Operating manual

10101
00110
11001

BySoft CAM

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1 Introduction



The BySoft CAM software allows you to efficiently and intuitively create cutting plans for laser and water cutting machines as well as bending and sorting plans. The integrated Parts and Job Manager helps you keep that essential top-level view of your data always to hand. The software is made up of several modules, which can be open and worked in simultaneously. If you have any questions about the program, the integrated online help will help you *(See: ›Help on the Online Help facility‹ Page 40)*.

The following topics will help you to learn to use BySoft CAM quickly or to deepen your understanding:

- **Structure and layout:** Familiarize yourself with the program's user interface.
- **Main functions:** Become familiar with the most important modules and functions for using the software.

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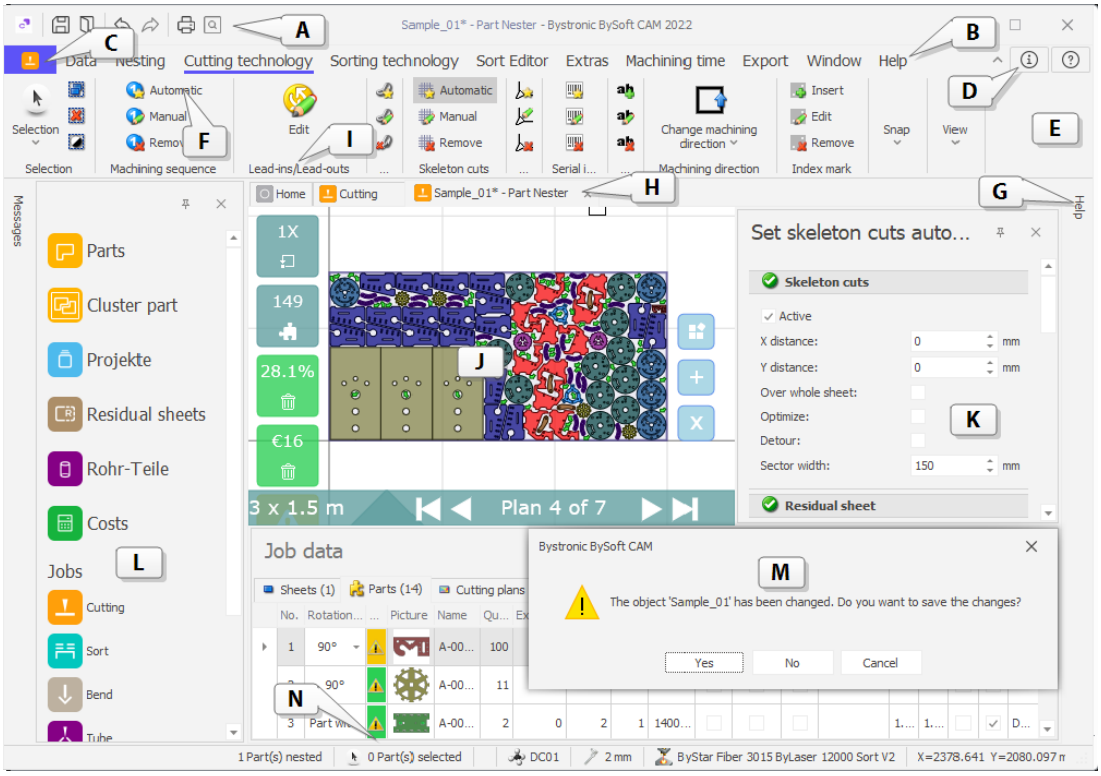
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2 Structure and layout

Thanks to its user-friendly interface, BySoft CAM is simple to use and easy to understand. The software user interface can be customized by moving and showing/hiding docking dialog windows. The settings are saved individually for each PC user account.

The software is composed of various modules. Licensed modules are enabled in the software and can be opened from the **Modules** dialog box.



A	Quick Access Toolbar	**B**	Menus in one module
C	Menu for file functions	**D**	Start icons for Help Explorer and Information window
E	Ribbon	**F**	Functions in the functional groups
G	Integral context-sensitive Help	**H**	Tabs for modules that are open
I	Functional groups in the Ribbon	**J**	Workspaces for a module
K	Docking dialog boxes	**L**	Selection dialog box for modules
M	Dialog boxes	**N**	Information in the status bar

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2.1 Menus and display elements

The menu bar displays all the functions that are available in the active view. The menu display is dynamic: only those menus and file functions are displayed that belong to the active module.

The software includes the following display elements:

- Modules
- Dialog boxes
- Docking dialog boxes
- Ribbon
- Status bar

2.1.1 Modules

The software is composed of various modules. Each module can be used to perform a separate, self-contained task. All the modules are displayed in the **Modules** dialog box, from where they can be opened.

Each open module is displayed in the program workspace as a separate object. When several modules are open at once, clicking on the tab of an object sitting in the background will bring this object to the front, where it can be worked in.

Working with two monitors

You can move each open docking dialog box onto a second monitor as a separate dialog box. This leaves you more space for working in the current module.

Moving a module view onto a second monitor

1. Click on the header of the docking dialog box and drag it with the mouse onto the second. monitor.
2. Drag the dialog box to the size you require.

Note: Use the **Reset windows** function in the **Window** menu to reset all the moved dialog boxes back to their default position.



Tip: Double-clicking in the header of a moved docking dialog box will dock the dialog box back in its default position in the software view.

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2.1.2 Dialog boxes

Dialog boxes are display windows in the software that can be positioned where you want them. As long as a dialog box is open, you cannot continue to work in the other software views. The following types of dialog boxes are used in the software:

Input dialog box

The input dialog boxes have a form-based layout. In the dialog boxes you can set various options or enter information. These dialog boxes can be closed with **OK** or **Cancel**.

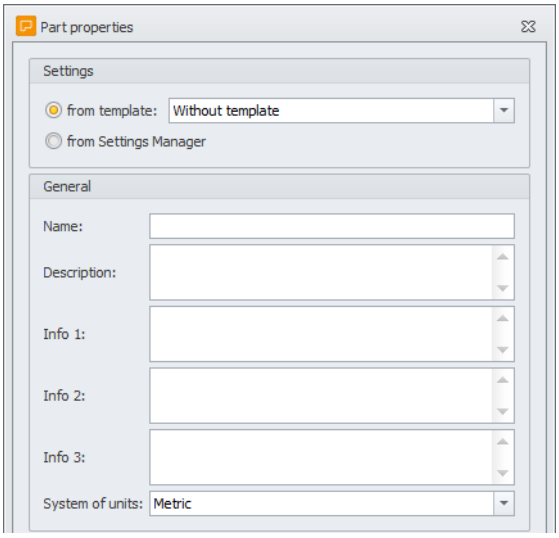


Fig. 2-1: Input dialog box for creating a new part

Prompt dialog box

Prompt dialog boxes act as a safeguard, for instance a prompt dialog box appears before a delete operation that is irreversible. These dialog boxes can be closed with **Yes**, **No** or **Cancel**.

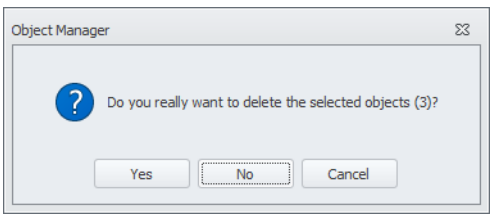


Fig. 2-2: Prompt dialog box when deleting an object in the Object Manager

Selection dialog box

In this dialog box you can select objects or files in an Explorer-type view. This dialog box can be closed with **Open** or **Cancel**.

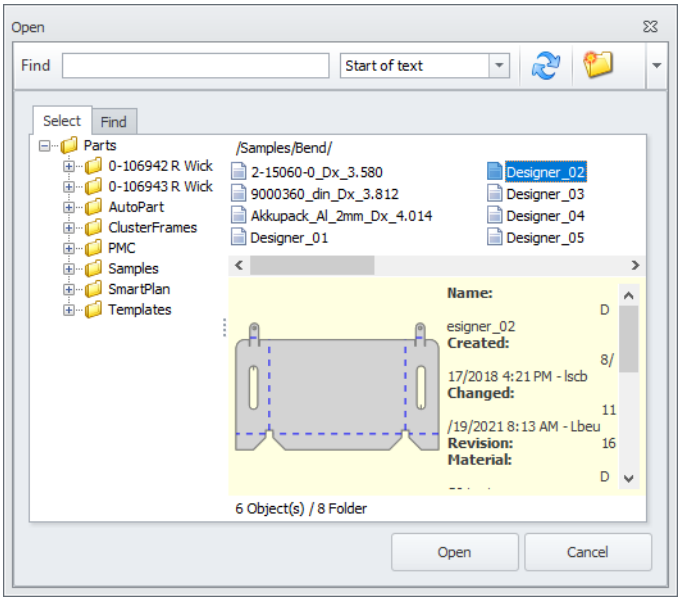


Fig. 2-3: Selection dialog box for opening a part

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Coordinates dialog box

This dialog box can be used with any function that requires the input of X and Y-coordinates. It opens automatically when you enter numbers from the keyboard.

Note: For the dialog box to open, the mouse pointer must lie within the object workspace.

Using the dialog box

1. Select the function you require.
2. Right-click in the workspace of the active object.
3. Use the keyboard to enter the X-coordinate. As soon as you touch a number key, the dialog box opens and the number is entered in the **X** field.
4. Press the **Tab** key to enter the Y-coordinate.
5. Press **Enter** to confirm the entry.

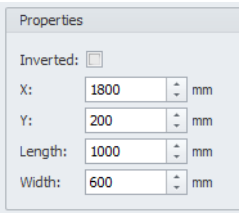


Fig. 2-4: Dialog box for entering X and Y-coordinates

2.1.3 Docking dialog boxes

Docking dialog boxes are dialog boxes that can be displayed permanently or temporarily. These dialog boxes are displayed at the sides of the software view or in the module. The position of these dialog boxes can be customized *(See: ›Arranging docking dialog boxes ‹ Page 1336)*.

Displaying docking dialog boxes

If a docking dialog box is displayed, it conceals the corresponding display area in the software. The dialog box can be fixed in order to keep it permanently displayed. The dialog box is then embedded in the user interface of the software. The display area of the relevant view is reduced in size by the size of the dialog box.

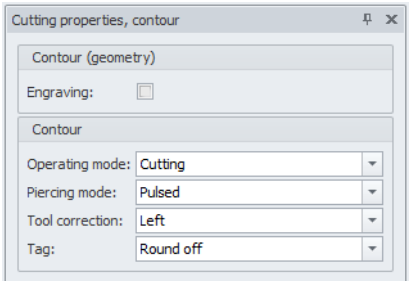


Fig. 2-5: Docking dialog box in the Part Editor module

> **Note:** The size and display settings for the docking dialog boxes are retained after closing the software.

Some docking dialog boxes can be shown or hidden using the **Window** selection function in the **Window** menu.

Displaying dialog boxes

- **Show dialog box**

  Move the mouse over the sensitive text of the docking bar.

- **Hide dialog box**

  Move the mouse into an area outside the dialog box.

- **Fix dialog box**

  1. Move the mouse over the sensitive text area of the docking bar in order to show the dialog box.
  2. Click on the icon to fix the dialog box. Click on the icon again to release the dialog box, which is then hidden automatically.

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2.1.4 Ribbon

The Ribbon is composed of one or more functional groups and the associated functions. Clicking on a menu will display the associated Ribbon.

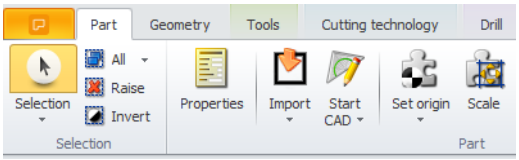


Fig. 2-6: The Ribbon in the **Geometry** menu of the Part Editor

Note: The Ribbon can be shown/hidden by clicking on the arrow icon at the right-hand edge of the menu bar. If the Ribbon is minimized, the Ribbon will be hidden again as soon as you click the mouse in a different area outside the Ribbon.

2.1.5 Quick access toolbar

This toolbar contains all the most commonly used functions for the module concerned. The quick access toolbar can be displayed above or below the Ribbon.

Functions in the toolbar:

- **Save:** *This function saves the current object.*
- **Open:** *This function opens a new object.*
- **Print:** *This function prints out the working documents for the part.*
- **Preview:** *This function starts the print preview.*
- **Undo:** *Undoes the last action.*
- **Restore:** *Restores the previous state.*



Fig. 2-7: Quick access toolbar

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2.1.6 Status bar

A status bar displays key information about the current object. The information is always shown in the bottom right edge of the software view whatever menu is selected.

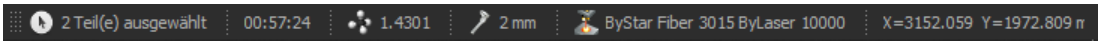


Fig. 2-8: Status bar in the Part Editor module

The status bar displays the following information depending on the module:

- Number of selected parts in the cutting plan
- 00:57:24 Job machining time
- Material of the part or plan
- Thickness of the part or plan
- X/Y dimensions of the part or sheet
- Effective weight of part
- Laser cutting machine assigned to the part or plan
- Bending machine assigned to the part
- Cutting process used for machining the part or plan
- X,Y X/Y-coordinates of the current mouse position with respect to the origin of the part or plan

Tip:

- If you move the mouse over the information area, the information is magnified and displayed with a short description.
- Click on an information area to call up the appropriate Settings dialog.

2.2 Global functions and dialog boxes

Global functions are functions that are used in more than one place within the software. For instance, the zoom functions are used in different menus in the Part Editor, Part Nester and Cutting Simulator modules. Similarly, the software uses dialog boxes and menus that appear in each module.

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2.2.1 Main menu

The Main menu displays all the menus available for the active module. Clicking on a menu will open the corresponding Ribbon.

File functions

Certain file functions are available for each module. The functions are displayed dynamically, i.e. only those functions are displayed that are available in the currently active module. The file functions can be displayed by clicking on the top left icon in the Main menu *(See: ›File functions‹ Page 49)*.

2.2.2 System settings



How do I open this dialog box?

Select: **File functions**

This dialog box contains the display preferences for the software user interface, the data-storage settings and settings for which modules to display.

Note: These settings can only be changed by users who are logged onto the PC with the **Administrator** user group.

Tabs in the System settings dialog box:

- Display
- Data storage
- Modules and functions
- Connection settings
- Extras

2.2.2.1 Display

How do I open this tab?

Select: **File functions -> System settings**

In this tab you can set the basic display preferences for the software.

Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

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General

- **System of units**

  Defines the system of units used in the software.

- **Appearance**

  Defines the color layout used for displaying the software. Changes in appearance take effect immediately and the software is displayed accordingly.

- **Language**

  Determines the language version of the software. All installed language versions are displayed in the selection menu.

> ☞ **Note:** After changing the language version, the software must be restarted to apply the language.

- **Invert direction of mouse wheel zoom**

  Inverts the zoom direction of the mouse wheel in the workspace of the following modules.

  - Part Nester
  - Part Editor
  - Tube Editor
  - Tube Nester
  - Auto Part
  - Cutting Simulator

> ☞ **Note:** If the layout is changed, the new layout is displayed immediately.

Settings for numerical values

- **Decimal places for length**

  Specifies the display of decimal places for all lengths.

- **Decimal places for angle**

  Specifies the display of decimal places for all angles.

- **Increment for length**

  Defines the increment per mouse click when setting a length using the arrow symbol in an input field.

- **Increment for angle**

  Defines the increment per mouse click when setting an angle using the arrow symbol in an input field.

2.2.2.2 Data storage

How do I open this tab?

Select: **File functions -> System settings**

The paths for the system files used in the software are defined in this tab. The system files are data objects that are saved in the file system in different folders. Special index files for each data-object folder are created for managing this data storage system.

The system files are saved by default in the user account in which the software was installed.

General

In this area, the default path for the data objects is defined as the highest folder level. You can update the index files for all data folders by pressing the **Update index** button.

Clicking on the three dots beside the **Default path** input field lets you define a new default path for all data objects. Any change to the default path is automatically applied to all the data objects in the **Data directories** area.

Data directories

In this area you can define for each data object a path that differs from the default path.

If the **Default path** option is disabled, a new path can be defined for the object type in question. Alternatively, the path can also be entered manually in the input field.

☞ **Note:** When the **Default path** option is enabled, the default path settings are adopted.

☞ **Note:** When there are several workstations in use, save the system files on a network drive. This lets all the workstations access the same system files, ensuring that a consistent result is obtained for nesting and the machining-time calculation.

Changing the default path

1. Disable the **Default path** option
2. Click on the three dots beside the input field.

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3. In the Folder dialog box, select the required folder and confirm the change with **OK**.

Description of data objects

The following data objects are managed under the respective folders:

Folder (data object)	Description
System data (**System**) - **AngleMeasurementSystemModels** = angle measurement systems - **BackStopModels** = backgauge models for press brakes - **BendingMachineConfigs** = configuration files for press brakes - **BendingMachines** = press brakes - **BendingMaterials** = materials for bending - **CuttingMachines** = cutting machines - **CuttingMaterials** = cutting materials - **CuttingRuleSets** = cutting tables - **LiftingAidModels** = bending aids - **LowerTools** = lower tools (dies) for press brake - **Materials** = materials - **SheetFormatSets** = sheet sizes - **UpperTools** = top tools for press brake - **VrModels** = 3D models	All the system files for the respective objects
Parts (**Parts**)	All the parts in the system
Parts jobs (**PartJobs**)	All the jobs in the system
Tubular parts (**Tubes**)	All the tubular parts in the system
Residual sheets (**ResidualSheets**)	All the residual sheets in the system
Report templates (**ReportTemplates**)	All the report templates in the system
Cost calculation (**Costings**)	All cost calculations in the system
Bending jobs (**BendJobs**)	All bending Jobs in the system

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Folder (data object)	Description
Auto Part projects (**AutoPartProjects**)	All Auto Part projects in the system
Sort jobs (**SortJobs**)	All sort jobs in the system

Update index of all jobs in the system

Clicking on the **Regenerate index** button will regenerate the indexing of all objects.

In order to regenerate the index of a single object type, click on the arrow beside the **Regenerate index** button and select the object type you require.

Note: The index only needs to be regenerated if deletion/addition of data objects in the data folder was performed outside of BySoft CAM.

Tip: Indexing can take a very long time for large databases. It is then best to update the index during quiet times or overnight.

Regenerate index

You can update the index by pressing the **Regenerate index** button. The progress in percent is displayed during index regeneration.

Note: The index only needs to be regenerated if deletion/addition of data objects in the data folder was performed outside of BySoft CAM.

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2.2.2.3 Modules and functions

How do I open this tab?

Select: **File functions -> System settings**

In this tab you can define which modules are available in the software. This allows the creation of custom module configurations when using multiple workstations.

To remove or restore a module in the software, disable or enable the relevant module in the **Enabled** column.

The **License status** column displays with different colors whether a license is present for the module.

- **Light green:** License present. The module can be used.
- **Dark green:** The module is currently being used.
- **Red:** License not present. The module cannot be used.

The user-defined settings on this tab are saved in the user folder. The **Reset Settings** button resets all user-defined settings. A confirmation prompt is displayed that must be confirmed by pressing **OK.**

Note:
The tab is displayed only if the program is run with administrator rights.

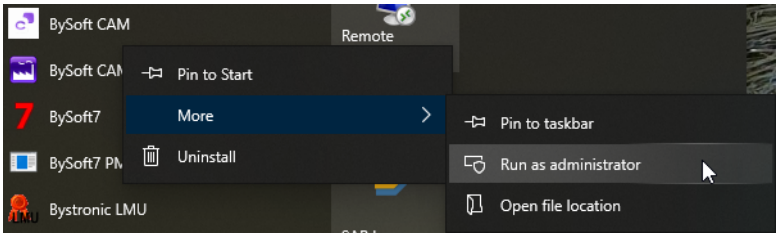


Fig. 2-9: Run program as administrator.

Tip: For workstations used purely for routine work, disable the **Administrator** module to prevent any changes to the Basic settings for system data.

2.2.2.4 Connection settings

How do I open this tab?

Select: **File functions -> System settings**

This tab is used to configure the connection settings for connecting to the database server and the BySoft CAM server. The servers can be specified using their server names or IP addresses.



Note: By default, the database and the BySoft CAM server will be found on the same server.

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2.2.2.5 Extras

How do I open this tab?

Select: **File functions -> System settings**

Additional functions can be enabled on this tab.

API

Determines where parts, jobs or bending jobs are stored that were created via a BySoft API. The directories are created automatically if they do not already exist in the file system.

Tilt optimization

Active

Enables or disables tilt optimization for the Part Editor and Part Nester modules *(See: ›Tilt optimization‹ Page 1039)*.

Tilt optimization prevents cutting gas pressure from causing parts to tilt or rise up after cutting if parts on the cutting grate have an unstable support.

This function prevents tilted parts from colliding with the cutting head and increases process reliability when cutting parts.

Global serial number

The global serial number is set when the function **Automatic serial number information** is selected in Part Nester in the **Cutting Technology** menu *(See: ›Automatic serial-number information‹ Page 1074)*. Each time the function is selected, the number is incremented by the number of respective parts in the cutting plan.

Active

If this option is enabled, **all** nested parts in a job on which the **Serial number information** data field is set will be engraved with a consecutive serial number.

The number is displayed in the adjacent field. The number can be adjusted manually. The next time a cutting plan is exported, the parts will be engraved with a serial number starting from the specified number.

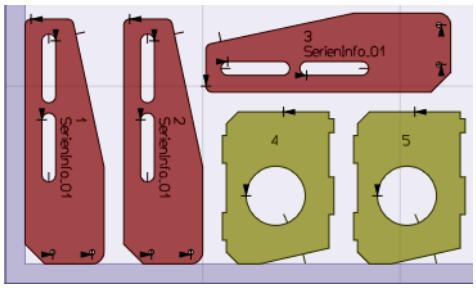


Fig. 2-10: Global serial numbers on all parts in a cutting plan

By part

If this option is enabled, a separate consecutive serial number is engraved on each nested part nested part in a job on which the **Serial-number information** field is set.

Otherwise, the behavior is the same as described under **Active**.

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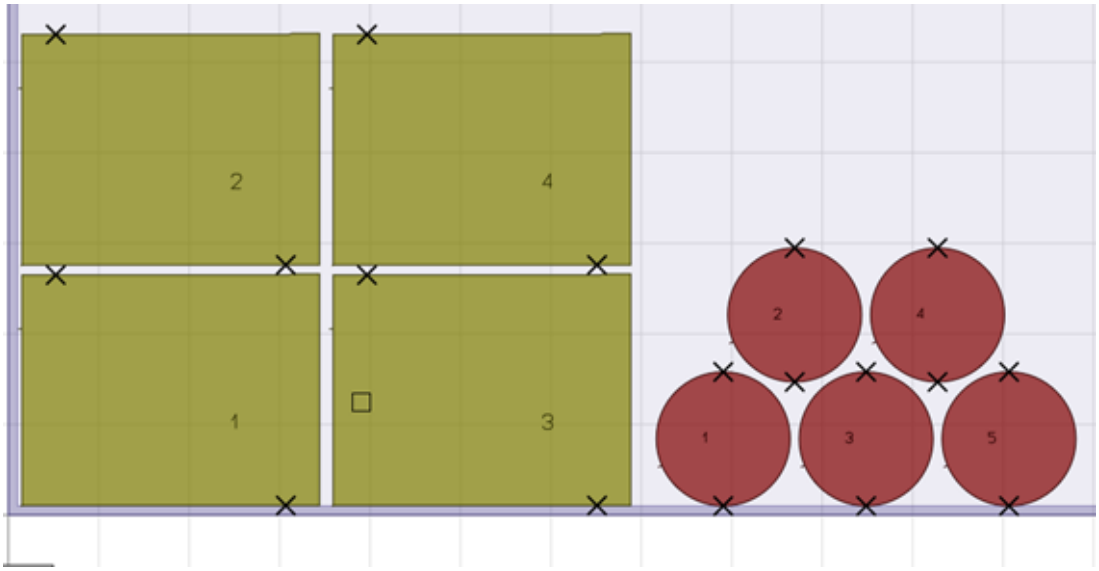


Fig. 2-11: Global serial numbers on different parts in a cutting plan



Note: The serial-number information for each part is stored in the ***PartsGlobalSerialNumbers.box*** file. The file is located in the following directory: ***Drive:\BystronicData\BySoftCam\System\General***

- **Synchronize with data store**

  If parts were deleted from the data store, they are also removed from the ***PartsGlobalSerialNumbers.box*** file via the button.

Cost tables

Import folder

> ⚠ **Attention!** For the import of Excel files to work correctly, the "Microsoft Access Database Engine 2016" is required.(https://www.-microsoft.com/download/details.aspx?id=54920).

Defines the path for automatic import of operation and material cost tables from an Excel file into the **Administrator** module. As soon as a table is copied into the directory, it is automatically imported into the **Cost Tables** form in Administrator.

In order for the tables to be imported without any errors, they must be set up with a certain structure. Also, the file name must match the definitions in the cost table.

To achieve this, export the corresponding tables from the **Cost Tables** form beforehand and edit the table according to your requirements.

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Edit operation costs from a cost table

1. In the **Available entries** area, select the cost table from which to export operation costs.
2. On the **Operations** tab, select the operation whose costs are to be exported to an Excel spreadsheet.
3. Select the header of the **Operation costs** table.
4. Select the **Export to MS Excel** function in the context menu.
5. In the file explorer, specify the location and file name of the table and press **Save** to confirm. In order for the table to be automatically re-imported into this directory, the file name must be specified as follows:
   - ***"Name of cost table"_"Name of operation"***_operationcosts.xls
   - Sample_Cutting_operationcosts.xlsx
6. Customize the table with your own values and then copy it to the directory specified here. The new values of the operation costs are automatically updated in the **Cost Table** form in Administrator.

bis und mit Dicke [mm]	Kosten/Einheit 1	Kosten/Einheit 2	Kosten/Einheit 3	Prozesskosten	Optionale Zeit
Werkstoff: 1.4301					
1	0.03	0	0.8	0	00:00:00
2	0.05	0	0.9	0	00:00:00
Werkstoff: AW5754					
2	0.09	0	0.76	0	00:00:00
Werkstoff: DC01					
1	0.05	0	0.79	0	00:00:00
1.5	0.05	0	0.8	0	00:00:00
2	0.08	0	0.92	0	00:00:00
3	0.17	0	0.99	0	00:00:00
Werkstoff: DD11					
4	0.32	0	0.99	0	00:00:00
5	0.58	0	1.13	0	00:00:00
Werkstoff: S235JR					
12	0.67	0	0.72	0	00:00:00

Fig. 2-12: Operation costs table

Edit material costs from a cost table

1. In the **Available entries** area, select the cost table from which to export the material costs.
2. Select the header on the **Material Costs** tab.
3. Select the **Export to MS Excel** function in the context menu.
4. In the file explorer, specify the location and file name of the table and press **Save** to confirm. In order for the table to be automatically re-imported into this directory, the file name must be specified as follows:
   - ***"Name of the cost table"***_operationcosts.xls
   - Sample_materialcosts.xlsx
5. Customize the table with your own values and then copy it to the directory specified here. The new values of the material costs are automatically updated in the **Cost table** form in Administrator.

bis und mit Dicke [mm]	Einheit	Kosten/Einheit	Verschnittvorgabe [%]
Werkstoff: 1.4301			
1	kg	5	40.00%
1.5	kg	5	40.00%
Werkstoff: AW5754			
2	kg	6.3	0.00%
Werkstoff: DC01			
3	kg	1.2	0.00%
2	kg	1.2	0.00%
1.5	kg	1.2	0.00%
1	kg	1.2	0.00%
Werkstoff: DD11			
5	kg	1.2	0.00%
4	kg	1.2	0.00%

Fig. 2-13: Material costs table

Note: After importing, an **Imported** subdirectory is created in the specified directory and the imported tables are moved to this directory. At the same time, a log file called **TemplateImport.log** is created.

Exporting system data

The **Export** button exports system data to two Excel files (part.csv, job.csv).

If exporting from a large database takes too long, you can press the **Cancel** button to cancel the export process. All parts or jobs exported up to that point are stored in the appropriate Excel file.

- **part.csv:** Contains information about all parts in the database.
- **job.csv:** Contains information about all jobs in the database.

Exporting system data

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1. Press the **Export** button.
2. In the dialog, specify the folder where you want to save the files.
3. Confirm your entry with **OK**.

Close application if inactive

This section defines automatic termination of the software.

Active

If this option is enabled, the software will be terminated automatically after a certain time. A confirmation dialog is displayed before closing.

- **Time period**

  Defines the time without any activity in the software after which a confirmation dialog for closing the software is displayed.

- **Time for prompt**

  Defines how long the confirmation dialog is displayed. After this time has elapsed without confirmation in the dialog, the software is automatically closed.

Close without prompt

If this option is activated, the software is closed without a prompt after the time has elapsed.



Tip: You can use this option to automatically release unused server licenses for the software.

2.2.3 Window

The **Window** menu is available in every module. It contains functional groups for displaying and arranging the layout of modules and docking dialog boxes. An additional functional group for controlling the display is included in the **Part Nester** and **Cutting Simulator** modules

2.2.3.1 Group: Shell

Where do I find the group?

Select the menu: **Window**

In this group you can specify how dialog boxes and modules are displayed and positioned.

Window

Where do I find the function?

Select the menu: **Window**

This function opens a dialog box from where you can show/hide various docking dialog boxes. These dialog boxes are always visible in the software interface.

The following dialog boxes can be shown or hidden:

- Modules
- Messages
- Dynamic help

Modules

How do I open this dialog box?

Select the menu: **Window > Shell > Window**

This docking dialog box contains all the modules that are enabled in BySoft CAM. Hover the mouse over the icons to show or hide the functions for the module.

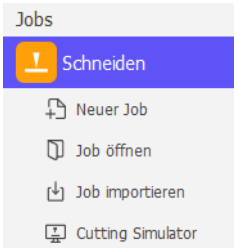


Fig. 2-14: Open dropdown field of the Part Nester module

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The folder structure with the objects of the selected module is displayed to the left of the module display in Object Manager. The functions and menus for managing objects are displayed in the header.

Modules which are enabled but for which there are no machines or components are displayed grayed-out in the following cases.

- The Tube Editor / Tube Nester modules are disabled.
  - No laser machine has the **rotary axis** option in Administrator.
- The **Tools** menu in Part Editor and the Tool Setup Optimizer module are disabled.
  - No bending machine available in Administrator.
- The Bend Robot module is disabled.
  - The Bend Robot program is not installed. The program can be launched from Part Editor in the **Bending** section -> **Export** menu.



Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

Sheet metal parts

Parts can be opened, imported or created from scratch in the Part Editor. Before a part can be imported or a new part created, certain details about the part must be entered or checked in the **Part properties** dialog.

- **New part:** Opens the **New part** dialog box in which you can record the master data for the new part.
- **Import part:** Opens the **Import part** dialog box for selecting the file you wish to import.

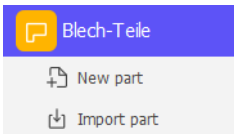


Fig. 2-15: Sheet metal parts section in the **Modules** docking dialog

Cluster part

Cluster parts can be opened and edited.



Fig. 2-16: Cluster part section in the **Modules** docking dialog

Tubular parts

You can open existing tube parts or import new ones in Tube Editor.

- **Import part:** Opens the **Import part** dialog box for selecting the file you wish to import.



Fig. 2-17: Tubular parts section in the **Modules** docking dialog

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Assemblies

In Auto Part, you can extract components from CAD assemblies and assign them a technology. Before an Auto Part project can be created, it is necessary to enter certain details about the project in the **Project properties** dialog.

- **New project:** Opens the **Project properties** dialog where you can specify the template and the name of the project.

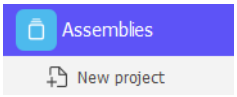


Fig. 2-18: Assemblies section in the **Modules** docking dialog

Residual sheets

All existing residual sheets are displayed and can be edited.

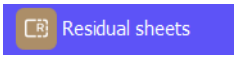


Fig. 2-19: Residual sheets section in the **Modules** docking dialog

Cutting

Jobs can be opened, imported or created from scratch in the Part Nester. Before a job can be imported or a new job created, certain details about the job must be entered/checked in the **Job properties** dialog box.

- **New job:** Opens the **Job Properties** dialog box in which you can record the master data for the new job.
- **Open job:** Calls up the **Open** dialog to select the appropriate part.
- **Import job:** Opens the **Import job** dialog box for selecting the job that you wish to import.
- **Cutting Simulator:** Opens the dialog for selecting a cutting plan to be simulated in the **Cutting Simulator** module. After selecting, the cutting plan is opened in Cutting Simulator.

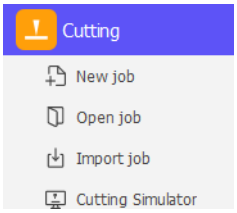


Fig. 2-20: Cutting section in the **Modules** docking dialog

Sort

Sort jobs can be opened, imported or created in Sort Job Editor. Before a job can be imported or a new job created, certain details about the job must be entered/checked in the **Job properties** dialog box.

- **New job:** Opens the **Job properties** dialog in which to enter the master data of the sort job.
- **Import job:** Opens the **Import job** dialog box for selecting the job that you wish to import.

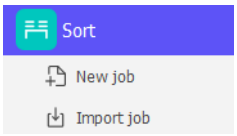


Fig. 2-21: Sort Job Editor section in the **Modules** docking dialog

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Bend

Bending jobs can be opened, imported or created from scratch in the Tool Setup Optimizer. Before a bending job can be imported or a new job created, certain details about the bending job must be entered/checked in the **Job properties** dialog box.

- **New job:** Opens the **Job properties** dialog box in which you can record the master data for the new bending job.
- **Import job:** Opens the **Import job** dialog to select the bending job that you wish to import.

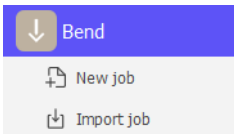


Fig. 2-22: Bending section in the **Modules** docking dialog

Tube

Tube jobs can be opened, imported or created from scratch in the Tube Nester. Before a tube job can be imported or a new tube job created, certain details about the tube job must be entered/checked in the **Job properties** dialog box.

Details

- **New job:** Opens the **Job Properties** dialog box in which you can record the master data for the new tube job.
- **Open job:** Calls up the **Open** dialog to select the appropriate tubular job.
- **Import job:** Opens the **Import job** dialog box for selecting the job that you wish to import.

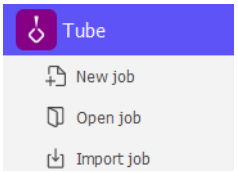


Fig. 2-23: Tube section in the **Modules** docking dialog

Costs

Use the “Cost Calculator” to perform cost calculations for flat and tubular parts. You can use these cost calculations to create quotations for customers or to optimize production costs, for example.

- **New cost calculation:** Opens the **Cost calculator properties** dialog box in which you can enter the master data for the new cost calculation. Press **OK** to create a new cost calculation based on the master data.
- **Import cost calculation:** Opens the dialog for importing an existing cost calculation.

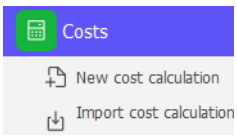


Fig. 2-24: Cost Calculator section in **Modules** docking dialog

Plant Manager

The **Plant Manager** application, **PM** for short, provides a fully or semi-automated procedure for creating production jobs from an ERP system and assigning them to a machine. You also have the option of intervening in the order process at any time.

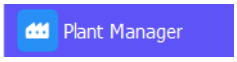


Fig. 2-25: Plant Manager area in the **Modules** docking dialog box

Parts removal

Parts removal can simplify clearing cut jobs. Parts removal is connected to Plant Manager.

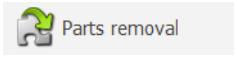


Fig. 2-26: Designer area in the **Modules** docking dialog box

Messages



How do I open this dialog box?

Select the menu: **Window > Shell > Panels**

This docking dialog box logs error messages and general information about work carried out on modules. This can be used to check what work has been performed on a part or job.

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Note: All information is deleted after closing the program.

Dynamic Help facility



How do I open this dialog box?

Select the menu: **Window > Shell > Panels**

This dialog box is used to display all the context-sensitive Help texts, i.e. Help texts that open in a specific context. The icons in the header can be used to run display and navigation functions *(See: ›Icons and functions in Help Explorer‹ Page 43)*.

Tip: To enlarge the display area, the docking dialog box can be dragged with the mouse to a different position on the screen or onto a second monitor *(See: › Working with two monitors‹ Page 5)*.

Reset windows



Where do I find the function?

Select the menu: **Window > Shell**

This function resets all user-defined window settings for docking dialog boxes back to the original factory settings.

Modules that are open



Where do I find the function?

Select the menu: **Window > Shell**

This function opens a selection box displaying all the modules that are open. Each module is shown with the object name and the relevant module name. If an object has not been saved yet, a * symbol is shown after the object name.



Tip: This function provides a quick way of selecting the module require when a large number of modules are open at once.

2.2.3.2 Group: Part Nester

Where do I find the group?

Select the menu: **Window**

The default layout for the program window can be restored using the **Reset windows** function in this group.



Note: This group is only displayed in the Part Nester module and in the Simulator.

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2.2.4 Help

The **Help** menu is available in every module. It contains the Help function group with all of the available help functions.

2.2.4.1 Help on the Online Help facility

How do I open this dialog box?

Select the menu: **Help > Help**

The web-based online help is an integrated part of the program. The software and the Online Help are displayed in the language version set for the operating system. The language version can be changed temporarily within the help at any time. The online help is designed responsively and can therefore also be displayed on mobile devices without any problems.

Possible ways of displaying the Help facility

The software Help facility can be displayed in the following ways:

- Displaying the entire Help facility in a Help Explorer view using the icon in the menu bar

  The size and position of the display area can be customized. The size and position is saved after closing.
- Context-sensitive display of the Help facility in the **Help** docking dialog box

  The docking dialog box is displayed by default on the right-hand side in the software window. The size of the dialog box is saved automatically.
- Tooltip display for all functions

  The tooltip (**A**) is automatically displayed when you hover the mouse over the desired function. The tooltip is displayed for all functions in the ribbon and the file functions.

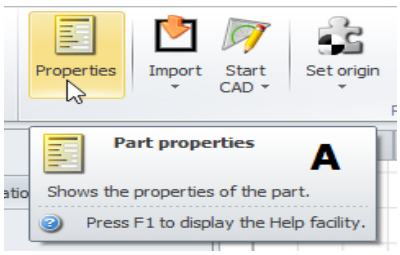


Fig. 2-27: Displaying the tooltip for a function

Show home page in Help Explorer

The Help Explorer consists of a home page and a content area.

to customize the Help user interface. The customized user-interface settings can be saved in a profile under **Window > Settings**

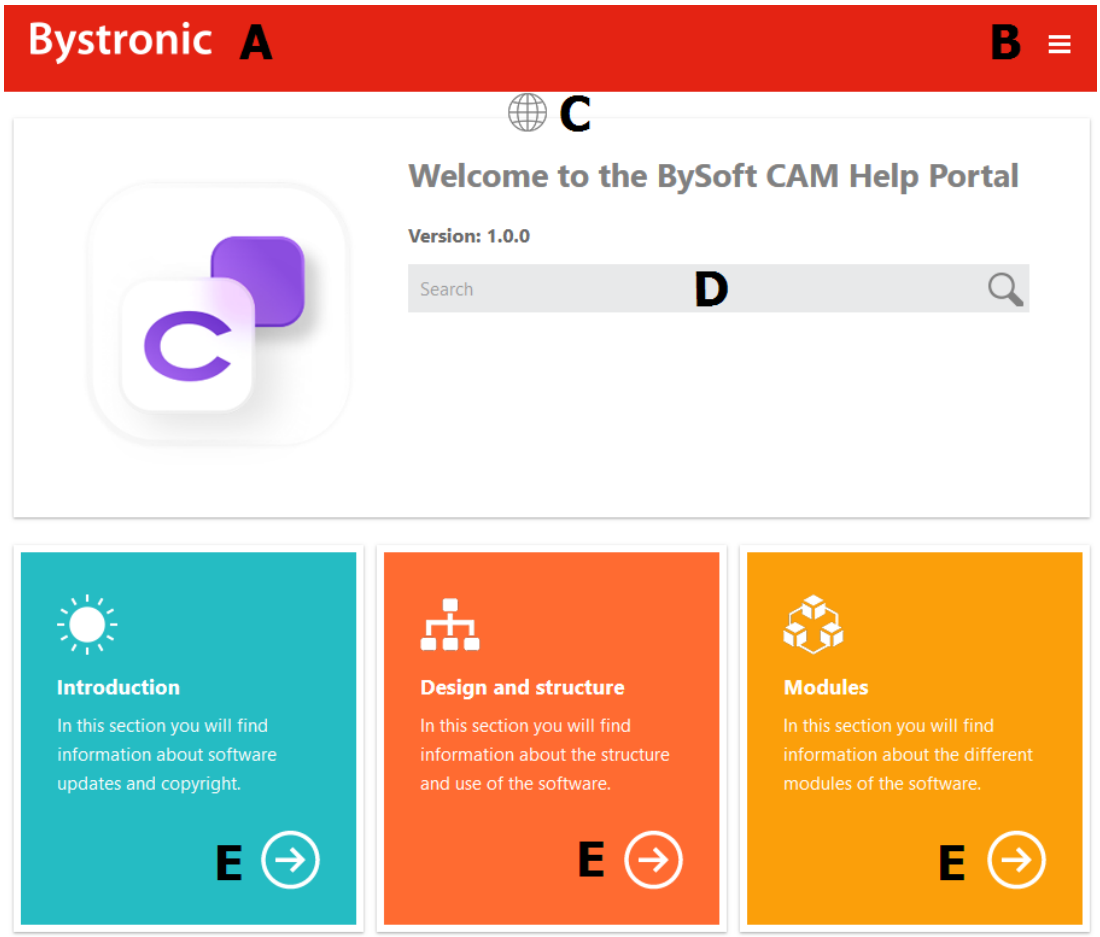


Fig. 2-28: Home page display areas

A	Text link to the home page
B	Slide menu icon in tablet and mobile view
C	Help language selection icon
D	Input box for full-text search
E	Link button to the respective tile topics

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Displaying the integrated context-sensitive Help

The context-sensitive Help is displayed in the **Help** docking dialog box.

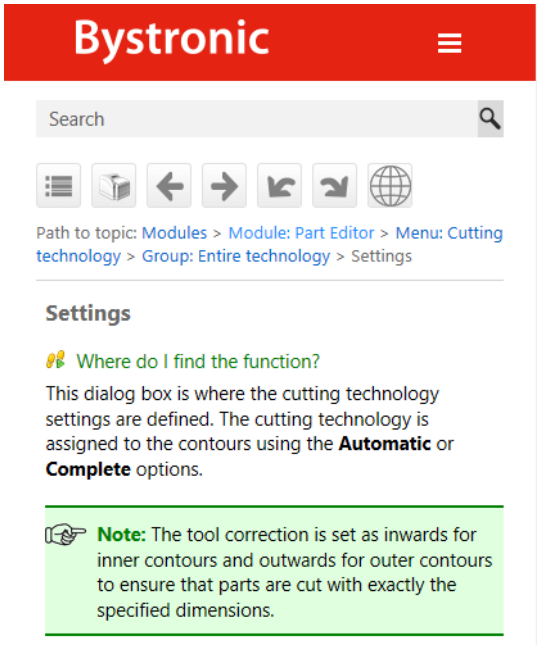


Fig. 2-29: Displaying the context-sensitive Help in the docking dialog box

Opening the context-sensitive Help

- Move the mouse over a function and press the **F1** key.
- Click in a docking dialog box or a tab.
- Press the **F1** key in a dialog box. A separate Help window will open.

Options for displaying the Help dialog box

- **Show dialog box**

  Move the mouse over the sensitive text of the docking bar.
- **Hide dialog box**

  Move the mouse into an area outside the dialog box.
- **Fix dialog box**

  1. Move the mouse over the sensitive text area of the docking bar in order to show the dialog box.
  2. Click on the icon to fix the dialog box. Click on the icon again to release the dialog box, which is then hidden automatically.

Displaying important information

Tip: Tips to make it easier to work with the software.

Note: Additional information for working with the software that helps you gain a better understanding.

Attention! Information that must be heeded to avoid malfunctions or irretrievable data loss.

Icons and functions in Help Explorer

Icon	Functions	Display	Description
	Contents	Navigation area	Opens the table of contents in a sidebar for structured display of topics in the Online Help. Each structure is displayed in three levels.
	**Find**	Navigation area	Searches the entire Help system to find required terms and displays the search results in the content area in order of how often the term appears in them.
	**Show all**	Toolbar	Opens all the drop-down texts in the displayed topic.
	**Hide all**	Toolbar	Closes all the opened drop-down texts in the displayed topic.
	**Previous**	Toolbar	Jumps to the previous topic in the table of contents structure.
	**Next**	Toolbar	Jumps to the next topic in the table of contents structure.
	**Back**	Toolbar	Jumps to the topic that was displayed before the current topic.
	**Next**	Toolbar	Jumps to the topic that was displayed after the current topic. If the displayed topic is the last one selected, the function has no effect.

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Icon	Functions	Display	Description
	**Print**	Toolbar	Prints out the topic displayed in the browser window.
	**Language change**	Toolbar	Displays all available language versions of the Help. Selecting an entry displays the topic in the selected language.
 Related to	**Related topics**	Help topic	This navigation mode links together similar topics and displays them in a popup window. Related topics are always displayed at the end of a topic.
	**Scroll up**	Help topic	Scrolls the display back to the beginning of the topic. The icon is displayed only when the content exceeds the display window by a certain size.
Jobs	**Hyperlinks**	Help topic	Hyperlinks are highlighted in blue and underlined in the text. Hyperlinks can be used to jump from one topic to another.
Code:	**Expanding text**	Help topic	Expanding texts are highlighted in blue but not underlined. Clicking on an expanding text displays extra information on the source text.

2.2.4.2 Quick Support



How do I open this dialog box?

Select the menu: **Help > Help**

This function establishes a secure online connection between your PC and Bystronic Customer Service. This makes it possible to discuss problems and questions directly on the screen. To establish the connection, call your responsible Customer Service department and tell them the ID (**A**) and the password (**B**).

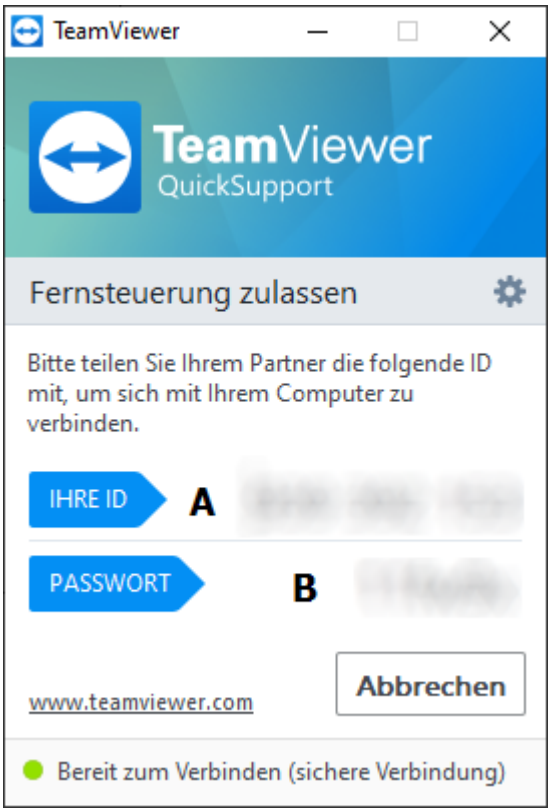


Fig. 2-30: Quick Support

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2.2.4.3 Customer portal



Where do I find the function?

Select the menu: **Help > Help**

This function initiates the login process in the Bystronic Software Portal. In the portal you can:

- Download updates for your software
- Download documents and operating instructions
- Watch tutorials about software functions
- Get answers to the most frequently asked questions about BySoft
- Check your software licenses

☞ **Note:** Before using the portal for the first time, you will receive an email from the local Bystronic support with login credentials for the portal (user name/-password).

Login is performed in the following steps:

1. Enter user name and password.

   ☞ **Note:** When logging in for the first time, you will be redirected and have to enter your own password.

2. Press the LOGIN button.

2.2.4.4 About



How do I open this dialog box?

Select the menu: **Help > Help**

About

This tab lists information about the operating system and the modules used by BySoft CAM. Please provide this information when Support asks for the current software version. The program version is shown at the beginning of the tab.



Note: Please use the **Send** function to provide Bystronic Customer Service with the information when needed.

3rd party libraries

This tab shows all third-party libraries and licensing information.

Conditions and supports

This tab displays the End User License Agreement for Bystronic machine software (EULA). The license agreement can be printed out if required.

Functions in the dialog:

- **Send**
  The contents of the tab can be sent by e-mail.
- **Save**
  The contents of the tab are stored in a text file.
- **Close**
  Closes the dialog box.
- **Print**
  Prints the end user license agreement.

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2.2.5 Docking dialog boxes

Global dialog boxes are displayed in all views and modules. They can be shown or hidden. Most docking dialog boxes can be displayed using the **Window** function in the **Window** menu.

2.2.6 File functions

The global file functions are available in several modules in the software. Most of the file functions are used in the Part Editor and Part Nester modules *(See: ›File functions‹ Page 14)*.

2.2.6.1 Open

This function opens a dialog box to search for and open an object. Various tabs are available for the object search.



Tip: The function can also be launched from the quick access toolbar *(See: ›Quick access toolbar‹ Page 11)*.

Locating objects

- **Select**

  To search the job database manually.

  Select the object in the **Select** tab and then click the **Open** button.

- **Find**

  To search using the **Search** box in the header.

  Enter the name of the object in the **Search** box and press Enter to confirm. The Search tab displays all objects found in the **Search result** section.

- **Advanced search** (Part Nester + Tube Nester only)

  To search for parts in jobs.

  Enter the name of the part in the **Part name** box and press Enter to confirm. All parts found are displayed in the top section. All jobs containing the parts found are displayed in the bottom section.

You can use a list box to restrict the search according to certain criteria.

- **Entire text**

  The program searches for a text that matches the character string exactly.

- **Start of text**

  The program searches for a text that starts with this character string.

- **Part of text:**

  The program searches for a text that contains this specified character string.

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Editing objects on multiple workstations

If the database is located on a network, an object cannot be edited by more than one workstation at once.

For instance, if a part is open in Part Editor, the same part on a different BySoft workstation can only be opened in read mode. A message to this effect is displayed. If the part is modified on this workstation, it has to be saved under a new name in the Object Manager.

What each button does:

Icon	Description
	This function displays the object from the previous search. You need to have performed the Search function at least once before this function can be used.
	This function displays the object from the subsequent search. You need to have performed the Search function at least once before this function can be used.
	This function opens the **Search** dialog to search for an object based on certain filter criteria.
	This function takes you one level up in the tree view. This function is only enabled when the root folder is not selected.
	This function can be used to refresh the display of the objects by reloading the contents of the folders.
	This function lets you customize the view for the objects in the Object Manager windows to suit your needs.
	This function can be used to display in the active window an Information pane containing extra information on the selected object or folder.
	This function lets you display in Explorer those objects that have the «Hidden» attribute.
	This function can be used to create a new folder.
	This function lets you rename the selected folder.
	This function can be used to delete the selected folder. The folder can only be deleted if it not longer contains any objects.

2.2.6.2 Save



Where do I find the function?

Select: **Main menu**

This functions saves the current object.



Tip: The function can also be launched from the quick access toolbar *(See: ›Quick access toolbar‹ Page 11)*.

2.2.6.3 Save all



Where do I find the function?

Select: **Main menu**

This function saves all opened objects. If there are objects open that have not been saved yet in the Object Manager, the **Save as...** dialog box opens in which you can save the object in the Object Manager.

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2.2.6.4 Save as...



Where do I find the function?

Select: **Part Editor >** 

This function can be used to save the current object in a new folder or under a new name.

In the **Select** tab, specify the folder in which you wish to save the object. In the **Name** field you can enter a new name for the object. When you click the **Save** button, the object is saved in the selected folder.

What each button does:

Icon	Description
	This function displays the object from the previous search. You need to have performed the Search function at least once before this function can be used.
	This function displays the object from the subsequent search. You need to have performed the Search function at least once before this function can be used.
	This function opens the **Search** dialog to search for an object based on certain filter criteria.
	This function takes you one level up in the tree view. This function is only enabled when the root folder is not selected.
	This function can be used to refresh the display of the objects by reloading the contents of the folders.
	This function lets you customize the view for the objects in the Object Manager windows to suit your needs.
	This function can be used to display in the active window an Information pane containing extra information on the selected object or folder.
	This function lets you display in Explorer those objects that have the «Hidden» attribute.
	This function can be used to create a new folder.
	This function lets you rename the selected folder.
	This function can be used to delete the selected folder. The folder can only be deleted if it not longer contains any objects.

2.2.6.5 Quick print



Where do I find the function?

Select: **Main menu > Working documents / Print labels**

This function sends the print job directly to the default printer in the operating system. There is no chance to set print preferences.

2.2.6.6 Preview



Where do I find the function?

Select: **Main menu > Working documents / Print labels**

This function opens the active object in a print preview. The preview window contains several options for modifying the document. You can also use the **Print**  option to print out the document from the preview.

Note: If the **Centre punch mark** attribute is set on a contour, the center punch mark is displayed as a black dot in the center of the contour. The contour itself is shown as a gray line.

Modifying a document

The preview window provides the following options for modifying the document:

-  **Find:** *You can search for specific words or numbers in the document.*
-  **Page setup:** *You can change the paper size, page orientation and margins.*
-  **Scale contents:** *The parts and jobs can be shown larger or smaller on the page.*
-  **Save document:** *Saves the document in various formats*
-  **Send document:** *The document can be sent as an email in various formats.*

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2.2.6.7 Undo



Where do I find the function?

Select: **Toolbar >** 

Reverses the operation that was last performed.

2.2.6.8 Restore



Where do I find the function?

Select: **Toolbar >** 

Redo function for restoring the operation that was last reversed using “undo”.

2.2.6.9 Close

Where do I find the function?

Select: **Main menu**

Closes the program. If you have not saved all the objects, a dialog box appears in which you can save these objects before closing.

2.2.7 Functional groups

Functional groups are where several similar functions are grouped in the Ribbon. Global functional groups are shared by several modules and menus.

2.2.7.1 Machining time

This group contains all the functions for calculating machining time.

Remove machining time



Where do I find the function?

Select: **Module > Machining time > Machining time**

This function removes the calculated machining time again.

Edit parameters



Where do I find the function?

Select: **Module > Machining time > Machining time**

This function opens the parameter set specified in the **Cutting parameters** field for editing in **Parameter Editor**. Parameter editing can be used to optimize machining time.

Proceed as follows to check the change in BySoft CAM:

1. Save the modified parameter in Parameter Editor.
2. Click **Calculate** in the **Machining time** menu.



Caution! The modified values are saved directly to the parameter set. The changes therefore have a direct impact on the machine's machining process if the machine also uses the parameters!



Note: Only edit the parameters if you can gauge the effects of the changes.



Tip: Use this function to optimize machining time.

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2.2.7.2 Selection

This group displays all the available selection functions. These functions can be applied to all elements of an object.

Select



This function lets you select one or more elements of an object. The mouse pointer is displayed as crosshairs  . To select an element, click with the crosshairs on the element concerned.

- **Add elements**

   Shift + right-click

  Use this key combination to add extra elements to a selection.

- **Deselect elements**

   Ctrl + right-click

  Use this key combination to deselect individual elements from a selection.

- **Invert elements**

   Ctrl + Alt + right-click

  This keyboard shortcut inverts the element selection.

Selecting several elements

These functions let you select several elements of an object at once.

- **Cross**
  Dragging the selection window from right to left will select all those elements that lie inside the window or lie across the window frame.
- **Inside**
  Dragging the selection window from left to right will select all those elements that lie entirely inside the window.
- **Outside**
  Dragging the selection window while holding down the **Shift + Ctrl** keys will select all those elements that lie outside the window and do not cross the window frame.

All

This function selects all the elements of the selected attribute.



Note: The selection functions can be used in combination with other functions.

Tip: After running this function, you can use **left-click + Ctrl** key to deselect elements again individually.

Thread

Select this function to select all contours defined as threads on a part.

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Open contours

Select this function to select all open contours on a part.

Application:

- To engrave all open contours.

Inside contours

Select this function to select all inside contours on a part.

Application:

- To add microjoints to all inside contours.

Outside contours

Select this function to select all outside contours on a part.

Equal

Selects all instances of the selected object.

Application:

- Select all texts on a part
- Select all bend lines
- Select contours with the same operating mode

☞ **Note:** This function is only enabled if at least one object is selected.

Remove

This function removes all the elements from a selection. The ESC key also performs the same function.



Note: You cannot undo the selection removal.

Invert

This function deselects all selected elements while selecting all unselected elements.



Tip: Running the function again will restore the original selection.

Same sector

Use this function to select all the parts which are located in the same sector as the marked part.



Note: This function is only displayed for the cutting plans of ByVention machines.

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2.2.7.3 View

This group contains all the functions that can be used to change the current view of a part or job.

The various functions and views are classified as follows:

Zoom functions

The zoom functions can be used to magnify or reduce objects in the graphics window.



Tip: You can also perform the zoom functions using the mouse thumb wheel.

Moving the view

The arrow symbols are provided in this group for moving the view of an object.



Tip: You can use the center mouse button to move the object freely in the current view. The view is moved by dragging and dropping with the mouse.

Special view

The **Special view** function is used to display various view options. The special-functions display can be specified by clicking on the arrow symbol and enabled/disabled by clicking on the symbol above it.

Zoom -

This function reduces the current image segment to half its size. You can also perform this function using the mouse thumb wheel.



Tip: Use this function to display more elements of an object.

Zoom +

This function enlarges the current image segment to twice its size. You can also perform this function using the mouse thumb wheel.



Tip: Use this function to display a detailed view of the object.

Zoom window

You can use the mouse to drag a window around the object to define the display area. When this function is enabled, the cursor is shown as a magnifying glass.





Tip: Use this function to magnify specific areas of an object.

Zoom all

This function displays the maximum size of the object in the working window.



Tip: Use this function to switch from the detailed view into the full view of the object.

Zoom selection

This function zooms the selected contour to the size of the working area of the active module.



Note: If multiple contours are selected, each click displays the next contour.

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Tip: Use this function to display very small contours. This can be the case if any contours smaller than 1mm are found and selected when checking the part *(See: ›Contours shorter than 0.1 mm‹ Page 394)*.

Pan left

This function moves the object to the left by 1/5 of the size of the workspace window.



Tip: Use this function to reveal elements on the right of the object that are not visible.

Pan right

This function moves the object to the right by 1/5 of the size of the workspace window.



Tip: Use this function to reveal elements on the left of the object that are not visible.

Up

This function moves the object up by 1/5 of the size of the workspace window.



Tip: Use this function to reveal elements at the bottom of the object that are not visible.

Down

This function moves the object down by 1/5 of the size of the workspace window.



Tip: Use this function to reveal upper elements of the object that are not visible.

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2.2.7.4 Snap

This group contains all the functions for picking geometry points of an object. For instance, you can use these functions to measure distances of a geometry or to set a new origin on a part.

Note: The snap functions can only be used together with another function.

Snap point preview

When using snap functions, the snap point is displayed as an orange dot on a contour next to the mouse cursor as a preview. Left-click to select the snap point.

Note: Only the snap points of the selected snap functions are displayed.

Orthogonal



If this function is enabled, the cursor can only be moved horizontally in the X direction or vertically in the Y direction.

This functions allows you to draw axis-parallel lines and align contours parallel to the axis. The direction is defined by moving the mouse in the appropriate direction.

Tip: This function can be combined with grid and snap functions.

Next



This function snaps to the nearest point on the selected element. The element must lie inside the cursor box.

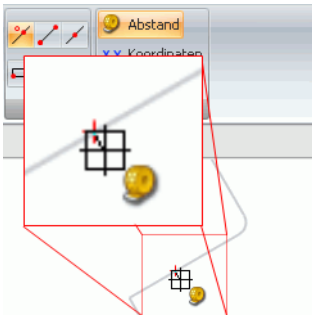


Fig. 2-31: Snapping to the nearest point on an element

End point

This function snaps to the end point of the selected element lying closest to the point you have clicked on.

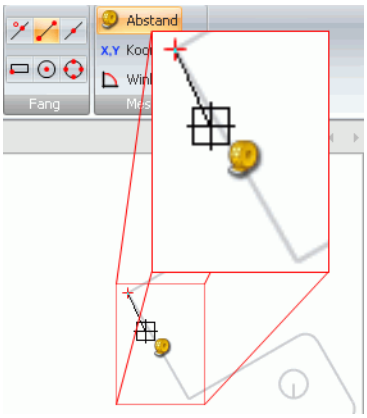


Fig. 2-32: Snapping to the end point of an element

Midpoint

This function snaps to the midpoint of the selected element.

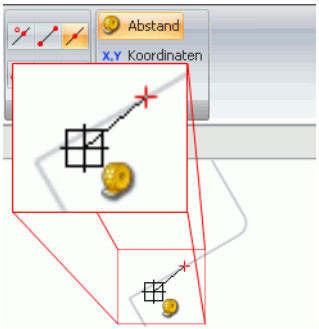


Fig. 2-33: Snapping to the midpoint of an element

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Center of gravity

This function snaps to the center of gravity of a contour. The function can also be used on inside contours.



To snap to the center of gravity of a contour after selecting the desired measurement or drawing function, move the cursor over the part contour (**A**) and confirm the displayed snap point (**B**) with the left mouse button.

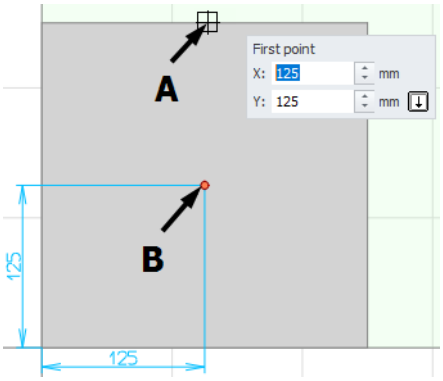


Fig. 2-34: Snapping to the center of gravity of a rectangle.

Quadrant

This function snaps to a point that lies precisely on the quadrant of a circle or arc (0, 90, 180, 270 degrees).



It always picks the quadrant point that lies closest to the picked point on the segment. For arc elements, you can only select points that are actually on the arc segment.

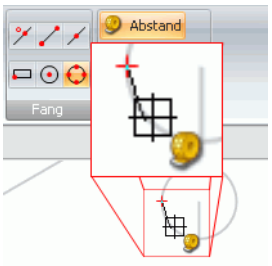


Fig. 2-35: Snapping to the quadrant of a circle

Note: This function can only be applied to circles or arc elements.

Origin

This function snaps to the origin of the selected object. This function is used in the Part Editor and Part Nester modules for various objects.

Note: The origin is shown as a red cross.

Part Editor

The origin of the text is picked. The origin of the text is defined by the **Text origin** option in the text functions of the **Geometry** menu.

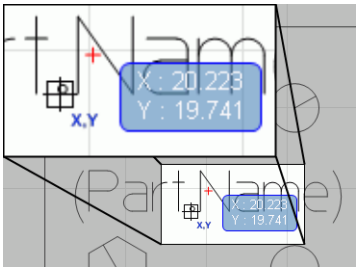


Fig. 2-36: Snapping to the origin of a text with the **X, Y coordinates** measurement function.

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Part Nester

Clicking on the contour of a part in the cutting plan will pick the origin of the part. Use the **Set origin** function in the Part Editor module to change the origin.

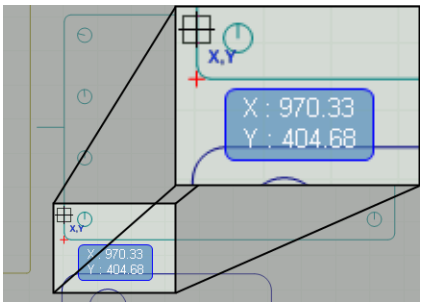


Fig. 2-37: Snapping to the origin of a part with the **X, Y coordinates** measurement function.

Center

This function snaps to the center of an arc or a circle. For very large arcs, the center can be a very long way away from the arc contour.

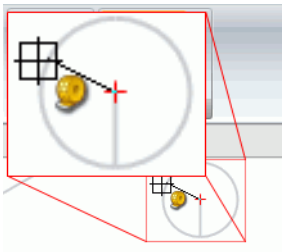


Fig. 2-38: Snapping to the center of a circle

Note: This function can only be applied to circles or arc elements.

Perpendicular

This function snaps to the perpendicular point of the selected element, which is the point lying at right-angles to the contour.



Distance from the center of a circle to a contour

1 = Center of the circle

2 = Selected contour to which you wish to measure the distance

3 = Perpendicular distance measured to the center of the circle

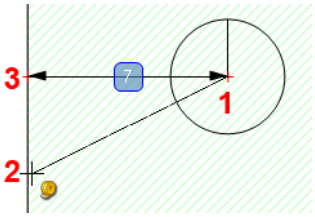


Fig. 2-39: Snapping to the perpendicular point with the Distance function

Grid

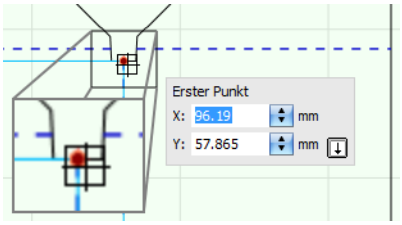
This function snaps to the point on an element that lies closest to the point you have picked on the grid. The grid pitch is set in the Part Editor in **Settings** in the **2D view** tab under the **Snap grid** option.



Intersection point

This functions snaps to the point of intersection of two intersecting or abutting geometries.





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Tangent

This function snaps to tangential points of circular or arc geometries.



Tip: Tangential lines between circles or arcs can be created using the **Tangent** option of the **Line** drawing function.

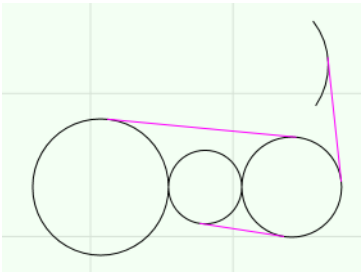


Fig. 2-40: Tangential lines and connections between geometries

2.2.7.5 Snap once

The snap functions in the group can only be used for one-time snapping. After selecting a snap function, the normal snap functions are disabled.

After performing the snap function, the one-time snap function is disabled and the normal snap functions are re-enabled.

2.2.7.6 Measure

This group contains all the functions that can be used to display information about the parts geometry.

Coordinates

This function measures the distance of the X and Y coordinates from the origin of the object.

X,Y

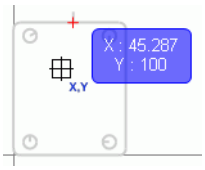


Fig. 2-41: Measuring the coordinates on a part

Distance



This function measures the distance between any two points. Any of the Snap functions can be used to specify the points. For measurement points lying at an angle (**A-B**), the ΔX and ΔY dimensions are also displayed.

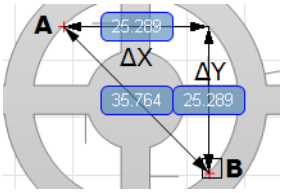


Fig. 2-42: Dimensions shown for measurement points lying at an angle

Measuring the distance using two points

1. Pick the first point on the contour (**1**).
2. Pick the second point on the contour (**2**).

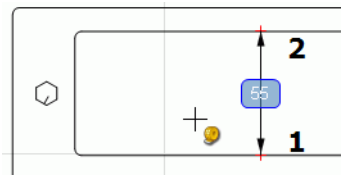


Fig. 2-43: Distance between two lines

Angle



This function can be used to display the angle between two lines  or three points  . The selected lines or points are marked in red. The position of the mouse pointer determines whether the inside or outside angle is displayed.

Note: The angle between two lines can also be measured between two bend lines or the X and Y coordinates.

Tip: The Snap functions can be used to pick the points or lines.

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Displaying the angle between two lines

1. Pick the first line (**1**).
2. Pick the second line (**2**).

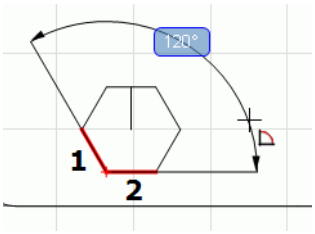


Fig. 2-44: Angle between two lines

Displaying the angle between three points

1. Pick the center point (**1**).
2. Pick the first point of the angle (**2**).
3. Pick the second point of the angle (**3**).

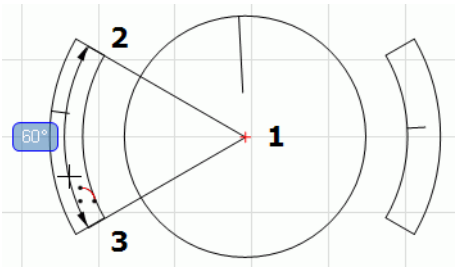


Fig. 2-45: Angle of an arc

3D orthogonal

This function measures the distance between two points in three-dimensional space.

Note: Orthogonal measurement allows you to measure up to four distances.

- If the two points are located on one axis, only one distance is shown.

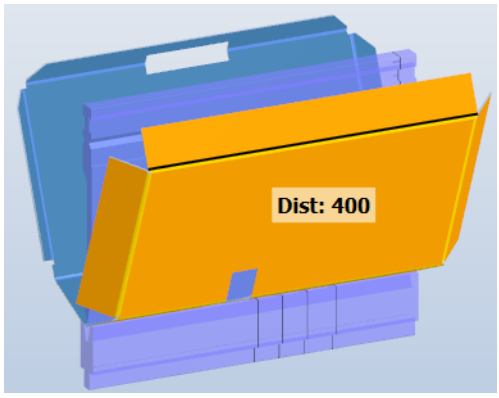


Fig. 2-46: Length on one axis

- If the two points are located on the same plane, only three distances are displayed.

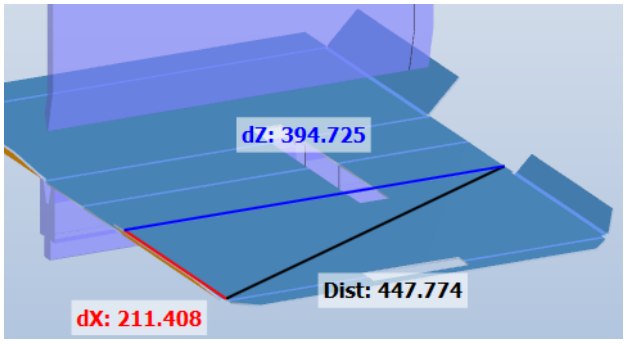


Fig. 2-47: Length on one plane

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- If the two points are located in three dimensions, four distances are shown.

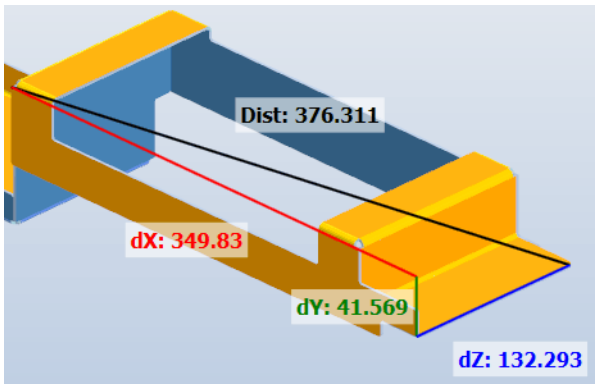


Fig. 2-48: Length in three dimensions

- **Dist:** = Distance between the two points
- **dX:** = Delta X distance
- **dY:** = Delta Y distance
- **dZ:** = Delta Z distance

3D perpendicular

This function measures the vertical distance from a point to a selected surface.



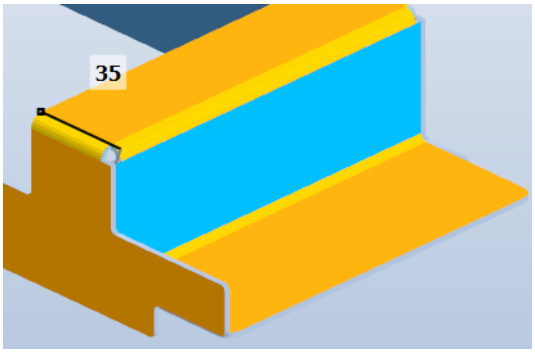


Fig. 2-49: Vertical distance from a surface to a point

3D flange length



This function displays the length of a flange in 3D view. To display the length of the flange, select the **3D flange length** function and the appropriate flange.

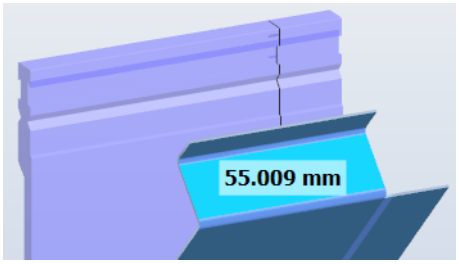


Fig. 2-50: Flange length display in 3D view

If bend lines are situated at right angles to each other, the dimension of the two side lengths is shown (**A**).

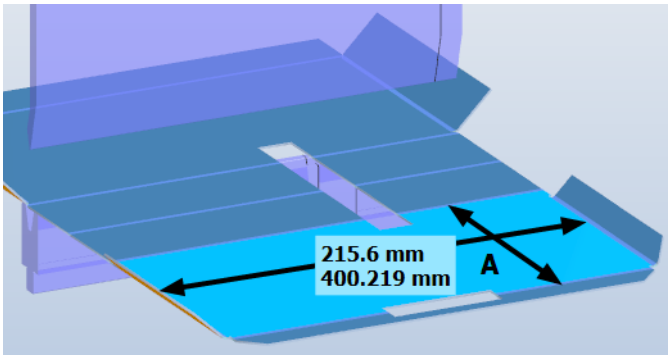


Fig. 2-51: Flange length display for three bend lines

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2.2.8 Shortcuts

In this dialog you can define a keyboard shortcut for all functions of the current module. Functions that have not yet been assigned a keyboard shortcut have the entry **(none)** in the **Shortcut** column.

Assigning a keyboard shortcut

1. Click in the **Shortcut** column and select a keyboard shortcut from the drop-down list.
2. Press **OK** to confirm the selection. The keyboard shortcut for the function only applies in the respective module.

Note:

- Make sure not to assign the same keyboard shortcut to multiple functions.

  Exception: You can assign the same keyboard shortcut to the same functions in multiple menus.
- Press the **Reset** button to reset all customer keyboard shortcuts.

Tip: To only display all functions that have not yet been assigned a keyboard shortcut, enter **(none)** in the **Shortcut** column's filter row.

2.2.9 Functions in tables

The items displayed in a table can be restricted using a filter or search function. This provides a convenient way of displaying the contents of the table based on certain criteria.

Search function (Plant Manager -> Cutting jobs / Bending jobs)

The search function can be used to search for specific jobs in all tables. The search results can be displayed based on certain criteria using the **Display** listbox.

Using the search function

1. Right-click on the **Job** field.
2. Enter the search term in the input field (**A**) of the search function.
3. Select how you want to display results in the **Display** listbox (**B**).
4. Press the **Search** button.

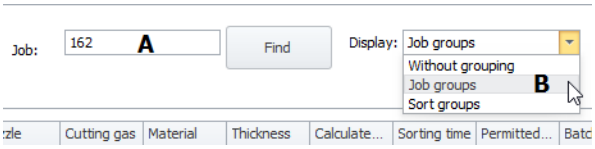


Fig. 2-52: Entering numbers to find jobs.

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Set filter (Plant Manager)

Filters can be set in more than one column at once. Set filters are highlighted in **red** in the filter row.

Using the filter function

1. Right-click in the column header of the table.
2. If the filter row is not displayed, select the **Show auto filter row** function from the shortcut menu. The filter row is displayed below the table header.
3. Click in the filter row (**A**) in the table column that you wish to filter.
4. Enter the character string to be used as the filter. The data contained in the column automatically changes to match the filter entry.
5. To hide the filter row, select the **Hide auto filter row** function from the shortcut menu.

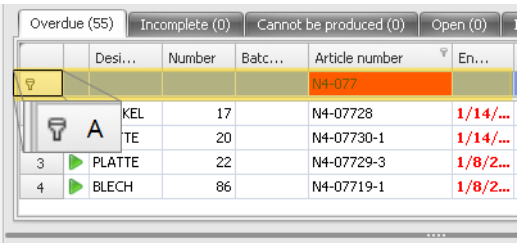


Fig. 2-53: Filter row in a tab

Note: To remove the filter again, delete the characters from the field.

Exporting tables to MS Excel

All the tables in BySoft CAM can be exported to Microsoft Excel so that the user can then reuse the contents of these tables as required.

> **Note:** Images in tables are exported alongside.

Exporting tables

1. Right-click in the column header of the table.
2. Select the **Export to MS Excel** option from the shortcut menu.
3. In the Save dialog box, specify the file name and location in which to save the file, then click **Save** to save the file.

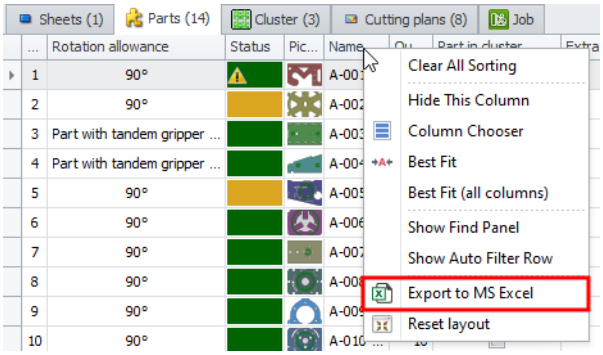


Fig. 2-54: Export function for all tables

	A	B	C	D	E	F
1	Nr.	Bild	Name	Tafeldimension	Durchläufe	Verschnitt
2	1		Sample_flat-01	3 x 1.5 m	1	18.46%
3	2		Sample_flat-02	3 x 1.5 m	2	13.34%
4	3		Sample_flat-03	3 x 1.5 m	1	39.43%
5	4		Sample_flat-04	3 x 1.5 m	1	73.34%

Fig. 2-55: Exported table in Excel

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2.2.10 Main functions

This summary shows you the most important functions for working with the software. Use these functions to help you work efficiently with the application.

Working using the keyboard

Alt key

To select the function or menu you require, press the **Alt** key and then the appropriate letter or letter combination (**A**) that is displayed over the menu or function. You can use this “hotkey” facility to operate all the functions and menus in the software from the keyboard.

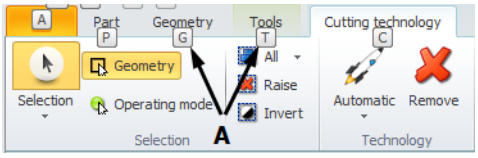


Fig. 2-56: Displaying the letters for function selection from the keyboard

Pressing the **Alt** key again will hide the letters, and you can no longer use the keyboard for selection.

Once you have opened a menu using the keyboard, only those functions available in the menu are accessible from the keyboard. Pressing the **Alt** key twice will let you select again a main function or another menu using the keyboard.

Keyboard shortcuts

Certain functions can only be accessed using a keyboard shortcut. If a function can be opened using a keyboard shortcut, the shortcut is displayed in the Quick Help or in the relevant dialog box for the function.

Multiple selection of objects

The following keyboard and mouse combinations can be used to select objects from a list:

- **Ctrl + A:** *Selects all the objects in the list.*
- **Shift + left-click:** *Selects all the objects between the first and second selection.*
- **Ctrl + left-click:** *Selects various objects from a list.*

Global shortcut menus

These functions are available when working in any module.

Ribbon

To display the shortcut menu, right-click in the Ribbon area.

- **Show quick access toolbar above/below the Ribbon:** *Displays the quick access toolbar above or below the Ribbon.*
- **Minimize the Ribbon:** *The Ribbon is only displayed when you click on a menu. Clicking on another element of the user interface will hide the Ribbon again.*

Closing modules

To display the shortcut menu, right-click on the tab of a module.

The displayed functions can be used to close opened modules. The available Close functions that are displayed depend on the type and number of opened modules.

- **Close all:** *Closes all the opened modules.*
- **Close all the others:** *Closes all the modules except the selected module.*
- **Close all [module name]:** *Closes all the opened modules of the same type.*
- **Close all the others [module name]:** *Closes all the opened modules of the same type except the selected module.*

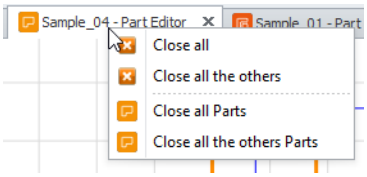


Fig. 2-57: Displaying the Close functions for multiple PartJobs

Attention! If modules are identified that have not been saved yet, a prompt dialog box is displayed. If the dialog box is confirmed with **OK**, all the modules that have not been saved yet are closed. Any changes made in these modules are lost! Select the **Cancel** button and then the **Save all** function to save any modules that have not been saved yet.

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3 Module: Start page



This module displays the last opened objects of the various modules in a quick-launch list. You can define the number of objects to display in the settings *(See: ›Settings‹ Page 86)*.

Moving the mouse over the object's name displays the object in a preview graphic on the right. The main object details are also shown below the preview.

To open the object, click on the appropriate object name in the list. If the object is no longer available, the corresponding entry is removed.

Note: This module is opened automatically after launching the software.

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View

The objects can be displayed in Start View in two different ways via a context menu.

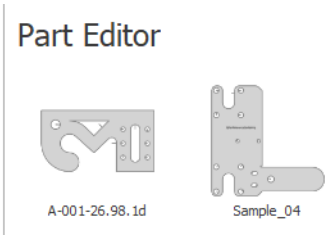
- **Grid view**

  The objects are displayed one below the other for the respective module.



- **Tile view**

  The objects are displayed as a graphic in a tile for the respective module.



Quick search

Click on the input box next to the magnifying glass icon to enter the name of the object you are looking for. The objects retrieved are displayed dynamically based on the string entered below the input box.

Tip: First the program looks for objects that begin with this string and then for objects containing the string in their name. In databases with a very large number of objects, this may cause a delay in displaying objects containing the string.

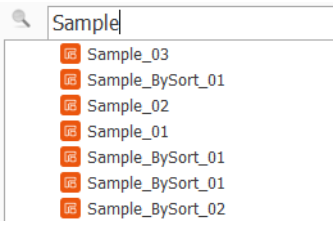


Fig. 3-1: Number of objects found

Tip: Use the up and down arrows to quickly select the appropriate object in the object display. Press **Enter** to open the highlighted file.

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3.1 Settings



How do I open this dialog box?

Select: **Start page**

In this dialog you can define view settings for the user interface in the **Start page** module.



Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

Number of objects

- **Quantity**

  Sets the maximum number of objects displayed on the **Start page**.

- **Mode**

  Determines in which mode the specified number of objects are displayed on the **Start page**.

  - **Total number of objects displayed**

    The specified maximum number of objects for all modules is displayed.

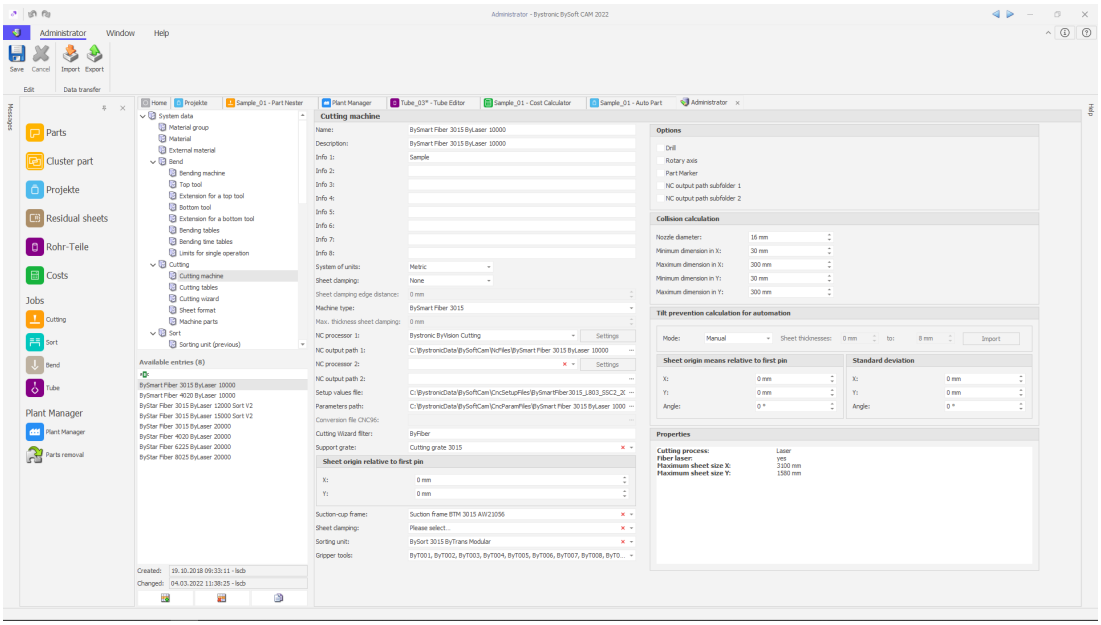
  - **Number of objects per module**

    The specified maximum number of objects per module is displayed.

4 Module: Administrator

How do I open this tab?

Select: **File functions**



The Administrator is used for managing the database, materials, sheet formats, cutting machines and bending machines and the Cutting Wizard.

The left-hand area of the module displays the tree structure containing the configurable objects. Clicking on an object displays the settings for this object on the right-hand side. The **Available entries** area displays all the entries that exist for the given object.

Note: If a modified object has not been saved yet, a dialog box for saving the change appears as soon as another object or module is selected.

Tip: Click on an area of the header to call up the **Shortcuts** function in the context menu. The **Shortcuts** dialog can be used to define keyboard shortcuts for all functions of the current module.

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4.1 Functions

Where can I find these functions?

Select: **Modules > Administration > Administrator**

These functions can be used to edit the objects in the Administrator. They can be launched using the relevant icon in the menu bar or in the relevant areas within the Administrator.

General functions for system data

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

4.1.1 Import



Where do I find the function?

Select the group: **Modules > Administration > Administrator > Data transfer**

This function can be used to import additional entries from an XML file into the **System data** and **Report templates** objects. The import path is saved for the particular object type.



Note: Any errors that occur when importing the objects are displayed in the **Messages** dialog box.

Importing entries

1. In the **System data** or **Report templates** area, click in the form into which you wish to import the relevant objects.
2. Press the **Import** button.
3. In the **Import folder** field, select the folder containing the objects you wish to import.
4. Enable the option **Overwrite objects of same name** if you want to overwrite objects that have the same name.
5. In the **Select objects** window, select the objects you wish to import.
6. Press **OK** to confirm the selection. The objects are imported and displayed in the **Available entries** field.

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4.1.2 Export



Where do I find the function?

Select the group: **Modules > Administration > Administrator > Data transfer**

This function can be used to export the entries for the selected **System data** and **Report templates** objects into an XML file or all system data into a ZIP file. The export path is saved for the particular object type.



Note: Any errors that occur when exporting the objects are displayed in the **Messages** dialog box.

Export individual entries

1. In the **System data** or **Report templates** area, click in the form into which you wish to import the relevant objects.
2. Press the **Export** button.
3. In the **Export folder** field, select the folder into which the objects will be exported.
4. Enable the option **Overwrite objects of same name** if you want to overwrite objects that have the same name.
5. Under **Selected objects**, select the objects that you wish to export.
6. Press **OK** to confirm the selection. The objects are exported into the folder.

Export all system data

1. Select the top entry **System data**.
2. Press the **Export** button.
3. In the **Select Folder** dialog, select the export folder to which the data should be exported.
4. Press the **Select folder** button to confirm the selection. The system data is exported in a ZIP file to the specified folder.

4.1.3 Save



Where do I find the function?

Select the group: **Modules > Administration > Administrator > Edit**

Saves any changes in the active object.

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4.1.4 Cancel



Where do I find the function?

Select the group: **Modules > Administration > Administrator > Edit**

Reverses (undoes) any changes made in the active object.

4.2 System data



Where can I find these functions?

Select: **Modules > Administration > Administrator**

The system data is managed in this area. The system data can be grouped into the following two areas.

Production means

These objects reflect the production means available to you and must be configured accordingly. The following objects count as production means:

- Material
- Sheet format
- Cutting machines
- Bending machines

Support tools

These objects are used for support functions used in preparing for production.

The following objects count as support functions:

- Cutting tables
- Bending tables
- Cutting wizard
- Cost tables
- Report templates

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4.2.1 Material group

Where do I find this form?

Select: **Modules > Administration > Administrator > System data**

Material groups can be used to combine materials consisting of the same material but with different properties. Material groups are used in Settings Manager for nesting and export settings.

Materials

This dialog box is used to assign materials to the selected material group.



Attention! Save the added materials before selecting a new material in the **Available entries** pane. Otherwise, the materials you added will be lost.

- **Add**

  Adds the selected material from the **Available materials** column to the **Materials in group** column.

- **Remove**

  Removes the selected material from the **Materials in group** column.

Create new material group

1. Use the  button to add a new material group to the **Available entries** pane.
2. Enter a name for the material group in the **Name** field.
3. Select the materials to assign to the material group in the **Available materials** pane.
4. Press the **Add** button to add the materials to the group.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

What each field does:

- **Name**

  This name is displayed in the **Available entries** pane and in all the selection fields for the material in the software.

  ☞ **Note:** The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g. DIN designation for material). (e.g. DIN designation for material).

- **Info 1–8**

  Extra information about the current object can be entered in these fields.

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4.2.2 Material

Where do I find this form?

Select: **Modules > Administration > Administrator > System data**

This form in the **Available entries** pane displays all the materials available in the software. You can create a new material using the  or  button. *(See: ›Creating a new material‹ Page 1340)*.

When creating cutting plans and bending plans, the software needs to select the appropriate cutting parameters and cutting rule set for the particular material. This means that the correct cutting table must be assigned once to each material in the **Cutting tables** field.

If the material will also be used for bending parts, assign the appropriate bending table to this material in the **Bending tables** field. If you do not assign a cutting table or bending table to the material, the default parameters are used for creating cutting and bending plans.



Note: Remove unused materials from the materials list so that only those materials still available in production can be used to create cutting and bending plans.



Tip: If several materials can be cut with the same cutting parameters, assign the same cutting table to these materials.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

What each field does:

- **Name**

  This name is displayed in the **Available entries** pane and in all the selection fields for the material in the software.

  ☞ **Note:** The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g. DIN designation for material).

- **Info 1–8**

  Extra information about the current object can be entered in these fields.

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

  The selected system of units applies to all thicknesses of the material.

- **Density**

  This field displays the density of the material. This value is used to calculate the weight of parts and sheets.

- **Bending tables**

  In this field you assign to the selected material the bending table that contains the settings for the relevant bending parameters.

- **Cutting tables**

  In this field you assign to the selected material the cutting table that contains the settings for the available material thicknesses and the filters for the cutting parameters and cutting wizard.

- **Magnetic**

  This checkbox can be used to specify whether the material has magnetic properties. This property is used for sorting parts with magnetic grippers.

- **Material group**

  The material can be assigned to an existing material group by using the selection box. New material groups can be created in the **Material group** form. The material groups are used in Settings Manager for the nesting and export settings.

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4.2.3 Bending



Where can I find these functions?

Select: **Modules > Administration > Administrator**

System data for press brakes is managed in this area.

Bending machine

Where do I find this form?

Select: **Modules > Administration > Administrator > System Data > Bending**

This form displays the definitions for the selected bending machines. Only the **NC processor** for the machine program and the **NC output folder** for the storage location of the bending plans can be set individually in this form.

- The Bendpart format is output for the NC processor **Bystronic ByVision Bending BendPart**. This format exports every bending part as separate bending job for ByVision Bending. You can define whether to compress the export file or not in the **Settings** dialog.
- For the NC processor **Bystronic ByVision Bending BendJob/BendPart**, the program outputs two different export formats for bending jobs depending on the particular application.
  - The export function in Tool Setup Optimizer outputs the BendJob format. All bending jobs from Tool Setup Optimizer are exported in the right order to a file for ByVision Bending.
  - The export in Part Editor outputs the **Bendpart** format. This format exports every bending part as separate bending job for ByVision Bending.

☞ **Note:** The form displays the entries for the machine that is currently selected in the **Available entries** pane.

Top tool

Where do I find this form?

Select: **Modules > Administration > Administrator > System Data > Bending**

This form displays all the valid options for the selected top tool. The data in this form is for information purposes only and cannot be changed.

☞ **Note:** The form displays the entries for the top tools that are currently selected in the **Available entries** pane.

Bottom tool

Where do I find this form?

Select: **Modules > Administration > Administrator > System Data > Bending**

This form displays all the valid options for the selected bottom tool. The data in this form is for information purposes only and cannot be changed.

☞ **Note:** The form displays the entries for the bottom tools that are currently selected in the **Available entries** pane.

Bending tables

Where do I find this form?

Select: **Modules > Administration > Administrator > System Data > Bending**

This form contains data that is needed in order to create bending plans for the material concerned. The information in the bending tables is preset and cannot be changed.

The upper section of the view contains all the general information on the selected material in the bending table. The two table views contain details on the various sheet thicknesses, radii and bend angles.

- **Grain direction compensation**

  Defines the default angle for grain direction compensation for all bend lines of a part. This setting is applied by default to all bending parts if the option is enabled *(See: ›Grain direction compensation (whole part)‹ Page 630)*.

Bending-time table

Where do I find this form?

Select: **Modules > Administration > Administrator > System Data > Bending**

This form contains all bending-time tables. These tables are used to calculate bending time in Part Editor *(See: ›Calculate bending time‹ Page 652)*.

You can import new bending-time tables into Administrator using the **Import** function.

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Note: You can only create and edit tables in the ByVision Bending Administrator.

Limits for individual operation

Where do I find this form?

Select: **Modules > Administration > Administrator > System Data > Bending**

This form contains all specific details for automatically determining the limits for individual operation.

If a bending part exceeds a value in this table, the number of operators is set to 2. The number of operators is displayed in bending part properties in Part Editor *(See: ›Bending‹ Page 381)*.

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4.2.4 Cutting



Where can I find these functions?

Select: **Modules > Administration > Administrator**

System data for laser cutting machines is managed in this area.

Cutting machine

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Cutting**

All the machines available for production are defined in this form. Two laser and water jet cutting machines are already entered as standard models. You can create new machines using the  button. *(See: ›Creating a new cutting machine‹ Page 1342)*.



Note: The form displays the entries for the machine that is currently selected in the **Available entries** pane.



Tip: For new entries, use the standard machines and then adjust the settings to your requirements. Then delete all unused machines from the list. This stops anyone accidentally assigning parts or plans to a machine that is not actually available for production.

What each field does:

- **Name**

  This name is displayed in the **Available entries** pane and in all the selection fields for the machine throughout the software.

  Note: The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g., laser power or machine format).

- **Info 1–8**

Extra information about the current object can be entered in these fields.

- **Machine type**

  The machine type that applies to the selected machine can be chosen in this field. The machine type determines what options are available, the cutting process and the sheet format for the machine.

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- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

- **Sheet clamping**

  In this field you specify on which side the sheet clamping mechanism is located. For Byspeed machines, the **Left** attribute must always be selected. For sheet clamping systems specific to your company used on water jet cutting machines, all 4 sides can be selected.

- **Sheet clamping edge distance**

  This is where you set the margin at the sheet edge in which parts are not allowed to be nested. The edge distance only needs to be specified for Byspeed machines running the **ByVision P8102** controller version or earlier. This is because automatic retraction of the clamping jaws is not supported in these early versions.

  A value of **zero** means that parts can be nested over the entire sheet.

- **Max. thickness sheet clamping**

  In this field you specify the maximum thickness up to which the sheets are still clamped.

- **Support grate offset X/Y**

  These fields are used to define the position of the first pin on the first support grate (**A**) relative to the sheet origin (**B**).

  The alignment is used to calculate tilt optimization so that the support points of the support grate in BySoft CAM match the actual support points relative to the sheet origin on the machine.

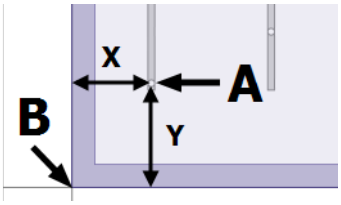


Fig. 4-1: Support grate offset

- **Repositioning dividing position 1/2**

  The dividing position defines at what position a division is made in the cutting plan for repositioning machines.

- **Repositioning shift value 1/2**

  The shift value defines by what distance from the absolute origin the cutting plans are repositioned to the left on repositioning machines.

- **NC processor 1**

  In this field you define the cutting processor for generating the cutting plan. Select the processor that corresponds to the controller type on your machine.

  For Bystronic laser and water jet cutting machines, use the following cutting processors:

  - Bystronic Astes 4 (BVC) *(See: ›Bystronic Astes4‹ Page 105)*
  - Bystronic BZT Bvc
  - Bystronic ByVision (NCP)
  - Bystronic ByVision (deprecated)
  - Bystronic CNC 96
  - Bystronic CNC 96 (deprecated)
  - Bystronic FMG-Sort *(See: ›Bystronic FMG-Sort‹ Page 105)*
  - Bystronic ByVision Cutting (BVC)
  - Bystronic SlotMachine Bvc
  - Autocad DXF

  The cutting processor is used to output DXF files for import into an external postprocessor.

- **NC output path 1**

  This field is used to enter the path to the output folder for Bystronic cutting plans.

- **NC processor 2**

  This field is used to define the post-processor for generating cutting or sorting plans for sorting systems or external systems.

  For sorting and external systems, use the following cutting processors:

  - **Bystronic FMG-Sort**

    The cutting processor is required when connecting a Bystronic FMG sorting system to a Bystronic laser cutting machine for cutting plans. You can define how the postprocessor outputs cutting plans in NCP or BVC format in the **NC processor** box.

  - **Bystronic Astes4**

    The cutting processor is required to connect an Astes4 sorting system to a Bystronic laser cutting machine.

    - **Import folder**

      Enter the directory in which to export an XML and the associated DXF file in this field.

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- **Cutting grate type**
  Enter the material of which the cutting grate consists in this type field based on Astes4 specifications (e.g. Copper, Steel, Stainless steel)
- **Advanced material output**
  Disabled: The material type (e.g. DC01) is entered in the <Material> field in the XML file (default setting).
  Enabled: A string consisting of material type, material thickness, X dimension and Y dimension is entered in the <Material> field of the XML file (e.g. DC01_3_3000_1500)
- **Subfolder per plan**
  A subdirectory is created for each cutting plan.
- **Replace material with materialnumber**
  Replaces the value of the material in the XML file with the value of the material number.

- **NC output path 2**
  This field is used to enter the path to the output folder for external system cutting or sorting plans.
- **Setup values**
  In this field you define the path to the Setup values file for the machine. These details are needed to calculate the machining time.
- **Parameter path**
  In this field you define the path to the cutting parameters that apply to this machine. These details are needed to calculate the machining time.
- **Conversion file CNC96**
  In this field you define the path to the conversion file for CNC96 parameters. Using the file name, select the conversion file that applies to your particular machine *(See: ›Creating a cutting machine fitted with CNC96 controller‹ Page 1343)*.

☞ **Note:** The conversion file is needed for the cutting-time calculation, because the conventions are not compatible between the CNC96 and CNC02 controller.

Cutting Wizard filter

This field contains a filter for each standard machine. The filter is used in several of the software modules to preselect the Cutting Wizard. The filter consists of the machine name and laser power, which appear in the name of the Cutting Wizard (**1**).

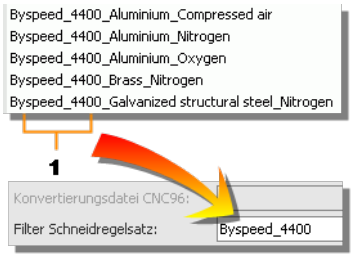


Fig. 4-2: Specifying a filter from the name of a Cutting Wizard (Byspeed_4400).

Using the filter:

- Part Editor = **Cutting technology** > **Automatic**
- Part Nester = **Export** >**Export settings** > **Preprocessor** > **Cutting Wizard**



Fig. 4-3: Cutting Wizard display filtered by “machine”

Support grate, Suction cups, Sheet clamping

The selection box shows all CAD geometries for the support grate, the suction-cup layout, and the sheet clamping. You must assign the appropriate geometry to the selected field.

Display of the appropriate machine components can be enabled in the special display in Part Nester *(See: ›Special view‹ Page 1182)*.

Sorting unit

You can assign a sorting unit to the current machine in this field.

Gripper tools

You can assign the existing gripper tools to the machine in this selection box. When exporting sorting plans, only parts with the tools selected here are exported in the sorting plan and sorted on the machine.

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Options

In this area you can enable some additional machine options.

Drill

Enable this option if a drill is fitted to a water jet cutting machine. If this option is enabled, the **Pilot drill** option can be used in the Cutting technology.

Rotary axis

Enable this option if the selected machine has a rotary axis. This means that the machine can also be used for cutting tubular parts. If this option is enabled, the **Rotary axis** tab is displayed.

Part Marker

- If this option is enabled, the labels in the cutting plan are output as a print job for Part Marker in the cutting plan.
- If this option is disabled, the labels in the cutting plan are output as an engraving in the cutting plan based on the **Parts ID** settings in the post-processor export settings.

☞ **Note:** The **Print part labels** option must be enabled in the post-processor export settings for exporting labels *(See: ›Print part labels‹ Page 1175)*.

NC output path 1 subfolder

Activate this option to use a subfolder with the name of the exported job for NC output path 1. All cutting plans in the job are saved to this subfolder.

NC output path 2 subfolder

Activate this option to use a subfolder with the name of the exported job for NC output path 2. All output files of postprocessor 2 are saved to this subfolder.

Collision calculation

The figures in the boxes are used for collision calculation for positioning paths in a cutting plan. If a positioning path runs across a cut contour whose area is inside minimum and maximum dimensions, this contour is flagged as a **Collision risk** in the part check *(See: ›Risk of collision‹ Page 397)*.

Nozzle diameter

Indicates the external diameter of the nozzle on the machine's cutting head.

This option defines the width of the corridor (**A**) along the positioning path (**B**) of the cutting head. The program checks whether there are any part contours (**C**) inside this corridor.

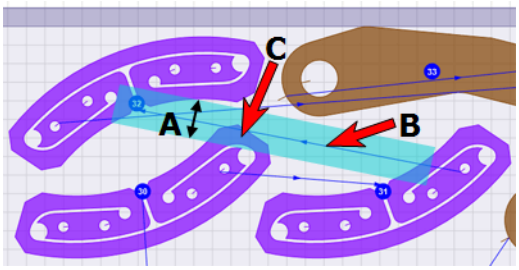


Fig. 4-4: Width of check corridor on cutting-head positioning path

Min./Max. dimension X

If there are any parts inside this corridor in the X direction, the cutting head is raised when it moves over the cut parts.

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Min./Max. dimension Y

If there are any parts inside this corridor in the Y direction, the cutting head is raised when it moves over the cut parts.

Tilt optimization calculation

The settings are used to calculate tilt optimization in Part Nester in conjunction with an automation system.

Mode

- **Manual**

  The average sheet origin and a possible angle of skew must be entered in the **Mean values** section.

- **History data**

  The average sheet origin and a possible angle of skew are calculated based on the data in an XML file. The XML file contains the identified positions of cut sheets that were cut on the selected machine. The XML file can be imported for the machine by pressing the **Import** button.

  The two fields **Sheet thickness** and **to** are used to define an import filter for a certain thickness range.

> ☞ **Note:** Thicknesses greater than **8 mm** are irrelevant for tilt optimization since parts with these thicknesses no longer tilt.

Mean values

- **X/Y**

  Defines the position of the sheet origin (**B**) relative to the first pin (**A**) on the first support grate.

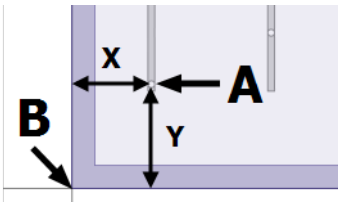


Fig. 4-5: Support grate offset X/Y

- **Angle**

  Defines the sheet's angle of skew.

Standard deviation

- **X/Y**

  Calculated standard deviation of the position of the sheet origin (**B**) relative to the first pin (**A**) on the first support grate based on the imported XML data.

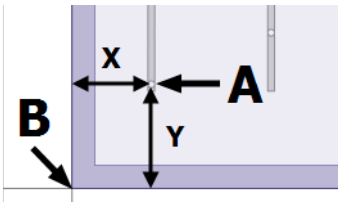


Fig. 4-6: Support grate offset X/Y

- **Angle**

  Calculated standard deviation of the sheet's angle of skew based on the imported XML data.

Properties

This area displays additional information about the selected **Machine type**. This information provides a summary of the cutting process, the maximum sheet size and extra functions of the machine that are supported.

- For laser machines, the machine's laser technology is also displayed.
- For water jet cutting machines, this area also shows the number, the separation and the arrangement of the cutting heads for multihead machining.

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Cutting tables

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Cutting**

This form contains data that is needed in order to create cutting plans for the material concerned. You can create a new cutting table using the  or  button. *(See: ›Creating a new cutting table‹ Page 1341)*.

You can create as many thicknesses as you like for each cutting table. You can create a new thickness in the **Cutting tables** area using the  or  button.



Tip: Use the  button to create new thicknesses because then you only need to change the value for the thickness each time.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

What each field does:

- **Name**

  This name is displayed in the **Available entries** pane and in the **Material** form in the **Cutting tables** selection field.

  Note: The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g., laser power or machine format).

Info 1–8

Extra information about the current object can be entered in these fields.

Filter for cutting parameters

In this field you can specify a filter for each material in order to select the parameters. Filters for all the standard parameters are already included. The filter consists of the material name, which appears in the name of the parameter set. The filter is used in the Part Editor and Part Nester modules to preselect the cutting parameters for calculating the machining time.

Note: The filter applies to all thicknesses in the table.

Cutting Wizard filter

In this field you can specify a filter for each Cutting Wizard. Filters are already provided for all Cutting Wizards supplied with the software. The filter is used in several of the software modules to preselect the Cutting Wizard. The filter consists of the material name, which appears in the name of the Cutting Wizard.

Example:

Filter entered = Structural steel

Name of Cutting Wizard = Byspeed_4400_**Structural steel**_Oxygen

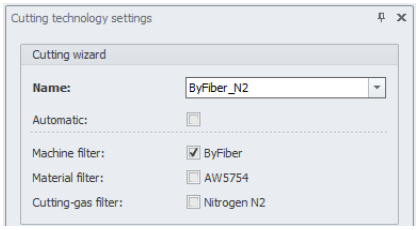


Fig. 4-7: Cutting Wizard display filtered by machine and material

Using the filter:

- Part Editor = **Cutting technology** > **Automatic**
- Part Nester = **Export** >**Export settings** > **Preprocessor** > **Cutting Wizard**

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What each column means:

- **Thickness**

  This column displays all the thicknesses available for the material. To add a new thickness for the material, click the  icon.

- **Parts protective distance**

  In this column, the value for the parts protective distance is entered for the **Laser** and **Water** technologies. This minimum separation is as close as the parts will be nested against each other.

- **Sheet edge**

  In this column, the value for the clearance around the sheet edge is entered for the **Laser** and **Water** technologies. This minimum distance is as close as the parts will be nested against the edge of a sheet.

- **Filter for cutting parameters**

  In this field you can specify a filter for each thickness of the material in order to help select the parameters. The filter consists of the material name and relevant thickness, which appear in the name of the respective parameter set.

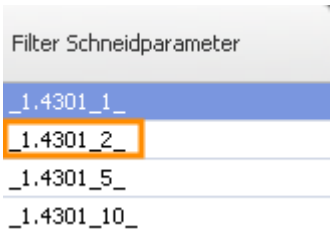


Fig. 4-8: Filter entry per thickness for stainless steel

Example:

- Filter entered for the material **Stainless steel** 2mm = 1.4301_2_
- Material name in the parameter name = SPEED3015_8105_ 4400_**1.4301_2_**5.0_N2.PAR

The filter in this column has a higher priority than the filter in the input field of the table and is therefore used as a finer filter on the parameters.

Using the filter:

- Part Editor / Part Nester = **Machining time** > **Cutting parameters**

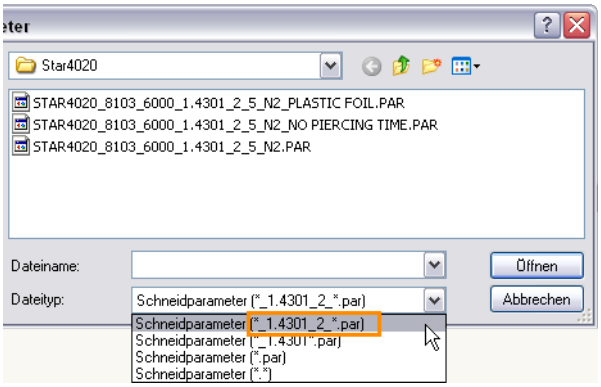


Fig. 4-9: Display of parameter sets in the **Cutting parameters** dialog box filtered by machine, material and thickness.

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Cutting Wizard

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Cutting**

This form in the **Available entries** pane displays all the Cutting Wizards contained in the system. You can create a new Cutting Wizard using the  or  button. Use the two arrow buttons to move rules up or down within the rule table.



Fig. 4-10: Arrow buttons for moving rules

Using Cutting Wizards

Cutting Wizards are used to assign technologies automatically to parts or cutting plans. Cutting rules can be created for the following geometry elements:

- Cutting contours
- Cutting lead-ins and lead-outs
- Cutting corners

Using Cutting Wizards in your work has the following advantages:

- Speeds up programming
- Reduces potential errors
- Requires less expert knowledge
- Optimizes the cutting process
- Cutting rules are easy to create and edit

Notes for production

Most companies work with their own particular production range of materials, thicknesses and parts.

In order to improve cutting quality and to optimize your cutting plans, you need to incorporate your cutting experience on the machine in the Cutting Wizards. If you consistently add your machining experience to the Cutting Wizards, you will see a rapid improvement in the quality of cutting plans. After a short period of time, the Cutting Wizards will be optimized for your production range, letting you benefit from the advantages.

In the long term, these steps will speed up production and increase the efficiency of your cutting machine.



Tip: If different people are responsible for programming cutting plans and for operating the machine, it will be helpful to both parties if each person experiences the other person's area of work for a while. This will help their understanding of the tasks involved.

Creating a new Cutting Wizard

1. Click the  button in the **Available entries** pane.
2. Enter a name for the new Cutting Wizard in the **Name** field.



Note: Use the following name convention: **Machine name_Laser power_Material_Cutting gas** (cutting gas only for laser machines)

3. Enter a more specific description for the Cutting Wizard in the **Description** field (optional).
4. Define the system of units and the cutting process for the Cutting Wizard.
5. Click the  button in the **Thickness** area.
6. Enter a thickness for the Cutting Wizard and press **OK** to confirm.
7. Click the  button in the **Rules for lead-ins/lead-outs** area.
8. Define the rules for the lead-in/lead-out type for the current thickness.
9. Click the  button in the **Cutting rules** area.
10. Define the rules for cutting the current thickness.
11. Repeat steps **5** to **9** until you have recorded the rules for all thicknesses of the material.
12. Save the settings.

Note: You can create as many cutting rules as you like for each Cutting Wizard. If the ranges specified in the table overlap each other, the rule that applies first in the relevant area will be used.

Cutting Wizards are provided for the most common machine and material combinations. You can, however, create your own Cutting Wizards or modify existing ones. The Cutting Wizards are displayed in the **Available entries** pane.

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What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

What each field does:

- **Name**

  This name is displayed in the **Available entries** pane and in all the selection fields for the machine throughout the software.

  ☞ **Note:** The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g., laser power or machine format).

- **Info 1–8**

  Extra information about the current object can be entered in these fields.

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

  The selected system of units applies to all thicknesses of the material.

- **Cutting process**

  In this field you specify the applicable cutting process (laser/water) for the object.

Thickness

This area displays all the thicknesses of the material that are defined for the cutting rules.

Note: If you cut a sheet with a thickness not defined in the table, the options for the next larger thickness in the table are used.

Example:

- Material thickness used for cutting: **22 mm**
- Available material thicknesses: **< 20 mm, 25 mm >**
- Rule used in the Cutting Wizard: **25 mm**

Attention! If a thickness is selected in the Part Editor or Part Nester module that is greater than the largest thickness defined in the table, you cannot use the Cutting Wizard. You cannot cut this raw material with the existing parameters. You must create a new cutting rule containing suitable options for this material thickness.

Rules for lead-ins/lead-outs

This area contains the settings for the lead-in and lead-out of a contour. These settings specify the conditions under which the options are meant to be used.

What each column means:

- **Contour type**

  Defines the contour type for which the cutting rule applies.

  #### Attribute

  - **As required:** The settings for the rule for the selected thickness apply to all contours.
  - **Inside contour:** The settings for the rule for the selected thickness apply only to the inner contour.
  - **Outer contour:** The settings for the rule for the selected thickness apply only to the outer contour.

- **Contour shape**

  Defines the contour shape for which the cutting rule applies.

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Attribute

- **As required:** The rule settings for the selected thickness apply to all contour shapes.
- **Circle:** The rule settings for the selected thickness apply only to the circular contour.
- **Rectangle:** The rule settings for the selected thickness apply only to the rectangular contour.

- **greater than hole diameter**

  Specifies the lower limit for calculating the area.

- up to and including hole diameter

  Defines the upper limit for calculating an area. This area may correspond to a circle, triangle, square or polygon. On the basis of the hole diameter, the program determines up to what area the rule for cutting the lead-in shall apply.

☞ **Note:** If a value of **0** is entered in the field **greater than hole diameter** and **up to and including hole diameter**, this rule is applied to all open contours. The value * means **infinite**.

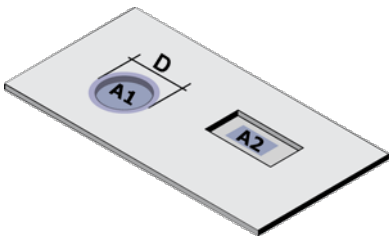
Example: Use pulsed cutting for lead-in

Pulsed up to and including hole diameter

Hole diameter D = 2mm

Area **A** = $D/2^2$ x Pi = $1mm^2$ x 3.1415 = **$3.14mm^2$**

- If the area of the inside contour **A1** is smaller than the area of a circle of diameter **D**, the lead-in is cut in pulsed mode.
- If the area of the inside contour **A2** is greater than the area of a circle of diameter **D**, the lead-in is cut in **CW** mode.



- **Lead-in type**

  Determines the lead-in type for the cutting rule.

- **Lead-out type**

  Determines the lead-out type for the cutting rule.

> **Note:** These options are usually only used for water jet cutting. Usually no lead-out cut is used in laser cutting.

- **Operating mode (lead-in and lead-out)**

  Defines which operating mode is used to cut the lead-in and lead-out.

Attribute

- **Default:** The parameters for machining the material are assigned by the preprocessor during export of the cutting plan, but not before this stage.
- **Cutting:** The CW parameters are used for machining the material.
- **Operating mode 2:** This operating mode is used to make a quality cut (water jet cutting only).
- **Pulse:** The pulsed parameters are used for machining the material. Pulsed operation can improve the machining quality for certain materials or for tight contours by reducing the amount of heat input to the material (laser cutting only).
- **Engraving:** The cutting parameters for engraving the material are used. Closed inner contours that have this machining mode are no longer displayed as a cut-out in the Part Editor.
- **Process macros:** The relevant process macro is used for machining the material.
  - Laser process macro
  - Water process macro

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Cutting rules

In this section you can specify the settings for cutting a contour and the conditions for assigning process macros.

What each column means:

- **Contour type**

  Defines the contour type for which the cutting rule applies.

  #### Attribute

  - **As required:** The settings for the rule for the selected thickness apply to all contours.
  - **Inside contour:** The settings for the rule for the selected thickness apply only to the inner contour.
  - **Outer contour:** The settings for the rule for the selected thickness apply only to the outer contour.

- **greater than hole diameter**

  Specifies the lower limit for calculating the area.

- **Tag**

  Defines the tag for which the cutting rule applies.

  #### Attribute

  - **Center punch mark:** The cutting rule only applies to center punch marks.
  - **Pre-piercing:** The cutting rule only applies to pre-piercing.
  - **Round off:** The cutting rule only applies to corner roundings.
  - **Looping:** The cutting rule only applies to corner loops.
  - **Scrap cut-down:** The cutting rule only applies to scrap cut-down.
  - **Scanning:** The cutting rule only applies to the scanning cutting process.

- **Piercing mode**

  Defines the piercing mode for the contour.

  #### Attribute

  - **Default:** The piercing mode is assigned by the preprocessor when the cutting plan is exported, but not before this stage.
  - **Normal:** Piercing is performed at full power.

- **Without time:** The piercing time saved in the cutting parameters on the machine is ignored.
- **Pulsed:** Piercing is performed with pulsed power using the piercing parameters specified in the cutting parameters (laser only).
- **Drilled:** Pilot drilling is performed using the drill parameters specified in the cutting parameters (water only).

☞ **Note:** Pulsed piercing produces smaller piercing holes than normal piercing.

- **up to and including hole diameter**

Defines the upper limit for calculating an area. This area may correspond to a circle, triangle, square or polygon. On the basis of the hole diameter, the program determines up to what area the rule for cutting the contour shall apply.

☞ **Note:** If the value **0** is entered in the field **from hole diameter** and **up to and including hole diameter**, this rule is applied to all open contours. The value * means **infinite**.

Example: Process macros

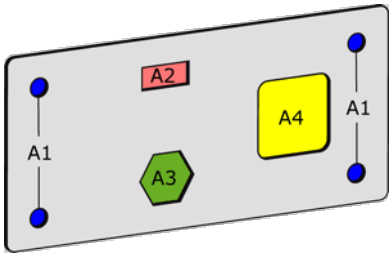
Process macro 1

1. Hole diameter entered for process macro 1 = **10mm**. This corresponds to an area of **15.7mm^2**.
2. The area of the contour **A1** to be cut is **9.4 mm^2**. This corresponds to a hole diameter of **6mm**.
3. Because the hole diameter of **A1** is smaller than the setting of **10mm**, the contour is cut with process macro 1.

Process macro 2

1. Hole diameter entered for process macro 2 = **30mm**. This corresponds to an area of **47.1 mm^2**.
2. The area of the contour **A2** to be cut is **26.7 mm^2**. This corresponds to a hole diameter of **17mm**.
3. Because the hole diameter of **A2** is smaller than the setting of **30mm** and greater than the value of **10mm** set in process macro 1, the contour is cut with process macro 2 (area **A = (L/2)^2 x pi**).

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- **Operating mode**

  Defines which operating mode is used to cut the contour.

  In ByVision, the current operating mode is displayed during cutting in the graphics window in the **WORK** view. This means that the machine operator can always see which cutting parameter is currently active.

Attribute

- **Default:** The parameters for machining the material are assigned by the preprocessor during export of the cutting plan, but not before this stage.
- **Cutting:** The CW parameters are used for machining the material.
- **Operating mode 2:** This operating mode is used to make a quality cut (water jet cutting only).
- **Pulse:** The pulsed parameters are used for machining the material. Pulsed operation can improve the machining quality for certain materials or for tight contours by reducing the amount of heat input to the material (laser cutting only).
- **Engraving:** The cutting parameters for engraving the material are used. Closed inner contours that have this machining mode are no longer displayed as a cut-out in the Part Editor.
- **Process macros:** The relevant process macro is used for machining the material.
  - Laser process macro
  - Water process macro

Engrave operating mode

In this area, you can define the default operating mode for engravings.

Engrave operating mode

This option is used to specify the operating mode for all engravings. If the **Default** operating mode is selected, the operating mode which is set on the contour of the engraving is used.

Corner mode

The settings for cutting corners are defined in this section. These settings are only used if the option **Use Cutting Wizard** is enabled for the corner mode in the Export settings.

Selection mode

Use this option to select the corner mode. There are various attributes available for cutting corners.

- **Use selection:** The specified corner mode is only applied to those corners whose contours have the **Round off** or **Looping** tag All other corners are cut normally *(See: ›Tag‹ Page 684)*.
- **Round off:** Cuts those corners that lie in the range 5°- 170° with the radius specified here, provided they have no radius in the cutting plan.
- **Looping:** Cuts those corners that lie in the range 15°- 135° with a loop, provided they have no radius in the cutting plan.

Note: If Looping is enabled, corners in inner contours that have no radius are also cut with a loop. To avoid this, assign the inner contour corners a radius of half the kerf width *(See: › Finding the tool radius‹ Page 1360)*.

Tip: This option is particularly useful for very sharp corners such as saw blades.

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Looping length

This option specifies the extent of the loop (**L**) from the corner of the contour.

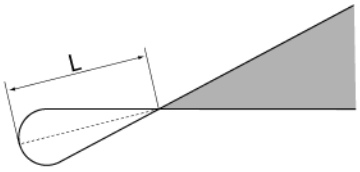


Fig. 4-11: Looping length when cutting sharp corners

Rounding radius for outside corners / inside corners

These options are used for specifying the rounding radii for outside corners (**A**) and inside corners (**B**).

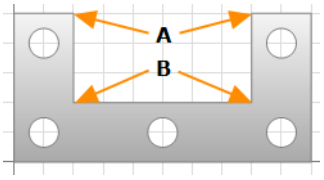


Fig. 4-12: Rounding radii for outside and inside corners

Sheet format

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Cutting**

This form in the **Available entries** pane displays all the sheet formats contained in the system. You can create a sheet format using the  or  button.

You can create as many sheet sizes as you like for each sheet format. You can create a new format size in the **Sheet format** area using the  button.

Note: The sheet format information is used in conjunction with the material for nesting jobs in Part Nester *(See: ›Sheets‹ Page 1191)*.

Creating a new sheet format

1. Click the  button in the **Available entries** pane.
2. Enter a name for the new sheet format in the **Name** field.
3. Enter a more specific description for the sheet format in the **Description** field (optional).
4. Specify the system of units for the sheet format.
5. Click the button  in the **Sheet format** area.
6. Enter a **Length** and **Width** for the format and confirm with **OK**.
7. repeat steps **5** and **6** until you have recorded all the sheet sizes.
8. Save the settings.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

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What each field does:

- **Name**

  This name is displayed in the **Available entries** pane and in all the selection fields for the sheet format.

  ☞ **Note:** The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g., laser power or machine format).

- **Info 1–8**

  Extra information about the current object can be entered in these fields.

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

  The selected system of units applies to all the format sizes.

Add sheet format

How do I open this dialog box?

Select: **Modules > Administration > Administrator > System data > Cost tables > Operations > Operations**

In this dialog box, you can specify the sheet format by entering **Length** and **Width** in the relevant system of units.

Confirming with **OK** enters the sheet format in the **Sheet format** form.

☞ **Note:** Subsequent changes cannot be made to a format once created. To modify a format, you must first delete the format from the form and then recreate it.

Machine parts

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Cutting**

This dialog is used to import geometry data of certain machine parts as a display template into BySoft CAM. Press  or  to create a new machine part *(See: ›Creating a new machine part‹ Page 1345)*

You can display the drawings in Part Nester using the **Special view** and use them for aligning parts on a cutting plan during manual nesting.

The following machine parts are supported:

- Support grate (shuttle table)
- Sheet clamping (shuttle table)
- Suction-cup frame (ByTrans, ByLoader)

What each button does:

- **Import geometry**
  Opens a dialog box for importing an existing geometry.
- **Edit geometry**
  Opens the current geometry for editing in Part Editor.

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What each field does:

- **Name**

  Indicates the name of the machine part used.

  ☞ **Note:** The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g., laser power or machine format).

- **Info 1–8**

  Extra information about the current object can be entered in these fields.

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

4.2.5 Sort



Where can I find these functions?

Select: **Modules > Administration > Administrator**

System data for press brakes is managed in this area.

Sorting unit (previous)

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort**

This form is used to define all the properties of the sorting unit. You can create a new sorting unit using the  or  button.



Attention! All configurable settings must match the ByVision Cutting controller so as to avoid conflicts when moving to positions.

Available entries

In this section you can save various configurations for the selected **Machine type**. Each configuration consists of a combination of the settings in the **Stacking site configuration** and **Stack carrier** sections.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

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Sorting unit

The settings in this section define the basic configuration of the BySort system.

What each field does:

- **Name**

  Defines the name of the sorting unit.

  ☞ **Note:** The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g., laser power or machine format).

- **Info 1–3**

  Extra information about the current object can be entered in these fields.

- **Machine type**

  Defines the type of machine used in the sorting system. The description has the following meaning:

1	2	3	4	5	6	7	8	9
3015	-	D	T	R	-	CP	-	2

Position	Description	Attributes
1	Size of the machine: 3000x1500 4000x2500	 3015 4020
2	-	-
3	Number of sorting heads: One Two	 S D
4	ByTrans layout: T layout L layout	 T L

Position	Description	Attributes
5	ByTrans rotation: Not rotated Rotated 180°	 - R
6	-	-
7	Available stacking areas: C=Cassette P=Pallet W=Carriage	 CP CW CPW
8	Only for special version	-
9	Special version: Version with 2 pallet rows	 2

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

  The selected system of units applies to all thicknesses of the material.

- **Tool change time**

  Defines the average time in seconds for a tool change in production.

- **Part gripping time**

  Defines the average time to grip a part in production. This time includes lowering, gripping of the part, and raising again to travel position.

- **Part placing time**

  Defines the average time to place a part in production. This time includes lowering to stacking position, putting down the part, and raising again to travel position.

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- **Positioning time**

  Average time taken by sorting heads from part pick-up to stacking position in the top position.

- **Minimum lift-off height**

  Defines the lift-off height after gripping the part before the sorting head may be moved horizontally.

  - Min. 75 mm
  - Max. 500 mm
  - Default 100 mm

- **Number of different gripper tools**

  Defines the number of gripper tools in the tool change station.

Stacking areas

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort -> Sorting unit**

These settings are used to display limits and to check whether the sorting head axis positions can be reached. In ByVision Cutting they are overwritten by the settings configured there.

The following types of stacking areas is available:

- **Cassette**

  The parts are stacked in the ByTrans Cross DUH cassette.

- **Pallet**

  Parts are stacked on one or more pallets. Pallets are also containers with an edge, for example crates.

Note: The dimensions of the selected stacking area are displayed as a dotted blue line in the **Stacking area layout** section *(See: ›Stacking area layout‹ Page 161)*.

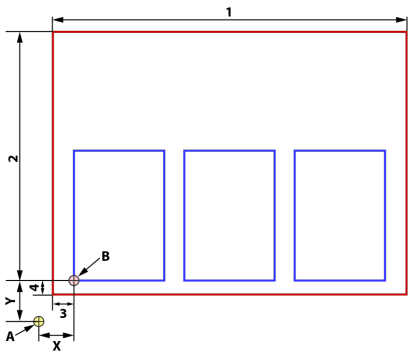


Fig. 4-13: Dimensions for stacking areas

A	Sorting unit origin
B	Stacking area origin
X	X distance to stacking area origin
Y	Y distance to stacking area origin
1	Length (stacking area)
2	Width (stacking area)
3	X offset (to stacking area origin)

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4 Y offset (to stacking area origin)

Stacking site configuration

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort -> Sorting unit**

This section is used to create configurations for pallet, cassette and carriage stacking sites.

Stacking site configurations can be selected in the **Sort Editor** menu in the Part Nester module for creating sorting plans *(See: ›Add‹ Page 817)*.

☞ **Note:** Multiple stacking sites can be defined per stacking area.

Create stacking site

1. Press the  button to create a new stacking site.
2. Enter a name for the stacking site in the **Name** box.
3. Click on the **Pallet** box and select the necessary stack carriers for the stacking site.
4. If necessary, repeat this step to define stacking sites for **Carriage** and **Cassette**.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

What each field does:

- **Name**

  Name of stacking site. The entry can be selected freely.
- **Pallet**

  Selection box of configured stack carriers for the stacking sites.
- **Cassette**

Selection box of configured stack carriers for the stacking sites.

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Zones for tool station / disruptive contours

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort -> Sorting unit**

In this section you can configure zones for tool areas or disruptive contours for all existing stacking areas. The created tool stations are displayed as rectangles on the **Machine layout** tab along with the appropriate name (**A**).

☞ **Note:** Only the assigned tool stations are displayed for the selected stacking area.

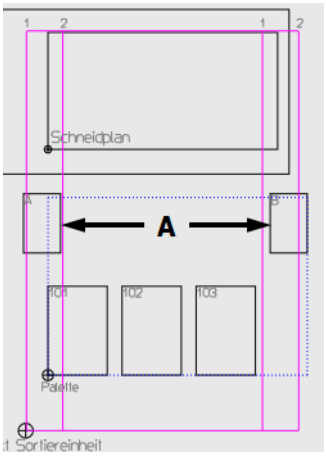


Fig. 4-14: Tool stations in sorting area

What each field does:

- **Name**

  Designation of tool station or disruptive contour.

- **Origin X**

  Origin of the tool station in the X direction. Origin X is indicated from the origin of the sorting unit.

- **Origin Y**

  Origin of the tool station in the Y direction. Origin Y is indicated from the origin of the sorting unit.

- **Length**

  Length of the tool station or disruptive contour.

- **Width**

  Width of the tool station or disruptive contour.

- **Stacking areas**

  Stacking area to which the tool station or disruptive contour is assigned.

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Machine layout

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort -> Sorting unit**

This tab displays the machine layout configured. The machine layout consists of the contours of the appropriate system. The contour must match the original dimensions of the system. In addition to the machine itself, this also includes the pallet, carriage or cassette stacking sites.

- The red lines (**a**) show the maximum range that can be reached by sorting heads 1 and 2 as per the specifications for the gripper tools.
- The dotted blue line (**b**) shows the dimensions of the selected stacking area.

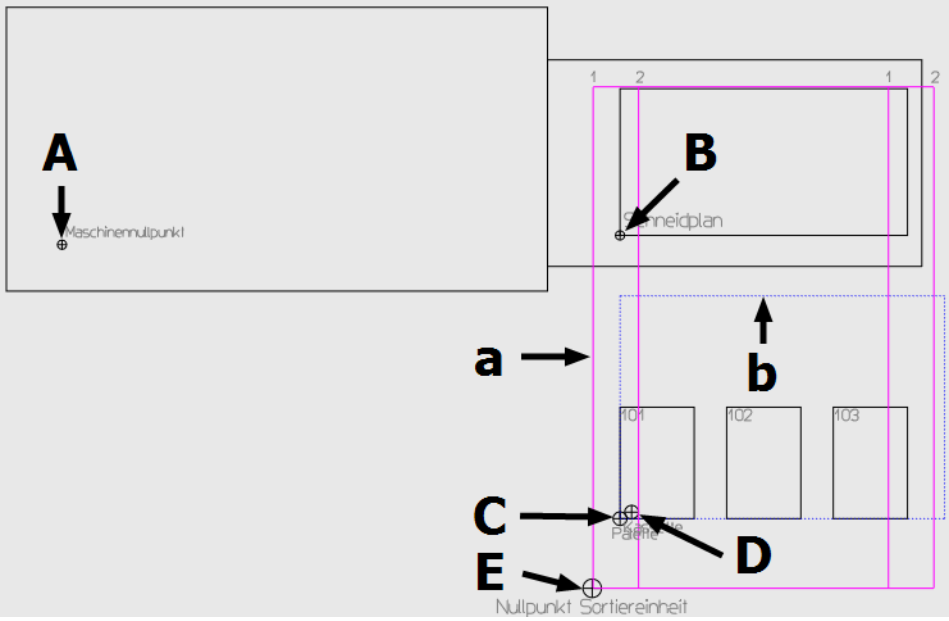


Fig. 4-15: Machine layout with configured origins

Origins:

A	Machine
B	Cutting plan
C	Pallet
D	Cassette
E	Sorting unit

Sorting head

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort -> Sorting unit**

This tab can be used to import the contour of the sorting head as a geometry. This contour is needed for the collision test between the sorting head and the stack *(See: ›Check stacking site definition‹ Page 819)*. The X and Y dimensions of the cutting head are shown in the appropriate fields.

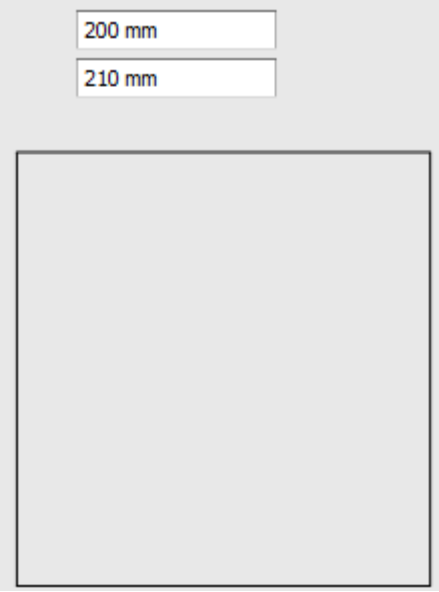


Fig. 4-16: Contour of a sorting head from above

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Properties

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort -> Sorting unit**

This tab displays the various positions and properties of the sorting system. The properties are for information purposes only and cannot be changed.

Explanation of properties:

- **Min. gripper tool separation**

  Minimum clearance of the sorting heads to each other (**A**).

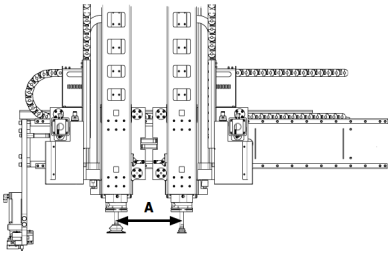


Fig. 4-17: Minimum clearance between sorting heads

- **Min. X position gripper tool 1**

  Defines the lower limit of the gripper tool's travel path in the X direction.

  The range for gripper tool 2 results from the minimum gripper tool clearance.

> **Note:** Defines the X value of the origin of the sorting unit.

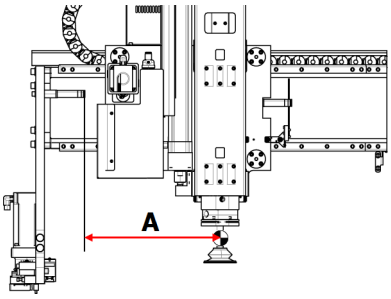


Fig. 4-18: Minimum position of gripper tool 1 in X the direction

- **Max. X position gripper tool 1**

  Defines the upper limit of the gripper tool's travel path in the X direction.

The range for gripper tool 2 results from the minimum gripper tool clearance.

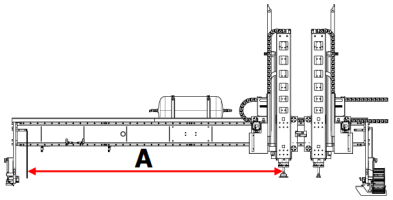


Fig. 4-19: Maximum position of gripper tool 1 in X the direction

Min. Y position gripper tool 1

Defines the front limit of the gripper tool's travel path in the Y direction (**A**).

The ranges for gripper tool 1 and 2 are identical.

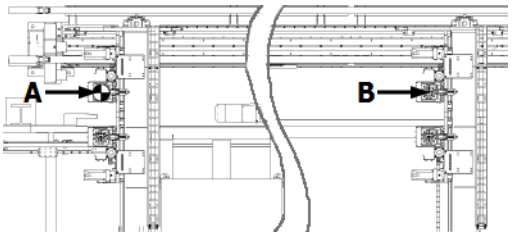


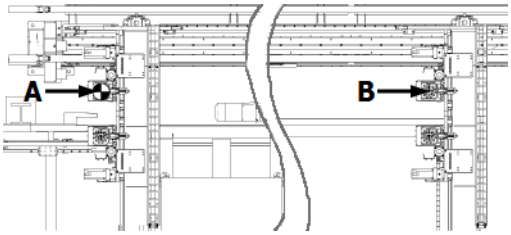
Fig. 4-20: Minimum position on machine layout

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- **Max. Y position gripper tool 1**

  Defines the rear limit of the gripper tool's travel path in the Y direction (**B**).

  The ranges for gripper tool 1 and 2 are identical.



Maximum position on machine layout

- **Min. Z position gripper tool 1**

  Defines the lower limit of the gripper tool's travel path in the Z direction (**A**). The position is specified without an inserted gripper tool.

  The ranges for gripper tool 1 and 2 are identical.

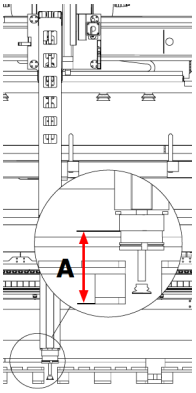


Fig. 4-21: Minimum Z position on machine layout

Max. Z position gripper tool 1

Defines the upper limit of the gripper tool's travel path in the Z direction (**A**). The position is specified without an inserted gripper tool.

The ranges for gripper tool 1 and 2 are identical.

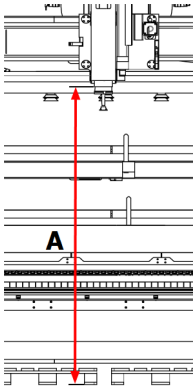


Fig. 4-22: Maximum Z position on machine layout

Plan origin XYZ

Defines the effective position of the cutting plan in the sorting unit in the X/Y/Z direction.

2 gripper tools present

Defines whether the sorting unit is operated with one or two sorting heads. Accordingly, one or two sections are displayed in the preview.

Maximum sorting head load (part weight)

Displays the maximum part weight for a sorting head.

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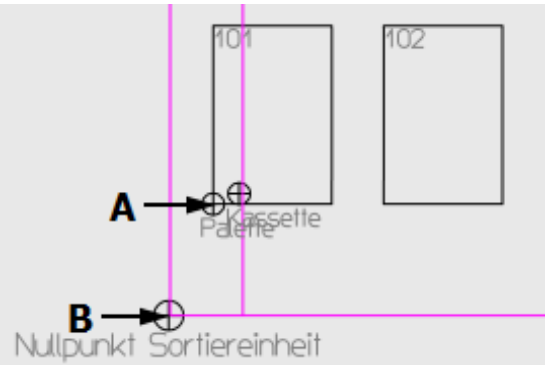
Stack carrier

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort -> Sorting unit**

The stack carrier is the area on which parts can be stacked. Multiple stack carriers can be defined per stacking area.

If the origin of the stacking area (**A**) is inside the origin of the sorting unit (**B**), you can also specify negative values for the positions of the stack carrier in order to make best use of the stacking area.



Create stack carrier

1. Press the button to create a new stack carrier.
2. Enter a numerical ID for the stack carrier in the **ID** box.
3. Enter the respective origin for the stack carrier in the **Origin X/Y/Z** boxes.
4. Enter the maximum permissible stack height on the stack carrier in the **max. stack height** box.
5. Enter the size of the stack carrier in the **Length** and **Width** boxes.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

What each field does:

- **Id**

  Stack carrier ID. The ID must be unique to prevent any confusion. The ID must consist of digits only.

- **Description**

  Field for a more specific description of the stack carrier.

- **Origin X/Y/Z**

  Indicates the origin of the stack carrier in the X/Y/Z direction.

> **Note:** The values for the respective origin are the dimension relative to the origin of the stacking area.

- **max. stack height**

  Maximum stack height on stack carrier.

- **Length**

  Length of stack carrier in the X direction.

- **Width**

  Width of stack carrier in the Y direction.

- **Thickness of frame**

  Defines the thickness of the stack carrier side frame if the stack carrier has a frame (e.g. crate).

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Sorting unit

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort**

This form is used to define all the properties of the sorting unit. You can create a new sorting unit using the  or  button.

Attention! All configurable settings must match the ByVision Cutting controller so as to avoid conflicts when moving to positions.

Available entries

In this section you can create different sorting configurations for the selected **Machine type**. Each configuration consists of a combination of the settings in the **Configuration of gripping zone** section and **Configuration of stacking area** as well as the associated stack carriers.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

Sorting unit

The values in this section define the basic configuration of the sorting unit in the sorting system.

What each field does:

- **Name**

  Defines the name of the sorting unit.

  > **Note:** The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g., laser power or machine format).

- **Info 1–3**

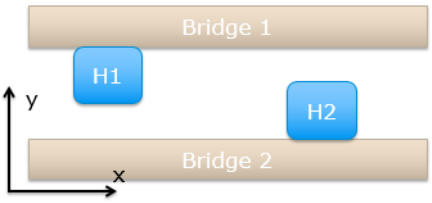
  Extra information about the current object can be entered in these fields.

- **Machine type**

  Defines the type of machine used in the sorting system.

  - **BySort Dual Bridge A**

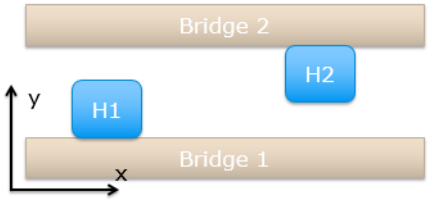
    Sorting unit with two bridges and two sorting heads (H1/H2). The bridge with sorting head H1 is located behind the bridge with sorting head H2.



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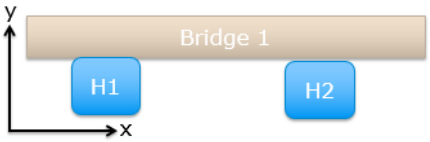
BySort Dual Bridge B

Sorting unit with two bridges and two sorting heads (H1/H2). The bridge with sorting head H1 is located in front of the bridge with sorting head H2.



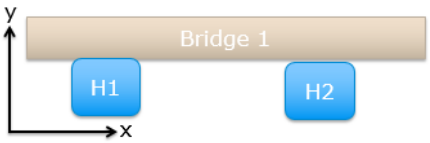
BySort on ByTrans Cross

Previous sorting unit with one bridge and two sorting heads.



BySort on ByTrans Modular

Sorting unit with one bridge and two sorting heads.



BySort Single Head

Sorting unit with one bridge and one sorting head.



- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

- **Minimum lift-off height**

  Defines the lift-off height after gripping the part before the sorting head may be moved horizontally.

  - Min. 75 mm
  - Max. 500 mm
  - Default 100 mm



Note: This value is applied only if the **Use previous postprocessor** option is enabled.

- **Use previous post processor (.scp)**

  The sorting information is output by the postprocessor in SCP format.

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Configuration of gripping zone

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Sorting unit**

This section defines the gripping zone from which the cut parts are picked up by the sorting heads.



Note: The gripping zone is the area on the laser cutting machine where the cut cutting plan is located.

Collision zones

The collision zones determine the limits of the gripping zone over a cutting plan. The collision zone must not be crossed by a sorting head with a tool, nor by a part. The rectangle of the collision zones must therefore be slightly larger than the maximum cutting area of the machine.

The values all refer to the virtual origin of the cutting plan.

- **Collision zone X-min:**

  Defines the minimum X dimension of the gripping zone (**B**) relative to the origin (**A**).

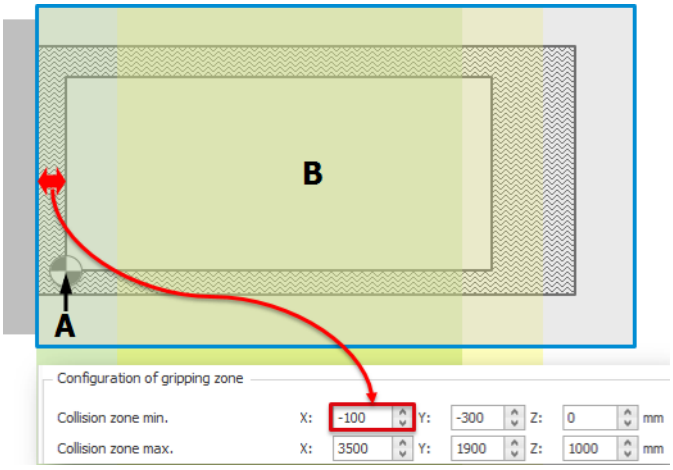


Fig. 4-23: minimum X dimension of the gripping zone to the origin

- **Collision zone Y-min:**
  Defines the minimum Y dimension of the gripping zone (**B**) relative to the origin (**A**).

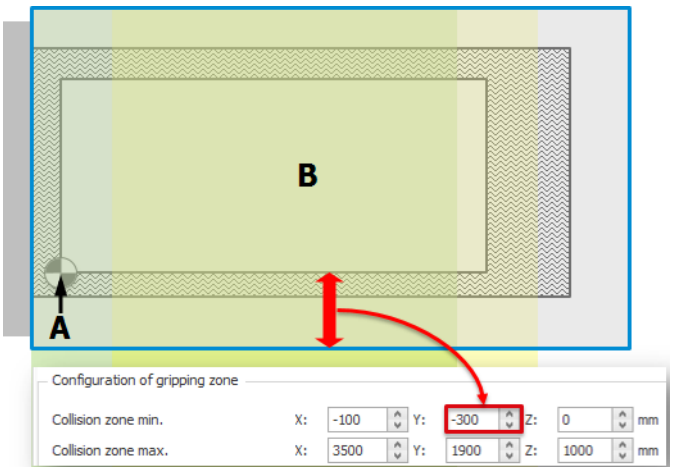


Fig. 4-24: minimum Y dimension of the gripping zone to the origin

- **Collision zone Z-min:**
  Defines the minimum Z dimension of the gripping zone.
- **Collision zone X-max:**
  Defines the maximum X dimension of the gripping zone (**B**) relative to the origin (**A**).

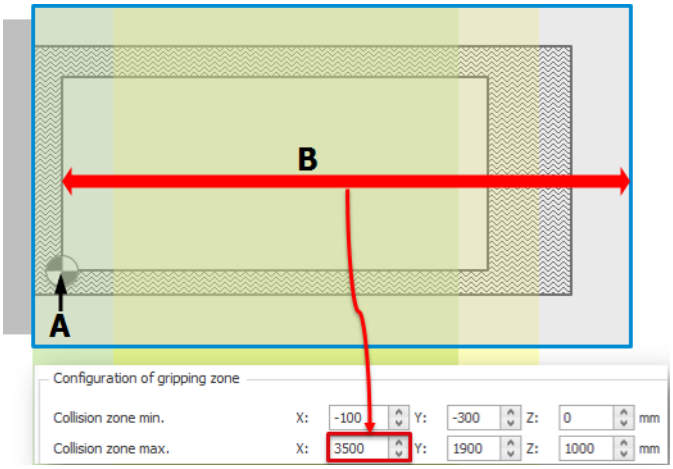


Fig. 4-25: maximum X dimension of the gripping zone to the origin

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Collision zone Y-max:

Defines the maximum X dimension of the gripping zone (**B**) relative to the origin (**A**).

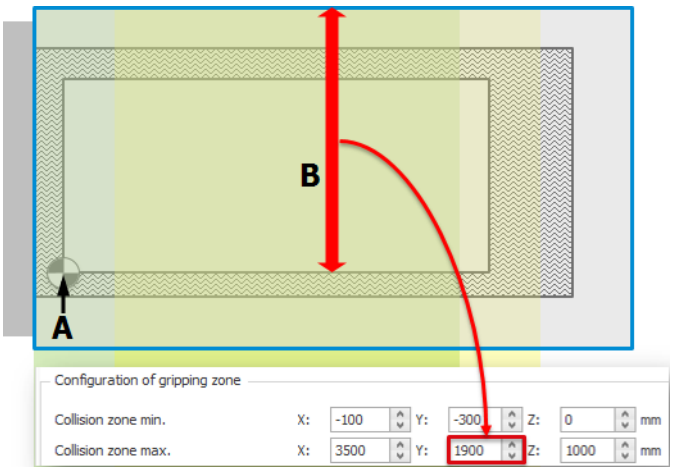


Fig. 4-26: maximum Y dimension of the gripping zone to the origin

Collision zone Z-max:

Defines the maximum Z dimension of the gripping zone.

Approximate position X/Y/Z

Defines the X/Y/Z dimension from the origin of the sorting unit (**B**) to the origin of the cutting plan (**A**).

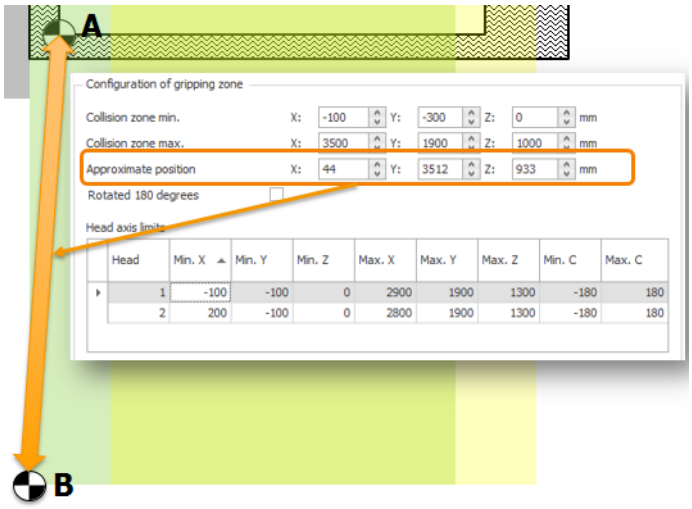


Fig. 4-27: Dimension from sorting unit origin to cutting plan origin

Rotated by 180 degrees

The laser cutting machine is rotated by 180°. The distances for the collision zones to the cutting plan origin (**A**) correspond to that of a non-rotated laser cutting machine.

The approximate position of the sorting unit origin (**B**) to the cutting plan origin (**A**) is given with positive values.

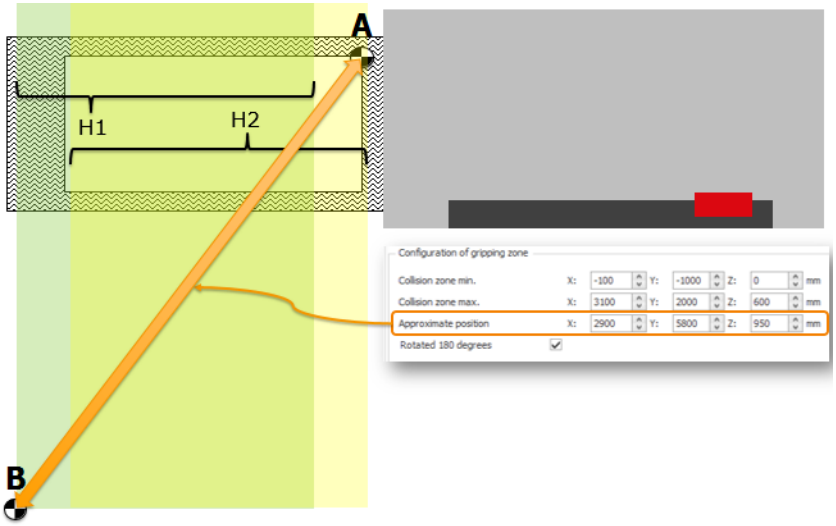


Fig. 4-28: Rotated laser cutting machine

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Head axis limits

In this section you can define the limits of movement of the axes of all existing sorting heads in the gripping zone. The axis limits mean that the sorting head must not travel beyond them.

- **Head**

  Displays the number of the sorting head.

- **Min. X**

  Defines the minimum distance of sorting head **H1/H2** to the origin (**A**) in the X direction.

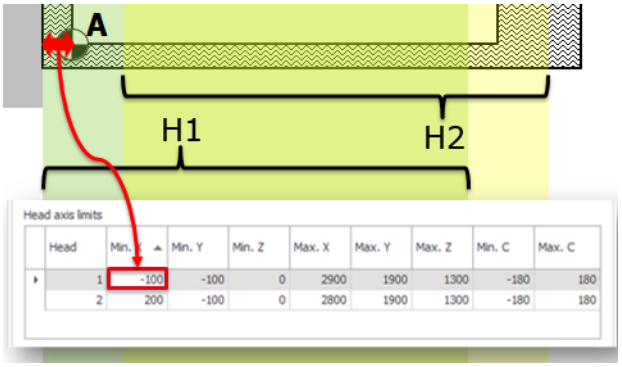


Fig. 4-29: Minimum distance of sorting head **H1** in the X direction

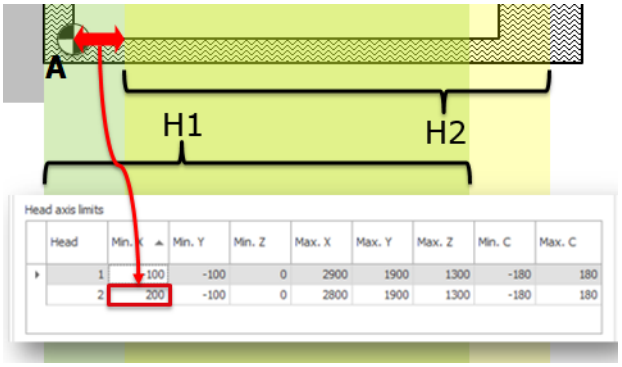


Fig. 4-30: Minimum distance of sorting head **H2** in the X direction

Min. Y

Defines the minimum distance of the sorting heads (**H1/H2**) to the origin (**A**) in the Y direction.

Note:
If the sorting heads are mounted on one bridge, this value is the same for both sorting heads.

If the sorting heads are mounted on two bridges, the values for the sorting heads are different.

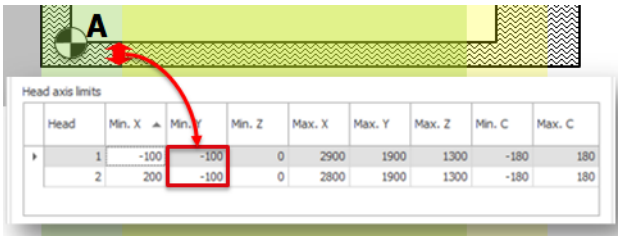


Fig. 4-31: Minimum distance of sorting head **H1/H2** in Y direction

Min. Z

Defines the minimum distance of the sorting head in the Z direction to the collision zone Z-min.

Max. X

Defines the maximum distance of sorting head (**H1/H2**) to the origin (**A**) in the X direction.

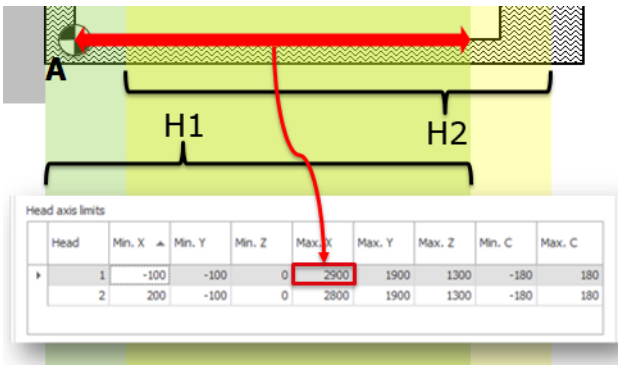
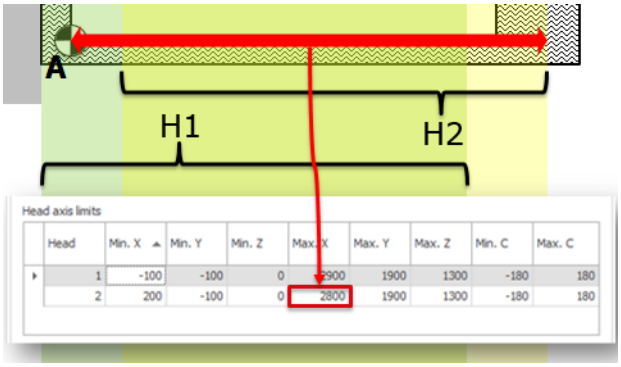


Fig. 4-32: Maximum distance of sorting head **H1** in the X direction

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Maximum distance of sorting head **H2** in the X direction

- **Max. Y**

  Defines the maximum distance of the sorting heads (**H1/H2**) to the origin (**A**) in the Y direction.

> **Note:**
> If the sorting heads are mounted on one bridge, this value is the same for both sorting heads.
>
> If the sorting heads are mounted on two bridges, the values for the sorting heads are different.

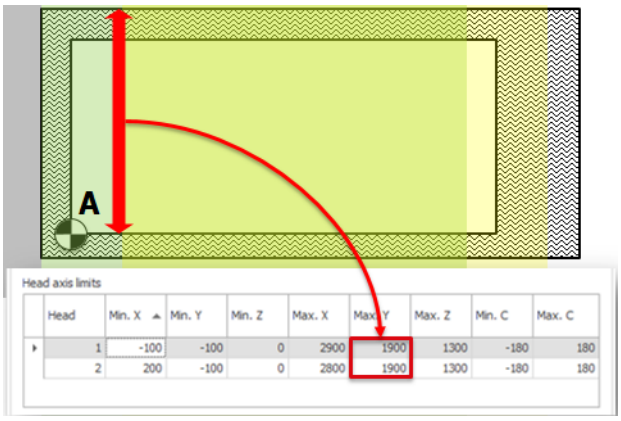


Fig. 4-33: Maximum distance of sorting head **H1/H2** in the Y direction

- **Max. Z**

  Defines the maximum distance of the sorting head in the Z direction to the collision zone Z-min.

- **Min. C**

  Defines the minimum angle of rotation (**E**) of the sorting head (**S**). The default value is **-180°**.

- **Max. C**

Defines the maximum angle of rotation (**F**) of the sorting head (**S**). The default value is **180°**.

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Stacking areas

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Sorting unit**

In this section you can create stacking areas for one or more stack carriers. For example, you can create a stacking area for pallets and another for AGVs.

- **Name**

  This field indicates the name of the stacking area.

- **Type**

  This field is used to select the type of stacking area.

  ☞ **Note:** Parts on the cassette, conveyor belt and AGV stacking areas are automatically removed by a system controller.

  - **Carriage**

    The parts are stacked on a carriage. The carriage can consist of multiple levels.

  - **Cassette**

    The parts are stacked in a cassette. The cassette can be part of a store system.

  - **Floor**

    The parts are stacked on the floor or on a base. By default, Euro 1 pallets are used for stacking on a base. Containers with an edge, e.g. crates, can also be used.

  - **Belt conveyor**

    The parts are placed on a conveyor belt.

  - **AGV**

    The parts are stacked on a stacking area on an AGV.

- **Description**

  You can enter an optional description of the stacking area in this field.

- **GLN**

  The global location number is used to uniquely identify the location of the stacking area in the sorting area.

Stacking area layout

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Sorting unit**

This tab displays the configured layout of the stacking area and the head axis limits for the existing sorting heads. The layout selected in the **Stacking areas** section is displayed in each case. The contour must match the original dimensions of the system. These include the stacking areas of pallets, trolleys, cassette, conveyor belt or AGV.

Note: The areas of the sorting heads are displayed with a red and blue dotted line and the number of the respective sorting head.

Tip: The mouse scroll wheel can be used to zoom in or out of the stacking area. Double-clicking the left mouse button centers the stacking area in the display again.

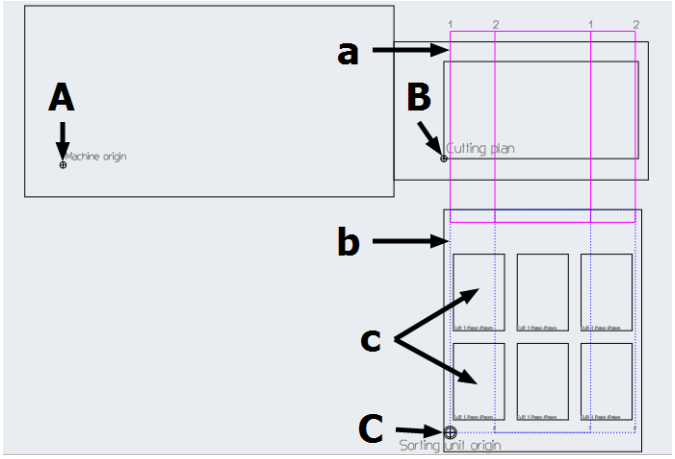


Fig. 4-34: Layout of the stacking area with configured origins

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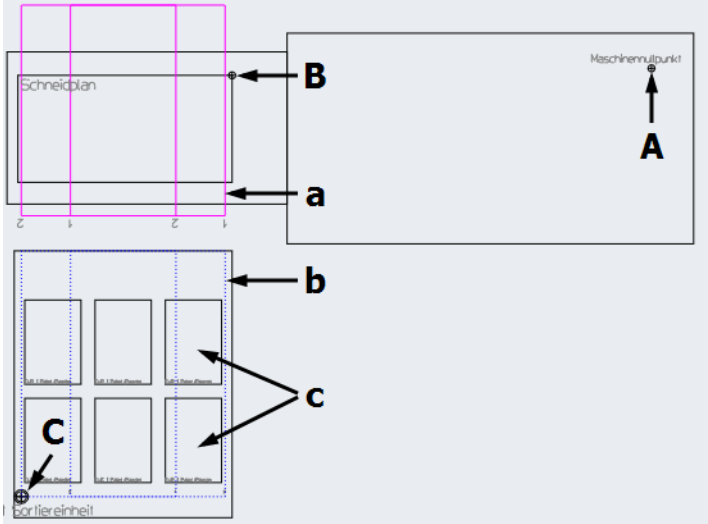


Fig. 4-35: Layout of the stacking area with configured origins with rotated machine layout

Key:

A	Machine origin
B	Cutting plan origin
C	Sorting unit origin
a	Maximum sorting head areas for removing parts
b	Maximum sorting head areas for stacking parts
c	Stack carriers in the stacking area (e.g. pallets)

Properties

This tab displays the various positions and properties of the sorting system. The properties are for information purposes only and cannot be changed.

Configuration of stacking area

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Sorting unit**

This section defines the stacking area in which parts can be stacked. One or more stacking carriers can be positioned in a stacking area for stacking parts.

Collision zones

The collision zones define the boundaries of the stacking area.

- **Collision zone X-min:**

  Defines the minimum X dimension of the stacking area (**B**) relative to the origin (**A**).

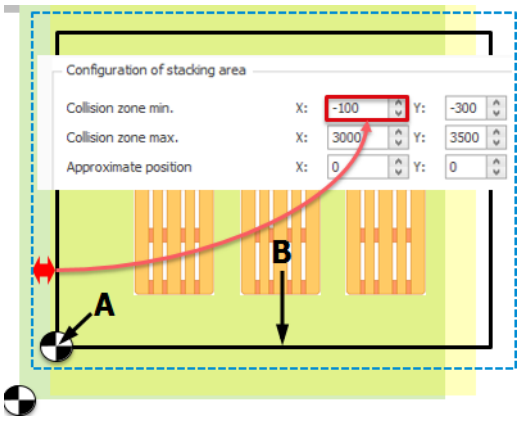


Fig. 4-36: minimum X dimension of the stacking area to the origin

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- **Collision zone Y-min:**

  Defines the minimum Y dimension of the stacking area (**B**) relative to the origin (**A**).

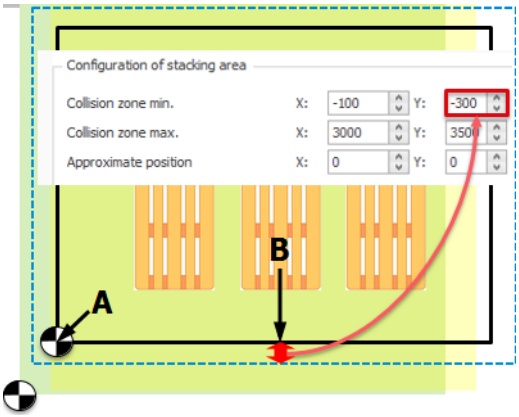


Fig. 4-37: minimum Y dimension of the stacking area to the origin

- **Collision zone Z-min:**

  Defines the minimum Z dimension of the stacking area.

- **Collision zone X-max:**

  Defines the maximum X dimension of the stacking area (**B**) relative to the origin (**A**).

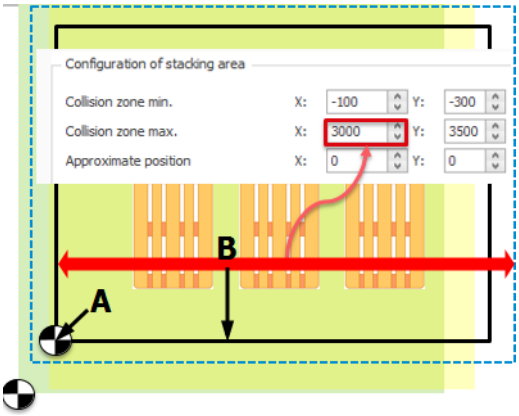


Fig. 4-38: maximum X dimension of the stacking area to the origin

Collision zone Y-max:

Defines the maximum Y dimension of the stacking area (**B**) relative to the origin (**A**).

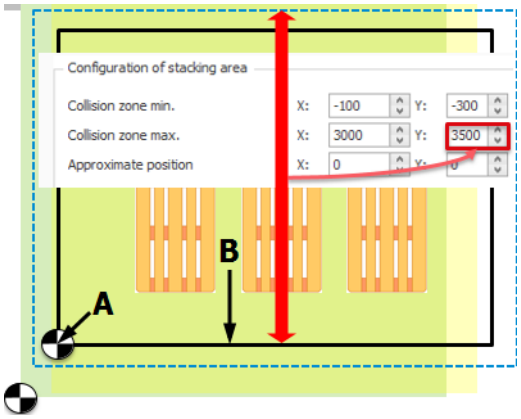


Fig. 4-39: maximum Y dimension of the stacking area to the origin

Collision zone Z-max:

Defines the maximum Z dimension of the stacking area.

Approximate position X/Y/Z

Defines the X/Y/Z dimension from the origin of the sorting unit (**B**) to the origin of the stacking area (**A**).

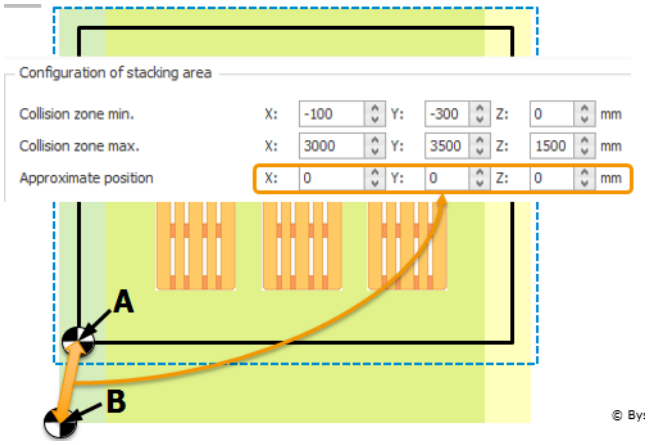


Fig. 4-40: Dimension from the sorting unit origin to the stacking area origin

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Head axis limits

In this section you can define the limits of movement of the axes of all existing sorting heads (**1**/**2**).

- **Head**

  Displays the number of the sorting head.

- **Min. X**

  Defines the minimum distance of sorting head **1** to the origin (**A**) in the X direction.

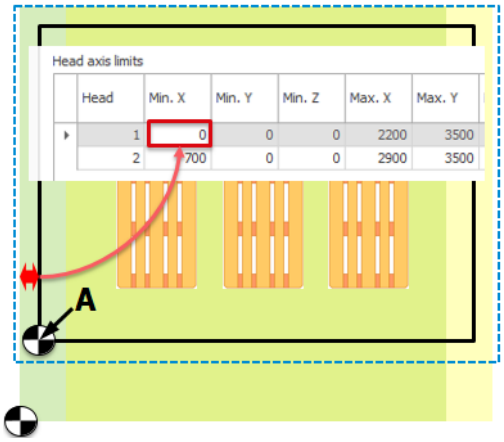


Fig. 4-41: minimum distance of sorting head **1** to the origin in the X direction

Defines the minimum distance of sorting head **2** to the origin (**A**) in the X direction.

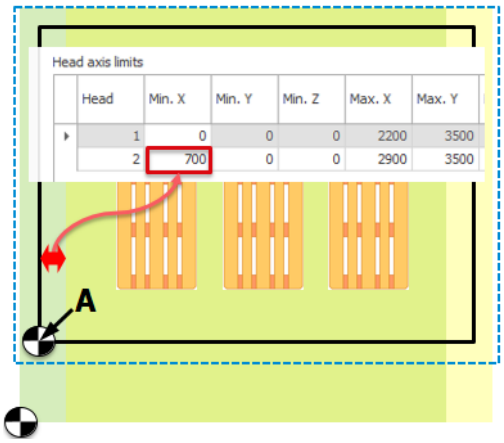


Fig. 4-42: minimum distance of sorting head **2** to the origin in the X direction

- **Min. Y**

Defines the minimum distance of sorting heads **1** and **2** to the origin (**A**) in the Y direction.

> **Note:** For systems with one bridge and two sorting heads, the distances for sorting head 1 and 2 are identical.

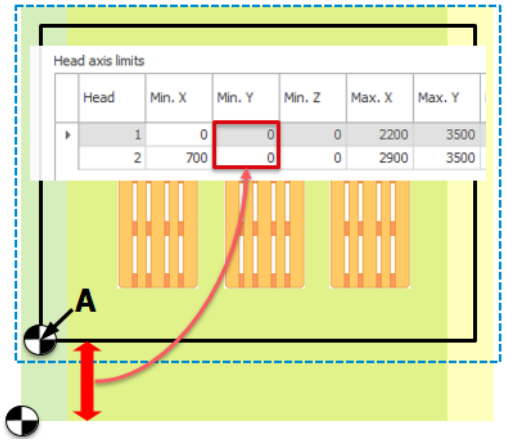


Fig. 4-43: minimum distance of the sorting heads to the origin in the Y direction

- **Min. Z**

  Defines the minimum distance of the sorting head in the Z direction to the collision zone Z-min.

- **Max. X**

  Defines the maximum distance of sorting head **1** to the origin (**A**) in the X direction.

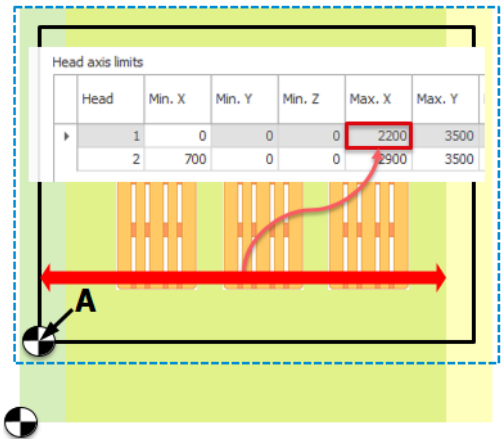


Fig. 4-44: maximum distance of sorting head **1** to the origin in the X direction

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Defines the maximum distance of sorting head **2** to the origin (**A**) in the X direction.

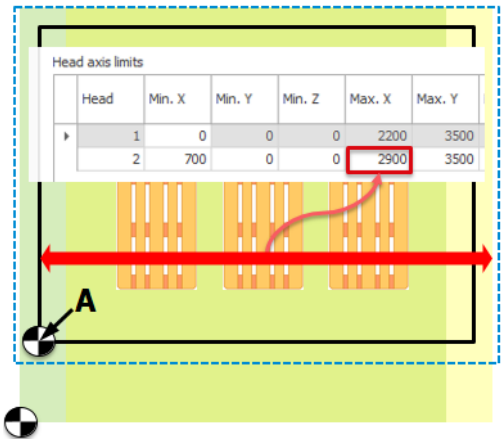


Fig. 4-45: maximum distance of sorting head **2** to the origin in the X direction

- **Max. Y**

  Defines the maximum distance of sorting heads **1** and **2** to the origin (**A**) in the Y direction.

> **Note:** For systems with one bridge and two sorting heads, the distances for sorting head 1 and 2 are identical.

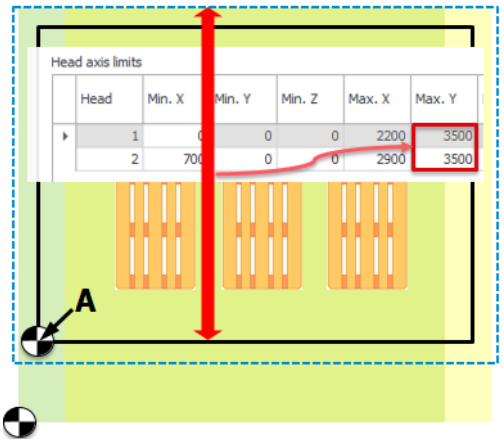


Fig. 4-46: maximum distance of the sorting heads to the origin in the Y direction

- **Max. Z**

  Defines the maximum distance of the sorting head in the Z direction to the collision zone Z-min.

- **Min. C**

Defines the minimum angle of rotation (**C**) of the sorting head (**S**). The default value is **-180°**.

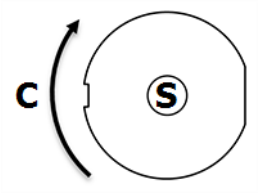


Fig. 4-47: minimum angle of rotation of the sorting head

Max. C

Defines the maximum angle of rotation (**C**) of the sorting head (**S**). The default value is **180°**.

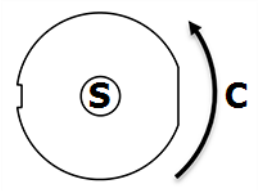


Fig. 4-48: maximum angle of rotation of the sorting head

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Stack carrier

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Sorting unit**

The stack carrier is the area on which parts can be stacked. Multiple stack carriers (**P1-P3)** can be defined per stacking area (**A**).

If the origin of the stack carrier is outside the origin of the stacking area (**A**), negative values can also be specified for the positions of the stack carrier in order to make optimum use of the stacking area.

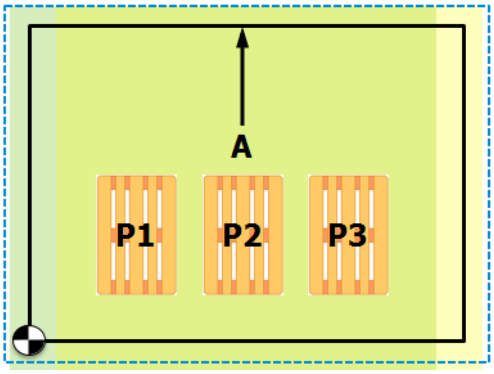


Fig. 4-49: Stack carrier in a stacking area

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

What each field does:

- **Type**

  Field for selecting a predefined stack carrier. Stack carriers can be created or edited in the **Stack carrier** form *(See: ›Stack carrier‹ Page 195)*.

- **Description**

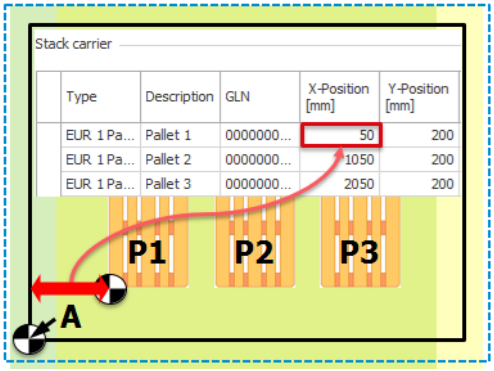
  Field for a more specific description of the stack carrier.

- **GLN**

  The global location number is used to uniquely identify the location of the stack carrier in the stacking area.

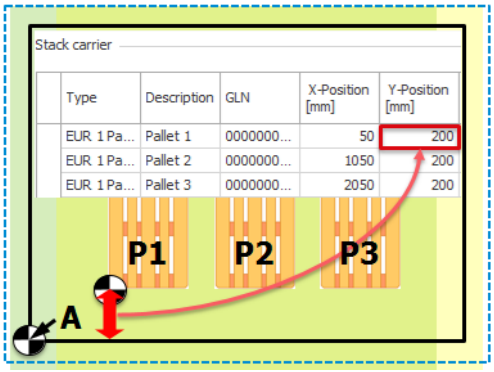
- **Position X, [mm]**

  X position from the origin of the stacking area (**A**) to the origin of the stack carrier (**P1**) in the stacking area.



- **Position Y, [mm]**

  Y position from the origin of the stacking area (**A**) to the origin of the stack carrier (**P1**) in the stacking area.



- **Rotated 90°**

  Rotates the stack carrier 90° clockwise.

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Manage gripper tool stations

How do I open this dialog box?

Select: **Modules > Administration > Administrator > System data > Sort > Sorting unit**

This dialog is used to define the gripper tool stations in the sorting system for each sorting head.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

Gripper tool stations

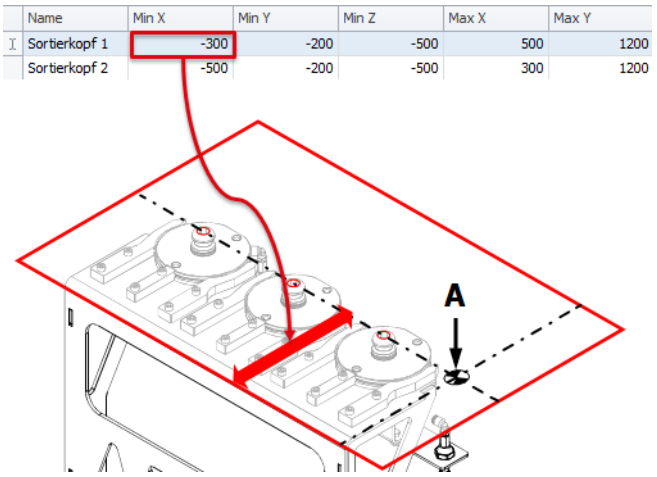
The settings in this section define the collision zone for the gripper tool stations. The origin of the gripper tool station can be freely defined and is determined during machine teach-in.

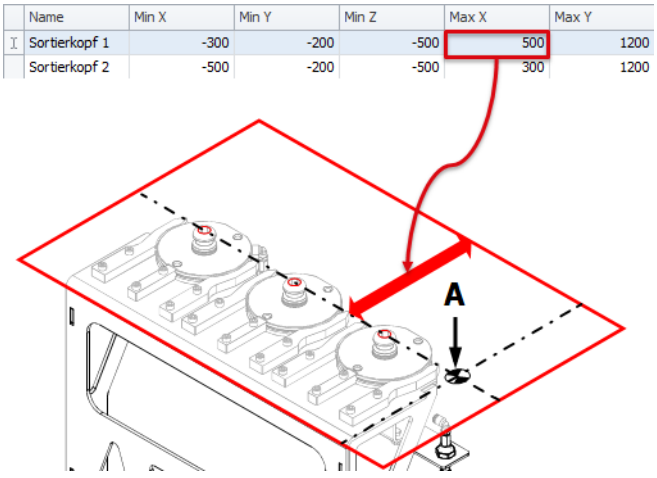
Name

Defines the name of the gripper tool station. The name is displayed in the **Sort job properties** in the **Gripper tool station** section of the Part Nester and Sort Job Editor modules *(See: ›Sort‹ Page 803)*.

Min X / Max X

Defines the minimum and maximum X dimension from the origin of the gripper tool station (**A**).

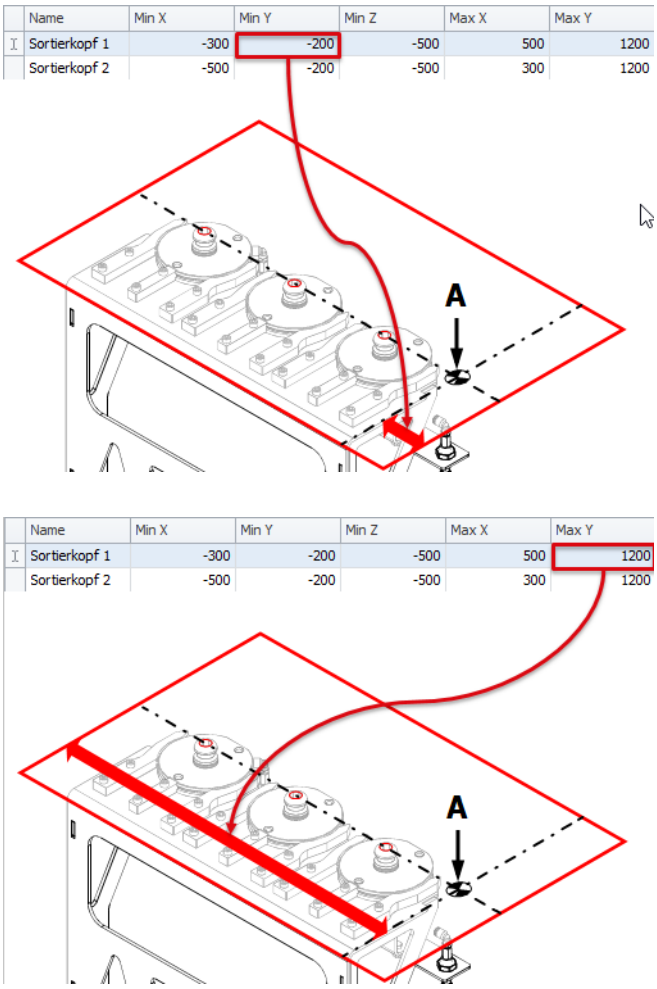




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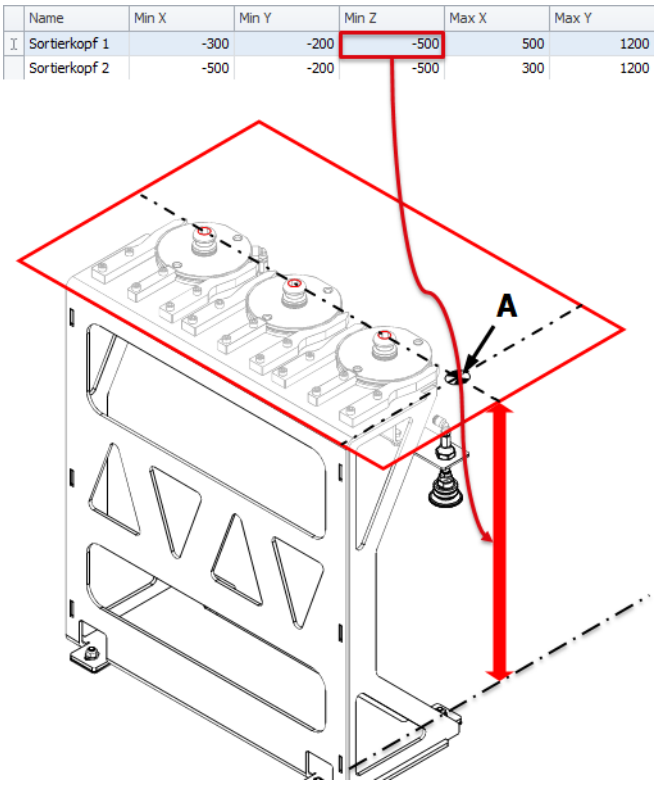
Min Y / Max Y

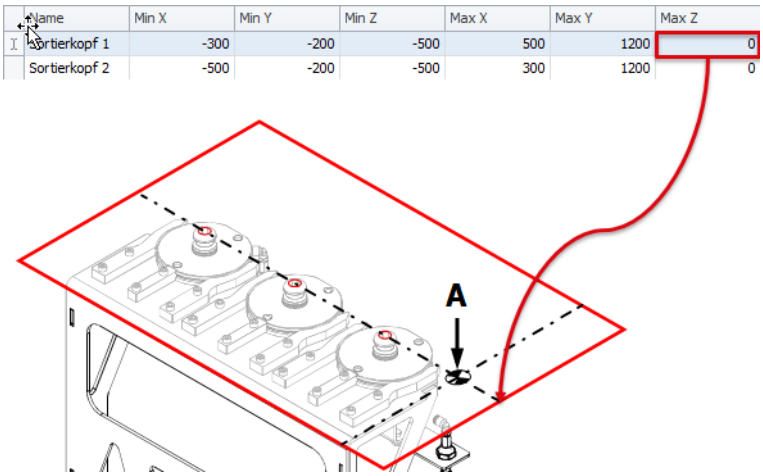
Defines the minimum and maximum Y dimension from the origin of the gripper tool station (**A**).



Min Z / Max Z

Defines the minimum and maximum Z dimension from the origin of the gripper tool station (**A**).





Head 1 / 2

Defines the sorting head for which the settings of the gripper tool station apply.

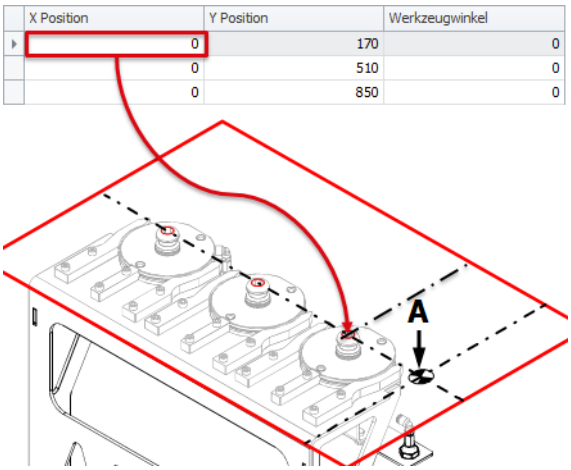
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Gripper tool slots

The settings in this section define the number and positions of the tool slots in a gripper tool station.

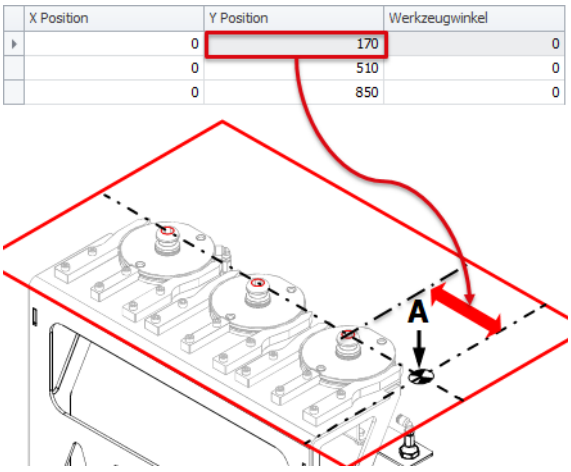
X position

Defines the X position from the origin of the gripper tool station (**A**) to the center of the gripper tool slot.



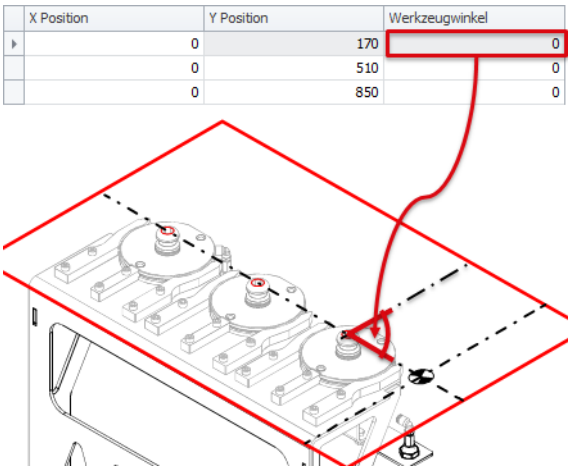
Y position

Defines the Y position from the origin of the gripper tool station (**A**) to the center of the gripper tool slot.



Tool angle

Defines the angle of the gripper tool in the gripper tool slot. The angle is 0° for the left-hand gripper tool slot and 180° for the right-hand gripper tool slot.



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Gripper tool

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort**

This form contains information needed to define gripper tools. Press the  or  button to create a new gripper tool.

Preconfigured gripper tools are available by default. However, you can also create custom gripper tools. Gripper tools can be created for gripping one part at a time or multiple parts simultaneously.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

What each field does:

- **Name**

  Specifies the name of the gripper tool.

  > **Note:** The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g., laser power or machine format).

- **Info 1–3**

  Extra information about the current object can be entered in these fields.

System of units

The system of units for the current object is defined in this field (mm/inches)

The selected system of units applies to all thicknesses of the material.

Load bearing capacity

Displays the gripper tool's maximum load bearing capacity. This is automatically calculated from the information in the **Gripper** section.

Spare distance

The spare distance is the distance from the outside edge of the suction cup to the contour of a part. This distance is used to position the suction cups over the part in order to ensure that it is picked up reliably. If the distance is smaller, the suction cup is displayed red (**A**).

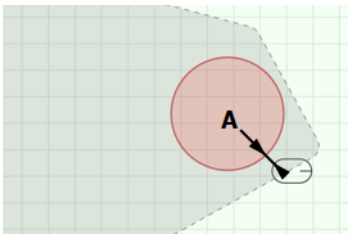


Fig. 4-50: Insufficient spare distance between suction cup and contour

Height

Specifies the height (**H**) of the gripper tool. The height is measured from the upper edge of the mounting plate to the lower edge of the magnet or suction cup in the relieved state minus 5 mm.

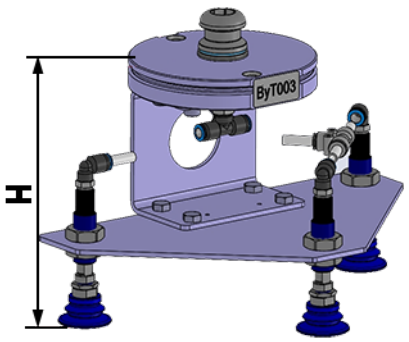


Fig. 4-51: Height of a gripper tool

Sorting unit (previous) compatible

Indicates whether the selected gripper can also be used with the **sorting unit (previous)**. This setting is for information only and cannot be edited.

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Gripper tool contour

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Gripper tool**

The contours of the gripper tool are displayed in this section. The contours of a gripper tool consist of the outer contour of the tool and the gripper hand. The gripper hand can in turn consist of one or more grippers. The contours can be imported by pressing the appropriate buttons or edited in Part Editor.

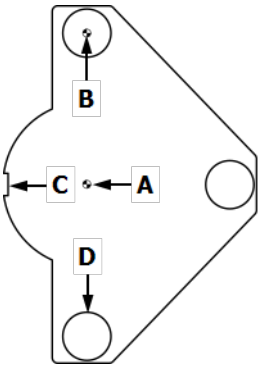


Fig. 4-52: Principle diagram of a static gripper tool

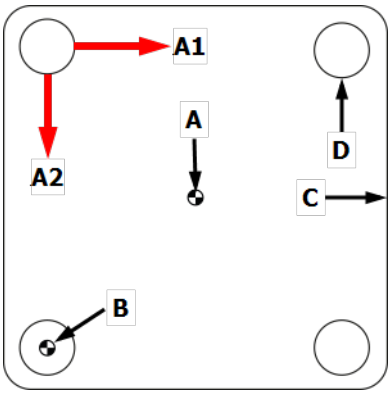


Fig. 4-53: Principle diagram of a gripper tool with multiple axes

Key:

A1	Axis 1 in X direction	B	**Gripper hand origin**
A2	Axis 2 in Y direction	C	**Outer contour of gripper tool**
A	Gripper tool origin	D	**Outer contour of gripper hand**

- **Outer contour**

  The outer contour of the gripper tool (**C**) can be imported or edited in Part Editor.

- **Outer contour of gripper hand**

  The gripper hand of the gripper tool (**D**) can be imported or edited in Part Editor. The contour of a gripper hand can consist of one or more grippers.



Note: The gripper display is dynamic and corresponds to the settings in the columns at the bottom.

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Gripper groups

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Gripper tool**

This section is used to define the type of gripper used for the gripper tool. Several gripper types can be specified for one gripper tool. This is required for the following applications:

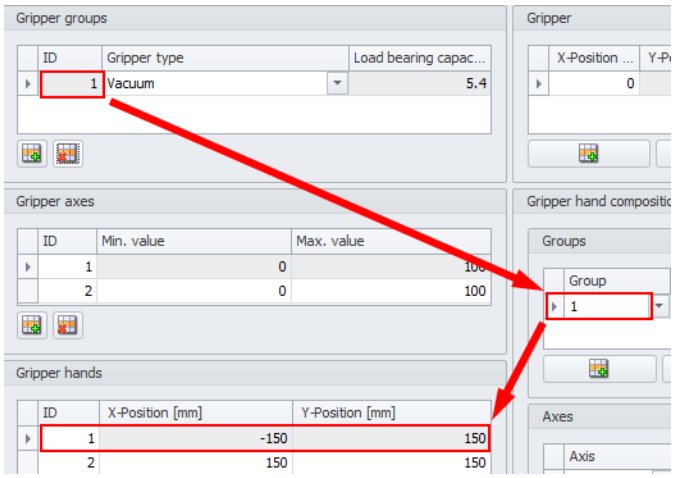
- Different sizes of the same type of gripper are used for the tool.
- Different types of grippers are used for the tool (vacuum gripper, magnet gripper).

Each gripper group applies to a specific gripper defined in the **Gripper** section.

What each field does:

ID

The ID is required for multi-part grippers in order to assign the gripper type in the **Groups** section to the selected gripper hand in the **Gripper hands** section.



Gripper type

Defines the type of gripper tool.

Vacuum

Gripping is performed by means of vacuum. The suction grippers must not be positioned over holes.

Magnet

Gripping is performed by means of a magnet. The magnet grippers may be positioned over holes.

Not defined

Reserved for future gripper tools types.

Load bearing capacity

The value is taken from the information in the **Gripper** section.

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Gripper axes

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Gripper tool**

This section is used to create the existing axes for a multi-part gripper. Each axis is defined with an ID and a minimum and maximum value for the movement range of the axis. The value can correspond to that of the motor's coordinate system or be a % of the movement range.

The minimum and maximum values are multiplied by the value of the coefficient in the **Axes** section and give the dimension of the axis drive measuring system.

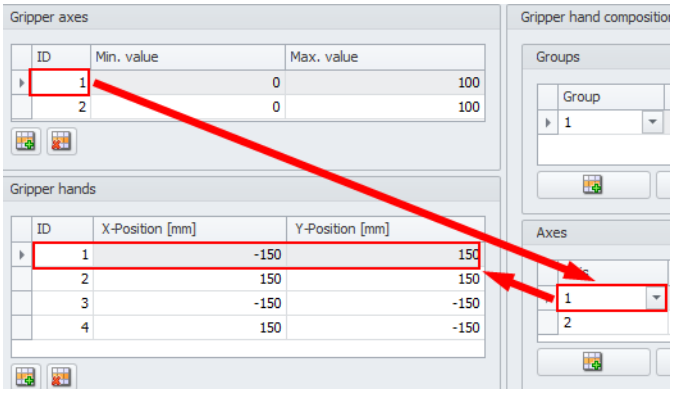
Example:

Max. value = 100		
Coefficient X = **1**	Coefficient X = **0.5**	Coefficient X = **0**
Coefficient Y = **0**	Coefficient Y = **0**	Coefficient Y = **-1**
Max. value X = 100 x 1 = **100**	Max. value X = 100 x 0.5 = **50**	Max. value X = 100 x 0 = **0**
Max. value Y = 100 x 0 = **0**	Max. value Y = 100 x 0 = **0**	Max. value Y = 100 x -1 = **-100**

What each field does:

ID

The ID is needed for multi-part grippers in order to assign the axis with the appropriate coefficient in the **Axes** section to the selected gripper hand in the **Gripper hands** section. When creating a gripper axis, the ID is generated automatically.



Min. value

Minimum value of the movement range of the gripper axis.

Max. value

Maximum value of the movement range of the gripper axis.

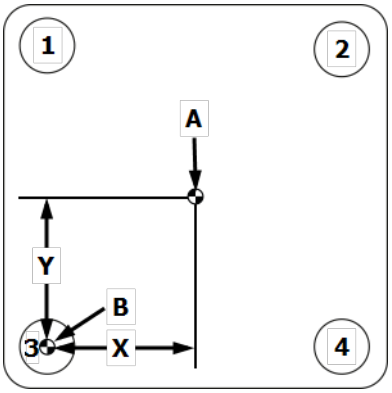
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Gripper hands

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Gripper tool**

This section is used to create all gripper hands on the gripper tool and define their position relative to the origin of the gripper tool.



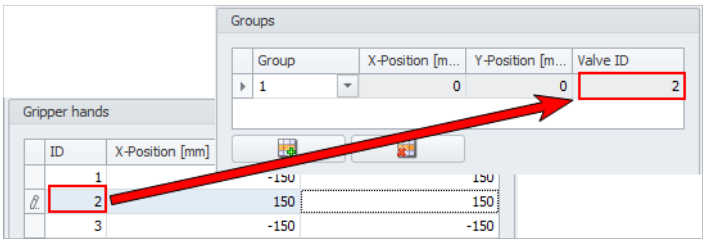
Key:

A	Gripper tool origin
B	Gripper hand origin
X	Gripper hand X distance
Y	Gripper hand Y distance
1-4	Gripper hand 1–4

What each field does:

ID

The ID is required to define the valve ID for multi-part grippers.



X position [mm]

Distance of the gripper hand in the X direction relative to the origin of the gripper tool.

Y position [mm]

Distance of the gripper hand in the Y direction relative to the origin of the gripper tool.

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Gripper

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Gripper tool**

This section provides details about the gripper being used.

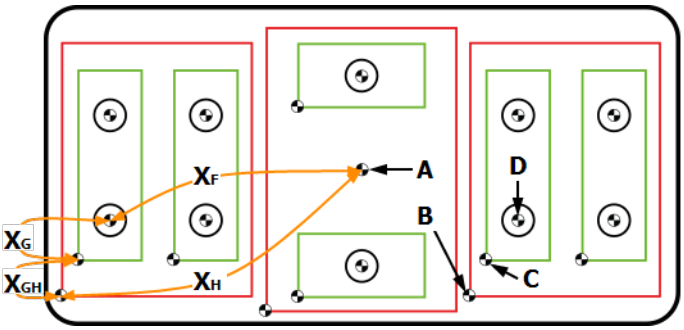
- For gripper tools in which all grippers are permanently mounted on the tool, an entry must be created for each gripper.
- For gripper tools in which the grippers can be moved along axes as gripper hands, only one entry is required (multi-part grippers). The X/Y positions of the individual grippers are defined in the **Gripper hands** section and have a default value of 0.

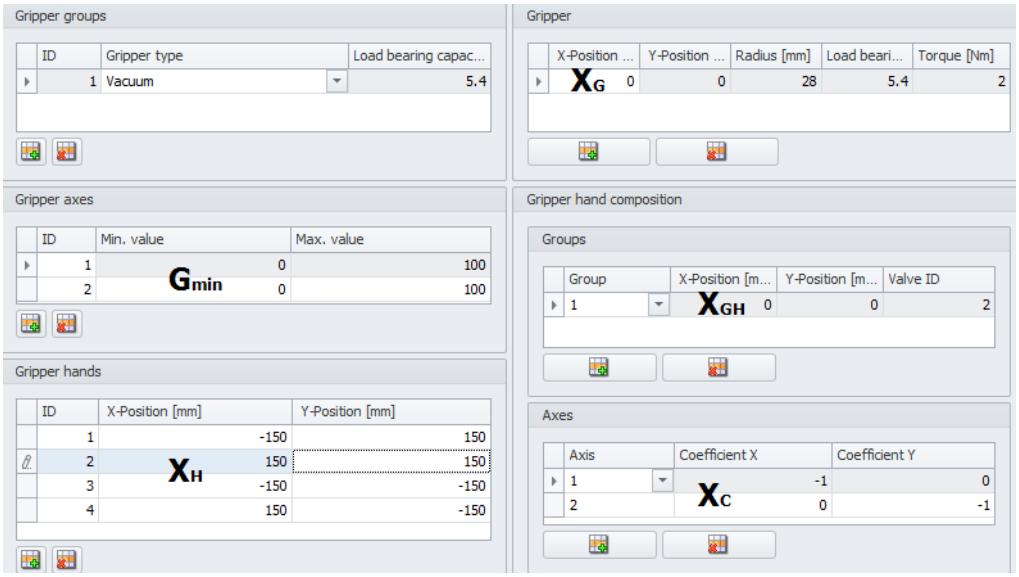
☞ **Note:** For multi-part grippers, the X/Y positions refer to all available gripper hands.

The end position of a gripper in multi-part grippers is calculated according to the following formula:

$$X_F = X_G + X_H + X_{GH} + \Sigma(\text{axis values} * X_C)$$

Note: The same formula is used to calculate the Y coordinates of the gripper





Key:

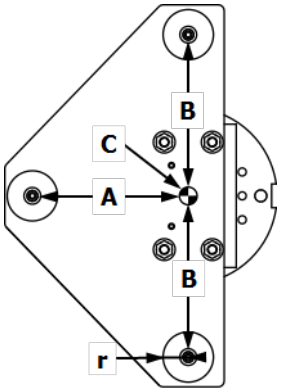
A	Gripper tool origin
B	Origin of gripper hands
C	Origin of groups
D	Gripper origins
X_F	This dimension refers to the origin of the gripper tool (**A**)
X_G	This dimension refers to the virtual origin of the group (**C**)

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X_H	This dimension is relative to the origin of the gripper tool (**A**)
X_{GH}	This dimension is relative to the virtual origin of the gripper hand (**B**)
G_{min}	Specifies the minimum or default value for the axis (axis values)
X_C	X-axis coefficient for all axes

What each field does:

- **X position**
  X distance of grippers (**A**) relative to the virtual origin of the group (**C**).
- **Y position**
  Y distance of the gripper (**B**) relative to the virtual origin of the group (**C**).
- **Radius**
  Radius of the gripper (**r**).



- **Load bearing capacity**
  Maximum load-bearing capacity of the gripper.

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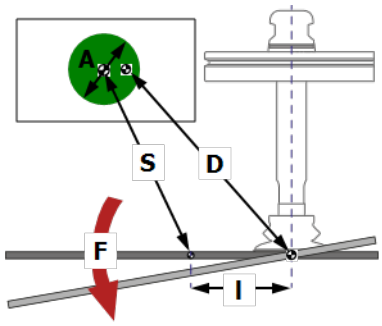
Torque

Defines the maximum permissible force (**F**) before the part tilts when lifted when the part is gripped eccentrically to its center of gravity. The smaller the weight of the part, the larger the area (**A**) in which the gripper can be placed outside the part's center of gravity.

The torque is calculated by multiplying the distance (**I**) from the center of the gripper (**D**) to the part's center of gravity (**S**) by the weight of the part.

The value is used to display the stability of a gripper in Part Editor (red/orange/green). It is only taken into account if all grippers of a gripper tool are arranged on one line.

☞ **Note:** Additional forces caused by the movement or rotation of the sorting heads are not taken into account. Likewise, the value has no direct influence on the sorting process on the machine.



Key:

A	The offset range of the part's center of gravity to the center of the gripper, determined by the torque, in which the part can still be gripped reliably.
D	Center of the gripper around which the part rotates when offset from the part's center of gravity.
I	Offset from the center of the gripper to the part's center of gravity
F	Torque of the part around the center of the gripper
S	Center of gravity of part

Gripper hand composition

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort > Gripper tool**

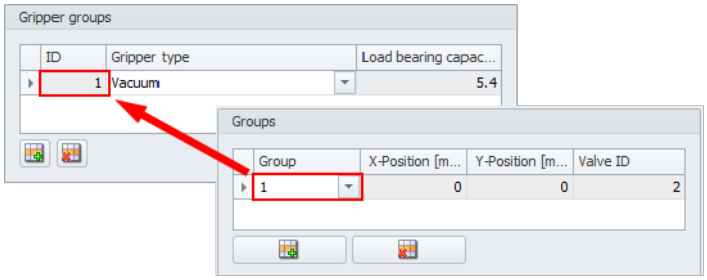
Groups

In this section, gripper hands are connected to form groups. Each gripper hand is assigned to a group. It is possible to create several groups with different gripper hands.

What each field does:

- **Group**

  Selection box for the gripper group ID. The group can only be selected if there are several groups in the **Gripper groups** section.



- **X position [mm]**

  Defines the X position of the group relative to the origin of the gripper hand.

- **Y position [mm]**

  Defines the Y position of the group relative to the origin of the gripper hand.

- **Valve ID**

  Defines the valve ID of the group. The ID must be known to the machine and defined in the machine.

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Axes

This section indicates how the selected gripper hand can be moved in the **Gripper Hands** section. The X/Y coefficients indicate the direction of the axis according to the X/Y coordinates of the groups in a gripper hand.

An entry must be created for each axis. In the **Axis** selection box, the entry must be assigned to the axis in the **Gripper axes** section. The coefficient is multiplied by the min./max. value in the **Gripper axes** section and gives the movement range of the axis *(See: ›Gripper axes‹ Page 184)*.

Meaning of the coefficients:

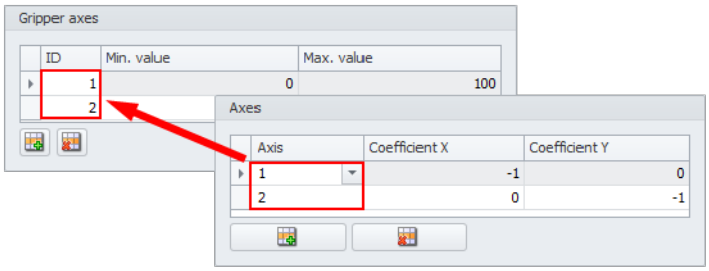
- **0** = No movement of the axis
- **1** = Movement of the axis in the positive direction
- **-1** = Movement of the axis in the negative direction

Note: Only numeric values can be used.

What each field does:

Axis

Defines the ID of the axis from the **Gripper axes** section.



X coefficient

Defines the X coefficient for the movement of the respective axis.

Y coefficient

Defines the Y coefficient for the movement of the respective axis.

Stack carrier

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort**

This form is used to define all stack carrier properties. You can create a new stack carrier type using the  or  button.

Available entries

In this section you can save various stack carrier types. Each type of stack carrier consists of different dimensions and weights defined in the **Stack carrier type** section.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

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Stack carrier

The settings in this section define the type and dimensions of the stack carrier. The configured stack carrier can be selected in the **Sorting Unit V2** form in the **Stack carrier** section in order to define a stacking area *(See: ›Type‹ Page 171)*.

What each field does:

- **Name**

  Specifies the name of the stack carrier.

  ☞ **Note:** The name is essential for the program to be able to save the record.

- **Description**

  You can enter a more detailed description of the current object in this field. (E.g.: Standard pallet).

- **Info 1–3**

  Extra information about the current object can be entered in these fields.

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

- **Carrier type**

  Defines the stack carrier type. The following stack carrier types can be selected:

  ☞ **Note:** The images are examples and may differ from the actual carrier type.

  - **Pallet**

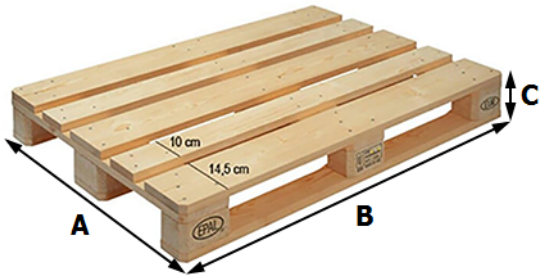


Fig. 4-54: Euro 1 pallet

Cassette

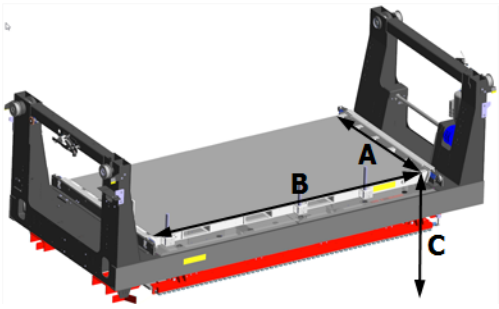


Fig. 4-55: DUH cassette in Bytrans Cross

Box

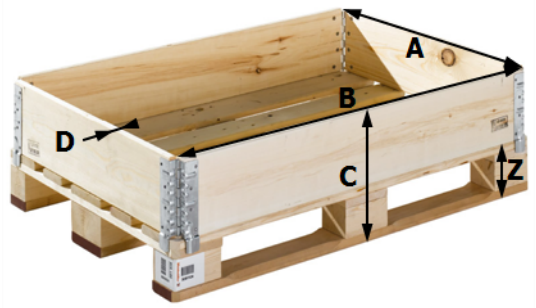


Fig. 4-56: Euro 1 pallet with frame

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- **Pillar pallet**

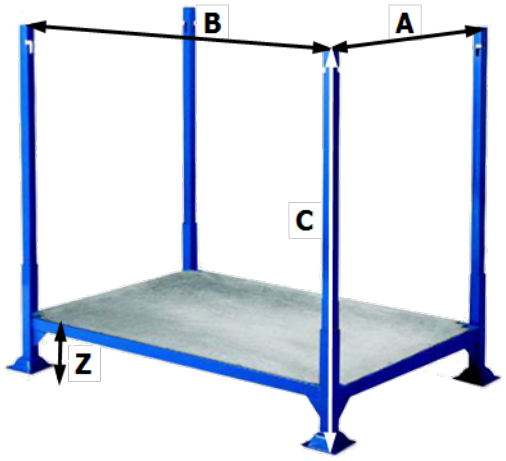


Fig. 4-57: Pillar pallet

- **Sorting rule**

  Defines how the parts are placed in the stack carrier.

  The sorting rule display for the stack carrier can be displayed via the special display **Additional stack carrier info** in Sort Editor or Sort Job Editor.

  - **Stacked**

    The parts are placed on the stack carrier as a stack.

  - **Distributed**

    The sorting head places the parts randomly within the inner limits of the stack carrier.

    ☞ **Note:** This option can be used only with the **box** carrier type, otherwise the parts would fall off the stack carrier.

- **Width**

  Defines the width of the frame around the stack carrier in the Y direction (**A**).

- **Length**

  Defines the length of the frame around the stack carrier in the X direction (**B**).

- **Height**

  Defines the height of the stack carrier (**C**). For cassettes, the height of the stacking area must be given relative to the ground.

- **Bottom offset (Z)**

  Defines the height from the bottom of the stacking area to the bottom of the pallet (**Z**).

- **Thickness of frame**

  Defines the width of the frame (**D**) around the carrier. The frame is displayed as in the right preview image. This dimension is required only for boxes with a frame.

- **Max. stack height**

  Defines the maximum possible stacking height on the stack carrier.

- **Maximum loading weight**

  Defines the maximum possible stack weight of the carrier. The maximum loading weight is specified by the manufacturer in the carrier properties.

- **Empty weight**

  Defines the empty weight of the carrier. The empty weight is specified by the manufacturer in the carrier properties.

Sorting zones

Where do I find this form?

Select: **Modules > Administration > Administrator > System data > Sort**

This form can be used to define sorting zones. You can create a new sorting zone using the  button.

Sorting zones can be used to sort parts together on a stacking site for subsequent processes such as bending, welding or painting. Sorting zones can also be used to sort parts belonging to an assembly onto a pallet together. You can create sorting zone names according to your own requirements.

- **Name**

  Field for entering a name for the sorting zone.

- **Color**

  Color assigned to the sorting zone. The color is automatically assigned when you create a new sorting zone. Click on the color icon to specify a custom color.

  The color is used in Sort Job Editor to indicate the target area. A target area can be a pallet or cassette, for example.

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4.2.6 Label

Where can I find these functions?

Select: **Modules > Administration > Administrator**

This form displays all fonts and their properties for writing labels. The fonts are required by the **Labels** function for the **Text** element in Part Editor *(See: ›Font‹ Page 497)*.

If no fonts have been installed, they must be imported using the Import function in Administrator.

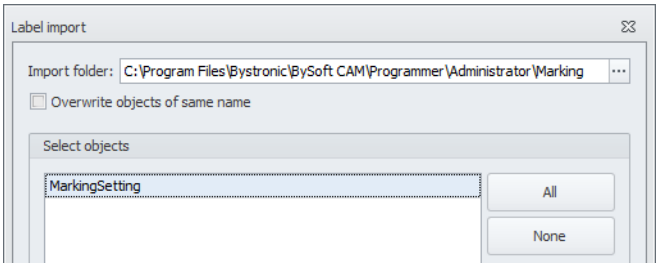


Fig. 4-58: Import path for fonts

Note: These settings are for viewing only and cannot be changed.

4.2.7 Drill



Where can I find these functions?

Select: **Modules > Administration > Administrator**

This section is used to manage system data for drilling.

Thread

Where do I find this form?

Select: **Modules > Administration > Administrator > System Data > Drilling**

This form displays all thread sizes for tapping threads that can be used on ByFlex. These settings are required for drilling functions in Part Editor.

 **Note:** The settings for **threads** and **core hole diameter** cannot be edited.

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4.2.8 Available entries

This pane displays all the objects that exist for the selected element. A new object appears in this pane as soon as it has been created and saved.

Filtering entries

The header contains a field (**A**) for filtering entries. To filter entries, enter the character string to be used as the filter. Filtering of entries starts as soon as you start entering filter characters. To remove the filter, delete the entry from the filter bar.

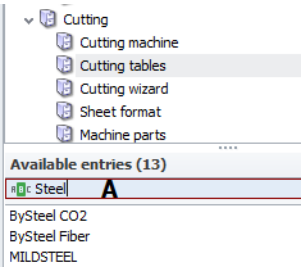


Fig. 4-59: Filter function in the **Available entries** pane

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

4.3 Settings Manager

Where do I find this form?

Select: **Modules > Administration > Administrator**

Configure the settings in this form to assign the appropriate parameters and settings automatically when creating new parts or jobs.

- **System of units**

  Define the system of units (metric/English) for dimension settings in Settings Manager in this field. The default system of units is **Metric**.

Tip: Highlight a row and call up the context menu to quickly configure row options *(See: ›Creating row entries from the drop-down menu‹ Page 204)*.

The Settings Manager can be used for the following modules:

- **Cutting wizard**

  This section is used to define which cutting rule set to use for the cutting technology. The settings apply to all part and tube modules.

  Tip: Right-click on the **Cutting Wizard** entry to import the Cutting Wizard settings for CO_2 laser machines and waterjet cutting machines into Settings Manager using the **Complete** function.

- **Part Editor**, **Tube Editor**

  To ensure that these settings are used in the Part Editor and Tube Editor modules, the **From Settings Manager** must be enabled when you first set the machine.

- **Part Nester**, **Tube Nester**, **Tool Setup Optimizer**

  To ensure that these settings are used in the Part Nester, Tube Nester, and Tool Setup Optimizer modules, the **From Settings Manager** option must be enabled when you create a new job.

  Note: In order for the Part Nester settings to be used in Plant Manager, the **Settings Manager** option must be enabled in **Job settings**, in the **Settings for** pane .

Procedure for creating settings tables

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1. Use the default option setting in the first row to get the desired result. You can edit the default setting in the appropriate box.
2. If the default setting cannot be used for all applications, insert additional table rows to configure special cases.
3. Test the settings in the **Test setting** section *(See: ›Test settings‹ Page 208)*.

Attention! When switching from templates to the Settings Manager, create the settings tables first. Only switch to the Settings Manager after testing the tables.

Creating multiple entries in tables

In certain columns you can create multiple attribute entries to reduce the number of rows in the table. You can select the attributes by means of a checkbox in the context menu for the particular option ( ).

1. Select the cell in which to create multiple entries.
2. Click on the triangle at the right edge of the cell. The program displays a selection box with the cell attributes.
3. Select the appropriate attributes and press **OK** to confirm the selection.

Creating row entries from the drop-down menu

If you highlight a row, you can define the entries from the drop-down menu.

1. Click on the edit icon at the right edge of the row to highlight the row ( ).
2. Right-click to call up the context menu with the available options.
3. Click on the triangle at the right edge of the appropriate option. The program displays a selection box with the cell attributes.
4. Select the appropriate attributes and press **OK** to confirm the selection. The selected attributes are displayed in the table.

Settings Manager

This section displays all configuration options for the selected function in table format.

Select a function to display the available settings in a table. The first row in the table contains the default values for the selected function. By clicking on a cell, you can modify the corresponding option attribute.

The table columns show the key criteria followed by the options. Key criteria are shown **gray** in the default row and cannot be changed (**A**). The options in the default row are the default values (**B**).

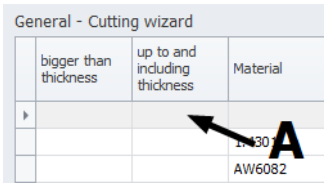


Fig. 4-60: Key criteria

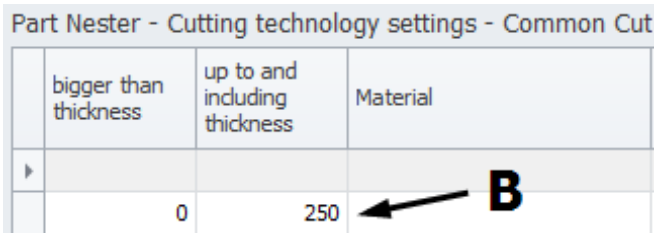


Fig. 4-61: Options in the default row



Note: If the system cannot find any rows with matching key criteria, it uses the attributes of the default row.

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Sample configuration for the cutting wizard

Material	Machine	Cutting process	Cutting gas	Cutting rule set
1.4301	BySpeed-Pro	Laser Fiber	AIR	BySpeed-Pro_4400_ 1.4301_AIR
1.4301(1)	BySpeed-Pro	Laser Fiber	N_2	BySpeed-Pro_4400_ 1.4301_N2
AW5754	BySpeed-Pro	Laser Fiber	AIR	BySpeed-Pro_4400_ AW5754_AIR
AW5754	BySpeed-Pro	Laser Fiber	O_2	BySpeed-Pro_4400_ AW5754_O2
1.4301(2)	Bystar Fiber	Laser Fiber	N_2	Bystar_3000_1.4301_N2

Example:

A job with the material 1.4301 for the machine BySpeed-Pro (**1**) is cut on the machine Bystar Fiber (**2**).

The system automatically selects the correct cutting rule set for the new machine.

Editing table entries

Enter “wildcards” to group or combine attributes in a row.

Example:

- An * stands for any number of valid characters.
  - 1.4301*
  - AW*
  - *steel
  - By*3015
- Use a , to define multiple materials in one table row.
  - 1.4301, DC01, DD11
- A ? stands for exactly one character. Multiple ? stand for exactly that number of valid characters.
  - ByStar ????
  - 1.????

Tip: You can right-click to call up the context menu to edit multiple table rows and their attributes at once *(See: ›Creating multiple entries in tables‹ Page 204)*!

Explanation of key criteria

The following key criteria are displayed depending on the function selected:

- **greater than thickness**
  Defines the lower limit for a thickness.
- **up to and including thickness**
  Defines the upper limit for a thickness.
- **Material**
  Defines the material. Any of the materials in the **Material** form can be selected *(See: ›Material‹ Page 96)*.
- **Material group**
  Defines the material group. All materials that are assigned to the material group are included *(See: ›Material group‹ Page 94)*.

> ☞ **Note:** You can specify either the **Material** or the **Material group**. It is recommended to specify a material group consisting of materials.

- **Cutting process**
  Defines the cutting process used. The **Laser** attribute applies to CO_2 and fiber lasers.
- **Cutting machine**
  Defines the cutting machine.

> ☞ **Note:** You can specify either the **Cutting process** or the **Cutting machine** field.

- **Bending machine**
  Defines the bending machine.
- **Cutting gas**
  Defines the cutting gas.
- **Abrasive**
  Defines the abrasive for the cutting process.

> ☞ **Note:** For waterjet cutting machines only.

What each button does:

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Adding a table row

Use the buttons at the bottom left edge to add, delete or copy table rows.

-  Adds a new table row to the current form.
-  Deletes the selected table row in the current form.
-  Activates editing of the selected field.

Prioritizing a table row

Use the arrow buttons at the bottom edge to move table rows and organize them in order of priority. The system uses the attributes of the first row in which the search criteria match the key criteria.

-  Moves the table row one priority up.
-  Moves the table row one priority down.



Tip: You can define overlapping thickness ranges in table rows. In this case, the table row with the higher priority is used.

Importing and exporting table rows

Use these buttons to exchange single rows or entire tables between multiple BySoft CAM installations.

-  Exports the highlighted table rows or the entire table content.
-  Imports an existing rules file based on certain options.

Copying a table row

Use these buttons to copy table rows to other tables with the same dependencies.

-  Copies the selected table row to the clipboard.
-  Pastes the table row from the clipboard into another table.

Generate key criteria

-  Launches the **Generate key criteria** dialog to automatically create table rows with the settings for the selected option.

Test settings

The test settings for the created tables can be defined here and applied to existing tables.

-  This button applies the cutting or bending machine settings to the table. The result from the test will be shown in the **Result** area.

  The system searches all tables for the first table row that matches all key criteria. Empty key criteria are ignored.

After running the test, use the two buttons to jump to functions with a matching table row.

- Shows the setting for the previous match.
-  Shows the setting for the next match.



Note: The functions with a match are indicated by a green dot in the selection area. The first matching row is highlighted in the table view for the selected function. Its attributes are applied to new parts or jobs.

Result

Shows the result of the table test. Matching values will be highlighted and shown. The attributes of the default row are applied to all undefined attributes in table rows.

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4.3.1 Cutting Wizard configuration

How do I open this dialog box?

Select: **Modules > Administration > Administrator > Settings Manager > Cutting Wizard**

This dialog is used to automatically import the Cutting Wizard settings for the selected machine. You can only import settings for machines with CO_2 laser and waterjet cutting machines.

Machines

Select all machines for importing in this section. The **Cutting Wizard filter** column displays the name of the Cutting Wizard for the appropriate machine.

Note: Only CO^2 laser cutting machines and waterjet cutting machines are displayed for which a Cutting Wizard exists.

Tip: Press the keyboard combination **CTRL+A** to select all machines.

Grouping

This section is used to define how to use the material groups for Cutting Wizard grouping.

- **Include material groups**

  This option uses material groups for Cutting Wizard grouping.

  Attention! If materials in the same material group have different Cutting Wizards, the grouping is only performed for the first material in this group. The other materials are inserted as separate rows with the appropriate material criteria.

- **Combine materials**

  This option combines all materials with the same Cutting Wizard in one row (`DC01;DD11`).

Mode

This section is used to define how to perform migration.

- **Add**

  This option adds all data to existing rows.

  > **Note:** Use this option to perform migration when you have added new materials.

- **Overwrite**

  This option overwrites all existing rows of CO_2 laser cutting machines and waterjet cutting machines.

Data backup

You can enter a path to save a backup file of the current Cutting Wizard configuration here.

- **Folder**

  This option defines the folder in which to save the backup file.

  The file name contains the date and time to identify the time of migration. This file contains all migrated rows from the Cutting Wizard.

  > **Note:** The export is always created before the migration process.

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4.3.2 Generate key criteria

How do I open this dialog box?

Select: **Modules > Administration > Administrator > Settings Manager**

This dialog is used to automatically generate table rows for the selected key criteria. There are several different settings for each key criterion. You can only define one setting at a time for each key criterion.

After pressing **OK** to confirm, the program automatically generates the table entries.



Note: The number of table rows depends on the selected settings. One row is created for each possible combination of selected settings.

4.3.3 Import settings

How do I open this dialog box?

Select: **Modules > Administration > Administrator > Settings Manager**

Use this dialog to import Settings Manager settings for the selected function from another BySoft CAM installation. The results of import are displayed in another dialog.

- **Import file**

  Path for the import file (rules file).

- **Add**

  Adds all rows below the existing rows

- **Merge**

  When merging, the rows to import are verified as follows:

  - If the key criteria do not exist in the target table, the row is appended to the end of the table
  - If the key criteria exist in the target table and the **Overwrite** option is not enabled, the row is not added.
  - If the key criteria exist in the target table and the **Overwrite** option is enabled, the options in the row of the target table are overwritten.

- **Replace all**

  Deletes all rows of the target table and adds the rows to import to the target table.

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4.3.4 Export settings

How do I open this dialog box?

Select: **Modules > Administration > Administrator > Settings Manager**

Use this dialog to export Settings Manager settings for another BySoft CAM installation.

- **Export file**

  Defines the export path for the rules file. The file name is generated automatically from the technology and function module.

- **All settings**

  Exports all rows of the selected function.

- **Selected**

  Exports all selected rows of the selected function.

4.4 API import configuration

Where do I find this form?

Select: **Modules > Administration > Administrator**

The settings in this dialog can be used to create different API import configurations.

Available entries

This section is used to create different API import configurations.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

API import configuration

In this section you enter details for the name, description, and additional information of the API import configuration.

What each field does:

- **Name**
  Specifies the name of the API import configuration.

  > **Note:** The name is essential for the program to be able to save the record.

- **Description**
  You can enter a more detailed description of the current object in this field.
- **Info 1–8**
  Extra information about the current object can be entered in these fields.

Geometry

Scaling factor

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This option specifies the scaling of the geometry elements.

Example:

- **0.5** = half the size
- **1** = no scaling
- **2** = twice the size

Contour tolerance

Using this option, several contours which lie at a certain distance from one another can be joined automatically to form a common contour. The contour tolerance causes all contours for which the distance is smaller than or equal to the value specified in the **Contour tolerance** field to be joined together.

Reducing the default value lowers the recognition rate. Increasing this value raises the recognition rate.

Origin

This option moves the bottom left corner of the bounding rectangle of each new part to the origin.

Delete duplicate elements

This option deletes duplicate drawing elements in the drawing. It always deletes the smaller element or element of equal size.

Automatic rotation

This option allows you to rotate parts during import by 90° based on their part geometry. The important factor is the part's bounding rectangle. The following attributes are available:

- **No rotation**
  Part geometries are imported without any rotation.
- **Rotate part onto wider side**
  If the part is higher than it is wide, it is rotated 90° clockwise.
- **Rotate part onto narrower side**
  If the part is wider than it is high, it is rotated 90° counterclockwise.

Filters for geometry selection

This tab displays all the defined filters together with the assigned technology options. You can create new filters using the **Add** button. Any number of filters can be defined.

Mode

These options can be used to include or exclude defined filters for importing layers.



Note: All specified filters can only be included or excluded!

- **Include**

  All specified layers will be imported in one part. Layers that are not specified are ignored.

- **Exclude**

  All specified layers in a part are ignored. Layers that are not specified will be imported.

What each button does:

- **Add**

  Opens the **Add geometry filter** dialog box in which you can create a new filter.

- **Edit**

  Opens the **Edit geometry filter** dialog box in which you can edit the selected filter.

- **Delete**

  Deletes the selected filter.

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4.5 DXF configuration export

Where do I find this form?

Select: **Modules > Administration > Administrator**

Use the settings in this form to assign specific layers to the part geometries during DXF export using the **Export** function *(See: ›Export part‹ Page 336)*.

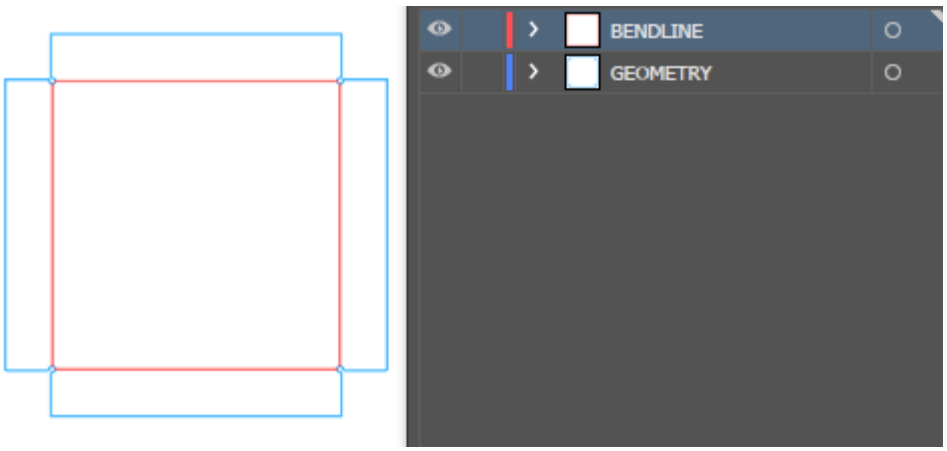


Fig. 4-62: Exported DXF file with the “bending line” and “geometry” layers

Available entries

This section is used to create various layer filters for DXF export.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

Export configuration

In this section you enter details for the name, description, and additional information of the DXF configuration.

What each field does:

- **Name**
  Specifies the name of the DXF configuration.

  ☞ **Note:** The name is essential for the program to be able to save the record.

- **Description**
  You can enter a more detailed description of the current object in this field.
- **Info 1–8**
  Extra information about the current object can be entered in these fields.

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Layer filters

All layer filters which you have created are displayed in this section. Use the **Add** button to create a new filter *(See: ›Add layer‹ Page 221)*. You can use the arrow keys to move the filters in the order of their application during import.

What each column means:

- **Type**

  Indicates the type of filter used (e.g.: **cutting technology** / **geometry**). If both filter types are selected, the **Combined** type is displayed.

- **Name**

  Specifies the name of the layer filter (e.g.: ***Layer Geometry***).

- **Filter**

  Specifies the filter attribute for the layer filter. If both filter types are activated, the attributes for both filter types are displayed (e.g.: ***Cutting_Normal contour***).

What each button does:

- **Add**

  Opens the **Add Layer** dialog to create a new layer filter.

- **Edit**

  Opens the selected layer filter in the **Add Layer** dialog for editing.

- **Delete**

  Deletes the selected layer filter from the table.

- 

  Moves the selected layer filter up one place

- 

  Moves the selected layer filter down one place

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4.5.1 Add layer



How do I open this dialog box?

Select: **Modules > Administration > Administrator > Export DXF configuration > Layer filter**

This dialog is used to add or edit a layer of an export configuration. Up to two filter types with specific attributes can be assigned to the layer.

- **Name**

  Specifies the name of the layer.

- **Cutting technology**

  Determines the cutting technology that will be assigned to the layer during export.

  > **Note:** Only one or all cutting technologies can be assigned to the layer at a time.

- **Geometry**

  Determines the geometry elements that are assigned to the layer during export.

  > **Note:** Only one or all geometry elements can be assigned to the layer at a time.

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4.6 Cost tables

Where do I find this form?

Select: **Modules > Administration > Administrator > System data**

This form contains the details of the cost calculation that are used as a template in the Cost Calculator module.

Tip: In the **Available entries** area create a separate cost table for each currency required. This allows you to create cost calculations in your own currency or a foreign currency.

Example:

- *(See: ›Creating a cost table‹ Page 1347)*
- *(See: ›Generating operation for programming‹ Page 1348)*
- *(See: ›Generating operation for drilling‹ Page 1349)*

What each field does:

Name

This name is displayed in the **Available entries** pane and in all the selection fields for the machine throughout the software.

Note: The name is essential for the program to be able to save the record.

Description

You can enter a more detailed description of the current object in this field. (E.g., laser power or machine format).

Info 1–8

Extra information about the current object can be entered in these fields.

Machine type

The machine type that applies to the selected machine can be chosen in this field. The machine type determines what options are available, the cutting process and the sheet format for the machine.

System of units

The system of units for the current object is defined in this field (mm/inches)

- **Currency symbol**

  Enter the currency symbol in this box. This symbol is used for all currencies in the cost calculation.

- **Positioning path**

  The option takes the positioning paths into account for the calculation of costs. The option is enabled by default.

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4.6.1 Operations

How do I open this tab?

Select: **Modules > Administration > Administrator > System data > Cost tables**

This tab contains details of the various operations. This information is used for the cost calculations in the Cost Calculator module.

Operation

In this area you can enter all the operations that are needed for producing parts. At least one entry must exist in order to use the cost table.

Because any name can be chosen for each operation, the different types of operation are color-coded for easier recognition.

Color	Operation type
	Cutting
	Bending
	Set-up
	User-defined
	Area/Weight

Note: The **User-defined** operation type lets you customize all the fields in the table.

Tip: You can enter a name of your choice in the **Name** field for each operation. For the **Cutting** and **Bend** operations, enter in this field the name of the machine on which the operation shall be used.

Adding a new operation

1. Select the  button.
2. Select the operation type you require.
3. Press **OK** to confirm the selection.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

What each field does:

- **Name**
  Name designating the operation.
- **Label 1-3**
  Label for the corresponding operation units.
- **Unit 1-3**
  Unit for the appropriate operation.

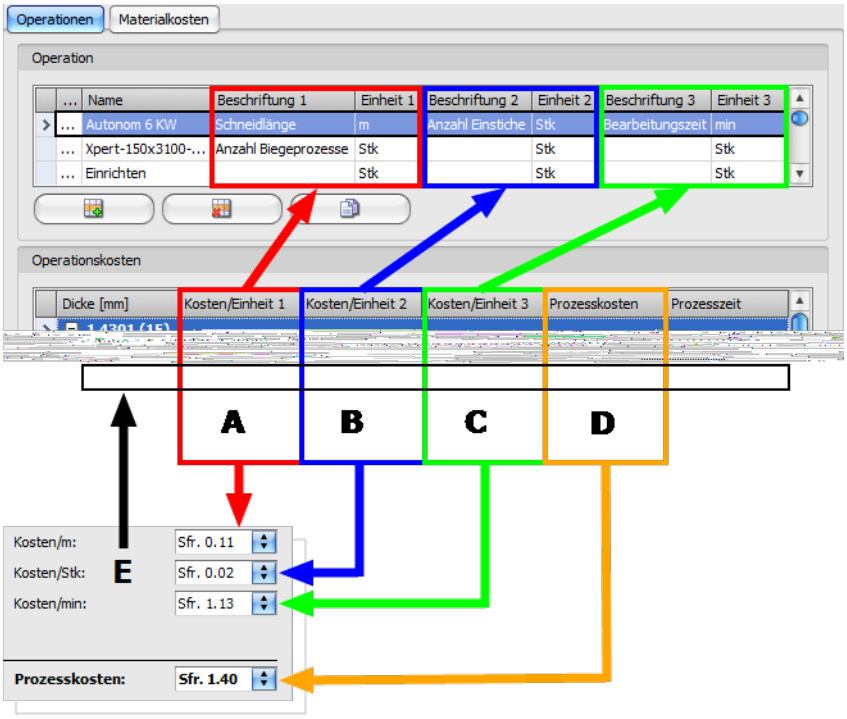
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Operation costs

The operation costs are assigned to the relevant operation in the **Operations** area. The operation costs are needed for calculating the process costs.

Assigning the operation costs to an operation

This graphic shows the assignment of the **Costs/unit 1-3** fields to the fields in the **Operation** area and in the Cost Calculator module.



A	Costs for cutting per meter
B	Costs per piercing cut
C	Costs for the machining time per minute
D	Process costs
E	Shows the costs for the selected thickness (1 mm) in the Cost Calculator module

Editing table values in Excel

You can also edit the table values in Excel and then import them back into the cost table.

Attention! Importing the table is permanent - it cannot be undone.

1. Click the table header in the **Operation costs** area
2. Select the **Export to MS Excel** function from the shortcut menu.
3. In the dialog box, specify the file name and location and confirm with the **Save** button.
4. Open the file in Excel and modify the values or add new ones.
5. Save the file.
6. Select the  button in the cost table.
7. Select the modified Excel file in the dialog box and confirm your selection with **Open**. The Excel file is imported into BySoft CAM and overwrites the current operating costs in the cost table.

Adding a new material for the operation costs

1. Select the  button.
2. Select the material you require.
3. Press **OK** to confirm the selection.
4. Enter the figures for the thickness and costs per unit in the first row.
5. Use the  button to copy the row as many times as you need thicknesses for the material.
6. Adjust the figures for the thickness and costs per unit in the rows you have added.

Tip: You can also use the Automatic function to add a new material *(See: ›Creating a cost table‹ Page 1347)*.

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What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.
	Opens all the closed Operation costs.
	Closes all the opened Operation costs.
	Opens the dialog box for importing operation costs from an Excel table.

What each field does:

- **up to and including thickness**

  Added material with the material thicknesses used. The entry for thickness indicates the threshold – relative to the previous thickness - up to which operation costs are valid.

  Example thicknesses:

  - Thickness = 5mm
  - Previous thickness = 3mm

  -> Operation costs for a thickness of **5** mm are used for thicknesses of **3.1 – 5** mm.

- **Costs/unit 1-3**

  Costs per unit 1-3 for the selected operation and the relevant material thickness.

- **Process costs**

  Used as a default value when a geometry is still not available for a part in the cost calculation, which would prevent the process costs being calculated automatically.

  ☞ **Note:** Calculated process costs in the Cost Calculator overwrite the default value from the cost table.

- **Optional time**

  Additional times that accrue for the machining process. The time is calculated per part in the cost calculation. It has no impact on the costs and is used only for orientation.

Examples of user-defined operation costs

- *(See: ›Generating operation for programming‹ Page 1348)*
- *(See: ›Generating operation for drilling‹ Page 1349)*

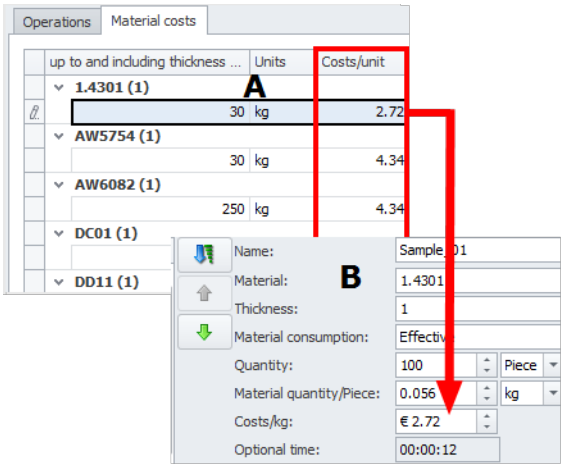
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4.6.2 Material costs

How do I open this tab?

Select: **Modules > Administration > Administrator > System data > Cost tables**

This tab contains details of the material costs. The costs are defined per unit for each thickness of the material. The material costs are needed for calculating the costs per part in the Cost Calculator.



A	Costs per unit for the relevant thickness and material.
B	Display of the costs per unit in the Cost Calculator

Adding a new material for the material costs

1. Select the  button.
2. Select the material you require.
3. Press **OK** to confirm the selection.
4. Enter the figures for the thickness and costs per unit in the first row.
5. Use the  button to copy the row as many times as you need thicknesses for the material.
6. Adjust the figures for the thickness and costs per unit in the rows you have added.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.
	Opens the closed Material costs.
	Closes the opened Material costs.
	Opens the dialog box for importing material costs from an Excel table.

What each field does:

- **Material/Thickness**
  Added material with the material thicknesses used.
- **Unit**
  Units for the relevant material thickness.
- **Costs/unit**
  Costs per unit for the relevant material thickness.
- **Waste target**
  The icon for waste in Part Nester has the appropriate color based on waste settings.

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4.6.3 Select operation type

How do I open this dialog box?

Select: **Modules > Administration > Administrator > System data > Cost tables > Operations > Operations**

In this dialog box you can select the operation type for the cost table. The following operation types are available for selection:

Operation	Color	Description
Cutting		Used for the cutting costs of laser and water jet cutting machines.
Bending		Used for the bending costs of parts.
Set-up		Used for the setup costs in the machining processing.
User-defined		Used for user-defined costs.
Area/Weight		Used to define costs by area or weight.

4.6.4 Select material

How do I open this dialog box?

Select: **Modules > Administration > Administrator > System data > Cost tables**

In this dialog box you can select the material for the operation and material costs, and then add them to the relevant area by pressing **OK**.



Tip: If a material you require is not available, you will need to create it as a new material in the **Material** form of the Administrator module. After restarting the Administrator, the new material is available to select.

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4.6.5 Cost tables

How do I open this dialog box?

Select: **Modules > Administration > Administrator > System data > Cost tables > Edit**

In this dialog box, the operation and material costs for each material and thickness available in BySoft CAM can be generated automatically for the selected cost table.

The cost tables are needed as templates for each cost calculation in the Cost Calculator module.



Note: The values for the material and operation costs apply to all the selected materials and thicknesses. These can be specified or modified later in the cost table.



Tip: Where material costs are the same for all thicknesses, import each material separately so that the costs only need to be entered once for all thicknesses of a material.

Creating materials for an operation

1. Select the materials and the associated thicknesses.
2. Enter for the material the material costs for the selected units (only if each material is imported separately).
3. Select the operation to which the material costs apply. If there is no operation available, you must create the operation you require from scratch in the **Operations** tab of the cost table.
4. Select the mode for adopting the specified values and adopt the values using **OK**.

Material

Shows all the materials available for selection in the cutting tables.

Thickness

Shows all the thicknesses of the selected materials available for selection in the cutting tables.

Operation

Shows all the operations available for selection in the cost table.

Material costs

The values in these fields define the costs per unit for the selected material and selected thickness. This information is displayed in the cost table in the **Material costs** tab.

What each field does:

- **Unit**
  Defines the unit for the material.
- **Costs/unit**
  Defines the costs per unit of the material.

Operation costs

The values in these fields define the operation costs for the selected operation. This information is displayed in the cost table in the **Operations** -> **Operation costs** tab.

What each field does:

- **Process costs**
  States the process costs for the operation.
- **Process time**
  States the process time for the operation.
- **Costs/unit 1-3**
  States the costs per unit 1-3.

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Mode

The mode specifies how the values are adopted in the cost table.

What each option does:

- **Add**

  This option adopts only those values for material and operation costs that are still missing from the cost table.

  Note: Existing values are not overwritten.

  Tip: Use this option to add new thicknesses for a material or to insert missing values.

- **Overwrite**

  This option means that all the values specified for material and operation costs are adopted in the cost table.

  Note: Existing values are overwritten.

  Tip: Use this option to overwrite the cost tables with new values for material and operation costs.

4.7 Report templates



Where do I find this form?

Select: **Modules > Administration > Administrator > System data**

This form displays all the report templates available for the element. You can edit report templates, create new templates or delete existing ones. Use Report Designer to edit report templates or create new ones. A default template is already provided for each print element.



Tip: You can create your own Report templates by using Report Designer to customize the template to suit your own requirements. Report Designer can be started from the BySoft CAM installation directory. Use the button in the **Available entries** area to copy a template.

What each button does:

Icon	Description
	Adds a new object to the current form.
	Deletes the selected object from the current form.
	Adds a copy of the selected object to the current form.

What each field does:

- **Name**

  The name of the report template must be specified in this field. This name is displayed in the **Available entries** pane and in all the list boxes for the report template throughout the software.

  Note: If a new template is created, it is essential to enter a name so that the software can save the element.

- **Description**

  You can enter a more detailed description of the current object in this field.

- **Info 1–8**

  Extra information about the current object can be entered in these fields.

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5 Module: Object Manager

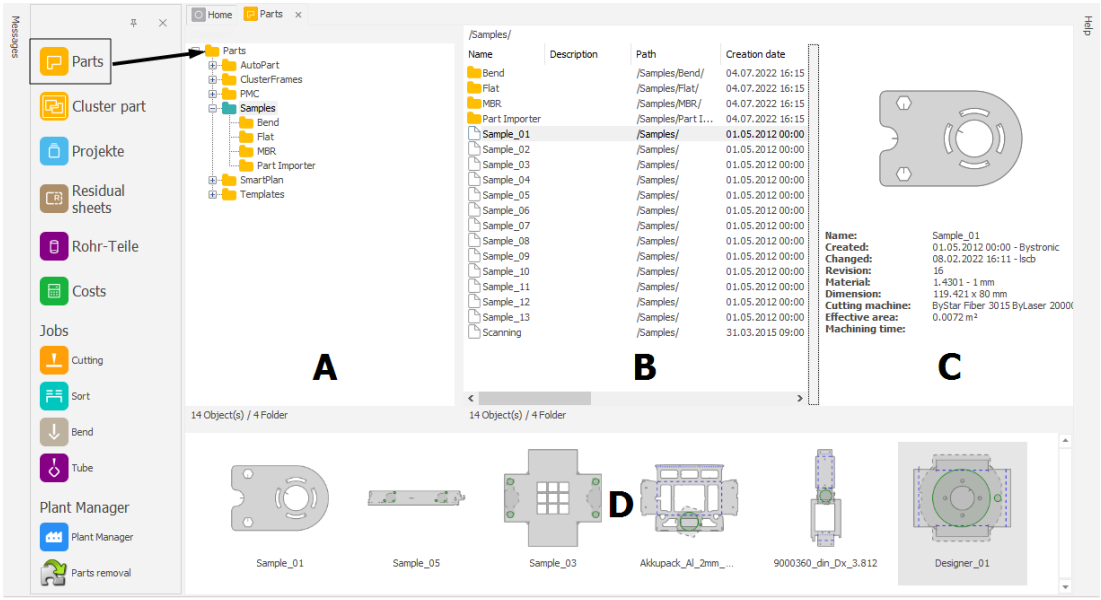
How do I open this view?

Select: **Module**

In Object Manager you can manage all objects and structures of BySoft CAM. The layout of Object Manager is similar to Windows Explorer.

Selecting a module in the **Modules** dialog displays the respective structure and content in several windows in the workspace.

Layout of windows in Object Manager



Key:

A	Window displaying the folder structure of the selected module.
B	Window for displaying the saved objects of the selected module. The type of display can be customized in the **View** menu using the **View** function *(See: ›View‹ Page 267)*.
C	Window for displaying information about the selected object. The window can be shown or hidden in the **View** menu using the **Information pane** function *(See: ›Information pane‹ Page 269)*.
D	Window for displaying the last selected objects.

Like in Windows Explorer, you can use keyboard commands to copy (**Ctrl+C**), cut (**Ctrl+X**) and paste (**Ctrl+V**) selected objects in the Object Manager too.

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Several windows are available for displaying a tidy view of the object structure and preview graphics. The view can be specified separately for each window. Use the **Tab** key to toggle between the two windows.

Note: The object data is not saved in a database but as a file in the Windows file system. This means that the files can also be accessed via the standard Windows Explorer tool.

Tip: Display the folder structure in the left window and the contents of the selected folder in the right window. To do this, select **Details** view for the right window from the **View** menu. To synchronize both windows, run the **Join windows function.**

Tip: Click on an area of the header to call up the **Shortcuts** function in the context menu. The **Shortcuts** dialog can be used to define keyboard shortcuts for all functions of the current module.

Arrangement of the folder structures

In principle you can design any folder structure you wish for each object. In practice, however, it is best to use the same folder structures for parts and jobs to make it easier to navigate the two databases.

Tip: If you need to create a new object structure, arranging the folders by customer as the top level has proved useful in many cases.

What each column means:

Name

Shows the name of the object. The name is used for unique identification of an object.

Description

Shows a more detailed description of the active object.

Creation date

Shows the date on which the object was created.

Created by

Shows the user who created the object.

Modification date

Shows the date on which the object was last modified.

Modified by

Shows the user who made the last change to the object.

Revision

Shows the revisions to the object.

Attributes

Shows which attributes are set for the object.

r = write protected

h = hidden

Info 1-3

Displays additional information about the respective jobs.

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5.1 Group: File

Where can I find these functions?

Select: **Module > Parts**

This group contains the functions that can be used to define the structure for each object type in Object Manager. You can also import, export or delete objects from here.

5.1.1 New residual sheet



Where do I find the function?

Select: **Module > File**

This function creates a new residual sheet in the selected folder. Define the material, dimensions, quantity and storage location for the residual sheet in the **New Residual Sheet** dialog.

In order to be able to perform the function, the **Residual sheets** function must be selected in the **Navigation** section.

Note: This function is only suitable for rectangular residual sheets.

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5.1.2 Open



Where do I find the function?

Select: **Module > File**

This function opens all selected objects in the appropriate module. Folders cannot be opened.



Note: This function can only be used when an object has been selected.

5.1.3 Rename



Where do I find the function?

Select: **Module > File**

This function lets you rename the selected object or folder. After entering the new name, the change must be confirmed with **Enter**.

 **Note:** This function can only be used when an object has been selected.

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5.1.4 Edit attributes



Where do I find the function?

Select: **Module > File**

This function can be used to assign certain object attributes to an object.

You change an object attribute by running the function and then enabling or disabling the relevant attribute in the **Edit attributes** dialog box.

Object attributes

- **Write-protected**

  Prevents objects from being edited in the Part Editor or Part Nester. This means that the object can only be opened as a copy and edited in this form.

5.1.5 Copy



Where do I find the function?

Select: **Module > File**

This function lets you copy one or more objects into another folder. If more than one object is selected, a prompt dialog box is displayed before copying.



Note: This function can only be used when at least one object has been selected and the **Join windows** function is disabled.



Tip: Objects can also be copied into another folder by dragging with the mouse.

Copying an object into another folder

1. In the left-hand window, select the folder into which you wish to copy the object.
2. In the right-hand window, select the object that you wish to copy.
3. Select the **Copy** function from the Ribbon. The selected object is copied into the selected folder.

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5.1.6 Move



Where do I find the function?

Select: **Module > File**

This function can be used to move objects or entire folders into the selected folder in the second window.



Note: The **Join windows** function must be disabled. Otherwise the move function cannot be performed because the source and destination folders are always the same. A message to this effect is displayed.

5.1.7 New folder



Where do I find the function?

Select: **Module > File**

This function creates a new folder below the current level.

You can create a new folder by running the function and entering a name for the folder in the **New folder** dialog box.

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5.1.8 Delete



Where do I find the function?

Select: **Module > File**

This function can be used to delete an object or folder.

Before a folder can be deleted, it must be emptied of all objects by deleting or moving them. This avoids losing all the objects in an important folder if it is accidentally deleted.



Note: You can also delete several folders at once, except in the **Tree view**.



Attention! Deleting objects and folders is permanent - it cannot be undone. A Prompt dialog box is displayed before deletion.

5.1.9 Properties



How do I open this dialog box?

Select: **Module > File**

In this dialog you can enter additional descriptions and information for the selected object in fields 1–8.

The entries for the object are displayed in the appropriate module under the **Properties** function.

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5.1.10 Residual sheet properties



How do I open this dialog box?

Select: **Module > File**

In this dialog you can configure all settings for the selected residual sheet.

- **Number of sheets**
  Defines the number of residual sheets with the same geometry.
- **Storage location**
  Defines the storage location of the residual sheet.
- **Product number**
  Indicates the number of the product to which the residual sheet belongs.
- **Batch number**
  Indicates the batch number of the residual sheet.

5.1.11 Print



Where do I find the function?

Select: **Modules > File**

This function can be used to print the working documents for the selected object.

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5.2 Group: Navigation

Where can I find these functions?

Select: **Module > Parts**

This group contains all the functions that can be used to navigate the Object Manager folder tree.

You can use the **Parts**, **Parts Jobs** and **Residual sheets** buttons to display the corresponding objects in Object Manager.

5.2.1 Refresh



Where do I find the function?

Select: **Module > Navigation**

This function can be used to refresh the display of the objects by reloading the contents of the folders. Restarting the Object Manager will also reload the contents of all folders.



Note: Use this function if object files have been added or deleted outside the Object Manager.

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5.2.2 Back



Where do I find the function?

Select: **Module > Navigation**

This function takes you to the previous view. You need to have changed the view at least once before this function can be used.

5.2.3 Forward



Where do I find the function?

Select: **Module > Navigation**

This function takes you to the next view. You need to have performed the **Back** function at least once before the **Forward** function can be used.

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5.2.4 Upwards



Where do I find the function?

Select: **Module > Navigation**

This function takes you one level up in the tree view. This function is only enabled when the root folder is not selected.

5.2.5 Root folder



Where do I find the function?

Select: **Module > Navigation**

This function is used to switch to the root folder in the topmost level.

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5.2.6 Right-hand folder



Where do I find the function?

Select: **Module > Navigation**

This function displays the contents of the selected folder in the left-hand window, and the subfolders in the right-hand window.

5.2.7 Left-hand folder



Where do I find the function?

Select: **Module > Navigation**

This function displays the contents of the selected folder in the right-hand window, and the subfolders in the left-hand window.

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5.3 Group: Selection

Where can I find these functions?

Select: **Module > Parts**

This group contains all the functions that can be used to search for and select objects in Object Manager.

5.3.1 Find



Where do I find the function?

Select: **Modules > View**

This function can be used to search for objects in Object Manager. Before running the search, you need to enable the required search option.

Search criteria

You can search for one or more objects by entering a text in the top field. Further search options for locating objects can be specified in the lower area.

You can use a list box to restrict the search according to certain criteria.

- **Entire text**
  The program searches for a text that matches the character string exactly.
- **Start of text**
  The program searches for a text that starts with this character string.
- **Part of text:**
  The program searches for a text that contains this specified character string.

In the middle section you can search by creation date, changes and revisions. You can limit the search using the appropriate calendars or input boxes.

- **Write-protected**
  Search for objects with the attribute **write-protected**.
- **Duplicate names**
  Search for objects with duplicate names.
- **Search for folders**
  Search for names of folders. All other search options are disabled.



Note: The program saves the configured search criteria and uses them again next time you open the Search dialog.



Tip:
Search with multiple search criteria

Several search criteria can be entered in each field. They must be separated by a semicolon without a space in between.

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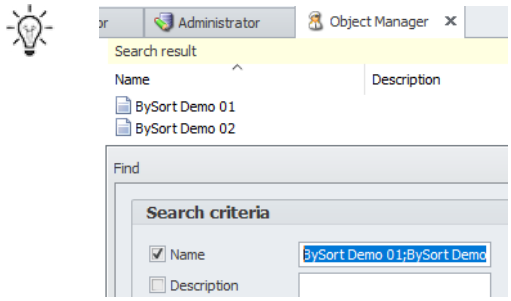


Fig. 5-1: Search with multiple search criteria

Search with wildcards

You can search for objects using the * wildcard in all fields (**A**). To do so, the field criterion **Part of text** (**B**) must be selected.

Example:

- **Sample*2mm***

  This searches for files that start with the designation **Sample** and file names containing the string **2mm**. The wildcard after 2mm means that characters must follow.

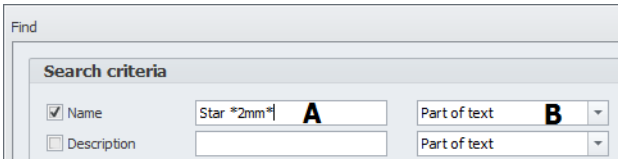


Fig. 5-2: Search with wildcards

5.3.2 Select all



Where do I find the function?

Select: **Modules > View**

This function selects all the objects and folders in the active window.

 **Note:** This function cannot be used in the Tree view.

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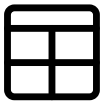
5.4 Group: View

Where can I find these functions?

Select: **Module > View**

This group contains all the functions that can be used to set the viewing options for objects in the middle window.

5.4.1 View



Where do I find the function?

Select: **Modules > View**

This function lets you customize the view for the objects in the Object Manager windows to suit your needs. Clicking on the **View** function opens up various view options.

View options

- **Icons**
  The objects are displayed in a horizontal layout using large icons.
- **List**
  The objects are displayed in a vertical layout using small icons.
- **Details**
  The objects are displayed in a vertical layout with additional information.

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5.4.2 Join windows



Where do I find the function?

Select: **Modules > View**

This function can be used to join together the two windows. If you change the folder in one window, the adjacent window will display the contents of this same folder.

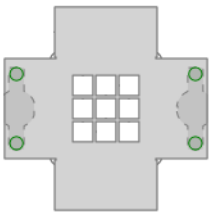
5.4.3 Information pane

Where do I find the function?

Select: **Modules > View**

This function can be used to display in the active window an Information pane containing extra information on the selected object or folder.

- If an object is selected, a preview graphic of the object is displayed together with additional information.
- If a folder is selected, extra details about the folder are displayed.



Name:	Sample_03
Created:	01.05.2012 00:00 - Bystronic
Changed:	06.07.2022 11:47 - Lbeu
Revision:	13
Material:	DC01 - 1 mm
Dimension:	791.76 x 791.76 mm
Cutting machine:	ByStar Fiber 3015 ByLaser 20000
Effective area:	0.4118 m²
Machining time:	00:00:25

Fig. 5-3: Displaying the Information pane for parts

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5.4.4 Last used objects



Where do I find the function?

Select: **Modules > View**

This function displays all last used objects of the selected module are displayed as a graphic under Object Explorer.



Tip: Double-click on the graphic to open the selected object in the appropriate module.

5.4.5 Display folder



Where do I find the function?

Select: **Modules > View**

This function displays the subfolders of the selected folder in the right window. If the **Connect window** option is selected, the objects saved under it will be displayed in the right window.

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5.5 Group: Options

Where can I find these functions?

Select: **Module > Archive**

This group contains the functions that can be used to define the structure for each object type in Object Manager. You can also import, export or delete objects from here.

5.5.1 Import



Where do I find the function?

Select: **Modules -> Archive -> Options**

This function can be used to import objects of the current object type into the selected folder.

Importing objects

1. Select the module to import objects to (e.g. **Parts**).
2. Select the folder into which the objects will be imported.
3. Click the **Import** function in the **Archive** menu.
4. In the **Import folder** field, select the folder containing the objects you wish to import.
5. Under **Select objects**, select the objects that you wish to import.
6. Press **OK** to confirm the selection. The objects are imported into the current folder.

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5.5.2 Archive



Where do I find the function?

Select: **Modules > Archive > Options**

This function can be used to export objects of the current object type into a specified folder for archiving purposes. Existing links to the material and the machine and to the nesting settings are retained.



Attention! The file name of archived objects must not be changed outside BySoft CAM. because this will mean that they cannot be imported again. An error message to this effect is displayed.

- **Export folder**
  File explorer for selecting the folder in which to save objects.
- **Export format**
  Selection box to define the format in which to archive parts.



Note: Displayed only when archiving parts and residual sheets.

- **Overwrite objects with same name**
  Overwrites objects with the same name in the designated folder.
- **Selected objects**
  Selection area for objects to be archived.

Archiving parts

1. Select the **Parts** module ().
2. Select the folder containing the parts to be archived.
3. Click the **Archive** function.
4. In the **Export folder** field, select the folder in which to archive the parts.
5. In the **Export format** field, select the export format in which you want to save the parts.
6. Enable the option **Overwrite objects of same name** if you want to overwrite parts that have the same name.
7. Under **Selected objects**, select the parts to be archived.
8. Press **OK** to confirm the selection. The parts are archived in the specified folder.



Note: Residual sheets can be archived in the same way as parts.

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Archiving jobs

1. Select the **Cutting** job module ( ).
2. Select the folder containing the jobs to be archived.
3. Click the **Archive** function.
4. In the **Export folder** field, select the folder in which the jobs are to be archived.
5. Enable the option **Overwrite objects of same name** if you want to overwrite jobs that have the same name.
6. Under **Selected objects**, select the jobs to be archived.
7. Press **OK** to confirm the selection. The jobs are archived in the specified folder.

☞ **Note:** Tubular jobs, cost calculations, bending jobs, cluster parts, and sort jobs can be archived in the same way as jobs.

5.5.3 MS Excel

Where do I find the function?

Select: **Modules > Archive > Options**

This function can be used to export the entries of the selected objects to Microsoft Excel.

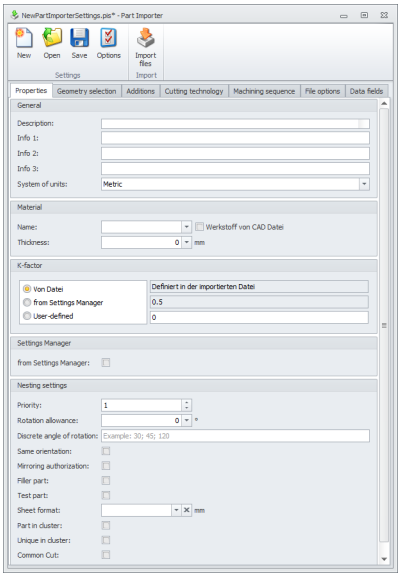
Exporting objects

1. Select the entries of parts to export to MS Excel.
2. Click the **MS Excel** function.
3. Select the file name and path for exporting the Excel file in the **Save as** dialog.
4. Click **Save** to confirm. The file is exported to the selected folder.

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6 Module: Part Importer



You can use this program to import geometry files into BySoft CAM on the basis of settings made in an import file. The settings are made in sequence in the relevant tabs.

During import, the geometry data can be filtered and assigned technology properties, and the machining sequence and lead-ins/lead-outs can be set.

Note: The entries and settings in the tabs are applied to all imported parts and geometry files.

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Import parts

The example shows the basic procedure for importing parts or geometry files. It does not include specific settings for geometry selection for DXF files.

1. Run the Part Importer application from the program folder **Bystronic** > **BySoft CAM** > **Tools**.
2. In the **Properties** tab, specify the system of units and the material including relevant thickness.
3. In the **Cutting technology** tab, set the cutting process and machine that will be used to cut the parts.
4. In the **Name** field, specify the applicable Cutting Wizard for the geometries. You can use the fields under the name to filter the cutting rule sets and so speed up selection.
5. Define the lead-in and lead-out for the parts.
6. In the **Machining sequence** tab, specify the sequence in which the contours of the parts shall be machined.
7. In the **File options** tab, specify the storage location for the parts in the BySoft CAM parts folder.
8. Select the **Import files** function.
9. Select the relevant files in the **Select import files** dialog box and confirm your selection with **Open**. All the selected files are imported.

DXF import via the command line

You can also import DXF files in batch mode via the command line. The command entered in the command line runs the Part Importer in the background, which then imports the DXF files into BySoft CAM.

If the command is entered in the command line from the BySoft CAM installation folder, there is no need to specify the path to the **PartImporter.exe** file. There a various ways of importing the DXF files.

Importing a single file

This example shows the command line for importing a single file.

- **Notation:** *PartImporter.exe -s=<SettingsFile> -file=<SourceFile>*
- **Example:** *"C:\Programme\Bystronic\BySoft CAM\PartImporter.exe" - s="C:\I-ImportData\Settings\DXFImporterSettings.pis" -file="C:\ImportData\CAD\1-06717.dxf"*

Importing files from a folder

This example shows the command line for importing all the files from a folder.

- **Notation:** *PartImporter.exe -s=<SettingsFile> -dir=<SourceDirectory>*
- **Example:** *"C:\Programme\Bystronic\BySoft CAM\PartImporter.exe" -s="C:\I-ImportData\Settings\DXFImporterSettings.pis" -dir="C:\ImportData\CAD"*

Importing files from a file list

This example shows the command line for importing files from various folders using a file list.

- **Notation:** *PartImporter.exe -s=<SettingsFile> -list=<SourceListFile>*
- **Example:** *"C:\Programme\Bystronic\BySoft CAM\PartImporter.exe" -s="C:\I-ImportData\Settings\DXFImporterSettings.pis" -list-t="C:\ImportData\DXFImportList.txt"*



Fig. 6-1: Path details for DXF files in a file list

Note: The quotation marks in the path details only need to be inserted if the folder name contains spaces.

Tip: To avoid needing to re-enter the command line every time, you can save the command line in a batch file (*.bat).

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6.1 Settings

This group contains all the settings for importing parts.

6.1.1 New import file



Where do I find the function?

Select: **Part Importer > Settings**

This function uses a default configuration to create a new configuration file for importing geometry data. You can customize the configuration to suit your personal requirements and then save it for future use.

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6.1.2 Open import file



Where do I find the function?

Select: **Part Importer > Settings**

This function opens an existing configuration file. In Explorer, select the import file and confirm your selection with **OK**.

6.1.3 Save import file



Where do I find the function?

Select: **Part Importer > Settings**

This function saves the import file. If the file has not been saved before, a dialog box opens in which you can specify the file name for saving the file under.

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6.1.4 Options



Where do I find the function?

Select: **Part Importer > Settings**

This function saves the import file. If the file has not been saved before, a dialog box opens in which you can specify the file name for saving the file under.

6.2 Import

This group contains all the file functions for importing parts.

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6.2.1 Import files



Where do I find the function?

Select: **Part Importer > Import**

This function can be used to import parts from various file formats into BySoft CAM. It is also possible to import several parts at once.

To import the parts, select the relevant files in the **Select import files** dialog box and confirm your selection with **Open**. The files are imported into the BySoft CAM parts folder.

The parts can be imported from the following file formats:

- **AutoCAD DXF files (*.dxf):** *This is the format for importing certain elements from a DXF file (See: ›Extracting parts from a DXF file‹ Page 1389).*
- **AutoCAD DWG files (*.dwg):** *This is the format for importing certain elements from a DWG file.*
- **Bysoft 7 part files (*.part):** *This is the format for re-importing an exported BySoft CAM part.*
- **Bysoft DIN files (*.din):** *This format is used to import parts from Bysoft 5.3 or earlier.*
- **Bysoft6 Bypart files (*.bpd; *.zip):** *This is the format for importing the geometry and technology of a part from Bysoft6 (See: ›Importing a part from Bysoft6‹ Page 1388).*
- **DSTV-NC files (*.nc; *.nc1):** *This format can be used to import NC files as flat parts.*
- **Encapsulated PostScript files (*.eps):** *This format is used to import graphics in EPS format.*
- **MMC BYBEND 7 files (*.bpx):** *This is the format for re-importing a bending part exported from BySoft CAM.*
- **Raster Images:** *This format can be used to convert a bitmap image into a vector graphic.*
- **Step files (*.step; *.stp):** *This format can be used to import CAD files in step format.*

Supported elements for importing from DXF or DWG files

When performing a DXF or DWG import, only certain elements can be imported into BySoft CAM.

DXF/DWG elements supported in BySoft CAM

ARC	MTEXT (just single-line text)
CIRCLE	POLYLINE
ELLIPSE (converted into a contour)	SPLINE (converted into a contour)
INSERT	TEXT
LINE	VERTEX
LWPOLYLINE	

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6.3 Properties

Where do I find the function?

Select: **Part Importer > Settings**

This tab is where the basic properties of the parts are defined. The material and thickness must be defined as a minimum.

General

Quantity

This field determines the number of instances with which each part is imported. The number of instances is displayed in Part Editor in the **Part Properties** dialog.

Description

You can enter a more detailed description of the imported parts in this field.

Info 1–8

Additional information on the imported parts can be entered in these fields.

System of units

The system of units (mm/inch) for the imported parts is set in this field.

☞ **Note:** The parts can only be nested in jobs that use the same system of units.

Material

Name

This field displays the name of the material. Clicking on the arrow symbol on the right lets you select the material from a list.

Thickness

This field displays the thickness of the material. Clicking on the arrow symbol on the right lets you select a thickness from a list. All the thicknesses available for the material are displayed. If you require a different thickness, enter it in the field from the keyboard.

From Settings Manager

If this option is used, certain properties of the cutting technology and the machining sequence are determined by Settings Manager during import. The appropriate sections and tabs are disabled in Part Importer.

Requirements:

- The settings for the cutting technology in Settings Manager for Part Editor are complete.

Nesting settings

Priority

You can use this option to set the order in which the different parts are to be nested. Entering a priority is optional. Enter a number between 1 and 99 for the priority, Where 1 stands for highest priority and 99 for lowest priority. No priority is assigned if 0 is entered.



Attention! The use of different priorities does not usually result in optimum nesting of the cutting plan.



Note: This function is only used for automatic nesting.

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Rotation allowance

This field defines the increment of the angle of rotation in degrees. The part can be rotated during nesting through this angle.



Tip: A rotation allowance can be entered by clicking on the input field or by selecting a set angle value via the left arrow button.

Discrete angles of rotation

You can enter multiple angles of rotation in this box. The part can be rotated during nesting through this angle. The angles of rotation must be separated from each other by semicolons (**;**).



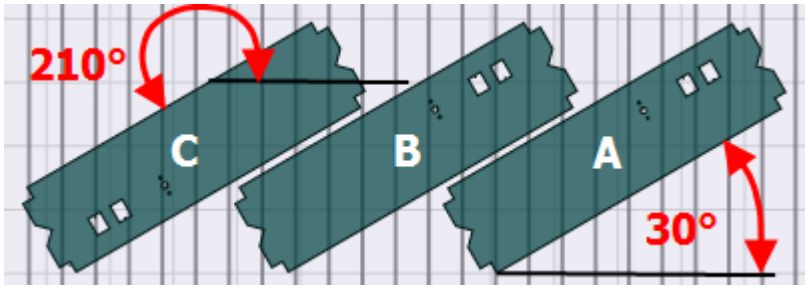
Tip: If this field contains angles of rotation, the angle of rotation in **Rotation allowance** is ignored (not for cluster nesting).

Same orientation

This option nests all instances of the part in the same orientation on the sheet.

Example:

If the part is nested with an angle of **30°** (**A**), the other instances can only be nested with the same angle (**B**) or with an angle of **210°**=180°+30° (**C**).



Application:

- Parts with same direction of grinding
- The correction value only needs to be entered on the bending machine for the first bending part. Because the alignment of the grain direction is the same, it is not necessary to enter a correction for all other parts.

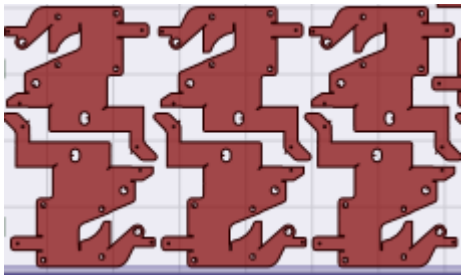


Fig. 6-2: Same orientation of a bending part on the sheet

Mirroring authorization

This option permits the part to be mirrored about the vertical axis during nesting.

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Filler part

Enable this option if you want to use this part to fill the areas leftover from nesting, or if the part is often needed in larger, unspecified quantities.

Test part

Flags the part as a test part. In Part Nester all instances of the part are nested as test parts *(See: ›Nesting a test part‹ Page 1375)*.

Sheet format

Use this option to assign an existing sheet format to the part. Depending on the size of the parts, this may help reduce sheet waste. If you nest the part on a different sheet format in Part Nester, the **Parts** tab displays a warning.

Part in cluster

This option specifies that the part can only be nested in a framed cluster *(See: ›Framed cluster‹ Page 1213)*.

Unique in cluster

This option specifies the part can only be nested in a cluster. It is not possible to nest it outside a cluster.

Take cutouts into account

These options determine whether or not to apply cutouts that were set on a part in Part Editor.

- **Default**

  If the part contains cutouts, they will be taken into account for nesting if the parameter is enabled in the nesting settings.

- **No**

  No cutouts are taken into account for nesting.

  Attention! The cutouts of a part are changed if the contour of another part is nested in a cutout. Based on the contour, the cutout is divided by the contour.

- **Yes**

  If the part contains cutouts, they are always taken into account for nesting.

  Note: The cutouts are taken into account even if the parameter is disabled in the nesting settings.

Common Cut

This option allows the combining of parallel contours during nesting based on certain criteria. These criteria are defined in the export settings for the pre-processor in the Part Nester module *(See: ›Common Cut‹ Page 1153)*.

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6.4 Geometry selection

This tab is where the import preferences are defined for importing the geometry elements. The **Filters for geometry selection** can be used to define various filter types.

If a geometry element matches the requirements of a filter, the element attributes of the filter are applied to the geometry element. If no filters are defined, all the geometry data is imported, but no technology is assigned to the geometry elements.

Geometry

Scaling factor

This option specifies the scaling of the geometry elements.

Example:

- **0.5** = half the size
- **1** = no scaling
- **2** = twice the size

Contour tolerance

Using this option, several contours which lie at a certain distance from one another can be joined automatically to form a common contour. The contour tolerance causes all contours for which the distance is smaller than or equal to the value specified in the **Contour tolerance** field to be joined together.

Reducing the default value lowers the recognition rate. Increasing this value raises the recognition rate.

Origin

This option moves the bottom left corner of the bounding rectangle of each new part to the origin.

Delete duplicate elements

This option deletes duplicate drawing elements in the drawing. It always deletes the smaller element or element of equal size.

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Automatic rotation

This option allows you to rotate parts during import by 90° based on their part geometry. The important factor is the part's bounding rectangle. The following attributes are available:

- **No rotation**
  Part geometries are imported without any rotation.
- **Rotate part onto wider side**
  If the part is higher than it is wide, it is rotated 90° clockwise.
- **Rotate part onto narrower side**
  If the part is wider than it is high, it is rotated 90° counterclockwise.

Align automatically

This option aligns imported parts parallel to the X axis on their longest side (**A**). This option is compatible with all other alignment options.

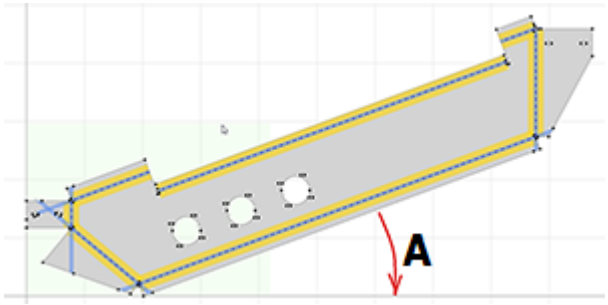


Fig. 6-3: Alignment of a part on the X axis

☞ **Note:** If any geometry filters are defined, the program searches for the duplicate drawing elements for each filter and then deletes them.

Filters for geometry selection

This tab displays all the defined filters together with the assigned technology options. You can create new filters using the **Add** button. Any number of filters can be defined.

Mode

These options can be used to include or exclude defined filters for importing layers.

☞ **Note:** All specified filters can only be included or excluded!

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- **Include**

  All specified layers will be imported in one part. Layers that are not specified are ignored.

- **Exclude**

  All specified layers in a part are ignored. Layers that are not specified will be imported.

What each button does:

- **Add**

  Opens the **Add geometry filter** dialog box in which you can create a new filter.

- **Edit**

  Opens the **Edit geometry filter** dialog box in which you can edit the selected filter.

- **Delete**

  Deletes the selected filter.

6.4.1 Add geometry filter

In this dialog box you can define geometry filters for the import. In addition, you can assign certain element attributes to those geometry elements captured by the filter.

General

Name

To create a filter, enter the name of the filter in the **Name** field. The name must match the name used in the drawing for the filter type.

Special name *

The name ***** assigns this geometry filter to all geometry elements that cannot be assigned a defined filter name. The character can only be used on its own in the field **Name** and not in combination with other characters.



Attention! All filter types defined in the following machining sequence are no longer taken into account.



Tip: Use this name to adopt geometries to which you do not want to assign a cutting technology. To do so, select the **None** option in the **Contour Cutting Properties** section.

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Filter type

The filter types are processed in the following order. It does not matter which order the filters are listed in the **Filters for geometry selection** section.

If you create several filters of the same filter type, all filters are processed together. If you have created a filter named *, it is always processed at the end of the appropriate filter type.

- **Layer**

  This filter is applied to a specific layer name in the drawing. Layers are different levels in the drawing that contain geometry elements.

  ☞ **Note:** The name is case sensitive.

- **Color**

  This filter is applied to a specific color number in the drawing.

- **Line type**

  This filter is applied to a specific line type in the drawing. The main line types are:

  - Solid line
  - Dashed line
  - Dash-dotted line

  ☞ **Note:** The names of the line types are language dependent. Find out the correct notation for the line types in advance.

- **Block**

  This filter is applied to a specific block in the drawing. A block is a linked group of geometry elements, which can lie on different layers in a drawing.

Contour properties

Engraving

If this option is enabled, all contours matching the properties of the geometry filter are executed as engravings. No lead-in is set when machining the contour.

Z height

This option allows you to define an inside contour as an extrusion or embossing (**A**). As a result, the program detects any collisions caused by the extrusion when calculating the bend.

The extrusion can be created by entering positive or negative values on the top and bottom of the bending part.

☞ **Note:** The Z dimension is always measured from the bottom of the part (**B**). Therefore, it is necessary to factor in the thickness of the material if values are positive (**C**).

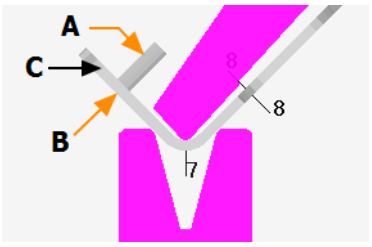


Fig. 6-4: Extrusion of an inside contour as a bolt

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Cutting properties, contour

None

This option does not add a cutting technology to the imported contours.

Copy from Wizard

This option copies the operating mode and piercing mode for the respective technology from the Cutting wizard.

Define manually

This option can be used to overwrite the **Operating mode** and **Piercing mode** cutting properties in the BySoft CAM Cutting wizard. This ensures that the geometry elements of the particular filter type are always machined with the same operating mode and piercing mode.

☞ **Note:** Only those cutting properties of the geometries that are captured by the filter are overwritten.

6.5 Additions

Use this tab to define the properties for cutouts during import.

Insert cut-outs

Cutouts can be used for parts that should be removed from the sheet by a sorting system. The point of these cutouts is to prevent the part from tilting, as a result of sharp corners, when being lifted out.

Cutout

Max width

This option defines the maximum width starting from which a cutout will be set (**A**).

Min depth

This option defines the minimum depth starting from which a cutout will be set (**B**).

Distance

This option defines the distance between the cutout and the contour (**C**). By default, the distance is equal to the kerf width.

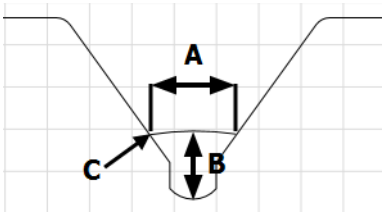


Fig. 6-5: Cutout on an outer contour

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Clustering

These options are used to automatically assign the **Part in cluster** attribute to a part with a certain size. This part can be used for cluster nesting in Part Nester.

- **Detect parts for clusters**

  Enables automatic detection of clusters.

- **Max. width**

  Defines the maximum width of a part's bounding rectangle up to which the **Part in cluster** attribute is assigned to the part.

- **Max. height**

  Defines the maximum height of a part's bounding rectangle up to which the **Part in cluster** attribute is assigned to the part.



Note: If the dimension of a part's bounding rectangle corresponds to the specified maximum height or width, the **Part in cluster** attribute is also assigned to the part.

Gripper

- **Set grippers automatically**

  If this option is enabled, a gripper is set on each part for sorting if possible.

Text

With these options, any text can be placed on the imported parts by entering coordinates. The text is defined in a separate dialog box. This is the same as for the **Add text** function in the PartEditor*(See: ›Formatting‹ Page 492)*.

Several texts can be created in one list. The texts can be added, edited, or deleted again.



Note: The texts are added to all imported parts at the defined positions. The cutting technology **engraving** is applied to all texts.

Add QR code

In this section you can define a QR code with various details and a specific size for each extracted cutting or bending part. If possible, the QR code is placed in the middle of the part. In certain cases you will need to correct it manually.

Values

Defines the content of the QR code. Click on the required values in the drop-down field to define the specifications for the QR code.



Tip: In Part Editor, you can add additional information to the QR code for each part using the **Edit parts ID** function *(See: ›Edit‹ Page 504)*.

Size

Sets the size of the placeholder for the QR code on the part. The size of the engraved QR code depends on the amount of content. The more information a QR code contains, the larger it is.

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6.6 Cutting technology

This tab is where the cutting technology is defined for all the imported geometry elements.

Enable the **No cutting technology** option if you wish to import the geometry elements without including the cutting technology.



Note: A cutting technology can only be imported if a machine and a Cutting Wizard have been specified.

General

This section is used to configure general settings for importing parts.

Cutting process

Defines the cutting process for imported parts.

Machine

Defines the machine on which to cut the imported parts.

Cutting gas

Defines the cutting gas with which to cut the imported parts.

Cutting parameter

Defines the cutting parameter with which to cut the imported parts.

No technology on outside contour

Removes the cutting technology from the outside contour of parts.



Attention! Only the inside contours of parts are cut.

Microjoints

Mode

The options in this section determine how many microjoints are automatically set on the contours.

Distributed

Use this option to define the number of microjoints to place automatically on outside and inside contours. The positions of the microjoints are displayed in a preview as gray crosses on the contour. The number of microjoints is distributed evenly along the contour.

Microjoint

- **Number of outside contours**
  Defines the number of microjoints to place on outside contours.
- **Number of inside contours**
  Defines the number of microjoints to place on inside contours.
- **Min. contour dimension**
  Defines the minimum length of a contour as of which microjoints are placed on the contour.
- **Max. contour dimension**
  Defines the maximum length of a contour up to which microjoints are placed on the contour.

End point

Sets a microjoint at the end point of all selected contours.

No microjoints

This option disables automatic placement of microjoints.

Intelligent

This option places microjoints on the contour depending on segment sizes.

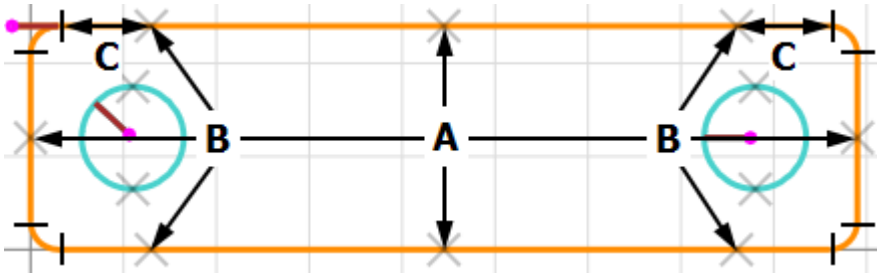


Fig. 6-6: Microjoints placed on segments of a contour

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Segments

- **Min. segment length**

  If the segment is larger than this value, it is taken into account for placing micro-joints (**A**).

- **Max. segment length**

  If the segment is smaller than this value, it is taken into account for placing microjoints (**A**).

- **Max. segment length with one microjoint**

  - If the segment is larger than **Min. segment length** and smaller than this value, a microjoint is placed in the middle of the segment (**A**).
  - If the segment is smaller than **Max. segment length** and larger than this value, an additional microjoint is placed on each segment at each beginning and end of the segment (**B**).

- **Distance to end of segment**

  This value specifies the distance (**C**) to the beginning and end of the segment where the additional microjoints are placed, if this is possible based on the other values of the **Max. segment length with one microjoint** option.

Cutting wizard

In this area you define the Cutting Wizard to apply to the geometry elements.

The properties of the Cutting Wizard can be displayed and edited in the BySoft CAM Administrator module in the **Cutting Wizard** form.

Name

Name of the cutting wizard to be used for the import.

Automatic

This option uses the cutting wizard based on the settings in Settings Manager. All other options are hidden.

Requirements:

- In **Settings Manager**, a rule with the Laser **ByFiber_N2** cutting rule set must be defined in the **Cutting wizard** section.

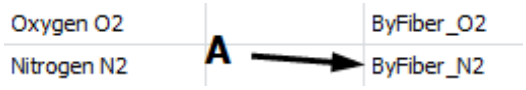


Fig. 6-7: ByFiber_2 cutting rule set

Machine filter

This option restricts the cutting wizard to the selected machine.

Material filter

This option restricts the cutting wizard to the selected material.

Cutting gas

This option restricts the cutting wizard to the selected cutting gas.

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Lead-in/Lead-out

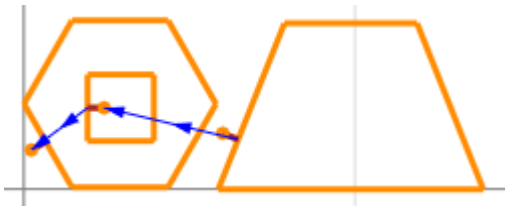
This area contains the options for setting lead-ins and lead-outs on a contour. Depending on what attribute is selected, it is possible to achieve dynamic positioning during the cutting process.

Position

Use this option to specify the position for automatic placement of the lead-ins and lead-outs.

- **Along the machining sequence**

  The software first calculates the shortest possible positioning path between the contours, and then places the lead-ins automatically so that they point towards the nearest contour. This prevents the tool traveling into the holes of contours that have already been cut when it is traveling in lowered positioning mode.



- **Coming from left**



- **Coming from right**



- **Coming from above**



- **Coming from below**



- **First segment**

  The lead-in is placed in the center of the first segment of the contour.



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Place lead-in in corners

This option places the lead-in of an outer contour in a corner of the contour.

☞ **Note:** The position of the lead-in depends on the setting in the **Position** option.



Fig. 6-8: Lead-in in a corner of the contour with **position coming from above**

Optimize machining direction

This option is used to adapt the contour machining direction to produce the minimum change in direction between the contour lead-in and the traversing path. This avoids unnecessary deceleration and increases the machining speed.

When the machining direction is changed, the tool correction is also changed automatically to the correct side (left/right).

☞ **Note:** This option is only used for the lead-in types **Direct** and **Hook**.

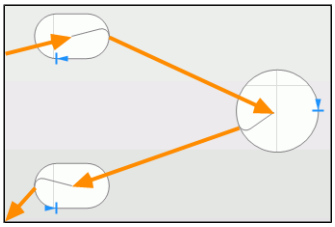


Fig. 6-9: Optimized contour machining direction to suit traversing path

Cutting properties, text / parts ID

In this pane you define the machining mode for text that has been set on the surface of a part in the **Geometry** menu. The properties that are set are applied to all texts on a part.

Operating mode

- **Default:** The parameters for machining the material are assigned by the pre-processor during export of the cutting plan, but not before this stage.
- **Cutting:** The CW parameters are used for machining the material.
- **Operating mode 2:** This operating mode is used to make a quality cut (water jet cutting only).
- **Pulse:** The pulsed parameters are used for machining the material. Pulsed operation can improve the machining quality for certain materials or for tight contours by reducing the amount of heat input to the material (laser cutting only).
- **Engraving:** The cutting parameters for engraving the material are used. Closed inner contours that have this machining mode are no longer displayed as a cut-out in the Part Editor.
- **Process macros:** The relevant process macro is used for machining the material.
  - Laser process macro
  - Water process macro

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Piercing mode

- **Default:** The piercing mode is assigned by the preprocessor when the cutting plan is exported, but not before this stage.
- **Normal:** Piercing is performed at full power.
- **Without time:** The piercing time saved in the cutting parameters on the machine is ignored.
- **Pulsed:** Piercing is performed with pulsed power using the piercing parameters specified in the cutting parameters (laser only).
- **Drilled:** Pilot drilling is performed using the drill parameters specified in the cutting parameters (water only).

☞ **Note:** Pulsed piercing produces smaller piercing holes than normal piercing.

Cutting properties without lead-in/lead-out or microjoint

Detect contact edge

This option detects the segment for the contact edge of a bending part (**A**) and sets the **No lead-in or microjoint** option on the segment *(See: ›No lead-in or microjoint‹ Page 692)*. This option prevents setting the lead-in (**B**) on the segment of a contact edge.

Restriction:

It is only possible to set the **No lead-in or microjoint** option on a segment automatically if the backgauges can be positioned on a single straight segment.

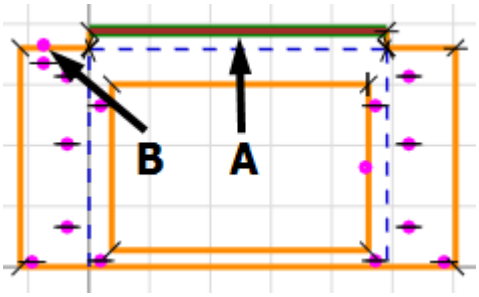


Fig. 6-10: Contact edge with “no lead-in / microjoint” attribute

Cutting time

Automatic Cutting Time Calculation

This option automatically calculates cutting time in the background when importing all parts.

This option can be used under the following conditions:

- The machine is configured
- The material and thickness have been configured on the **Properties** tab.

Calculating the machining time

A parameter file is required to calculate cutting time. The following values are required to obtain the parameter file:

Machine: contains the path to the parameter files.

Material: has a cutting table containing the filters to retrieve the parameter file.

Thickness: defines the appropriate parameter filter from the cutting table.

Gas: Gas is configured for the part and is required to filter the parameter (the system has configured the default gas based on the cutting table).

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6.7 Machining sequence

This pane contains several options for specifying the order in which contours are machined. The choice of attribute can have a huge impact on the machining time for the part.

Mode

Constructing sequence

The machining sequence is set by the order in which the contours are constructed.

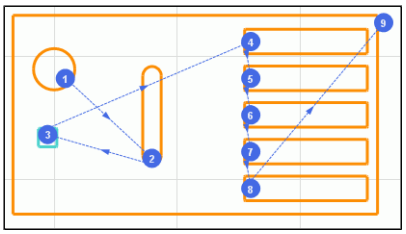


Fig. 6-11: Machining sequence set by constructing sequence

Optimized positioning path

The sequence for the contours is computed to give the shortest overall distance for all traversing paths together.

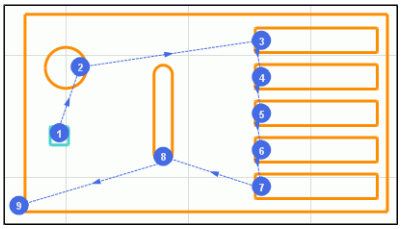


Fig. 6-12: Optimized machining sequence

Vertical - From left to right

The contours are processed vertically (up and down) starting from the lower left and progressing to the top right.

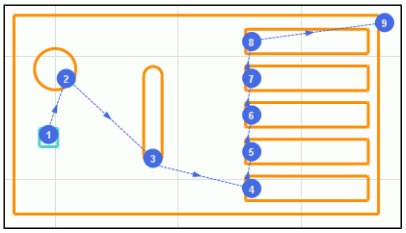


Fig. 6-13: Vertical machining sequence from left to right

Vertical - From right to left

The contours are processed vertically (up and down) starting from the lower right and progressing to the top left.

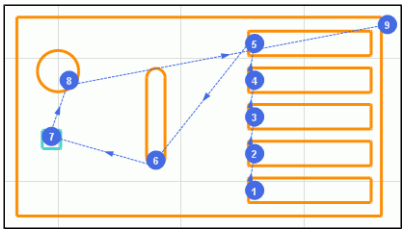


Fig. 6-14: Vertical machining sequence from right to left

Horizontal - From bottom to top

The contours are processed horizontally (from right to left) starting from the bottom right and progressing to the top left.

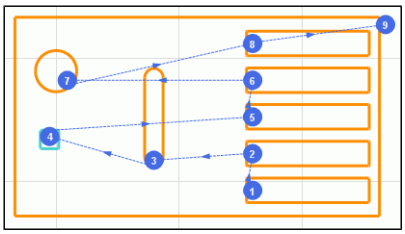


Fig. 6-15: Horizontal machining sequence from bottom to top

Horizontal - From top to bottom

The contours are processed horizontally (from right to left) starting from the top right and progressing to the lower left.

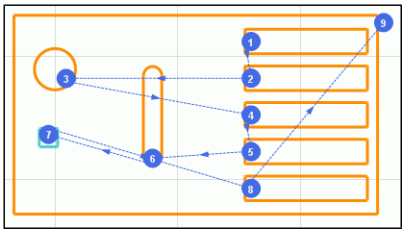


Fig. 6-16: Horizontal machining sequence from top to bottom

Random order

The sequence for the contours is defined by a random sequence generator. Select **Random order** to avoid too much heat entering the material when laser-cutting tightly packed contours or heavy-gage sheets.

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Note: This setting usually results in the longest machining time.

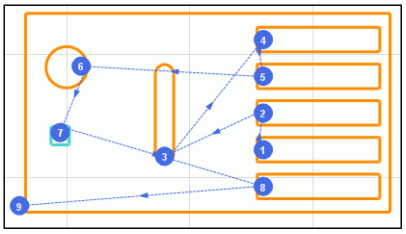


Fig. 6-17: Machining sequence with random order

Increment

This option can be used to set the increment between machining sequence numbers when the contours are actually machined.

- **Increment 1:** All contours are machined in accordance with the computed machining sequence (default).
- **Increment >1:** The actual machining sequence is applied to every nth contour of the computed machining sequence.

Using the increment option for a perforated sheet

When the increment is set to 2, only every second hole in the hole sequence is cut. The remaining holes are cut in the second pass in the given sequence.

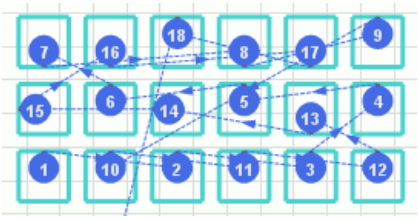


Fig. 6-18: Machining sequence for a perforated sheet with increment of 2

Tip: Use an increment >1 to distribute the generated heat more evenly in perforated sheets.

Group by sectors

Use this option to divide the total area of the part into sectors. Inside each sector, the identified contours are processed as a group using the machining sequence set under **Mode**.



Note: The sectors are displayed in a preview as blue and white areas defined by the dimensions entered in the **Grid X/Y** fields.

This option can be used for the following applications:

- To achieve a shorter positioning path for a part that has numerous different operating modes.
- For a part which contains numerous small inner contours in heavy-gage material, in order to distribute the heat input to the material more evenly. In this case, the **Random order** mode is recommended in order to distribute the sectors randomly over the part.

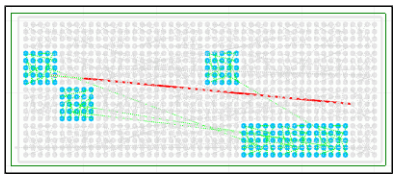


Fig. 6-19: Machining a perforated sheet by sectors

Grouping by operating mode, laser/water

The contours are processed in the order set for the operating modes. You can set the order to optimize the heat input to the material and to increase process reliability. Use the two arrow buttons to set the order of the operating modes.

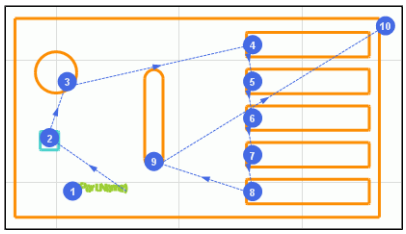


Fig. 6-20: Machining sequence based on the operating-mode order engraving, process macro 1 and cutting

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Group by size

This option sets the machining sequence in the part by contour size. All inside contours are processed using the machining sequence set in **Mode**.

- **Small contours first**

  The machining sequence goes from the smallest inside contours to the biggest inside contours.

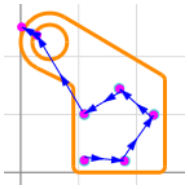


Fig. 6-21: Machining sequence with “Small contours first” mode

- **Large contours first**

  The machining sequence goes from the biggest inside contours to the smallest inside contours.

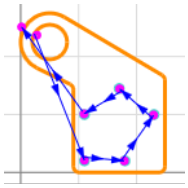


Fig. 6-22: Machining sequence with “Large contours first” mode

6.8 File options

This tab is where the storage location is defined for the files being imported into BySoft CAM. It is also where you specify what happens when two files have the same name.

Data storage

To define the storage location of the files, enter the path to the parts folder in the **Sub-folder** field or select the folder using the ··· button in the **Find folder** dialog box.

File name

If when importing the parts it is found that a file name already exists, the following options are available:

- **Ignore file**
  The file with the same file name is not imported.
- **Overwrite file**
  The existing file is overwritten by the file with the same file name.
- **Add serial number to file name**
  The file with the same file name is imported with a serial number in the file name.

Note: Only one option can be applied per import operation.

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6.9 Data fields

Where do I find the function?

Select: **Part Importer > Settings**

On this tab you can define how to import text elements in the drawing as attributes into BySoft CAM. Use the **$** character as a placeholder for the text element to use.

Attention! The information for the data fields must exist as text elements in the drawing. Text elements from contours are not recognized!

Example:

In this example, the material identifier **DC01** is used for the part in BySoft CAM.

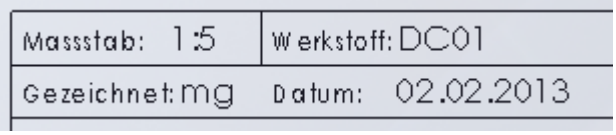


Fig. 6-23: Text element with material identifier in a drawing

- Text element in drawing = material: **DC01**
- Input in **Material** field = material:**$**

delete data fields found

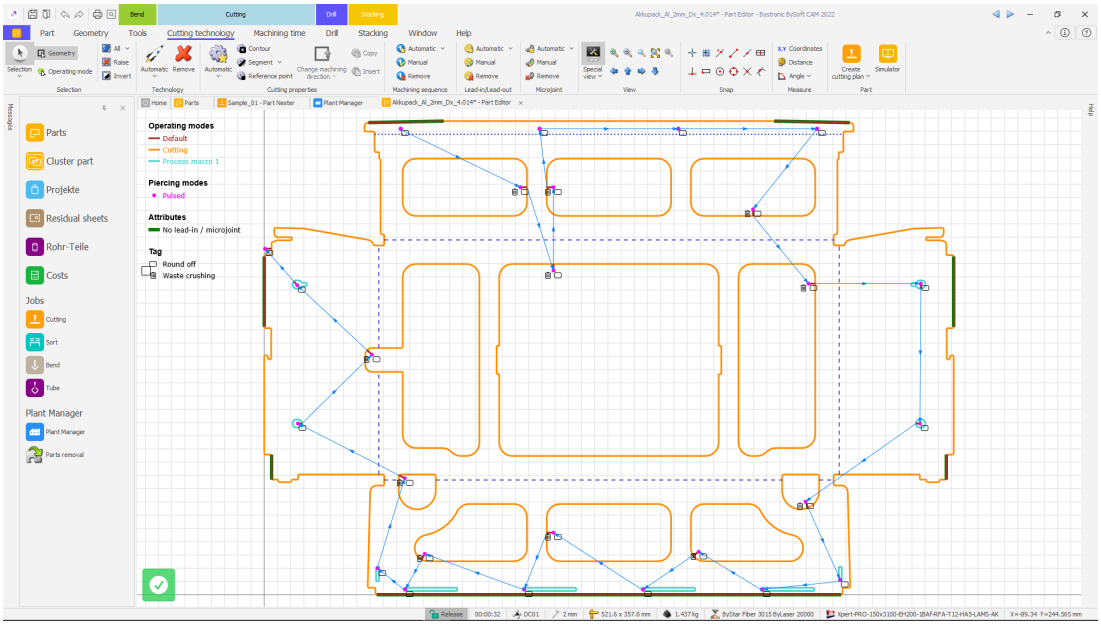
This option does not apply texts from imported data fields to the part contour.

Note: If this option is not enabled, all texts from imported data fields are applied to the part as an engraved contour.

7 Module: Part Editor

Where can I find the module?

Select: **Modules >**



The Part Editor is a module in which existing parts can be edited, new parts created, or parts imported from a different format. It is also where you can assign a cutting technology to parts. The editing process for parts basically follows the sequence of menus from left to right.

Note: In Part Editor, you can only make limited changes to the geometry of a part. More extensive changes to the geometry of a part can be made in an external CAD program by running the **Start CAD** function from the **Part** menu.

Tip: Click on an area of the header to call up the **Shortcuts** function in the context menu. The **Shortcuts** dialog can be used to define keyboard shortcuts for all functions of the current module.

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Part Editor consists of the technology sections **Cutting**, **Bending**, **Drilling** and **Stacking**. They are displayed or not depending on the licenses in use. The **Bending** area is only available if the relevant license has been purchased and bend lines have been created on the part. To open the part in the relevant technology area, click on the appropriate button in the header. The areas are displayed as a button in the header.

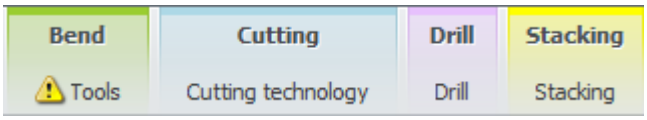


Fig. 7-1: Technology areas displayed in the Part Editor

Preparing a flat part for the nesting process

This procedure is recommended for importing a new flat part into Part Editor and then preparing it for the nesting process:

1. In the **Geometry** menu, set the origin in the lower left corner of the part, if possible.
2. Use the **Check** function to check whether the part contains any open contours.
3. In the **Cutting technology** menu, use the **Entire technology, automatic** function to assign the cutting technologies to the part.
4. Use the **Check** function to check that all the contours have been assigned a cutting technology.
5. Save the part.

Display contour properties

You can display the properties of a contour in a pop-up window by moving the cursor (**A**) over the contour. This function is available in every menu in which contours are displayed.

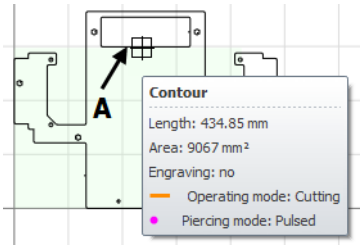


Fig. 7-2: Contour properties display

Displaying the properties of bend lines

You can display the properties of a bend line in a pop-up window by moving the cursor (**A**) over the bend line. This function is available in every menu in which bend lines are displayed.

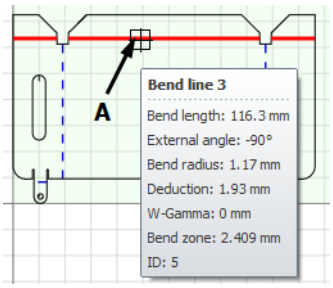


Fig. 7-3: Displaying the properties of a bend line

Icons in the workspace

Symbol	Description
	Displays the status of the part geometry. The status is updated in real time. The status is only displayed if the **In the background** option is enabled in the settings *(See: ›In the background‹ Page 347)*. **Note:** For the bending technology, the status must be determined manually using the **Check** function in the **Bend** section. **Symbol 1:** The geometry of the part is OK. The part can be produced. **Symbol 2:** The geometry of the part is not OK. It may be possible to produce the part. **Symbol 3:** The geometry of the part is not OK. The part cannot be produced. **Tip:** Click on this icon to call up the **Verify part** dialog with detailed information about the status of the part.
	Navigation bar for switching between multiple gripper tools. **Note:** The navigation bar is only displayed in the **Stack** menu.

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General menus

- File functions
- Part
- Geometry
- Window
- Help

Menus in the Bending area

- Tools
- Bending sequence
- Tool setup plan
- Backgauges
- Bending aids
- LAMS (Angle measurement system)
- Simulation

Menus in the Cutting area

- Cutting technology
- Machining time

Menus in the Stacking section

- Stack

7.1 File functions

The functions in this menu relate to the files and user interface of the module.



Functions for the current part

These functions relate solely to the current part.

- Save
- Save as...
- Save as residual sheet
- Save as template
- Save as default template
- Print working documents
  - Quick print
  - Preview
- Print labels
  - Preview
  - Quick print
- Export
- Undo
- Restore
- Close

General functions

These functions are available to run irrespective of the current part.

- Open
- Save all
- Import
- Settings
- System settings

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7.1.1 Part properties

How do I open this dialog box?

Select: **Part Editor >** 

This dialog is displayed while performing the following functions:

- For importing a part using a BySoft CAM add-in.
- When importing an external file.
- After selecting the **New part** function.

Various details must be specified before the part is generated.

- When importing a DXF file consisting only of contours, the import mode is skipped. The part is opened directly in geometry mode.
- When importing a DXF file that contains individual segments, the part is opened in import mode.

To import a new part, it is essential to enter the name and any possible part template. All the other details can be entered or modified later in the **Part** menu using the **Properties** function in the **Part** group as well as when saving the part.

Creating a new part involves the following basic steps:

1. Create a new part in BySoft CAM.
2. Specify the part properties in the **Part properties** dialog box.
3. Specify the cutting technology (only if not using a cutting table).
4. Assign the cutting technology to the part.
5. Save the part in the database.

What each field does:

Settings

- **from template**

  A template for the part can be selected in this field. By default, the **Standard** template is used. All of the settings in the template are applied to the new part.

- **from Settings Manager**

  The part settings are used by Settings Manager in Administrator.

General

Name

The name of the part must be entered in this field.

> **Note:** The name is essential for the program to be able to save the record.

Description

You can enter a more detailed description of the current object in this field.

Info 1–8

Extra information about the current object can be entered in these fields.

System of units

The system of units for the current object is defined in this field (mm/inches)

> **Note:** The part can only be nested in jobs that use the same system of units.

Material

Name

This field displays the name of the material. Clicking on the arrow symbol on the right lets you select the material from a list.

Thickness

This field displays the thickness of the material. Clicking on the arrow symbol on the right lets you select a thickness from a list. All the thicknesses available for the material are displayed. If you require a different thickness, enter it in the field from the keyboard.

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7.1.2 Save as residual sheet

Where do I find the function?



Select: **Part Editor >** 

Use this function to save the current part as a residual sheet in the **ResidualSheets** folder. This function can be used to construct any form of residual sheet and then save it as a residual sheet.

7.1.3 Save as template



Where do I find the function?

Select: **Part Editor >** 

Use this function to save the current part as a template in the **Templates** folder. You can create additional subfolders below this folder for saving part templates.

Use the **Save as** dialog box to save the template. A default part name is suggested in the **Name** field which is composed of the part settings of the machine, material, thickness and the system of units.



Note: Part templates are managed in the same way as standard parts in **Object Manager**.



Tip: Creating part templates can cut the time needed for importing new parts, because recurring settings in new parts do not need to be customized each time.

Examples of uses for templates:

- Several parts of an assembly with the same material and thickness for recurring cutting jobs.

Save part as template

1. Create a new part without a contour.
2. Specify the part's properties in the **Part** menu.
3. Specify the technology in the **Cutting technology** menu.
4. For a bending part, you can optionally specify the bending technology in the **Tools** menu.
5. From the File functions, select the **Save as template** function.
6. Enter the name of the new part in the **Name** field or accept the suggested name.
7. Press **Save** to save the part as a template.

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7.1.4 Save as default template



Where do I find the function?

Select: **Part Editor >** 

Use this function to save the current part as the **Default** in the **Templates** folder. This part template can be used each time a new part is imported.



Note: Part templates are managed in the same way as standard parts in **Object Manager**.



Tip: Use the default template to specify those part settings that occur most often.

7.1.5 Save as cluster frame



Where do I find the function?

Select: **Part Editor >** 

Use this function to save the current part as a cluster frame in the **ClusterFrames** folder. You can create additional child folders below this folder for saving cluster frames.

Use the **Save as** dialog box to save the template. A default part name is suggested in the **Name** field which is composed of the part settings of the machine, material, thickness and the system of units.

 **Note:** Cluster frames are managed in the same way as standard parts in **Object Manager**.

Examples of uses for cluster frames:

- Nesting several parts of an assembly with the same material and thickness to sort them together in a cluster frame.

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7.1.6 Print working documents



Where do I find the function?

Select: **Part Editor >** 

This function can be used to print out the working documents for the current part.

Printer

In this area you can select the printer and set your printer preferences.

Print

In this area you can specify preferences for the print layout.

- **Print template**
  Specifies what layout is used for the printed document.
- **Number of copies**
  Specifies how many copies of the document are printed.

Preview

This button previews the print document. The document can also be printed or emailed from the Preview dialog box.

7.1.7 Print labels



Where do I find the function?

Select: **Part Editor >** 

This function can be used to print out labels for the current part.

Printer

In this area you can select the printer and set your printer preferences.

Print

In this area you can specify preferences for the print layout.

- **Print template**
  Specifies what layout is used for the printed document.
- **Number of copies**
  Specifies how many copies of the document are printed.

Labels

Defines the number of labels that shall be printed for the part.

Preview

This button previews the print document. The document can also be printed or emailed from the Preview dialog box.

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7.1.8 Export part



Where do I find the function?

Select: **Part Editor >** 

This function can be used to export the current part from BySoft CAM into a different format. The following export formats are supported:

- **AutoCAD DXF files (*.dxf):** *With this file type, the part is exported in DXF format. Only the geometry elements of the parts are exported.*
- **Bysoft 7 files (*.part):** *With this format, the part is exported in BySoft CAM format. This export does not include bending data.*
- **MMC BYBEND 7 files (*.bpx):** *With this format, the bending part is exported in BySoft CAM format.*
- **CYCAD DXF (*.dxf):** *This format exports all geometry and bending information into a DXF file. This format is used to import into the PC-1200 controller from **Cybelec**.*
- **Mesh STL (*.stl):** *This format makes it possible to display a 3D part as a 3D object in another system (ERP, MES, etc.).*

 **Tip:** Use this function to transfer parts between different applications.

Export part

1. Select the part that you wish to export.
2. Select the **Export part** function.
3. In the **Export part** dialog box, select the folder in which you wish to save the part.
4. In the **File type** selection field, select the format in which you want to save the part.
5. Press **OK** to confirm the selection. The file is saved in the folder under the specified file name and format.

7.1.9 Settings



How do I open this dialog box?

Select: **Part Editor >** 

In this dialog box you can define all the view settings for the **Part Editor** user interface and the presets for importing parts.



Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

Tabs in the Settings dialog box:

- 2D view
- 3D view
- Cutting properties
- CAD
- Dimensioning
- Verify part
- Options

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7.1.9.1 2D view

How do I open this tab?

Select: **Part Editor >**  **> Settings**

This tab contains the options for customizing the 2D user interface in the **Cutting** technology area.

Attention! Do not use luminous colors in the color preferences because these cause eye strain after a prolonged period.

Note: When customizing the color options, some elements will no longer be visible if you select identical or similar colors for different color options.

Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

Background

These options set the color of the background display in Part Editor.

- **Import**

  Specifies the background color for the view while the import functions are running.

- **Default**

  Specifies the background color for the main view in the menus.

- **Profile**

  Specifies the color of the tubular section displayed in the workspace of the Tube Editor.

Grid

These options are used to specify the display preferences for the grid. A visible grid and a snap grid are used. The visible grid is displayed in the workspace using the **Display** option. The snap grid is not visible and can be enabled/disabled using the **Align with grid** function.

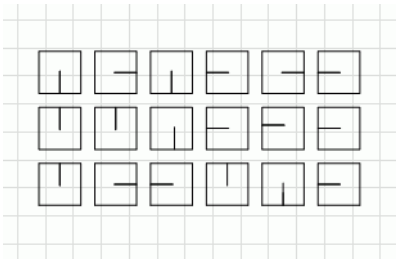


Fig. 7-4: Background display for visible grid

- **Displays**

  Displays a square grid in the workspace. The grid helps with orientation.

- **Grid size**

  Specifies the minimum grid size for the optical grid When zooming in, the grid is displayed down to this minimum size.

- **Import**

  Specifies the grid color for the import view.

- **Default**

  Specifies the grid color for the standard view.

- **Crosshairs**

  Displays the cursor additionally as crosshairs. The crosshairs are displayed over the full workspace.

- **Snap grid**

  If the **Align with grid** snap function is enabled, the value specified here defines the snap grid pitch, i.e. the gap between snap points. The default value is 1mm. The grid can be used for measurement functions or placing text, for example.

Axes

These options are used to specify the display preferences for the axes.

- **Displays:** Displays the origin of the object as axes crosshairs.
- **Import:** Specifies the color of the axes for the import view.
- **Default:** Specifies the color of the axes for the standard view.

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Reference rectangle

These options are used to display a rectangle that can be used for a size comparison with the displayed part. The reference rectangle is defined by default to be landscape format in paper size **A3**. You can customize the size using the **Width** and **Height** input fields.

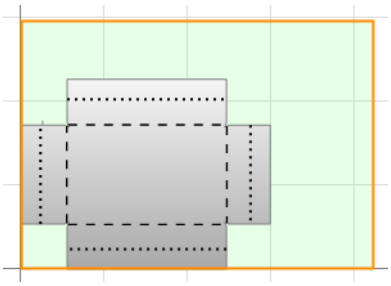


Fig. 7-5: Background display for reference rectangle

- **Displays:** Displays the reference rectangle as a colored area.
- **Width:** Specifies the width of the reference rectangle.
- **Height:** Specifies the height of the reference rectangle.
- **Import:** Specifies the color of the reference rectangle for the import view.
- **Default:** Specifies the color of the reference rectangle for the standard view.

Bending

These options are used to set the color display for the Entry zone (**A**) and the Bend zone (**B**). The zones can be displayed in the **Tools** and **Bending sequence** menus.

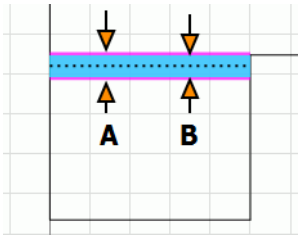


Fig. 7-6: Color display of the entry zone and bend zone

Simplified drawing

These options can be used to speed of contour drawing as of a certain number of segments. Drawing is sped up by means of several options that simplify contour geometry.

- **Simplify:**
  Arcs are approximated with straight lines.
- **Smooth:**
  Reference points are removed.
- **Hide Contours:**
  If the **Shade part** is enabled, outside contours are not drawn.



Note: If a particular option is disabled, no contour simplification performed with this option.

Use the **Reset** button to restore the default settings for the tab.

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7.1.9.2 3D-View

How do I open this tab?

Select: **Part Editor >** **> Settings**

This tab contains the color options for customizing the 3D user interface in the **Bending** technology area.

You can set a transparency level in percent for displaying parts of the machine. Use this transparency feature to reveal bending parts hidden behind the bending tools.

Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

Automatic toggling before/after the process

- If this option is enabled, the 3D simulation displays the situation before and after the bending process.
- If the option is disabled, the 3D simulation displays all bending processes, either only before or after the process.

Note: Use the **Before bending** button in the view bar to select the view before or after the bending process.

Use the **Reset** button to restore the default settings for the tab.

7.1.9.3 Cutting properties

How do I open this tab?

Select: **Part Editor >** **> Settings**

This tab contains the color options for displaying the cutting properties on a part. The color options can be defined for the **Operating mode** and the **Piercing mode**. The cutting properties are only displayed if a cutting technology has been assigned to the part in the **Cutting technology** menu.

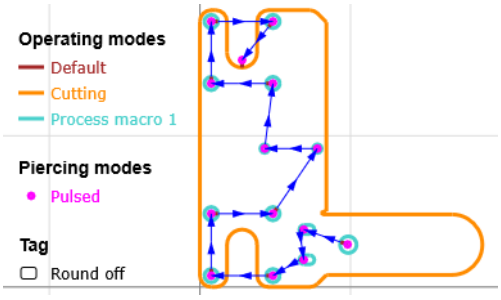


Fig. 7-7: Displaying the cutting properties on a part

Use the **Reset** button to restore the default settings for the tab.

7.1.9.4 CAD

How do I open this tab?

Select: **Part Editor >** **> Settings**

This tab contains options for opening an external CAD program for creating new parts or editing contours of existing parts. You have the option of opening a BySoft CAM part in an alternative CAD tool.

Options for alternative CAD

These options can only be selected if the **Parts editor** is set to **Alternative CAD**.

> **Note:** When using an alternative CAD program, BySoft CAM converts the part into the DXF format before transferring the file to the CAD tool.

- **Program**
  In this field you specify the path and file name of the executable file via which the alternative CAD software is launched.
- **Arguments**

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In this field you can specify the arguments to be used when opening the file in the specified software.

- **Working directory**

  In this field you can specify the folder to be used as the working folder for the software.

- **Accept changes automatically**

  If this option is enabled, all the changes made to a part in an external CAD program are adopted automatically in BySoft CAM.

Examples:

Alternative CAD	Program	Arguments
ByPart (Bysoft6)	C:\Program Files (x86)\Bysoft6\programs\ByPart.exe	/user="system" /import="$(Filename)" /file="$(Filename)"
DraftSight	C:\Program Files (x86)\Dassault Systemes\DraftSight\bin\DraftSight.exe	"$(Filename)"

7.1.9.5 Dimensioning

How do I open this tab?

Select: **Part Editor >** **> Settings**

This tab contains options for dimensioning parts in the Part Editor module. The details affect the dimensioning functions in the **Geometry** menu *(See: ›Insert dimensioning‹ Page 509)*.

☞ **Note:** Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

General

This is the area where you define the color for dimensioning.

Text

The information in this section refer to the dimensioning text displayed.

Height

This option defines the size of the dimensioning text.

Precision for length

This option configures the decimal places for length dimensions.

Precision for angle

This option configures the decimal places for angle dimensions.

The information in the fields below are displayed before or after the dimension text.

- **Prefix** = text before dimension text
- **Suffix** = text after dimension text

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Linear

The text is displayed for horizontal or vertical dimensionings.

Aligned

The text is displayed for diagonal dimensionings.

Radius

The text is displayed for radius dimensionings.

Diameter

The text is displayed for circle dimensionings.

Arc length

The text is displayed for arc dimensionings.

Angle

The text is displayed for angle dimensionings.

Construction line

Distance

This option specifies the distance of the construction line tot he contour.

Prolongation

This option extends the construction line beyond the dimension arrow.

Arrow

Length

This option specifies the length of the construction line arrow.

Use the **Reset** button to restore the default settings for the tab.

7.1.9.6 Verify part

How do I open this tab?

Select: **Part Editor >**  **> Settings**

This tab contains all the options for checking parts for the Part Editor module.

General

In the background

This option automatically performs a background part check whenever the geometry changes. The result is displayed with an icon in the workspace *(See: ›Icons in the workspace‹ Page 325)*.

Disable this option:

to increase performance on weak computers.

Minimum contour length

This option defines the minimum length of a contour as of which the part check displays the error message **Contours shorter than x mm**.

Short segmentation

Number of segments

This option is used to set the number of excessively short segments in a contour starting at which the part check will display the following error message: **Short segmentation**.

Minimum segment length

This option is used to set the minimum segment length. If the segment length of a number of segments equal to the number in the **Number of segments** option falls below this minimum segment length, the part check will display the following error message: **Short segmentation**.

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7.1.9.7 Options

How do I open this tab?

Select: **Part Editor >** **> Settings**

This tab contains additional options for the Part Editor module.

General

Display cutting time in minutes

This option displays the cutting time in decimal format in minutes. If this option is not enabled, the cutting time is displayed in the «HH:MM:SS» format.

- Option enabled: **343.9min**
- Option not enabled: **05:43:54**

Tip: Use this option for calculating the total of several cutting times by hand.

Automatic Cutting Time Calculation

When this option is used, the cutting time for a new or modified part will be automatically calculated in the background. This means that the cutting time for the part can be automatically provided to other systems such as Plant Manager.

The cutting parameter is selected automatically. If multiple cutting parameters exist in the same combination of machine, laser, material, thickness, etc., it is not possible to calculate cutting time. The parameter must be selected manually in the **Machining time** menu.



Tip:
Add a **$** character to the end of a parameter to set it as default. If there are multiple identical parameter values with different machining attributes, it is automatically selected for calculating cutting time.

Example:

Correct

- ByStarFiber3015_L605_6000_AW5754_1.5_N2_**$**.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2_**$**Quality.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2_Quality**$**.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2_Quality_**$**.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2_Quality_**$**_XY.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2_Quality_**$**Z.PAR

Wrong

- ByStarFiber3015_L605_6000_AW5754_1.5_N2**$**.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2**$**_Quality.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_**$**_N2.PAR



Note: If this option is enabled, the **Calculate** and **Remove** functions are disabled in the **Machining time** menu. Machining time can only be calculated automatically if the correct cutting parameter exists!

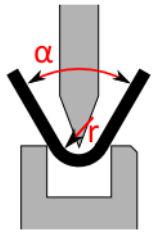
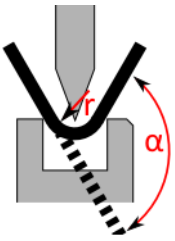
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Automatic bending line correction

This option either lengthens or shortens bend lines whose end points do not lie exactly on the outer contour, depending on the situation.

Angle definition

This option can be used to define the input of a bend angle based on the internal or external angle. This setting applies to all bend angle figures in the **Bend** section.

Defining an angle based on the internal angle	Defining an angle based on the external angle
	

Limit for number of parts extracted

This option is used to set the maximum number of parts for extraction that will be automatically imported and opened in Part Nester.

This option is used only when importing parts with the **Open parts** mode. When this mode is used, all extracted parts are opened in Part Editor after being imported.

Attention! If the number of parts is too big (approx. more than 40), there may be a memory overflow.

Dimensioning for drawing

This option displays distances as dimensions next to or in the contour when drawing contours.

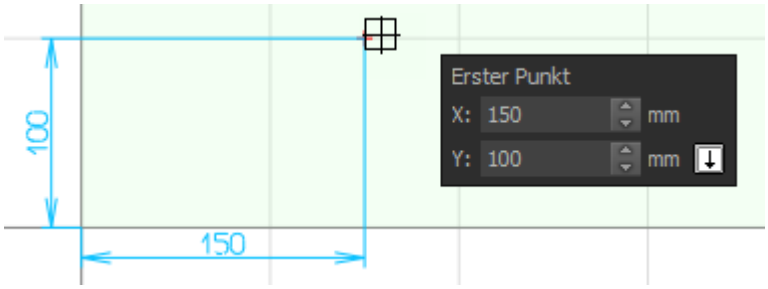


Fig. 7-8: Dimensions of the distance from the origin

Automatically connect bend lines

This option automatically connects two or more bends to form a bend that is on a common axis as soon as you switch to the **Tools** menu in the **Bending** section.

Export configuration

In this section you can select an export configuration for layer filters via the **Name** selection field. This configuration assigns certain layers to the part contours during a DXF export.

You can create the export configuration for layer filters in Administrator *(See: ›DXF configuration export‹ Page 218)*.

Bend deduction

Update automatically

When this option is enabled, the bend allowance of the bend line is recalculated automatically when a new tool with a different bend allowance is used. If a tool is used with a different bend allowance, a warning is displayed.

Tip: Deactivate this option if the part has already been cut and all that remains is for the bending technology to be applied automatically to the part.

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Check width of the bend zone

This option can be used to disable the bend zone check if the **Update automatically** option is enabled.

Disabling this option permits bends in which bend allowance of the Bend zone is much larger than the original bend zone. This is the case where the bend radius is greater (**A**) than it was constructed (**B**).

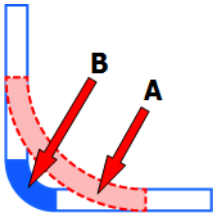


Fig. 7-9: Greatly increased bend radius of a bend

Disable this option:

- If you want to bend bends with a tool with a larger radius than the constructed bend radius.
- If the bend was constructed without a radius (bend allowance = 0).

Check corner relief

Use this option to disable the corner relief check if the geometry is shortened using the **Update bend allowance** function in the **Tools** menu.

Disable this option:

If geometries with corner reliefs that are too small or with no corner reliefs cause an error message during shortening but you still want to shorten them.

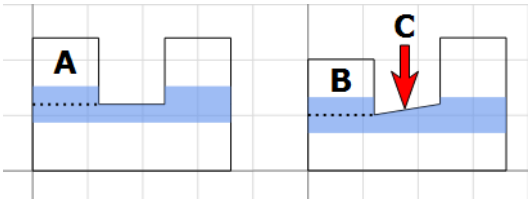


Fig. 7-10: Geometry without corner relief before (**A**) and after (**B**) shortening

Attention! If corner reliefs are not present or too small, this may result in oblique edges (**C**).

Nest bending parts

Same orientation

When you enable this option, all instances of the part created subsequently in the Part Nester module are nested in the same orientation on the sheet. This option is intended for bending parts for which the grain direction of the sheet must be taken into account to ensure high bending accuracy.

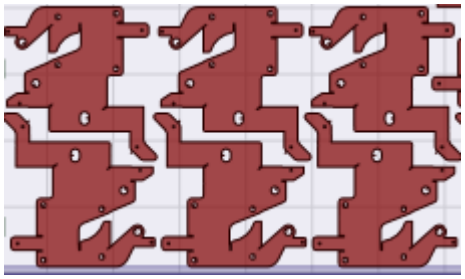


Fig. 7-11: Same orientation of a bending part on the sheet

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7.2 Import part



Where do I find the function?

Select: **Modules > Part Editor > Import part**

This function can be used to import a part or a contour from various file formats. If the part you wish to import exists as a DXF file in a CAD drawing, the part can be extracted from the drawing using a range of selection options.



Note: Developed views of bending parts can also be imported as a DXF file. The bend lines are defined using the **Bend** function.

A part can be imported from the following file formats:

- **AutoCAD DXF files (*.dxf):** *This is the format for importing certain elements from a DXF file (See: ›Extracting parts from a DXF file‹ Page 1389).*
- **AutoCAD DWG files (*.dwg):** *This is the format for importing certain elements from a DWG file.*
- **Bysoft 7 part files (*.part):** *This is the format for re-importing an exported BySoft CAM part.*
- **Bysoft DIN files (*.din):** *This format is used to import parts from Bysoft 5.3 or earlier.*
- **Bysoft6 Bypart files (*.bpd; *.zip):** *This is the format for importing the geometry and technology of a part from Bysoft6 (See: ›Importing a part from Bysoft6‹ Page 1388).*
- **DSTV-NC files (*.nc; *.nc1):** *This format can be used to import NC files as flat parts.*
- **Encapsulated PostScript files (*.eps):** *This format is used to import graphics in EPS format.*
- **MMC BYBEND 7 files (*.bpx):** *This is the format for re-importing a bending part exported from BySoft CAM.*
- **Raster Images:** *This format can be used to convert a bitmap image into a vector graphic.*
- **Step files (*.step; *.stp):** *This format can be used to import CAD files in step format.*



Note: To import an entire Bysoft6 database, use the **Migration Wizard** tool. To import several parts, use the **Part Importer** tool. These tools can be launched from the **Bystronic** > **BySoft CAM** > **Tools** software folder.



Tip: The function can also be launched from the quick access toolbar *(See: ›Quick access toolbar‹ Page 11)*.

Supported elements for importing from DXF or DWG files

When performing a DXF or DWG import, only certain elements can be imported into BySoft CAM.

DXF/DWG elements supported in BySoft CAM

ARC	MTEXT (just single-line text)
CIRCLE	POLYLINE
ELLIPSE (converted into a contour)	SPLINE (converted into a contour)
INSERT	TEXT
LINE	VERTEX
LWPOLYLINE	

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7.2.1 Group: Selection

Where do I find the function?

Select: **Modules > Part Editor > Import**

This group displays all the available selection functions. These functions can be applied to all elements of an object.

7.2.1.1 Select class

Where do I find the function?



Select: **Modules > Part Editor > Import > Selection**

The function can be used to select a particular class in a DXF file. A different color is used to display each class. Clicking on the geometry of a particular class will select all the geometry elements of the same class.

The following classes are available:

- Default
- Layer
- Color
- Blocks
- Line type

Extracting a part by multiple selection

In many cases, a part has to be extracted from a DXF file by multiple selection of geometry and drawing elements that are no longer needed. To do this, proceed as follows:

1. Select the **Class** function.
2. Select the view function for which the fewest unwanted geometry elements are selected.
3. Select the **Invert** function.
4. Delete the selected geometries.
5. Use another view function to select the remaining geometry elements that are no longer needed.
6. Delete the selected geometry elements.



Note: If a selection cannot be made using the view functions, select the **Geometry** function and delete manually the geometry elements that are no longer needed.

7.2.2 Group: Geometry

Where do I find the function?

Select: **Modules > Part Editor > Import**

This group displays all the available geometry functions. These functions can be applied to all elements of an object to be imported.

7.2.2.1 Use filter



Where do I find the function?

Select: **Modules > Part Editor > Import > Geometry**

This function applies the geometry selection settings from the last import to the geometries. The settings for layer, color, blocks and lines are applied.



Tip: Use this function when importing multiple parts that have the same layer structure.

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7.2.3 Group: Engraving

Where do I find the function?

Select: **Modules > Part Editor > Import**

This group shows all functions for setting and deleting engravings.

7.2.3.1 Set



Where do I find the function?

Select: **Modules > Part Editor > Import > Engraving**

This function assigns the **Engraving** cutting technology to all contours of the selected layer. This technology is displayed to the right of the layer key (**A**).



Fig. 7-12: Engraving set on a layer.

7.2.3.2 Remove



Where do I find the function?

Select: **Modules > Part Editor > Import > Engraving**

This function removes the engraving from all layers.

7.2.4 Group: View

Where do I find the function?

Select: **Modules > Part Editor > Import**

This group contains all the functions that can be used to change the current view of an import file. The group also contains several functions that can be used to select the geometries for importing parts.

7.2.4.1 Special view



Where do I find the function?

Select: **Modules > Part Editor > Import > View**

This function can be used to display various elements on the part to be imported. The elements are shown/hidden by clicking on the relevant option. The elements can be displayed separately or simultaneously.

You can display the following elements:

- **Key:** *Displays the existing elements based on the selected view function as a key in the drawing (e.g.: displays all existing layers). Selected elements in the drawing are indicated by a rectangle in the key.*

7.2.4.2 Default



Where do I find the function?

Select: **Modules > Part Editor > Import > View**

This function displays the geometry in a drawing without any color-highlighting of the construction parameters.

Tip: Use this function for geometries that will not need re-editing after import.

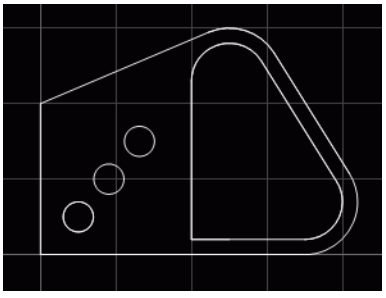


Fig. 7-13: Displaying geometries without color-highlighting

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7.2.4.3 Layer



Where do I find the function?

Select: **Modules > Part Editor > Import > View**

This function uses colors to highlight all the layers in the drawing.



Tip: Use this function to select part geometries when the geometries have been drawn on a different layer from all the others.



Fig. 7-14: Displaying the layers in a drawing

7.2.4.4 Colors



Where do I find the function?

Select: **Modules > Part Editor > Import > View**

This function displays all the colors used in the drawing.



Tip: Use this function to select part geometries when the geometries have been drawn in different colors.

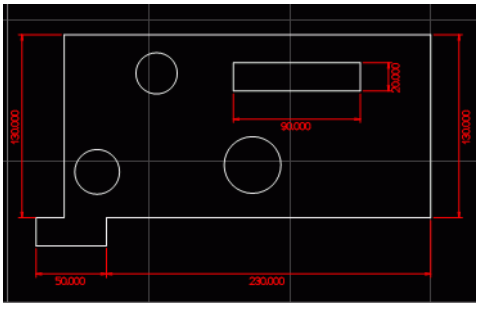


Fig. 7-15: Displaying the dimensions in the color of the geometry

7.2.4.5 Blocks



Where do I find the function?

Select: **Modules > Part Editor > Import > View**

This function displays all blocks in the drawing in different colors.



Tip: Use this function to select part geometries when the geometries have been drawn in different blocks.

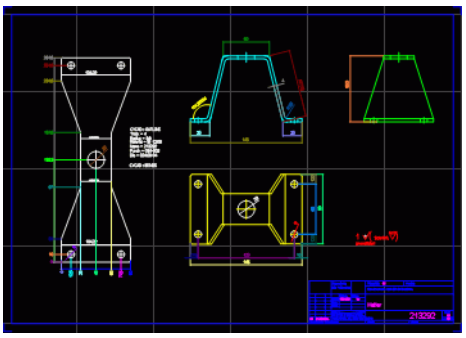


Fig. 7-16: Displaying the blocks in different colors

7.2.4.6 Line type



Where do I find the function?

Select: **Modules > Part Editor > Import > View**

This function displays all line types in the drawing in different colors. It is mostly used for selecting construction lines in a drawing. This is why the function can usually only be employed with another selection.



Tip: Use this function to select part geometries when the geometries have been drawn in different line types.

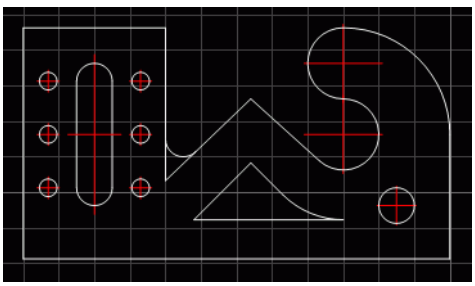


Fig. 7-17: Displaying line types in the drawing

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7.2.5 Group: Create part

Where do I find the function?

Select: **Modules > Part Editor > Import**

This group contains all the functions for importing geometries. Depending on the situation, the geometry can be imported directly or as a selection.

7.2.5.1 No selection



Where do I find the function?

Select: **Modules > Part Editor > Import > Create part**

This function imports all the geometries that exist as contours. Overlapping geometries are deleted.



Tip: Use this function when all unwanted geometries have been deleted or when the geometry elements can be imported without any re-editing.

7.2.5.2 From selection



Where do I find the function?

Select: **Modules > Part Editor > Import > Create part**

This function imports the selected geometries as a contour. Overlapping geometries are deleted.

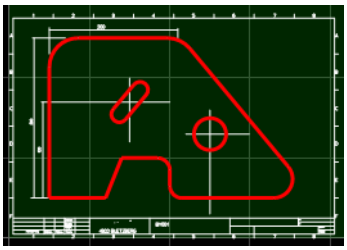


Fig. 7-18: Selecting a contour for import



Tip: Use this function when you only wish to import the selected geometry elements.

7.2.5.3 Create part



Where do I find the function?

Select: **Modules > Part Editor > Import > Create part**

This dialog box contains options for importing parts. The contour tolerance for the English system of units can be specified separately.

Settings

Contour tolerance, metric/English

Using this option, several contours which lie at a certain distance from one another can be joined automatically to form a common contour. The contour tolerance causes all contours for which the distance is smaller than or equal to the value specified in the **Contour tolerance** field to be joined together.

Reducing the default value lowers the recognition rate. Increasing this value raises the recognition rate.

Move to origin

This option moves the bottom left corner of the bounding rectangle of each new part to the origin.

Correcting branches manually

If a branch is identified between segments when importing a part, then if this option is enabled, the branch to be followed after the node must be specified manually *(See: ›Resolving a branch point during DXF import‹ Page 1329)*.

If this option is disabled, the program automatically selects the next segment to continue along.



Tip: Use this option in order to specify yourself which branch to follow when importing parts.

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Resolve contours

If there are duplicate geometries on a contour, the shorter of the two geometries is deleted.

Attention! If a cutting technology is already set on a contour, it is lost when the contour is resolved.

Connect bend lines

This option is used to connect two or more bends automatically that lie on a common axis into one bend.



Fig. 7-19: Connected bends on one bend line

7.3 Menu: Part

Where do I find the menu?

Select: **Modules > Part Editor**

This menu is where the properties are defined for the part. Changing the contour of the entire part is only possible using the **Scale** function. Changes to individual contour elements can be made using the **Start CAD** function or in the **Geometry** menu.

Functional groups

The function bar contains the following functional groups:

- Selection
- Extract
- Part
- View
- Snap
- Measure

Functions for changing the contour

These functions are used for changing the contour of a part:

- Start CAD
- Scale

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7.3.1 Group: Extract

Where do I find the group?

Select: **Modules > Part Editor > Part**

This group contains all the functions for extracting contours. Depending on the function, one or more contours or parts can be extracted from a drawing.

7.3.1.1 Extract manually



Where do I find the function?

Select: **Modules > Part Editor > Part > Extract**

You can use this function to extract one or more contours from a part to open or save as a new part. The extracted part is created with the same name as the original part.

Extracting and opening a new part

1. Select the contours that you wish to extract as a new part.
2. In the **Extract** group select the **Manual** function.
3. Select the **Open parts** function in the **Mode** section.
4. Click the **Apply** button. The selected contours are opened as a new part for editing.

Extracting and saving a new part

1. Select the contours that you wish to extract as a new part.
2. In the **Extract** group select the **Manual** function.
3. Select the **Save parts** function in the **Mode** section.
4. Click the **Apply** button. Saves the selected contours as a new part to the path given in **Directory**.



Tip: By selecting all the contours, you can produce a copy of the part with a new name.

What each field does:

Mode

- **Open part**
  Opens the selected contours as a new part in Part Editor.
- **Save parts**
  Saves the selected contours as a new part to the path given in **Directory**. The part is not opened in Part Editor.
- **Directory**
  Specifies the directory in which to save the selected contours as a new part. The path is used if the **Save parts** option is selected.

Parts

- **Quantity**
  This field always displays the quantity **1** as only one part can be extracted.
- **Name**
  Enter the name for the extracted part in this field.
- **Move to origin (0.0)**
  This option moves the bottom left corner of the bounding rectangle of each new part to the origin.
- **Description**
  You can enter a more detailed description of the current object in this field.
- **Info 1-3**
  Extra information about the current object can be entered in these fields.

Material

- **Name**
  This field displays the name of the material. Clicking on the arrow symbol on the right lets you select the material from a list.
- **Thickness**
  This field displays the thickness of the material. Clicking on the arrow symbol on the right lets you select a thickness from a list. All the thicknesses available for the material are displayed. If you require a different thickness, enter it in the field from the keyboard.

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7.3.1.2 Extract automatically



Where do I find the function?

Select: **Modules > Part Editor > Part > Extract**

This function is useful when working with a file containing multiple parts from an imported DXF drawing. The function lets you save these parts automatically as individual parts. Before the selected parts can be extracted, it is first necessary to configure the appropriate options in the **Extract** docking dialog *(See: ›Extracting parts from a DXF file‹ Page 1389)*).

Click on **Apply** to start automatic extraction of the parts. Each extracted part is created with the specified name and an additional serial number.

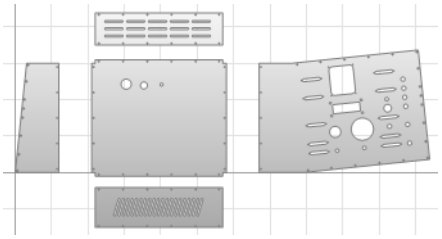


Fig. 7-20: Imported parts from a DXF drawing



Note: Up to **20** parts can be extracted at once.



Tip: Specify the cutting technology of the parts before extracting the parts. Therefore, it is no longer necessary to define the cutting technology for each separate part.

Extracting and saving several parts

1. Select the parts that you wish to extract from the drawing.
2. In the **Extract** group select the **Automatic** function.
3. Select the **Save parts** function in the **Mode** section.
4. Select the directory where you want to save the parts.
5. Optionally, specify additional information about the parts in the **Parts** section.
6. Check the material and thickness for the parts.
7. Click the **Apply** button. Saves the selected parts to the path given in **Directory**.

What each field does:

Mode

- **Open part**
  Open every selected part in Part Editor, where you can edit it.
- **Save parts**
  Saves every selected part to the path given in **Directory**. The parts are not opened in Part Editor.
- **Directory**
  Specifies the directory in which to save the selected parts. The path is used if the **Save parts** option is selected.

Parts

The information in the fields is assigned to all extracted parts.

- **Parts found**
  Displays the number of outside contours found.
- **Name**
  Enter the name for all extracted parts in this field.
- **Move to origin (0.0)**
  This option moves the bottom left corner of the bounding rectangle of each new part to the origin.
- **Quantity**
  This field can be used to specify the number of parts that will be transferred to Part Nester when a cutting plan is created from the part in the **Cutting technology** menu.
- **Description**
  This field can be used to specify a more detailed description of the parts.
- **Info 1-3**
  Additional information about the parts can be entered in these fields.

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Material

- **Name**

  This field displays the name of the material. Clicking on the arrow symbol on the right lets you select the material from a list.

- **Thickness**

  This field displays the thickness of the material. Clicking on the arrow symbol on the right lets you select a thickness from a list. All the thicknesses available for the material are displayed. If you require a different thickness, enter it in the field from the keyboard.

7.3.2 Group: Part

Where do I find the group?

Select: **Modules > Part Editor > Part**

This group contains functions that can be applied to the entire part. Each part is assigned certain properties as metadata. The metadata can be displayed and edited using the **Properties** function.

7.3.2.1 Part properties



How do I open this dialog box?

Select: **Modules > Part Editor > Part > Part**

This docking dialog box displays the properties of the current part. The properties can be displayed and edited in the relevant tab for the technology.

The following tabs are available:

- General
- Cutting

  In the case of a new part, the **Cutting** tab will not be displayed until there is a cutting technology on the part.
- Bending

  In the case of a new part, the **Bend** tab will not be displayed until there is a bending technology on the part.
- Attachment

General

How do I open this tab?

Select: **Modules > Part Editor > Part > Part > Part properties**

This tab displays the basic properties of the current part.



Note: If a part name has not been specified yet under **Name**, when you save the part, the **Save as** dialog box will open in which you can enter a name to save the part under.

Settings

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- **Apply Settings Manager**

  This button applies the Settings Manager settings to the part based on the options selected in the drop-down box.

  - **Cutting and bending**

    Applies the settings for cutting and bending in Settings Manager to the part.

  - **Cutting**

    Applies the cutting settings in Settings Manager to the part.

  - **Bending**

    Applies the settings for bending in Settings Manager to the part.

General

The general information describes the part and the system of units that are used. When the part is saved, various data is created automatically under the **Created**, **Changed** and **Revision** options, which cannot be changed manually.

- **Name**

  Shows the name of the part. The part name can only be changed subsequently in the **Object Manager** if the part has not already been nested at that point.

- **Path**

  Shows the path to the object.

- **Created**

  Displays the date and time at which the object was created and the name of the user who created it.

- **Changed**

  Displays the date and time at which the object was last saved and the name of the user who saved the object.

- **Revision**

  Displays the current revision number for the object. The revision number is increased by a count value every time the object is saved.

- **Quantity**

  This field can be used to specify the number of parts that will be transferred to Part Nester when a cutting plan is created from the part in the **Cutting technology** menu.

- **Description**

  You can enter a more detailed description of the current object in this field.

- **Info 1-8**

  Extra information about the current object can be entered in these fields.

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

☞ **Note:** The part can only be nested in jobs that use the same system of units.

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Material

The information about the material is used for calculating the weight and for assigning the cutting technology.

- **Name**

  This field displays the name of the material. Clicking on the arrow symbol on the right lets you select the material from a list.

- **Thickness**

  This field displays the thickness of the material. Clicking on the arrow symbol on the right lets you select a thickness from a list. All the thicknesses available for the material are displayed. If you require a different thickness, enter it in the field from the keyboard.

Geometry

This section displays details of the geometry and weight of the part. The weight can only be displayed if the density of the material has been specified in the system data.

Profile (Tube Editor)

In this section you can specify the profile type and its dimensions for the tubular part.



Note: As soon as there is a contour in the work area, the profile type can no longer be changed.

Cutting

How do I open this dialog box?

Select: **Modules > Part Editor > Part > Part > Part properties**

This tab displays the cutting properties assigned to the part.

Cutting process

The information about the machine is needed in order to assign the cutting technology and to calculate the machining time.

Machine

This field displays the machine that shall be used to machine the object. Clicking on the arrow symbol on the right lets you select a different machine from a list. This list only contains those machines that are configured in the system. If you require a new machine, you must create it in the **Administrator** module in the **Cutting machines** form *(See: ›Cutting machine‹ Page 102)*.

Cutting gas

The cutting gas is used in the **Cutting technology settings** as a filter for the Cutting Wizard *(See: ›Cutting gas / Abrasive filter‹ Page 666)*.

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Cutting parameter

The cutting parameter specification is used for the cutting time calculation. The **Cutting parameter** dialog box will show the available cutting parameters based on the material, thickness, machine, and cutting gas specifications.

If no cutting parameter is found in the specified path, the dialog box can be used to specify the appropriate parameter. The path specifications for the cutting parameters for the relevant machine are defined in Administrator *(See: ›Parameter path‹ Page 106)*.

Abrasive (ByJet)

The abrasive is used in the **Cutting technology settings** as a filter for the Cutting Wizard.

Nesting settings

The nesting settings are needed for nesting the part in the Part Nester module.

Priority

You can use this option to set the order in which the different parts are to be nested. Entering a priority is optional. Enter a number between 1 and 99 for the priority, Where 1 stands for highest priority and 99 for lowest priority. No priority is assigned if 0 is entered.



Attention! The use of different priorities does not usually result in optimum nesting of the cutting plan.



Note: This function is only used for automatic nesting.

Rotation allowance

This field defines the increment of the angle of rotation in degrees. The part can be rotated during nesting through this angle.



Tip: A rotation allowance can be entered by clicking on the input field or by selecting a set angle value via the left arrow button.

Discrete angles of rotation

You can enter multiple angles of rotation in this box. The part can be rotated during nesting through this angle. The angles of rotation must be separated from each other by semicolons (;).



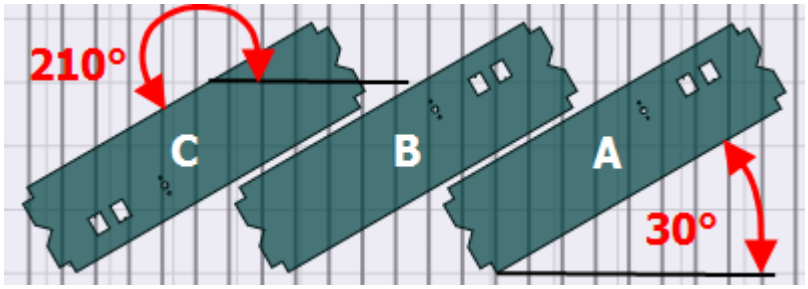
Tip: If this field contains angles of rotation, the angle of rotation in **Rotation allowance** is ignored (not for cluster nesting).

Same orientation

This option nests all instances of the part in the same orientation on the sheet.

Example:

If the part is nested with an angle of **30°** (**A**), the other instances can only be nested with the same angle (**B**) or with an angle of **210°**=180°+30° (**C**).



Application:

- Parts with same direction of grinding
- The correction value only needs to be entered on the bending machine for the first bending part. Because the alignment of the grain direction is the same, it is not necessary to enter a correction for all other parts.

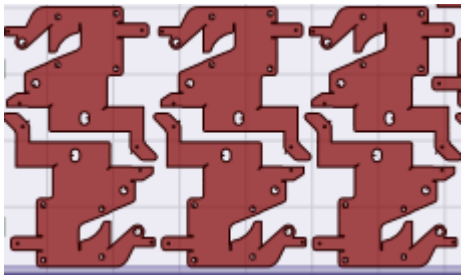


Fig. 7-21: Same orientation of a bending part on the sheet

Mirroring authorization

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This option permits the part to be mirrored about the vertical axis during nesting.

Test part

Flags the part as a test part. In Part Nester all instances of the part are nested as test parts *(See: ›Nesting a test part‹ Page 1375)*.

Sheet format

Use this option to assign an existing sheet format to the part. Depending on the size of the parts, this may help reduce sheet waste. If you nest the part on a different sheet format in Part Nester, the **Parts** tab displays a warning.

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Part in cluster

This option specifies that the part can only be nested in a framed cluster *(See: ›Framed cluster‹ Page 1213)*.

Part in cluster

This option specifies that the part can only be nested in a framed cluster *(See: ›Framed cluster‹ Page 1213)*.

Filler part

Enable this option if you want to use this part to fill the areas leftover from nesting, or if the part is often needed in larger, unspecified quantities.

Take cutouts into account

These options determine whether or not to apply cutouts that were set on a part in Part Editor.

- **Default**

  If the part contains cutouts, they will be taken into account for nesting if the parameter is enabled in the nesting settings.

- **No**

  No cutouts are taken into account for nesting.

  Attention! The cutouts of a part are changed if the contour of another part is nested in a cutout. Based on the contour, the cutout is divided by the contour.

- **Yes**

  If the part contains cutouts, they are always taken into account for nesting.

  Note: The cutouts are taken into account even if the parameter is disabled in the nesting settings.

Take flexible tandem gripping into account

This option can be used to nest large parts to be sorted with tandem grippers with a specific angle of rotation on a cutting plan in Part Nester.

Common Cut

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This option allows the combining of parallel contours during nesting based on certain criteria. These criteria are defined in the export settings for the pre-processor in the Part Nester module *(See: ›Common Cut‹ Page 1153)*.

Tube Editor nesting settings

This information is only needed for nesting tubular parts in the Tube Nester module.

Roll allowed

When nesting the part, it can be rotated through a certain angle about the tubular axis. For circular tubes, every angle is possible for rolling the part. For rectangular or polygonal tubes, only certain roll angles are possible (e.g. rectangular tube = 180°, hexagonal tube = 60°).

Reverse allowed

When nesting the part, it can be turned through 180° about the vertical axis *(See: ›Reverse part‹ Page 1288)*.

Bending

How do I open this tab?

Select: **Modules > Part Editor > Part > Part > Part properties**

The press brake (bending machine) assigned to the part, the bending method and the tools are defined in this tab.

Bending process

Machine

Defines the bending machine on which you wish to process the bending part. Clicking on the arrow symbol on the right lets you select a different bending machine from a list. This list only contains those bending machines that are available in the system.



Tip: You can create bending machines that do not exist yet in Administrator.

Number of operators

Defines the number of operators required to bend the part. Criteria for determining the number of operators are size, bend length, weight and complexity of the bending part.

Automatic

If the option is enabled, the number of operators for the bending part is determined automatically based on a table of limits *(See: ›Limits for individual operation‹ Page 101)*

Bending-time table

Defines the table for automatic calculation of bending time. You can edit tables in Administrator *(See: ›Bending-time table‹ Page 99)*.

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Bending model

Min. area of inside contour

Bending parts with many small holes can slow down the 3D display and the export of the bending part considerably. To prevent this, you can use this option to limit the display of small inner contours.

All inner contours which are smaller than the value entered will no longer be displayed in the 3D display of the bending part.

Note: Hidden inner contours will also no longer be exported when the bending plan is exported. Consequently, these contours are also not displayed on the bending machine.

Tip: To determine the area of an inner contour which is to be hidden, select the **Contour properties** function in the **Geometry** menu.

Compatible relief

When you export bending programs for ByVision 1.x, errors may occur in the 3D display. Enable this option to prevent this.

General

Notes

You can enter general bending information for the machine operator in this box. The information is displayed on the bending machine in the bending program.

Attachment

How do I open this tab?

Select: **Modules > Part Editor > Part > Part > Part properties**

On this tab you can link any files or websites with the part. Double-click to run files. Links can also be opened in ByVision Bending. This allows you to give the operator additional information (e.g. original drawing).

Note: Files are only saved as links. In order for the operator to be able to open these files in ByVision Bending, they must be saved on a network drive to which the operator has access.

Tip: You can drag and drop the files from a file explorer into the dialog.

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7.3.2.2 Import geometry



Where do I find the function?

Select: **Part Editor > Geometry > Contour**

This function can be used to import geometries in all known file formats into a part. It deletes the existing geometry.

Application:

Creating several parts and assigning geometries to the parts should be done in two steps.

- **Keep existing geometry**

  This imports the new geometry into the part without deleting the existing geometry.

7.3.2.3 Start CAD



Where do I find the function?

Select: **Modules > Part Editor > Part > Part**

This function opens the current part in an external CAD program. The geometry of the part can then be post-edited in this program. When you close the CAD program, any changes to the geometry of the part are adopted in BySoft CAM.



Attention! If the modified part is already nested in one or more jobs, the changes will also be reflected in these jobs. Check these jobs to make sure that the modified contours do not overlap other parts. The changes do not affect cutting plans that have already been created.

7.3.2.4 Launch Bend Robot



Where do I find the function?

Select: **Modules > Part Editor > Part > Part**

This function opens the current part in the Bend Robot program. This program is used to program robot machining of bend processes.



Attention! If the bending part contains embedded elements, a message is displayed. It may not be possible to edit the part in Bend Robot. Cancel exporting to Bend Robot and assign the correct elements to the bending part in BySoft CAM.



Note:

- If the bending part does not exist in Bend Robot, a new Bend Robot project is automatically created for the part.
- If the bending part exists in Bend Robot, the program automatically opens the appropriate RoboPart in Part Manager.

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7.3.2.5 Set origin



Where do I find the function?

Select: **Modules > Part Editor > Part > Part**

This function can be used to set the origin of a part in a new position of the part. You have the following options for setting the origin of a part in a new position:

- Click on any position to set the origin here.
- Use the Snap functions to set the origin at a specific position on the contour.
- Specify the position of the origin using X and Y-coordinates.
- Set the origin on one of the points of the bounding rectangle.

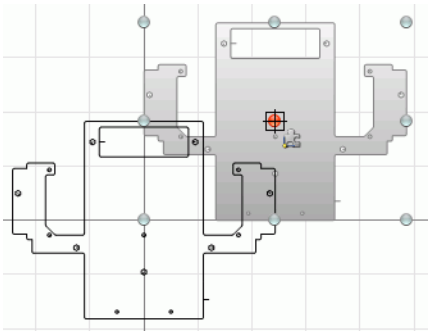


Fig. 7-22: Preview of the new origin on a part

7.3.2.6 Scale



Where do I find the function?

Select: **Modules > Part Editor > Part > Part**

This function can be used to scale the size of a part. Scaling is performed using the **Scaling factor** or the preset **Scale of original drawing**.



Note: The scaled part is shown as a preview based on the selected scaling factor.

Scaling a part to its original size

The preset scaling factors provide a simple way of scaling back to its original size a part that was imported from a drawing in a certain scale.

Scaling a part using a scale

Assume that we have imported a part with the scale 2:1. The part is therefore twice as large as the original.

1. Select the **Scale** function.
2. Under **Scale of original drawing**, select the scale **2:1**. The part is halved in size.
3. Click on the **Apply** button.



Note: Since the preset scaling factors relate to the original drawing, scaling is performed as the inverse of the selected factor.

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Reference

When scaling using a reference dimension, a new dimension can be assigned to an element of a part. All elements of the part are then scaled in proportion to this dimension.

The **Measure** option can be used to measure the reference dimension. Use the Snap functions to do this.

Scaling a part using a reference dimension

1. Select the **Measure** option.
2. Click on the first point of the reference dimension to be measured. Use the Snap functions to do this.
3. Click on the second point of the reference dimension to be measured. The **Reference length** field displays the measured length.
4. Enter the new length in the **New length** field.
5. Click on the **Apply** button.

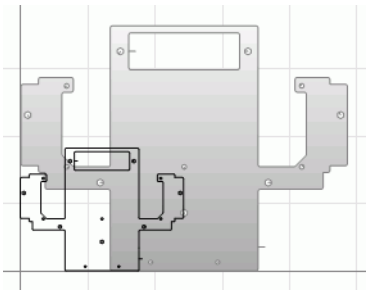


Fig. 7-23: Preview of the part with the scaling factor 2:1

What each option does:

- **Scaling factor**

  You can enter any scaling factor you wish in this field.

  - Values > 1 increase the size of the contour.
  - Values <1 decrease the size of the contour.

- **Scale of original drawing**

  You can use these scaling factors to scale the imported parts back to their original size.

7.3.2.7 Rotate



Where do I find the function?

Select: **Modules > Part Editor > Part > Part**

You can use this function to rotate the part through a specified angle. You specify the angle of rotation in the Angle of rotation field. You can rotate the part in 15° steps about the relevant center of rotation by clicking and dragging with the mouse.



Tip: Click on the right arrow next to the button to rotate the part ± 45° or 90°.

The part is rotated about the origin by default. You have the following options for specifying the center of rotation for a part:

- Clicking on any position to set the center of rotation here.
- Using the Snap functions to set the center of rotation at a specific position on the contour.
- Specifying the center for rotation by entering the X/Y-coordinates in the **Center X/Y** fields.



Note: Alternatively, you can use a reference angle to define the angle of rotation.

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Rotation using a reference angle

1. Select the **Measure** option.
2. Define the starting point for the angle on a contour. Use the Snap functions to do this.
3. Define the finishing point for the angle on a contour. The measured angle is displayed in the **Reference angle** field.
4. Enter the new angle in the **New angle** field.

Alternatively, you can also specify the difference angle between the old and new angle of rotation in the **Angle of rotation** field.

5. Click on the **Apply** button to adopt the angle of rotation.

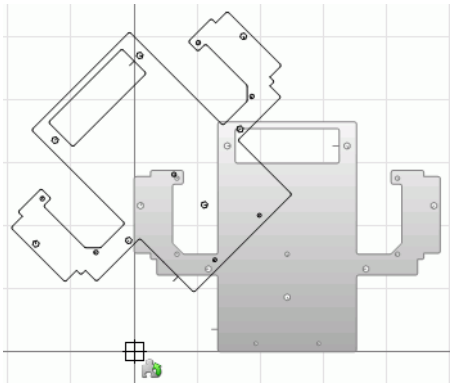


Fig. 7-24: Rotating a part about the origin

7.3.2.8 Rotate profile



Where do I find the function?

Select: **Module > Tube Editor > Tube > Tube**

This function can be used to rotate the profile of a rectangular tube through 90° so that the width becomes the height and vice versa.

The function can be used for rectangular tubes with the same dimensions but different developed views so that they can be nested together.

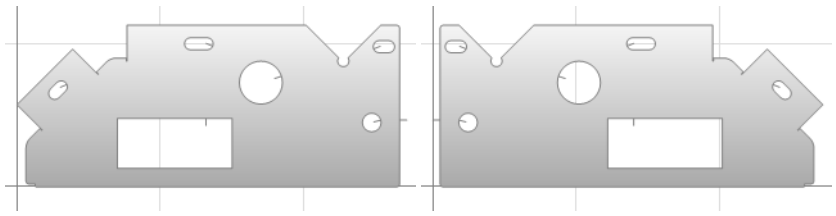
7.3.2.9 Reflect about Y-axis



Where do I find the function?

Select: **Modules > Part Editor > Part > Part**

This function mirrors the part about the origin on the Y-axis.



Note: The origin of the part is not reflected and therefore remains in the same position.

Fig. 7-25: Part reflected about vertical axis (horizontal reflection)

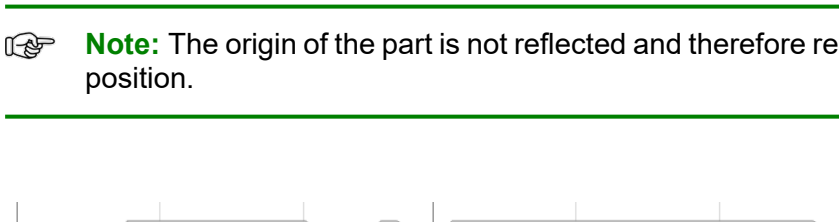
7.3.2.10 Reflect about the X-axis



Where do I find the function?

Select: **Modules > Part Editor > Part > Part**

This function is used to reflect the part about the vertical axis passing through the origin of the part.



Note: The origin of the part is not reflected and therefore remains in the same position.

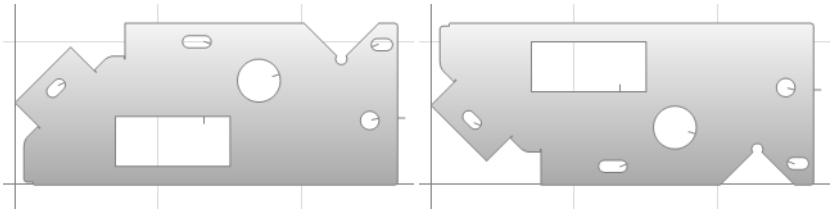


Fig. 7-26: Part reflected about horizontal axis (vertical reflection)

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7.3.2.11 Verify part



Where do I find the function?

Select: **Modules > Part Editor > Part / Cutting technology > Part**

This function checks the current part against various criteria. The **Verify part** docking dialog box displays the criteria used in the check.

☞ **Note:** The settings for checking criteria are saved for each part. This allows you to assign different testing criteria to parts.

Clear warnings

This function can be used to reset all current warnings. Once the function is run, the status will change to green. This reset status will be saved for the part.

☞ **Note:** As soon as changes are made to the part, the reset status will be reset.

Application:

Use this function if you have checked all the warnings and have found them to be irrelevant (e.g., warnings when there are open contours).

Links to verification criteria

If a criterion is associated with an error or warning, this is indicated by a red or yellow dot respectively. The text **select** and **edit** appears on the right beside the verification criterion.

- **select:** *Selects all elements or objects in the working area that are valid for the verification criterion.*
- **edit:** *Opens the appropriate edit function to remedy the problem.*

☞ **Note:** Only those criteria are displayed that apply to the current part.

Activating / Deactivating test criteria

If you do not want to check one or more test criteria, you can use the right button to deactivate them.

- ☑ Test criterion is applied
- ☐ Test criterion is not applied

Status of verification criteria

Status	Geometry / Cutting	Bending
	The verification criterion gets the green light. The part can be nested.	The verification criterion gets the green light. The part can be bent.
	There are one or more warnings for the verification criterion. The part can be nested.	There are one or more warnings for the verification criterion. The part can be bent.
	One or more errors have been identified for the verification criterion. The part cannot be nested.	One or more errors have been identified for the verification criterion. The part cannot be bent.
	The criterion has not been checked yet.	The criterion has not been checked yet.

Geometry

These examples show the basic procedure to follow for parts contours identified as having errors

Contours present

Displays the quantity of all contours of the part. Clicking on the text **Select** will select all the contours of the part.

Outside contours

Displays all outside contours of the part. This message is displayed if more than one outside contour was found. Clicking on the text **Select** will select all outside contours of the part.

Text outside of part

At least one text or text field is located outside of the part.

1. Select the text which is located outside of the part.
2. Move the text to a position inside the part using the **Move** function in the **Geometry** menu.

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Mirrored text

The part has been assigned the **Mirroring authorization** nesting option although there is text on the part.

- Remove mirroring authorization for the part in the properties.
- Remove the text from the part.

Open contours

The part contains one or more open contours.

1. Select the **Geometry** menu.
2. Select the open contours.
3. Click on the **Close** function. If this is not possible, close the contour in an external CAD program.
4. Save the change.

Contours shorter than 0.1 mm

The part has at least one contour that is shorter than the value specified for contours. Minimum contour length can be changed in the settings *(See: ›Minimum contour length‹ Page 347)*.

Possible steps:

- Delete contour
- Redraw contour

Short segmentation

The contour has one or more segments that are shorter than the value specified for segments *(See: ›Short segmentation‹ Page 347)*.

Possible steps:

- Delete reference points
- Smooth contour
- Clean up contour
- Redraw contour

Contour spikes

The tip of the contour is too acute and cannot be cut.

Possible steps:

- Delete acute contour
- Change critical angle *(See: ›Critical angle‹ Page 452)*
- Redraw contour

Overlapping contour(s)

Contours in a part overlap or touch each other.

1. Select the **Geometry** menu.
2. Select the contour that is overlapping another contour.
3. Delete the contour or select the **Move** function in order to move the contour out of the overlap.
4. Save the change.

Invalid geometries

The parts contains invalid geometries. The geometries are usually not visible.

1. Use the **Select** link to select the geometries.
2. Select the **Delete** link. This deletes the invalid geometries.

Cutting

These examples show the basic procedure to follow when cutting problems are identified.

Contours/texts without cutting technology

Contours or texts in a part do not have a cutting technology.

1. Select the **Cutting technology** menu.
2. Select the contours that do not have a cutting technology.
3. Select the **Automatic** function from the **Entire cutting technology** group.
4. Click **Apply** to assign the cutting technology to the contour.

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Contours without tool correction

At least one contour of a part does not have a tool correction.

1. Select the **Cutting technology** menu.
2. Select the contours that do not have a tool correction.
3. Select the **Contour** function from the **Cutting properties** group.
4. In the **Cutting properties, contour** dialog box, assign the appropriate correction to the contours.

Contours without machining sequence or with same machining sequence

At least one contour of a part does not have a machining sequence or has a double machining sequence.

1. Select the **Cutting technology** menu.
2. Select the **Complete** function from the **Machining sequence** group.
3. Click **Apply** to assign the machining sequence to the contour.

Lead-ins/lead-outs intersect contours

At least one lead-in/lead-out intersects a contour.

1. Use the **select** link to select the contour.
2. Select the **Cutting technology** menu.
3. Select the **Manual** function from the **Lead-in/lead-out** group.
4. Move the lead-in to a different position or shorten the lead-in until it no longer intersects the contour.

Lead-ins/lead-outs inverted on segment side

At least one contour lead-in/lead-out is on the wrong side.

- The lead-in/lead-out of an outside contour is on the inside of the contour
- The lead-in/lead-out of an inside contour is on the outside of the contour

1. Use the **select** link to select the contour.
2. Select the **Cutting technology** menu.
3. Select the **Manual** function from the **Lead-in/lead-out** group.
4. Select the correct segment side in the **Alignment** section.

☞ **Note:** If the segment side is intentional on semi-finished products, for example, you can ignore this message. The part can still be cut.

Lead-ins on “Cut first” segment

At least one lead-in is not on a segment with the **Cut first** attribute.

1. Use the **select** link to select the contour.
2. Select the **Cutting technology** menu.
3. Select the **Automatic** function from the **Lead-in/Lead-out** group. The lead-in is automatically moved to the segment with the **Cut first** attribute.

Risk of collision

Cut contours are flagged with the **Collision risk** status if the nozzle passes over them again after cutting.

Only those contours are flagged as dangerous that are between the minimum and maximum X/Y dimension. The size range is defined in the Cutting machine dialog in Administrator *(See: ›Collision calculation‹ Page 108)*.



Note: Only parts smaller than twice the distance between two cutting grates can cause a collision.

ByJet

Invalid tool grouping

In multi-head cutting in a single part, the contours belonging to the same tool group are not congruent.

Example: Square and triangle.

- Assign congruent contours to the tool group *(See: ›Requirements for multi-head cutting‹ Page 1383)*.

Invalid tool distance

In multi-head cutting in a single part, tool distance is outside the minimum and maximum range.

- Assign the contours to a different group or cutting head that is inside the range *(See: ›Multi-head cutting in a single part‹ Page 1382)*.

Bending

These examples show the basic procedure for bending problems

Warnings

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General warnings exist. The active warnings can be displayed using the **Select** link.

Bend line does not end on contour

At least one bend line does not end on the contour of the part.

1. Select the bend lines.
2. Select the **Correct** function from the **Geometry** menu. The bend lines will be extended until they reach the contour.

Contours in the entry zone

Contours of a part are located in the entry zone of bend lines.

- Move the contours out of the entry zone.
- Insert relief cuts.
- Use pressing tool.

Overlapping bend zones

There are one or more overlapping bend zones on the bending part.

- In the CAD program, create corner reliefs on the bend lines concerned
- In the CAD program, enlarge the corner reliefs of the bend lines concerned.
- Use tool with smaller radius.

Violation of minimum side length

It has one or more sides which are shorter than the bend entry zone.

- Select a bending tool with a smaller entry zone.
- Made the side longer.

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Z-bend

The program has detected two bend lines with different directions of bending that are too close together. The Z-bend can only be made with pre-bending in order to avoid a collision.

- Use the **Pre-bending** entry to insert a pre-bend in the bending process.

Example:

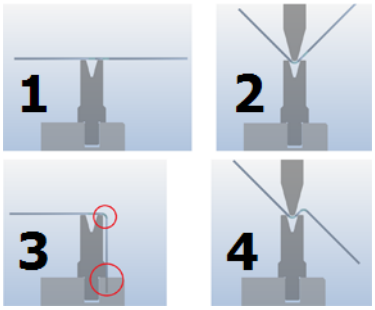


Fig. 7-27: Z-bend without pre-bending

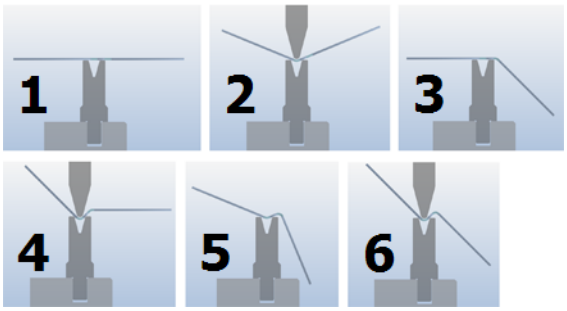


Fig. 7-28: Z-bend with pre-bending

Technology areas: Tools, Bending sequence, Tool setup plan, Breach of the 75% rule, Backgauges, Bending aid, Angle measurement system, Bending data

There are one or more warnings or error messages for the technology areas concerned. The messages are displayed in dialog boxes in the corresponding menus. The action to take depends on the message for the particular area.

☞ **Note:** To display the message, move with the mouse in the relevant dialog box over the icon in the **Status** column (/).

7.3.3 Group: View

Where do I find the group?

Select: **Module > Part Editor > Part / Geometry**

This group contains all the functions that can be used to change the current view of the part.

The various functions and views are classified as follows:

Zoom functions

The zoom functions can be used to magnify or reduce objects in the graphics window.



Tip: You can also perform the zoom functions using the mouse thumb wheel.

Moving the view

The arrow symbols are provided in this group for moving the view of an object.



Tip: You can use the center mouse button to move the object freely in the current view. The view is moved by dragging and dropping with the mouse.

Special view

The **Special view** function is used to display various view options. The special-functions display can be specified by clicking on the arrow symbol and enabled/disabled by clicking on the symbol above it.

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7.3.3.1 Special view



Where do I find the function?

Select: **Modules > Part Editor > Part / Geometry > View**

This function can be used to display various elements on the contour of the part. The elements are shown/hidden by clicking on the relevant option. The elements can be displayed separately or simultaneously.

You can display the following elements:

- **Contour reference points:** *Displays the reference points on all contours of a part.*
- **Bounding rectangles:** *Displays the bounding rectangle for all contours of a part.*
- **Machining direction:** *Displays the machining direction for all contours of a part.*
- **Open contours:** *Displays all open contours of a part.*
- **Bend zone:** *Shows the area of the bend zone on a bend line as a gray area.*
- **Entry zone:** *Shows the area of the entry zone on a bend line as a yellow area.*

7.3.3.2 Shade part



Where do I find the function?

Select: **Modules > Part Editor > Part / Geometry > View**

This function can be used to improve the display of the area of a part by shading it with a gray color ramp. The color used for shading cannot be changed.

7.4 Menu: Geometry

Where do I find the menu?

Select: **Modules > Part Editor**

In this menu, geometry functions can be applied to the part. Only limited changes to the geometry are possible. Fundamental changes to the geometry can only be made using the Start CAD function.

Functional groups

The function bar contains the following functional groups:

- Selection
- Contour
- Bend line
- Corner relief
- Text
- Parts ID
- Dimensioning
- View
- Snap
- Measure

Functions for changing the geometry

These functions can be used to change the geometry of a part:

- Delete
- Close
- Move
- Copy
- Rotate
- Stretch
- Scale
- Round off
- Offset
- Smooth
- Insert text

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Functions for changing the bend lines

- Define bend
- Correct bend line

7.4.1 Group: Properties

Where do I find the group?

Select: **Modules > Part Editor > Geometry**

This group contains functions that display the properties of various geometry elements.

7.4.1.1 Geometry properties



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function displays the cutting properties for the selected contour.

Geometry

This option can only be applied to construction lines or individual segments of a contour. The function is disabled for closed contours.

Construction geometry

Defines the selected lines or line segments as a construction geometry. No cutting technology can be set on construction geometries.

Application:

- To assist construction
- Geometries in the cutting plan that are not to be cut

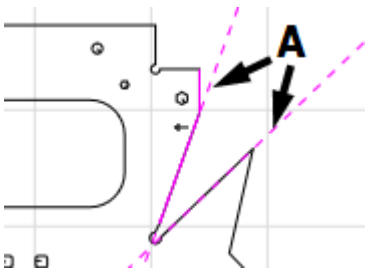


Fig. 7-29: Construction line (**A**) as construction geometry on a part

Contour

Engraving

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This option can be used to control the weight calculation and nesting in inner contours. This option is identical to the **Engraving** option in the **Cutting technology** menu*(See: ›Engraving‹ Page 681)*.

If for an inner contour, the **Engraving** option is enabled and the **Shade part** option is enabled, the inner contour is no longer transparent but is displayed as a shaded area of the part.

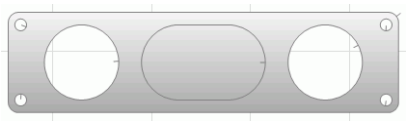


Fig. 7-30: Middle inside contour with **Engraving** option enabled

☞ **Note:** For contours with the **Engraving** operating mode, the **Engraving** option is enabled by default.

Bend marking

This option defines the contour as a bend mark *(See: ›Bend marking‹ Page 476)*.

Sortable waste

This option can be used to define inside contours of a part as sortable waste. If the part is cut on a machine with a sorting unit, the inside contour can be sorted to a stacking site.

The sortable waste parts can be stacked to a stacking site via the **Waste parts** tab in the following modules.

- Part Nester -> Sort Editor -> Sort data
- Sort Job Editor -> Job data

Z height

This option allows you to define an inside contour as an extrusion or embossing (**A**). As a result, the program detects any collisions caused by the extrusion when calculating the bend.

The extrusion can be created by entering positive or negative values on the top and bottom of the bending part.

☞ **Note:** The Z dimension is always measured from the bottom of the part (**B**). Therefore, it is necessary to factor in the thickness of the material if values are positive (**C**).

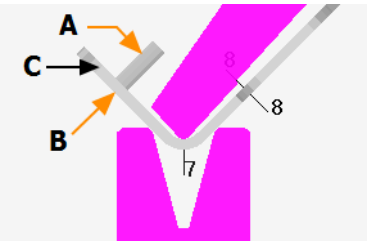


Fig. 7-31: Extrusion of an inside contour as a bolt

The options can be used for the following applications:

- **Weight calculation:** *If the **Engraving** option is disabled for an inner contour, the area of the inner contour is included in the weight calculation.*
- **Nesting in inner contours:** *If the **Engraving** option is enabled for an inner contour, parts can no longer be nested in the area of the inner contour.*
- **Hiding inner contours for bending parts:** *To determine the area of an inner contour for bending parts so that it can be hidden in the Parts properties for the 3D display using the **Min. area of inside contours** option.*

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- **Bending parts with embossing:** *Takes embossing on a bending part into account with the **Z height** option, e.g. bolts or nipples when calculating bending.*

☞ **Note:** This function has no effect on the automatic assignment of the cutting technology. An inner contour remains an inner contour and as such is still assigned the **Cutting** operating mode.

Tolerance

These options can be used to set a tolerance range on the selected contour. The geometry is set by the postprocessor in the middle of the tolerance range. By default, **0** is used for both values.

☞ **Note:** The option is only suitable for contours for which the same tolerance range may apply over the whole contour.

Typical applications for contours:

- Round hole
- Square / rectangle
- Slot

Upper value

This option is used to specify the upper tolerance. You can use positive or negative values. The value must be greater than or equal to the lower value.

Lower value

This option is used to specify the lower tolerance. You can use positive or negative values. The value must be less than or equal to the upper value.

Example:

Tolerance in a round hole

- Diameter: **30 mm**
- Upper value: **0.1 mm**
- Lower value: = **-0.3 mm**

1. The above tolerances are assigned to a round hole of 30 mm.

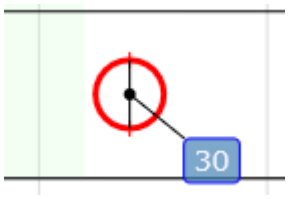
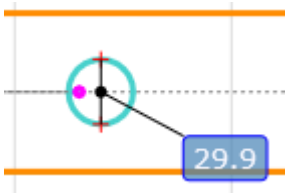


Fig. 7-32: Circular contour in Part Editor

2. The postprocessor creates a contour in the middle of the tolerance range.



Diameter moved to middle of tolerance range in cutting plan.

7.4.1.2 Segment properties



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function displays the properties of the selected segment.

This provides a simple way of obtaining information about the start and end points, midpoint and dimensions of the selected segment without needing to measure them first using the measure functions.

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7.4.2 Group: Draw

Where do I find the function?

Select: **Modules > Part Editor > Geometry**

This group contains all the functions for drawing geometries.

7.4.2.1 Draw line

Where do I find the function?



Select: **Modules > Part Editor > Geometry > Draw**

Draws one or more lines. You can draw lines with the mouse or by entering coordinates, lengths and angles in the input dialog (**A**).

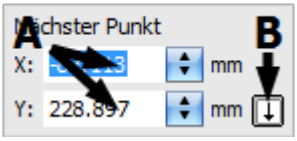


Fig. 7-33: Draw line input dialog



Note: After confirming the coordinates, press ENTER or left-click to continue the function.



Tip: All snap functions are available when using the mouse to draw lines.

Keyboard commands

- **ESC** = Cancels the drawing command.
- **CTRL + Z** = Deletes the last geometry drawn.
- ↓ = Displays a dialog with various options for drawing geometries (**B**). Depending on the number of geometries drawn, only the applicable options are available.

Options

- **Close**

  Closes the polyline by drawing a line from the end point to the start point of the polyline.
- **Tangent**

  Draws a tangential line between two geometries.

- **Back**
  Deletes the last line drawn.
- **Absolute**
  Defines a point by entering absolute coordinates.
- **Relative**
  Defines a point by entering relative coordinates.
- **Polar absolute**
  Defines a point by entering an absolute length and an angle.
- **Polar relative**
  Defines a point by entering a relative length and an angle.

Drawing a line by entering absolute coordinates



Fig. 7-34: Contour with coordinate definition

1. Select the **Line** function.
2. Enter X/Y coordinates **20,120** for the starting point.
3. Press ENTER to confirm.
4. Enter X/Y coordinates **20,20** for the next point.
5. Press ENTER to confirm.
6. Enter X/Y coordinates **50,20** for the next point.
7. Press ENTER to confirm.
8. Enter X/Y coordinates **70,55** for the next point.
9. Press ENTER to confirm.
10. Enter X/Y coordinates **70,120** for the next point.
11. Press ENTER to confirm.
12. Select the arrow key ↓.
13. Select the **Close** option to return the polyline to the starting point.

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Drawing a line with polar coordinates

The following example shows how to draw a polyline using polar coordinates. To enter a polar coordinate, enter the distance and an angle.

By default, angles get bigger counterclockwise and smaller clockwise. To perform a counterclockwise movement, enter a negative angle.

☞ **Note:** Entering 10<315 has the same result as entering 10<-45.

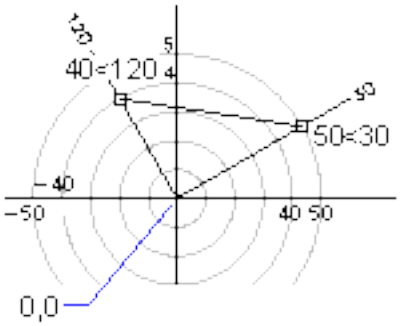


Fig. 7-35: Coordinate system for polar coordinates

1. Select the **Line** function.
2. Select the arrow key ↓.
3. Select the **Polar absolute** option.
4. Enter a length of **40 mm** and an angle of **120°**.
5. Press ENTER to confirm.
6. Select the arrow key ↓.
7. Select the **Polar absolute** option.
8. Enter a length of **50 mm** and an angle of **30°**.
9. Press ENTER to confirm.

Drawing horizontal and vertical lines with the mouse

Enable the **Orthogonal** snap function to draw horizontal or vertical lines. This means that the polyline can only be set in the horizontal or vertical direction.

7.4.2.2 Draw arc



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Draw**

Draws an arc clockwise or counterclockwise.

Note: After confirming the coordinates, press ENTER or left-click to continue the function.

Tip: All snap functions are available to set the start and end point.

Keyboard commands

- **ESC** = Cancels the drawing command.
- **CTRL + Z** = Deletes the last geometry drawn.
- ↓ = Displays a dialog with various options for drawing geometries (**B**). Depending on the number of geometries drawn, only the applicable options are available.

Options

- **Absolute**

  Defines a point by entering absolute coordinates.
- **Relative**

  Defines a point by entering relative coordinates.
- **Polar absolute**

  Defines a point by entering an absolute length and an angle.
- **Polar relative**

  Defines a point by entering a relative length and an angle.
- **Radius**

  Defines the radius of the arc between the two points. This setting must be greater or equal to half the distance between the two points.

  Note: This option is used to draw an arc element up to half the circumference of a circle.

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Midpoint:
Defines the midpoint of the arc between the two points.

> **Note:** This option is used to draw an arc element of semicircle size or greater.

Examples

Drawing an arc with a radius



Fig. 7-36: Arc with arc radius

1. Select the **Arc** function.
2. Enter X/Y coordinates **50,50** for the starting point.
3. Press ENTER to confirm.
4. Enter X/Y coordinates **100,70** for the end point.
5. Press ENTER to confirm.
6. Enter an arc radius of **60**.
7. Press ENTER to confirm.

7.4.2.3 Draw circle



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Draw**

Draws a circle by defining radius or diameter.

Note: After confirming the coordinates, press ENTER or left-click to continue the function.

Tip: All snap functions are available when setting the midpoint.

Keyboard commands

- **ESC** = Cancels the drawing command.
- **CTRL + Z** = Deletes the last geometry drawn.
- ↓ = Displays a dialog with various options for drawing geometries (**B**). Depending on the number of geometries drawn, only the applicable options are available.

Options

- **Radius**
  Draws a circle by defining a radius.
- **Diameter**
  Draws a circle by defining a diameter.

Examples

Drawing a circle by defining a radius

1. Select the **Circle** function.
2. Enter X/Y coordinates **0,0** for the midpoint.
3. Press ENTER to confirm.
4. Enter a radius of **50**.
5. Press ENTER to confirm.

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7.4.2.4 Draw ellipse

Where do I find the function?

Select: **Modules > Part Editor > Geometry > Draw**

Draws an ellipse defined by the midpoint and two axis end points. The position of the two axis end points defines the angular position of the ellipse.

Note: After confirming the coordinates, press ENTER or left-click to continue the function.

Tip: All snap functions are available for setting the midpoint and axis end points.

Keyboard commands

- **ESC** = Cancels the drawing command.
- **CTRL + Z** = Deletes the last geometry drawn.
- ↓ = Displays a dialog with various options for drawing geometries (**B**). Depending on the number of geometries drawn, only the applicable options are available.

Examples

Drawing an ellipse

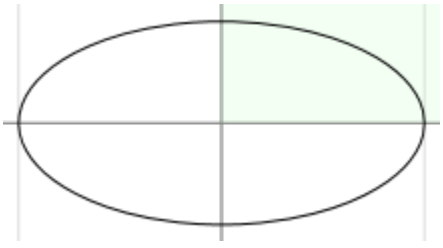


Fig. 7-37: Ellipse with axis end points

1. Select the **Ellipse** function.
2. Enter X/Y coordinates **0,0** for the midpoint.
3. Press ENTER to confirm.
4. Enter X/Y coordinates **100,0** for axis end point 1.
5. Press ENTER to confirm.

6. Enter X/Y coordinates **0,50** for axis end point 2.
7. Press ENTER to confirm.

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7.4.2.5 Draw rectangle



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Draw**

Draws a rectangle by entering two corner points.

Note: After confirming the coordinates, press ENTER or left-click to continue the function.

Tip: All snap functions are available for setting the end points.

Keyboard commands

- **ESC** = Cancels the drawing command.
- **CTRL + Z** = Deletes the last geometry drawn.
- ↓ = Displays a dialog with various options for drawing geometries (**B**). Depending on the number of geometries drawn, only the applicable options are available.

Options

- **Absolute**
  Defines a point by entering absolute coordinates.
- **Relative**
  Defines a point by entering relative coordinates.
- **Polar absolute**
  Defines a point by entering an absolute length and an angle.
- **Polar relative**
  Defines a point by entering a relative length and an angle.

Examples

Draw rectangle

1. Select the **Rectangle** function.
2. Enter X/Y coordinates **0,0** for the first corner point.
3. Press ENTER to confirm.

4. Enter X/Y coordinates **100,30** for the second corner point.
5. Press ENTER to confirm. The program draws a rectangle with a length of **100** and a height of **30**.

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7.4.2.6 Draw polygon



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Draw**

Draws a polygon with up to 100 sides of equal length.

Note: After confirming the coordinates, press ENTER or left-click to continue the function.

Tip: All snap functions are available to set the midpoint or start and end point.

Keyboard commands

- **ESC** = Cancels the drawing command.
- **CTRL + Z** = Deletes the last geometry drawn.
- ↓ = Displays a dialog with various options for drawing geometries (**B**). Depending on the number of geometries drawn, only the applicable options are available.

Options

- **Absolute**
  Defines a point by entering absolute coordinates.
- **Relative**
  Defines a point by entering relative coordinates.
- **Polar absolute**
  Defines a point by entering an absolute length and an angle.
- **Polar relative**
  Defines a point by entering a relative length and an angle.

Examples

Drawing a polygon with a midpoint

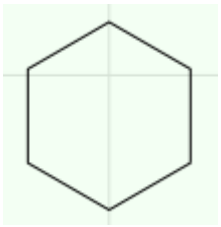


Fig. 7-38: Polygon with midpoint and incircle radius

1. Select the **Polygon** function.
2. Define the number of sides of the polygon as **6**.
3. Press ENTER to confirm.
4. Enter X/Y coordinates **100,80** for the midpoint.
5. Press ENTER to confirm.
6. Enter an incircle radius of **40**.
7. Press ENTER to confirm.
8. Enter **90°** for the angle of the start side of the polygon.
9. Press ENTER to confirm.

Drawing a polygon by defining sides

This input method defines the length and position of a side by entering start point, end point and angle.

1. Select the **Polygon** function.
2. Define the number of sides of the polygon as **6**.
3. Press ENTER to confirm.
4. Select the arrow key ↓.
5. Select the **Side** option.
6. Enter X/Y coordinates **50,50** for the starting point.
7. Press ENTER to confirm.
8. Enter X/Y coordinates **80,50** for the end point.
9. Press ENTER to confirm.
10. Enter **0°** for the angle of the start side of the polygon.
11. Press ENTER to confirm. The program draws the polygon with six sides of length **30 mm**.

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7.4.2.7 Draw slot



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Draw**

Draws a slot with the specified dimensions and the midpoint at the center of the slot. Enter an rounding radius to influence the appearance of the slot as follows:

- **Rounding radius is 0**
  Draws a rectangle.
- **Rounding radius is less than half the width**
  Draws a rectangle with rounded corners.
- **Rounding radius equals half the width**
  Draws a slot.

 **Note:** The default rounding radius is always half the width of the slot.

 **Note:** After confirming the coordinates, press ENTER or left-click to continue the function.

 **Tip:** All snap functions are available when setting the midpoint.

Keyboard commands

- **ESC** = Cancels the drawing command.
- **CTRL + Z** = Deletes the last geometry drawn.
- ↓ = Displays a dialog with various options for drawing geometries (**B**). Depending on the number of geometries drawn, only the applicable options are available.

Options

- **Absolute**
  Defines a point by entering absolute coordinates.
- **Relative**
  Defines a point by entering relative coordinates.
- **Polar absolute**
  Defines a point by entering an absolute length and an angle.

Polar relative

Defines a point by entering a relative length and an angle.

Examples

Drawing a slot with defaults



Fig. 7-39: Slot with defaults

1. Select the **Slot** function.
2. Enter a slot length of **100**.
3. Press ENTER to confirm.
4. Enter a slot width of **30**.
5. Press ENTER to confirm.
6. Enter a rounding radius of **15**.
7. Press ENTER to confirm.
8. Enter X/Y coordinates **60,30** for the midpoint.
9. Press ENTER to confirm.
10. Enter a slot angle of **0°**.
11. Press ENTER to confirm.

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7.4.2.8 Draw construction line



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Draw**

Draws a construction line using different methods.

Note: After confirming the coordinates, press ENTER or left-click to continue the function.

Tip: All snap functions are available for setting points.

Keyboard commands

- **ESC** = Cancels the drawing command.
- **CTRL + Z** = Deletes the last geometry drawn.
- ↓ = Displays a dialog with various options for drawing geometries (**B**). Depending on the number of geometries drawn, only the applicable options are available.

Options

- **Parallel**
  Draws a construction line parallel to a segment.
- **Bisector**
  Draws a construction line as a bisector between two lines.
- **Perpendicular bisector**
  Draws a construction line as a perpendicular bisector between two lines.
- **Absolute**
  Defines a point by entering absolute coordinates.
- **Relative**
  Defines a point by entering relative coordinates.
- **Polar absolute**
  Defines a point by entering an absolute length and an angle.
- **Polar relative**
  Defines a point by entering a relative length and an angle.

Examples

Drawing a construction line with default settings

1. Select the **Construction line** function.
2. Enter X/Y coordinates for the first point.
3. Press ENTER to confirm.
4. Enter X/Y coordinates for the second point.
5. Press ENTER to confirm.

Drawing a construction line parallel to a segment

1. Select the **Construction line** function.
2. Select the arrow key ↓.
3. Select the **Parallel** option.
4. Select the segment to draw a parallel construction line to.
5. Enter the X/Y distance to the segment.
6. Press ENTER to confirm.

Construction line as a bisector between two lines.

1. Select the **Construction line** function.
2. Select the arrow key ↓.
3. Select the **Bisector** option.
4. Select the first line.
5. Select the second line.

Draw a construction line as a line segment bisector between two points

1. Select the **Construction line** function.
2. Select the arrow key ↓.
3. Select the **line segment bisector** option.
4. Select the first point on a line.
5. Select the second point on a line.

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7.4.2.9 Draw construction geometry



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Draw**

Activates the mode for drawing construction geometries.

Construction geometries are displayed with broken lines and, unlike normal contours, consist of separate geometry elements. Construction geometries can be used to create intersections for trimming contours or geometry elements.

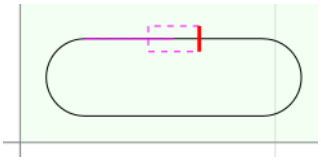


Fig. 7-40: Construction geometry over slot

7.4.3 Group: Edit

Where do I find the function?

Select: **Modules > Part Editor > Geometry**

This group contains all the functions for editing geometries.

7.4.3.1 Trim



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Edit**

Trims a geometry element up to the point of intersection with another geometry.

Trimming a geometry elements

1. Select the **Trim** function.
2. Holding the left mouse button to drag a line over the geometry element.

7.4.3.2 Stretch



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Edit**

Stretches lines or arcs up to defined edges.

Arcs can only be stretched in the direction of rotation. It is not possible to stretch them in the other direction.

7.4.3.3 Connect



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Edit**

Converts connected base elements into contours.

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7.4.4 Group: Contour

Where do I find the group?

Select: **Modules > Part Editor > Geometry**

This group contains functions that can be applied to the contour of a part.

7.4.4.1 Autoform



Where do I find the function?

Select: **Modules > Part Editor > Contour**

This function provides a simple means of creating contours from predefined geometric shapes *(See: ›Apply autoform‹ Page 1378)*.

Mode

In this area, you can specify the manner in which the shape (**A**) is applied to the original contour (**B**).

- **Cut**

  This option is used to cut the part having the selected shape and which is located in the material out of the contour (**C**).

- **Add**

  This option is used to add the part of the selected shape which is not located in the material to the contour (**D**).

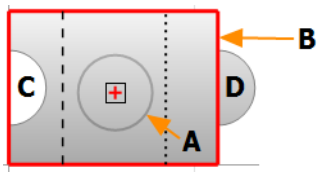


Fig. 7-41: Ways of applying shapes to a contour

Shape

In this area you select the type of contour shape that you want to create. The contour of the selected Autoform shape is displayed in the preview pane. The required parameters are displayed in the preview diagram as letters beside the relevant dimensioning lines, and can be set in the **Parameters** area.

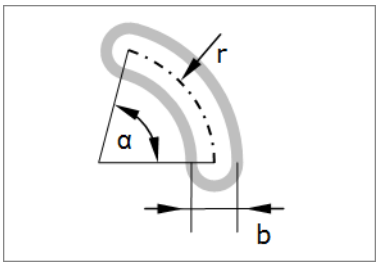


Fig. 7-42: Graphic preview of the shape of a bent slot

Special shape for auxiliary backgauge:

This shape can be used for bending parts where no stops for the bend line can be set on the outside contour due to the geometry of the part. To align the auxiliary backgauge to the corresponding bend line for which you want to create a stop, activate the **Align to bend line** option.

The auxiliary backgauge has two narrow tabs. The tab width (d) must be dimensioned so that the tabs can be easily removed from the part after bending.

Adding a backgauge to a contour

1. Select the **auxiliary backgauge** autoform in the **Shape** section.
2. Select **Add** mode to add the auxiliary backgauge to the part contour.
3. Activate the **Align to bend line** option.
4. Move the tabs of the auxiliary backgauge to the position of the outside contour opposite the bend line and click the mouse at this position. The auxiliary backgauge is connected to the outside contour.

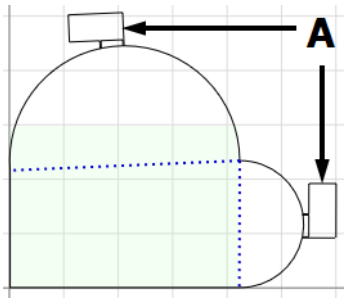


Fig. 7-43: Auxiliary backgauges (**A**) on an asymmetrical part contour

Position

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In this section you can define the position of the part with respect to the origin using X and Y-coordinates and the rotation of the contour. The reference point for the distance from the origin is always the center of the contour.

Alignment

This option is used to define the position at which to align the contour. Click on the appropriate point in the matrix at which to align the contour. Alignment is performed to the contour's bounding rectangle.

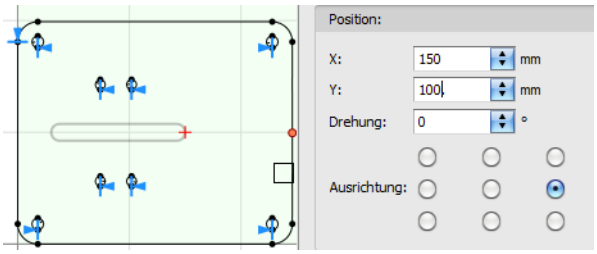


Fig. 7-44: Alignment position of a slot, center right

Tube Editor

Sector

This option can be used to move or insert contours, segment points or texts into the respective sectors of the tubular sections. The sectors are displayed on the left-hand side of the developed view of the tube (**A**).

Circular tubes are composed of four sectors. A rectangular or polygon tube contains the same number of sectors as it has faces.

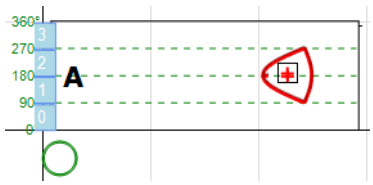


Fig. 7-45: Diagram showing the sectors for a circular tube

Tube angle

This option positions the geometry at a specific angle on the round tube.

Note: This option can only be used for round tubes.

Parameters

The parameters that are needed to create the particular Autoform shape are set in this area.

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7.4.4.2 Cutting waste into smaller sizes



Where do I find the function?

Select: **Modules > Part Editor > Contour**

This function is used to split inside contours by defining manual separating cuts. This ensures that cut-outs fall through the cutting grate in order to avoid collisions with the cutting head caused by material standing up.



Note: The separating cuts can only be set between segment end points.



Tip: For the postprocessor to be able to automatically cut waste into smaller sizes, the **Cut waste into smaller sizes** tag must be set on the contour in the menu **Cutting technology -> Cutting properties -> Contour** *(See: ›Tag‹ Page 684)*.

Mode

This option defines the direction in which the separating cut can be set.

- **Rectangular**

  Separating cuts can be set horizontally and vertically.

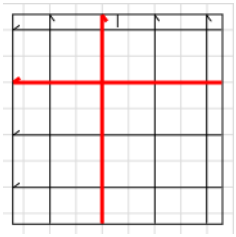


Fig. 7-46: Rectangular separating cuts in an inside contour

- **Horizontal**

  Separating cuts can only be set horizontally.

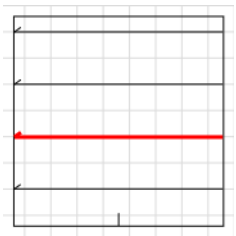


Fig. 7-47: Horizontal separating cuts in an inside contour

Vertical

Separating cuts can only be set vertically.

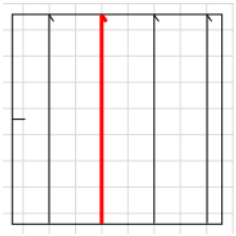


Fig. 7-48: Vertical separating cuts in an inside contour

Angle

Separating cuts are set at a certain angle to the segment end point. The angle of the separating cut is defined in the **Angle** option.

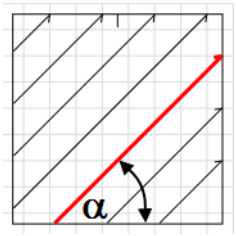


Fig. 7-49: Angled separating cuts in an inside contour by defining an angle (α).

Zigzag

Separation cuts are set as zigzags along the cutout. In order to be able to set a zigzag separation cut, the lead-in must be set on the narrow side of the cutout.

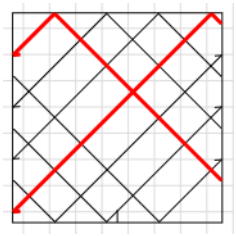


Fig. 7-50: Zigzag separating cuts in an inside contour

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Settings

Distance

This option defines the distance to the contour up to which the separating cut is performed (**A**). The kerf width is the default setting.

- **Lead-in**

  This option specifies the length of the lead-in on a separating cut (**B**). If length is **0**, no lead-in is performed, instead piercing is performed on the separating cut itself.

- **Angle**

  This option defines the angle of the separating cut if **Angle** mode is selected.

- **Offset**

  This option decreases the distance from the opposite side of the lead-in to the contour (**C**). The distance at the end of the separating cut results from the values of the **Distance** - **Offset** options.

  For cutting waste into smaller sizes, the side on which piercing begins is cut deeper than the side on which piercing ends. Therefore, this option can be used to place the distance of the end of piercing closer to the contour so that it is easier to break off the waste.

> **Note:** If **Zigzag** mode is selected, the offset is only applied to the end of the separating cut.

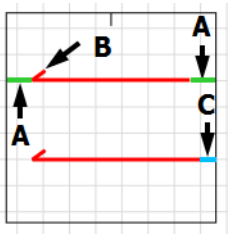


Fig. 7-51: Waste cut settings

7.4.4.3 Copy



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function copies the selected contour onto the clipboard. Several contours can also be selected and copied at once.

The **Paste** function or **Ctrl+V** shortcut will paste the copied contours back in the same position in the current part. At the same time, the **Move contour** docking dialog box is displayed, which lets you move the contour to the required position.

Duplicating inner contours of a part

1. Select the contours that you wish to duplicate.
2. Select the **Copy** function.
3. Select the **Insert** function. The copied contours will be inserted back in the same position in the part.
4. Move the contours to the required position.



Tip: The copied contours can also be inserted in another part using the **Ctrl+V** shortcut.

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7.4.4.4 Insert



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function pastes the contour from the clipboard into the current part using the original coordinates. If the contour is pasted into the same part, the **Move contour** docking dialog box opens automatically.

Pasting a contour from the clipboard into the same part

1. Select the **Insert** function. The contour is pasted over the original contour.
2. Select the base point of the pasted contour.
3. In the **Move contour** dialog box, specify the point that you wish to move the copied contour to.
4. Click on the **Apply** button.



Tip: The contour can also be moved using the Snap functions.

7.4.4.5 Delete



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function deletes the selected contour of the part.



Tip: You can select several contours at once using the **Shift** key and then delete them all together.

Set engraving



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function applies the **engraving** cutting technology to all selected contours.



Note: The inner contour is displayed filled-in on engravings.

Remove engraving



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function removes the **engraving** cutting technology from all selected contours.

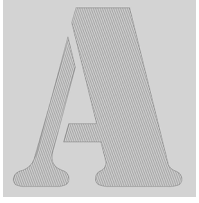
Hatching



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function applies hatching to the selected contour. You can define the spacing and angle of the lines for hatching.



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Application:

Note: The function can only be used if a contour is selected.

- To highlight engraved fonts or areas on a part.

Automatic groove

Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function is used to create groove on the selected circle contour. The groove can be created simultaneously on several selected circles.

Assign groove to a circle contour

1. Select the desired circle contour.
2. Configure the appropriate settings.
3. Press **Apply** to assign the groove to the circle.

Note: The cutting technology detects the groove on the circular contour and automatically places a pierce at its center.

- **Depth (a)**
  Determines the depth of the groove.
- **Width (y)**
  Determines the width of the groove.
- **Radius (r)**
  Determines the radius of the groove to the arc. The greater the value, the smoother the transition from groove to arc.
- **Rotation**
  Determines the midpoint of the groove on the circle.
  An angle of **0°** places the midpoint of the groove at the bottom end and **90°** at the top end of the circle.

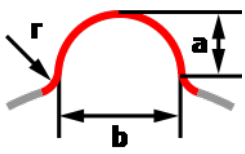


Fig. 7-52: Adjustable groove dimensions

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7.4.4.6 Edit



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function opens a drop-down menu from where you can select various editing functions for existing contours and segments.

The following edit functions are available:

- Resolve
- Separate
- Separate multiple
- Close
- Round off
- Stretch
- Scale
- Offset
- Move
- Mirror
- Rotate
- Row
- Clean
- Smooth
- Insert reference point
- Remove reference point

7.4.4.7 Create



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

Coming soon…

Settings



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit > Create > Settings**

This dialog contains options for creating contours. The contour tolerance for the English system of units can be specified separately.

Settings

Contour tolerance, metric/English

Using this option, several contours which lie at a certain distance from one another can be joined automatically to form a common contour. The contour tolerance causes all contours for which the distance is smaller than or equal to the value specified in the **Contour tolerance** field to be joined together.

Reducing the default value lowers the recognition rate. Increasing this value raises the recognition rate.

Correcting branches manually

If the program detects a branching of segments when you create contours, it is necessary to enter the branch point manually with this option enabled *(See: ›Resolving a branch point during DXF import‹ Page 1329)*.

If this option is disabled, the program automatically selects the next segment to continue along.

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7.4.4.8 Resolve



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function resolves all the selected contours into their separate geometric elements.



Note: Text and data fields are not resolved!

Connecting geometric elements

In a gear wheel, the geometric elements of the outer contour are only partially connected.

1. Select the **Resolve** function.
2. Reimport the part into BySoft CAM using the **No selection** function.

Note: If there are still not enough connected geometric elements after the import, run the **Resolve** function again and increase the value of the **Contour tolerance** option in the settings.

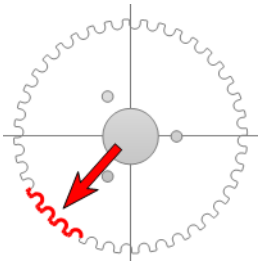


Fig. 7-53: Part in which the geometries of the outer contour are not connected

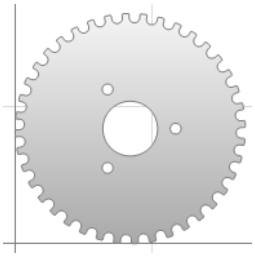


Fig. 7-54: Connected contour after resolution and reimporting

7.4.4.9 Separate



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function can be used to separate the contour into several contour segments. Click to split the contour at a certain point.

If the function is applied to a contour which has not yet been separated, the starting point of the contour is also split.



Tip: Use this function for multihead cutting on a part in order to separate corners of long parts. By separating, it is possible to assign two cutting heads to two parallel contour segments (**A**) *(See: ›Multi-head cutting in a single part‹ Page 1382)*).

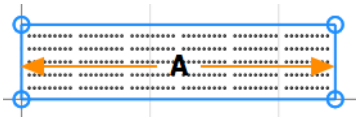


Fig. 7-55: Separated corners of an outer contour

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7.4.4.10 Separate multiple

Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function can be used to separate a contour into several contour segments in one step. The segments are distributed evenly over the entire contour.

Note: If the function is applied to a contour which has not yet been separated, the starting point of the contour is also split.

Contour, separate multiple

1. Use the mouse to select the contour to split into several contour segments.
2. Enter the desired number of contour segments in the **Quantity** field.
3. Click on the **Apply** button.

Tip: Use this function for multihead cutting on a part in order to split up a long straight contour into several contour segments (**A**). By separating, it is possible to assign several cutting heads to a very long contour *(See: ›Multi-head cutting in a single part‹ Page 1382)*).

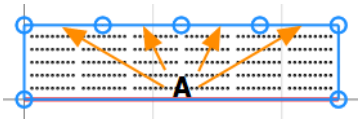


Fig. 7-56: Contour separated into several segments

7.4.4.11 Close contour

Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function can be used to close the end of an open contour by linking it to the start of the contour with a straight line. To close an open contour, select the contour and then click the **Close** function. The end of the contour is linked to the start of the contour by a straight-line segment.

7.4.4.12 Round off



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function can be used to round off or chamfer the corners of contours. Rounding-off or chamfering can only be applied to reference points on the contour.

Note: The options can only be applied to closed contours.

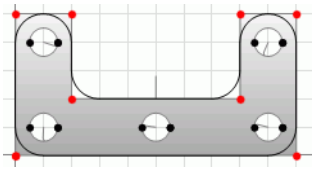


Fig. 7-57: Rounding off corners with a radius

- **Rounding**

  When this option is enabled, all selected reference points on the contours are rounded off with the value set in the **Radius** field.

  Corners that have already been rounded are not rounded off again.

- **Chamfer**

  When this option is enabled, all corners on the contours are chamfered. The value in the **Distance** field specifies the distance, viewed from the corner point, at which the chamfer shall start.

Tip: You can use the **Shift** key to select multiple reference points and the **Ctrl** key to remove the selection again.

Rounding off corners

1. Select the **Round off** function.
2. Select the contour whose corners you wish to round off.
3. Select the reference points where you want rounding-off to be performed.
4. Enter the rounding radius in the **Radius** field.
5. Click on the **Apply** button.

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7.4.4.13 Stretch



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function is used to move the reference points of a contour, allowing you to expand or compress regions of a contour.

Moving the reference points changes the contour in the X or Y-direction, according to the direction of the movement. The selected reference points can be moved using the mouse or by entering coordinates. Movement using the mouse can be performed in either the X-direction or Y-direction.

 **Note:** If there are any bending lines between the reference points to stretch, they are also moved.

You have the following options for selecting the reference points:

- Clicking on a single reference point.
- Dragging a rectangle with the mouse to select several reference points.

Moving the reference points of a contour

1. Select the **Stretch** function.
2. Select the reference points of the contour that you wish to move.
3. Select the base point of the reference points (optional).
4. Specify the move point for the reference points.
5. Click on the **Apply** button.

Base point

The base point of the selected element is displayed as absolute X and Y-coordinates in the **Base point** area.

You have the following options for selecting the base point:

- Clicking on any position.
- Left-clicking on a point in the alignment array.
- Selecting a Snap function.

Absolute/relative coordinates

You have the following options for selecting the move point:

- Clicking on any position.
- Entering a new X and Y value in the **Absolute coordinates** or **Relative movement** areas.
- Selecting a sector in the **Absolute coordinates** area (Tube Editor).
- Selecting a tube angle for round tubes (Tube Editor)
- Selecting a Snap function.

Tip: Information about the next step is displayed in the status bar at the lower left.

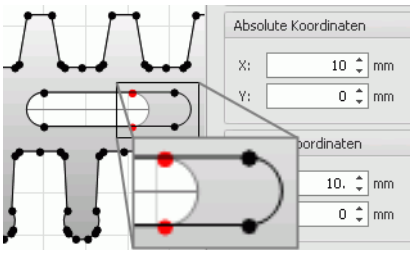


Fig. 7-58: Lengthening a slot by moving the reference points

Vector

This option is used to move contour reference points using the vectors **Angle** and **Distance**.

Move reference points via vectors

1. Select the **Stretch** function from the **Edit** function selection.
2. Select the reference points that you wish to move.
3. In the **Distance** box enter the distance you want to move the selected reference points.
4. In the **Angle** box enter the reference angle you want to move the contour. Select an angle of **0** for a horizontal shift.
5. Click on the **Apply** button.

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Angle

Defines the angle by which to move the selected reference point.

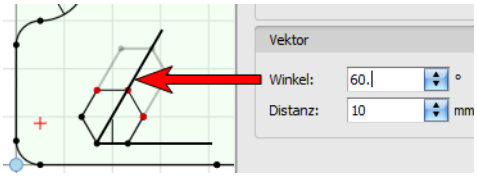


Fig. 7-59: Reference point shift with an angle

Distance

Defines the distance of the shift based on the selected reference point. A negative figure moves the reference point left.

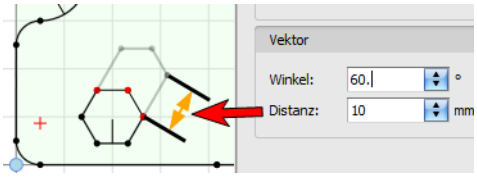


Fig. 7-60: Distance of reference point shift

Tube Editor

Sector

This option can be used to move or insert contours, segment points or texts into the respective sectors of the tubular sections. The sectors are displayed on the left-hand side of the developed view of the tube (**A**).

Circular tubes are composed of four sectors. A rectangular or polygon tube contains the same number of sectors as it has faces.

☞ **Note:** The X and Y-dimensions in the fields for the absolute coordinates always refer to the selected sector.

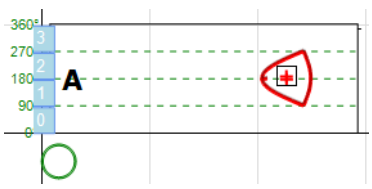


Fig. 7-61: Diagram showing the sectors for a circular tube

7.4.4.14 Scale



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function is used to scale the size of the contour. Scaling is performed using the **Scaling factor** or the preset **Drawing scale**.

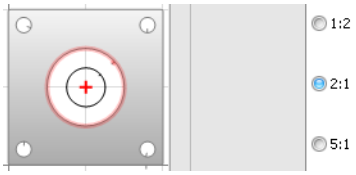


Fig. 7-62: Halving the size of a circular cut-out.

Scaling using a scaling factor.

1. Select the contour that you want to scale in size.
2. Select the **Scale** function.
3. Select the base point for the scaling (optional).
4. Specify the scaling factor for the contour in the **Scaling** area.
5. Click on the **Apply** button.



Note: The scaled contour is shown as a preview based on the selected scaling factor.

You have the following options for selecting the base point:

- Clicking on any position.
- Left-clicking on a point in the alignment array.
- Selecting a Snap function.

Scaling a contour to its original size

The preset scaling factors provide a simple way of scaling back to its original size a contour that was imported from a drawing in a certain scale.

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Scaling a contour using a scale

Assume that we have imported a contour with the scale 2:1. The contour is therefore twice as large as the original.

1. Select the **Scale** function.
2. Under **Scale of original drawing**, select the scale **2:1**. The contour is halved in size.
3. Click on the **Apply** button.



Note: Since the preset scaling factors relate to the original drawing, scaling is performed as the inverse of the selected factor.

Reference

When scaling using a reference dimension, a new dimension can be assigned to an element of a contour. All elements of the contour are then scaled in proportion to this dimension.

The **Measure** option can be used to measure the reference dimension. Use the Snap functions to do this.

Scaling a contour using a reference dimension

1. Select the contour that you want to scale in size.
2. Select the **Scale** function.
3. Select the **Measure** option.
4. Select the base point for the scaling (optional).
5. Click on the first point of the reference dimension to be measured. Use the Snap functions to do this.
6. Click on the second point of the reference dimension to be measured. The **Reference length** field displays the measured length.
7. Enter the new length in the **New length** field.
8. Click on the **Apply** button.

- **Scaling factor**

  You can enter any scaling factor you wish in this field.

  - Values > 1 increase the size of the contour.
  - Values <1 decrease the size of the contour.

- **Scale of original drawing**

  You can use these scaling factors to scale the individual, imported contours back to their original size.

7.4.4.15 Offset



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function can be used to enlarge or reduce the size of a contour by a specified amount. The corner transitions for the contour can also be changed at the same time. The **Critical angle** option defines the criterion for the type of corner transition.

- **Mode**

  These options determine how the offset is applied on the highlighted contours.

  - **Relative**

    The offset is added to or subtracted from the existing dimension of the highlighted contours.

  - **Absolute**

    The offset is applied equally to all highlighted contours. This allows you to set contours with different diameters to the same diameter.

- **Offset**

  This value defines the amount by which the contour is enlarged or reduced. A positive value enlarges the contour and a negative value reduces the size of the contour. Depending on which option is selected in the **Mode** section, the value of **absolute** or **relative** is applied to all highlighted contours.

- **Copy**

  This option inserts the mirrored contour as a copy of the original contour.

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- **Corner transition**

  These options determine as of which angle a corner is executed as a rounding or as a chamfer.

  - **Rounding**

    The corner transition is rounded off. The rounding-off radius equals the value set in the **Distance** field.

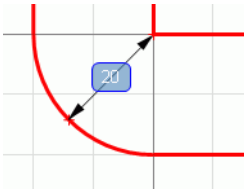
    

    Fig. 7-63: Corner transition rounded off with a radius of 20 mm

  - Chamfer

    The corner transition is chamfered. The distance of the chamfer from the original corner equals the value set in the **Distance** field.

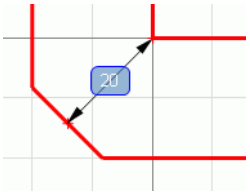
    

    Fig. 7-64: Corner transition chamfered over a distance of 20 mm

- **Critical angle**

  This option determines the angle up to which the corner transition is applied to the contour.

7.4.4.16 Move



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function moves the selected contour by a specified distance from the original position. Several contours can also be selected and moved at once.

Move contour

1. Select the contour that you wish to move.
2. Select the **Move** function from the **Edit** function selection (optionally **Ctrl+M**).
3. Select the base point of the contour (optional).
4. Specify the move point for the contour.
5. Click on the **Apply** button.



Note: If you move the contour while holding down the left mouse button, the contour will be permanently displayed. If Snap functions are enabled, the base point automatically snaps to all the snap points on the contours over which the base point is moved.



Tip: The contour can also be moved using the Snap functions.

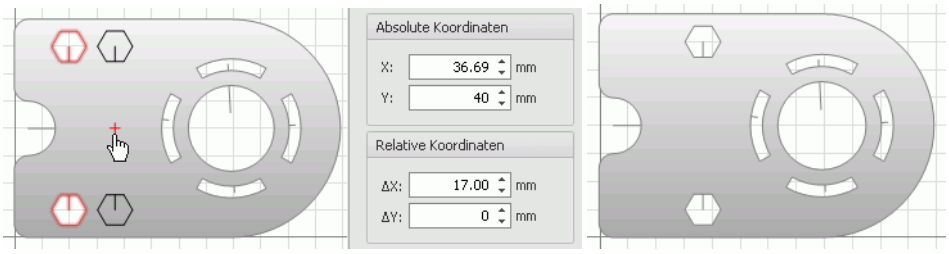


Fig. 7-65: Moving contours by 17mm by entering a relative X-coordinate

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Base point

The base point of the selected element is displayed as absolute X and Y-coordinates in the **Base point** area.

You have the following options for selecting the base point:

- Clicking on any position.
- Left-clicking on a point in the alignment array.
- Selecting a Snap function.

Absolute/relative coordinates

You have the following options for selecting the move point:

- Clicking on any position.
- Entering a new X and Y value in the **Absolute coordinates** or **Relative movement** areas.
- Selecting a sector in the **Absolute coordinates** area (Tube Editor).
- Selecting a tube angle for round tubes (Tube Editor)
- Selecting a Snap function.



Tip: Information about the next step is displayed in the status bar at the lower left.

Vector

This option is used to move contours or bend lines using the vectors **Angle** and **Distance**.

Press **Parallel** to reset the angle to **0**.



Tip: This function also allows a parallel shift to a reference segment.

Move contour via vectors

1. Select the contour that you wish to move.
2. Select the **Move** function from the **Edit** function selection.
3. In the **Distance** box enter the distance you want to move the selected contour.
4. In the Angle box enter the reference angle you want to move the contour. Select an angle of **0** for a horizontal shift.
5. Click on the **Apply** button.

Parallel shift of angled contour

1. Select the contour that you wish to move.
2. Select the **Move** function from the **Edit** function selection.
3. Press **Parallel**. Enter the angle of the inclined contour in the **Angle** box.
4. In the **Distance** box enter the distance you want to move the selected contours.
5. Click on the **Apply** button.

Angle

Defines the angle by which to move the selected contour.

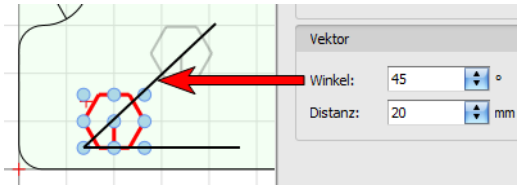


Fig. 7-66: Contour shift in an angle

Distance

Defines the distance of the shift based on the selected contour. A negative figure moves the contour left.

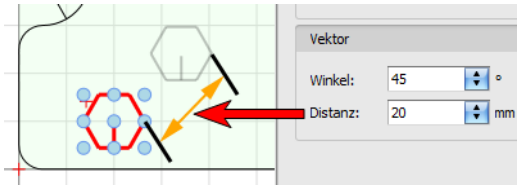


Fig. 7-67: Distance of contour shift

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Tube Editor

Sector

This option can be used to move or insert contours, segment points or texts into the respective sectors of the tubular sections. The sectors are displayed on the left-hand side of the developed view of the tube (**A**).

Circular tubes are composed of four sectors. A rectangular or polygon tube contains the same number of sectors as it has faces.

☞ **Note:** The X and Y-dimensions in the fields for the absolute coordinates always refer to the selected sector.

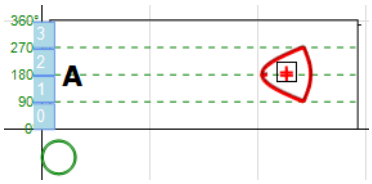


Fig. 7-68: Diagram showing the sectors for a circular tube

Tube angle

This option positions the geometry at a specific angle on the round tube.

☞ **Note:** This option can only be used for round tubes.

Moving a contour over tube angles

1. Select the contour that you wish to move.
2. Select the **Move** function in **Contour** > **Edit**.
3. Enter the **Tube angle** for moving in **Absolute coordinates**.
4. Click on the **Apply** button. The contour is moved by the specified angle on the tube.

7.4.4.17 Mirror



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function mirrors the selected contour at a specified distance from the original position. You can also select multiple contours to mirror in one go.

Procedure…

1. Select the contour to mirror (**A**).
2. Select the **Edit** -> **Mirror** function in the **Contour** group.
3. Select the type of alignment for the axis of mirroring.
4. Select the position of the axis of mirroring (**B**) using the snap functions on the contour (**C**). Optionally, you can also define the position of the axis of mirroring by entering the X/Y position and the angle in the dialog.
5. Click on the **Apply** button. The mirrored contours are applied to the part (**D**).

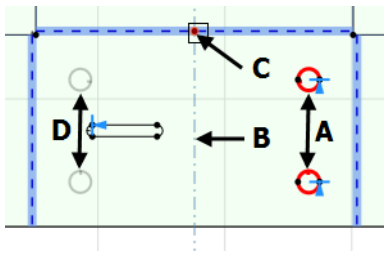


Fig. 7-69: Horizontal mirroring of circular contours

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Mirror

- **Type**

  Defines the direction in which to mirror a contour.

  - **Horizontal**

    Mirrors the contour horizontally around a vertical axis of mirroring.

  - **Vertical**

    Mirrors the contour vertically around a horizontal axis of mirroring.

  - **User-defined**

    Mirrors the contour around a user-defined axis of mirroring.

- **Copy**

  If this option is enabled, a copy of the contour is mirrored around the axis of mirroring. The highlighted contour is retained.

- **X/Y**

  Defines the position of the axis of mirroring based on the origin of the part.

- **Angle**

  Defines the angle of the axis of mirroring.

  - **Measure**

    Use this option to define the angle of the axis of mirroring by specifying two points on the contour. Any of the snap functions can be used to specify the contour points.

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7.4.4.18 Rotate



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function rotates the selected contour about the center of rotation through a specified angle. The angle of rotation can be set using the mouse or by entering an angle in the **Angle of rotation** field. When using the mouse to set the angle of rotation, the angle is changed in steps of **15°**.

Note: The angle of rotation is shown as a blue semi-transparent area.

Rotating a contour using an angle of rotation

1. Select the contour that you wish to rotate.
2. Select the **Rotate** function.
3. Use the keyboard to enter the X-coordinate. As soon as you touch a number key, the dialog box opens and the number is entered in the **X** field.
4. Press the **Tab** key to enter the Y-coordinate.
5. Press **Enter** to confirm the entry.
6. In the **Angle of rotation** field, enter the angle through which you want to rotate the contour. Alternatively, you can rotate the contour in steps by left-clicking and dragging with the left mouse button pressed.
7. Click on the **Apply** button.

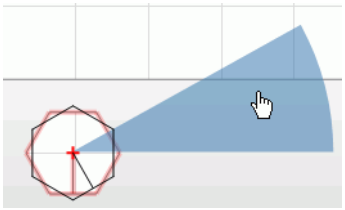


Fig. 7-70: Rotating a contour using the mouse

You have the following options for selecting the center of rotation:

- Entering the X and Y-coordinates.
- Clicking on any position.
- Left-clicking on a point in the alignment array.
- Selecting a Snap function.

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Reference

When rotating a contour using the **Reference** option, it is easy to assign a new angle to the element of a contour without needing to know the reference angle of the element.

The **Measure** option can be used to measure the reference angle. Use the Snap functions to do this.

Rotating a contour using the Reference option

1. Select the contour that you wish to rotate.
2. Select the **Rotate** function.
3. Set the center of rotation about which the contour will be rotated.
4. Select the **Measure** option.
5. Click on the start point of the contour element to be measured.
6. Click on the end point of the contour element to be measured. The measured reference angle is displayed in the **Reference angle** field.
7. Enter the new angle in the **New angle** field.
8. Click on the **Apply** button.

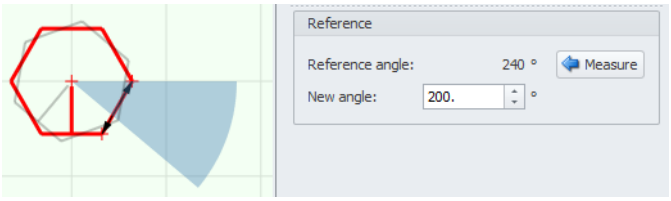


Fig. 7-71: Using the reference angle to assign a new angle of 30° to the contour

7.4.4.19 Row



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

Using this function, you can duplicate contours as rows (rectangular) or around a center point (polar). Depending on the selected mode, the appropriate parameters are displayed.

Rectangular parameters

Contours are duplicated in the X- and Y-direction.

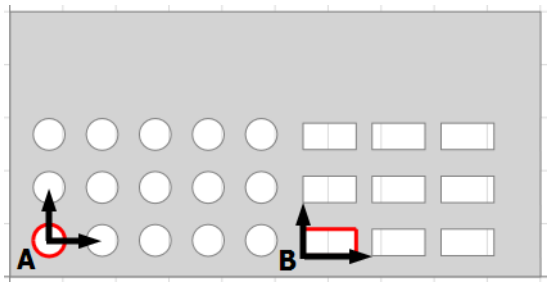
Number X/Y

Enter the number of contours in the X and Y direction in these fields.

Distance X/Y

Enter the distance of contours in the X and Y direction in these fields. The minimum distance must be at least the X/Y dimension of the bounding rectangle, otherwise the contours will overlap.

Fig. 7-72: Distance for circular and rectangular contours



Key:

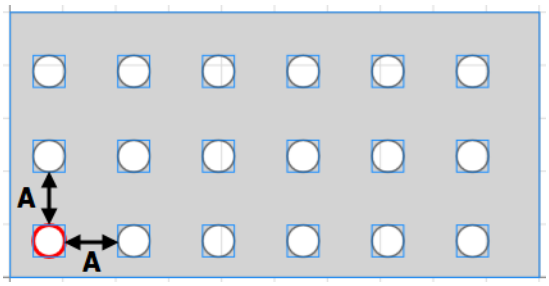
A	X/Y distance for circular contours from midpoint to midpoint.
B	X/Y distance for rectangular contours from origin to origin of the contour.

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Bounding rectangle

This option calculates the X/Y distance based on the bounding rectangle of the contours. The distance is the dimension between the two rectangles.

Fig. 7-73: Distance for contours with bounding rectangle.



Key:

A	X/Y distance between bounding rectangle

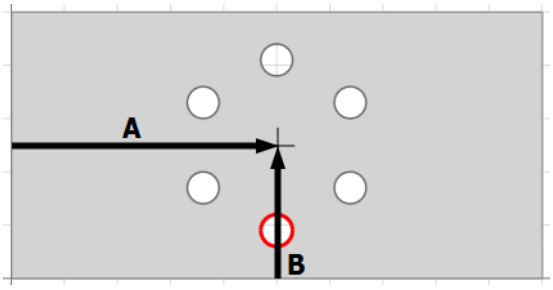
Polar parameters

The contours are duplicated in a circle around a center.

Center X/Y

In this field enter the X/Y distance of the center of rotation based on the origin around which the contours are arranged. This center us marked with a cross.

Fig. 7-74: Center for polar arrangement of contours.



Key:

A	X distance from origin to center.
B	Y distance from origin to center.

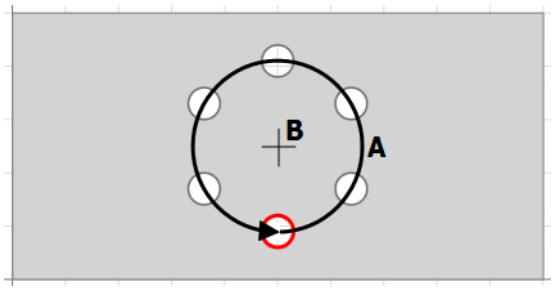
Quantity

Enter the number of contour duplications in this field.

Angle

In this field enter the angle of rotation around which the contours are arranged. A value of **360°** arranged the contours in a full circle.

Fig. 7-75: Arrangement of circular contours with an angle of rotation of 360°



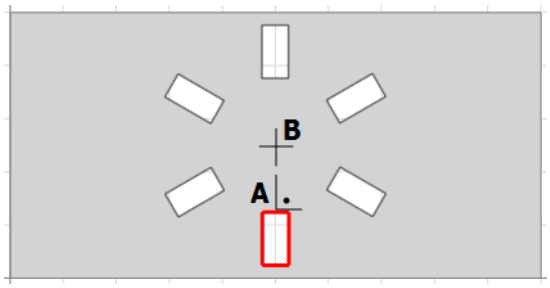
Key:

A	Angle of rotation
B	Center of rotation

Rotate geometry

This option rotates the contours facing the center. The side that is to face towards the center must be at right angles (**A**) to the center of rotation (**B**).

Fig. 7-76: Rectangle aligned around the center of rotation



Key:

A	Right angle to center of rotation
B	Center of rotation

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7.4.4.20 Clean



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour > Edit**

This function removes superfluous reference points from the selected contour without changing the contour geometry.



Tip: This function can also remove segments with a length of 0mm.

7.4.4.21 Smooth



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function can be used to reduce automatically the number of straight and arc-shaped elements of a contour. Smoothing is applied to all selected contours.

The **Preview** button displays the result of the current setting on the contour. Press the **Apply** button to adopt the settings.



Note: Before running the **Smooth** function, select all the contours that you wish to smooth.



Tip: Use this function for automatic post-editing of scanned and irregular contours containing numerous contour elements. Try out the various settings of the two options before adopting the smoothed contours.

Maximum values

- **Linear deviation**

  This option reduces connected geometry elements that have a linear deviation less than or equal to this value. This reduction is applied to all straight geometry elements.

- **Chordal deviation**

  This option reduces connected geometry elements that have a chordal deviation less than or equal to this value. This reduction is applied to circles, arcs or arc elements.

7.4.4.22 Insert reference point



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function can be used to set additional reference points on a geometry. Reference points are indicated by a black dot on the geometry.

To insert a reference point, left-click on the position on the contour in which you want to set the reference point.

Note: You can set various machining attributes for reference points in the **Cutting technology** menu *(See: ›Reference point‹ Page 696)*.

Settings

Offset

This option can be used to set a reference point at a certain distance to a point on the contour. A negative figure sets the reference point before the selected point on the contour.

Setting a reference point with offset

1. Enter the distance to the appropriate point on the contour in the **Offset** box.
2. Select the point on the contour. The reference point is set at the specified distance to this point.

Tip: You can use this option for water jet cutting, for example with process macro **3** to use a different macro when approaching a lead-out (**1**). Using a slower cutting speed reduces water jet overrun and thus ensures that the cut goes right through at the end of the contour.

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7.4.4.23 Remove reference point



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Contour**

This function can be used to remove manually set reference points again from a contour. Left-click on the manually inserted reference point to remove it.



Note: You can only remove reference points that were added using the **Insert reference point** function.

7.4.5 Group: Bending

Where do I find the group?

Select: **Modules > Part Editor > Geometry**

This group contains all the functions that can be used to edit or correct bends.

7.4.5.1 Edit



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Bend**

This function opens a dropdown menu from where you can select various editing functions for bend lines.

The following edit functions are available:

- Insert
- Edit
- Correct
- Automatic
- Connect
- Resolve

7.4.5.2 Insert bend line



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Bend > Bend line**

This function can be used to insert bend lines using the snap functions either using the mouse or by entering coordinates.

In the **Bend** area, you can assign the appropriate bend type to the bend line.



Note: Use the **Edit** function to make any subsequent alterations to an existing bend line.

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Bend

Normal bending

This option sets the selected bend line to be a normal bend. Normal bending is the standard method for creating a bend.

- **Bend angle**

  Specifies the bend angle of the bend based on the outside angle (**α**). The bend angle can be positive or negative.

- **Bend radius**

  Specifies the internal bend radius (**r**) for the bend line. The bend radius is set by the choice of bending tool.

- **K-factor**

  The K-factor determines the position of the imaginary neutral fiber in the sheet. Values between 0 and 1 can be used.

Position of the neutral fiber in the sheet	Value
Inside	0
Center	0.5
Outside	1

☞ **Note:** The K-factor has a direct influence on bend allowance. A **smaller** K-factor increases bend allowance. A **larger** K-factor decreases bend allowance.

- **Bend allowance**

  The user can change the value for the bend allowance in this field.

  DIN button:

  This button enters the bend allowance based on DIN6935 in the **Bend allowance** field. The bend allowance depends on the bend radius.

Hemming

This option sets the selected bend line to be a hemming bend. Hemming bends the two sides in two bending steps with a bend angle of 180°. Hemming is performed in accordance with the DIN standard. After bending, a small gap will remain along the bend line.

Bending steps for a hemming bend:

1. Pre-bending

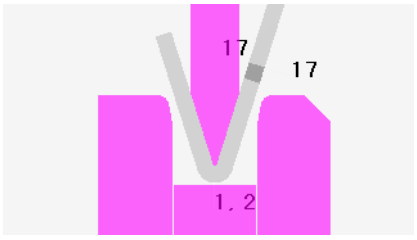


Fig. 7-77: Pre-bending process

2. Hemming

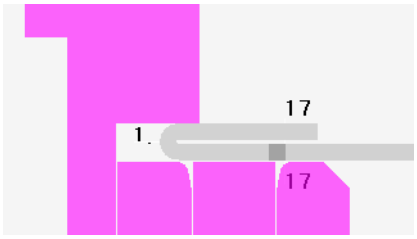


Fig. 7-78: Hemming process

- **Bend angle**

  Specifies the pre-bending angle (**α**) for the first bend step. The default pre-bending angle is 145°.

- **Bend radius**

  Specifies the internal bend radius (**r**) for the bend. A value of **0** means that the sides are pressed flat together.

☞ **Note: K-factor** and **bend allowance** can be specified as with normal bends.

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Start / End

Enter the start and end point of the bend line in these fields by entering the X and Y coordinates. The coordinates refer to the origin of the part.

Press the **Apply** button to assign the bend line to the part based on the coordinates entered.

☞ **Note:** If the end points of the bend line are not exactly on a contour, apply the **Correct** function to the bend line.

Parallel

These options are used to insert bend lines parallel to a contour or other bend lines.

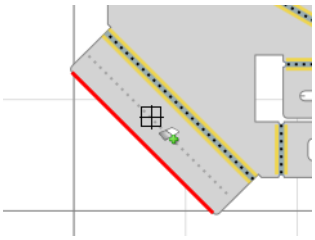


Fig. 7-79: Parallel bend line to contour segment

Procedure…

1. Press the **Select** button in the **Parallel** section.
2. Select the contour segment that you wish to move parallel to the bend line.
3. Enter the distance to the contour segment in the **Offset** box.
4. 1Select the **Allow for bend allowance** option to allow for bend allowance.
5. Click Apply to set the bend line.

Options

Reference

Displays the reference angle to the selected contour segment.

Select

this button selects the contour segment for the new bend line.

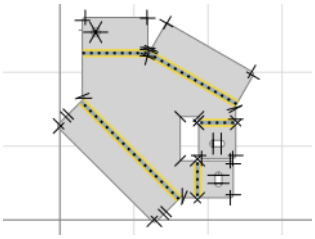


Fig. 7-80: Contour segments for inserting parallel bend lines on a part

Offset

The figure in this box defines the distance of the bend line to the selected contour segment (**A**).

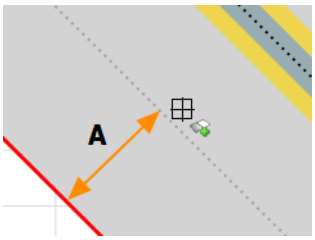


Fig. 7-81: Distance from selected contour segment

Allow for bend allowance

When inserting a parallel bend line, the bend allowance for flange length can be factored in. The distance specified in **Offset** is reduced by half the bend allowance (**A**).

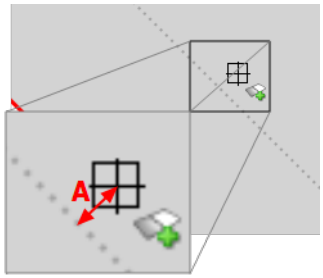


Fig. 7-82: Half of bend allowance of bend line relative to mouse cursor

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7.4.5.3 Edit bend line



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Bend > Bend line**

This function can be used to change the properties of bend lines. During the DXF import of a bending part, bending properties can be assigned to each line that is intended to be used as a bend line.

In the **Bend** area, you can assign the relevant bend type to the selected bend line.



Note: The **Apply** button will assign the specified bending properties to the selected bend line.

Bend types

Normal bending

This option sets the selected bend line to be a normal bend. Normal bending is the standard method for creating a bend.

- **Bend angle**

  Specifies the bend angle of the bend based on the outside angle (**α**). The bend angle can be positive or negative.

- **Bend radius**

  Specifies the internal bend radius (**r**) for the bend line. The bend radius is set by the choice of bending tool.

- **K-factor**

  The K-factor determines the position of the imaginary neutral fiber in the sheet. Values between 0 and 1 can be used.

Position of the neutral fiber in the sheet	Value
Inside	0
Center	0.5
Outside	1

☞ **Note:** The K-factor has a direct influence on bend allowance. A **smaller** K-factor increases bend allowance. A **larger** K-factor decreases bend allowance.

- **Bend allowance**

  The user can change the value for the bend allowance in this field.

  DIN button:

  This button enters the bend allowance based on DIN6935 in the **Bend allowance** field. The bend allowance depends on the bend radius.

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Hemming

This option sets the selected bend line to be a hemming bend. Hemming bends the two sides in two bending steps with a bend angle of 180°. Hemming is performed in accordance with the DIN standard. After bending, a small gap will remain along the bend line.

Bending steps for a hemming bend:

1. Pre-bending

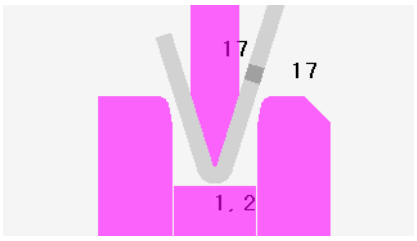


Fig. 7-83: Pre-bending process

2. Hemming

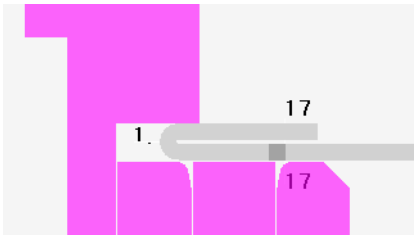


Fig. 7-84: Hemming process

- **Bend angle**

  Specifies the pre-bending angle (**α**) for the first bend step. The default pre-bending angle is 145°.

- **Bend radius**

  Specifies the internal bend radius (**r**) for the bend. A value of **0** means that the sides are pressed flat together.

☞ **Note: K-factor** and **bend allowance** can be specified as with normal bends.

7.4.5.4 Correct bend line



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Bend > Bend line**

This function is used to correct the length of those bend lines whose end points do not lie exactly on the outer contour, either lengthening or shortening the length according to the situation.

7.4.5.5 Invert bend line



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Bend > Bend line**

This function inverts all selected bend lines in their direction of bending.

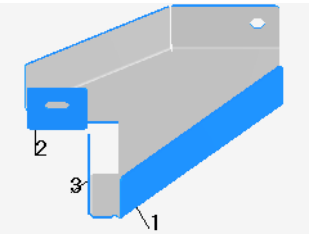
Inverting bend lines

1. Select all bend lines to invert.
2. Select the **Invert** function.

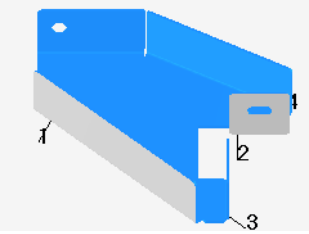


Tip: Use this function to create a mirrored bending part.

Normal bending parts



Mirrored bending part with inverted bend lines



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7.4.5.6 Bend marking



How do I open this dialog box?

Select: **Modules > Part Editor > Geometry > Bend > Bend line**

This function is used to place a circle mark on the leg of a bend. The mark is engraved on the part and displayed as an extrusion in the 3D display in the bending program.

- **Diameter**
  defines the diameter of the bend mark.
- **Extrusion**
  defines the depth of extrusion of the bend mark.



Note: A negative figure sets the extrusion on the opposite side of the part.



Tip: Use this function to distinguish similar sides in a bending process more easily.

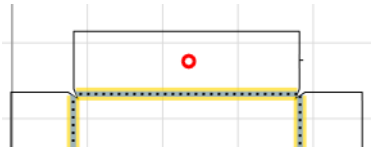


Fig. 7-85: Bend mark on a side in 2D view

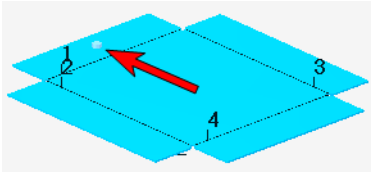


Fig. 7-86: Bend mark on a side in 3D view

7.4.5.7 Relief cut



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Bend > Bend line**

Use this function to set a relief cut on or next to a bend line. Relief cuts are required to avoid deformation of holes in the entry zones of the bend.

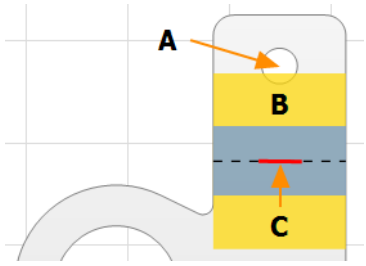


Fig. 7-87: Relief cut on a bend line

A	Hole cut-out
B	Entry zone (yellow)
C	Relief cut

Setting a relief cut

1. Select the **Relief cut** function.
2. Holding the **shift key** select the bend line and the contour for which you want to create a relief cut.
3. Define the width of the relief cut. A value of **0** corresponds to the cutting gap.
4. Click **Apply** to assign the relief cut to the part.

Note: If the part has already been assigned a cutting technology, it needs to be set again for the relief cut.

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Selection

- **Geometries in entry zone**
  Relief cuts are only generated for geometries in the entry zone.
- **All geometries**
  Relief cuts are generated for all geometries.

Length

- **Proportion in entry zone**
  A relief cut is only generated for the length of the geometry in the entry zone.
- **Overall geometry**
  A relief cut is generated for the entire length of the geometry.

Safety

Depending on the selection in the **Length** section, this value is either added to or subtracted from the relief cut.

A value of **0** corresponds to the length of the relief cut as indicated in the **Length** section.

Width

This value defines the width of the relief cut.

A value of **0** corresponds to the width of the cutting gap.

Offset

This value moves the relief cut away from the bend line towards the contour.

A value of **0** corresponds to the position of the relief cut on the bend line.

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7.4.6 Group: Corner relief

Where do I find the group?

Select: **Modules > Part Editor > Geometry**

This group contains all the functions that can be used to edit or delete the corner reliefs of a bending edge.

7.4.6.1 Manual corner relief



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Corner relief**

Use this function to place a corner relief at the end of bend lines. In addition, the shape and size of existing corner reliefs can also be changed.

Creating corner reliefs for bend lines

1. Select the **Manual** corner relief function.
2. You can specify the corner relief options in the **Manual corner relief** dialog box.
3. Use the mouse to move the selection box to the end of the bend line or to the intersection point of two bend lines. The corner relief is displayed as a gray contour in a preview.
4. Confirm the selection with the left mouse button.

Tool

Size

This option specifies the size of the selection box with which the corner relief will be selected.

Corner relief

Shape

Use this option to specify the shape of the corner relief. The default shape is **V-shaped**.

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V-shaped	Rectangular	Circular
		

Safety

This option lengthens (**A**) or shortens (**B**) the intrusion of a corner relief into the bend zone. The value **0** corresponds to a corner relief of the same size as the bend zone (**C**).

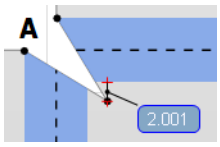


Fig. 7-88: Lengthening of the corner relief by 2mm

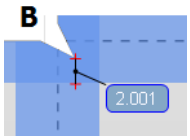


Fig. 7-89: Shortening of the corner relief by 2mm

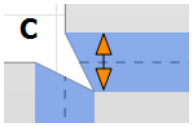


Fig. 7-90: Corner relief with a value of 0

Aperture

This option determines the size of the aperture of a corner relief based on the contour reference point. The size of the aperture is half the dimension given (**A**) in both directions relative to the contour reference point (**B**).

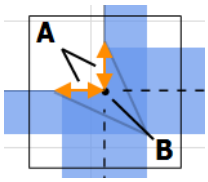


Fig. 7-91: Size of the aperture of a corner relief

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7.4.6.2 Remove corner relief



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Corner relief**

This function removes the selected corner relief from the bend line. To remove the corner reliefs, proceed as follows:

1. Select the **Remove** corner relief function.
2. Use the mouse to move the selection box over the corner relief.
3. Confirm the selection with the left mouse button.

7.4.7 Group: Cutouts

Where do I find the group?

Select: **Modules > Part Editor > Geometry**

This group contains functions that can be used to set cutouts on sharp corners. Cutouts can be used for parts that should be removed from the sheet by a sorting system. The point of these cutouts is to prevent the part from tilting, as a result of sharp corners, when being lifted out.

7.4.7.1 Automatic



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Cutouts**

This function is used to automatically set cutouts on all inner corners of the outer contour as per the configured options. Cutouts are displayed in a preview on the part. Press the **Apply** button to set the cutouts on the part.

Cutout

Max width

This option defines the maximum width starting from which a cutout will be set (**A**).

Min depth

This option defines the minimum depth starting from which a cutout will be set (**B**).

Distance

This option defines the distance between the cutout and the contour (**C**). By default, the distance is equal to the kerf width.

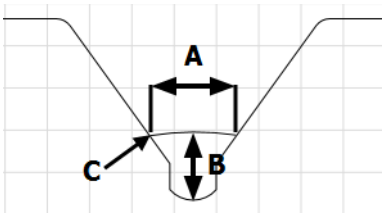


Fig. 7-92: Cutout on an outer contour

Geometry

Arc

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This option performs the cutout as an arc.

Line

This option performs the cutout as a line.

Adjust to contour vertices

An attempt is made to match the end of the cutout (**A**) to the vertices of the contour (**B**). All previous parameters are taken into account.

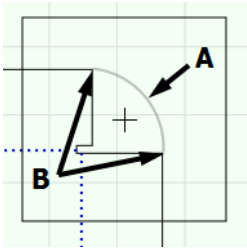


Fig. 7-93: Cut-out between two contour vertices

7.4.7.2 Insert



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Cutouts**

This function can be used to set cutouts on all inner corners of the outer contour, as per the configured options, with a selection box.

Tool

Size

This option defines the size of the selection box used to select the corner for the cutout (**D**).

Cutout

Max width

This option defines the maximum width starting from which a cutout will be set (**A**).

Min depth

This option defines the minimum depth starting from which a cutout will be set (**B**).

Distance

This option defines the distance between the cutout and the contour (**C**). By default, the distance is equal to the kerf width.

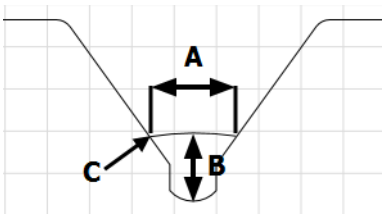


Fig. 7-94: Cutout on an outer contour

Geometry

Arc

This option performs the cutout as an arc.

Line

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This option performs the cutout as a line.

Adjust to contour vertices

An attempt is made to match the end of the cutout (**A**) to the vertices of the contour (**B**). All previous parameters are taken into account.

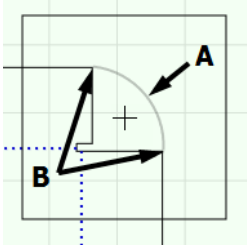


Fig. 7-95: Cut-out between two contour vertices

7.4.7.3 Manual



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Cutouts**

This function can be used to manually set cutouts on all corners of the outer contour. The size of the cutout is defined by the position of the selection box at the contour tip. The cutout is set as a segment of the incircle of the selection box.



Tip: To define the size of the cutout, drag the selection box along the contour tip.

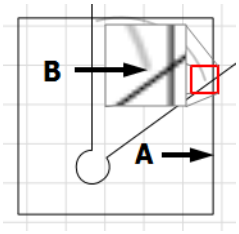


Fig. 7-96: Manually set cutout on an outer contour

Tool

Size

This option defines the size of the selection box used to select the corner for the cutout (**D**).

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Cutout

Distance

This option defines the distance between the cutout and the contour (**B**). By default, the distance is equal to the kerf width.

Outer contour only

If this option is enabled, it will only be possible to set cutouts on outer contours.

Snap center of circle

If this option is enabled, the cutout (**C**) is snapped to the nearest vertex of the outer contour (**D**). The midpoint of the circle (**E**) of a cutout is placed on the bisector with the radius of the incircle of the selection box (**F**) from the vertex.

Insert separation cut

This option inserts a separation cut (**A**) in the center of the cutout.

The option reduces the risk of the cutout damaging the fork of the twin-fork system or the cutting grate during the automatic removal process.

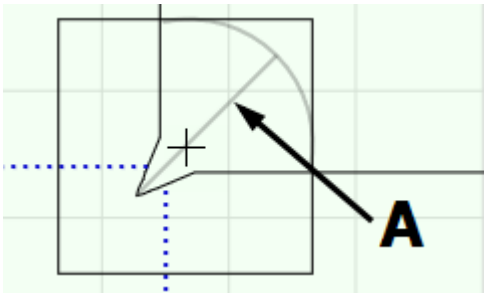


Fig. 7-97: Separation cut in a cutout

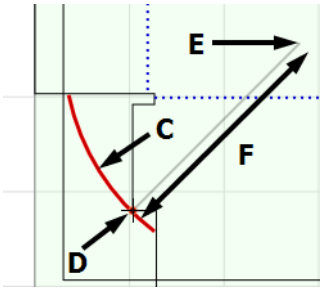


Fig. 7-98: Cutout on an outer contour

7.4.8 Group: Text

Where do I find the group?

Select: **Modules > Part Editor > Geometry**

This group contains all the functions that can be used to set any texts on a part. Using text information to identify parts or batch sizes in an order can help quality assurance in the production process.

Note: When creating engraved text, make sure that the text will still be visible after any finishing processes such as painting or galvanizing.

7.4.8.1 Insert



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Text**

Use this function to place a custom text on the part using the mouse or by entering coordinates. You can use texts and data fields of various categories and a combination of both.

If the inserted text is cut, the post-processor automatically sets straight 5 mm lead-ins on the contours in Part Nester when exporting.

The mode for operating the text is configured in the cutting technology settings *(See: › Cutting properties, text‹ Page 666)*.

Note: If **mirroring authorization** is set on a part with text, the ***Mirrored text*** warning is displayed.

Tip: You can position the text quickly and easily by left-clicking on a location inside the contour of the part. If the text has not yet been positioned, it is displayed centered in the view window as soon as the mouse leaves the view window.

Tip: In a DXF file you can define text information for a normal text element. The import function automatically converts the text into the appropriate text *(See: ›Define layer ID for labels in DXF file‹ Page 1381)*.

Placing text on a part

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1. Select the **Insert text** function.
2. Enter the desired text or the data field in the **Text** field.
3. Format the text then arrange it on the part using the relevant options.
4. Click the **Insert** button to apply the text to the part, then save the part.

Insert data field

1. To do so, click on the  icon at the right end of the input field.
2. Select the data field in the corresponding category.

> **Note:** The particular contents of the data field is not set on the part until the cutting plan is generated.

Restrict data field display

You can restrict the data field content display using an additional code entry. There are two ways of restricting this:

Version 1:

- Display a string from a certain character, for example the name of the part.
- Name = **samplename2**
- Text field = $(Part.Name**[7]**)
- String displayed = **name2**

Version 2:

- Display a string from one character to another, for example the name of the part.
- Name = **samplename2**
- Text field = $(Part.Name**[7:10]**)
- String displayed = **name**

> **Note:** Texts are only generated from data fields when the data fields concerned actually contain text.

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What each field does:

- **Part:** All available data fields for the un-nested part are displayed.
- **Nesting part:** All available data fields for the nested part are displayed.
- **Sheet:** All available data fields of one sheet are displayed.
- **Nesting part reference:** The **Serial-number information** data field is displayed. This data field can be used to assign consecutive numbers to a series of parts. The serial-number information is assigned in the **Cutting technology** menu of the Part Nester module *(See: › Group: Serial-number information‹ Page 1074)*.

Formatting

In this area you can format the text entered in the **Text** field and also align the text on the part.

What each option does:

- **Text**

  In this field you can enter or re-edit the text. The added data fields can also be entered in this field. These data fields are defined as code and therefore must not be changed.

- **Data field**

  This option inserts predefined data fields in the text input area.

> ☞ **Note:** You can place text and data fields together in any combination. Any change to a text is displayed in the workspace as a preview.

- **Font**

  This option is used to define the text font. All fonts installed in the system are displayed.

> ☞ **Note:** For fonts that are to be cut out, only fonts that do not have inside contours are suitable. There are types of special laser fonts for these fonts.

- **Line text type**

  This option only displays fonts that consist only of a line. These fonts minimize inscription time during engraving.

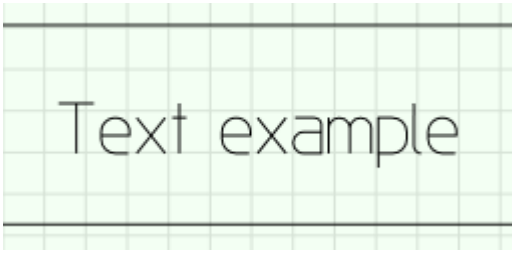


Fig. 7-99: Text with Bystronic line font

- **Size**

  This option changes the font size of the entered text.

- **Sector** (Tube Editor)

  This option can be used to move or insert contours, segment points or texts into the respective sectors of the tubular sections. The sectors are displayed on the left-hand side of the developed view of the tube (**A**).

  Circular tubes are composed of four sectors. A rectangular or polygon tube contains the same number of sectors as it has faces.

> **Note:** The X and Y-dimensions in the fields for the absolute coordinates always refer to the selected sector.

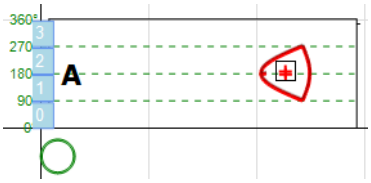


Fig. 7-100: Diagram showing the sectors for a circular tube

- **X/Y**

  These options let you specify the position of the text on the part using X and Y-coordinates.

- **Tube angle** (Tube Editor)

  This option positions the geometry at a specific angle on the round tube.

- **Rotation**

  This option rotates the text through a specified angle.

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- **Mirror**

  This option mirrors the text around the alignment point (red cross).

- **Bold**

  This option is used to increase the text contour size slightly.

- **Italic**

  This option is used to italicize the text contour.

- **Position at center of gravity**

  This option positions the text at the part's center of gravity.

- **Alignment**

  This option is used to define the position in which you wish to arrange the text. Click on the particular point in the array that corresponds to the text alignment you require.



Fig. 7-101: Data field arranged in lower left corner

Tip: The Snap functions can also be used for aligning the text *(See: ›Snap‹ Page 64)*.

- **Label text**

  This option prints the text on the material with an inkjet printer. No cutting technology is assigned to the text. This option can only be used if an inkjet printer has been configured for the machine.

Note: Use this option if you need to remove the text from the material.

7.4.8.2 Edit

Where do I find the function?

Select: **Modules > Part Editor > Geometry > Text**

This function can be used to edit an existing text. This function cannot be used until an existing text has been selected. Otherwise the function works in the same way as the **Insert text** function.

7.4.8.3 Convert

Where do I find the function?

Select: **Modules > Part Editor > Geometry > Text**

This function converts any text into contours.

Use suitable fonts (e.g. Laser Old Style) to cut out text. With these fonts, the inside contours of letters remain connected to the outer contour so that they do not fall out of the letters.



Fig. 7-102: Laser font of letters with inside contours

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7.4.9 Group: Label

Where do I find the group?

Select: **Modules > Part Editor > Geometry**

This group contains all the functions that can be used to set any labels on a part.

7.4.9.1 Insert



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Label**

Use this function to place labels identifying parts on the part using the mouse or by entering coordinates.

Labels are printed on the appropriate part in the cutting plan on the machine using the Part Marker module. They can be printed on mild steel, all non-ferrous metals, aluminum, and with some restrictions on foil-covered sheet metal.

On each label you can use texts and data fields from various categories and a combination of both. The size and contents of the labels are displayed as gray areas in the workspace.



Fig. 7-103: Labels with two text fields and a Data Matrix field.

Attention! Labels are lost when you reimport a BVC cutting plan!

Note: Labels are only displayed in the Part and Geometry menus. Printing can go up to **10 mm** from the edge of the sheet.

Tip: In a DXF file you can define text information for labels, part identification, or normal text. The import function automatically converts the content of the layer ID into the appropriate element *(See: ›Define layer ID for labels in DXF file‹ Page 1381)*.

Template

In this section you can save a new label as a template or load a label template.

- **Template**

  A template for labels can be selected in this field.

- **Save**

  Saves changes to the current label.

- **Save as...**

  Saves the label as a template under a user-defined name.

Elements

You can configure the print elements for the labels in this section. You can use two kinds of elements for labels.

ABC

This print element is used to print text. You can enter text in alphanumeric ASCII format.

- **Contents**

  You can enter alphanumeric characters in ASCII format in this field. If you select one or more data fields using the **Data fields** option, they are also entered in the field.

- **Data field**

  Use the **Insert** selection box to insert data fields into the label *(See: › Insert data field‹ Page 491)*.

> ☞ **Note:** The Part Marker section prints the current time and date of printing the respective fields on the part. The batch number is copied from the store system.

- **Character width**
  - **Fixed**: Each character takes up the same amount of pixels.
  - **Proportional**: The number of pixels is adapted for the each character width. An **I** takes up fewer pixels than a **W**.
- **Font**

  Defines the font for the text element.

> ☞ **Note:** To change the font size, select a different font.

- **Max. number of characters**

  This setting defines the length of the text element.

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> **Note:** The numbers of characters must be at least equal to the maximum number of characters expected in the **Contents** field. If the element uses more characters than are defined here, a message is displayed on export and the label element will not be exported.

- **Position X**

  X position of the text element in the label.

- **Position Y**

  Y position of the text element in the label.

Code

The print element can be used to define barcodes and 2D codes.

- **Type**
  - **Data Matrix**

    A 2D code according to the ECC200 standard (square).

  - **QR code**

    A 2D code.

  - **Barcode**

    A linear code (code 39).

- **Contents**

  You can enter characters in ASCII format in this field. If you select one or more data fields using the **Data fields** option, they are also entered in the field.

- **Data field**

  Use the **Insert** selection box to insert data fields into the label *(See: › Insert data field‹ Page 491)*.

- ☞ **Note:** In the Part Marker section you insert the current time and date of printing into the code. The batch number is copied from the store system.

- **Height**

  If the selected code type is a **Barcode**, you can enter the height of the barcode in this field.

- **Error correction**

  Error correction can only be defined for the **QR code** code type. The larger the selected error correction, the larger the QR code will be printed.

  - **L** = QR code can still be read up to 7% missing data points.
  - **M** = QR code can still be read up to 15% missing data points.
  - **Q** = QR code can still be read up to 25% missing data points.

- **Point size**

  Point size can only be defined for the **Data Matrix** and **Barcode** code type.

  - **Data Matrix**

    Defines the number of drops to print per code pixel. The number is the selected point size squared. A setting of **2** prints four drops per code pixel. This setting should be used by default.

  - **Barcode**

    Defines the number of drops to use for the thinnest line width.

- **Position X**

  X position of the code in the label.

- **Position Y**

  Y position of the code in the label.

- **Maximum number of characters**

  This setting defines the length of the code element.

☞ **Note:** The numbers of characters must be at least equal to the maximum number of characters expected in the **Contents** field. If the element uses more characters than are defined here, a message is displayed on export and the label element will not be exported.

- **Character set**

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Defines the character set to use for the code element. The more characters the character set contains, the larger the code element.

- **Numeric**
  Only numbers are allowed.
- **Alphanumeric**
  Letters, numbers and special characters may be used:

SP	!	"	#	$	%	&	'	(	)	*	+	,	-	.	/
0	1	2	3	4	5	6	7	8	9	:	;	<	=	>	?
@	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
P	Q	R	S	T	U	V	W	X	Y	Z	[	\	]	^	_
`	a	b	c	d	e	f	g	h	i	j	k	l	m	n	o
p	q	r	s	t	u	v	w	x	y	z	{	|	}	~	

- **Extended alphanumeric**
  The following additional special characters may be used:

NBSP	¡	¢	£	¤	¥	¦	§	¨	©	ª	«	¬	SHY	®	¯
°	±	²	³	´	µ	¶	·	¸	¹	º	»	¼	½	¾	¿
À	Á	Â	Ã	Ä	Å	Æ	Ç	È	É	Ê	Ë	Ì	Í	Î	Ï
Ð	Ñ	Ò	Ó	Ô	Õ	Ö	×	Ø	Ù	Ú	Û	Ü	Ý	Þ	ß
à	á	â	ã	ä	å	æ	ç	è	é	ê	ë	ì	í	î	ï
ð	ñ	ò	ó	ô	õ	ö	÷	ø	ù	ú	û	ü	ý	þ	ÿ

Label

You can define the position of the label in this section.

- **X**
  X position of the label relative to the origin of the part.
- **Y**
  Y position of the label relative to the origin of the part.
- **Rotation**
  Angle of rotation of the label on the part.
- **Size**
  Displays the label size.
- **Apply**

Applies changes to an existing label.

- **Insert**

  Adds a new label to the part.

7.4.9.2 Edit



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Label**

This function can be used to edit an existing text. This function cannot be used until an existing label has been highlighted. Otherwise the function works in the same way as the **Insert label** function.

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7.4.10 Group: Parts ID

Where do I find the group?

Select: **Modules > Part Editor > Geometry**

This group contains functions that can be used to set a parts ID on the part as a data matrix code.

7.4.10.1 Insert



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Parts ID**

Use this function to place symbols or text as a barcode on the part using the mouse or by entering coordinates.

Text

The same options are available as in the **Insert text** function *(See: ›Insert‹ Page 489)* for entering text or code fields.



Tip: Use a maximum of **4** lowercase letters or **10** numbers for an optimum barcode size.

Formatting

The parts ID is displayed as a square matrix and is just a placeholder.

Size

Defines the size of the barcode in the workspace. The value does not affect the actual size of the barcode on the part. The barcode is only generated in Part Nester when the preprocessor exports the cutting plan.

X/Y

Defines the X and Y coordinates for the position of the barcode on the part.

Rotation

Rotates the barcode by the specified angle.

Barcode

Defines the type of barcode. By default, the barcode is created as a DataMatrix.

Type

Defines the type of the barcode on the part.

- **DataMatrix**: The parts ID is created as a DataMatrix code.

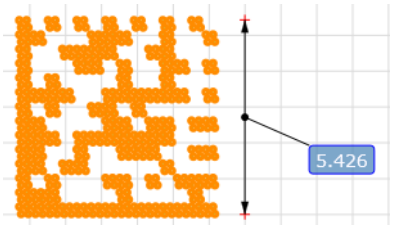


Fig. 7-104: Data matrix code of 8 characters

- **QR Code**: The parts ID is created as a QR code.

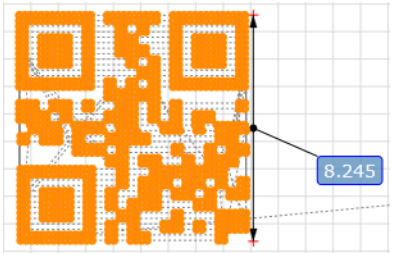


Fig. 7-105: QR code of 8 characters

Note: In order for the preprocessor to generate the barcode on export, the **Parts ID** option must be enabled in the preprocessor settings *(See: › Insert data field‹ Page 491)*. By default, the barcode is assigned the **Engraving** mode.

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7.4.10.2 Edit



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Parts ID**

This function can be used to edit an existing parts ID. This function cannot be used until an existing parts ID has been selected. Otherwise the function works in the same way as the **Add parts ID** function.

7.4.11 Group: Dimensioning

Where do I find the group?

Select: **Modules > Part Editor > Geometry**

This group contains all the functions for dimensioning contours on a part.

7.4.11.1 Automatic dimensioning



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Dimensioning**

Use this function to automatically insert dimensions on a part. Selecting this function calls up the **Edit dimensions** dialog to define the criteria for automatic dimensioning.

Format

This section is used to define the dimensioning format. The dimensioning format defines the arrangement of dimension lines on the part in accordance with ISO 129-1.

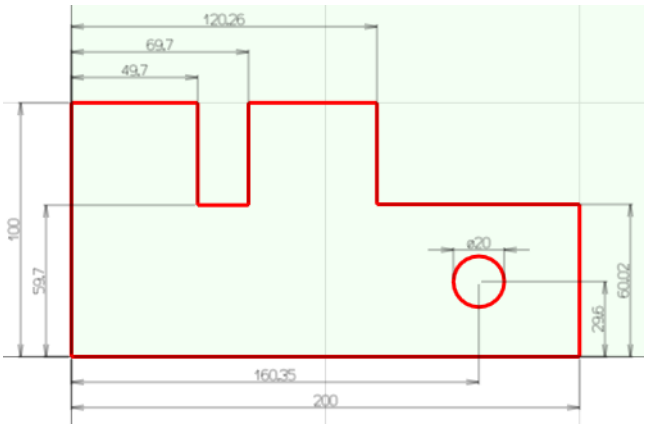
Type

Parallel

The dimension lines are drawn parallel in one, two or three orthogonal directions or concentrically.

This mode of dimensioning is preferable when dimensions in the same direction have the same starting point.

Fig. 7-106: Parallel dimensions on a part



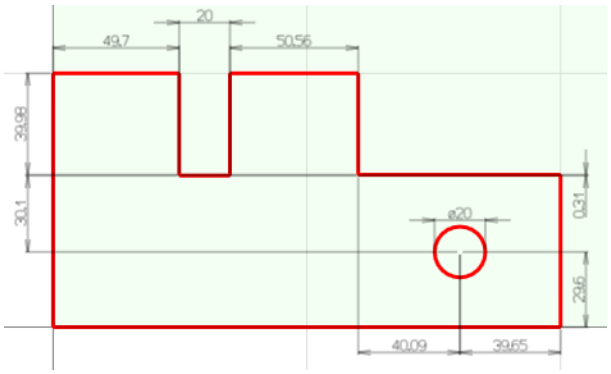
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Running

If space is limited or in case of special requirements, running dimensioning can be used.

Running dimensioning is simplified form of parallel dimensioning.

Fig. 7-107: Running dimensioning on a part

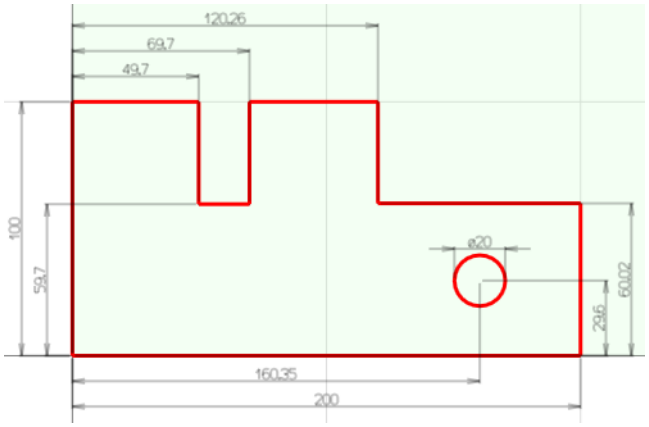


Alignment

Automatic

Dimension lines are aligned automatically based on the selected dimensioning format.

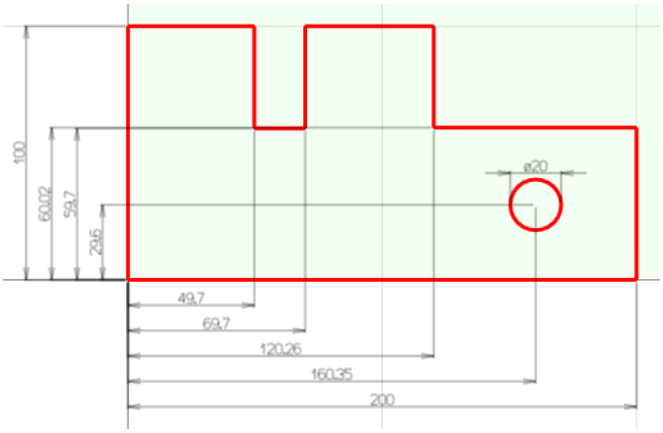
Fig. 7-108: Automatic alignment of dimension lines on a part



Origin

Dimension lines are aligned to the origin based on the selected dimensioning format.

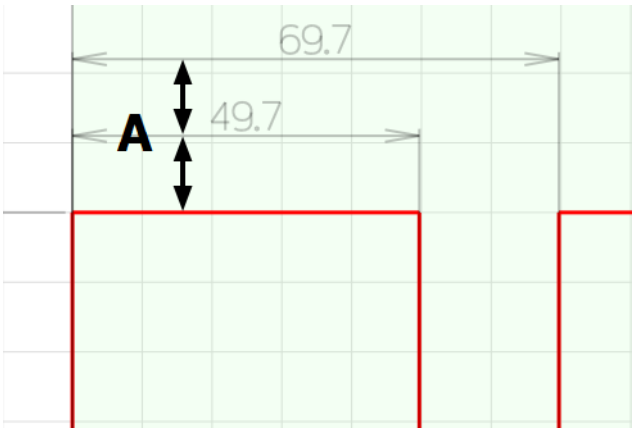
Fig. 7-109: Alignment of dimension lines on a part



Distance

Defines the parallel distance (**A**) of the dimension arrows to each other and to the contour.

Fig. 7-110: Distance of dimension lines to a part

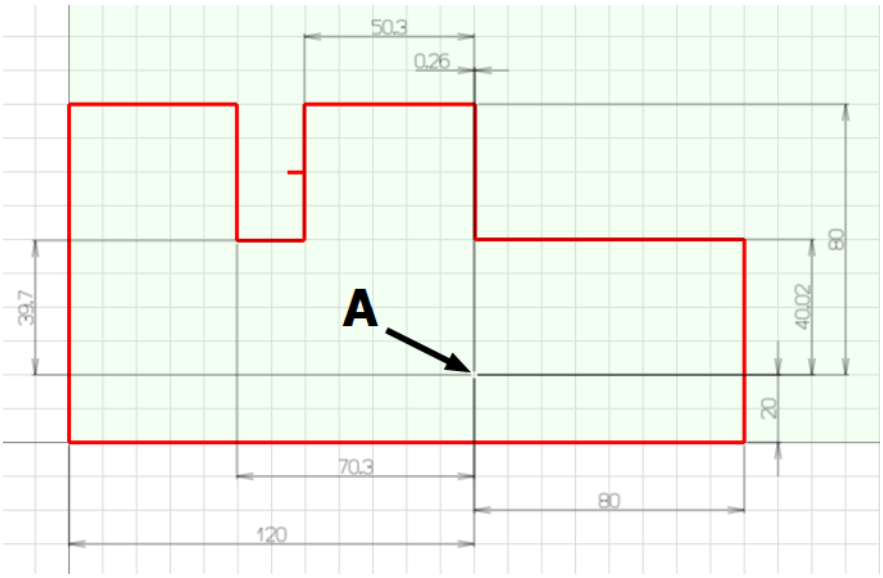


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Origin

This section is used to define the origin for alignment of dimension lines by entering it or by using the mouse in the workspace.

Fig. 7-111: Manually defined origin for dimension lines (**A**)



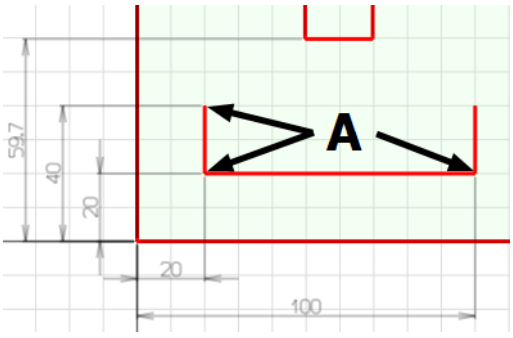
Options

This section is used to define the dimensioning of open contours.

Open contours

If this option is enabled, open contours are also dimensioned (**A**).

Fig. 7-112: Dimension lines on open contours



7.4.11.2 Insert dimensioning



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Dimensioning**

Use this function to insert dimensionings on a part. You can insert the following types of dimensioning:

- Unit of length
- **Arc:** *The first click on an arc dimensions the radius, the second the diameter.*
- Radius
- **Diameter:** *The first click on a circle dimensions the diameter, the second the radius.*
- Angle

Attention! When inserting, the program distinguishes between whether a snap point or a segment is selected.

Note: To select snap points, activate the appropriate selection functions.

Inserting a dimensioning for a unit of length

1. Select the **Insert** function.
2. Pick the starting point of the line segment.
3. Pick the end point of the line segment.
4. Using the mouse, position the distance of the dimensioning to the line segment.
5. Left-click to apply this position.

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Inserting a dimensioning for an arc

1. Select the **Insert** function.
2. Pick the starting point of the arc segment.
3. Pick the end point of the arc segment.
4. Click on the arc segment.
5. Using the mouse, position the distance of the dimensioning to the arc segment.
6. Left-click to apply this position.

☞ **Note:** The measurement refers to the length of the arc segment.

Inserting a dimensioning for a radius

1. Select the **Insert** function.
2. Select the arc segment.
3. Use the mouse to position the radius measurement.
4. Left-click to apply this position.

Inserting a dimensioning for a diameter

1. Select the **Insert** function.
2. Click on the contour of the circle to display the diameter.
3. Use the mouse to position the diameter.
4. Left-click to apply this position.

Inserting a dimensioning for an angle

1. Select the **Insert** function.
2. Select the first line segment.
3. Select the second line segment.
4. Using the mouse, position the distance of the dimensioning to the line segment.
5. Left-click to apply this position.

7.4.11.3 Edit dimensioning



How do I open this dialog box?

Select: **Modules > Part Editor > Geometry > Dimensioning**

Use this dialog to edit or modify dimensionings. The changes to the dimensioning are displayed gray in a preview (**A**).

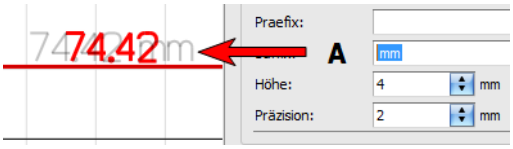


Fig. 7-113: Preview of a changed dimensioning

Note: Press **Delete** or use the **Delete contour** function to remove dimensionings.

Tip: You can configure global changes to dimensioning attributes in the Part Editor settings *(See: ›Dimensioning‹ Page 345)*.

Example:

1. Select the **Edit** dimensioning button.
2. Select the dimensioning on the part that you wish to edit.
3. Enter the changes in the appropriate boxes in the **Edit dimensioning** dialog.
4. Click **Apply** to save the changes.

Text

User-defined

This option overwrites the dimensioning text with the text input in this box.

Prefix

This option prefixes the dimensioning text with the text input here.

Suffix

This option appends the text input here to the dimensioning text.

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Height

This option defines the size of the dimensioning text.

Precision

This option configures the decimal places for dimensioning.

Construction lines

Distance

This option specifies the distance of the construction line tot he contour.

Prolongation

This option extends the construction line beyond the dimension arrow.

Arrow

Length

This option specifies the length of the construction line arrow.

7.4.11.4 Move dimensioning



Where do I find the function?

Select: **Modules > Part Editor > Geometry > Dimensioning**

Use this function to change the distance of the dimensioning to the dimensioned contour with the mouse.

Moving the dimensioning of a contour

1. Select the Move function.
2. Select the dimensioning that you wish to move.
3. Use the mouse to position the dimensioning.
4. Left-click to apply this position.

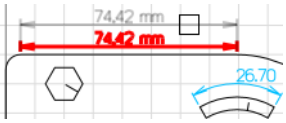


Fig. 7-114: Changing the distance of the dimensioning to the contour

7.5 Bending

In this area you can assign to imported bending parts the bending technology for Bystronic press brakes. To access the Bending area, click the **Bend** view in the software header or click the **Tools** menu. Additional menus are displayed. The menus are arranged in the order required for specifying the bending technology.

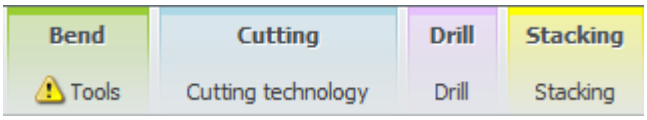


Fig. 7-115: Area display in the header

Status indicator

A status indicator showing the status of the relevant process step is displayed before each menu name in the **Bending** area.

- = All criteria in the menu are met.
- = The criteria in the menu have not been verified yet.
- = The criteria in the menu have not been set yet or are linked to errors.

Note: The **Bending** area is only available if the relevant license has been purchased and the part contains bending information.

Displaying messages

All messages, warnings and errors are displayed in a message window. The message window is displayed in the lower left corner for a certain period of time. It disappears again once this time has elapsed.

Tip: The message window is permanently displayed while the mouse is lying over it.



Fig. 7-116: Warning displayed in a message window

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Menus

The following menus are displayed in the **Bending** area:

- Bending sequence
- Tool setup plan
- Backgauges
- Bending aids
- LAMS (Angle measurement system)
- Simulation

The states of the bending data in the menus are indicated by different icons preceding the menu label.

-  Bending data are OK.
-  Bending data must be checked again.
-  There is one or more warnings.
-  Warnings were cleared.
-  There is one or more errors.
-  Errors were cleared.

7.5.1 View bar

The View bar can be used to manage the display in the 3D window. The View bar is available is every bending menu except the **Tools** menu. The View bar is divided into the following areas:

Tip: The view bar can be positioned horizontally or vertically next to the view window to suit the user's needs.

Tool view

This area controls the display of various machine components. The buttons have the following meanings from left to right:

- **Display bending machines:** *Enables/disables display of the machine.*
- **Display top tools:** *Enables/disables display of the top tools.*
- **Display bottom tools:** *Enables/disables display of the bottom tools.*

Alignment

In this area you specify the orientation you require for the view in the display window. The buttons have the following meanings from left to right:

- **Isometric:** *An isometric projection view is shown in the 3D window.*
- **Front elevation:** *A front elevation view is shown in the 3D window.*
- **Side elevation:** *A side elevation view is shown in the 3D window.*
- **Plan view:** *A plan view is shown in the 3D window.*

Zoom functions

- **Zoom all** *Zooms to all visible objects in the 3D window.*
- **Zoom station:** *Zooms to the active bending station in the 3D window.*
- **Zoom part:** *Zooms to the part in the 3D window.*
- **Zoom collision:** *Zooms to the current collision in the 3D window.*

Center

- **Center automatically:** *Centers the view of the bending part in the 3D window.*

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7.5.2 Navigation bar

The Navigation bar can be used to simulate the bending process and the movement of machine components. The Navigation bar is available in every bending menu except the **Tools** menu. The Navigation bar is divided into the following areas:

Bending process

This area controls the display and sequence of the bending processes. The buttons have the following meanings from left to right:

- Switches to the first process.
- Switches to the previous process.
- Switches to the next process.
- Switches to the last process.
- Starts the simulation.
- Stops the simulation. Continues the simulation when pressed again.
- Stops the simulation.



Speed

The simulation speed can be set with the slide control.



Simulating movements and collisions

This area is used to control the simulation of various machine components. The buttons have the following meanings from left to right:

- Displays the current bending process before it has been performed.
- Displays a collision during the bending process.
- Shows the movement of the backgauges, bending aids, and LAMS.



7.5.3 View window

The view for the bending menus is composed of a View window and a docking dialog box.

View window

Excepting the **Tools** menu, the main window displays in a 3D view the bending part and all components fitted on the machine.

The display in the main window can be changed from the View bar and Navigation bar.

Moving the 3D view using the center scroll button on the mouse

- The center scroll button on the mouse rotates the 3D view.
- The center scroll button on the mouse + the **Ctrl** key moves the view horizontally or vertically.

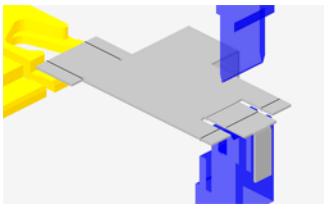


Fig. 7-117: 3D view in the main window

Docking dialog box

Depending on the menu, the docking dialog box displays the bending processes and the views showing the bending sequence and station plan.

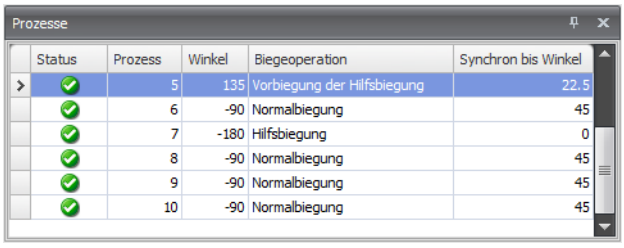


Fig. 7-118: Docking dialog box in the **Backgauges** menu

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7.5.4 Group: Technology

Where do I find the group?

Select: **Modules > Part Editor > Tools**

The functions in this group can be used to specify the entire bending technology settings and assign them automatically to the bending part.

7.5.4.1 Set bending technology automatically



Where do I find the function?

Select: **Modules > Part Editor > Bend > … > Technology**

This function is used to assign the entire bending technology settings to the bending part that have been defined in the **Bending technology settings** docking dialog box. The fully automatic procedure for creating the bending technology involves the following process steps:

1. Specify bending tools.
2. Create bending sequence.
3. Create a tool setup plan.
4. Calculate backgauges.
5. Configure bending aids (only if fitted).
6. Configure angle measurement system (only if fitted).

7.5.4.2 Complete all bending technology settings



Where do I find the function?

Select: **Modules > Part Editor > Bend > … > Technology**

This function calculates all the bending technologies that have not been set yet. Technologies that have already been set are retained and are not recalculated.



Tip: You can also press the **F3** button on the keyboard to run the **Complete** function.

7.5.4.3 Bending technology settings

How do I open this dialog box?

Select: **Modules > Part Editor > Bend > … > Technology**

This dialog box is where the technology settings are created for the processes, bending sequence, tool setup plan, backgauges, bending aids and angle measurement system.

The settings are applied to the bending part after running the function **Set bending technology automatically** or clicking on the **Apply** button.

Tools

Mode

- **Default tools**

  This option sets default tools for all bending processes.

- **Radius**

  For each bending process, this option uses the tool pair which results in a radius that matches the programmed radius (on the bend line) as closely as possible.

- **Bend allowance**

  For each bending process, this option uses the tool pair which results in a bend allowance that matches the programmed radius bend allowance (stored on the bend line) as closely as possible.

Tolerance

This option defines the tolerance for **Radius** and **Bend allowance** allow in which the same tools can be used.

If there are different bend allowances or radii inside this tolerance, the same tools are used for the bends.

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Top / bottom tool

You can change the default tools for the part in the selection boxes. Detailed information about the selected tool is displayed below the two selection boxes.



Note: Bending tools can only be changed if **Default tools** mode is selected.

Bending sequence

In this area you specify the options for selecting the tools and creating the bending sequence. An optimum bending sequence can be calculated by making suitable settings.

Check whether bending part will collide with itself

This option prevents the sides of a part from touching. Only use this option for special cases because it seriously restricts the possible bending sequences or even excludes any bending sequences.



Tip: If sides are bent towards each other then these may touch along the edge. If this option is enabled, a collision warning is displayed even though the bend could be made without a problem. You will need to disable this option to avoid this warning.

Select top tool automatically

This option means that another top tool is selected when the number of bending sequences specified under the option **Maximum number of bending sequences** cannot be found using the standard tool.

Disable this option to keep the top tools unchanged during automatic computation of the bending sequence.

Assess backgauges

This option determines the bending sequence on the basis of the better position of the rear backgauges.



Note: This option can only be selected if a number >1 has been set for the option **Maximum number of bending sequences**.

Max. number of bending sequences

This option sets the maximum number of bending-sequence solutions that you wish the program to find for the parts. If this number is reached, the best solution is assigned to the part as the bending sequence.



Note: Increasing the number can lead to considerably longer computing times for complex bending parts.

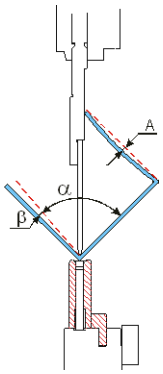
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Elastic tolerance

Caution! Only use this function for thin sheets (up to about 2mm). For thicker sheets and larger tolerance angles, the tool or the bending machine may be damaged if it collides with the bending part with too great a force.

If slight contact needs to be made with the top or bottom tool in order to produce a bend, then enter the tolerance value (in degrees) in the option field. You can enter a maximum tolerance angle of 15°. This tolerance value is then added to the target value of the bend angle when checking the angles. No collision message is issued inside this value.

Note: The deformation of the material when it collides with the top or bottom tool must remain within the elastic region of the material. Otherwise the material is permanently deformed!



A	elastic deformation region
α	maximum bend angle without collision
β	tolerance

Position of center of gravity of part

This option specifies whether the center of gravity of the part is preferably in front of or behind the bending tool for the bending sequence.

☞ **Note: Center of gravity, front:** Remember for very long sides the high angular velocity during bending.

☞ **Note: Center of gravity, rear:** For very long sides, make sure that the bending part rests on the backgauge finger to prevent sagging of the side.

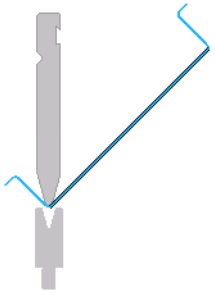


Fig. 7-119: Center of gravity of part in front of the bending tool

Bend length

This option specifies whether short or long sides of a part shall preferentially be bent first.

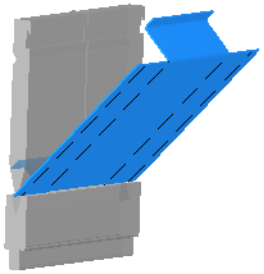


Fig. 7-120: Short bend length first

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Center of part

This option specifies which bend shall preferentially be made first based on the distance from the center of the part.

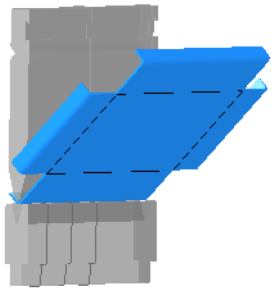


Fig. 7-121: Those further from center of part first

Use alternative tools

If alternative tools are assigned to the standard tools of the appropriate machine in Bending Administrator, these tools are also used to calculate the bending sequence.

Assessment criteria

This setting affects the weight attached to a criterion when calculating the bending sequence. The higher a criterion is placed, the more weight is given to this criterion in the calculation. You can use the arrow buttons to shift the weighting order of the criteria.

☞ **Note:** The criteria correspond to the options above and the properties that have been set for them. The **Sheet handling** option means that optimum bending of the part is possible.

Station

In this area you define the criteria for automatic computation of the bending stations.

Minimum distance

This option sets the minimum distance (**A**) between multiple stations. If the bending part collides with another bending station, move the distance of the bending station that the part collides with until the collision is removed.

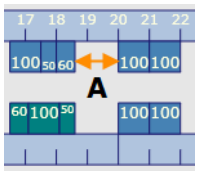


Fig. 7-122: Distance between two bending stations

Grid

This option generates a grid for positioning the bending tools on a station plan. The position always refers to the left-hand position of a bending station.

☞ **Note:** Hämmerle machines have a fixed mechanical positioning grid of 50mm. The grid pitch value for these machines must therefore always be divisible by 50.

Max. shortfall of station length

Maximum length by which a station may be shorter than bend length.

Minimize number of segments

This option tries to minimize the number of segments on the station after calculating the optimum number of segments on the station.

☞ **Note:** Disable this option if you do not wish to minimize segments.

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Backgauges

In this area you define the criteria for automatic computation of the backgauges for the bending process.

Minimum edge length

This option sets the minimum length of an edge for which a backgauge can be selected. A default length of 10 mm is set. Only those numbers of the contact edges that meet this minimum length can be selected using the **Contact edge** option in the **Edit backgauges** menu.

Retraction limit

This setting defines the lower limit as of which a retraction is performed. A retraction of less than this value is not performed and is set to **0**.

Application:

This setting permits small collisions without a retraction. This is often used in thick sheets. The default minimum retraction is 0.2 mm *(See: ›Retraction‹ Page 607)*.

Auto-snap

This option enables automatic snapping of existing backgauges.

Note: This option is only possible if a backgauge finger for corner stops is fitted.

Use one backgauge only

This option is used for automatic calculation of backgauges for the bending part with only one backgauge.

Tip: This avoids calculating a non-ideal backgauge configuration for small parts.

Optimize

Optimization reduces backgauge axis movements to a minimum. This increases the efficiency of bending processes.

- **Z position**

  The program tries to use the same backgauge positions in the Z direction.

- **Position window**

  Specifies the maximum distance of the axis in the Z direction in which the backgauges may be automatically moved.

- **Final bend**

  For bending processes that do not use any backgauges, the X position of the previous bending process is used.

  Bends without backgauges:

  - Auxiliary bend
  - Pre-bending
  - Hemming

- **Parked position**

  This option prevents the unused backgauge from moving to Z parked position on short backgauge edges. The program tries to use the Z backgauge position from the previous bending process. If this is not possible, the unused backgauge is parked immediately alongside the used backgauge.

Bending aid

This area is only displayed if the bending machine selected in the menu **Part** > **Properties** > **Bend** has at least one bending aid.



Tip: Use bending aids to support large, bulky or heavy sheets during the bending process.

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Use

The selected attribute is applied to all available bending aids.

- **Positioned**

  This attribute is used to position all available bending aids horizontally, if this is technically possible from a bending viewpoint. During the bending process, the bending aid **does not** perform a supporting tilting movement in parallel with the bending process.

  Note: The front bending aids must be positioned manually on the machine for safety reasons.

  Tip: Use this attribute when you want to use the bending aid as a sheet rest.

- **Parked**

  This attribute is used to move all available bending aids to the parked position. The bending aid remains in the parked position for the current process until the **Park** attribute is manually disabled using the **Edit** function from the **Bending aids** menu, or until another use has been selected.

- **Enabled**

  This attribute is used to assign a bending aid automatically to all bending processes, if this is technically possible from a bending viewpoint. All bending aids fitted are included. During the bending process, the bending aid **does** perform a supporting tilting movement in parallel with the bending process.

- **W position**

  The program tries to use the same positions of the bending aids in the Z direction.

- **Parked position**

  This option prevents unused bending aids from moving to Z parked position. The program tries to use the position from the previous bending process for unused bending aids. If this is not possible, the unused bending aid is parked immediately alongside the used bending aid.

  Note: This option can only be used for rear bending aids.

Angle measurement system

This area is only displayed if the bending machine selected in the menu **Part** > **Properties** > **Bend** is fitted with an angle measurement system (LAMS).



Tip: Use this option when working on expensive materials or when bending small numbers of heavy-gage and large parts.

Use

This option defines the basic setting for the angle measurement system.



Note: If the angle measurement system is equipped with motorized positioning, then the system is positioned automatically.

- **Positioned - All measurements on**

  The angle measurement system is enabled for all bending processes and positioned on the current bending station.

- **Positioned - All measurements off**

  The angle measurement system is disabled for all bending processes and positioned on the current bending station.

- **Positioned - First bend angle measured per angle**

  The angle measurement system is enabled and positioned on the current bending station. If several identical bending angles are bent successively, only the first bend angle is ever measured until a new angle is bent.

- **Parked**

  You use this attribute to move the Angle measurement system into the parked position. The angle measurement system remains in the parked position in the current process until the **Park** attribute is manually disabled using the **Edit** function from the **LAMS** menu, or until another use has been selected.

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Every n'th bending part

For a series of identical bending parts, you can enter here the number of parts after which you want the angle measurement system to check the part.



Note: This option only works when the attribute **Positioned - All measurements on** or **Positioned - First bend angle measured per angle** is selected for the **Use** option.

Mode

The spring-back measurement is needed to correct for the spring-back of the material during bending.

- **Measurement On**

  The spring-back angle is measured automatically.

- **Database**

  The spring-back angle from the database is used.

- **Manual**

  The spring-back angle must be entered manually via the **Angle** option.

Angle

If the **Manual** attribute is selected for the **Spring-back angle**, the spring-back angle can be entered manually in this field.

Optimize

- **Z position**

  If possible, the program tries to use the same LAMS positions in the LZ direction.

- **Position window**

  Specifies the maximum distance of the axis in the LZ direction in which the automatic angle measurement system may be moved.

Sheet thickness compensation

Mode

- **Off**
  Sheet thickness measurement is switched off.
- **On**
  Sheet thickness measurement is switched on for all processes.
- **Adopt**
  Sheet thickness measurement is switched on for the first process. All following processes are set to **Accept** mode.

Direction of rolling compensation (whole part)

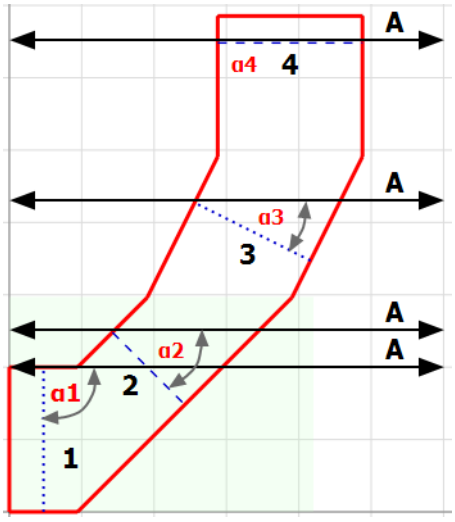
Grain direction compensation accounts for the grain direction of the sheet when bending, taking into account the angle (**α1-4**) between grain direction (**A**) and the bend lines (**1-4**).

Example:

- Angle correction = **2°**
- Grain direction = **0°**

Angle	Degrees	Compensation
α1	90°	2°
α2	45°	1°
α3	27°	0.6°
α4	0°	0°

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A
α4
4
A
α3
3
A
A
α2
α1
2
1

Mode

- **Off**
  No grain direction compensation is performed.
- **Database**
  The value for angle correction is used from the bending table in Administrator.
- **Manual**
  The values for angle correction and grain direction can be entered manually.

Angle correction

You can enter a manual correction value. The angle correction can only be entered in manual mode.

Grain direction

Defines the grain direction of the part with the angles **0°** and **90°**.

- **0°** = grain direction in the horizontal direction
- **90°** = grain direction in the vertical direction

And angle of **0°** is used by default.

7.5.4.4 Check bending technology

Where do I find the function?



Select: **Modules > Part Editor > Bend > … > Technology**

This function checks all menus in the **Bend** section and refreshes their statuses accordingly.

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7.6 Menu: Tools

Where do I find the menu?

Select: **Modules > Part Editor**

In this menu you define the bending tools that you wish to use to produce the bends. The bending tools can be assigned manually or automatically.

Functional groups

The function bar contains the following functional groups:

- Selection
- Technology
- Tools
- Bends
- View
- Auxiliary functions

7.6.1 Group: Tools

Where do I find the group?

Select: **Modules > Part Editor > Tools**

The functions in this group are used to assign the relevant bending tool to the bending processes. The tool can be assigned manually or automatically. It is also possible to edit bending tools that have already been set.

7.6.1.1 Tool cabinet



Where do I find the function?

Select: **Modules > Part Editor > Tools > Tools**

This function can be used to assign tool cabinets to the bending part.

Tool cabinets can only be assigned to the bending part if tool cabinets are configured for the selected bending machine. Otherwise, an error message is shown.



Tip: Tool cabinets can be created for the appropriate bending machine in ByVision Bending Administrator.

- If tool cabinets are enabled, only the appropriate number of tools configured for the bending machine are used to calculate setup plans.
- If no tool cabinets are enabled, all tools (any number) assigned to the bending machine are used to calculate setup plans.

The **Tool cabinet** dialog displays all available tool cabinets. Click on the checkbox to assign the tool cabinets to the bending part. The selection is applied immediately.

Prioritizing tool cabinets

If there are multiple tool cabinets for a bending machine, tool use is prioritized based on the order of tool cabinets in the dialog. All tools in the first tool cabinet are used first. Then the next tools, and so on.

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7.6.1.2 Set tools automatically



Where do I find the function?

Select: **Modules > Part Editor > Tools > Tools**

This function is used to assign the selected default tools automatically to each bending process. The tools are determined in the following steps:

1. The program tries to set the default tools.
2. If the default tool is not valid for a bend, all tools are determined. The upper and bottom tool with the smallest change of bend allowance compared with the current bend allowance is assigned to the bend.

 **Note:** Tools already set are removed.

7.6.1.3 Tools settings

How do I open this dialog box?

Select: **Modules > Part Editor > Bend > Tools > Tools > Automatic > Settings**

This dialog box is used to configure settings for automatic assignment of tools.

The settings are applied to the bending part after running the **Automatic** or **Complete** function.

Mode

- **Default tools**

  This option sets default tools for all bending processes.

- **Radius**

  For each bending process, this option uses the tool pair which results in a radius that matches the programmed radius (on the bend line) as closely as possible.

- **Bend allowance**

  For each bending process, this option uses the tool pair which results in a bend allowance that matches the programmed radius bend allowance (stored on the bend line) as closely as possible.

Tolerance

This option defines the tolerance for **Radius** and **Bend allowance** allow in which the same tools can be used.

If there are different bend allowances or radii inside this tolerance, the same tools are used for the bends.

Top / bottom tool

You can change the default tools for the part in the selection boxes. Detailed information about the selected tool is displayed below the two selection boxes.

☞ **Note:** Bending tools can only be changed if **Default tools** mode is selected.

7.6.1.4 Complete all tools settings



Where do I find the function?

Select: **Modules > Part Editor > Tools > Tools > Automatic**

This function is used to assign the tools automatically to all the bending processes, based on the criteria for material, thickness, bend angle etc. contained in the database settings.

☞ **Note:** Tools that have already been assigned are retained unchanged.

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7.6.1.5 Check tools



Where do I find the function?

Select: **Modules > Part Editor > Tools > Tools**

This function checks whether the bending processes can be bent with the assigned tools. The result of the check is displayed in a message window.



Note: Tools that already have the OK status Dimensioning () are not checked again.



Fig. 7-123: Message window displayed after a check

7.6.1.6 Edit tools



How do I open this dialog box?

Select: **Modules > Part Editor > Tools > Tools**

This function can be used in the **Edit tools** dialog box to assign to the currently selected bending process a top tool and bottom tool for the selected **Bending method**.

Clicking the **Apply** button runs a check of the chosen tools and applies them to the bending process. If the tools are not suitable for the bending process, the last valid tools are used.

Show all tools

This option shows all tools that have a different bend allowance. Enable this option if you wish to select tools that have a different bend allowance.



Attention! If a bend allowance changes, the geometry of the part is not corrected accordingly. Hence changing the bend allowance results in a change in the side length of the part.



Note: This option should be disabled by default in order to ensure that only tools having the same bend allowance are used.

Method

In this section you select the bending and hemming method for the selected bending process.

Bending methods

- Air bending
- Coining
- 3-point bending

Hemming

If the selected bend is of the **Hemming** type, you can select the method in the **Hemming** selection dialog.

- **Standard embossing:** Hemming is performed with normal pressing force. After bending, a small gap remains along the bend line (**A**).
- **Strong embossing:** Hemming is performed with increased pressing force. After bending, almost no gap remains along the bend line (**B**).

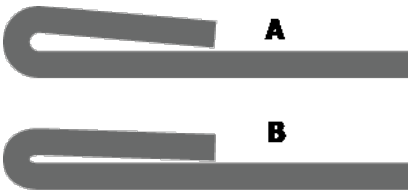


Fig. 7-124: Hemming methods

☞ **Note:** In order to enable the dialog for hemming, it is necessary to select the **Coining** method in the **Bending method** selection dialog or additionally, for Hämmerle bending machines, **3-point**.

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Adapter 1/2

Tool adapters for the top and bottom stations can be defined in these tabs.

Adapters can be used for the following use cases:

- As an extension of the bending tool in order to prevent collisions with the machine beam
- As a tool adapter in order to use bending tools from a different manufacturer.

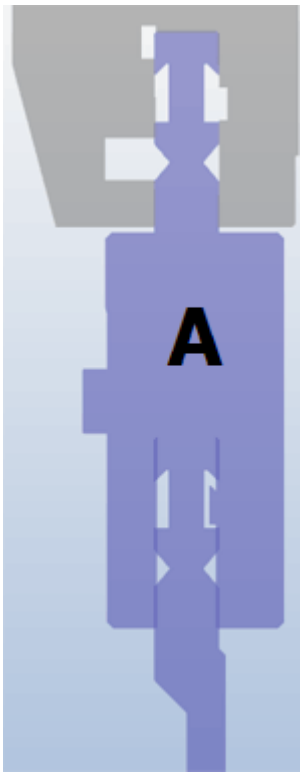


Fig. 7-125: Adapter 1 in top station (**A**)

7.6.1.7 Clear warning



How do I open this dialog box?

Select: **Modules > Part Editor > Tools > Tools**

This function acknowledges and clears the warning for the selected bending process.

Displaying warnings for tools

The warning is displayed as a symbol after the tool name in the **Processes** dialog box. Details about the warning are displayed when you move the mouse over the warning symbol.

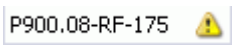


Fig. 7-126: Warning display prior to acknowledgment

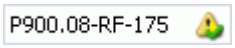


Fig. 7-127: Warning display after acknowledgment

Note: If the message is not acknowledged, another tool suitable for the bending process must be selected.

Tip: Use this function for warnings that can be ignored for the selected bending process. Acknowledging the warning tells the system that the tool can be used for the bending process.

7.6.1.8 Clear errors



How do I open this dialog box?

Select: **Modules > Part Editor > Tools > Tools**

This function is used to clear errors from allocated tools.

Note: This function is only enabled if the pending error can be cleared.

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7.6.2 Group: Bends

Where do I find the group?

Select: **Modules > Part Editor > Tools**

The functions in this group can be used to join bend lines together or to resolve joined bend lines back into separate bend lines.

7.6.2.1 Update bend allowance



Where do I find the function?

Select: **Modules > Part Editor > Tools > Bends**

This function corrects the bend allowances of the geometry based on the set tools.



Tip: Use this function after changing a tool or a corner relief to adapt the bend allowance of a bend.

Without geometry change

This sub-function transfers bend allowance and the resultant radius of the tool combination to the bend line without shortening or stretching the geometry.



Tip: Use this sub-function to assign the bend allowance to the bend line if the tool is known and the geometry is already correctly shortened.

7.6.2.2 Autoconnect bends



Where do I find the function?

Select: **Modules > Part Editor > Tools > Bends**

This function is used to autoconnect all bends on a common axis. This lets you bend several sides in one bending process*(See: ›Connect bends‹ Page 543)*

7.6.2.3 Connect bends



Where do I find the function?

Select: **Modules > Part Editor > Tools > Bends**

This function is used to connect two or more bends that lie on a common axis into one bend. This lets you bend several sides in one bending process.

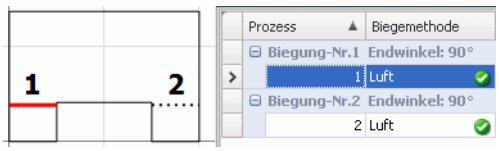


Fig. 7-128: Two separate bends on one bend line



Fig. 7-129: Connected bends on one bend line

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Connecting bends that have a contour between the side pieces

The bends can still be connected even when there is a contour (**A**) between the side pieces.

In order to be able to bend the connected bend lines at the same time, the bending station must contain empty segments in the area of the contour **A**. Alternatively, the bend can also be bent between two bending stations spaced the appropriate distance apart.

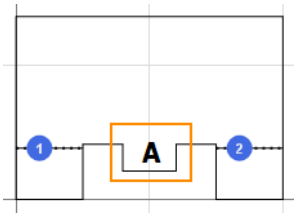


Fig. 7-130: Contour between two bends

Connecting two bend lines into one bend

1. Use **Shift** + left mouse button to select all the bends on a bend line that you wish to connect.
2. Select the **Connect** function.
3. The selected bends are connected together.

7.6.2.4 Resolve bend

Where do I find the function?



Select: **Modules > Part Editor > Tools > Bends**

This function resolves all connected bends on the selected bend line back into separate bends. After resolving the bends, each side can be bent separately again.

7.6.3 Group: Stepped bends

Where do I find the group?

Select: **Modules > Part Editor > Tools**

The functions in this group can be used to join several bend lines into stepped bends or to resolve a stepped bend back into separate bend lines.

7.6.3.1 Combine into stepped bend



Where do I find the function?

Select: **Modules > Part Editor > Tools > Stepped bend**

This function is used to connect two or more parallel into one stepped radius bend.

1. Left-click and drag out a frame to mark all bends in the step bend.
2. Select the **Connect** function.
3. The bends are converted to a stepped radius bend.



Tip: Define the bending sequence of the step bend so that the bending process starts with the bend line furthest from the contact edge and ends with the shortest. This bends the radius towards the operator (**A**). This allows you to set the part to the same contact edge for every step bend.

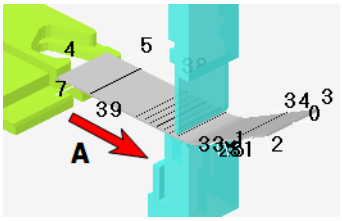


Fig. 7-131: Stepped radius bend bending sequence

Connecting step bends has the following advantages:

- The ram moves up only slightly between the bends so that the sheet is just about no longer clamped. This speeds up the bending process.
- If you enter an angle correction after bending the stepped radius bend, it is applied proportionately to all bend lines.

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7.6.3.2 Resolve stepped bend



Where do I find the function?

Select: **Modules > Part Editor > Tools > Stepped bend**

This function converts all selected step bends into normal bends.

7.6.3.3 Convert stepped bend



Where do I find the function?

Select: **Modules > Part Editor > Tools > Stepped bend**

This function can be used to transform a bend into a stepped bend which can no longer be bent through its bend radius using a tool.

The red line in the graphic preview shows the curve of the stepped bends in relation to the original bend radius of the part.

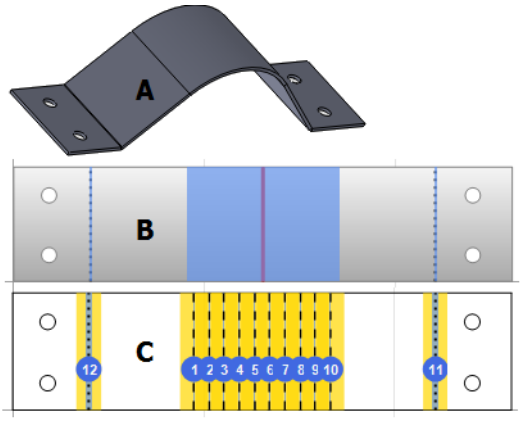


Fig. 7-132: Converting a bend radius into a stepped bend

A	Bending part with a large bend radius
B	Imported bending part with bend zone displayed
C	Bend radius converted into stepped bends

Converting a large bend radius into a stepped bend

1. Import the bending part (**A**) into the Part Editor.
2. Select the bend line which is to be converted into a stepped bend (**B**).
3. Select the **Convert** function in the **Step bend** group.
4. In the **Step bend conversion** docking dialog box, specify the type, the number of bends and the tools.
5. Use the **Apply** button to assign the stepped bend of the bend line to (**C**).

General

Bend zone limited

This option prevents the entry zone of the first and last step bend overlaps the entry zone of an adjacent bend (**D**).

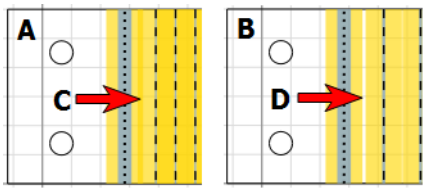


Fig. 7-133: Max. number of step bends without (**A**) and with (**B**) limited bend zone

☞ **Note:** This option can only be used with the **Tangential inside** type.

Type

The edges and areas of a stepped bend do not run completely parallel to the actual bend radius. Use this option to specify whether the bend radius should run along the outer or inner side of the stepped bend. The default selection is the attribute **Tangential inside**.

☞ **Note:** This option must only be altered if another part is to be added to fit along the inside or outside of the bend.

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Number of bends

Use this option to specify the number of bends to be used to create the radius. Use the **Maximum** button to enter the maximum possible number of bends in the **Number of bends** field.

The higher the number of bends, the more exact the bend radius and the more complex the bend will be.

☞ **Note:** The maximum possible number of bends is dependent on the tool settings.

Top / bottom tool

Use this option to specify the top and bottom tools for the stepped bend.

Line spacing

Displays the distance of the bend lines of a step bend to each other. Increase or decrease the number of bends to change the line spacing displayed. Smaller spacing results in a more accurate arc of the step bend.

Model

These options allow you to create step bends that are only performed at both ends of a rectangular contour.

Application:

These options can be used to create clamps or tubes that are first bent and then rolled.

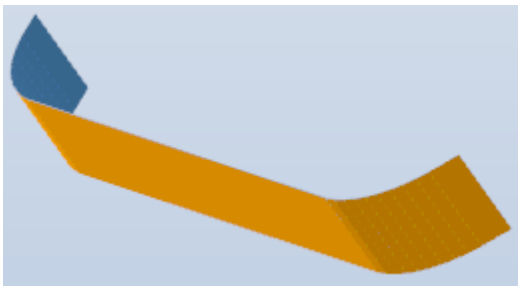


Fig. 7-134: Step bends at ends of contour

Limit

Enables the options for limiting step bends.

Number of bends

Limits the step bends performed on both sides of the contour to the number indicated.

Length

Displays the length of the area in which the defined number of step bends is located. The length is the same on both sides.

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7.6.4 Group: View

Where do I find the group?

Select: **Modules > Part Editor > Tools/Bending sequence**

This group contains all the functions that can be used to change the current view of the bending part.

The various functions and views are classified as follows:

Zoom functions

The zoom functions can be used to magnify or reduce objects in the graphics window.

Tip: You can also perform the zoom functions using the mouse thumb wheel.

Moving the view

The arrow symbols are provided in this group for moving the view of an object.

Tip: You can use the center mouse button to move the object freely in the current view. The view is moved by dragging and dropping with the mouse.

Special view

The **Special view** function is used to display various view options. The special-functions display can be specified by clicking on the arrow symbol and enabled/disabled by clicking on the symbol above it.

7.6.4.1 Special view

Where do I find the function?

Select: **Modules > Part Editor > Tools/Bending sequence > View**

This function can be used to display various elements on the bend line of the part. The elements are shown/hidden by selecting the relevant element option. The elements can be displayed separately or simultaneously.

You can display the following elements:

- **Bend zone:** *Shows the area of the bend zone on a bend line.*
- **Entry zone:** *Shows the area of the entry zone on a bend line.*
- **Bend angle:** *Shows the bend angle of the bend line.*
- **Bend allowance:** *Shows the bend allowance dimension for the bend line.*
- **Bend:** *Shows the construction sequence for the bends. The number is used as unique identification of the bend on a part. It has no other significance.*
- **Process:** *Shows the sequence of the set bending processes.*

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7.6.5 Tools processes

How do I open this dialog box?

Select: **Modules > Part Editor > Tools**

This dialog box contains a table displaying the assignment of the tools to each of the bending processes. The currently selected bending process is shown in the 2D view of the Part Editor as a red bending line. One or more bending operations can be assigned to each bending process.

You can use the context functions **Expand all groups** and **Collapse all groups** to open up or re-close all the bending processes.



Tip: Double-clicking on a bending process opens the **Edit tool** dialog box in which you can edit the tool for the selected process.

What each column means:

Bend line

Shows the order of the bend lines in the bending process.

Final angle

If the side is bent with a pre-bend, the column displays the final angle with which the side is bent.

Process

A serial number indicates the sequence of the bending processes.

Bending operation

Shows the type of bend in the bending process.

- Normal bending
- Hemming
- Pre-bending
- Auxiliary bend
- Stepped radius bend

Bending method

Displays the bending method that is used to bend the process.

Angle

Shows the angle of the bend. An angle of **0** is only used for the final bend in a hemming operation.

Top tool

Displays the top tool used for the process and a status indicator for the tool.

Bottom tool

Displays the bottom tool used for the process and a status indicator for the tool.

Tool setup plan

Shows the tool setup plan number for the bend.

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Calculation

Shows the method used to calculate the bend allowance value.

Calculation methods:

- **DB** = With empirical values from the database.
- **Curve generator** = With a curve generator.
- **DIN** = With the DIN 6935 formula.
- **DB/DIN** = Mixed calculation basis.

Station

Shows the number of the station on the tool setup plan for the bend.

Top station, adapter 1/2

Shows the adapter for top stations 1 and 2

Bottom station, adapter 1/2

Shows the adapter for bottom stations 1 and 2

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7.6.6 New bending technology

This dialog box is displayed when the **Bending** area or the **Tools** menu has been selected but no bending machine has been assigned to the bending part.

Once the bending machine has been defined and confirmed by pressing the **OK** button, the display switches to the **Bending** section in the **Tools** menu.

Bending process

Machine

Defines the bending machine on which you wish to process the bending part. Clicking on the arrow symbol on the right lets you select a different bending machine from a list. This list only contains those bending machines that are available in the system.



Tip: You can create bending machines that do not exist yet in Administrator.

Number of operators

Defines the number of operators required to bend the part. Criteria for determining the number of operators are size, bend length, weight and complexity of the bending part.

Automatic

If the option is enabled, the number of operators for the bending part is determined automatically based on a table of limits *(See: ›Limits for individual operation‹ Page 101)*

Bending-time table

Defines the table for automatic calculation of bending time. You can edit tables in Administrator *(See: ›Bending-time table‹ Page 99)*.

Bending technology settings

from Settings Manager

The settings for bending technology are used by Settings Manager in Administrator.

General

General bending information about the part can be entered in the **Notes** field for the machine operator. The information is displayed on the bending machine in the bending program.

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7.7 Menu: Bending sequence

Where do I find the menu?

Select: **Modules > Part Editor**

In this menu you define the sequence in which you wish the bends to be made. The bending sequence can be assigned manually or automatically.

The display in the main window can be changed from the View bar and Navigation bar.

Functional groups

The function bar contains the following functional groups:

- Selection
- Bending sequence
- Auxiliary bend
- Pre-bending
- Rotate
- View

7.7.1 Group: Bending sequence

Where do I find the group?

Select: **Modules > Part Editor > Bending sequence**

This group contains functions that can be used to assign the bending sequence to the part automatically or manually. A check function lets you check all bending sequences for collisions.

7.7.1.1 Bending sequence settings



How do I open this dialog box?

Select: **Modules > Part Editor > Bending sequence > Bending sequence**

In this area you specify the options for selecting the tools and creating the bending sequence. An optimum bending sequence can be calculated by making suitable settings.



Tip: Click on the **Automatic** function to apply the settings to the bending part.

Check whether bending part will collide with itself

This option prevents the sides of a part from touching. Only use this option for special cases because it seriously restricts the possible bending sequences or even excludes any bending sequences.



Tip: If sides are bent towards each other then these may touch along the edge. If this option is enabled, a collision warning is displayed even though the bend could be made without a problem. You will need to disable this option to avoid this warning.

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Select top tool automatically

This option means that another top tool is selected when the number of bending sequences specified under the option **Maximum number of bending sequences** cannot be found using the standard tool.

Disable this option to keep the top tools unchanged during automatic computation of the bending sequence.

Assess backgauges

This option determines the bending sequence on the basis of the better position of the rear backgauges.

☞ **Note:** This option can only be selected if a number >1 has been set for the option **Maximum number of bending sequences**.

Max. number of bending sequences

This option sets the maximum number of bending-sequence solutions that you wish the program to find for the parts. If this number is reached, the best solution is assigned to the part as the bending sequence.

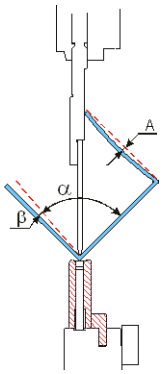
☞ **Note:** Increasing the number can lead to considerably longer computing times for complex bending parts.

Elastic tolerance

Caution! Only use this function for thin sheets (up to about 2mm). For thicker sheets and larger tolerance angles, the tool or the bending machine may be damaged if it collides with the bending part with too great a force.

If slight contact needs to be made with the top or bottom tool in order to produce a bend, then enter the tolerance value (in degrees) in the option field. You can enter a maximum tolerance angle of 15°. This tolerance value is then added to the target value of the bend angle when checking the angles. No collision message is issued inside this value.

Note: The deformation of the material when it collides with the top or bottom tool must remain within the elastic region of the material. Otherwise the material is permanently deformed!



A	elastic deformation region
α	maximum bend angle without collision
β	tolerance

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Position of center of gravity of part

This option specifies whether the center of gravity of the part is preferably in front of or behind the bending tool for the bending sequence.

☞ **Note: Center of gravity, front:** Remember for very long sides the high angular velocity during bending.

☞ **Note: Center of gravity, rear:** For very long sides, make sure that the bending part rests on the backgauge finger to prevent sagging of the side.

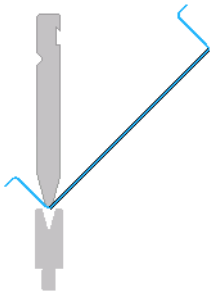


Fig. 7-135: Center of gravity of part in front of the bending tool

Bend length

This option specifies whether short or long sides of a part shall preferentially be bent first.

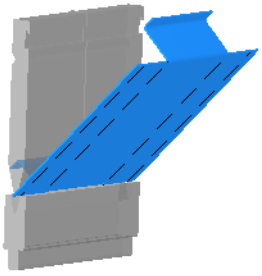


Fig. 7-136: Short bend length first

Center of part

This option specifies which bend shall preferentially be made first based on the distance from the center of the part.

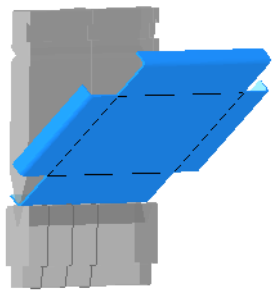


Fig. 7-137: Those further from center of part first

Use alternative tools

If alternative tools are assigned to the standard tools of the appropriate machine in Bending Administrator, these tools are also used to calculate the bending sequence.

Assessment criteria

This setting affects the weight attached to a criterion when calculating the bending sequence. The higher a criterion is placed, the more weight is given to this criterion in the calculation. You can use the arrow buttons to shift the weighting order of the criteria.

☞ **Note:** The criteria correspond to the options above and the properties that have been set for them. The **Sheet handling** option means that optimum bending of the part is possible.

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7.7.1.2 Set bending sequence automatically

Where do I find the function?

Select: **Modules > Part Editor > Bending sequence > Bending sequence**

This function is used to compute the optimum bending sequence based on the criteria in the **Bending sequence settings** dialog box, and then to assign the sequence to the bend line.

7.7.1.3 Check bending sequence

Where do I find the function?

Select: **Modules > Part Editor > Bending > Bending sequence > Bending sequence**

This function checks the set bending sequence for collisions. The result of the check is displayed in a message window.



Fig. 7-138: Message window

You can take the following actions if a collision is identified:

- Change the bending sequence until there are no more collisions.
- In the settings, disable the option **Check whether bending part will collide with itself**.
- In the settings, enable the options **Rotate top tool automatically** and **Select top tool automatically**.
- In the settings, enter an elastic tolerance, provided this allows the bending part to be bent *(See: ›Bending sequence settings‹ Page 557)*.

7.7.1.4 Edit bending sequence



How do I open this dialog box?

Select: **Modules > Part Editor > Bending sequence > Bending sequence**

This function can be used to move the bends to any position in the bending sequence.

- Moves the selected bend up one place.
- Inverts the order of all bends. All bending processes must be highlighted to perform this function.
- Moves the selected bend down one place.

Shaded areas of the bending sequence indicate where the selected bend cannot be moved into. For instance, a hemming bend cannot be bent before its corresponding pre-bend.

Note: Alternatively, you can move the bends by clicking and dragging with the mouse. The currently active bend line is highlighted in red in the **2D view** dialog box.

Tip: Several bends can be moved at once using the **Shift+mouse** key combination.

The currently selected bend can be previewed in the 3D window. To be able to preview the bend, the **Before bending** button must be disabled in the view bar.



Fig. 7-139: Before bending button disabled in the view bar

Attention! Bend lines that have no status indicator cannot normally be added to the bending sequence without causing a collision.

Note: If the status indicator is a question mark , the bend line has not been checked yet. Run the **Check** function to check the bending sequences for collisions.

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What each column means:

Status

Displays the status of the bend line in the bending sequence.

Symbol	Description
	The bending sequence for the bend line has not been defined yet and can be added to the bending sequence without causing a collision.
	A bending sequence has been assigned to the bend line.
	The bend can only be made with a collision.
	The bend with a collision has been accepted.
	The bending sequence has not been checked yet (only for **Edit** function).

Process

A serial number indicates the sequence of the bending processes.

Angle

Shows the angle of the bend. An angle of **0** is only used for the final bend in a hemming operation.

Bending operation

Shows the type of bend in the bending process.

- Normal bending
- Hemming
- Pre-bending
- Auxiliary bend
- Stepped radius bend

7.7.1.5 Set bending sequence

How do I open this dialog box?

Select: **Modules > Part Editor > Bending sequence > Bending sequence**

This function can be used to set the bending sequence manually on a part.

Double-clicking on any bend line in the table will set this bend line as the next bend in the bending sequence. The active bend line is highlighted in red in the **2D view** dialog box.

Tip: The bending sequence can also be set in the **2D view** docking dialog box.

Bend lines that have already been assigned to a bending sequence are separated from unassigned bend lines by a black bar. All assigned bend lines lie above the black bar.

Using the black bar

- One or more bending sequences can be reset by moving the black bar **upwards**.
- One or more bending sequences can be set by moving the black bar **downwards**.

The currently selected bend can be previewed in the 3D window. To be able to preview the bend, the **Before bending** button must be disabled in the view bar.



Fig. 7-140: Before bending button disabled in the view bar

Attention! Bend lines that have no status indicator cannot normally be added to the bending sequence without causing a collision.

Note: Clicking on the **Set** function again will reset all the set bending sequences.

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What each column means:

Status

Displays the status of the bend line in the bending sequence.

Symbol	Description
	The bending sequence for the bend line has not been defined yet and can be added to the bending sequence without causing a collision.
	A bending sequence has been assigned to the bend line.
	The bend can only be made with a collision.
	The bend with a collision has been accepted.
	The bending sequence has not been checked yet (only for **Edit** function).

Process

A serial number indicates the sequence of the bending processes.

Bending operation

Shows the type of bend in the bending process.

- Normal bending
- Hemming
- Pre-bending
- Auxiliary bend
- Stepped radius bend

Angle

Shows the angle of the bend. An angle of **0** is only used for the final bend in a hemming operation.

7.7.1.6 Accept collision



Where do I find the function?

Select: **Modules > Part Editor > Bending sequence > Bending sequence**

This function is used to accept the collision for the selected bend. It is only enabled if the selected bending process will cause a collision with the part itself or a machine component.

7.7.2 Group: Auxiliary bend

Where do I find the group?

Select: **Modules > Part Editor > Bending sequence**

This group contains functions that can be used to assign additional auxiliary bends to a bend or remove them again.

7.7.2.1 Insert auxiliary bend

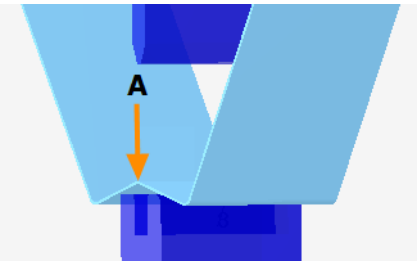


How do I open this dialog box?

Select: **Modules > Part Editor > Bending sequence > Auxiliary bend**

This function adds an auxiliary bend to the selected bend line.

An auxiliary bend is composed of at least two and at most three bend steps. In the first step, the auxiliary bend is pre-bent to the specified bend angle. In a second, later step, the auxiliary bend is bent back by the specified bend angle.



Auxiliary bend (**A**) for long sides prior to bending back

Adding an auxiliary bend to a bend line

1. Select the bend line to which you wish to add an auxiliary bend.
2. Select the **Add** function from the **Auxiliary bend** group.
3. Specify the offset from the bend line and the bend angle for the auxiliary bend.
4. Press the **Apply** button to set the auxiliary bend.



Tip: If one auxiliary bend is not enough, an extra pre-bend step can be added to the auxiliary bend so that the auxiliary bend can be made in two pre-bending steps.

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Adding an extra pre-bend to an auxiliary bend

1. Select the auxiliary bend to which you wish to add a second pre-bend.
2. Select the **Add** function from the **Pre-bending** group.
3. Specify the bend angle for the second pre-bend.
4. Press the **Apply** button to set the second pre-bend on the auxiliary bend.

Offset

This option is used to define the distance of the auxiliary bend from the selected bend line (**A**). The preview of the auxiliary bend is shown as a black line (**B**) in the 2D view.

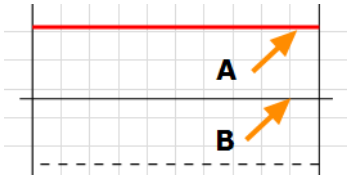


Fig. 7-141: Preview of an auxiliary bend

Bend angle

This option is used to define the bend angle (**A**) of the auxiliary bend. You can enter a positive or negative bend angle.

Tip: Choose as large an angle as possible for the pre-bend. An angle between 135° and 179° is recommended.

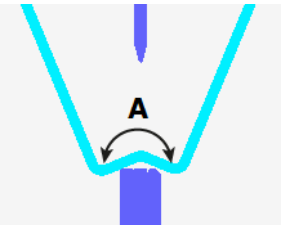


Fig. 7-142: Bend angle of an auxiliary bend

Note: The mechanical deformation of the material at the bend point means that the position of the auxiliary bend can still be seen after bending back. So only use this bending method for parts that do not need a high-quality surface finish.

Tip: Use this bending method for bending long sides whose bend lines lie too close together for a normal bend.

7.7.2.2 Remove auxiliary bend



Where do I find the function?

Select: **Modules > Part Editor > Bending sequence > Auxiliary bend**

This function removes the selected auxiliary bend. This function can only be used when an auxiliary bend is set on the bend line.

7.7.2.3 Invert auxiliary bend



How do I open this dialog box?

Select: **Modules > Part Editor > Bending sequence > Auxiliary bend**

This function inverts the bend angle of the selected auxiliary bend. After this function has been executed, the auxiliary bend will be bent in the opposite direction.

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7.7.3 Group: Pre-bending

Where do I find the group?

Select: **Modules > Part Editor > Bending sequence**

This group contains functions that can be used to assign additional pre-bends to a bend or remove them again.

7.7.3.1 Insert pre-bend



Where do I find the function?

Select: **Modules > Part Editor > Bending sequence > Pre-bending**

This function is used to create a bend in two bending steps. A pre-bend is made before the actual bend. letting you produce bends that can only be bent in two steps.



Attention! The bend angle must always be bent in the same direction as the final bend.



Tip: Use pre-bending for complex bending parts to avoid collisions during the bending process.

Adding a pre-bend to a bend line

1. Select the bend line to which you wish to add a pre-bend.
2. Select the **Add** function from the **Pre-bending** group.
3. Specify the bend angle for the pre-bend.
4. Press the **Apply** button to set the pre-bend.

Bend angle

This option is used to define the bend angle (**A**) of the pre-bend. At a later stage, the finished bend with the final bend angle (**B**) will be produced in a second step.

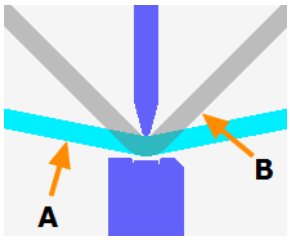


Fig. 7-143: Bend angles of a pre-bend and final-bend

7.7.3.2 Remove pre-bend



Where do I find the function?

Select: **Modules > Part Editor > Bending sequence > Pre-bending**

This function removes the selected pre-bend. This function can only be used when a pre-bend is set on the bend line.

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7.7.4 Group: Rotate

Where do I find the group?

Select: **Modules > Part Editor > Bending sequence**

This group contains functions that can be used to rotate tools or the bending part. Rotation lets you achieve bends that would otherwise be impossible or could only be made with a collision.

7.7.4.1 Rotate top tool



Where do I find the function?

Select: **Modules > Part Editor > Bending sequence > Rotate**

This function rotates the top tool for the current process through 180° in the tool holder. This function can be used for elbow-shaped and asymmetric top tools.

7.7.4.2 Rotate bottom tool



Where do I find the function?

Select: **Modules > Part Editor > Bending sequence > Rotate**

This function rotates the bottom tool for the current process through 180° in the tool holder. This function can be used for asymmetric bottom tools.

Note: This function cannot be used for Hämmerle bottom tools because they cannot be rotated.

7.7.4.3 Rotate part



Where do I find the function?

Select: **Modules > Part Editor > Bending sequence > Rotate**

This function rotates the bending part for the current process through 180° about the vertical axis.

Tip: Use this function, for example, to place the center of gravity of a part in front of or behind the bending tool to make handling easier during the bending process.

7.7.5 2D view

How do I open this dialog box?

Select: **Modules > Part Editor > Bending sequence**

This dialog box displays the developed view of the bending part including all bend lines. The bend lines are shown as dotted or dashed lines. The active bend line is highlighted in red.

The currently selected bend can be previewed in the 3D window. To be able to preview the bend, the **Before bending** button must be disabled in the view bar.



Fig. 7-144: Before bending button disabled in the view bar

Setting the bending sequence in the 2D view

If the **Set** function is selected, the relevant status indicator is displayed for each bend line. Clicking on a bend line will add it to the bending sequence. The bend lines are added to the sequence in the order in which you click on them.

Set bend lines are shown in gray. The position within the bending sequence is indicated on the bend line by numbers together with the relevant status.

Tip: Selecting the **Set** function again will remove the entire bending sequence again.

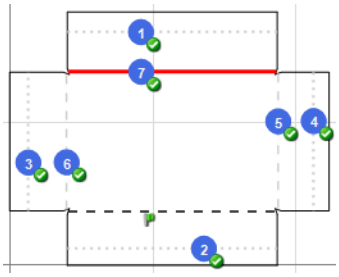


Fig. 7-145: Partially set bending sequence on a part

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7.8 Menu: Tool setup plan

Where do I find the menu?

Select: **Modules > Part Editor**

The tool setup plan for the bending part is created in this menu. A tool setup plan can consist of one or more station plans.

The display in the main window can be changed from the View bar and Navigation bar.

Click and drag to move the part manually on the bending station. Drag the mouse to one of the three points to align the part to left, center or right of the station.

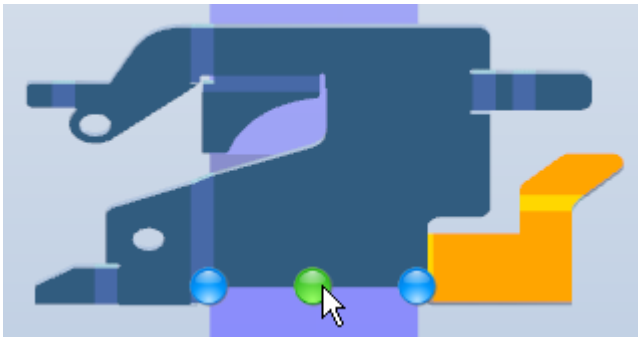


Fig. 7-146: Manually moving a part on the bending station

Functional groups

The function bar contains the following functional groups:

- Tool setup plan
- Station plan
- Station

7.8.1 Group: Tool setup plan

Where do I find the group?

Select: **Modules > Part Editor > Tool setup plan**

This group contains the functions that can be used to create a tool setup plan automatically. A tool setup plan can consist of one or more station plans.

7.8.1.1 Setup plan settings



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Settings**

This function is used to specify the criteria for automatic computation of the tool setup plan.

Minimum distance

This option sets the minimum distance (**A**) between multiple stations. If the bending part collides with another bending station, move the distance of the bending station that the part collides with until the collision is removed.

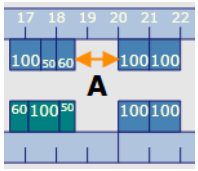


Fig. 7-147: Distance between two bending stations

Grid

This option generates a grid for positioning the bending tools on a station plan. The position always refers to the left-hand position of a bending station.

☞ **Note:** Hämmerle machines have a fixed mechanical positioning grid of 50mm. The grid pitch value for these machines must therefore always be divisible by 50.

Max. shortfall of station length

Maximum length by which a station may be shorter than bend length.

Minimize number of segments

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This option tries to minimize the number of segments on the station after calculating the optimum number of segments on the station.



Note: Disable this option if you do not wish to minimize segments.

7.8.1.2 Set tool setup plan automatically



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Tool setup plan**

This function automatically creates the tool setup plan based on the criteria specified in the **Setup plan settings** dialog box. The creation process involves the following operations:

- Creation of the necessary station plans.
- Creation of stations on the station plans.
- Assignment of bending processes to the stations.

Note: When creating station plans automatically, the 75% rule for 3-point bending is always observed on Hämmerle bending machines. On Beyeler machines or when using air bending on Hämmerle bending machines, you can therefore post-optimize the setup by editing the stations manually.

7.8.1.3 Import tool setup plan



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Tool setup plan**

This function is used to import the tool setup plan from an existing bending part into the current bending part. It imports all segments with the associated tool definition from the selected bending part.

Note: If the existing tool setup plan uses different tools, it is necessary to assign the tools for the imported tool setup plan manually.

Import tool setup plan

1. Select the **Import** function.
2. In the Parts database, select the part whose tool setup plan that you wish to import.
3. Press **Open** to confirm the selection. The imported tool setup plan is displayed in the **Tool setup plan** dialog.

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7.8.1.4 Check tool setup plan



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Tool setup plan**

This function checks whether all the processes have been assigned a bending station. The result of the check is displayed in a message window.

7.8.2 Group: Station plan

Where do I find the group?

Select: **Modules > Tool Setup Optimizer**

This tab displays the station plan of the selected bending part with the related information.

7.8.2.1 Insert station plan



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Station plan**

This function adds a new station plan to the tool setup plan. The two blue arrow buttons in the **Tool setup plan** dialog box can be used to switch between multiple station plans.

Using different tools on one station plan

Attention! When using different tools on one station plan there is a collision risk.

Different tools can be used on the same station plan under the following provisos:

- They must be of the same height.
- They must be of the same type (e.g. bending tool or hemming tool).

7.8.2.2 Remove station plan



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Station plan**

This function removes the current station plan from the tool setup plan. If the tool setup plan only contains one station plan, this function is disabled.

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7.8.2.3 Center station plan



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Station plan**

This function centers the selected station on the current station plan.

- **All**

  This option centers all stations on the current station plan.



Tip: Use this option if you have chosen a new machine with a different machine length.

7.8.3 Group: Station

Where do I find the group?

Select: **Modules > Part Editor > Tool setup plan**

This group contains the functions that can be used to add bending stations automatically or manually to a station plan and to edit these bending stations. You can assign and/or edit the respective bending processes for each bending station.

7.8.3.1 Insert Station



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Station**

This function adds a new station to the station plan.

If the station plan is empty, the new station is added at the left-hand end of the station plan.

- If no station is selected, the new station is added to the right of the last station on the right, at the preset station separation.
- If a station is selected, the new station is inserted to the right of the selected station, at the preset station separation. If the gap to the next station is not large enough to fit the preset station separation, no station is inserted.

7.8.3.2 Remove station



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Station**

This function removes the selected station from the station plan.

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7.8.3.3 Edit station



How do I open this dialog box?

Select: **Modules > Part Editor > Tool setup plan > Station**

This function can be used to make manual changes to the position, length and segmentation of the selected station. The position of the top tool or bottom tool can be defined jointly or separately.

The length of segments can be changed in the **Segmentation** area. Right-clicking on a segment opens a pop-up menu containing all length segments available for the tool.

☞ **Note:** On Hämmerle bending machines, tools with a segment length of 55-95mm or horn tools can only be used at the beginning or end of a bending station. If you attempt to set one of these segments in the inside area of a bending station, the following error message appears: **Interleaving of segments invalid.**.

Setting a new length and position for a station

1. Select the station.
2. Select the **Edit** function.
3. Enable the **Synchronous position** option in order to assign the changes automatically to the bottom station as well.
4. Use the **Left-hand position** option to move the left-hand side of the station to the position you require on the station plan.
5. Use the  button to add new segments or open the pop-up menu to select a new length for an existing segment until you achieve the station length you require.

Synchronous position

If this option is enabled, any change made to the position of a top tool or bottom tool is applied synchronously to the other tool in each case.

Left-hand position/Right-hand position

This option is used to define the position of the station on the station plan. The distance dimension entered refers to the left-hand/right-hand side of the station.

The icon after the input box for **Left-hand position** centers the top station on the bottom one.

Note: On Hämmerle bending machines, the segmentation of the tool holder means that the position can only be set in steps of 50mm.

Length

This option shows the total length of the station. The length is calculated from the sum of the individual segments.

Segment automatically

This function automatically creates a new bending station for the current bending process. This takes into account station segmentation settings and the use of horn tools in the drop-down menu.

Drop-down menu for **Segment automatically** settings.

Segmentation

Defines the segmentation strategy.

- **Do not exceed length**

  The maximum length of the bending station is the bend length of the current bending process.

  Note: Bend length is displayed in parentheses.

- **Minimize number of segments**

  The bending station is created with as few segments as possible.

Horn tool

Defines the strategy for using horn tools.

- **None**

  No horn tool is used.

- **Left**

  A horn tool is used on the left side of the bending station.

- **Right**

  A horn tool is used on the right side of the bending station.

- **On both sides**

  A horn tool is used on both sides of the bending station.

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Tool

Provided that more than one tool can be selected, this option lets you change the tool for the bending process.

Rotated



Tip: This option defines the bending tool as rotated. This option is only available if in the **Bending sequence** menu, the top or bottom tool has been rotated for one or more bending processes.

If a bending station has this option assigned, those bending processes that are bent on this station with a rotated tool must be assigned using the **Assign process** function.



Note: If the tools are rotated for all the bending processes, this option is displayed as selected for all processes, but is disabled.

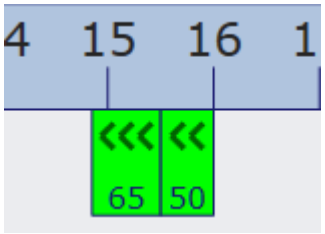


Fig. 7-148: Display a rotated top tool in the station plan

Check segmentation

This option checks interleaving of the bending tools on a 3-point bending machine. Only tools with a grid of 50mm can be used in tool selection.

Exception:

if a station with a horn tool is inserted the wrong way round, the left side of the tool is not on the 50mm grid. The segmentation check can be disabled for this exception.



Note: This option is only available for **Hämmerle** machines.

Segmentation

This area shows all the tool segments used in the station. The length or type of a segment can be changed in the pop-up menu by right-clicking on a segment.

Segment types

- Standard segment (**A**)
- Segment with left/right horn (**B**)
- Station with empty segment (**C**)

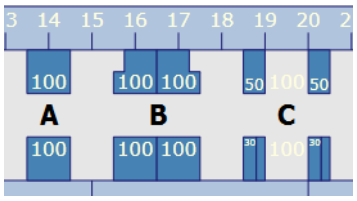


Fig. 7-149: Segment types on a station plan

Buttons

- = Adds a new segment to the station.
- = Removes the selected segment from the station.
- = Moves the selected segment one position down the order.
- = Moves the selected segment one position up the order.
- = Reverses the order of the tool segmentation for the station.

Tip: You can also move segments using Drag&Drop.

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7.8.3.4 Move to previous station



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Station**

This function moves the selected station to the previous station plan.

☞ **Note:** This function is only enabled if the current station plan is number two or higher.

7.8.3.5 Move to next station



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Station**

This function moves the selected station to the next station plan.

☞ **Note:** The function is only enabled if there is another station plan after the current station plan.

7.8.4 Group: Assignment

Where do I find the group?

Select: **Modules > Part Editor > Tool setup plan**

This group contains functions that can be used to manually/automatically add, edit or delete bending sequence assignments to the bending stations.

You can assign and/or edit the respective bending processes for each bending station.

7.8.4.1 Automatically assign process



Where do I find the function?

Select: **Modules > Part Editor > Tool setup plan > Assignment**

This function automatically assigns as yet unassigned bending processes to the bending stations.



Note: This function can only be used if at least one bending process has not been assigned to a station.

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7.8.4.2 Assign process manually



How do I open this dialog box?

Select: **Modules > Part Editor > Tool setup plan > Assignment**

In this dialog box you can assign new bending processes to a bending station or change the assignment of bending processes to a bending station.

The **Synchronize station selection** option automatically selects the bending station in the **Tool setup plan** dialog box which the selected bending process will use. Use this option as a visual check of the assigned bending stations.



Tip: The simplest way to assign processes to the bending stations is by Drag&Drop. Using **Ctrl** or **Shift** + left mouse button you can select several processes at once.

Assigned

How do I open this tab?

Select: **Modules > Part Editor > Tool setup plan > Station > Assign process**

This tab shows all the bending processes that have already been assigned to a bending station.



Note: The processes can be assigned to another bending station.

Assigning a process to another bending station.

- Select the **Station** column and click on the arrow.
- Drag the process onto another bending station in the **Station plan** section.

Offset

In this area you can define the position of the part on the bending station. Specifying different positions is used to avoid collisions when computing the bends.

Attention! If the position on the bending station is changed when backgauges have already been set, you must run the **Reposition axes** function from the **Backgauges** menu for the process. This will prevent incorrect positioning of the bending part for rounded edges and snapped-on corners.

Note: These settings have no impact on the machine when backgauges run parallel to the bend line.

You have the following options for positioning a bending side on the station:

- **Automatic:** *Automatically positions the side in the optimum position on the station.*
- **Left:** *Moves the side to the left-hand end of the station. Use this option to prevent tongues colliding with the station during bending.*
- **Centered:** *Centers the side in the station. Use this option as the standard option for air bending.*
- **Right:** *Moves the side to the right-hand end of the station. Use this option to prevent tongues colliding with the station during bending.*
- **User-defined:** *Moves the side to a user-definable position. Use this option on stations containing empty segments to adjust the work piece position to suit the station or for 3-point bending in order to comply with the 75% rule.*
- **Process:** *Assigns the selected position only to the current process.*
- **Station:** *Assigns the selected position to all processes on the station.*

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What each column means:

Bend line

Displays the number and status of the bend line.

-  Bending process can be performed on the station.
-  It may not be possible to perform the bending process on the station. Run the Check function to check the process.

Process

A serial number indicates the sequence of the bending processes.

Plan

Displays the number of the station plan on which the process will be bent.

Bend length

Shows the length of the side.

Station

Displays the number of the bending station on which the process will be bent.

Offset

Displays the position of the bending part with respect to the left-hand side of the bending station.

- **Positive value:** Side is inside the bending station.
- **Negative value:** Side overlaps with the bending station.

Pro Station

How do I open this tab?

Select: **Modules > Part Editor > Tool setup plan > Station > Assign process**

This tab shows all assigned bending processes for the selected station in the **Station plan** section.

Note: The processes can be assigned to another bending station.

Assigning a process to another bending station.

- Select the **Station** column and click on the arrow.
- Drag the process onto another bending station in the **Station plan** section.

Offset

In this area you can define the position of the part on the bending station. Specifying different positions is used to avoid collisions when computing the bends.

Attention! If the position on the bending station is changed when backgauges have already been set, you must run the **Reposition axes** function from the **Backgauges** menu for the process. This will prevent incorrect positioning of the bending part for rounded edges and snapped-on corners.

Note: These settings have no impact on the machine when backgauges run parallel to the bend line.

You have the following options for positioning a bending side on the station:

- **Automatic:** *Automatically positions the side in the optimum position on the station.*
- **Left:** *Moves the side to the left-hand end of the station. Use this option to prevent tongues colliding with the station during bending.*
- **Centered:** *Centers the side in the station. Use this option as the standard option for air bending.*
- **Right:** *Moves the side to the right-hand end of the station. Use this option to prevent tongues colliding with the station during bending.*
- **User-defined:** *Moves the side to a user-definable position. Use this option on stations containing empty segments to adjust the work piece position to suit the station or for 3-point bending in order to comply with the 75% rule.*

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- **Process:** *Assigns the selected position only to the current process.*
- **Station:** *Assigns the selected position to all processes on the station.*

What each column means:

Bend line

Displays the number and status of the bend line.

- Bending process can be performed on the station.
- It may not be possible to perform the bending process on the station. Run the Check function to check the process.

Process

A serial number indicates the sequence of the bending processes.

Plan

Displays the number of the station plan on which the process will be bent.

Bend length

Shows the length of the side.

Station

Displays the number of the bending station on which the process will be bent.

Offset

Displays the position of the bending part with respect to the left-hand side of the bending station.

- **Positive value:** Side is inside the bending station.
- **Negative value:** Side overlaps with the bending station.

Not assigned

How do I open this tab?

Select: **Modules > Part Editor > Tool setup plan > Station > Assign process**

This tab lists all the bending processes that have not been assigned to a bending station yet.

Assigning a process to a bending station:

- Select the **Station** column and click on the arrow.
- Drag the process onto another bending station in the **Station plan** section.

What each column means:

Bend line

Displays the number and status of the bend line.

Process

A serial number indicates the sequence of the bending processes.

Top tool

Shows the top tool being used.

Bottom tool

Shows the bottom tool being used.

Bend length

Shows the length of the side.

Station

In this column you can click the arrow to assign a bending station to the process.

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7.8.4.3 Remove process assignment



How do I open this dialog box?

Select: **Modules > Part Editor > Tool setup plan > Station**

This function removes the current process from the station selected in the **Tool setup plan** dialog box. The removed process is moved to the **Not assigned** tab in the **Assign process** docking dialog box.

7.8.4.4 Remove station's process assignment



How do I open this dialog box?

Select: **Modules > Part Editor > Tool setup plan > Station**

This function removes all assigned processes from the station selected in the Tool setup plan dialog box. The removed processes are moved to the **Not assigned** tab in the **Assign process** docking dialog box.

7.8.4.5 Clear errors



How do I open this dialog box?

Select: **Modules > Part Editor > Tool setup plan > Station**

This function is used to clear errors in the setup plan.

☞ **Note:** This function is only enabled if the pending error can be cleared.

7.8.4.6 Clear warning



How do I open this dialog box?

Select: **Modules > Part Editor > Tool setup plan > Station**

This function acknowledges and clears the warning for the selected bending process.

7.8.5 Station plan

How do I open this dialog box?

Select: **Modules > Part Editor > Tool setup plan**

This dialog box displays the station plan and all the stations that are present. If more than one station plan is being used, you can switch between the various station plans using the two arrow keys. The 3D view always shows the station currently highlighted.

For each station, the station number is displayed between the top and bottom tool.

You can drag&drop stations to move them. If the **Synchronous** option is enabled in the **Edit Station** dialog, the bottom tool is moved in synch with the top tool. Otherwise, you can move the tools separately.

Note: On Hämmerle bending machines, the grid pitch of the tool holder is taken into account when moving.

Tip:

- Double-clicking on the station opens the **Edit station** dialog box in which you can make changes to the station.
- When the mouse is moved over a station, a pop-up window is displayed. This window displays the tool, the length and the left position of the top and bottom stations of the station respectively.
- Click and drag to move the bending station on the station plan.

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7.9 Menu: Backgauges

Where do I find the menu?

Select: **Modules > Part Editor**

In this menu you create the backgauges for all the bending processes. Backgauges are needed for precise positioning of the bending part prior to making a bend. The backgauges can take the form of a right and left backgauge finger minimum.

Tip: The backgauges can be moved manually using the mouse.

The display in the main window can be changed from the View bar and Navigation bar.

Functional groups

The function bar contains the following functional groups:

- Backgauges

7.9.1 Group: Backgauges

Where do I find the menu?

Select: **Modules > Part Editor > Backgauges**

This group contains all the functions that can be used to assign the backgauges to the respective bending process automatically or manually. You can run a check function to check that the backgauges are working correctly.

7.9.1.1 Set backgauges automatically



Where do I find the function?

Select: **Modules > Part Editor > Backgauges > Backgauges**

This function positions the backgauges automatically on the bending edges of the individual processes, based on the criteria set in the **Backgauge settings** dialog box.

7.9.1.2 Backgauge settings



How do I open this dialog box?

Select: **Modules > Part Editor > Backgauges > Backgauges**

This function is used to specify the criteria for automatic computation of the required backgauges for all bending processes.

Minimum edge length

This option sets the minimum length of an edge for which a backgauge can be selected. A default length of 10 mm is set. Only those numbers of the contact edges that meet this minimum length can be selected using the **Contact edge** option in the **Edit backgauges** menu.

Retraction limit

This setting defines the lower limit as of which a retraction is performed. A retraction of less than this value is not performed and is set to **0**.

Application:

This setting permits small collisions without a retraction. This is often used in thick sheets. The default minimum retraction is 0.2 mm *(See: ›Retraction‹ Page 607)*.

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Auto-snap

This option enables automatic snapping of existing backgauges.

Note: This option is only possible if a backgauge finger for corner stops is fitted.

Use one backgauge only

This option is used for automatic calculation of backgauges for the bending part with only one backgauge.

Tip: This avoids calculating a non-ideal backgauge configuration for small parts.

Optimize

Optimization reduces backgauge axis movements to a minimum. This increases the efficiency of bending processes.

- **Z position**

  The program tries to use the same backgauge positions in the Z direction.

- **Position window**

  Specifies the maximum distance of the axis in the Z direction in which the backgauges may be automatically moved.

- **Final bend**

  For bending processes that do not use any backgauges, the X position of the previous bending process is used.

  Bends without backgauges:

  - Auxiliary bend
  - Pre-bending
  - Hemming

- **Parked position**

  This option prevents the unused backgauge from moving to Z parked position on short backgauge edges. The program tries to use the Z backgauge position from the previous bending process. If this is not possible, the unused backgauge is parked immediately alongside the used backgauge.

7.9.1.3 Complete all backgauge settings



Where do I find the function?

Select: **Modules > Part Editor > Backgauges > Backgauges**

This function positions the backgauges automatically on the bending edges of the individual processes, based on the criteria set in the **Backgauge settings** dialog box.



Note: Backgauge positions that have already been created for the respective processes are retained unchanged.

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7.9.1.4 Check backgauges



Where do I find the function?

Select: **Modules > Part Editor > Backgauges > Backgauges**

This function checks the backgauges for collisions with the backgauges themselves, with the bending part and with the machine. It also checks whether valid tools and tool setup plans are present.

Note: The check function only needs to be used if processes have the error status or not yet checked status .

7.9.1.5 Edit backgauges



How do I open this dialog box?

Select: **Modules > Part Editor > Backgauges > Backgauges**

In this dialog box you can change the position of the backgauges or set a new backgauge for the selected bending process.

Invalid combinations of backgauge positions are identified by an error symbol .

There are two basic types of backgauge fingers:

- **Standard backgauge finger**

  Cannot be used for corner stops. On some machines it is possible to combine it with the backgauge finger for corner stops.

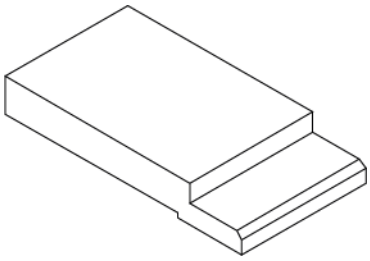


Fig. 7-150: Sketch showing the standard backgauge finger

Backgauge finger for corner stops

Can be used for all backgauges (optional).

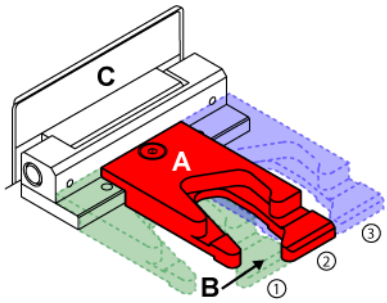


Fig. 7-151: Sketch showing backgauge finger for corner stops

The following explanations apply to the backgauge finger for corner stops, because this is the type most often used. All the information also applies to the standard backgauge finger, except when dealing with a corner stop.

All backgauges

The options in this section can be assigned at the same time to all backgauges of the selected bending process.

Park: all

This option moves all active backgauges to parked position.

Contact edge: select

This option assigns all backgauges are assigned to the selected contact edge.

Reference point: select

This option defines the reference point for all backgauges.

Snap: all

This option selects the **Snap** option for all backgauges.

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Backgauge 1/2

This section is used to configure all options of the appropriate backgauge finger for the currently selected process.

The backgauge finger (**A**) can be arranged on the backgauge arm (**C**) in three different positions. The opening between the backgauge fingers (**B**) can also be changed from left to right side by changing the fingers on the backgauge arms.

Tip: If you are working on bending parts whose contact edges lie close together, it is best to fit the backgauge fingers in the inner position (**1**) with the opening facing outwards.

Symbols for backgauge types in the workspace

Symbol	Description
	Snap is enabled. The backgauge snaps into a corner on the part correctly.
	Snap is disabled. The backgauge is on a straight edge on the part.
	Snap is disabled. The backgauge is on an inclined edge on the part.
	This icon flags the backgauge as a master backgauge. The master backgauge determines the position of the R axis for all backgauges present. **Note:** The master backgauge is only used for backgauges with a common R axis.

Park

This option is used to move the particular backgauge arm into the parked position.

If this **Parked** option is enabled, all other options for the backgauge finger are disabled except the **Contact edge** option.

Note: If the **Parked** option is enabled and a different contact edge is selected for this backgauge finger, the **Parked** option is disabled again and the backgauge finger set at the selected contact edge.

Snap

With this option, the backgauge finger is snapped onto the relevant corner of the contact edge. At least 3 stop points (**A**) are needed to fix the part precisely.

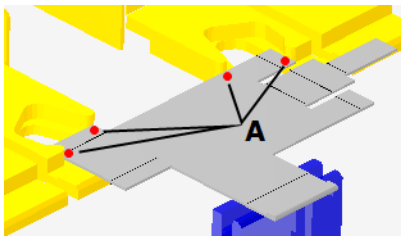
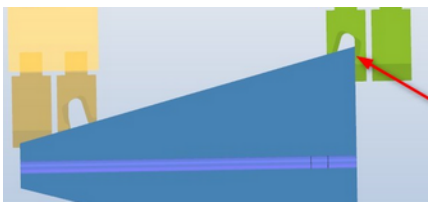


Fig. 7-152: Stop points when snapping onto corners

Note: This option is only possible if a backgauge finger for corner stops is fitted.

Tip:

- Choose this option for rounded edges or corner stops.
- If it is not possible to provide a corner stop because of the current position of the backgauge finger, it may still be possible to provide a stop by changing the backgauge fingers on the backgauge arms.
- If the selected backgauge edge is not parallel to the bend line, try to snap the backgauge that is located further back in the X direction.



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Fig. 7-153: Snapping a backgauge if not parallel to bend line

Contact edge

The **Select** button is used to determine the contact edge for the current bending process. All valid contact edges are marked on the bending part (**A**). Only valid contact edges can be selected.

Press the **Select unsafe** button to select any edge of the part as a contact edge. This option is only intended for borderline cases where it is difficult to find a suitable contact edge.

Note: Selecting this option zooms in on the contact edges.

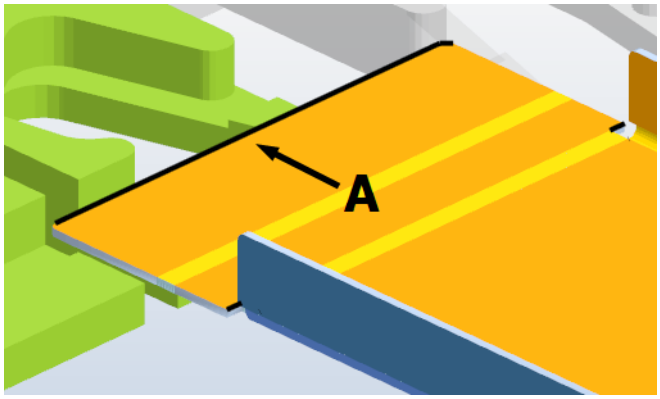


Fig. 7-154: Contact edge on bending part

Reference point

This option defines which element of the backgauge finger will rest against the contact edge of the bending part. Each reference point on the backgauge finger is marked by a sphere.

Note: Selecting this option hides the part and zooms in on the active backgauge finger.

Tip: To make a quick selection, use the drop-down menu next to the **Select** button to choose the number of the reference point.

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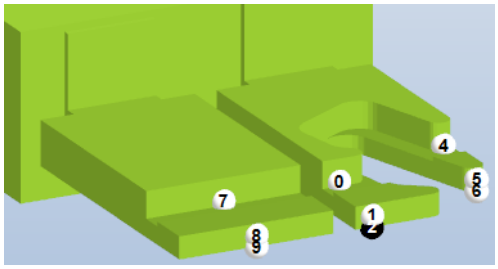


Fig. 7-155: Reference points on the backgauge finger for corner stops

Sensor

Use this option to enable or disable the sensor on the backgauge finger. The option is only enabled if there is a sensor on the backgauge finger.

The sensor detects whether the sheet is touching the backgauge finger correctly. If the sheet is not touching the backgauge finger correctly when the sensor is on, the following bending process cannot be performed.

Retraction

This option specifies the distance by which the backgauge finger is retracted from the contact edge before the bend is made. Retractions are used in order to prevent collisions between bending part and backgauge finger during the bending process.

Retractions must be set if an already bent part, whose bent side is pointing down (**B**), shall be pushed against the backgauge finger (**1**).

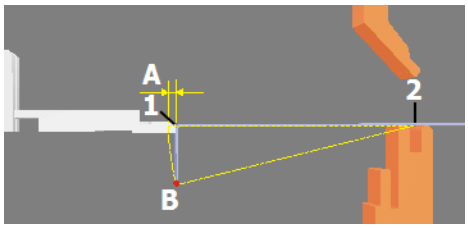


Fig. 7-156: Using a retraction (**A**) for a backgauge when the part has a side bent downwards

A	Minimum distance for backgauge finger retraction
B	Distance to end point of bent side
1	Contact edge
2	Pivot point of the side (**B**) during bending

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Z-retraction

This option defines the extension dimension of the side backgauge finger.

R position

This option defines the position of the backgauge finger on the R axis (**A**). The value defines the vertical position of the backgauge finger.

☞ **Note:** If the backgauge fingers are located on a common R axis, this value applies to all backgauge fingers. The backgauge finger for which the value applies is marked by an asterisk.

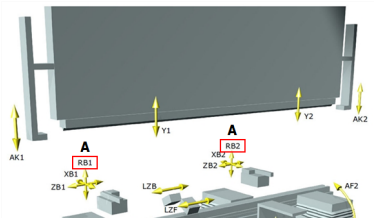


Fig. 7-157: R axes of backgauges

Z position

This option defines the position of the backgauge finger against the contact edge of the bending part. There are two ways of positioning the backgauge finger:

- By entering the position in the **Z position** field.
- By moving the backgauge finger in 3D view.

Use the two arrow buttons to move to the minimum or maximum Z position of the axis.

7.9.1.6 Reposition axes



How do I open this dialog box?

Select: **Modules > Part Editor > Backgauges > Backgauges**

This option recalculates the positions of the backgauge axes for all processes, and repositions the backgauges as necessary.

Use this option when the following changes have been made:

- If the position of a bending station has been changed after being set.
- If the position of a bending part on a bending station has been changed after being set.

7.9.1.7 Optimize backgauges



Where do I find the function?

Select: **Modules > Part Editor > Backgauges**

This function attempts to reduce axis movement of the backgauges between processes. Optimization is performed based on the settings in the **Optimize settings** dialog.

Note: The option is only enabled if all backgauges have a valid status .

Optimize backgauge settings

How do I open this dialog box?

Select: **Modules > Part Editor > Backgauges**

This dialog is used to configure the optimization settings for backgauges.

Optimization reduces backgauge axis movements to a minimum. This increases the efficiency of bending processes.

The **Optimize** function assigns the settings to all backgauges of the bending processes.

Note: Running the **Automatic** function will also apply the settings to the bending processes.

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Optimize

- **Z position**

  The program tries to use the same backgauge positions in the Z direction.

- **Position window**

  Specifies the maximum distance of the axis in the Z direction in which the backgauges may be automatically moved.

- **Final bend**

  For bending processes that do not use any backgauges, the X position of the previous bending process is used.

  Bends without backgauges:

  - Auxiliary bend
  - Pre-bending
  - Hemming

- **Parked position**

  This option prevents the unused backgauge from moving to Z parked position on short backgauge edges. The program tries to use the Z backgauge position from the previous bending process. If this is not possible, the unused backgauge is parked immediately alongside the used backgauge.

7.9.1.8 Clear errors



How do I open this dialog box?

Select: **Modules > Part Editor > Backgauges > Backgauges**

This function is used to clear backgauge errors.

☞ **Note:** This function is only enabled if the pending error can be cleared.

7.9.2 Backgauge processes

How do I open this dialog box?

Select: **Modules > Part Editor > Backgauges**

This dialog box contains a table displaying the status of the backgauges for each of the bending processes. The currently selected bending process is shown in the 3D view.

Note: Use the CTRL or shift key to select multiple bending processes.

Tip: Double-clicking on a bending process opens the **Edit backgauges** dialog box in which you can edit the backgauge for the selected bending process.

Positioning backgauges using the foot pedal

If the backgauges will be moved forwards towards the operator for the next bending process, the operator must press the foot pedal to confirm the repositioning of the backgauges when the new position lies more than a certain distance away. The backgauges will then move to the relevant position.

Confirmation safeguards the operator from being hit by the bending aid during automatic positioning of bending parts with long sides in the rear machine area.

Note: The minimum distance at and above which confirmation using the foot pedal is required is defined in the bending database.

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What each column means:

Status

Displays the status of the backgauges in the bending sequence.

Symbol	Description
	A valid backgauge is assigned to the bending process.
	A valid backgauge has not yet been assigned to the bending process.
	The backgauge for the bending process has not been checked yet. Run the **Check** function.

Process

A serial number indicates the sequence of the bending processes.

Angle

Shows the angle of the bend. An angle of **0** is only used for the final bend in a hemming operation.

Bending operation

Shows the type of bend in the bending process.

- Normal bending
- Hemming
- Pre-bending
- Auxiliary bend
- Stepped radius bend

Bend line

Shows the number of the bend line on the part.

Bend length

Shows the length of the bend line on the part.

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7.10 Menu: Bending aids

Where do I find the menu?

Select: **Modules > Part Editor**

This menu is displayed if the selected bending machine is fitted with one or more bending aids.

The display in the main window can be changed from the View bar and Navigation bar.

Placing bending aids using the mouse

The bending aids can be positioned for each bending process using the mouse. If the bending aid is placed under a side, it automatically snaps to the underside.

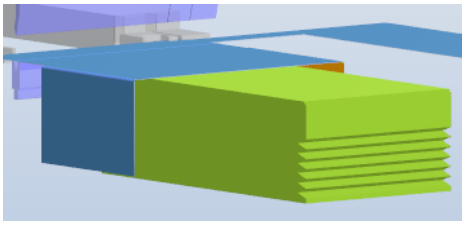


Fig. 7-158: Bending aid snapped under a side

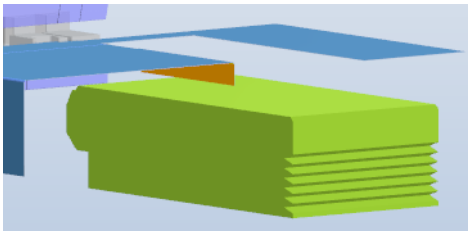


Fig. 7-159: Bending aid snapped to the edge of a side

If it is not possible to create the height with the bending aid, or if there is a collision between the side and the bending aid, an error message is output.

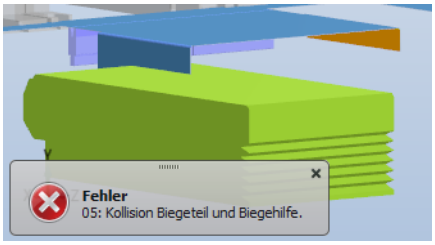


Fig. 7-160: Bending aid in collision with a side

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Tip: Use bending aids to support large, bulky or heavy sheets during the bending process.

Functional groups

The function bar contains the following functional groups:

- Bending aids

7.10.1 Group: Bending aids

Where do I find the group?

Select: **Modules > Part Editor > Bending aids**

This group contains all the functions that can be used to assign the bending aids to the respective bending process automatically or manually. You can run a check function to check that the bending aids are working correctly.

7.10.1.1 Bending aids settings



How do I open this dialog box?

Select: **Modules > Part Editor > Bending aids > Bending aids**

This function is used to specify the criteria for automatic computation of the required bending aids.

Use

The selected attribute is applied to all available bending aids.

- **Positioned**

  This attribute is used to position all available bending aids horizontally, if this is technically possible from a bending viewpoint. During the bending process, the bending aid **does not** perform a supporting tilting movement in parallel with the bending process.

  Note: The front bending aids must be positioned manually on the machine for safety reasons.

  Tip: Use this attribute when you want to use the bending aid as a sheet rest.

- **Parked**

  This attribute is used to move all available bending aids to the parked position. The bending aid remains in the parked position for the current process until the **Park** attribute is manually disabled using the **Edit** function from the **Bending aids** menu, or until another use has been selected.

- **Enabled**

  This attribute is used to assign a bending aid automatically to all bending processes, if this is technically possible from a bending viewpoint. All bending aids

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fitted are included. During the bending process, the bending aid **does** perform a supporting tilting movement in parallel with the bending process.

Optimize

- **W position**

  The program tries to use the same positions of the bending aids in the Z direction.

- **Parked position**

  This option prevents unused bending aids from moving to Z parked position. The program tries to use the position from the previous bending process for unused bending aids. If this is not possible, the unused bending aid is parked immediately alongside the used bending aid.

> **Note:** This option can only be used for rear bending aids.

7.10.1.2 Set bending aids automatically



Where do I find the function?

Select: **Modules > Part Editor > Bending aids > Bending aids**

This function positions the bending aids automatically on the bending edges of the individual processes, based on the criteria set in the **Bending aids settings** dialog box.

7.10.1.3 Complete all bending aid settings



Where do I find the function?

Select: **Modules > Part Editor > Bending aids > Bending aids**

This function applies and positions the bending aids automatically to all bending processes, based on the criteria set in the **Bending aids settings** dialog box.

> **Note:** Bending aids that have already been created for the respective processes are retained unchanged.

7.10.1.4 Check bending aids



Where do I find the function?

Select: **Modules > Part Editor > Bending aids > Bending aids**

This function checks the bending aids for collisions with the bending aids themselves, with the bending part and with the machine.

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☞ **Note:** The check function only needs to be used if processes have the error status or not yet checked status .

7.10.1.5 Edit bending aids



How do I open this dialog box?

Select: **Modules > Part Editor > Bending aids > Bending aids**

In this dialog box you can change the position of the bending aids or set a new bending aid for the selected bending process. Invalid settings are identified by an error symbol  .

The options are displayed in a dedicated area for each bending aid.

There are two types of bending aid:

- **Rear bending aids:** *Can be moved automatically in the W-axis along the tool holder.*
- **Front bending aids:** *For safety reasons cannot be moved automatically in the W-axis along the tool holder. The operator must adjust the position manually for the bending station concerned.*

Front / Rear

In this area you can configure the options separately for each front and rear bending aid.

Park

This option is used to move the particular bending aid into the parked position. If this option is enabled, all other options are disabled.

Activate

This option enables the particular bending aid for use in the selected bending process.

H position

This option is used to manually adjust the height of the bending aid to the respective bending process.



Tip: Use this option to optimize their height of bending aids if the automatic mechanism cannot find a good solution.

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W position

This option sets the distance of the reference point of the respective bending to the left outside edge of the press table. There are two ways of positioning:

- By entering the position in the **W position** field.
- By moving the slide control.

Synchronization for active bending aids

In this area you can configure the synchronized movement of the active bending aid with the ram during the backwards movement after making the bend.

Application:

- To prevent heavy or large bending parts slipping under the bottom tool in the rear area of the machine after bending.
- To prevent the bending part dropping onto the backgauge fingers when rear bending aids are in use.

Up to angle (α)

This option specifies the angle **α** for synchronous movement of the bending aids (**A**) back to their initial position after the bending process; the bending aids are moved synchronously with the ram at reduced speed until they reach this angle.

☞ **Note:** If the maximum angle is entered in the input field, synchronized movement is **not** performed. The bending aid and the ram move back at normal speed to their initial position.

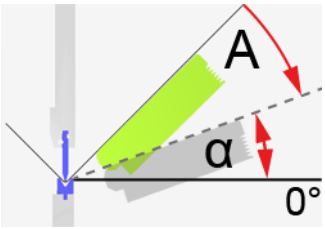


Fig. 7-161: Synchronized return movement of the bending aid (A) as far as angle α.

7.10.1.6 Reposition axes



Where do I find the function?

Select: **Modules > Part Editor > Bending aids > Bending aids**

This option recalculates the positions of all active bending aids for all processes, and repositions the bending aids as necessary.



Note: Positioning of the front bending aids on the machine must be done solely by the operator.

Use this option when the following changes have been made:

- If the position of a bending station has been changed after being set.
- If the position of a bending part on a bending station has been changed after being set.

7.10.1.7 Optimize bending aids



Where do I find the function?

Select: **Modules > Part Editor > Bending aids**

This function attempts to reduce axis movement of the backgauges bending aids between processes. Optimization is performed based on the settings in the **Optimize settings** dialog.



Note: The option is only enabled if all bending aids have a valid status  .

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Optimize bending aids settings

How do I open this dialog box?

Select: **Modules > Part Editor > Bending aids > Bending aids**

This function is used to specify the criteria for automatic calculation of the bending aids.

Optimize

- **W position**

  The program tries to use the same positions of the bending aids in the Z direction.

- **Parked position**

  This option prevents unused bending aids from moving to Z parked position. The program tries to use the position from the previous bending process for unused bending aids. If this is not possible, the unused bending aid is parked immediately alongside the used bending aid.

  ☞ **Note:** This option can only be used for rear bending aids.

7.10.2 Bending aids processes

Where do I find the group?

Select: **Modules > Part Editor > Bending aids**

This dialog box contains a table displaying the status of the bending aids for each of the bending processes. The currently selected bending process is displayed in the 3D view.



Tip: Double-clicking on a bending aid opens the **Bending aids settings** dialog box in which you can edit the bending aid for the selected bending process.

What each column means:

Status

Displays the status of the bending aids in the bending sequence.

Symbol	Description
	A valid bending aid is assigned to the bending process.
	A valid bending aid has not yet been assigned to the bending process.
	The bending aid for the bending process has not been checked yet. Run the **Check** function.

Process

A serial number indicates the sequence of the bending processes.

Angle

Shows the angle of the bend. An angle of **0** is only used for the final bend in a hemming operation.

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Bending operation

Shows the type of bend in the bending process.

- Normal bending
- Hemming
- Pre-bending
- Auxiliary bend
- Stepped radius bend

Synchronous up to angle

Displays the angle for synchronous movement of the bending aids back to their initial position after the bending process; the bending aids are moved synchronously with the ram at reduced speed until they reach this reference angle. *(See: › Synchronization for active bending aids‹ Page 620)*.

Front side active

Shows whether the bending aid is activated on the front side for this process.

Rear side active

Shows whether the bending aid is activated on the rear side for this process.

7.11 Menu: Compensation

Where do I find the menu?

Select: **Modules > Part Editor**

Compensation offsets fluctuations in sheet thickness and beam deformation during the bending process. For this purpose, a reference measurement is performed before bending. The difference between nominal and actual value is converted into a correction of the target Y position.



Tip: It is usually enough to run compensation during the part's first process and to apply the corrections to the following processes.

This menu is only available on the following machines if the appropriate option is enabled in Administrator:

- **Xpert:**
- **Xact:**

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7.11.1 Group: Compensation

Where do I find the group?

Select: **Modules > Part Editor > Compensation**

This group contains all the functions that can be used to edit compensation.

7.11.1.1 Compensation settings



Where do I find the function?

Select: **Modules > Part Editor > Compensation > Compensation**

This dialog box is where the settings are defined for automatic assignment of compensation.

Mode

- **Off**
  Sheet thickness measurement is switched off.
- **On**
  Sheet thickness measurement is switched on for all processes.
- **Adopt**
  Sheet thickness measurement is switched on for the first process. All following processes are set to **Accept** mode.

Note: Select the **Edit** function to determine the compensation of individual processes.

7.11.1.2 Automatic



Where do I find the function?

Select: **Modules > Part Editor > Compensation > Compensation**

This function automatically sets compensation to all of the part's bending processes based on the settings.

7.11.1.3 Compensation check



Where do I find the function?

Select: **Modules > Part Editor > Compensation > Compensation**

This functions checks whether the set compensations are valid for the processes.



Tip: Incorrect compensations can be corrected using the **Automatic** or the **Edit** function.

7.11.1.4 Edit compensation



How do I open this dialog box?

Select: **Modules > Part Editor > Compensation > Compensation**

In this dialog you can configure compensation settings for each bending process.

- **Advantage:** Increases accuracy for the respective bending process.
- **Disadvantage:** May increase part bending time significantly.



Note: Automatically assigning compensation overwrites these settings.

These options are only available on the following machines if the appropriate option is enabled in Administrator:

- **Xpert:** *Crowning can be performed dynamically and statically.*
- **Xact:** *Crowning can only be performed statically.*

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Sheet thickness compensation

The settings in this section define how sheet thickness fluctuation is compensated.

Mode

- **Off**
  Sheet thickness compensation is disabled for the selected process.
- **On**
  Sheet thickness compensation is enabled for the selected process.
- **Adopt**
  Sheet thickness compensation accepts the value measured by the process as indicated in the **Adopt for process** option.

Adopt for process

Determines the process from which the measured compensation is adopted for the selected process.



Note: Only processes that have the mode set to **On** may be selected.

Ram action in clamping point

- **Default**
  This is the default setting for ram behavior.

  If a retraction is programmed, the ram stops at the clamping point and retracts the backgauges before bending.



Attention! If a retraction is programmed, sheet thickness measurement for the selected process may fail! In this case, use the **Never pause** option.

- **Never pause**

  If a retraction is programmed, the ram does not stop at the clamping point to ensure sheet thickness measurement for compensation. The retraction is nevertheless performed!

⚠CAUTION

Risk of collision!

Use the **never** option for curved sides (**A**) underneath the backgauge finger (**B**). The side may collide with the backgauge finger and pull the part back or bend it.

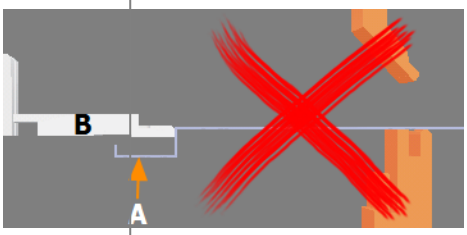


Fig. 7-162: Curved sides underneath the backgauge finger

Crowning

Mode

- **Dynamic crowning:** Sets crowning to **dynamic** for the selected process.
  - The CNC sets pressure automatically.
  - The greater the pressing force, the more pressure is built up for crowning.
  - Crowning pressure is present from the clamping point up to the end of the bend.
- **Static crowning:** Sets crowning to **static** for the selected process.
  - Pressure is calculated and set based on pressing force.
  - Pressure is already active when the ram leaves top dead center and remains until the end of the bend.

⚠ **Attention!** Using the **Automatic** function sets the **Static crowning** options back to the default setting **Dynamic crowning** on Xpert machines.

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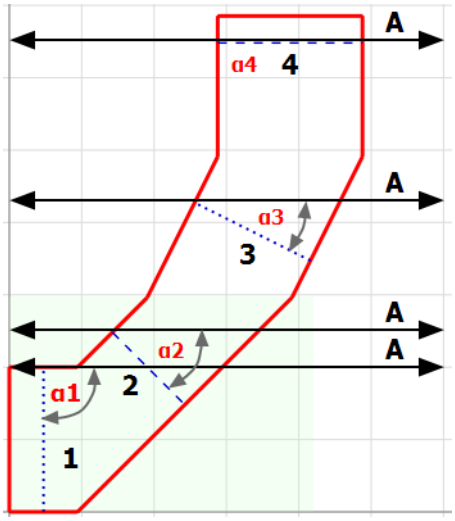
Grain direction compensation (whole part)

Grain direction compensation accounts for the grain direction of the sheet when bending, taking into account the angle (**α1-4**) between grain direction (**A**) and the bend lines (**1-4**).

Example:

- Angle correction = **2°**
- Grain direction = **0°**

Angle	Degrees	Compensation
α1	90°	2°
α2	45°	1°
α3	27°	0.6°
α4	0°	0°



Mode

- **Off**
  No grain direction compensation is performed.
- **Database**
  The value for angle correction is used from the bending table in Administrator.
- **Manual**
  The values for angle correction and grain direction can be entered manually.

Angle correction

You can enter a manual correction value. The angle correction can only be entered in manual mode.

Grain direction

Defines the grain direction of the part with the angles **0°** and **90°**.

- **0°** = grain direction in the horizontal direction
- **90°** = grain direction in the vertical direction

And angle of **0°** is used by default.

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7.11.2 Compensation

Where do I find the menu?

Select: **Modules > Part Editor > Bending > Compensation**

This dialog displays the status of compensation for the various bending processes in a table. The currently selected bending process is shown in the 3D view.

Tip: Double-clicking on a process opens the **Edit compensation** dialog box in which you can configure the compensation for the selected bending process.

What each column means:

Process

A serial number indicates the sequence of the bending processes.

Angle

Shows the angle of the bend. An angle of **0** is only used for the final bend in a hemming operation.

Bending operation

Shows the type of bend in the bending process.

- Normal bending
- Hemming
- Pre-bending
- Auxiliary bend
- Stepped radius bend

Mode

Displays the set compensation mode *(See: ›Mode‹ Page 628)*

Adopt for process

Displays the number of the process from which the compensation is adopted *(See: ›Adopt for process‹ Page 628)* The number **-1** means do not adopt.

Ram action in clamping point

Displays the value for ram action in clamping point *(See: ›Ram action in clamping point‹ Page 628)*

Grain direction compensation

Displays the value of grain direction compensation for the respective bending process *(See: ›Grain direction compensation (whole part)‹ Page 630)*.

Dynamic crowning

Displays the mode of dynamic crowning *(See: ›Crowning‹ Page 629)*

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7.12 Menu: LAMS

Where do I find the menu?

Select: **Modules > Part Editor**

This menu is displayed if the selected bending machine is fitted with an angle measurement system (LAMS).

Application:

Use the angle measurement system to bend extremely precise angles.



Tip: The LAMS can be moved manually using the mouse.

The display in the main window can be changed from the View bar and Navigation bar.

Functional groups

The function bar contains the following functional groups:

- Angle measurement system

7.12.1 Group: Angle measurement system

Where do I find the group?

Select: **Modules > Part Editor > LAMS**

This group contains all the functions that can be used to assign the angle measurement system to the respective bending process automatically or manually. You can run a check function to check that the angle measurement system is working correctly.

7.12.1.1 Angle measurement system settings



How do I open this dialog box?

Select: **Modules > Part Editor > LAMS > Angle measurement system**

This function is used to specify the criteria for employing the angle measurement system and its automated functions.

Use

This option defines the basic setting for the angle measurement system.



Note: If the angle measurement system is equipped with motorized positioning, then the system is positioned automatically.

- **Positioned - All measurements on**

  The angle measurement system is enabled for all bending processes and positioned on the current bending station.

- **Positioned - All measurements off**

  The angle measurement system is disabled for all bending processes and positioned on the current bending station.

- **Positioned - First bend angle measured per angle**

  The angle measurement system is enabled and positioned on the current bending station. If several identical bending angles are bent successively, only the first bend angle is ever measured until a new angle is bent.

- **Parked**

  You use this attribute to move the Angle measurement system into the parked position. The angle measurement system remains in the parked position in the current process until the **Park** attribute is manually disabled using the **Edit** function from the **LAMS** menu, or until another use has been selected.

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Every n'th bending part

For a series of identical bending parts, you can enter here the number of parts after which you want the angle measurement system to check the part.



Note: This option only works when the attribute **Positioned - All measurements on** or **Positioned - First bend angle measured per angle** is selected for the **Use** option.

Mode

The spring-back measurement is needed to correct for the spring-back of the material during bending.

- **Measurement On**

  The spring-back angle is measured automatically.

- **Database**

  The spring-back angle from the database is used.

- **Manual**

  The spring-back angle must be entered manually via the **Angle** option.

Angle

If the **Manual** attribute is selected for the **Spring-back angle**, the spring-back angle can be entered manually in this field.

7.12.1.2 Set angle measurement system automatically



How do I open this dialog box?

Select: **Modules > Part Editor > LAMS > Angle measurement system**

This function positions the LAMS automatically on the bending station of the individual processes, based on the criteria set in the **Angle measurement system settings** dialog box.

7.12.1.3 Complete all settings for angle measurement system



Where do I find the function?

Select: **Modules > Part Editor > LAMS > Angle measurement system**

This function applies the angle measurement system automatically to all bending processes that have not yet been assigned an angle measurement system, based on the criteria set in the **Angle measurement system settings** dialog box.

☞ **Note:** Those settings for the angle measurement systems that have already been assigned to a bending station are retained unchanged.

7.12.1.4 Check angle measurement system



How do I open this dialog box?

Select: **Modules > Part Editor > LAMS > Angle measurement system**

The check function only needs to be used if processes have the error status or not yet checked status .

7.12.1.5 Edit angle measurement system



How do I open this dialog box?

Select: **Modules > Part Editor > LAMS > Angle measurement system**

In this dialog box you can set or change the position of the angle measurement system for the selected bending process. Invalid settings are identified by an error symbol .

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Function

The functions of the angle measurement system are configured in this area.

Mode

This option enables/disables the angle measurement system. If the angle measurement system is disabled, the only adjustments you can make are to the position of the angle measurement system.

Adopt for process

The settings are adopted for the selected process.

Every n'th bending part

For a series of identical bending parts, you can enter here the number of parts after which you want the angle measurement system to check the part.

☞ **Note:** This option only works when the attribute **Positioned - All measurements on** or **Positioned - First bend angle measured per angle** is selected for the **Use** option.

Axis

This area is used for positioning the angle measurement system along the tool holder.

Park

This option is used to move the angle measurement system into the parked position. If this option is enabled, all other options for the angle measurement system are disabled.

Synchronous

This option allows you to position to front and rear angle measurement system separately. Additional options are shown.

LZ position

This option can be used to position the front and rear angle measurement system parallel to the bending station of the selected bending process.

LZB position

This option can be used to position the rear angle measurement system next to the bending station of the selected bending process.

LZF position

This option can be used to position the front angle measurement system next to the bending station of the selected bending process.

Spring-back measurement

In this area you can set the correction for material spring-back during bending.

Mode

The spring-back measurement is needed to correct for the spring-back of the material during bending.

- **Measurement On**
  The spring-back angle is measured automatically.
- **Database**
  The spring-back angle from the database is used.
- **Manual**
  The spring-back angle must be entered manually via the **Angle** option.

Angle

If the **Manual** attribute is selected for the **Spring-back angle**, the spring-back angle can be entered manually in this field.

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7.12.1.6 Reposition axes



How do I open this dialog box?

Select: **Modules > Part Editor > LAMS > Angle measurement system**

This option recalculates the positions of the angle measurement system for all processes, and repositions the system if necessary.

☞ **Note:** If the angle measurement system does not have a motorized drive, it must be positioned by the operator.

Use this option when the following changes have been made:

- If the position of a bending station has been changed after being set.
- If the position of a bending part on a bending station has been changed after being set.

7.12.1.7 Optimize angle measurement system



Where do I find the function?

Select: **Modules > Part Editor > LAMS > Angle measurement system**

This function attempts to reduce axis movement of the angle measurement system between processes. Optimization is performed based on the settings in the **Optimize settings** dialog.

☞ **Note:** The option is only enabled if the angle measurement system has a valid status  .

7.12.1.8 Optimize angle measurement system settings

How do I open this dialog box?

Select: **Modules > Part Editor > LAMS**

This dialog is used to edit the optimization settings for the angle measurement system.

The **Optimize** function assigns the settings to all LAMS positions of the bending processes.



Note: Running the **Automatic** function will also apply the settings to the bending processes.

Optimize

- **Z position**

  If possible, the program tries to use the same LAMS positions in the LZ direction.

- **Position window**

  Specifies the maximum distance of the axis in the LZ direction in which the automatic angle measurement system may be moved.

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7.12.2 Angle measurement system processes

Where do I find the menu?

Select: **Modules > Part Editor > Bending > LAMS**

This dialog box contains a table displaying the status of the angle measurement system for each of the bending processes. The currently selected bending process is shown in the 3D view.

Tip: Double-clicking on a process opens the **Angle measurement system settings** dialog box in which you can configure the angle measurement system for the selected bending process.

Positioning the angle measurement system using the foot pedal

If the angle measurement system will be moved into a different position for the next bending process, the operator must confirm the repositioning of the angle measurement system by pressing the foot pedal. The angle measurement system will then move to the relevant position.

Confirmation is used to prevent the angle measurement system colliding with the bending part. It is only used with a motor-driven angle measurement system.

Note: The minimum distance at and above which confirmation using the foot pedal is required is defined in the bending database.

What each column means:

Status

Displays the status of the angle measurement system in the bending sequence.

Symbol	Description
	A valid angle measurement system is assigned to the bending process.
	A valid angle measurement system has not yet been assigned to the bending process.
	The angle measurement system for the bending process has not been checked yet. Run the **Check** function.

Process

A serial number indicates the sequence of the bending processes.

Angle

Shows the angle of the bend. An angle of **0** is only used for the final bend in a hemming operation.

Bending operation

Shows the type of bend in the bending process.

- Normal bending
- Hemming
- Pre-bending
- Auxiliary bend
- Stepped radius bend

Mode

Shows whether LAMS is enabled for this process.

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7.13 Menu: Correction

Where do I find the menu?

Select: **Modules > Part Editor**

This menu allows you to correct punch and peripheral axis movements during bending processes.

The display in the main window can be changed from the View bar and Navigation bar.

Functional groups

The function bar contains the following functional groups:

- Correction

7.13.1 Group: Correction

Where do I find the group?

Select: **Modules > Part Editor > Correction**

This group contains the functions for editing the movement of punch and peripheral axes.

7.13.1.1 Edit correction



Where do I find the function?

Select: **Modules > Part Editor > Correction > Correction**

This dialog is used to correct upper dead center and for the start of peripheral axes for each bending process.



Caution! Moving peripheral axes can knock the bending part towards the operator and damage it. Observe the handling of the bending part when correcting peripheral axes. For bending parts with long sides behind the bending tool, use the **Delay** option to trigger movement of the peripheral axes by the operator.

Upper dead center

This option defines the distance for the punch to move up from the top edge of the bottom tool after the bending process.

- **min.:** Sheet thickness + 1.0 mm
- **max.:** < maximum position Y1/Y2

Switchover point Down

As of this point, the upper clamping bar switches from rapid speed to bending speed when moving down.

- **min.:** Safety point
- **max.:** < maximum position Y1/Y2

Clamping point

This option defines the position at which the bending part is clamped. Programmed retractions or bending aids are moved as of this point.

- **min.:** 50% sheet thickness
- **max.:** < switchover point

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Switchover point Up

As of this point, the upper clamping bar switches from bending speed to rapid speed when moving up.



Note: If **Bending speed, upwards** is set to a value **<10 mm/s**, **Switchover point Up** is automatically set to sheet thickness.

- **min.:** programmed Y position
- **max.:** Maximum position

Bending speed, downwards

This option defines the speed with which the machine moves down as of **Switchover point Down**.

- **min.:** 0.5 mm/s
- **max.:** 10.0 mm/s

Bending speed, upwards

This option defines the speed with which the machine moves up as of **Switchover point Up**.



Note: This speed only takes effect if **Switchover point Up** >BDC. If **Bending speed, upwards** is set to a value **<10 mm/s**, **Switchover point Up** is automatically set to sheet thickness.

Bending speed, downwards

This option defines the speed with which the machine moves down as of **Switchover point Down**.

- **min.:** 0.5 mm/s
- **max.:** 10.0 mm/s

Pressing force

This option determines the force to be exerted to bend the bending part.

- **min.:** 0 t
- **max.:** < max. machine pressing force / max. tool load

Dwell time

This option specifies how long the upper clamping bar stops at bottom dead center (BDC).

- **min.:** 0 s
- **max.:** 10s

BDC stop

If this option is enabled, the upper clamping bar stops at bottom dead center as long as the operator is pressing the control (pedal or two-hand button). As soon as the control is released, the upper clamping bar moves to the programmed upper dead center.

Start peripherals

This option defines the time as of when peripheral axes are allowed to move.

- **Clamping point**

  If the upper clamping bar passes the clamping point when moving up, the peripheral axes move to the next programmed position.
- **Upper dead center**

  If the upper clamping bar reaches upper dead center, the peripheral axes move to the next programmed position.
- **Delay**

  The peripheral axes only start to move when released by the pedal.

Pass counter

This option defines how often to perform the bending process. The default number is **1**. The number of passes for the bending process can be 2 or more for special applications.

Application:

- Series production of parts for which you want to perform the same bend on a given number of parts.

A number of **0** skips the bending process on the machine.

Application:

- If you want to perform the bending process on a different machine or with a Z tool.

7.13.1.2 Remove correction



Where do I find the function?

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Select: **Modules > Part Editor > Bend > Correction > Correction**

This function removes all corrections from the selected bending process.



Note: This function is only enabled if a correction was made on the bending process.

7.13.2 Processes correction

Where do I find the menu?

Select: **Modules > Part Editor > Bend > Correction**

This dialog displays the status of correction for the various bending processes in a table.

Note: If upper dead center or the start of the peripheral is corrected, this is displayed by a symbol in the column.

Tip: Double-clicking on a process opens the **Edit correction** dialog box in which you can configure the correction for the selected bending process.

What each column means:

Process

A serial number indicates the sequence of the bending processes.

Angle

Shows the angle of the bend. An angle of **0** is only used for the final bend in a hemming operation.

Bending operation

Shows the type of bend in the bending process.

- Normal bending
- Hemming
- Pre-bending
- Auxiliary bend
- Stepped radius bend

Upper dead center

Displays the measurement configured for upper dead center.

Start peripherals

Displays the starting point for moving the peripheral for the next bending process.

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7.14 Menu: Export

Where do I find the menu?

Select: **Modules > Part Editor**

In this menu, you can use the functions in the **Bending part** group to export the bending part to the assigned bending machine.

The display in the main window can be changed from the View bar and Navigation bar.



Tip: Use the simulator as a final check before exporting all the bending processes.

7.14.1 Group: Bending part

Where do I find the menu?

Select: **Modules > Part Editor > Simulation**

This group contains all the functions for exporting the bending part for the assigned bending machine.

7.14.1.1 Export bending part automatically



Where do I find the function?

Select: **Modules > Part Editor > Simulation > Bending part**

This function exports the current bending part into the relevant folder, based on the settings made in the **Bending part export settings** dialog box.

7.14.1.2 Export bending part manually



Where do I find the function?

Select: **Modules > Part Editor > Simulation > Bending part**

This function exports bending plans to a user-defined output folder. Running the function opens a dialog box in which you must specify the output folder.

After confirming with **OK**, the bending plan is exported to this folder.

The software runs a check of the bending plan in the background prior to export. If any errors are found in the bending plan, these are displayed in the **Results of plan check** dialog box as a list of the errors.

Select **Continue** if you still want to export the bending plan or **Cancel** if you do not want to carry out the export.

☞ **Note:** If a bending plan with the same name has already been exported, a confirmation prompt is displayed asking whether you wish to overwrite the existing plan.

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7.14.1.3 Calculate bending time



Where do I find the function?

Select: **Modules > Part Editor > Simulation > Bending part**

This function calculates bending time for the part.

Bending time for the part is displayed in the **Bending time** dialog. Specify the applicable table in the **Table** selection box. You can edit tables in Administrator *(See: ›Bending-time table‹ Page 99)*.

Note: Bending time is used in the Plant Manager module and in the Tool Setup Optimizer to create an optimize bending jobs.

Attention! Only calculate bending time after completely calculating the bending technology.

7.14.2 Bend Robot



Where do I find the function?

Select: **Modules > Part Editor > Bending > Export**

This function opens the current bending part in Bend Robot for further processing as a robot part. Any changes to components are saved and bending parts are automatically exported before the Robot Manager module is launched.



Note: If the part contains embedded objects, they are listed in a warning message.

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7.15 Cutting

In this area you can assign to parts a cutting technology for Bystronic laser cutting machines or water jet cutting machines. To access the Technology area, click the **Cutting** option in the software header or the **Cutting technology** menu. Additional menus are displayed.

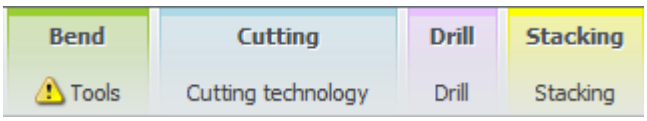


Fig. 7-163: Area display in the header

Note: The **Cutting** area is only available if the relevant license has been purchased.

Menus

The following additional menus are displayed in the **Cutting** area:

- Cutting technology
- Machining time

7.16 Menu: Cutting technology

Where do I find the menu?

Select: **Modules > Part Editor**

The cutting technology for the current part is configured in this menu.

The cutting technology is needed in order to tell the machine controller how you want the part machined. The cutting technology is composed of the cutting properties, the machining sequence and the lead-in and lead-out.

If the part is meant to be produced on different machines, the cutting technology for a different machine can be overwritten in the Part Nester module.



Attention! In order to be able to nest the part in the Part Nester module, the part must be assigned a cutting technology.

Functional groups

The function bar contains the following functional groups:

- Selection
- Entire technology
- Cutting properties
- Machining sequence
- Lead-in/Lead-out
- Microjoint
- View
- Snap
- Part

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7.16.1 New cutting technology

This dialog box is displayed if you switch from the **Part** or **Geometry** menu into the **Cutting technology** menu but no machine has been specified yet.

Specify the machine that you wish to use for cutting the part. After confirming with **OK**, the view switches to the **Cutting technology** menu in which you can assign the cutting technology to the part.

Cutting process

The information about the machine is needed in order to assign the cutting technology and to calculate the machining time.

Machine

This field displays the machine that shall be used to machine the object. Clicking on the arrow symbol on the right lets you select a different machine from a list. This list only contains those machines that are configured in the system. If you require a new machine, you must create it in the **Administrator** module in the **Cutting machines** form *(See: ›Cutting machine‹ Page 102)*.

Cutting gas

This field is used to specify the cutting gas with which to cut the part.

Cutting parameter

This field is used to specify the cutting parameter with which to cut the part.

Nesting settings

The nesting settings are needed for nesting the part in the Part Nester module.

Priority

You can use this option to set the order in which the different parts are to be nested. Entering a priority is optional. Enter a number between 1 and 99 for the priority, Where 1 stands for highest priority and 99 for lowest priority. No priority is assigned if 0 is entered.



Attention! The use of different priorities does not usually result in optimum nesting of the cutting plan.



Note: This function is only used for automatic nesting.

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Rotation allowance

This field defines the increment of the angle of rotation in degrees. The part can be rotated during nesting through this angle.



Tip: A rotation allowance can be entered by clicking on the input field or by selecting a set angle value via the left arrow button.

Discrete angles of rotation

You can enter multiple angles of rotation in this box. The part can be rotated during nesting through this angle. The angles of rotation must be separated from each other by semicolons (**;**).



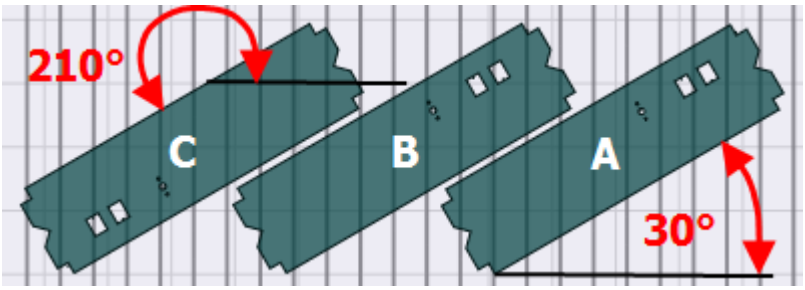
Tip: If this field contains angles of rotation, the angle of rotation in **Rotation allowance** is ignored (not for cluster nesting).

Same orientation

This option nests all instances of the part in the same orientation on the sheet.

Example:

If the part is nested with an angle of **30°** (**A**), the other instances can only be nested with the same angle (**B**) or with an angle of **210°**=180°+30° (**C**).



Application:

- Parts with same direction of grinding
- The correction value only needs to be entered on the bending machine for the first bending part. Because the alignment of the grain direction is the same, it is not necessary to enter a correction for all other parts.

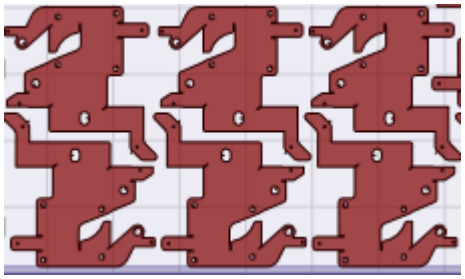


Fig. 7-164: Same orientation of a bending part on the sheet

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Mirroring authorization

This option permits the part to be mirrored about the vertical axis during nesting.

Filler part

Enable this option if you want to use this part to fill the areas leftover from nesting, or if the part is often needed in larger, unspecified quantities.

Test part

Flags the part as a test part. In Part Nester all instances of the part are nested as test parts *(See: ›Nesting a test part‹ Page 1375)*.

Sheet format

Use this option to assign an existing sheet format to the part. Depending on the size of the parts, this may help reduce sheet waste. If you nest the part on a different sheet format in Part Nester, the **Parts** tab displays a warning.

Part in cluster

This option specifies that the part can only be nested in a framed cluster *(See: ›Framed cluster‹ Page 1213)*.

Unique in cluster

This option specifies the part can only be nested in a cluster. It is not possible to nest it outside a cluster.

Take cutouts into account

These options determine whether or not to apply cutouts that were set on a part in Part Editor.

- **Default**

  If the part contains cutouts, they will be taken into account for nesting if the parameter is enabled in the nesting settings.

- **No**

  No cutouts are taken into account for nesting.

> ⚠ **Attention!** The cutouts of a part are changed if the contour of another part is nested in a cutout. Based on the contour, the cutout is divided by the contour.

Yes

If the part contains cutouts, they are always taken into account for nesting.

> **Note:** The cutouts are taken into account even if the parameter is disabled in the nesting settings.

Take flexible tandem gripping into account

This option can be used to nest large parts to be sorted with tandem grippers with a specific angle of rotation on a cutting plan in Part Nester.

Common Cut

This option allows the combining of parallel contours during nesting based on certain criteria. These criteria are defined in the export settings for the pre-processor in the Part Nester module *(See: ›Common Cut‹ Page 1153)*.

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7.16.2 Group: Selection

Where do I find the group?

Select: **Modules > Part Editor > Cutting technology**

This group displays all the available selection functions. These functions can be applied to all elements of an object.

7.16.2.1 Geometry



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Selection**

This function lets you select individual elements of a geometry.

You can select the following elements:

- Contours
- Segments
- Reference points



Note: This function is also used for importing geometries. During the import, the function can be used to select geometry elements that cannot be selected with the **Class** function.



Tip: Use the **Ctrl** key to select multiple geometry elements. Use the **Shift** key to deselect the selection of individual geometry elements.



Tip: Use the function **Selection** > **All** > **Identical** to select identical objects.In this selection mode you can also select geometry elements that have the same operating mode without having to switch to **Operating mode** selection mode.

7.16.2.2 Operating mode



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Selection**

This function selects all the elements of a geometry that have the same operating mode.



Tip: You can use this function, for instance, to edit the cutting technology for all engraved contours.

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7.16.3 Group: Entire technology

Where do I find the group?

Select: **Modules > Part Editor > Cutting technology**

The functions in this group can be used to assign a cutting technology automatically to the part. The technology is assigned via the Cutting Wizard and other options that are specified in the **Cutting technology, automatic** dialog box.

7.16.3.1 Automatic



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Technology**

This function automatically assigns the entire cutting technology to the current part.



Note: If no contour is selected, the technology is assigned to the entire part. If one or more contours are selected, the cutting technology is only assigned to this contour/these contours.

- **Complete**

  The cutting technology is only set on contours which do not have a technology yet.

- **Settings**

  You can define the cutting technology in the **Cutting technology settings** docking dialog.

7.16.3.2 Settings



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Technology**

This dialog box is where the cutting technology settings are defined. The cutting technology is assigned to the contours using the **Automatic** or **Complete** options.

Note: The tool correction is set as inwards for inner contours and outwards for outer contours to ensure that parts are cut with exactly the specified dimensions.

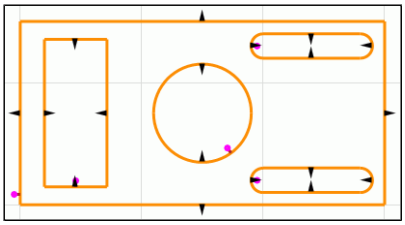


Fig. 7-165: Automatically assigned tool correction

Tip: The tool correction can be changed using the **Contour** function in the **Cutting properties** group.

Cutting wizard

In this pane you define the Cutting Wizard to apply to the contour of the part.

Name

The selected Cutting Wizard is displayed in the **Name** field. If there are several Cutting Wizards available, you can select another Cutting Wizard by clicking in the field.

Note: When selecting a cutting wizard, the system remembers the cutting wizard that was last selected. If no cutting wizard is found by setting filters, and if you disable the filters again, this box displays the cutting wizard that was last selected.

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Automatic

Enabling this box adopts the settings from Settings Manager. The filter definitions are hidden.

If no valid table is found, an error message is displayed to this effect.

Filter definitions

These filters can be used to restrict the choice of Cutting Wizards. The properties of the Cutting Wizard can be displayed and edited in the Administrator module in the **Cutting wizard** form *(See: ›Cutting Wizard‹ Page 116)*.

If a number of Cutting Wizards are found that match the filter criteria, the user must manually select the data record that has the appropriate cutting gas.

☞ **Note:** The filter definitions is defined in the **Part properties** > **General** / **Cutting** in the **Material** and **Cutting process** sections.

Cutting gas / Abrasive filter

This filter option can be used to define the Cutting Wizard more exactly. The cutting gas or, on ByJet machines, the abrasive displayed is defined in the **Part properties** dialog *(See: ›Cutting gas‹ Page 375)*.

Cutting properties, text

In this pane you define the machining mode for text that has been set on the surface of a part in the **Geometry** menu. The properties that are set are applied to all texts on a part.

Operating mode

- **Default:** The parameters for machining the material are assigned by the pre-processor during export of the cutting plan, but not before this stage.
- **Cutting:** The CW parameters are used for machining the material.
- **Operating mode 2:** This operating mode is used to make a quality cut (water jet cutting only).
- **Pulse:** The pulsed parameters are used for machining the material. Pulsed operation can improve the machining quality for certain materials or for tight contours by reducing the amount of heat input to the material (laser cutting only).
- **Engraving:** The cutting parameters for engraving the material are used. Closed inner contours that have this machining mode are no longer displayed as a cut-out in the Part Editor.
- **Process macros:** The relevant process macro is used for machining the material.
  - Laser process macro
  - Water process macro

Piercing mode

- **Default:** The piercing mode is assigned by the preprocessor when the cutting plan is exported, but not before this stage.
- **Normal:** Piercing is performed at full power.
- **Without time:** The piercing time saved in the cutting parameters on the machine is ignored.
- **Pulsed:** Piercing is performed with pulsed power using the piercing parameters specified in the cutting parameters (laser only).
- **Drilled:** Pilot drilling is performed using the drill parameters specified in the cutting parameters (water only).

☞ **Note:** Pulsed piercing produces smaller piercing holes than normal piercing.

Cutting properties, parts ID

This is the area where you specify the operating mode for the **Parts ID** option. The **Process macro 5** operating mode is used for text detection by default.

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Machining sequence

This pane contains several options for specifying the order in which contours are machined. The choice of attribute can have a huge impact on the machining time for the part.

Mode

Constructing sequence

The machining sequence is set by the order in which the contours are constructed.

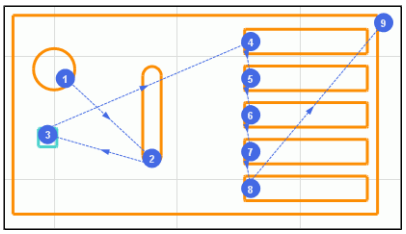


Fig. 7-166: Machining sequence set by constructing sequence

Optimized positioning path

The sequence for the contours is computed to give the shortest overall distance for all traversing paths together.

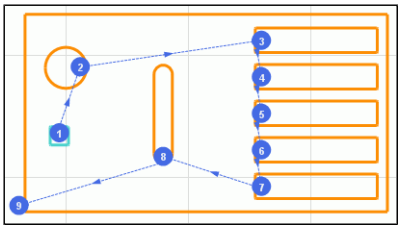


Fig. 7-167: Optimized machining sequence

Vertical - From left to right

The contours are processed vertically (up and down) starting from the lower left and progressing to the top right.

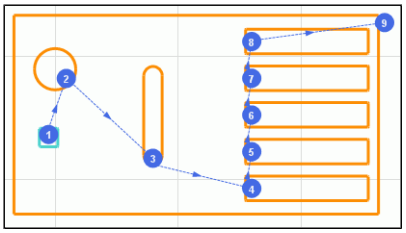


Fig. 7-168: Vertical machining sequence from left to right

Vertical - From right to left

The contours are processed vertically (up and down) starting from the lower right and progressing to the top left.

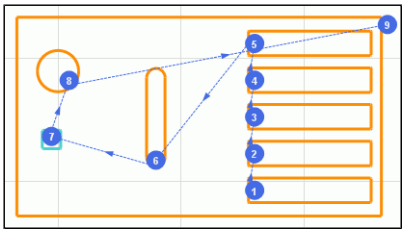


Fig. 7-169: Vertical machining sequence from right to left

Horizontal - From bottom to top

The contours are processed horizontally (from right to left) starting from the bottom right and progressing to the top left.

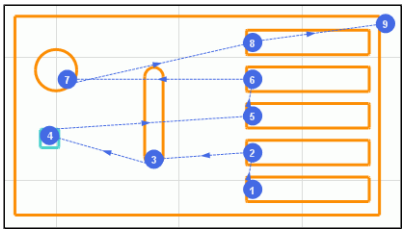


Fig. 7-170: Horizontal machining sequence from bottom to top

Horizontal - From top to bottom

The contours are processed horizontally (from right to left) starting from the top right and progressing to the lower left.

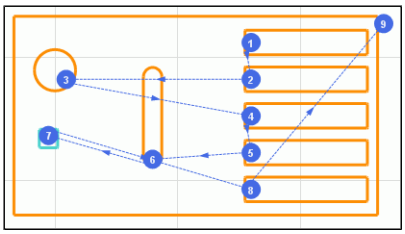


Fig. 7-171: Horizontal machining sequence from top to bottom

Random order

The sequence for the contours is defined by a random sequence generator. Select **Random order** to avoid too much heat entering the material when laser-cutting tightly packed contours or heavy-gage sheets.

☞ **Note:** This setting usually results in the longest machining time.

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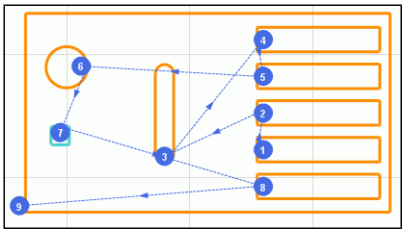


Fig. 7-172: Machining sequence with random order

Increment

This option can be used to set the increment between machining sequence numbers when the contours are actually machined.

- **Increment 1:** All contours are machined in accordance with the computed machining sequence (default).
- **Increment >1:** The actual machining sequence is applied to every nth contour of the computed machining sequence.

Using the increment option for a perforated sheet

When the increment is set to 2, only every second hole in the hole sequence is cut. The remaining holes are cut in the second pass in the given sequence.

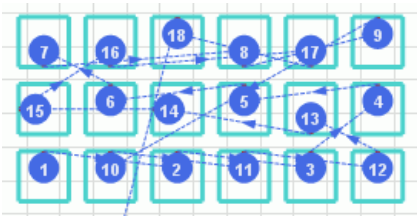


Fig. 7-173: Machining sequence for a perforated sheet with increment of 2

Tip: Use an increment >1 to distribute the generated heat more evenly in perforated sheets.

Right lead-out first (ByTube)

This option cuts the right lead-out on a tube first.

The sequence of cutting is as follows:

1. Right lead-out
2. Inside contours
3. Left lead-out

If this option is not enabled, the cutting sequence is performed as follows.

1. Inside contours
2. Right lead-out
3. Left lead-out

Group by sectors

Use this option to divide the total area of the part into sectors. Inside each sector, the identified contours are processed as a group using the machining sequence set under **Mode**.

Note: The sectors are displayed in a preview as blue and white areas defined by the dimensions entered in the **Grid X/Y** fields.

This option can be used for the following applications:

- To achieve a shorter positioning path for a part that has numerous different operating modes.
- For a part which contains numerous small inner contours in heavy-gage material, in order to distribute the heat input to the material more evenly. In this case, the **Random order** mode is recommended in order to distribute the sectors randomly over the part.

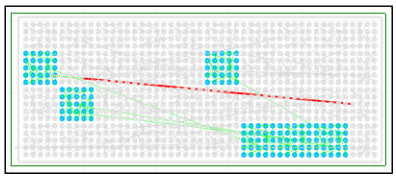


Fig. 7-174: Machining a perforated sheet by sectors

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Grouping by operating mode, laser/water

The contours are processed in the order set for the operating modes. You can set the order to optimize the heat input to the material and to increase process reliability. Use the two arrow buttons to set the order of the operating modes.

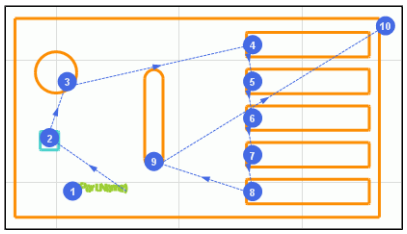


Fig. 7-175: Machining sequence based on the operating-mode order engraving, process macro 1 and cutting

Group by size

This option sets the machining sequence in the part by contour size. All inside contours are processed using the machining sequence set in **Mode**.

- **Small contours first**

  The machining sequence goes from the smallest inside contours to the biggest inside contours.

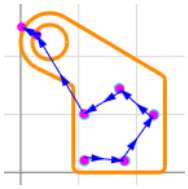


Fig. 7-176: Machining sequence with “Small contours first” mode

- **Large contours first**

  The machining sequence goes from the biggest inside contours to the smallest inside contours.

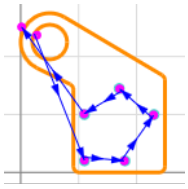


Fig. 7-177: Machining sequence with “Large contours first” mode

Lead-in/Lead-out

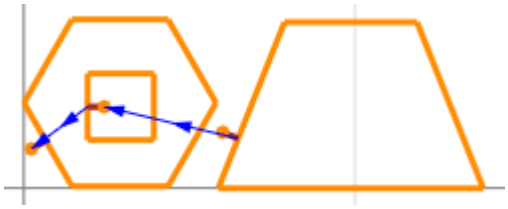
This area contains the options for setting lead-ins and lead-outs on a contour. Depending on what attribute is selected, it is possible to achieve dynamic positioning during the cutting process.

Position

Use this option to specify the position for automatic placement of the lead-ins and lead-outs.

- **Along the machining sequence**

  The software first calculates the shortest possible positioning path between the contours, and then places the lead-ins automatically so that they point towards the nearest contour. This prevents the tool traveling into the holes of contours that have already been cut when it is traveling in lowered positioning mode.



- **Coming from left**



- **Coming from right**



- **Coming from above**



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- **Coming from below**



- **First segment**

  The lead-in is placed in the center of the first segment of the contour.



Optimize machining direction

This option is used to adapt the contour machining direction to produce the minimum change in direction between the contour lead-in and the traversing path. This avoids unnecessary deceleration and increases the machining speed.

When the machining direction is changed, the tool correction is also changed automatically to the correct side (left/right).

Note: This option is only used for the lead-in types **Direct** and **Hook**.

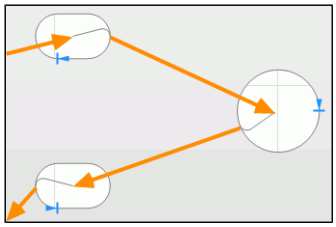


Fig. 7-178: Optimized contour machining direction to suit traversing path

Place lead-in in corners

This option places the lead-in of an outer contour in a corner of the contour.

☞ **Note:** The position of the lead-in depends on the setting in the **Position** option.



Fig. 7-179: Lead-in in a corner of the contour with **position coming from above**

Segments without lead-ins/lead-outs or microjoint

Detect contact edge

This option detects the segment for the contact edge of a bending part (**A**) and sets the **No lead-in or microjoint** option on the segment *(See: ›No lead-in or microjoint‹ Page 692)*. This option prevents setting the lead-in (**B**) on the segment of a contact edge.

Restriction:

It is only possible to set the **No lead-in or microjoint** option on a segment automatically if the backgauges can be positioned on a single straight segment.

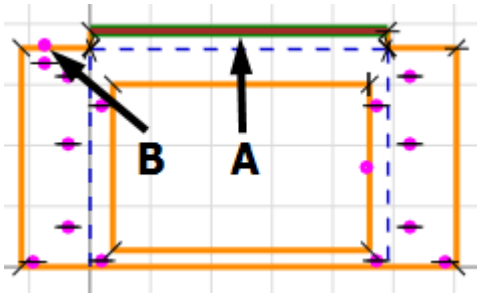


Fig. 7-180: Contact edge with “no lead-in / microjoint” attribute

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7.16.3.3 Remove



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Entire technology**

This function removes the entire cutting technology from the part. It also removes the technology from segments and reference points.



Note: If no contour is selected, the technology is removed from the entire part. If one or more contours are selected, the cutting technology is only removed from this contour/these contours.

7.16.4 Group: Cutting properties

Where do I find the group?

Select: **Modules > Part Editor > Cutting technology**

The functions in this group can be used to assign cutting properties of different contour elements manually to the part. The properties are assigned via the relevant docking dialog boxes.

☞ **Note:** At least one contour element must be selected in order to be able to edit a cutting property.

7.16.4.1 Automatic



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Cutting properties**

This function automatically assigns the cutting properties to the current part without recalculating the machining sequence, lead-ins and lead-outs, and microjoints.

☞ **Note:** If no contour is selected, the cutting property is assigned to the entire part. If one or more contours are selected, the cutting property is only assigned to this contour/these contours.

- **Complete**

  The cutting properties are only set on contours which do not have any cutting properties yet.

- **Settings**

  You can define the cutting property in the **Cutting properties settings** docking dialog.

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7.16.4.2 Cutting properties settings



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Cutting properties > Automatic**

This dialog box is where the cutting property settings are defined. The cutting technology is assigned to the contours using the **Automatic** or **Complete** options.

Note: The tool correction is set as inwards for inner contours and outwards for outer contours to ensure that parts are cut with exactly the specified dimensions.

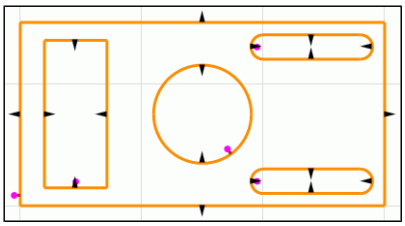


Fig. 7-181: Automatically assigned tool correction

Tip: The tool correction can be changed using the **Contour** function in the **Cutting properties** group.

Cutting wizard

In this pane you define the Cutting Wizard to apply to the contour of the part.

Name

The selected Cutting Wizard is displayed in the **Name** field. If there are several Cutting Wizards available, you can select another Cutting Wizard by clicking in the field.

Note: When selecting a cutting wizard, the system remembers the cutting wizard that was last selected. If no cutting wizard is found by setting filters, and if you disable the filters again, this box displays the cutting wizard that was last selected.

Automatic

Enabling this box adopts the settings from Settings Manager. The filter definitions are hidden.

If no valid table is found, an error message is displayed to this effect.

Filter definitions

These filters can be used to restrict the choice of Cutting Wizards. The properties of the Cutting Wizard can be displayed and edited in the Administrator module in the **Cutting wizard** form *(See: ›Cutting Wizard‹ Page 116)*.

If a number of Cutting Wizards are found that match the filter criteria, the user must manually select the data record that has the appropriate cutting gas.

☞ **Note:** The filter definitions is defined in the **Part properties** > **General** / **Cutting** in the **Material** and **Cutting process** sections.

Cutting properties, text

In this pane you define the machining mode for text that has been set on the surface of a part in the **Geometry** menu. The properties that are set are applied to all texts on a part.

Operating mode

- **Default:** The parameters for machining the material are assigned by the pre-processor during export of the cutting plan, but not before this stage.
- **Cutting:** The CW parameters are used for machining the material.
- **Operating mode 2:** This operating mode is used to make a quality cut (water jet cutting only).
- **Pulse:** The pulsed parameters are used for machining the material. Pulsed operation can improve the machining quality for certain materials or for tight contours by reducing the amount of heat input to the material (laser cutting only).
- **Engraving:** The cutting parameters for engraving the material are used. Closed inner contours that have this machining mode are no longer displayed as a cut-out in the Part Editor.
- **Process macros:** The relevant process macro is used for machining the material.
  - Laser process macro
  - Water process macro

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Piercing mode

- **Default:** The piercing mode is assigned by the preprocessor when the cutting plan is exported, but not before this stage.
- **Normal:** Piercing is performed at full power.
- **Without time:** The piercing time saved in the cutting parameters on the machine is ignored.
- **Pulsed:** Piercing is performed with pulsed power using the piercing parameters specified in the cutting parameters (laser only).
- **Drilled:** Pilot drilling is performed using the drill parameters specified in the cutting parameters (water only).

☞ **Note:** Pulsed piercing produces smaller piercing holes than normal piercing.

Cutting properties, parts ID

This is the area where you specify the operating mode for the **Parts ID** option. The **Process macro 5** operating mode is used for text detection by default.

Segments without lead-ins/lead-outs or microjoint

Detect contact edge

This option detects the segment for the contact edge of a bending part (**A**) and sets the **No lead-in or microjoint** option on the segment *(See: ›No lead-in or microjoint‹ Page 692)*. This option prevents setting the lead-in (**B**) on the segment of a contact edge.

Restriction:

It is only possible to set the **No lead-in or microjoint** option on a segment automatically if the backgauges can be positioned on a single straight segment.

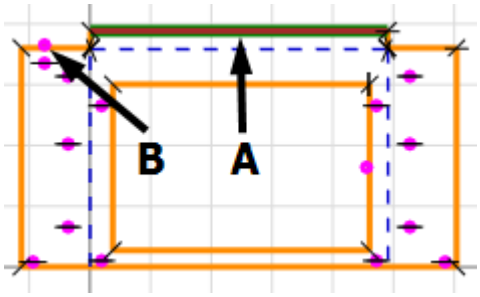


Fig. 7-182: Contact edge with “no lead-in / microjoint” attribute

7.16.4.3 Contour



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Cutting properties**

This function can be used to assign various cutting properties to the entire part or just to individual contours. Each of the properties can only be assigned to the complete contour.

- If no change is made to the relevant box, the cutting properties of the contour element concerned are retained.
- If a new cutting property is selected in a box, this cutting property is assigned to all selected contour elements.



Tip: Use the **Shift** key to select more than one contour element with the mouse. You can use the **Ctrl** key to deselect a contour element from a multiple selection.

Contour (geometry)

Engraving

This option is identical to the **Engraving** option in the **Geometry** menu*(See: ›Engraving‹ Page 405)*.

Behavior of cutting mode when changing the engraving option.

Activating or disabling the **Engraving** option does not change the operating mode configured.

Behavior of engraving mode when changing the operating mode.

Initial situation for engraving	Action	Result
disabled	Changing the operating mode from any mode to engraving.	The **Engraving** option is activated.
enabled	Changing the operating mode from any mode to engraving.	The **Engraving** option stays activated.
disabled	Changing the operating mode from engraving to any mode.	The **Engraving** option stays disabled.
enabled	Changing the operating mode from engraving to any mode.	The **Engraving** option is disabled.

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Contour

Operating mode

- **Default:** The parameters for machining the material are assigned by the pre-processor during export of the cutting plan, but not before this stage.
- **Cutting:** The CW parameters are used for machining the material.
- **Operating mode 2:** This operating mode is used to make a quality cut (water jet cutting only).
- **Pulse:** The pulsed parameters are used for machining the material. Pulsed operation can improve the machining quality for certain materials or for tight contours by reducing the amount of heat input to the material (laser cutting only).
- **Engraving:** The cutting parameters for engraving the material are used. Closed inner contours that have this machining mode are no longer displayed as a cut-out in the Part Editor.
- **Process macros:** The relevant process macro is used for machining the material.
  - Laser process macro
  - Water process macro

Note: If the operating mode of a contour is changed from **Cutting** to **Engraving**, the initial piercing cut and lead-in are automatically removed from the contour. For inner contours, the **Engraving** contour property is also set in the **Geometry** menu *(See: ›Engraving‹ Page 405)*.

Tip: Select the **Segment** function to change the operating mode for a specific element of a contour *(See: ›Segment‹ Page 690)*.

Piercing mode

- **Default:** The piercing mode is assigned by the preprocessor when the cutting plan is exported, but not before this stage.
- **Normal:** Piercing is performed at full power.
- **Without time:** The piercing time saved in the cutting parameters on the machine is ignored.
- **Pulsed:** Piercing is performed with pulsed power using the piercing parameters specified in the cutting parameters (laser only).
- **Drilled:** Pilot drilling is performed using the drill parameters specified in the cutting parameters (water only).



Note: Pulsed piercing produces smaller piercing holes than normal piercing.

Tool correction

These options can be used to specify the position of the tool (laser beam or water jet) with respect to the contour. Should you need to use a different tool correction for certain reasons, the tool correction can be changed using the following attributes:

- **None:** No tool correction performed. The cutting beam/jet is positioned in the center of the contour. No tool correction is displayed on the contour. This option is automatically applied to all open contours (**1**).
- **Left:** The correction is placed on the left-hand side of the contour viewed in the machining direction. This option is automatically applied to all inner contours (**2**).
- **Right:** The correction is placed on the right-hand side of the contour viewed in the machining direction. This option is automatically applied to all outer contours (**3**).

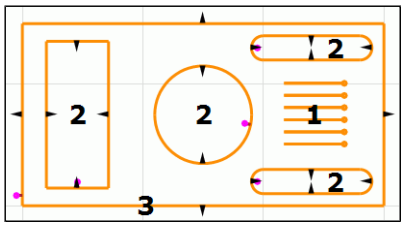


Fig. 7-183: Displaying the tool corrections on a part

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Attention! The contours of a part must not contain any segments that are shorter than the tool correction being used. Otherwise you may get unexpected cutting results or the CNC may crash. For contours with many short segments, use the **Smooth** function to reduce the number of segments of a contour *(See: ›Smooth‹ Page 464)*.

Note: The tool correction is always applied in the machining direction of the contour. You can display the machining direction of the contours using the **Special view** function.

Tip: You can display the tool correction for the contours using the **Special view** function in the **Cutting technology** menu.

- Tool correction, Laser CO_2= 0.2-0.5 mm
- Tool correction, Fiber laser = 0.1-0.6 mm
- Tool correction, Water = 0.8 mm

Tag

This option can be used to assign certain attributes to the contours of a part The attributes are not applied to the contours of the part until the export stage. The preprocessor in Part Nester applies the attributes, but only to those contours that are enabled.

The following attributes can be assigned to a tag:

Note: Tags are machined using the machining mode assigned to the contour.

- **Center punch mark**

  A center punch mark is set on the contour. In the exported cutting plan, a center punch mark is engraved in the center of the contour. The contour itself is not cut. This attribute is intended, for instance, for very small hole contours that will need to be drilled in the subsequent machining process. In the working documents, contours that have the center punch mark attribute are shown as a gray line with a black dot in the center *(See: ›Center punch mark‹ Page 1151)*.

- **Pre-piercing**

  The initial piercing cuts on the selected contours are pre-pierced before the contour is cut *(See: ›Pre-piercing‹ Page 1133)*.

- **Rounding off**

  Outside and inside corners are rounded with a specified radius *(See: ›Corner mode‹ Page 1130)*.

- **Looping**

  Sharp corners on an outer contour are cut with a corner loop *(See: ›Corner mode‹ Page 1130)*. This procedure offers the following advantages:

  - The axes travel with a continuous direction change when cutting a sharp corner.
  - The part itself is cut at a constant speed.
  - Axis overshoot and additional burring are avoided.
  - Heat input in the corner is reduced.

- **Scrap cut-down**

  For inner contours, a mesh of cuts is made within the contour before the actual inner contour is cut. This ensures that the waste pieces drop down through the cutting grating. It also avoids the risk of collisions with protruding parts, thereby improving process reliability. This option can only be used on inner contours *(See: ›Cutting waste into smaller sizes‹ Page 1141)*.

  Note: This option increase the machining time significantly. Parts cannot be nested in segments containing scrap cut-down.

- **Scanning**

  All rectangular or square contours are machined first in the X-direction and then in the Y-direction. For circular ring cut-outs, first those straight contours are cut for which the intersecting point runs through the center point of the circle. Afterwards, the arc segments of the contour are cut which are located on the same circular radius *(See: ›Scanning‹ Page 1135)*.

- **Vaporization**

  For film-coated sheets, the film on the contour is vaporized at low laser power *(See: ›Vaporization‹ Page 1139)*.

- **Thread**

  Threads are cut in contours with this attribute.

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Index marking

Defines an existing round contour hole on a blank as an index marking (**A**). There must be at least two holes per blank that can be used as index markings. The hole diameter must be between **5-45 mm**.

Before the part contour is cut, the index holes are detected with **Detection Eye** in order to determine the position of the part.

Attention! Index holes must not be recessed or have a chamfer as this makes detection with **Detection Eye** impossible or inadequate.

Note: Index markings are **not** assigned a cutting technology.

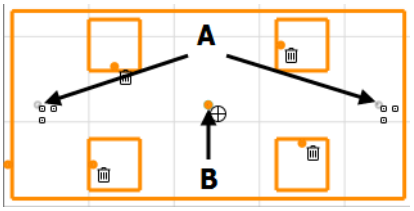


Fig. 7-184: Index and calibration marks on a part

Calibration marking

Defines a round contour on a blank as a calibration marking (**B**). This contour is cut in order to compensate for the expansion caused by the material being heated up with **Detection Eye**. The hole diameter must be between **5-45 mm**.

Before the part contour is cut, the calibration holes are detected with **Detection Eye** in order to determine the position of the part more accurately.

Note: Calibration marks can only be set as circular contours. Wherever possible, they should be set in the cutouts of a part.

Do not move lead-in

Locks the position of the lead-in so that the lead-in cannot be moved as a result of tilt optimization in the Part Nester module. Depending on requirements, this allows you to force the lead-in to be made in the same place.

Attention! This significantly decreases the efficiency of tilt optimization for the part in Part Nester.

Multiple engraving

This mark is used to execute contours with a certain width. The properties of the engraving are defined in the postprocessor *(See: ›Multiple engraving‹ Page 1179)*.

Tip: This option is particularly useful in order to make engravings more visible.

Keep cutting technology

If this flag is set, the post-processor does not change the cutting technology on the contour during export.

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Tube Editor

XYU mode

This option inhibits movement of the U-axis (rotary axis). The contour is cut solely by moving the cutting head in the X and Y-directions. This means that the contour is cut through the circular tube at right angles to the horizontal plane (**B + C**). The resultant through-hole runs along the full surface of the wall thickness.

The larger the radius of the through-hole, the closer the hole sides come to the edge of the tube, and hence the greater the material thickness to cut through. This means that the cut quality reduces towards the edge of the tube. The guide value recommended for the maximum radius of a through-hole (**x**) equals half the inside radius of the tube ($x_{max} = R_i/2$).

Note: XYU mode can only be used for circular tubes.

Tip: Use this option, for example, for thick-walled tubes to avoid cutting a wedge (**a**) on the outside of the tube.

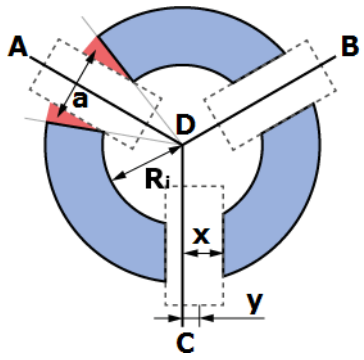


Fig. 7-185: Circular tube through-holes with and without XYU mode

A	through-hole cut with movement of the U-axis	**R_i**	tube inside radius
B	through-hole cut in XYU mode without movement of the U-axis	**a**	wedge-shaped cut-out region
C	through-hole cut in XYU mode with Y-offset	**x**	maximum radius for through-hole
D	center of through-holes	**y**	offset in Y-direction

Special considerations

For contours in XYU mode, the U axis always rotates relative to the lead-in position of the contour.

Attention! If the contour's lead-in is not contour-symmetrical to the X axis (**A**), this may lead to incorrect cutouts (**B**).

Note: If you only cut certain segments in XYU mode on the contour, the U axis always rotates relative to the midpoint of the distance from the start to end segment in XYU mode (**C**).

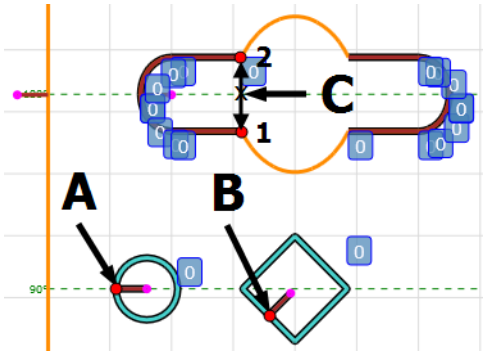


Fig. 7-186: Lead-ins in XYU mode

A	Contour's lead-in is symmetrical to the contour
B	Contour's lead-in is not symmetrical to the contour
C	Midpoint of the distance from the start to end segment in XYU mode

Y-offset

This option can be used with the enabled XYU mode to shift the through-hole away from the center in the Y-direction (**y**).

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7.16.4.4 Segment



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Cutting properties**

This function can be used to machine one or more segments of a contour with a different operating mode.

For instance, use the **Operating mode** option to set a process macro on sharp corners to avoid burn-off in corners. Another possible application for this option is when you wish to engrave part of a contour rather than cutting it, for instance.

- If no change is made to the relevant field, the cutting properties of the segment concerned are retained.
- If a new cutting property is selected in a field, this cutting property is assigned to all selected segments.



Tip: Use the **Shift** key to select more than one segment with the mouse. You can use the **Ctrl** key to deselect a segment from a multiple selection.

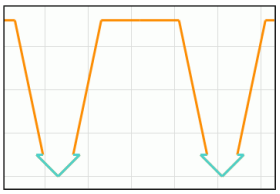


Fig. 7-187: Process macro 1 set on segments around sharp corners

Operating mode

- **Default:** The parameters for machining the material are assigned by the pre-processor during export of the cutting plan, but not before this stage.
- **Cutting:** The CW parameters are used for machining the material.
- **Operating mode 2:** This operating mode is used to make a quality cut (water jet cutting only).
- **Pulse:** The pulsed parameters are used for machining the material. Pulsed operation can improve the machining quality for certain materials or for tight contours by reducing the amount of heat input to the material (laser cutting only).
- **Engraving:** The cutting parameters for engraving the material are used. Closed inner contours that have this machining mode are no longer displayed as a cut-out in the Part Editor.
- **Process macros:** The relevant process macro is used for machining the material.
  - Laser process macro
  - Water process macro

Sensing OFF



Attention! If sensing is switched off, the cutting nozzle and part may collide.

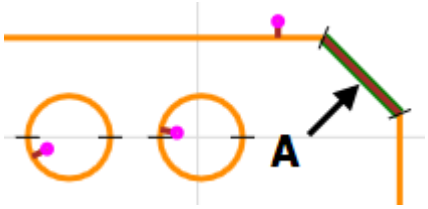
This option disables sensing on the machine. Switching off sensing on the machine is necessary or useful in the following situations:

- When cutting non-conductive material on a machine fitted with capacitive sensing (sensing MUST be off).
- In order to avoid scratching the material surface on a machine using tactile sensing (switching off sensing is optional).

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No lead-in or microjoint

This option is used to define segments on which no microjoint or lead-in must be set. Such segments are indicated by a green border (**A**).



Application:

- To avoid tilted parts due to postprocessor calculation in Part Nester.
- To avoid lead-in burring on the contact edge of bend parts.

  In the cutting technology settings you can use the **Detect contact edge** option to automatically set this option on a straight segment *(See: ›Segments without lead-ins/lead-outs or microjoint‹ Page 675)*.

Cut first

This option defines which segments on a contour to cut first.

If several segments of a contour are selected, they are connected to the segments in between by the shortest path. Connecting is also done via unselected segments if that is the shortest path.

The selected segments are highlighted in yellow and the corresponding attribute is displayed in the key (**A**).

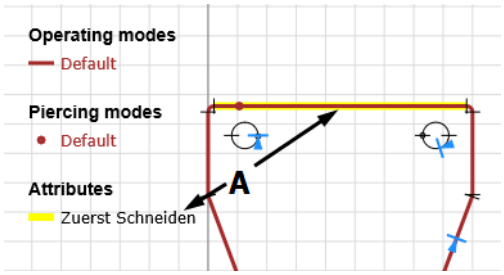


Fig. 7-188: Selected segment to be cut first

XYU mode (Tube Editor)

This option inhibits movement of the U-axis (rotary axis). The segment is cut solely by moving the cutting head in the X and Y-directions. This means that the segment is cut through the circular tube at right angles to the horizontal plane *(See: ›XYU mode‹ Page 688)*.

Insert segment

Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Cutting properties > Segment**

This function can be used to insert one or multiple segments on a contour and machined with a different operating mode than the contour.

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Remove segments

Segments can be removed again as follows:

- With the **Undo** edit function in the header if they were previously set.
- With the **Remove reference point** edit function in the **Geometry** menu.

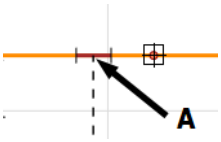


Fig. 7-189: Segment inserted on a contour

Operating mode

- **Default:** The parameters for machining the material are assigned by the pre-processor during export of the cutting plan, but not before this stage.
- **Cutting:** The CW parameters are used for machining the material.
- **Operating mode 2:** This operating mode is used to make a quality cut (water jet cutting only).
- **Pulse:** The pulsed parameters are used for machining the material. Pulsed operation can improve the machining quality for certain materials or for tight contours by reducing the amount of heat input to the material (laser cutting only).
- **Engraving:** The cutting parameters for engraving the material are used. Closed inner contours that have this machining mode are no longer displayed as a cut-out in the Part Editor.
- **Process macros:** The relevant process macro is used for machining the material.
  - Laser process macro
  - Water process macro

Sensing OFF

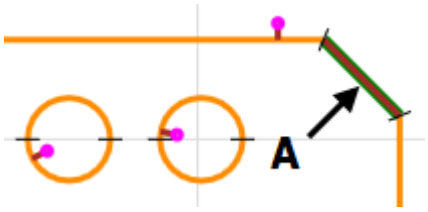
Attention! If sensing is switched off, the cutting nozzle and part may collide.

This option disables sensing on the machine. Switching off sensing on the machine is necessary or useful in the following situations:

- When cutting non-conductive material on a machine fitted with capacitive sensing (sensing MUST be off).
- In order to avoid scratching the material surface on a machine using tactile sensing (switching off sensing is optional).

No lead-in or microjoint

This option is used to define segments on which no microjoint or lead-in must be set. Such segments are indicated by a green border (**A**).



Application:

- To avoid tilted parts due to postprocessor calculation in Part Nester.
- To avoid lead-in burring on the contact edge of bend parts.

Length

Specifies the length of the segment to insert.

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7.16.4.5 Reference point



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Cutting properties**

This function can be used to set certain machine commands on reference points. Several commands at once can be used for each reference point. Reference points are located by default at the junction between two segments of a contour.



Note: If you wish to assign a machine command to a different point on the contour, you must set the required reference point in the relevant position on the contour using the **Add reference point** function in the **Geometry** menu *(See: ›Insert reference point‹ Page 465)*. Then you can use the **Reference point** function to assign the machine command to the reference point.

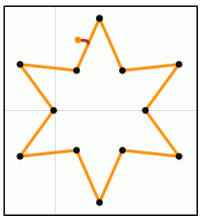


Fig. 7-190: Reference points on a star contour

Microjoint

This option is used to define the reference point as a microjoint.



Note: Microjoints can also be set on a contour using the **Set** function in the **Microjoint** group *(See: ›Manual‹ Page 725)*.

Dwell

This option causes the machine to wait at the reference point for a certain time before continuing the cutting process. The laser beam or water jet remains switched on during this time. The wait time is specified in the **Dwell time** cutting parameter. This option is used for:

- Blowing out molten material on laser machines in order to cut straight corners.
- Achieving a vertical water jet angle on water jet cutting machines in order to cut straight corners.

Optional stop

This option causes an interrupt at the position of the reference point before the cutting head is lowered. This option is enabled in ByVision using the **STOP SEL** function.

The operator can use this option when needing to intervene in the machining process to perform a task on the machine.

Looping

This option cuts a corner loop at the position of the reference point. Loops can only be cut at corners without a radius. The attribute is used to cut sharp corners.

Attention! Only use the attribute on reference points of outside contours, otherwise the corner loop will be cut inside the part.

Run FixMaster (ByJet only)

This option causes the pierce exit to be performed at reference point position and sets a gluing point at all reference points with the **Microjoint** point attribute on the same contour.

Note: Microjoints situated behind the last FixMaster execution point are ignored and no longer set.

Tip: To ensure that the **Run FixMaster** option is ideally set, use the **Manual** or **Automatic** functions in the **Microjoint** group.

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7.16.4.6 Change machining direction



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Cutting properties**

This function can be used to change the direction in which a contour is machined. The machining direction can be displayed using the **Special view** function. The beginning of a contour is represented with a bar and the end is represented with an arrowhead.



Attention! If the **Tool correction** option in the menu **Cutting technology** > **Entire technology** > **Settings** is set to **left** or **right**, the tool correction must also be changed after changing the machining direction.



Note: The machining direction is the direction in which the contour is machined on the machine.

7.16.4.7 Copy



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Cutting properties**

This function copies the following cutting properties of the highlighted contour.

- All properties defined in the **Contour** function.
- All properties defined in the **Segment** function.
- All properties defined in the **Lead-in/Lead-out** function.

You can use the **Insert** function to assign these cutting properties to other contours.



Tip: Use this function when making complex changes to cutting properties for multiple contours.

7.16.4.8 Insert



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Cutting properties**

Use this function to assigned the copied cutting properties to all highlighted contours.

7.16.5 Group: Machining sequence

Where do I find the group?

Select: **Modules > Part Editor > Cutting technology**

The functions in this group can be used to assign a machining sequence automatically or manually to the contours of a part.

7.16.5.1 Automatic

Where do I find the function?



Select: **Modules > Part Editor > Cutting technology > Machining sequence**

This function automatically assigns the machining sequence to the contours.



Note: If no contour is selected, the machining sequence is assigned to the entire part. If one or more contours are selected, the machining sequence is only assigned to this contour/these contours.

- **Complete**

  The machining sequence is only set on contours which do not have a machining sequence yet. Existing machining sequences are not re-assigned. The contour machining sequence is appended to the existing machining sequence.

  Note: Using this function may cause the outer contour not to be cut as the last contour in the machining sequence. To set the outer contour to the end of the machining sequence again, use the **Append** option in the **Manual machining sequence** docking dialog.

- **Settings**

  The options for setting the machining sequence can be defined in the **Machining sequence settings** docking dialog.

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7.16.5.2 Machining sequence settings

Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Machining sequence > Automatic**

You have a range of options for controlling how the machining sequence is assigned to contours. The chosen option or attribute can have a huge impact on the machining time for the part.

Mode

Constructing sequence

The machining sequence is set by the order in which the contours are constructed.

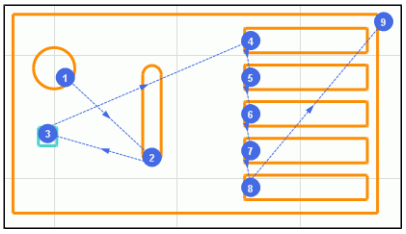


Fig. 7-191: Machining sequence set by constructing sequence

Optimized positioning path

The sequence for the contours is computed to give the shortest overall distance for all traversing paths together.

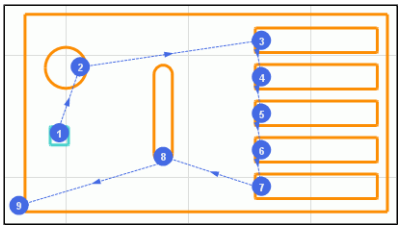


Fig. 7-192: Optimized machining sequence

Vertical - From left to right

The contours are processed vertically (up and down) starting from the lower left and progressing to the top right.

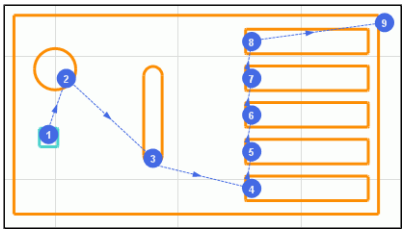


Fig. 7-193: Vertical machining sequence from left to right

Vertical - From right to left

The contours are processed vertically (up and down) starting from the lower right and progressing to the top left.

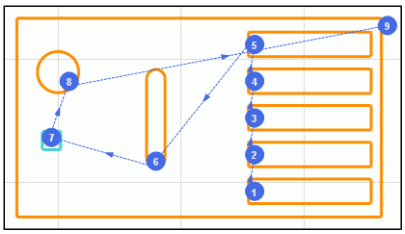


Fig. 7-194: Vertical machining sequence from right to left

Horizontal - From bottom to top

The contours are processed horizontally (from right to left) starting from the bottom right and progressing to the top left.

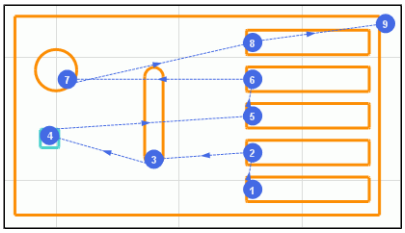
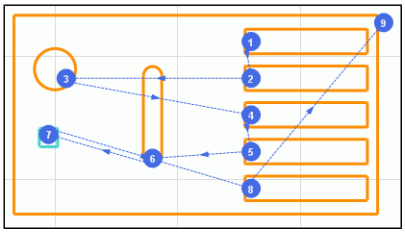


Fig. 7-195: Horizontal machining sequence from bottom to top

Horizontal - From top to bottom

The contours are processed horizontally (from right to left) starting from the top right and progressing to the lower left.



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Fig. 7-196: Horizontal machining sequence from top to bottom

- **Random order**

  The sequence for the contours is defined by a random sequence generator. Select **Random order** to avoid too much heat entering the material when laser-cutting tightly packed contours or heavy-gage sheets.

> **Note:** This setting usually results in the longest machining time.

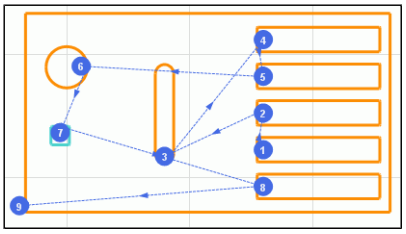


Fig. 7-197: Machining sequence with random order

Increment

This option can be used to set the increment between machining sequence numbers when the contours are actually machined.

- **Increment 1:** All contours are machined in accordance with the computed machining sequence (default).
- **Increment >1:** The actual machining sequence is applied to every nth contour of the computed machining sequence.

Using the increment option for a perforated sheet

When the increment is set to 2, only every second hole in the hole sequence is cut. The remaining holes are cut in the second pass in the given sequence.

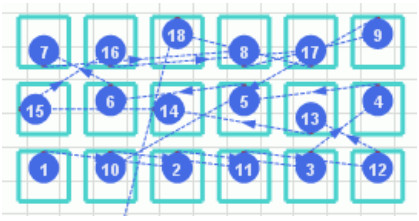


Fig. 7-198: Machining sequence for a perforated sheet with increment of 2



Tip: Use an increment >1 to distribute the generated heat more evenly in perforated sheets.

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Right lead-out first (ByTube)

This option cuts the right lead-out on a tube first.

The sequence of cutting is as follows:

1. Right lead-out
2. Inside contours
3. Left lead-out

If this option is not enabled, the cutting sequence is performed as follows.

1. Inside contours
2. Right lead-out
3. Left lead-out

Group by sectors

Use this option to divide the total area of the part into sectors. Inside each sector, the identified contours are processed as a group using the machining sequence set under **Mode**.

☞ **Note:** The sectors are displayed in a preview as blue and white areas defined by the dimensions entered in the **Grid X/Y** fields.

This option can be used for the following applications:

- To achieve a shorter positioning path for a part that has numerous different operating modes.
- For a part which contains numerous small inner contours in heavy-gage material, in order to distribute the heat input to the material more evenly. In this case, the **Random order** mode is recommended in order to distribute the sectors randomly over the part.

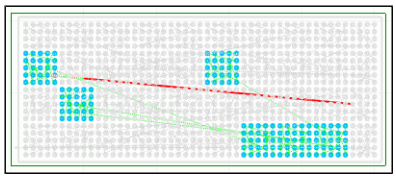


Fig. 7-199: Machining a perforated sheet by sectors

Grouping by operating mode, laser/water

The contours are processed in the order set for the operating modes. You can set the order to optimize the heat input to the material and to increase process reliability. Use the two arrow buttons to set the order of the operating modes.

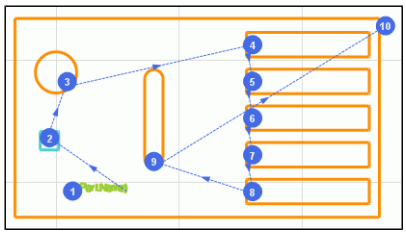


Fig. 7-200: Machining sequence based on the operating-mode order engraving, process macro 1 and cutting

Group by size

This option sets the machining sequence in the part by contour size. All inside contours are processed using the machining sequence set in **Mode**.

- **Small contours first**

  The machining sequence goes from the smallest inside contours to the biggest inside contours.

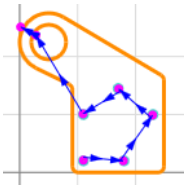


Fig. 7-201: Machining sequence with “Small contours first” mode

- **Large contours first**

  The machining sequence goes from the biggest inside contours to the smallest inside contours.

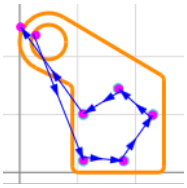


Fig. 7-202: Machining sequence with “Large contours first” mode

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7.16.5.3 Manual



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Machining sequence**

This function can be used to set the machining sequence manually for the contours of a part. The following options are available:

- Set the entire machining sequence manually.
- Post-edit an existing machining sequence.
- Insert new contours in an existing machining sequence.



Attention! When setting the machining sequence manually, make sure that the outside contour is always cut last. Otherwise it can cause collisions and imprecise cutting of inner contours if the part has already been severed from the sheet before cutting the inner contours.

The traversing path is indicated by a blue dotted line. The machining sequence is indicated by a blue number on the contour.

Each selected contour is highlighted and is given an ascending number. If a number has already been assigned to a contour, this contour is given a new number.



Note: A manually set machining sequence is not overwritten again by the automatic system. To set the machining sequence automatically on all contours, first delete the manually set machining sequences.



Tip: Before editing the machining sequence, first select the mode and then the contour that you wish to assign the selected mode to.

Mode

- **Append**

  The selected contour is added at the end of the machining sequence.
- **Insert**

  The selected contour is inserted in the machining sequence in front of the reference part.
- **Delete**

  The selected contour is removed from the machining sequence.

Reference contour

This button defines the reference contour. The reference contour is used for the **Insert** mode and is indicated by a red number.

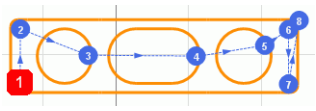


Fig. 7-203: Showing the reference contour (1) for manual insertion of a new contour in the machining sequence.

Inserting a contour in an existing machining sequence

1. Select the **Insert** option.
2. Select the **Reference contour** button.
3. Select the reference contour in front of which you wish to insert the new contour in the machining sequence. The reference contour is indicated by a red number.
4. Select the new contour. The contour is inserted before the reference contour.

Tip: You can also follow this procedure in order to assign a new machining sequence to contours that already have a machining sequence.

7.16.5.4 Remove

Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Machining sequence**

This function removes the machining sequence from the selected contour. If the contour lies between two contours in the machining sequence, the next contour in the machining sequence is joined to the one before it.

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7.16.6 Group: Lead-in/Lead-out

Where do I find the group?

Select: **Modules > Part Editor > Cutting technology**

The functions in this group can be used to assign the cutting technology for the lead-in and lead-out automatically or manually to the contours of a part.

7.16.6.1 Automatic



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Lead-in/Lead-out**

With this function, the lead-ins and lead-outs are automatically assigned to the contours.



Note: If no contour is selected, the selected lead-ins and lead-outs are assigned to the entire part. If one or more contours are selected, the lead-ins and lead-outs are only assigned to this contour/these contours.

- **Complete**

  The lead-ins and lead-outs are only set on contours which do not have any lead-ins and lead-outs yet. Existing lead-ins and lead-outs are not re-set.

- **Settings**

  You can define the type of lead-in/lead-out to use in the **Lead-in/lead-out settings** docking dialog.

7.16.6.2 Lead-in/lead-out settings



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Lead-in/lead-out > Automatic**

Define the contour lead-ins and lead-outs in this docking dialog box.

Mode

After Cutting Wizard

This option sets the lead-in and lead-out type on the part based on the Cutting Wizard selected in the **Cutting Wizard** section.

You can find information on the lead-in and lead-out type for the selected Cutting Wizard in **Administrator** -> **Cutting Wizard** in the **Lead-in type** column.

After manual configuration

This option sets the lead-in and lead-out type on the part based on the information in the **Lead-in** and **Lead-out** sections.

Lead-ins are displayed on the contour as a gray preview based on the appropriate setting (**A**).

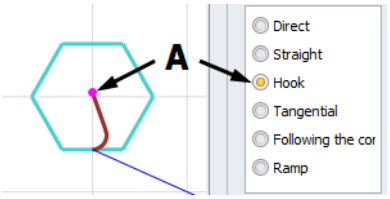


Fig. 7-204: Lead-in/Lead-out preview

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General

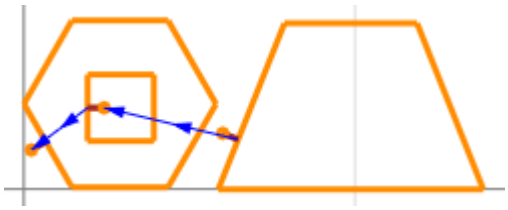
This area contains the options for setting lead-ins and lead-outs on a contour. Depending on what attribute is selected, it is possible to achieve dynamic positioning during the cutting process.

Position

Use this option to specify the position for automatic placement of the lead-ins and lead-outs.

- **Along the machining sequence**

  The software first calculates the shortest possible positioning path between the contours, and then places the lead-ins automatically so that they point towards the nearest contour. This prevents the tool traveling into the holes of contours that have already been cut when it is traveling in lowered positioning mode.



- **Coming from left**



- **Coming from right**



- **Coming from above**



- **Coming from below**



- **First segment**

  The lead-in is placed in the center of the first segment of the contour.



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Place lead-in in corners

This option places the lead-in of an outer contour in a corner of the contour.

☞ **Note:** The position of the lead-in depends on the setting in the **Position** option.



Fig. 7-205: Lead-in in a corner of the contour with **position coming from above**

Cutting Wizard

Name

The selected Cutting Wizard is displayed in the **Name** field. If there are several Cutting Wizards available, you can select another Cutting Wizard by clicking in the field.

☞ **Note:** When selecting a cutting wizard, the system remembers the cutting wizard that was last selected. If no cutting wizard is found by setting filters, and if you disable the filters again, this box displays the cutting wizard that was last selected.

Automatic

Enabling this box adopts the settings from Settings Manager. The filter definitions are hidden.

If no valid table is found, an error message is displayed to this effect.

Filter definitions

These filters can be used to restrict the choice of Cutting Wizards. The properties of the Cutting Wizard can be displayed and edited in the Administrator module in the **Cutting wizard** form *(See: ›Cutting Wizard‹ Page 116)*.

If a number of Cutting Wizards are found that match the filter criteria, the user must manually select the data record that has the appropriate cutting gas.



Note: The filter definitions is defined in the **Part properties** > **General** / **Cutting** in the **Material** and **Cutting process** sections.

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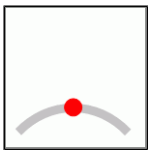
Lead-ins and lead-outs

Lead-in/Lead-out

The example uses of lead-ins given here are merely suggestions, and may differ in practice depending on the particular application. You can find information on the lead-in and lead-out for the particular material and thickness you are using in the Cutting Wizard in the **Lead-in type** column.

- **Direct**

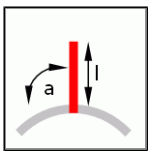
  The initial piercing cut is performed directly on the contour.

  This lead-in is mainly used on thin sheets up to about 1.5mm thick and for open contours, where impairing the contour does not matter too much. Rapid piercing directly on the contour makes this lead-in the choice for high-speed cutting. This is the default lead-out setting for laser machines.

- **Straight**

  A straight lead-in is set on the contour.

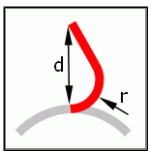
  

  - **length (l)** = length of the lead-in
  - **angle (a)** = angle of the straight line to the contour

  This lead-in is used for most contours. This lead-in causes the least damage to the contour but is rather slower than the **Hook** lead-in. This lead-in is also the best choice for microjoints and pulsed piercing. This is the default lead-out setting for water jet cutting machines.

- **Hook**

  This lead-in makes a hook shape with the contour.

  - **distance (d)** = perpendicular distance of initial piercing cut from contour
  - **radius (r)** = radius of the hook path taken to the contour.

This lead-in offers better dynamic performance compared with the **Straight** lead-in and can be used to reduce the heat build-up in thick sheets and difficult contours. The higher speed used for the lead-in means more damage to the contour.

- **Tangential**

  The lead-in is set in an arc tangentially to the contour.

> **Note:** If this type of lead-in is used, the position **Along the cut sequence** must be selected.



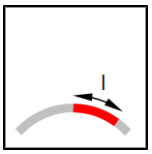
- **Length (l)** = length of the arc
- **radius (r)** = radius of the arc for the lead-in

- **Following the contour**

  The lead-in or lead-out is set directly following the contour.

  Application:

  As lead-out for water jet cutting so that the contour is reliably cut completely thanks to the drag. The pierce exit is performed by the specified length of the end cut after the contour lead-in.



- **Length (l)** = length of the lead-in or lead-out on the contour

- **Ramp**

  A straight lead-in is set with an arc at a tangent to the contour.

- **Length (l)** = total length of the lead-in including arc
- **Angle (a)** = angle of the straight line to the contour
- **Radius (r)** = radius of the arc

> **Note:** Only displayed if lead-ins are set manually.

Application:

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Improved influence on heat build-up during piercing.

- **Automatic**

  The type and position of the lead-in is set in correspondence to the contour shape. This means that each contour has the optimal lead-in and lead-in position assigned to it.



Note: Only displayed in the settings for automatic setting of lead-ins.

Operating mode

- **Default:** The parameters for machining the material are assigned by the pre-processor during export of the cutting plan, but not before this stage.
- **Cutting:** The CW parameters are used for machining the material.
- **Operating mode 2:** This operating mode is used to make a quality cut (water jet cutting only).
- **Pulse:** The pulsed parameters are used for machining the material. Pulsed operation can improve the machining quality for certain materials or for tight contours by reducing the amount of heat input to the material (laser cutting only).
- **Engraving:** The cutting parameters for engraving the material are used. Closed inner contours that have this machining mode are no longer displayed as a cut-out in the Part Editor.
- **Process macros:** The relevant process macro is used for machining the material.
  - Laser process macro
  - Water process macro

Segment side

This option defines which side of the contours the lead-ins are automatically set.

- **Normal**

  The largest contour is taken to be the outer contour for which the lead-in will be placed outside the contour. For all the other contours, the lead-in is set inside the contour.

- **Inverse**

  The largest contour is taken to be the inner contour for which the lead-in will be placed inside the contour. For all the other contours, the lead-in is set outside the contour.

> **Tip:** Select this function if the sheet is the work piece and the cut-outs are filled with filler parts.

- **Always outside**

  All contours are taken to be outer contours. The lead-in is always set outside the contour.

- **Always inside**

  All contours are taken to be inner contours. The lead-in is always set inside the contour.

Optimize machining direction

This option is used to adapt the contour machining direction to produce the minimum change in direction between the contour lead-in and the traversing path. This avoids unnecessary deceleration and increases the machining speed.

When the machining direction is changed, the tool correction is also changed automatically to the correct side (left/right).

> **Note:** This option is only used for the lead-in types **Direct** and **Hook**.

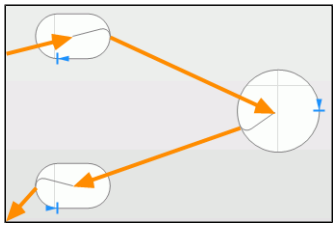


Fig. 7-206: Optimized contour machining direction to suit traversing path

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7.16.6.3 Manual lead-in/lead-out



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Lead-in/Lead-out**

This function can be used to edit, move or re-assign manually the lead-in and lead-out types for individual contours. The options are applied immediately to the selected contour.

Lead-ins are displayed on the contour as a gray preview based on the appropriate setting (**A**).

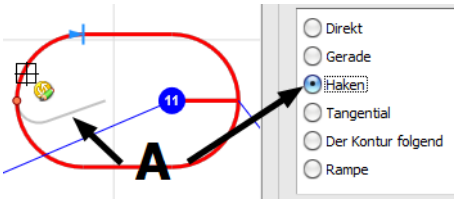


Fig. 7-207: Lead-in/Lead-out preview

- If a contour without a lead-in is highlighted, the dialog is opened in **Set** mode.
- If a contour with a lead-in is highlighted, the dialog is opened in **Edit** mode.



Note: Manually set lead-ins and lead-outs are no longer overwritten by the automatic system. To set lead-ins and lead-outs automatically on all contours, first delete the manually set lead-ins and lead-outs.



Tip: Use the **Shift** key to select more than one contour with the mouse. You can use the **Ctrl** key to deselect a contour from a multiple selection. The Snap functions can be used for placing the lead-ins.

Mode

The following options are available for changing lead-ins and lead-outs manually.

Edit

This option can be used to set new lead-ins on contours or edit existing lead-ins/lead-outs.

1. In the **Mode** area select the **Edit** option.
2. Then select the relevant contour. You can also select more than one contour at once.
3. Define the new attributes for the lead-in and lead-out.

Set

This option can be used to assign a predefined lead-in and lead-out to the contours.

1. In the **Mode** area select the **Set** option.
2. Define the lead-in and lead-out for the contours.
3. Select the relevant contours. The lead-in and lead-out are assigned to the selected contour.

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Alignment

Segment side

This option can be used to set the lead-in on the opposite side of a contour.

- **Left:** The lead-in is placed on the left-hand side of the contour viewed in the machining direction.
- **Right:** The lead-in is placed on the right-hand side of the contour viewed in the machining direction.

> **Note:** You can display the machining direction of the contours using the **Special view** function.

Lead-in/Lead-out

You use these options to assign the type of lead-in and lead-out manually to the selected contour. The options for the manual lead-ins and lead-outs are the same as those described for the **Lead-in/lead-out settings** function.

7.16.6.4 Remove



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Lead-in/Lead-out**

This function deletes all lead-ins and lead-outs from the selected contours.

> **Note:** If no contour is selected it deletes the lead-ins and lead-outs of contours of the part.

7.16.7 Group: Microjoint

Where do I find the group?

Select: **Modules > Part Editor > Cutting technology**

The functions in this group can be used to set microjoints on a contour or to remove them again.

Microjoints are areas of a contour that are not cut through. They keep the part joined to the material by a narrow web. Microjoints are used for materials in the thickness range 1-5mm. For larger thicknesses, the **weld joints** option must be used on the machine so that the parts can be broken out of the sheet easily.



Note: If the **Microjoint** option is disabled in the preprocessor, the width of the microjoints is defined by the parameters specified on the machine.

Microjoints prevent:

- Collisions with the cutting head arising from tilting parts.
- Small parts dropping down into the small-parts tray of the machine.
- Parts dropping out or tipping up during automatic sheet removal.
- Parts hitting the rake in an automation system.
- The cut termination being burned off when cutting stainless steel.

7.16.7.1 Automatic



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Microjoint**

This function automatically sets microjoints on the contour according to the number defined in the settings. The microjoints are distributed at equal intervals along the contour.



Note: If no contour is selected, the defined number of microjoints is assigned to all of the part's contours. If one or more contours are selected, the microjoints are only assigned to this contour/these contours.

- **Settings:** The number of microjoints to set can be defined in the **Microjoint automatic** docking dialog.

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7.16.7.2 Microjoint settings



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Microjoint > Automatic**

Define settings for automatic placement of microjoints in this dialog. The microjoints to be set according to the settings are displayed as gray crosses on the contour.

Press the **Apply** button to set the microjoints on the part and display them as black crosses.



Note: If no contour is selected, the microjoint settings are applied to all of the part's contours. If one or more contours are selected, the options are only assigned to this contour/these contours.

Mode

The options in this section determine how many microjoints are automatically set on the contours.

Distributed

Use this option to define the number of microjoints to place automatically on outside and inside contours. The positions of the microjoints are displayed in a preview as gray crosses on the contour. The number of microjoints is distributed evenly along the contour.

Microjoint

- **Number of outside contours**
  Defines the number of microjoints to place on outside contours.
- **Number of inside contours**
  Defines the number of microjoints to place on inside contours.
- **Min. contour dimension**
  Defines the minimum length of a contour as of which microjoints are placed on the contour.
- **Max. contour dimension**
  Defines the maximum length of a contour up to which microjoints are placed on the contour.

End point

Sets a microjoint at the end point of all selected contours.

No microjoints

This option disables automatic placement of microjoints.

Intelligent

This option places microjoints on the contour depending on segment sizes.

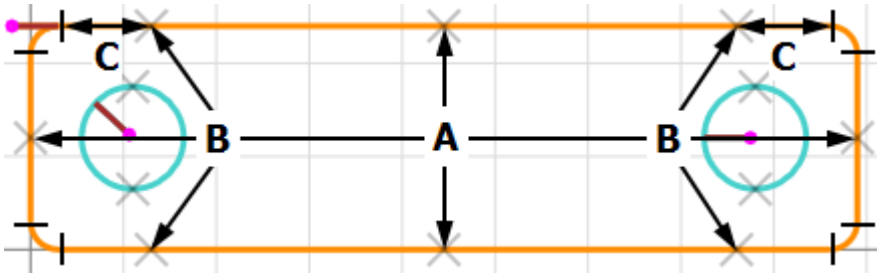


Fig. 7-208: Microjoints placed on segments of a contour

Segments

- **Min. segment length**

  If the segment is larger than this value, it is taken into account for placing microjoints (**A**).

- **Max. segment length**

  If the segment is smaller than this value, it is taken into account for placing microjoints (**A**).

- **Max. segment length with one microjoint**

  - If the segment is larger than **Min. segment length** and smaller than this value, a microjoint is placed in the middle of the segment (**A**).
  - If the segment is smaller than **Max. segment length** and larger than this value, an additional microjoint is placed on each segment at each beginning and end of the segment (**B**).

- **Distance to end of segment**

  This value specifies the distance (**C**) to the beginning and end of the segment where the additional microjoints are placed, if this is possible based on the other values of the **Max. segment length with one microjoint** option.

Run FixMaster (ByJet only)

The options in this section are used to set gluing points on the contours instead of microjoints.

Tip: If possible, position the pierce exit point in the corner of a contour. Re-piercing after placing gluing points causes least damage to the contour if performed in the corner of a contour.

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Active

Enabling this option causes a gluing point to be placed on all existing microjoints of the selected contour.

Before end

This option specifies the dimension at which the pierce exit is performed before finishing cutting in order for FixMaster to place the gluing points. The dimension should be at least 10 mm so that the part is still reliably held by the material.



Note: If the pierce exit point is moved close to the corner of a contour by the dimension entered in **Before end**, the pierce exit point automatically snaps to the corner.

7.16.7.3 Manual



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Microjoint**

This function can be used to set as many microjoints or FixMasters as you want on contours or to remove them.

 **Tip:** The Snap functions can be used for placing microjoints.

Set microjoint

1. Enable the **Set** function.
2. Left-click on the position on the contour in which you want to set a microjoint. The microjoint is indicated by a black cross on the contour.

Mode

Set

Sets the microjoint (**A**) on the selected contour.

If the **Execute FixMaster** option (**B**) is set on a contour, all microjoints are executed as gluing points (ByJet only).

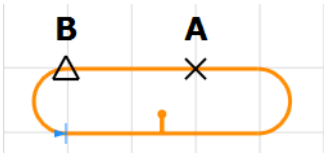


Fig. 7-209: Microjoint and FixMaster on a contour

Remove

Removes the selected microjoint or FixMaster from the contour.

Copy

Copies the microjoint or FixMaster from the contour of the selected part to the instances of the part in the cutting plan or in the entire job.

Run FixMaster (ByJet only)

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Specifies the pierce exit point (**B**) for executing the FixMaster and sets a gluing point on all microjoints defined (**A**).

Note: Microjoints situated behind the last FixMaster execution point are ignored and no longer set.

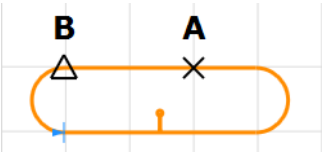


Fig. 7-210: Microjoint and FixMaster on a contour

General

Offset

This option can be used to set a microjoint at a certain distance to a point on the contour. A negative figure sets the microjoint before the selected point on the contour.

Set microjoint with offset

1. Enter the distance to the appropriate point on the contour in the **Offset** box.
2. Select the point on the contour. The microjoint is set at the specified distance to this point.

Tip: Use this option for water jet cutting to set a microjoint before the lead out. This means that the part stays attached to the sheet and is not damaged when the water jet pulls out.

7.16.7.4 Remove



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Microjoint**

Removes the microjoint from the contour.

To remove microjoints from a contour, enable the function and left-click on the relevant microjoint.

7.16.8 Group: Multihead

Where do I find the group?

Select: **Modules > Part Editor > Cutting technology**

The functions in this group can be used to set microjoints on a contour or to remove them again.

7.16.8.1 Automatic



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Multihead**

This function automatically assigns the available cutting heads to the contours and combines them in groups.

The cutting heads and groups assigned to the contours can be displayed with the special display **Tools** and **Groups**.



Note: If no contour is selected, all of the part's contours are taken into account when setting the cutting heads. If one or more contours are selected, only these contours are taken into account when setting the cutting heads.

Complete: The cutting heads and their groups are only set on contours to which no cutting heads have been assigned.



Attention! Tool correction (left/right) must always be set manually on all separated outer contours as **no** tool correction is set on open contours when automatically assigning cutting properties.

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7.16.8.2 Manual



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Multihead**

Use this function to assign cutting heads and groups manually to the contours or edit them. The cutting head and group assignment is applied to all selected contours immediately.

- **Tool**: Selection box to select the cutting head to use.
- **Group**: Input box to select the group number.
- **New group**: Button to assign the next free group number.



Tip: Use this function to optimize cutting heads and groups already assigned.

Assign contour to cutting head

1. Select the contours to assign to a cutting head.
2. Select the **Manual** function in the **Multihead** section.
3. Select the appropriate tool in the **Tool** selection box in the **Multi-head manual** dialog.

Assign contours to group

1. Select the contours to assign to a group.
2. Select the **Manual** function in the **Multihead** section.
3. Select the appropriate group number in the **Group** selection box in the **Multi-head manual** dialog.

Assign contours to new group

1. Select the contours to assign to a new group.
2. Select the **Manual** function in the **Multihead** section.
3. Press the **New group** button in the **Multi-head manual** dialog. The next free group number is assigned to the selected contours.

7.16.8.3 Remove multihead



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Multihead**

Deletes all assigned tools and groups from the contours.



Tip: To delete tools and groups from individual contours, select the contours and then the **Delete** function.

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7.16.9 Group: View

Where do I find the group?

Select: **Modules > Part Editor > Cutting technology**

This group contains all the functions that can be used to change the current view of the part.

The various functions and views are classified as follows:

Zoom functions

The zoom functions can be used to magnify or reduce objects in the graphics window.



Tip: You can also perform the zoom functions using the mouse thumb wheel.

Moving the view

The arrow symbols are provided in this group for moving the view of an object.



Tip: You can use the center mouse button to move the object freely in the current view. The view is moved by dragging and dropping with the mouse.

Special view

The **Special view** function is used to display various view options. The special-functions display can be specified by clicking on the arrow symbol and enabled/disabled by clicking on the symbol above it.

7.16.9.1 Special view



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > View**

This function can be used to display various information about the contour of the part. The information is shown/hidden by clicking on the relevant option.

The following information can be displayed:

- **Key:** *Displays in the top left of the workspace the operating and piercing modes used and the point attributes.*
- **Tag:** *Displays the tags that are set on the contours.*
- **Machining direction:** *Displays the machining direction for all contours of a part.*
- **Positioning path:** *Displays the traversing path taken by the tool to the next contour.*
- **Machining sequence:** *Displays the numbers of the machining sequence for the contours of a part.*
- **Point attributes:** *Shows the point attributes set on a contour.*
- **Tool correction:** *Shows the direction of tool correction as a black arrow.*
- **Tool:** *Shows the number of the tool with which the contour is cut for each contour.*
- **Groups:** *Shows the number of the associated group for each contour. All contours belonging to a group are cut simultaneously.*
- **XYU mode:** *Shows all contours cut in XYU mode with black outlines. If a Y offset is specified, it is displayed in a blue rectangle (See: ›XYU mode‹ Page 688).*

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7.16.10 Group: Part

Where do I find the group?

Select: **Modules > Part Editor > Cutting technology**

The functions in this group can be used to check parts and create a cutting plan from them.

7.16.10.1 Create cutting plan



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Part**

This function is used to create a cutting plan automatically from the part, which is then opened in the Part Nester module. Only contours with a technology are included when calculating sheet size. A sheet edge is also added.



Note: You can configure the job properties in the **Job properties** dialog. For instance, this lets you machine the part using another cutting machine or different technology settings.

Exporting a part as a cutting plan

1. Select the **Create cutting plan** function.
2. In the **Job properties** dialog box, specify the material, the thickness, and the machine that you want to use to cut the part.
3. Confirm your entries with **OK**.
4. Select the **Export** menu to export the part as a cutting plan.



Tip: To simulate the cutting process, use the **Simulator** function in the **Export** menu of the Part Nester.

Semi-finished product

This function also includes contours without a technology to create a cutting plan. No sheet edge is added.



Tip: This function is particularly suitable for subsequent cutting of semi-finished products (e.g. cutting ventilation holes in a pre-fabricated door).

7.16.10.2 Simulator



Where do I find the function?

Select: **Modules > Part Editor > Cutting technology > Part**

This function starts the simulator to simulate the part's cutting process for testing purposes.

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7.17 Menu: Machining time

Where do I find the menu?

Select: **Modules > Part Editor**

The machining time for the part is displayed in this menu.

Click on the **Calculate** button to calculate the machining time. The machining time is calculated from the assigned cutting parameters and the machine setup values.

If one of the values is not displayed, the machining time cannot be calculated. The setup value is assigned to the displayed machine in the **Cutting machine** form of the Administrator *(See: ›Creating a new cutting machine‹ Page 1342)*.

The machining time is composed of the times for the following operating modes:

- Positioning
- Cutting
- Pulse
- Engraving
- Start/end of cuts
- Process macros

If there is not a unique set of cutting parameters that can be assigned to the part, the specific file required must be defined in the **Cutting parameters** field. To do this click on the ··· button and select the parameter file you require from File Explorer.

Clicking on the black arrow opens a filter selection (**A**), which can be used to enlarge or reduce the number of files displayed.

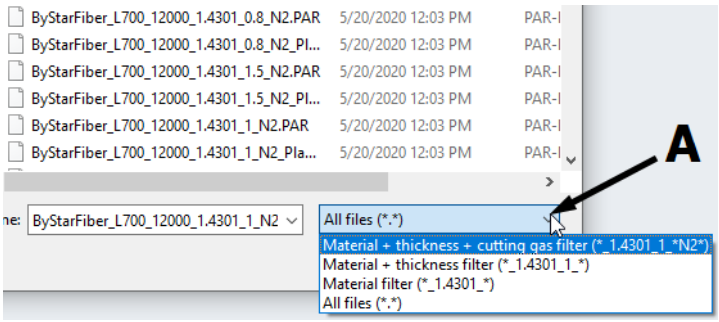


Fig. 7-211: Filter selection in File Explorer for cutting time calculation

Applications for calculating the machining time

Optimizing the machining time of the part

to check the machining time when the cutting technology is assigned manually, in order to optimize the machining time. In principle, an optimum balance between machining time and quality is achieved using the default parameters and the cutting rule sets.

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7.17.1 Calculate machining time



Where do I find the menu?

Select: **Modules > Part Editor > Machining time**

This function calculates the machining time for the current part and shows the result in various tables. If the part is changed, machining time is automatically deleted again.



Note: The machining time can only be calculated if the **Cutting parameters**, **Setup values** and **Machine** fields each contain an entry.

7.18 Drill

In this section you can assign a drill technology to circular contours inside parts.

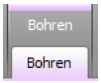


Fig. 7-212: Area display in the header

Note: The **Drill** section is only available if the relevant license has been purchased.

Menus

The following additional menus are displayed in the **Drill** section:

- Drill

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7.19 Menu: Drill

Where do I find the menu?

Select: **Modules > Part Editor**

The drill technology for the current part is configured in this menu.

The drill technology is required to machine holes, threads or recesses on the ByFlex machine.

In order to be able to nest the part in the Part Nester module, the part must be assigned a cutting technology.

Functional groups

The function bar contains the following functional groups:

- Selection
- Drilled hole
- Snap
- Special view
- Measure

7.19.1 Group: Drilled hole

Where do I find the group?

Select: **Modules > Part Editor > Drill**

This group contains all drill technologies that can be set on circular contours of a part.

7.19.1.1 Edit



Where do I find the function?

Select: **Modules > Part Editor > Drill > Drilled Hole**

7.19.1.2 Remove



Where do I find the function?

Select: **Modules > Part Editor > Drill > Drilled Hole**

This function removes the drill technology from holes.

- If no holes are highlighted, the function removes the drill technology from all holes in the part.
- If any holes are highlighted, the drill technology is only removed from the selected holes.

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7.19.2 Group: View

Where do I find the group?

Select: **Modules > Part Editor > Drill**

This group contains all the functions that can be used to change the current view of the part.

7.19.2.1 Special view



Where do I find the function?

Select: **Modules > Part Editor > Drill**

This function can be used to display various elements on the contour of the part. The elements are shown/hidden by clicking on the relevant option. The elements can be displayed separately or simultaneously.

You can display the following elements:

- **Process:** *Displays the drill processes set on a part's circular contours.*

7.20 Menu: Stack

Where do I find the menu?

Select: **Modules > Part Editor**

This menu is used to set stacking information for BySort.

Functional groups

The function bar contains the following functional groups:

- Selection
- Gripper
- Snap
- View

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7.20.1 Add automatically



Where do I find the function?

Select: **Modules > Part Editor > Stacking > Gripper**

This function automatically sets suitable gripper tools on the part.

If there are already gripper tools on the part, the program prompts you to **Replace existing gripper information**.

- **Yes**

  This deletes all existing gripper tools and replaces them with new ones.

- **No**

  The existing gripper tools are not deleted. Any additional, suitable gripper tools are added to the part.

- **Cancel**

  This cancels the **Automatic** function.

If it is not possible to assign a tool defined for the machines to the part, a dialog is shown with the message ***No gripper tool found for selected machine. Search for all available gripper tools?***. Select **Yes** to search for all gripper tools for the part in BySoft CAM.

7.20.2 Add / Edit



Where do I find the function?

Select: **Modules > Part Editor > Stacking > Gripper**

In this dialog you can add gripper tools to parts or edit existing gripper tools on a part.

When positioning a tool, the center of gravity of the tool is displayed as a red dot. Click on the center of gravity or drag the gripper tool out of the geometry display to position at the center of gravity.

Depending on the situation, grippers are displayed in the following colors:

- **Green:** Part can be gripped without any problems.
- **Orange:** Part can be gripped but is unstable and may tip over.
  - Position gripper at center of part.
  - Use a different gripper tool.
- **Red:** The part cannot be gripped.
  - Part is too heavy for the gripper.
  - Gripper is positioned on a contour.
  - Gripper is too far from the center of gravity.

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Gripper tool 1

This box displays the currently selected gripper tool 1 and its properties.

Tool

Selection box for gripper tools. The tools are created in Administrator *(See: ›Gripper tool‹ Page 178)*.

Note: For non-magnetic material, only vacuum grippers are displayed.

X/Y

Position of gripper tool on part.

If the gripper tool has multiple grippers, the center of the grippers should be at the center of gravity of the part.

Angle of rotation

You can use this function to rotate gripper tool 1 through a specified angle.

Optimum part stability is achieved when the longest axis of the gripper tool coincides with the longest axis of the part.

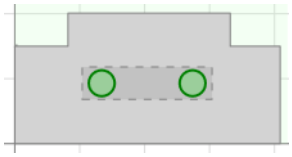


Fig. 7-213: Stably positioned gripper tool

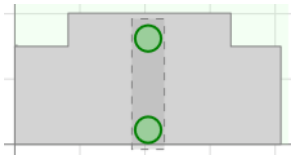


Fig. 7-214: Unstably positioned gripper tool

Tip: The angle of rotation for the gripper tool can also be set with **Ctrl** + **mouse wheel** before positioning.

Axis value

This value defines the axis value of the selected axis for multi-part grippers. It can be used to customize grippers for multiple parts.

Note: This option is only active for multi-part grippers.

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Gripper tool 2

Gripper tool 2 is used if the part needs to be gripped with two gripper tools (tandem pick). This may be the case with very long or very heavy parts.

When nesting, the tandem part can be rotated by **0°** or **180°** in Part Nester.

Active

Activates the 2nd gripper tool for positioning on the part.

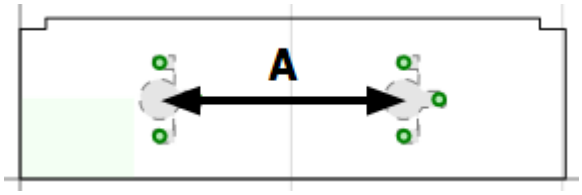
Tool

Selection box for gripper tools. The tools are created in Administrator *(See: ›Gripper tool‹ Page 178)*.

Note: For non-magnetic material, only vacuum grippers are displayed.

ΔX

Positions gripper tool 2 on the part at the specified distance (**A**) to gripper tool 1 in the X direction.



Tip: The distance between gripper tools can also be set using **Shift** + **mouse-wheel** before positioning.

Angle of rotation

Use this option to rotate gripper tool 2 through a specified angle.

Optimum part stability is achieved when the longest axis of the gripper tool coincides with the longest axis of the part.

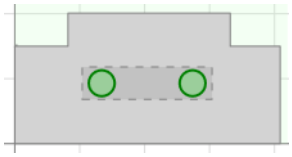


Fig. 7-215: Stably positioned gripper tool

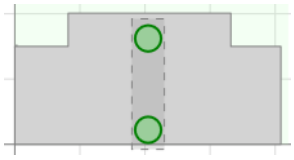


Fig. 7-216: Unstably positioned gripper tool

Tip: The angle of rotation for the gripper tool can also be set with **Ctrl** + **mouse wheel** before positioning.

Part rotated

This rotates the tandem part including the gripper tools by 90°. This allows the part to be rotated by a discrete angle of rotation of **90°** or **270°** when nesting in Part Nester *(See: ›Discrete angles of rotation‹ Page 1210)*.

Tip: If a non-rotated gripper tool is also created on the tandem part, the part can also be nested in Part Nester using a discrete angle of rotation of **0°** or **180°**.

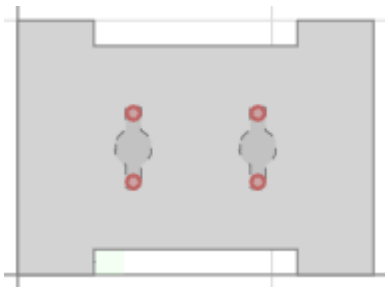


Fig. 7-217: Tandem part not rotated

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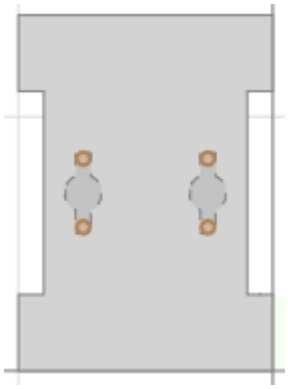


Fig. 7-218: Tandem part rotated

Axis value

This value defines the axis value of the selected axis for multi-part grippers. It can be used to customize grippers for multiple parts.

☞ **Note:** This option is only active for multi-part grippers.

Apply

This button applies the gripper settings to the part

7.20.3 Delete



Where do I find the function?

Select: **Modules > Part Editor > Stacking > Gripper**

This function deletes the current gripper tool from the part.

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7.20.4 Prioritize



Where do I find the function?

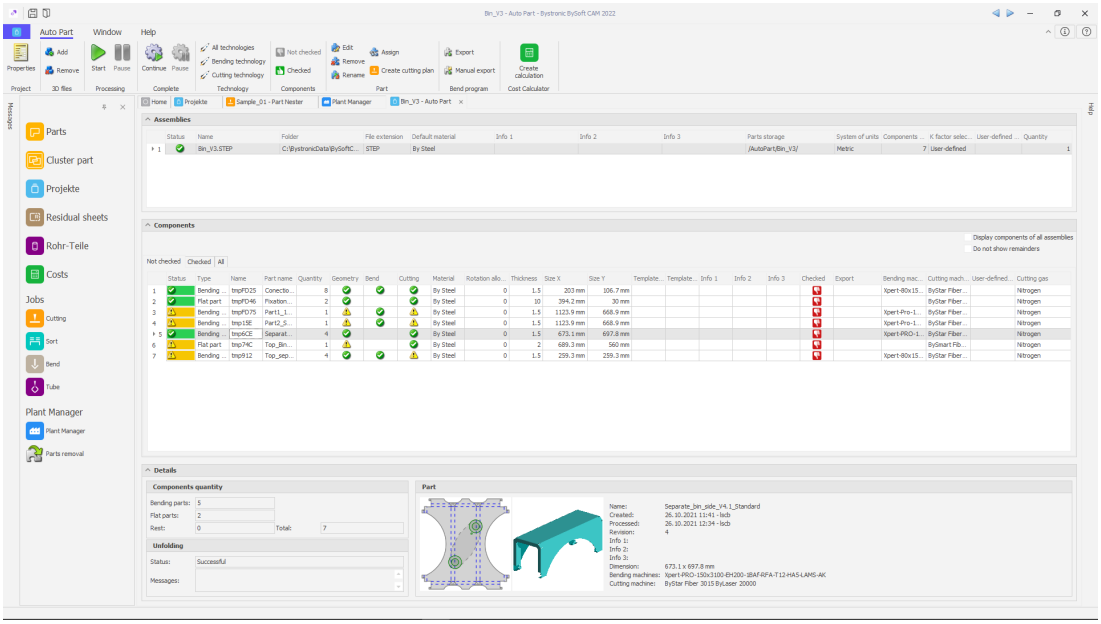
Select: **Modules > Part Editor > Stacking > Gripper**

This function sets the current gripper tool in first place on the tool selection list.

8 Module: Auto Part

Where can I find the module?

Select: **Modules >**



You can use this module to automate the following processes:

- Separate parts from 3D assemblies.
- Assign cutting and bending technology to parts.
- For bending parts, export the bending plan to the bending machine.

Tip: Click on an area of the header to call up the **Shortcuts** function in the context menu. The **Shortcuts** dialog can be used to define keyboard shortcuts for all functions of the current module.

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8.1 File functions

The functions in this menu relate to the files and user interface of the module.



Functions for the current project

These functions relate solely to the current project.

- New
- Open
- Save
- Save as...
- Save all
- Save as template
- Save as default template
- Settings
- System settings
- Close

8.1.1 Project properties

How do I open this dialog box?

Select: **Auto Part > New Project**

This dialog is displayed while creating a new project.

Template

Name

A template for the new project can be selected in this box. If no template is selected, the settings for the project components are taken from the Settings Manager.

General

Name

Enter a name for the project in this box.

> **Note:** The name is essential for the program to be able to save the record.

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8.1.2 Save as template



Where do I find the function?

Select: **Auto part >** 

This function can be used to save the current Auto Part file as a template with a custom name in the **Templates** folder. You can create additional subfolders below this folder for saving templates.

 **Note:** Auto Part templates are managed in **Object Manager**.

 **Tip:** Creating Auto Part templates can cut the time needed for importing parts as new assemblies as recurring settings do not need to be customized.

8.1.3 Save as default template



Where do I find the function?

Select: **Auto Part >** 

Use this function to save the current Auto Part file as the **Default** in the **Templates** folder. This template is used whenever you create a new Auto Part file *(See: ›Project properties‹ Page 753)*.

 **Note:** Auto Part templates are managed in **Object Manager**.

 **Tip:** Use the default template to specify the default settings for new projects in the properties.

8.1.4 Settings



How do I open this dialog box?

Select: **Auto Part >** 

In this dialog box you can define all settings for the AutoPart module.

- **Default parts storage**

  Defines the default storage for all imported parts.

  To define the storage location of the files, enter the path to the parts folder in the Subfolder field or select the folder using the button in the Find folder dialog box.

- **Module**

  Defines the unfolder for use in Auto Part. **Bystronic Unfolder** is selected by default. This usually achieves the best results.

- **From Settings Manager**

  The test criteria for the geometry and the cutting and bending technology are determined by the Settings Manager in Administrator.

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Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

8.2 Menu: AutoPart

Where do I find the menu?

Select: **Modules > Auto Part**

This menu contains all the functions and views for the AutoPart module.

Functional groups

The function bar contains the following functional groups:

- 3D files
- Processing
- Complete
- Technology
- Components
- Part
- Bend program

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8.2.1 Assemblies

Where do I find the function?

Select: **Modules > Auto Part**

This section shows all imported assemblies.

Before the start of processing, you can modify the **Material** and **Parts storage** for each assembly. Additional information about the assembly can be entered in the columns **Info1-3**. The information is applied to all separated parts of the assembly.

What each column means:

Status

Displays the status of the assembly.

Symbol	Description
	New assembly was added but has not been processed yet.
	Assembly is being processed.
	Assembly was processed successfully.
	An error occurred while reading or unfolding the assembly and caused processing to be aborted.

Name

Displays the name of the assembly.

Folder

Displays the path of the imported assembly.

Default material

Displays the material of the assembly. The material can be changed using a drop-down box.

File extension

Displays the file extension of the imported assembly (STP, SAT, IGS, etc.).

Info 1-8

Additional information about the current assembly can be entered in these boxes. The entries in the Info boxes are applied to every part of the assembly.

Info

Extra information about the current object can be entered in these fields.

Parts storage

Defines where the parts are stored in the BySoft CAM parts database. Click in the field to select where parts are stored in the parts database.

System of units

The system of units for the current object is defined in this field (mm/inches)

Number of components

Defines the number of instances of the assembly. The number of parts in the assembly is multiplied by this number.

K-factor selection

A drop-down list can be used to determine where the K-factor is taken from.

- **From file**

  The K-factor is read from a STEP file. The file must have the Bystronic STEP format.

- **From Settings Manager**

  The K-factor is determined based on the material and thickness from the Settings Manager.

- **User-defined**

  The K-factor is determined by entering a factor in the **User-defined K-factor** field.

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8.2.2 Components

Where do I find the function?

Select: **Modules > Auto Part**

Use this section to manage the components of the selected assembly. Depending on their check status, components are displayed on the **Not checked**, **Checked** or **All** tabs.

- **Display components of all assemblies**

  Displays the components of all assemblies in the **Components** section. If the function is disabled, only the components of the selected assembly are displayed.

- **Do not show remainders**

  Hides all parts in the **Components** section that are not cutting or bending parts.

What each column means:

Status

Displays the status of the component.

Symbol	Description
	OK: Part was created with complete technology. The part can be produced.
	Warning: Part was created with warnings in unfolding or technology.
	Error: Technology on part is incomplete or this is not a BySoft CAM part.
	Not available: Could not create an unfolding.
	Technology not up-to-date: The technology is not set or is not up-to-date. 1. Select the **Edit** function to set the technology manually. 2. Save the part and stop editing. 3. Run the **All technologies** function again.
	Component in queue: Component is in the queue and is locked for editing. The component can be removed from the queue!
	Component in processing: The component cannot be edited and removed from the queue.

Type

Displays the type of the component. There following types are available:

- **Rest:** Not a BySoft CAM part
- **Flat part:** A BySoft CAM part
- **Bending part:** A BySoft CAM part with bending information

Name

Displays the name of the component in the assembly. The name cannot be changed.

Part name

Shows the name of the part. The name can be changed.

Quantity

Shows the required production quantity of a part.

The number of parts can be changed.

Geometry

Displays the status of the geometry.

Bending

Displays the status of the bending technology.

Cutting

Displays the status of the cutting technology.

Material

Displays the name of the material used.

The material can be changed.

Thickness

Displays the thickness of the part.

Thickness can be changed.

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Rotation allowance

This field defines the increment of the angle of rotation in degrees. The part can be rotated during nesting through this angle.



Tip: A rotation allowance can be entered by clicking on the input field or by selecting a set angle value via the left arrow button.

Size X

Shows the dimension of the part in the X direction.

Size Y

Shows the dimension of the part in the Y direction.

Template part cutting

Displays the template part for the bending technology.

Template part bending

Displays the template part for the cutting technology.

Info 1-8

Displays additional information on the order. The information is adopted by the assembly.

Checked

Shows the check status of the part.

Export

Shows whether the part was exported as a bending part.

Bending machine

Defines the machine to bend the part with.

Cutting machine

Defines the machine to cut the part with.

8.2.3 Details

Where do I find the function?

Select: **Modules > Auto Part**

This section displays information about the components, the unfolding, and the part of the selected assembly.



Note: If there are multiple selections, the details of the last assembly are shown.

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8.2.4 Group: Project

Where do I find the group?

Select: **Modules > Auto Part**

This group contains all the functions for the AutoPart project.

8.2.4.1 Properties



How do I open this dialog box?

Select: **Modules > Auto Part > Project**

This docking dialog box displays the properties of the current Auto Part project. The properties can be displayed and edited on the appropriate tab. The following tabs are available:

- Process steps
- Unfolding
- File options
- Bending technology
- Cutting technology
- Geometry
- General

Process steps

How do I open this tab?

Select: **Modules > Auto Part > Project > Properties**

This tab is used to define the procedure when automatically importing 3D files.

Checkpoints

This section is used to define when to stop component processing for a check.

Note: If no option is selected, component processing is not interrupted.

- **Check after importing the geometry**
  Automatic processing is interrupted after importing the geometry (default setting).
- **Check after setting the geometry**

Automatic processing is interrupted after setting the bending technology.

- **Check after setting the cutting technology**
  Automatic processing is interrupted after setting the cutting technology.

Automatic processing

This section is used to define which steps to perform automatically.



Note: If no option is selected all steps are performed automatically.

- **Set bending technology**
  In automatic processing, the program tries to set the bending technology automatically (default setting).
- **Set cutting technology**
  In automatic processing, the program tries to set the cutting technology, automatic automatically (default setting).
- **Export bending parts**
  In automatic processing, the program tries to export a bending part automatically.

Unfolding

How do I open this tab?

Select: **Modules > Auto Part > Project > Properties**

This tab is used to define the default settings for unfolding bending parts.

Settings

- **K-factor**
  The K-factor determines the position of the imaginary neutral fiber in the sheet. Values between 0 and 1 can be used (default 0.5).
  - **Use defined K factor**
    If the option is enabled, the defined value is sent to **Bystronic Unfolder** for unfolding the files. If **SolidWorks Unfolder** is selected in the Auto Part settings, the option has no effect.
- **Maximum part thickness**
  Maximum thickness for parts detected as a geometry. Thicker parts are created as type **Other** (default 25 mm).
- **Maximum bending radius**
  Maximum bend radius for parts detected as a geometry (default 15 mm).

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File options

How do I open this tab?

Select: **Modules > Auto Part > Project > Properties**

This tab is used to define the procedure in the event of conflicts during the import process.

☞ **Note:** Only one option at a time may be selected for conflict resolution.

Conflict resolution

If when importing the parts it is found that a file name already exists, the following options are available:

- **Ignore file**
  The file with the same file name is not imported.
- **Overwrite file**
  The existing file is overwritten by the file with the same file name.
- **Add serial number to file name**
  The file with the duplicate file name is imported with a serial number in the file name (default setting).

Settings

- **Create one subfolder per assembly**
  If this option is enabled, one subfolder is created in the parts storage for each assembly that contains parts. The name of the subfolder is taken from the name of the assembly.

Bending technology

How do I open this tab?

Select: **Modules > Auto Part > Project > Properties**

This tab is used to define the settings for automatic assignment of the bending technology.

Settings

Calculate bend allowance

If this option is enabled, the program checks whether bend allowance for parts needs to be recalculated based on certain criteria.

- If so, bend allowance is recalculated and applied to the geometry.
- If not, the preset bend allowance is applied without modifying the geometry.

After calculating bend allowance, the Bending Wizard is used to calculate the bending sequence.

> **Note:** If the option is not enabled, only the Bending Wizard is used to calculate the bending sequence.

Bending machines

This view displays a list of all bending machines available for bending.

In the defined order of machines, the program checks whether it is possible to create a valid bending sequence for the respective bending part. If so, the bending part is assigned to the machine.

If not valid bending sequence is found, the bending part is assigned to the last machine in the list. A warning is output for the bending part in the **Components** section.

> **Note:** At least one bending machine must always be selected. Otherwise, it is not possible to identify a valid bending sequence.

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- **Add**
  Adds a new bending machine to the list.
- **Remove**
  Removes the selected bending machine from the list.
- **Up**
  Moves the selected bending machine up one place in the list.
- **Down**
  Moves the selected bending machine down one place in the list.

Cutting technology

How do I open this tab?

Select: **Modules > Auto Part > Project > Properties**

This tab is used to define the appropriate cutting machine for automatic assignment of the cutting technology.

Settings

- **Calculate cutting time**
  Automatically calculates the cutting time for the separated parts.
- **Cutting wizard automatic**
  Automatically selects the cutting wizard using the configuration in the Settings Manager.

Cutting machine

The selected cutting machine determines from which cutting machine to apply the technology to the separated parts.

Geometry

How do I open this tab?

Select: **Modules > Auto Part > Project > Properties**

This tab is used to define the geometry settings for the parts.

Settings

- **Move to origin**
  This option moves the bottom left corner of the bounding rectangle of each new part to the origin.

Cutout

Max width

This option defines the maximum width starting from which a cutout will be set (**A**).

Min depth

This option defines the minimum depth starting from which a cutout will be set (**B**).

Distance

This option defines the distance between the cutout and the contour (**C**). By default, the distance is equal to the kerf width.

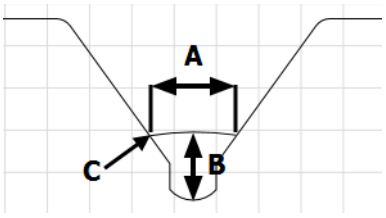


Fig. 8-1: Cutout on an outer contour

Clustering

These options are used to automatically assign the **Part in cluster** attribute to a part with a certain size. This part can be used for cluster nesting in Part Nester.

- **Detect parts for clusters**

  Enables automatic detection of clusters.

- **Max. width**

  Defines the maximum width of a part's bounding rectangle up to which the **Part in cluster** attribute is assigned to the part.

- **Max. height**

  Defines the maximum height of a part's bounding rectangle up to which the **Part in cluster** attribute is assigned to the part.

☞ **Note:** If the dimension of a part's bounding rectangle corresponds to the specified maximum height or width, the **Part in cluster** attribute is also assigned to the part.

Text

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With these options, any text can be placed on the imported parts by entering coordinates. The text is defined in a separate dialog box. This is the same as for the **Add text** function in the PartEditor*(See: ›Formatting‹ Page 492)*.

Several texts can be created in one list. The texts can be added, edited, or deleted again.



Note: The texts are added to all imported parts at the defined positions. The cutting technology **engraving** is applied to all texts.

General

How do I open this tab?

Select: **Modules > Auto Part > Project > Properties**

This tab is used to define how materials are used.

Material

- **Using material from CAD file**

  If this option is enabled, the program searches BySoft CAM for the material specified in the CAD file and, if it exists, uses.

  If the option is disabled or the specified material does not exist in BySoft CAM, the program uses the standard material for the assembly.

Gripper

- **Set grippers automatically**

  If this option is enabled, a gripper is set on each part for sorting if possible.

Log file

- **Active**

  Activates creation of a log file in CSV format. The log file can tell the ERP system which files could not be imported, for example.

- **Folder**

  Specifies the folder in which the log file is saved. By default, the log file is saved to “C:\BystronicData\BySoft CAM\AutoPartProjects\Log\”.

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Description

Columns index	Meaning	Comment	Example:
1	Event	Unfolded Added Bending Technology Added Cutting Technology Exported Part updated Part removed Part assigned see also below	Unfolded
2	Date	current date	09.07.2019
3	Time	current time	10:42:33
4	3D filename	Fullfileame of 3D component/assembly	C:\temp\sample.STEP
5	Component status	OK (green) Warning (yellow) Error (red) N.A. (white) Unknown (grey)	OK
6	Component type	Flat part Bend part Others	Bend Part
7	Component name	Name of the 3D part (not filename)	Holder
8	Part name	from BySoft CAM part	Holder_1
9	Part Order info	from BySoft CAM part	A324
10	Part Info 1	from BySoft CAM part	
11	Part Info 2	from BySoft CAM part	
12	Part Info 3	from BySoft CAM part	

Columns index	Meaning	Comment	Example:
13	Status geo-metry	OK Warning Error	OK
14	Status bending	OK Warning Error Unknown	OK
15	Status cutting	OK Warning Error Unknown	OK
16	Status export	OK Warning Error	OK
17	Material		1.4301
18	Thickness		2
19	Measurement system		Metric
20	Checked		Not checked

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Name of attribute

Normally, the name of the part from the assembly is used to unfold a 3D file.

In this section you can define attributes from a 3D file to replace the part name for unfolding with the corresponding value of the attribute.

Example:

If the part name is to consist of an identification number, the **Identity_Number** attribute must be entered in the **Group** field (see figure). If the option is enabled, the unfolded part of the assembly is given the part name **7053WK-07-02-51**.

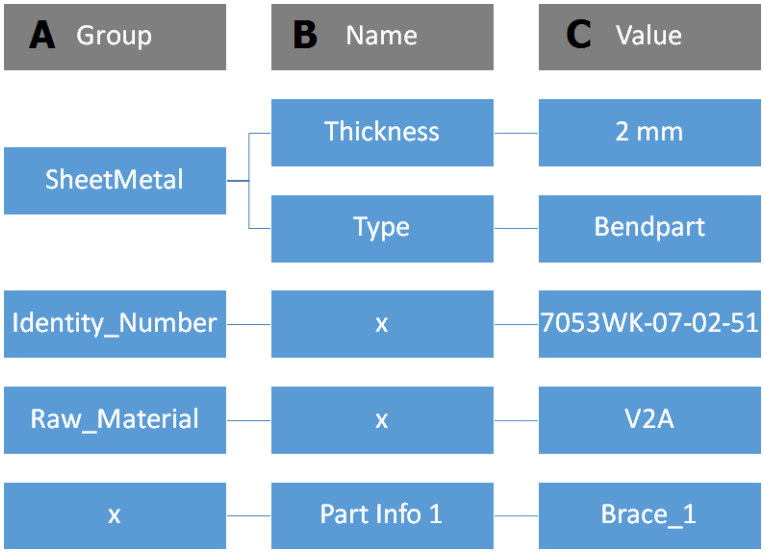


Fig. 8-2: Attributes in a tree structure of group (**A**), name (**B**) and value (**C**)

- **Active**
  Enables replacement of the file name with the value of an attribute.
- **Name**
  Name of the attribute whose value is to be used as the file name.
- **Group**
  Name of the group whose value is to be used as the file name.

Add QR code

In this section you can define a QR code with various details and a specific size for each extracted cutting or bending part. If possible, the QR code is placed in the middle of the part. In certain cases you will need to correct it manually.

Values

Defines the content of the QR code. Click on the required values in the drop-down field to define the specifications for the QR code.



Tip: In Part Editor, you can add additional information to the QR code for each part using the **Edit parts ID** function *(See: ›Edit‹ Page 504)*.

Size

Sets the size of the placeholder for the QR code on the part. The size of the engraved QR code depends on the amount of content. The more information a QR code contains, the larger it is.

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8.2.5 Group: 3D files

Where do I find the group?

Select: **Modules > Auto Part> Auto Part**

This group contains all the functions for adding and removing assemblies.

8.2.5.1 Add 3D files



Where do I find the function?

Select: **Modules > Auto Part> Auto Part > 3D files**

This function can be used to add 3D files to the Auto Part module as assemblies from a 3D-CAD. You can select multiple files at once.

The following formats are supported:

SolidWorks (*.sldprt;*.sldasm)
ACIS (*.sat)
IFC 2x3 (*.ifc)
IGES (*.iges;*.igs)
Inventor Part (*.ipt)
Inventor Assembly (*.iam)
Parasolid (*.x_t;*.x_b)
ProE/Creo Part (*.prt;*.xpr)
ProE/Creo Assembly (*.asm;*.xas)
Solid Edge Part (*.par;*.psm)
Solid Edge Assembly (*.asm)
STEP (*.step;*.stp)

Fig. 8-3: Auto Part import formats

☞ **Note:** This function can only be used if no processing is active.

8.2.5.2 Remove 3D files



Where do I find the function?

Select: **Modules > Auto Part> Auto Part > 3D files**

This function deletes the assemblies and their components selected in the **Assemblies** section.

☞ **Note:** It does not delete the parts created in the parts database!

8.2.6 Group: Processing

Where do I find the group?

Select: **Modules > Auto Part> Auto Part**

This group contains all the functions for processing assemblies.

8.2.6.1 Start



Where do I find the function?

Select: **Modules > Auto Part > Auto Part > Processing**

This function starts processing the selected assemblies. Processing consists of three processes:

1. 3D import of assemblies using the BySoft CAM add-in.
2. Analyze components and separation of parts.
3. For bending parts, create an unfolding.
4. Import DXF in BySoft CAM and save contour as part.



Note: During processing of assemblies, the function is disabled. Once assembly processing has finished, the function is enabled again.



Attention! This function can only be run for an Auto Part project. If you run the function in a second Auto Part project while it is already running, a warning is displayed.

8.2.6.2 Pause



Where do I find the function?

Select: **Modules > Auto Part > Auto Part > Processing**

This function is used to interrupt the processing process. Processing of the current assembly is completed first.

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8.2.7 Group: Complete

Where do I find the group?

Select: **Modules > Auto Part> Auto Part**

This group contains all the functions for the completing parts.

8.2.7.1 Continue



Where do I find the function?

Select: **Modules > Auto Part > Auto Part > Processing**

This function resumes component processing that was previously stopped. The checkpoint for stopping processing can be defined in the **Properties** on the **Process steps** tab *(See: ›Process steps‹ Page 764)*.

8.2.7.2 Pause



Where do I find the function?

Select: **Modules > Auto Part > Auto Part > Processing**

This function is used to stop processing components. This **Continue** function resumes component processing.

8.2.8 Group: Technology

Where do I find the group?

Select: **Modules > Auto Part> Auto Part**

This group contains all the functions for the setting technologies on components.

8.2.8.1 Sett all technologies



Where do I find the function?

Select: **Modules > Auto Part> Auto Part > Technology**

This function automatically sets the bending and cutting technology for the part.

8.2.8.2 Set bending technology



Where do I find the function?

Select: **Modules > Auto Part> Auto Part > Technology**

This function automatically sets the bending technology for the part.

8.2.8.3 Set cutting technology



Where do I find the function?

Select: **Modules > Auto Part> Auto Part > Technology**

This function automatically sets the cutting technology for the part.

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8.2.9 Components

Where do I find the group?

Select: **Modules > Auto Part> Auto Part**

This area shows all extracted components of an assembly.

- **Display components of all assemblies**
  Shows the components of all assemblies in an overall view.

8.2.9.1 Not checked



Where do I find the function?

Select: **Modules > Auto Part> Auto Part > Components**

This function is used to flag the part as not yet checked. This is the default status of all new parts in an assembly.

☞ **Note:** Parts with this status are displayed in the **Components** section on the in **All** and **Not checked** tabs.

8.2.9.2 Checked



Where do I find the function?

Select: **Modules > Auto Part> Auto Part > Components**

This function is used to flag the part as checked. This enables an overview of the current status of work. In terms of function, flagging has no effect on part processing.

☞ **Note:** Parts with this status are displayed in the **Components** section on the in **All** and **Checked** tabs.

8.2.10 Group: Part

Where do I find the group?

Select: **Modules > Auto Part> Auto Part**

This group contains all the functions for editing parts.

8.2.10.1 Edit part



Where do I find the function?

Select: **Modules > Auto Part > Auto Part > Part**

This function lets you edit the selected part in the Part Editor. The Auto Part module is locked as long as the part is open in Part Editor.

Use this function for the following changes to a part:

- For the cutting technology to customize the machine, thickness or material if no cutting rule set was found for this combination.
- For the bending technology to customize tools, bending sequence, tool setup plan, etc.



Note: After modifying the part, the appropriate technology must be reassigned to the part. This updates the status changes for geometry, bending and cutting.

8.2.10.2 Remove part



Where do I find the function?

Select: **Modules > Auto Part > Auto Part > Part**

This function removes the component's link to the saved part. After removing, the component's type is set to **Other**.



Tip: Use this function to assign a different part to the component.

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8.2.10.3 Rename part



Where do I find the function?

Select: **Modules > Auto Part > Auto Part > Part**

This function can be used to change the name of the part in the **Part name** column. The column is located in the **Components** section.

8.2.10.4 Assign part



Where do I find the function?

Select: **Modules > Auto Part > Auto Part > Part**

This function can be used to assign a different part to a component.

☞ **Note:** This function can only be used if the part was previously removed from the component.

8.2.11 Group: Bend program

Where do I find the group?

Select: **Modules > Auto Part> Auto Part**

This group contains all the functions for editing parts.

8.2.11.1 Export bending program



Where do I find the function?

Select: **Modules > Auto Part> Auto Part > Bending program**

This function automatically exports the bending program for the selected bending parts to the output directory of the bending machine.

8.2.11.2 Export bending program manually



Where do I find the function?

Select: **Modules > Auto Part> Auto Part > Bending program**

This function automatically exports the bending program for the selected bending parts to a custom directory.

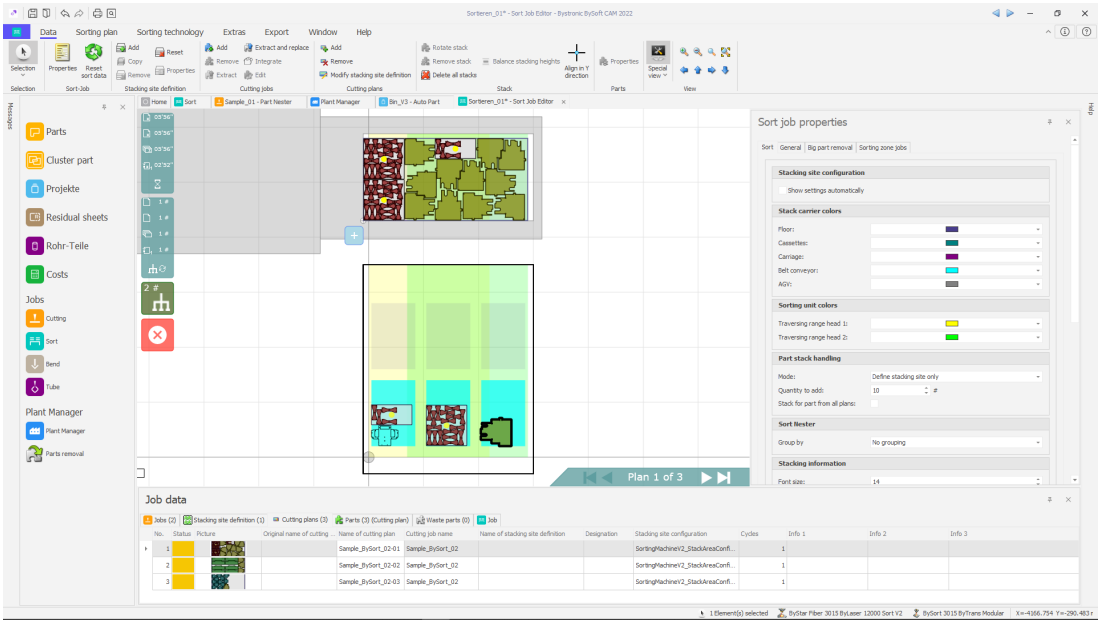
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9 Module: Sort Job Editor

Where do I find the menu?

Select: **Modules > Sort Job Editor**



In the Sort Job Editor module you can create sorting plans from several jobs. Sorting information is exported together with cutting plans by the postprocessor and imported into the machine.

Tip: **Ctrl+C** copies a stack to the clipboard. **Ctrl+V** pastes the copied stack onto the selected stacking site definition.

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Context menu functions

Creating a sorting plan

1. Select the **Data** menu.
2. Select the **Add** function in the **Cutting plans** section.
3. In the **Add cutting plans** dialog box, select the **Select all** option to assign all cutting plans of the job to a sorting plan.
4. In the lower selection box, select the stacking site configuration on which to stack the parts and press OK to confirm.
5. The cutting plans are assigned to the stacking site configuration and displayed in the **Job data** dialog on the **Cutting plans** tab.
6. In the **Nesting** menu, select the **Automatic** function. The sorting plan will be automatically created for each cutting plan based on the stacking site definition.

Copy a stack to a different stacking site definition.

1. Select the stack on the stacking site definition.
2. Press **Ctrl+C** to copy the stack to the clipboard.
3. Select the stacking site definition to paste the stack into on the **Stacking site definitions** tab.
4. Press **Ctrl+V** to paste the stack into the stacking site definition. The stack is pasted into the stacking site definition in its original position.

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Icons in the workspace

Various icons to call up information about the sorting plan are displayed on the left in every sorting plan. The display is refreshed automatically as soon as any change is made to the sorting plan.

Symbol	Description
	Sorting time for all the stacking site definitions in the job.
	Sorting time for the stacking site definition that is currently selected. If there is only one stacking site definition, the sorting time will be the same as that for the job.
	Sorting time for the cutting plan that is currently selected. If a cutting plan contains multiple cycles, sorting time is only displayed for one cycle.
	Icon for sorting times section
	Number of tool changes for all the stacking site definitions in the job.
	Number of tool changes for the stacking site definition that is currently selected. If there is only one stacking site definition, the number of tool changes is the same as that for the job.
	Number of tool changes for the cutting plan that is currently selected. If a cutting plan contains multiple cycles, the number of tool changes is only displayed for one cycle.
	Icon for tool change section
	Displays the number of gripper tools used in the sorting plan. **Tip:** Click on the icon to display the **Check gripper tools** dialog with detailed information about the gripper tools.

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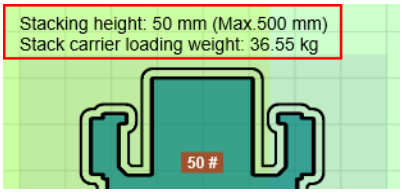
Symbol	Description
	Displays the status of the stacking site definition. The status is updated in real time. ■ **Symbol 1:** The stacking site definition is OK. The stacking site definition can be exported. ■ **Symbol 2:** The stacking site definition contains errors. The stacking site definition can be exported. ■ **Symbol 3:** The stacking site definition contains errors. The stacking site definition cannot be exported.  **Tip:** Click on this symbol to call up the **Check stacking site definition** dialog with detailed information about the status of the stacking plan.
	Navigation bar for switching between multiple cutting plans.

Stack carrier information

Additional information can be displayed for each stack carrier in the special display.

Information above the stack carrier:

- The height of the stack and the maximum possible stack height on the stack carrier.
- The weight of all stacks on the stack carrier.



Information below the stack carrier:

- Additional information about each part from the **Order information** column.



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Interactive buttons

When a part is selected in the stacking area, interactive buttons will be displayed. The buttons will remain visible as long as at least one part is selected.

Button	Description
	The number of parts in the stack is defined manually. Click the button to define the number of parts dynamically.
	The number of parts in the stack is defined dynamically. Click the button to define the number of parts manually.
	The parts are stacked with a small offset inside the broken line. To disable it, click on the button.
	The parts are stacked without an offset. Click the button to stack the parts with an offset.
	Adds new cutting plans to the sorting plan.
	Deletes the cutting plan from the sorting plan. To delete it, select the cutting plan and click on the button.

☞ **Note:** If the **Always display interactive buttons** option is enabled in the Part Nester settings, the buttons will be displayed permanently *(See: ›Always display interactive buttons‹ Page 906)*.

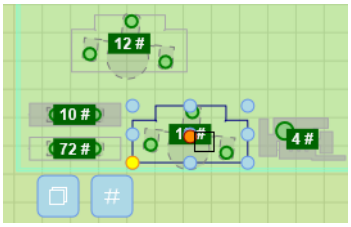


Fig. 9-1: Buttons displayed for a selected stack

Overlapping stacking areas

If there are overlapping stacking areas on the sorting system, each stacking area has its own button (**A**). Click this button to activate the respective stacking area.

The activated stacking area is indicated by a black line (**B**) and the disabled stacking area is indicated by a broken line (**C**).

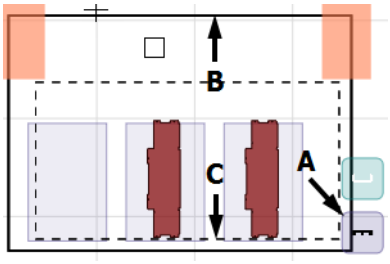


Fig. 9-2: Pallet stacking area

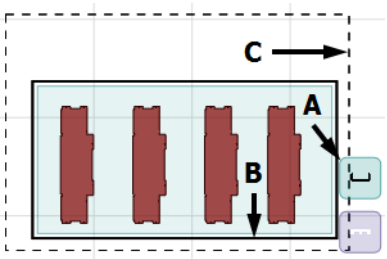


Fig. 9-3: Cassette stacking area (DUH ByTrans Cross)

Note: Stacks cannot be moved between stacking areas directly. To move a stack to another stacking area, first delete it and then drag it to another stacking area on the **Parts** tab.

Button	Description
	Activates the cassette stacking area.
	Activates the pallet stacking area.

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Overlapping stacks

If multiple stacks in parts groups are arranged on top of each other, you can highlight them with an additional border in the special display. To do so, select the **Display parts groups** option in the **special view**. The number of all parts in the group is displayed in the center of the parts group.

Hover the mouse over the group to display a pop-up dialog with information about the parts group. A maximum of 3 stacks can be displayed in the pop-up dialog.

Click once to move the entire group and twice to move a single stack out of the group. A single stack selection is indicated by a green dot.

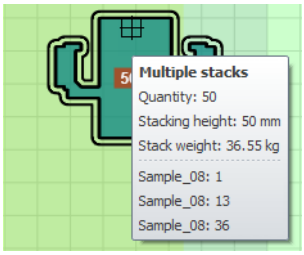


Fig. 9-4: Highlighted stack in a parts group

9.1 File functions

The functions in this menu relate to the files and user interface of the module.



Functions for the current job

These functions relate solely to the active job.

- Save
- Save as...
- Export
- Close

General functions

These functions are available to run irrespective of the active job.

- Open
- Save all
- Import
- Settings
- System settings

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9.1.1 Export sort job



Where do I find the function?

Select: **Sort Job Editor >**

This function exports the current job. This file can be re-imported into BySoft CAM using the **Import job** function. The following export formats are supported:

- BySoft CAM (*.sortjob)



Tip: Use this function to transfer jobs between different BySoft CAM workstations that do not have access to the same database.

Export job

1. In the **Export job** dialog box, select the folder in which you wish to save the job.
2. In the **File name** field, enter the name that you wish to save the job under (optional). The job name is used by default.
3. Confirm the selection with **OK**. The job is saved in the folder under the specified file name.

9.1.2 Import sort job



Where do I find the function?

Select: **Sort Job Editor >**

This function can be used to import a sort job in another format into BySoft CAM. All parts of an imported sort job are embedded in the job and are not saved in the parts database.

The following job formats are supported:

- BySoft CAM (*.sortjob)

Import sort job

This example describes the basic procedure for importing sort jobs.

1. Select the **Import job** function.
2. In the **Import job** dialog box, select the relevant file and confirm the selection with **Open**.
3. In the **Job properties** dialog box, confirm the job details with **OK**. The job is opened in Part Nester.
4. Use the **Save** function to save the sort job in the job database. The job is now available for further editing.



Tip: The function can also be launched from the quick access toolbar *(See: ›Quick access toolbar‹ Page 11)*.

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9.1.3 Settings



Where do I find the function?

Select: **Sort Job Editor>**

In this dialog box you can define the view settings for the **Sort Job Editor** user interface and the options for logging data about nesting operations.

Use the **Reset** button to restore the default settings for the tab.



Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

Tabs in the Settings dialog box:

- View
- Options
- Commands

9.1.3.1 View

How do I open this tab?

Select: **Sort Job Editor >** **> Settings**

This tab contains options for customizing the user interface of the **Sort Job Editor** module. Personal settings are saved separately for each user.

Use the **Reset** button to restore the default settings for the tab.

What each option does:

- **Show grid**

  Displays a square grid in the workspace. The grid helps with orientation.

- **Show axes**

  Displays the origin of the object as axes crosshairs.

- **Show crosshairs**

  Displays the cursor additionally as crosshairs. The crosshairs are displayed over the full workspace.

- **Grid size**

  Specifies the minimum grid size for the optical grid When zooming in, the grid is displayed down to this minimum size.

- **Snap grid**

  If the **Align with grid** snap function is enabled, the value specified here defines the snap grid pitch, i.e. the gap between snap points. The default value is 1mm. The grid can be used for measurement functions or placing text, for example.

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Setting the color

Colored parts

This option can be used to enable/disable the colored display of parts in the cutting plan and in the **Parts** tab. If this option is enabled, all parts are displayed in different colors to distinguish them more easily. If this option is disabled, all parts are displayed in a gray tone.



Attention! Do not use luminous colors in the color preferences because these cause eye strain after a prolonged period.



Note: When customizing the color options, some elements will no longer be visible if you select identical or similar colors for different color options.

The color preferences are used to set the color display of the background, axes and grid in the Part Nester module. The elements in a job can also be displayed in particular colors.

9.1.3.2 Options

How do I open this tab?

Select: **Sort Job Editor >** **> Settings**

Job name generator

Use the job name generator to automatically generate names for new jobs. If this option is enabled, a new job name is automatically input in the **Name** box in the **Job properties** dialog.

Tip: The use of a variable for **Job number** or **Date** prevents duplicate job names from being generated.

Active

Activates the job name generator.

Available variables

This section displays all the variables available for creating a job name. The variables can be entered in the **Name** field.

Name

Input the name and variables for the job name in this box.

Example

Displays the job name based on the definition in **Name** and **Current job number**.

Current job number

Lets you change the counter value. The number to be assigned next is always shown. If the counter value is changed, job numbering continues consecutively starting from the new number.

Attention! If you change the counter value manually, there is a risk that duplicate job names will be generated. This means that cutting plans may be overwritten when exported to the machine.

Press **Save** to apply the changed counter.

9.1.3.3 Commands

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How do I open this tab?

Select: **Part Nester >**  **> Settings**

This tab contains options for displaying buttons.

Always display interactive buttons

If this option is enabled, the interactive buttons are always displayed when selecting one or more parts in the cutting or sorting plan. If this option is disabled, the buttons are not displayed for the selected part until the space bar is pressed. The buttons remain visible so long as the space bar is pressed.

9.2 Menu: Data

Where do I find the menu?

Select: **Modules > Sort Job Editor**

In this menu, you can nest plans fully automatically based on the configured settings, and you can edit the sheets, parts and plans in the job.

Functional groups

The function bar contains the following functional groups:

- Sort-Job
- Stacking site definition
- Cutting jobs
- Cutting plans
- Stack
- Parts
- View

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9.2.1 Group: Sort-Job

Where do I find the group?

Select: **Modules > Sort Job Editor > Data**

This group contains all the functions for configuring sort job properties.

9.2.1.1 Reset sort data



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Sort job**

This function is used to reset invalid sort data.

9.2.1.2 Sort job properties



Where do I find the group?

Select: **Modules > Sort Job Editor > Data > Sort Job**

This function displays the properties of the current sort job on the tabs.

Sort

How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > Sort job > Sort job properties**

This dialog is used to configure settings for displaying the stacking site, sorting unit, and part stack.

Stacking site configuration

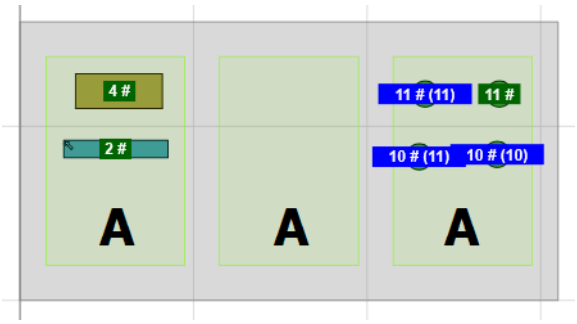
Show settings automatically

When switching stacking site definitions on the **Stacking site definition** tab, the appropriate settings are displayed in the **Stacking site definition settings** dialog. If this option is disabled, you must select the **Properties** function in the **Stacking site definition** group to display the stacking site definition.

Stack carrier colors

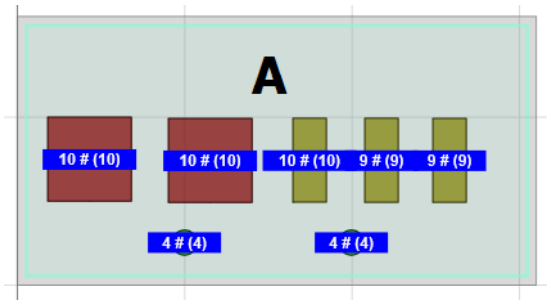
Pallets

Defines the color of the pallet places (**A**).

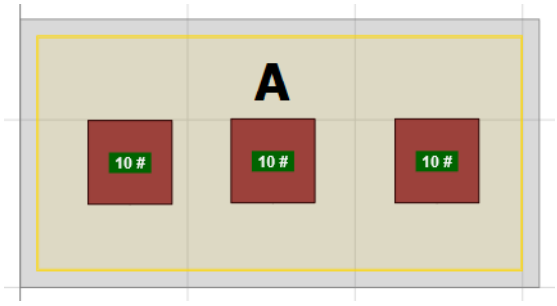


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- **Cassettes**
  Defines the color of the cassette places (**A**).

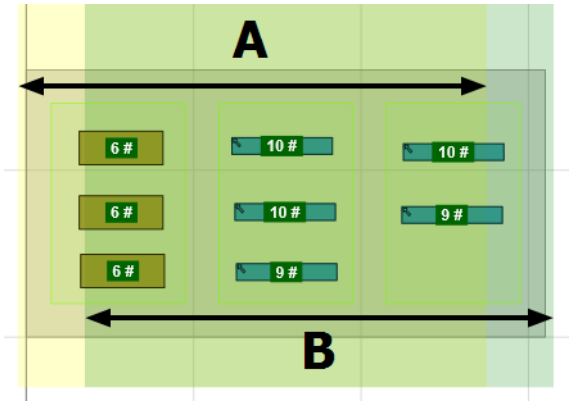


- **Carriage**
  Defines the color of the carriage places (**A**).



Sorting unit color

- **Head 1 range of travel**
  Defines the color of the range of travel of sorting head 1 (**A**)
- **Head 2 range of travel**
  Defines the color of the range of travel of sorting head 2 (**A**)



Part stack handling

- **Mode**
  - **Max. possible stack height**
    The maximum possible stack height is used for the stacks. If the number of parts is less than the maximum possible stack height, only one part stack can be set.
  - **Balanced stack heights**
    All available instances of a part are distributed evenly among all stacks.
  - **Add defined part quantity**
    Only a certain number of parts per stack is stacked, taking maximum stack height into account.
  - **Define stacking site only**
    Only a stacking site is defined for the part. This is the default setting.

> **Note:** By default, the option **Stacking site only** should be used as this results in the fastest sorting time. The other options should only be used if you need a certain number of parts per stack.

- **Quantity to add**
  Defines the number of parts per stack when **Add defined part quantity** mode was selected.
- **Stack for part from all plans**
  If a part is manually set as a stack and its instances are spread out across different cutting plans, the instances from the various cutting plans are sorted onto this stack.

> **Note:** Maximum stack height is defined in the part properties and for the stack carrier in the **Sort** section in Administrator.

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Sort Nester

- **Group by**
  - **No grouping**
    This is the default setting for Sort Nester.
  - **Order information**
    Stacks with the same order information are nested together.
  - **Info 1-3**
    Stacks with the same Info 1–3 are nested together.

Stacking information

The display settings for stacking information are defined in this section. The stacking information is displayed below the stacking site definition in the Sort Editor workspace.

- **Font size**
  Defines the maximum font size in pixels. Font size auto-adjusts up to this size when zooming in the workspace. At larger zoom levels it stays this size.
- **Field 1-3**
  Defines the contents of the display for the respective field. You can select columns **Order information** and **Info 1-3** on the **Parts** tab of the sort data. If the **Display nothing** attribute is selected, no stack information is displayed.

Sorting program

- **Allowance between parts**
  Defines the allowance per part to compensate for the difference in stack height due to burrs on the part and the gap between parts. The allowance corresponds to an empirical value and can be determined as follows using parts that have already been cut:

  Measured height of the part stack divided by the number of stacked parts minus part thickness.

Sequencing strategy

The sequence strategy determines which automatically determined removal sequence is used to sort cutting plan parts. The automatic removal sequence is defined for the cutting plan in the **Sorting technology** menu *(See: ›Set automatically‹ Page 845)*.

Consecutive

Sorting of the cutting plan starts alternately from the cutting plan side closest to the stacking site to the cutting plan side furthest from the stacking site.



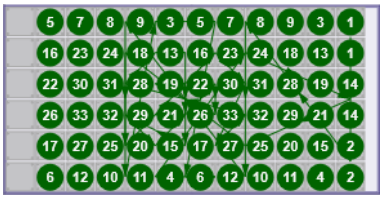
From inside

Sorting of the cutting plan starts in a spiral from the inside out.



From outside

Sorting of the cutting plan starts in an inward spiral from outside.



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Save sorting information on export

Activate this option when connecting to a robot system to remove parts from stack carriers by robot.

The information is stored in a separate file that can be read by the robot system. It contains all information about the current stacking situation. By default, the file is saved in the ***SortInfos*** directory under the default path. The default path is defined in **System settings** -> **Data storage**.



Note: During export, a Save dialog appears in which you can copy the file name from the sort job or create a user-defined one.

Cutting plan loading position

In some cases, especially with rotated machines, the cutting plans are not aligned with the origin of the cutting machine. The limits of the sheet stops may be on the other side of the shuttle table, which may result in a different position for sheets with smaller formats and affect the accessibility of parts for the sorting heads.

With these options it is possible to use a predefined orientation of cutting plans and a user-defined position of cutting plans in the order so as to improve accessibility of parts for the sorting heads.

- **Alignment**

  Defines the alignment of the sheet on the shuttle table. By default, **Bottom Left** is set. Changes to this setting are displayed directly in the machine layout preview.

  When **User-defined** is selected, the bottom two options are enabled. This allows you to set a user-defined sheet origin for the X and Y position.

  Example:

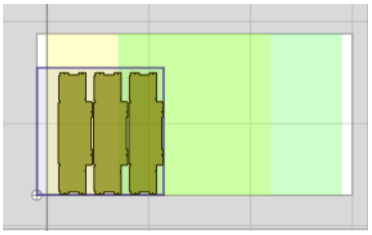


Fig. 9-5: Default setting, Bottom Left

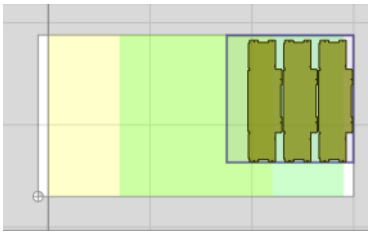


Fig. 9-6: Top Right setting

- **User-defined X position**

  Defines the sheet origin for the X position.

- **User-defined Y position**

  Defines the sheet origin for the Y position.

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Gripper tool stations

These options define whether the tools in the gripper tool stations are assigned automatically or manually. Specific settings can be made for each gripper tool station.

- **Gripper tool station**

  This field is used to select the gripper tool station for the particular sorting head to which the settings below apply.

  The name of the gripper tool stations is defined in Administrator under **System data** -> **Sorting** -> **Sorting unit** -> **Manage tool station** *(See: ›Gripper tool stations‹ Page 173)*.

- **Automatic composition**
  - If the option is **enabled**, the program automatically assigns the sorting tools to the gripper tool station.
  - If the option is **disabled**, the tools can be assigned manually to the respective gripper tool station in the **Composition** section.

- **Composition**

  This section is used to assign the tools to the selected gripper tool station. You can only select as many tools as the selected gripper tool station has tool slots.

> ☞ **Note:** If no tool is selected for the gripper tool station when this option is enabled, the program automatically assigns the sorting tools to the gripper tool station.

General



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > Sort job > Sort job properties**

General

The general information describes the job and the system of units that are used. When the job is saved, various data is created automatically under the **Created**, **Changed** and **Revision** options, which cannot be changed manually.

- **Name**

  Displays the name of the job. The job name can only be changed subsequently in the **Object Manager**.

- **Path**

  Shows the path to the object.

- **Created**

  Displays the date and time at which the object was created and the name of the user who created it.

- **Changed**

  Displays the date and time at which the object was last saved and the name of the user who saved the object.

- **Revision**

  Displays the current revision number for the object. The revision number is increased by a count value every time the object is saved.

- **Description**

  You can enter a more detailed description of the current object in this field.

- **Info 1–8**

  Extra information about the current object can be entered in these fields.

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

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Cutting process

This section displays the machine with which to process the cutting jobs. The information about the machine is needed in order to assign the cutting technology and to calculate the machining time.

☞ **Note:** Information about the machine and cutting gas are used in the export settings preprocessor in the **Cutting Wizard** section as a filter criterion for selecting the Cutting Wizard *(See: ›Filter definitions‹ Page 1129)*.

Machine

This field displays the machine that shall be used to machine the object. Clicking on the arrow symbol on the right lets you select a different machine from a list. This list only contains those machines that are configured in the system. If you require a new machine, you must create it in the **Administrator** module in the **Cutting machines** form *(See: ›Cutting machine‹ Page 102)*.

Sorting process

Sorting unit

This field displays the sorting unit assigned to the cutting machine.

⚠ **Attention!** When the sorting unit is changed, all existing batches in the sort job are deleted.

Reset

This button deletes all sorting plans created.

Gripper tools

This field assigns the gripper tools available in Administrator to the selected sorting unit.

☞ **Note:** Make sure that the selected tools are also present on the sorting unit.

Big part removal



How do I open this tab?

Select: **Modules > Sort Job Editor > Data > Sort job > Sort job properties**

On this tab you can configure the properties for removal of big parts.

Default values

- **Big parts**
  Location where big parts are stored in the automation module.
- **Max. stack height**
  Maximum stack height for big parts at the storage site.
- **Tolerance to gripper contour**
  Minimum clearance from contour of part required to activate a suction cup.
- **Min. part area**
  Minimum part area for detecting a part as a big part.

Sorting zone jobs



How do I open this tab?

Select: **Modules > Sort Job Editor > Data > Sort job > Sort job properties**

On this tab you can assign the parts in the cutting plans to sorting zone jobs. This allows you to stack parts with the same follow-up process together on the same stack carrier (e.g. bending, welding, painting, etc.).

You can also define a **target stacking area** for these parts, on which to stack the parts (e.g. pallet, cassette, cart, etc.).

The stack carriers are color-coded according to the color definition of the sorting zone job assigned to the respective parts. To do this, activate the special view and select the **Sorting zone jobs** option must be selected in the special view. The color definition can be edited in the **Color** column.

General

This section defines how to treat stacking areas for manual stacking of parts.

Mode

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- **Freely definable**

  Parts can be stacked on all available stack carriers. The target stacking area defined in the **Sorting zone jobs** section is not taken into account.

  ☞ **Note:** If stacks with different sorting zone jobs are sorted onto the same stack carrier, a warning message is shown.

- **Defined stacking areas**

  Parts can be stacked on all available stack carriers in the defined stacking area of the sorting zone job. The stack is displayed in the color of the associated sorting zone job

  ☞ **Note:** If stacks with different sorting zone jobs are sorted onto the same stack carrier, a warning message is shown. At the same time, an additional warning message is shown if stacks are sorted to the wrong stacking area.

Sorting zone jobs

Sorting zone jobs can be created in this section. The **Default** job shows all existing parts of the sorted cutting plans. Use the  icon to create custom sorting zone jobs.

☞ **Note:** Parts can be assigned to each sorting zone job in the **Parts** section.

Key to columns

- **Name**

  Defines the name of the sorting zone job. Use unique names for sorting zone jobs.

- **Sorting zone**

  Assigns the selected sorting zone to the sorting zone job. The existing sorting zones are defined in Administrator.

- **Color**

  Defines the color of the sorting zone job. The stacking areas with parts from different sorting zones are colored to match the appropriate target areas.

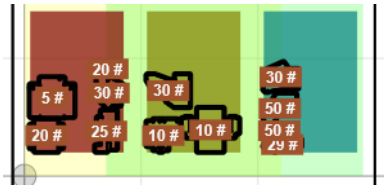


Fig. 9-7: Color coding of stacking areas with parts from different sorting zone jobs

Note: The color is displayed only if the special view is activated and the **Sorting zone jobs** option is selected.

Target stacking area

Assigns a stacking area to the sorting zone job for sorting the allocated parts.

Note: This default is only taken into account if **Defined stacking areas** mode is activated *(See: ›Defined stacking areas‹ Page 814)*.

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Parts

This section displays all parts of the cutting plans for the respective sorting zone job. By default, the parts are assigned to the **Default** sorting zone job.

To assign parts to the selected sorting zone job, use the  button and select the appropriate parts in the **Add parts** dialog.



Note: All parts assigned to sorting zone jobs will be removed from the **Default** sorting zone job.



Tip: Adding parts to sorting zone jobs affects the parts to be sorted and the sorting plan (stacking site definitions). Therefore, specify the sorting zone jobs directly after adding the cutting plans and only create the sorting plan afterwards.

9.2.2 Group: Stacking site definitions

Where do I find the group?

Select: **Modules > Sort Job Editor > Data**

This group contains all the functions for defining stacking sites.

9.2.2.1 Add



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > Stacking site definitions**

In this dialog you can add stacking site definitions to the job data.

If no name for the stacking site definition is input in the **Name** box, the system automatically enters the date and system time as the name.

Procedure…

1. Select the configuration for the stacking site definitions in the **Configuration** selection box.
2. In the **Name** box, enter the name of the stacking site definition and press **OK** to confirm.

9.2.2.2 Copy



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Stacking site definitions**

With this function you can add the stacking site definition selected in the **Job data** as a copy.

9.2.2.3 Remove



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Stacking site definitions**

This function removes the stacking site definition selected in the **Job data**.

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9.2.2.4 Reset



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Stacking site definitions**

This function removes all stacking site definitions created in the **Job data**.

Attention! It deletes all sorting plans and stacks created in the stacking site definitions.

9.2.2.5 Properties



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > Stacking site definitions**

You can edit settings for the selected stacking site definition in this dialog box. Press the **Apply** button to apply the edited setting for the selected stacking site definition.

Attention! If stacks have already been set on the stacking site definition, they are deleted when you edit the configuration.

9.2.2.6 Check stacking site definition



Where do I find the function?

Select: **Modules > Sort Job Editor > Export > Plans**

This function checks all sorting plans based on various criteria.

Links to verification criteria

If a criterion is associated with an error or warning, this is indicated by a red or yellow dot respectively. The **select** text appears on the right next to the check criterion.

- **select**
  Selects all elements or objects in the working area that are valid for the verification criterion.

 **Note:** Only those criteria are displayed that apply to the selected cutting plan.

Status of verification criteria

- **Green** = The stacking site definition is OK. The sorting plan can be exported.
- **Yellow** = The stacking site definition contains errors. The sorting plan can be exported with the errors. During the export, there will be a warning that will need to be acknowledged.
- **Red** = The stacking site definition contains errors. The sorting plan cannot be exported.

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Stacking site definition information

Information on the number of objects in the stacking site definition can be found in this area.

Status for laser cutting machines

This area lists all the criteria used to check the stacking site definition. The following example show the basic procedure used when a sorting plan does not meet criteria.

Note: The blue info bar will show the description or value for each selected criterion (figure).

Sorting time

The sorting time calculated in the background *(See: ›Icons in the workspace‹ Page 1092)*.

Number of sorted parts

Not all the parts in the cutting plan were sorted.

- Increase the number on the part stack.
- Increase the stack height on the part.
- Use the **Automatic** function to run automatic sorting.

Number of tool changes

Shows the number of tool changes for the current sorting plan.

- Use the number of tool changes to check the sorting plan's efficiency. The more tool changes, the longer the sorting process will take.

Number of gripper tools

The number of tools used exceeds the tool storage capacity.

- Use the automatic function to assign multiple parts to the parts in Part Editor.
- Only use tools that are present.

Removal sequence

More than two parts with the same removal sequence have been defined. Check the removal sequence in the **Cutting technology** menu.

- In the Cutting technology menu, run the **Automatic** function to reassign the removal sequence to the cutting plan.

Gripper tools on the parts

For one or more stacks, there are no gripper tools on the part.

1. Go to the **Parts** tab, select the part you want, and open it with the **Edit** context menu function in Part Editor.
2. Select the **Stack** menu.
3. Select the **Automatic** function.
4. Save and close the part.

Gripper tool check

Problems were detected with a gripper tool defined for a part.



Note: The error can refer to a tool that is not used in the sorting plan.

- The tool is not correctly positioned on the part.
- The tool's load capacity is in a critical range.
- The tool's maximum load capacity has been exceeded.
- The tool is not positioned in a stable manner on the part and may tilt over.

Microjoint test

Microjoints have been detected on an outer contour.

- Remove the microjoints from the part's outer contour.

Part sorting possibility

The sorting tool cannot reach the part.

- The configuration for the min./max. gripper tool positions in **Administrator** -> **Sorting unit** has been defined incorrectly *(See: ›Sorting unit (previous)‹ Page 131)*.
- The part stack on the stacking site cannot be reached by the tool at the set-down position.

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Set-down position

The ACTUAL tool cannot reach the set-down position.

- The configuration for the min./max. gripper tool positions in **Administrator** -> **Sorting unit** has been defined incorrectly *(See: ›Sorting unit (previous)‹ Page 131)*.
- Position the part stack within the stacking site.

Tandem gripper test

During tandem gripping, problems were detected with the distance between the gripper tools.

- The gripper distance is smaller than the minimum sorting head distance. Set the gripper tools on the part farther away from each other.
- The orientation of the nested part is not suitable for tandem gripping. Rotate the part on the sorting plan or on the stacking site to the same position.

Sorting head collision test

The sorting head will collide with one or more objects (e.g., crate edge).

- The sorting part is over an edge.
- The sorting part is too close to an edge. The tool may collide with the edge.
- Two stacks with different heights are too close to each other. The tool may collide with the taller stack during the depositing operation. This will be the case if the height difference between the stacks is equal to or greater than the height of the gripper tool.

Missing gripper tools

The gripper tools set on the part do not exist on the machine. Assign a gripper tool available on the machine to the part in Part Editor *(See: ›Add / Edit‹ Page 743)*. Also possible in Part Nester.

Disruptive contour collision test

A collision test was performed on the disruptive contour and detected a collision of the sorting head.

- For the **sorting unit (previous)**, the disruptive contours of the tool station are checked.
- For the **sorting unit**, the collision zone of the gripping zone and the stacking area is checked.

9.2.3 Group: Cutting jobs

Where do I find the group?

Select: **Modules > Sort Job Editor > Data**

This group contains all functions for cutting jobs in the Sort Job Editor.

9.2.3.1 Add



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Cutting jobs**

With this function you can add one or more jobs to the **Jobs** tab. The **Add Job** dialog is used to select the jobs and add them to the job list by pressing the **Open** button.

Selecting multiple jobs

- To select jobs in different folders, use the **Add** arrow buttons. The selected jobs will be added to the **Selected elements** section.
- Within a folder, you can select multiple jobs by pressing **Ctrl** + **left mouse button** and press the **Open** button to add them directly.
- All jobs found by the **Search** function are displayed on the **Search** tab and can be added from there.



Note: All selected jobs in the **PartJobs** folder and all jobs in the **Selected elements** section are added to the job list.

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Find

You can use the **Search** input box to find jobs with specific job names. Press **Enter** to display the search results on the **Find** tab.

You can use a list box to restrict the search according to certain criteria.

- **Entire text**
  The program searches for a text that matches the character string exactly.
- **Start of text**
  The program searches for a text that starts with this character string.
- **Part of text:**
  The program searches for a text that contains this specified character string.

Selected elements

This section shows all parts to be added to the **Jobs** tab.

- **Add search result**
  Option enabled: If the search result returns only one job, this job will be automatically added to the **Selected elements** section.

What each button does:

Button	Description
	Adds the selected jobs to the **Selected elements** section.
	Removes the jobs selected in the **Selected elements** section.

9.2.3.2 Remove



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Cutting jobs**

This function removes the selected job from the **Jobs** tab.



Tip: If you accidentally delete a job, use the **Undo** function to restore the deleted job.

9.2.3.3 Extract



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Cutting jobs**

This function saves a part job embedded in a sort job in the part job database. This is the case when a sort job was imported. The revision number stored on the part job is adopted.



Note:

- The function is only available for embedded part jobs.
- If there are jobs with the same part name in the selected folder, you can choose in a dialog whether to overwrite or skip these jobs.
- The parts contained in the part job remain embedded in the respective job. Extracting these parts must be done in Part Nester using the **Extract** part function *(See: ›Extract‹ Page 920)*.

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Extract an embedded job

1. Select the embedded job on the **Jobs** tab.
2. Click the **Extract** function in the context menu.
3. In the dialog, select the folder containing the job.
4. Press the **Open** button. The job will be saved in the specified folder.

9.2.3.4 Extract and replace



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Cutting jobs**

This function saves a part job embedded in a sort job in the part job database. The extracted part job is then used in the sort job.



Note:

- The function is only available for embedded part jobs.
- If there are jobs with the same part name in the selected folder, you can choose in a dialog whether to overwrite or skip these jobs.
- The parts contained in the part job remain embedded in the respective job. Extracting these parts must be done in Part Nester using the **Extract** part function *(See: ›Extract‹ Page 920)*.

Extract an embedded job

1. Select the embedded job on the **Jobs** tab.
2. Click the **Extract and replace** function in the context menu.
3. In the dialog, select the folder containing the job.
4. Press the **Open** button. The job will be saved in the specified folder.

9.2.3.5 Integrate



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Cutting jobs**

This function embeds all selected part jobs in the sort job. This deletes the link to the job and part database.

All changes to the cutting plan only affect the embedded cutting plans and parts.

 **Note:** Embedded jobs are indicated by the  icon.

 **Tip:** To save the part job back to the job database use the **Extract** function.

9.2.3.6 Edit



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Cutting jobs**

With this function you can edit the selected part job in Part Nester. As long as the job is open in Part Nester, the job is locked in Sort Job Editor.

 **Attention!** If any changes are made to the part job in Part Nester, this can affect the sorting plan, and it will therefore be checked again in Sort Job Editor after editing in Part Nester.

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9.2.4 Group: Sorting plan

Where do I find the group?

Select: **Modules > Sort Job Editor > Data**

This group contains all functions for creating stack-carrier-oriented sorting plans in Sort Job Editor.

9.2.4.1 Manage sorting process



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > Sorting plan**

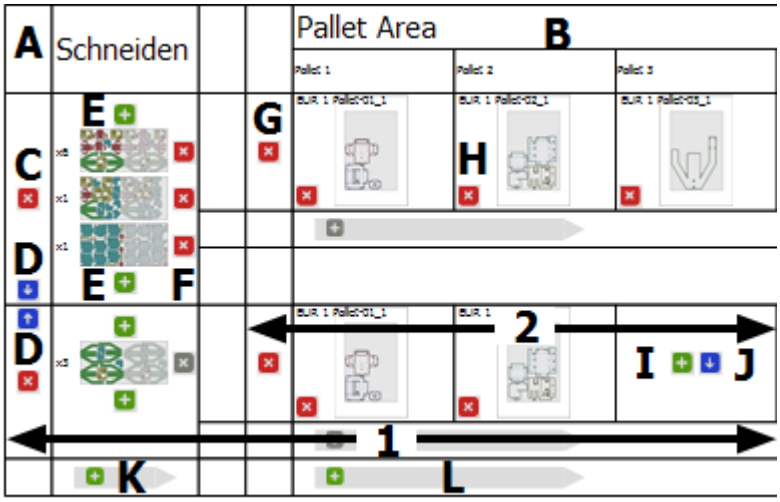
In this dialog you can create a sorting process manually or edit an existing sorting process. The dialog consists of the sections for cutting plans (**A**) and stack carriers (**B**). Each sorting cycle (**1**) is displayed on a separate line with the associated stacking areas (**2**).



Note: A cycle can use different stack carriers as long as they are defined in the sorting plans stacking area. A separate stacking area must be created for each stack carrier (**L**).

Adding cutting plans to a sorting cycle

1. In the **Cutting** section above or below the cutting plans, select the  icon.
2. In the **Add cutting plans** dialog, select the desired cutting plans.
3. Optionally, specify the passes of the cutting plans that you want to add.
4. Press **OK** to confirm the selection. The cutting plans are added to the sorting cycle.



Key:

A	Section for cutting plans	**G**	Delete stacking area
B	Section for stack carriers	**H**	Delete stack carrier
C	Delete cycle	**I**	Add stack carrier
D	Move cycle up or down	**J**	Copy stack carrier
E	Add cutting plan above or below	**K**	Create new cycle
F	Delete cutting plan	**L**	Add stacking area

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9.2.4.2 Add cutting plans



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Sorting plan > Sorting process -> Cutting ->  **

With this function cutting plans can be added to the sorting cycle.

- **Select all**
  Selects all cutting plans in the list.



Note: This function can only be selected if you have not yet added all cutting plans to the sorting process.



Tip: If a cutting plan has several passes, you can define the number of passes to add to the sorting cycle in the **Passes** column.

9.2.4.3 Sorting plan settings



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > Sorting plan**

This dialog is used to define the stacking areas for the sorting plan. The stacking areas are required for creating stacking areas for sorting plans.



Note: The available stacking areas are defined in Administrator on the **Sorting unit** form.

9.2.4.4 Automatic



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Sorting plan**

With this function automatically creates a sorting process based on the information in the **Define stack carrier sorting plans** dialog and displays it in the **Manage sorting process** dialog *(See: ›Manage sorting process‹ Page 828)*.



Note: Using the function is the easiest and best way to create a sorting process and should be used by default. Subsequent adjustment is usually not necessary.

9.2.4.5 Automatic sorting process settings



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > Sorting plan -> Automatic**

This dialog is used to define settings for automatic creation of sorting processes.

- **Max. allowed calculation time**

  Defines the time used for calculating the sorting process. By default, 10 seconds are used. Increasing the time usually does not produce better results.

- **Grouping**

  Takes the selected attribute into account for the order of sorting plans.

- **Automatic**

  If this option is enabled, the sorting plans are created automatically after the calculating the sorting processes.

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9.2.4.6 Create sorting plans



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Sorting plan**

This function automatically creates sorting plans based on the existing sorting process. If sorting plans already exist, the function cannot be selected.

9.2.4.7 Reset



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Sorting plan**

This function deletes the entire sorting process. Jobs and their cutting plans are retained.



Tip: An accidental reset can be undone by pressing the button or **Ctrl+Z**.

9.2.5 Group: Cutting plans

Where do I find the group?

Select: **Modules > Sort Job Editor > Data**

This group contains all the functions for editing cutting plans.

9.2.5.1 Add



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Cutting plans**

This function is used to add cutting plans to the job data and to add a stacking site definition.

Note: The function can only be selected if the **Cutting plans** tab does not yet contain all the cutting plans of the part jobs.

Select cutting plan

Select all

Selects all cutting plans in the list.

Tip: If a cutting plan has several passes, you can define the number of passes to add to the cutting plans in the **Passes** column.

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Stacking site definition

In this section you can assign a stacking site configuration to the selected cutting plans.

Edit stacking site definition

enabled: All existing stacking site definitions are displayed.

1. Click on the selection box for stacking site definitions and press **OK** to confirm your choice.

disabled: The program displays options for selecting a new stacking site definition.

1. Select the configuration for the stacking site definitions in the **Configuration** selection box.
2. In the **Name** box, enter the name of the stacking site definition and press **OK** to confirm.

Configuration

Selection box with all existing stacking site definitions.

Name

Input box for the name of the stacking site definition. If no name is entered, the program will automatically insert the current date and time when you press **OK** to confirm the selection.

Add cutting plans to sort data

1. Select the cutting plans in the top section.
2. Select the stacking site definitions for the cutting plans in the bottom section.
3. Press **OK** to confirm the selection.

9.2.5.2 Remove



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Cutting plans**

This function removes the selected cutting plans from the list.

9.2.5.3 Edit stacking site definition



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > Cutting plans**

This dialog box can be used to assign a new stacking site definition to the selected cutting plan. There are two ways to assign the stacking site definition to the cutting plan.

- **Edit stacking site definition**

  enabled: All existing stacking site definitions are displayed.

  1. Click on the selection box for stacking site definitions and press **OK** to confirm your choice.

  disabled: The program displays options for selecting a new stacking site definition.

  1. Select the configuration for the stacking site definitions in the **Configuration** selection box.
  2. In the **Name** box, enter the name of the stacking site definition and press **OK** to confirm.

  - **Configuration**

    Selection box with all existing stacking site definitions.

  - **Name**

    Input box for the name of the stacking site definition. If no name is entered, the program will automatically insert the current date and time when you press **OK** to confirm the selection.

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9.2.6 Group: Stack

Where do I find the group?

Select: **Modules > Sort Job Editor > Data**

This group contains all the functions for editing stacks.

9.2.6.1 Rotate stack



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > Stack**

You can use this dialog to rotate the selected stack through a specified angle. To rotate, enter an angle of rotation or use the mouse in the work area.

Procedure…

1. Select the stack to rotate in the stack area.
2. Select the rotation point on the stack.
3. Select the **Rotate stack** function.
4. Enter an angle of rotation in the **Angle of rotation** box.
5. Press **OK** to confirm the selection.

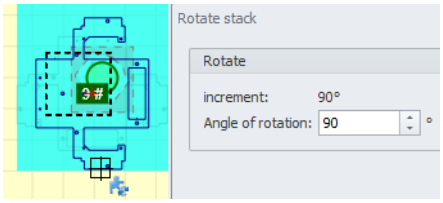


Fig. 9-8: Defining an angle of rotation using the input box



Tip: Select multiple stacks to rotate them together (Shift + right mouse button).

9.2.6.2 Remove stack



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Stack**

This function removes the selected stack from the stacking site definition.



Tip: Select multiple stacks to remove them together (Shift + right mouse button).

9.2.6.3 Stack insertion mode

Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Stack**

This function defines the mode for pasting copied stacks *(See: ›Sort job properties‹ Page 1098)*.

9.2.6.4 Copy stack



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Stack**

This function copies the highlighted stack onto the clipboard. The **Insert stack** function pastes the copied stack back onto a different stack carrier *(See: ›Insert stack‹ Page 837)*.

9.2.6.5 Insert stack



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Stack**

This function pastes the copied stack from the clipboard onto a different stack carrier. This stack is pasted onto the stack carrier under the mouse cursor.

Left-click or press the key combination **Ctrl+V** to paste. Each further mouse click adds another part to the stack. Press **ESC** to quit the **Insert stack** function.

The insertion point on the stack carrier corresponds to the position on the stack carrier from which the stack was copied. This allows parallel stacking of parts.

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9.2.6.6 Delete all stacks



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Stack**

This function removes all stacks from the stacking site definition.

9.2.6.7 Even out heights



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Stack**

This function automatically evens out the heights of stacks containing the same parts.

The function can only be used on stacks with a manually set number of parts. The number of parts is displayed blue for stacks with a manually set number of parts.

Procedure…

1. Choose a stack.
2. Select the **Even out heights** function.

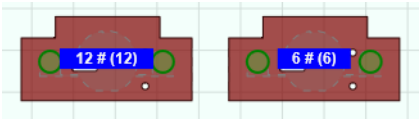


Fig. 9-9: Different number of parts

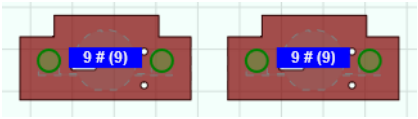


Fig. 9-10: Evened out number of parts

9.2.6.8 Arrange in Y direction



Where do I find the function?

Select: **Modules > Sort Job Editor > Data > Stack**

This function aligns the center of the current gripper on the part to be inserted in the Y direction to the center of the gripper of a part already set (**A**).

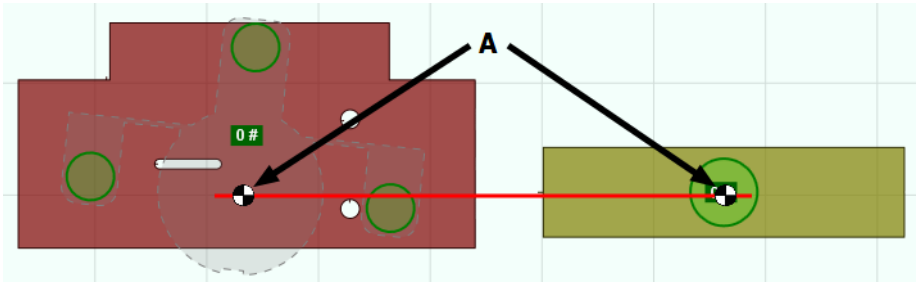


Fig. 9-11: Alignment of grippers in the X direction

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9.2.7 Group: Parts

Where do I find the group?

Select: **Modules > Part Nester > Sort Editor**

This group contains all the functions for automatic nesting of the parts.

9.2.7.1 Sort part properties



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > Parts**

In this dialog box you can define settings for the sort part.

General

- **Name**

  Displays the name of the sort part.

- **Revision**

  Displays the current revision number for the object. The revision number is increased by a count value every time the object is saved.

- **Color**

  Selection box for defining part color. All instances of the part are displayed this color in the sorting plan.

Stack settings

- **Rotation allowance**

  This field defines the increment of the angle of rotation in degrees. The part can be rotated by this angle during sorting.

- **Max. stack height**

  Defines the maximum stack height of the part on the stacking site.

- **Sort into box**

  This option stacks the part into a box on the stacking area when automatically creating a stacking site definition.

  Requirements:

  - The part fits in the box.
  - There is at least one box on the stacking area.

  If several parts have this option, each part is sorted into its own box.

To sort multiple parts into one box, enable the **Can be mixed with other parts** option.

> **Note:** If no box is available, the part is sorted onto another stack carrier, e.g. a pallet.

- **Can be mixed with other parts**
  This option can be used to stack multiple parts in one box.

> **Note:** This option can be used only if the **Sort into box** option is enabled.

Preferred gripper configuration

You can define the preferred gripper configuration for the selected part in the sorting part properties. This allows you to reduce the number of tool changes by using the same grippers in the sorting plan.

It can also be used to define the preferred tool for tandem gripping a large part.

> **Note:** A change of grippers is automatically updated on the part stacks in the sorting plan.

- **Configuration**
  Defines the preferred gripper tool for the part.
- **Tool 1**
  Shows the selected tool with the associated metadata.

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9.2.8 Group: View

Where do I find the group?

Select: **Modules > Sort Job Editor > Data**

This group contains all the view functions for the **Data** menu.

9.2.8.1 Special view



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Data > View**

This function shows/hides the options selected under the text button. The options can be used to display additional views on the object. Various options are available for selection depending on the module.



Tip: Use this function to get a quick overview of the sorting information in the current cutting plan.

The following information can be displayed:

- **Display parts group:** *Displays groups of overlapping parts with an additional border. The number of all parts in the group is displayed in the center of the parts group.*
- **Stack carrier height:** *Shows the height of the stack and the maximum possible stack height on the stack carrier. If there are multiple stacks on a stack carrier, the height of the largest stack is shown.*
- **Stack carrier weight:** *Shows the weight of all stacks on the stack carrier.*
- **Additional stack carrier information:** *Shows additional information about each part from the **Order information** column. The sorting rule is also displayed for each stack carrier.*
- **Stack quantities:** *Displays the number of parts per stack.*
- **Gripper:** *Displays the gripper set on the part.*
- **Show parts of all plans:** *Displays the direction of machining of all parts in a cutting plan*
- **Machine layout:** *Displays the machine layout as a gray area.*
- **Cutting plan:** *Displays the selected cutting plan at loading position on the shuttle table.*
- **Pick-place zones:** *Displays the whole BySort sorting area on the system.*
- **Sorting zone jobs:** *Displays the color-coded stacking sites where parts of sorting zone jobs are stacked.*

- **Suction-cup frame:** *Displays the suction cup layout of the suction cup frame on the automation system if it is assigned to the machine in Administrator. Use this display to nest parts manually, taking the suction-cup frame into account.*
- **Machining sequence:** *Displays the machining-sequence numbers of all the parts in the cutting plan.*
- **Removal sequence:** *Shows the removal sequences for the sorting process on the parts.*

9.2.8.2 Check sorting plan

How do I open this dialog box?

Select: **Modules > Sort Job Editor -> Data**

This dialog displays the status of all checked criteria in the sorting plan.

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9.3 Menu: Sorting technology

Where do I find the menu?

Select: **Modules > Sort Job Editor**

This menu is used to define the sorting technology for the current job.

The sorting technology consists of the nesting properties, removal sequence, and stack definitions.

Functional groups

The function bar contains the following functional groups:

- Selection
- Removal sequence
- Big part removal
- Gripper
- Snap
- View

9.3.1 Group: Removal sequence

Where do I find the group?

Select: **Modules > Sort Job Editor > Sorting technology**

The functions in this group can be used to assign a removal sequence for the sorting system manually or automatically to the parts in a cutting plan.

9.3.1.1 Set automatically



Where do I find the function?

Select: **Modules > Sort Job Editor > Sorting technology > Removal sequence**

This function can be used to automatically assign the removal sequence to the whole cutting plan. The sequence is only assigned to the current cutting plan.

Application:

This function should only be used to check the removal sequence when you want to define a removal sequence manually. For instance, when you want to remove certain parts last.



Note: An optimal removal sequence for the cutting plan is only possible if the sequence is only defined when the postprocessor exports the plan.



Tip: If the removal sequence is not displayed on the cutting plan, activate the **Removal sequence** entry in the special view (**CTRL+SHIFT+F10**).

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9.3.1.2 Set manually



Where do I find the function?

Select: **Modules > Sort Job Editor > Sorting technology > Removal sequence**

This function can be used to manually set the removal sequence for parts in a cutting plan.

The positioning path is indicated by a green line. The machining sequence is indicated by a number on each part that has been assigned a removal sequence.

Application:

Use this function to remove certain parts from the cutting plan first.

All parts that have not been assigned a removal sequence are assigned a removal sequence by the postprocessor when you expert the job. Warning **Some parts do not have a removal sequence number** is displayed before export.



Note: Before setting the removal sequence, first select the mode and then the part that you wish to assign the selected mode to.



Tip: If the removal sequence is not displayed on the cutting plan, activate the **Removal sequence** entry in the special view (**CTRL+SHIFT+F10**).

Mode

You can use the modes in this area to create the removal sequence manually or to modify an existing sequence.

Parallel removal

The selected parts are highlighted for parallel removal and are therefore given the same removal number.

Append

Adds the selected part to the end of the removal sequence.

Insert

The selected part is inserted in the removal sequence in front of the reference part.

> **Note:** The reference part is indicated by a red number.

Set reference part for insertion

The selected part is specified as the reference part for the **Insert** option.

> **Note:** The reference part is indicated by a red number.

Delete individually

The selected part is removed from the removal sequence.

9.3.1.3 Remove

Where do I find the function?

Select: **Modules > Sort Job Editor > Sorting technology > Removal sequence**

This function removes the removal sequence from the current cutting plan.

If no removal sequence is set but a sorting plan is defined, the removal sequence is defined by the postprocessor when you export.

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Tip: If you accidentally delete the removal sequence, use the **Undo** function to restore it.

9.3.2 Group: Big part removal

Where do I find the group?

Select: **Modules > Sort Job Editor > Sorting technology**

This group contains all functions for activating suction cups for removing big parts.

9.3.2.1 Set active grippers automatically



Where do I find the function?

Select: **Modules > Sort Job Editor > Sorting technology > Big part removal**

These functions automatically define big parts and activate the suction cups for gripping big parts.

- **Min. contour separation to suction cups**
  Minimum clearance from contour of part required to activate a suction cup.
- **Min. part area**
  Minimum part area for detecting a part as a big part.
- **Define big parts**
  Automatically defines the big parts in the active plan or in all plans of a job.
- **Determine active grippers**
  Automatically activates the suction cups on the big parts in the active plan or in all plans of a job.



Note: If a plan contains sort data, it is not possible to perform big part removal. The **Big part removal disabled** section is displayed. Press the **Remove sort data** button to remove sort data from the job. You can then apply big part removal to the job.

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9.3.2.2 Set active grippers manually



Where do I find the function?

Select: **Modules > Sort Job Editor > Sorting technology > Big part removal**

These functions can be used to manually define big parts and activate the suction cups for gripping big parts.

- **Min. contour separation to suction cups**
  Minimum clearance from contour of part required to activate a suction cup.
- **Min. part area**
  Minimum part area for detecting a part as a big part.

Mode

- 
  Enables or disables the selected part for removal of big parts.
- 
  Enables or disables the selected suction cup for gripping big parts.

9.3.2.3 Remove



Where do I find the function?

Select: **Modules > Sort Job Editor > Sorting technology > Big part removal**

This function removes the active suction cups for the big parts.

9.3.3 Group: Gripper

Where do I find the group?

Select: **Modules > Sort Job Editor > Sorting technology**

> **Note:** This group contains all the functions for assigning one or more gripper tools to the part for sorting.

9.3.3.1 Automatic



Where do I find the function?

Select: **Modules > Sort Job Editor > Sorting technology > Gripper**

This function automatically sets single-part or multi-part grippers on the plan or the entire job. The gripper tools are set on the parts based on the selected options.

Single-part gripper

If this function is active, the grippers selected under the **Gripper tools** option are set on the parts. If more than one gripper tool is selected, the

- **Overwrite existing grippers**

  All existing gripper tools on the parts are deleted before new gripper tools are set.

- **Gripper tools**

  Select the gripper tools to set on the parts in the selection box.

> **Note:** For non-magnetic material, all magnet grippers are disabled and cannot be selected.

Multi-part gripper

If this function is active, the multi-part gripper defined under the **Tool** option is set on the parts. If the **Single-part gripper** section is also activated, the multi-part gripper is set on the parts first. If it is not possible to grip parts with the multi-part gripper, a single-part gripper is used for these parts.

> **Tip:** If it is not possible to grip parts with the multi-part gripper, it may be possible to achieve a better result by changing the axis values.

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> **Note:** It is not possible to use multi-part grippers on a machine with a first generation sorting unit.

- **Overwrite single-part grippers**

  Deletes all single-part gripper tools set on parts and replaces them with multi-part grippers.
- **Overwrite multi-part grippers**

  Deletes all multi-part gripper tools set on parts and replaces them with a new multi-part gripper.
- **Tool**

  Selection box for gripper tools. The tools are created in Administrator *(See: ›Gripper tool‹ Page 178)*.
- **Axis value**

  This value defines the axis value of the selected axis for multi-part grippers. It can be used to customize the gripper for a particular part.

  - **Axis**

    Determines the axis for which the axis value can be set.
- **Automatic Selection**

  Gripper tools are set on the selected parts of the current plan according to the settings.
- **Automatic Plan**

  Gripper tools are set on the parts of the current plan according to the settings.
- **Automatic Job**

  Gripper tools are set on the parts of all plans of the job according to the settings.

9.3.3.2 Manual



Where do I find the function?

Select: **Modules > Sort Job Editor > Sorting technology > Gripper**

In this dialog you can add grippers to cutting plans or edit existing grippers.

- **Add**

  This option allows you to add and position gripper tools on the part and define certain properties.
- **Edit**

  This option allows you to edit the gripper tool set on the part and edit certain properties.
- **Apply for all parts**

  This option applies gripper settings to all instances of the selected part.

When positioning a tool, the center of gravity of the tool is displayed as a red dot. Click on the center of gravity or drag the gripper tool out of the geometry display to position at the center of gravity.

Depending on the situation, grippers are displayed in the following colors:

- **Green:** Part can be gripped without any problems.
- **Orange:** Part can be gripped but is unstable and may tip over.
  - Position gripper at center of part.
  - Use a different gripper tool.
- **Red:** The part cannot be gripped.
  - Part is too heavy for the gripper.
  - Gripper is positioned on a contour.
  - Gripper is too far from the center of gravity.

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Gripper tool 1

This box displays the currently selected gripper tool 1 and its properties.

Tool

Selection box for gripper tools. The tools are created in Administrator *(See: ›Gripper tool‹ Page 178)*.

Note: For non-magnetic material, only vacuum grippers are displayed.

X/Y

Position of gripper tool on part.

If the gripper tool has multiple grippers, the center of the grippers should be at the center of gravity of the part.

Angle of rotation

You can use this function to rotate gripper tool 1 through a specified angle.

Optimum part stability is achieved when the longest axis of the gripper tool coincides with the longest axis of the part.

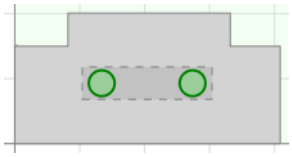


Fig. 9-12: Stably positioned gripper tool

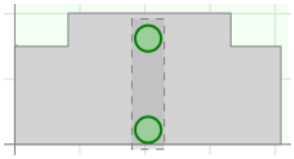


Fig. 9-13: Unstably positioned gripper tool

Tip: The angle of rotation for the gripper tool can also be set with **Ctrl** + **mouse wheel** before positioning.

Axis value

This value defines the axis value of the selected axis for multi-part grippers. It can be used to customize grippers for multiple parts.



Note: This option is only active for multi-part grippers.

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Gripper tool 2

Gripper tool 2 is used if the part needs to be gripped with two gripper tools (tandem pick). This may be the case with very long or very heavy parts.

When nesting, the tandem part can be rotated by **0°** or **180°** in Part Nester.

Active

Activates the 2nd gripper tool for positioning on the part.

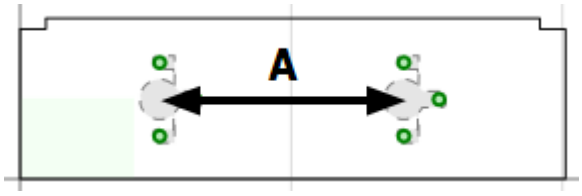
Tool

Selection box for gripper tools. The tools are created in Administrator *(See: ›Gripper tool‹ Page 178)*.

Note: For non-magnetic material, only vacuum grippers are displayed.

ΔX

Positions gripper tool 2 on the part at the specified distance (**A**) to gripper tool 1 in the X direction.



Tip: The distance between gripper tools can also be set using **Shift** + **mouse-wheel** before positioning.

Angle of rotation

Use this option to rotate gripper tool 2 through a specified angle.

Optimum part stability is achieved when the longest axis of the gripper tool coincides with the longest axis of the part.

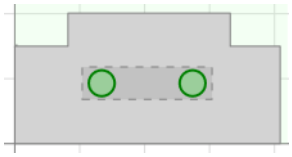


Fig. 9-14: Stably positioned gripper tool

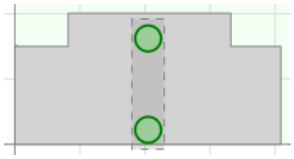


Fig. 9-15: Unstably positioned gripper tool

Tip: The angle of rotation for the gripper tool can also be set with **Ctrl** + **mouse wheel** before positioning.

Part rotated

This rotates the tandem part including the gripper tools by 90°. This allows the part to be rotated by a discrete angle of rotation of **90°** or **270°** when nesting in Part Nester *(See: ›Discrete angles of rotation‹ Page 1210)*.

Tip: If a non-rotated gripper tool is also created on the tandem part, the part can also be nested in Part Nester using a discrete angle of rotation of **0°** or **180°**.

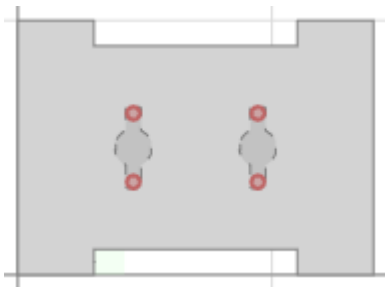


Fig. 9-16: Tandem part not rotated

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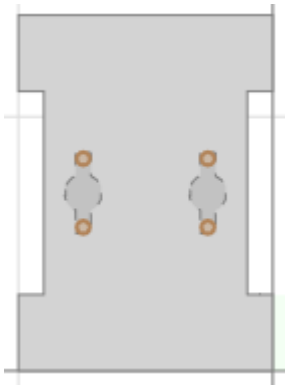


Fig. 9-17: Tandem part rotated

Axis value

This value defines the axis value of the selected axis for multi-part grippers. It can be used to customize grippers for multiple parts.

 **Note:** This option is only active for multi-part grippers.

Apply

This button applies the gripper settings to one or all instances of a part in the cutting plan

9.3.3.3 Remove



Where do I find the function?

Select: **Modules > Sort Job Editor > Sorting technology > Gripper**

This function is used to remove the gripper tools set by the **Automatic** or **Manual** function from the parts in the current plan.

9.4 Job data

How do I open this dialog box?

Select: **Modules > Sort Job Editor**

This dialog contains all objects used in the current part job.

At least one element of the following objects must be present in each part job in order to create and export a sorting plan:

- A cutting plan containing at least one nested part
- A stacking site configuration

The objects are displayed in a table in the corresponding tabs. The number of objects in the tab is displayed in brackets.



Note: You can customize how the columns are displayed in each of the tabs *(See: ›Customizing table columns‹ Page 1334)*.

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9.4.1 Jobs



How do I open this tab?

Select: **Modules > Sort Job Editor > Job data**

This tab displays part jobs as a table. All existing part jobs can be used for the cutting and sorting process.

What each column means:

No.

Shows the number of the object.

Status

Displays the status of the part job.

Picture

Previews the object in a diagram.

Name of part job

Displays the name of the part job.

Info 1-8

Extra information about the current object can be entered in these fields.

9.4.2 Stacking site definition

How do I open this tab?



Select: **Modules > Sort Job Editor > Job data**

This tab contains a table showing all the stacking site definitions in a job. All the functions in the **Stacking site definition** group affect this tab and can be run directly from within the group or via the context menu. Clicking on a stacking site definition will open it in cutting plan view.

The entries in the following columns can be modified by hand:

- **Stacking site definition name**
- **Info 1**

Add cutting plans to sort data

1. Select the **Add** function in the **Stacking site definition** group.
2. Select the stacking site configuration in the **Configuration** box.
3. In the **Name** box, enter the name of the stacking site definition.
4. Confirm your entries with **OK**.

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What each column means:

No.

Shows the number of the object.

Picture

Previews the object in a diagram.

Stacking site definition name

Displays the name of the stacking site definition defined for the cutting plan. You can define the name in the properties of the stacking site definition.

Stacking site configuration

Displays the stacking site configuration assigned to the stacking site definition.

Cycles

Displays the number of repetitions with which the cutting plan is cut.

Info 1-8

Extra information about the current object can be entered in these fields.

Number of plans

Shows the number of cutting plans stacked on the stacking site definition.

Stack carrier sorting strategy

Defines the sequence in which parts are stacked on different stack carriers.

- **Regular**

  Parts are stacked alternately on the different stack carriers. As a result, the stacks are evenly stacked to their final height.

  This strategy has the highest process reliability, as it prevents the gripper tool from colliding with other stacks and is therefore used by default.

- **Stack carrier-wise**

  Stacks are stacked completely one after another per stack carrier.

  This strategy allows you to remove stack carriers from the sorting zone for clearing before all parts have been stacked.

9.4.3 Cutting plans

How do I open this tab?



Select: **Modules > Part Nester > Job data**

This tab contains a table showing all the cutting plans for sorting. All the functions in the **Cutting plans** group affect this tab and can be run directly from within the group or via the context menu. Clicking on a cutting plan will open the cutting plan in the cutting plan view.

The entries in the following columns can be modified by hand:

- **Name of cutting plan**
- **Info 1-8**

Add cutting plans to sort data

1. Select the **Add** function in the **Cutting plans** group.
2. Select the cutting plans in the top section.
3. Select the stacking site definitions for the cutting plans in the bottom section.
4. Press **OK** to confirm the selection.

What each column means:

No.

Shows the number of the object.

Picture

Previews the object in a diagram.

Name

Shows the name of the cutting plan.

Stacking site definition name

Displays the name of the stacking site definition defined for the cutting plan. You can define the name in the properties of the stacking site definition.

Stacking site configuration

Displays the stacking site configuration assigned to the stacking site definition.

Cycles

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Displays the number of repetitions with which the cutting plan is cut.

Info 1-8

Extra information about the current object can be entered in these fields.

9.4.3.1 Split up cycles



Where do I find the function?

Select: **Modules > Part Nester > Sort Editor > Cutting plans**

With this function you can split plans with multiple passes into separate plans.

- **Original plan pass**

  Displays the number of passes of the selected plan.
- **Create plans with pass 1**

  Creates a separate cutting plan for each pass.
- **Plan passes, copies**

  In this field you can define a custom number of plans and passes. By default, one plan is defined for each pass.

> **Note:** The details for each plan must be separated by a semicolon and the total must correspond to the pass of the original plan.

Example:

- Original plan pass = **4**
- Desired copies of plan = **2**
- Entries in the field **Plan passes, copies** = **2;2**
- Result = **The plan is split into two plans with pass 2**.

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9.4.4 Parts



How do I open this tab?

Select: **Modules > Part Nester > Job data**

This tab displays all parts of the currently selected cutting plan as a table. All the functions in the **Part** group of the **Data** menu relate to this tab and can be run directly from within the group or via the shortcut menu.

The colors of the parts are assigned automatically by the software. Use the **Color** option in the **Properties** to change the color of a part manually to improve the view in the cutting plan.

Set parts on stacking site

You can drag and drop parts as stacks onto the stacking site. Clicking the left mouse button again adds another stack of the part. The number of parts per stack is reduced according to the stack settings.

If part stacks of sorting zone jobs are defined incorrectly, a warning is displayed.

- Part stacks from different sorting zone jobs are stacked on the same stack carrier.
- Part stacks are stacked in an incorrect target stacking area.



Attention! The part contour must be inside the stacking site definition, otherwise it may tip over after stacking.

What each column means:

No.

Shows the number of the object.

Status

Displays the status of the parts in the nesting process.

Picture

Previews the object in a diagram.

Info 1-8

Extra information about the current object can be entered in these fields.

If the part is a cluster containing different parts, the information texts of the respective parts are shown in the field separated by semicolons.

Order information

This field is not intended for any particular purpose. You can use this field to enter any information you wish.

Name

Shows the name of the part. The name is used for unique identification of a part.

Size

Shows the size of the bounding rectangle of a part.

Parts

Shows the available number of parts for sorting in the selected cutting plan.

Sorted parts

Displays the sorted number of parts in the sorting plan.

Rotation allowance

Indicates the increment of the angle of rotation by which the part can be rotated.

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9.4.5 Waste parts



How do I open this tab?

Select: **Modules > Sort Job Editor > Job data**

This tab displays all waste parts that can be sorted. Waste parts can be inside contours of parts, skeleton sheets or residual sheets.

To sort manually, select the waste part and drag it to a sorting position on the stack carrier.

 **Note:** In order for these parts to be sorted, the respective objects must be defined as sortable waste parts.

What each column means:

No.

Shows the number of the object.

Status

Displays the status of the parts in the nesting process.

Picture

Previews the object in a diagram.

Name

Displays the name of the waste part. The name is used for unique identification of a waste part.

Size

Shows the size of the bounding rectangle of a part.

Sorted parts

Displays the sorted number of parts in the sorting plan.

Parts

Shows the available number of parts for sorting in the selected cutting plan.

9.4.6 Stack carrier sorting plans



How do I open this tab?

Select: **Modules > Sort Job Editor > Job data**

This tab displays all stack carriers created for sorting plans.



Tip: Select the **Edit** function from the context menu to edit the stacks on the stack carrier.

9.4.6.1 Edit stack carrier sorting plans



How do I open this dialog box?

Select: **Modules > Sort Job Editor > Job data -> Stack carrier sorting plans**

This dialog is used to create or edit stack carriers for sorting plans. The stack carriers are required for creating the sorting process.

The dialog consists of the following sections:

Sorting zone jobs

This section displays all sorting zones defined under the **Sort job properties** function. By default, the parts of the jobs are assigned to the **Default** zone.



Note: Select a sorting zone to display all parts assigned to the zone in the **Parts** section.

Select the icon in the **Add** column to add the part to the selected stack carrier in the **Stack carrier** section and automatically nest on it. If the part cannot be nested, the part is placed in the center of the stack carrier.

Stack carrier sorting plans

This section contains all the functions for creating and editing stack carriers. The top section displays the stack carriers created with the added parts. Select the icon in the **Remove** column to remove the part on the stack carrier. The two buttons at the top can be used to open or close all stack carriers at the same time.

The lower section contains the function for creating stack carriers as well as a preview of the selected stack carrier.

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- **Delete**

  Deletes the selected stack carrier
- **Copy**

  Creates a copy of the selected stack carrier
- **Stack carrier area type**

  Selection box for the stack carrier area
- **Stack carrier**

  Selection box for the stack carrier type
- **Add stack carrier**

  Creates a new stack carrier based on the settings for the stacking area and the stack carrier type.
- **Locked**

  Prevents automatic nesting of part stacks on the stack carrier. This option can be selected to perform subsequent manual nesting of part stacks.

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9.4.7 Assigned parts



How do I open this tab?

Select: **Modules > Sort Job Editor > Job data**

This tab displays all parts present on the selected stack carrier. In the dialog, you can change the number of parts and also specify additional stacks required. The additional stacks are displayed on the **Stack carrier** tab.



Note: Additional stacks can only be created if the maximum instances of the part on the sorting carrier are reduced so that they are available for additional stacks.

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9.4.8 Stack carrier



How do I open this tab?

Select: **Modules > Sort Job Editor > Job data**

This tab displays all stack carriers created for the sorting process.

9.4.8.1 Edit stack carrier



Where do I find the function?

Select: **Modules > Sort Job Editor > Job data -> Stack carrier**

This function can be used to edit the stacks on the stack carriers. The stacks can be deleted or moved within the gray area.

-  Ends editing of stack carriers
-  Prevents nesting of additional part stacks on the current stack carrier

9.4.9 Job



How do I open this tab?

Select: **Modules > Sort Job Editor > Job data**

This tab displays summary information about the part job.

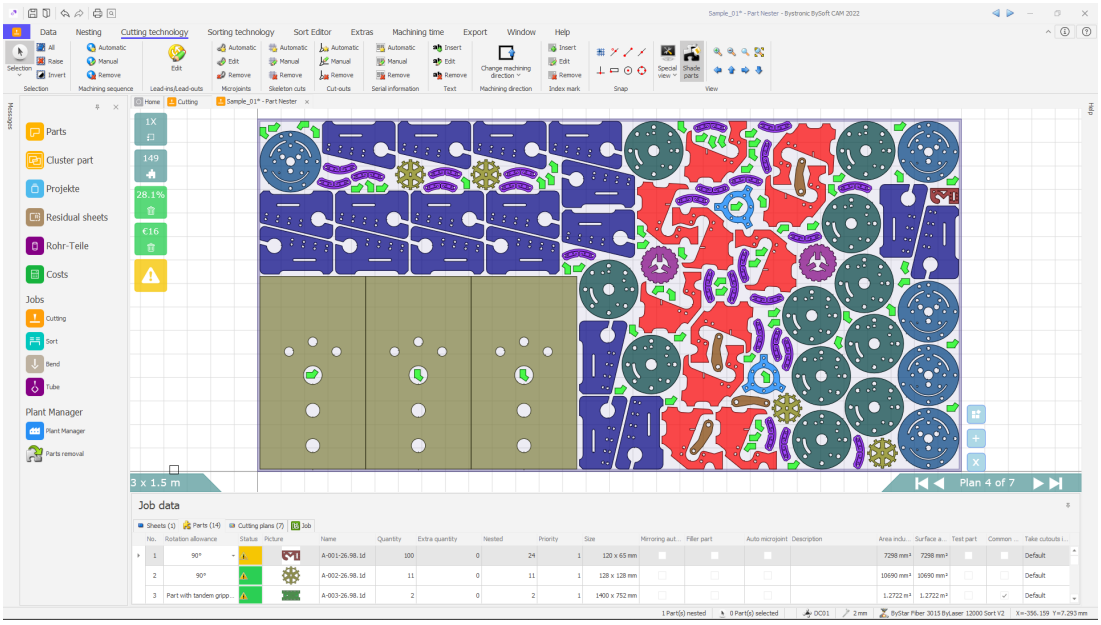
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10 Module: Part Nester

Where can I find the module?

Select: **Modules >**



The Part Nester is a module in which jobs can be produced from parts and can be exported as a cutting plan. The generation process for jobs basically follows the sequence of menus from left to right.

Note: If the job contains parts with the **Warning** or **Error** category, this is indicated by a suitable message which is displayed for a certain period of time on opening the job.

Tip: If a part is selected in the cutting plan, the appropriate part is highlighted on the **Parts** tab of the job data.

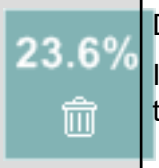
Tip: Click on an area of the header to call up the **Shortcuts** function in the context menu. The **Shortcuts** dialog can be used to define keyboard shortcuts for all functions of the current module.

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Cutting plan information

Each cutting plan displays graphically on the left-hand side the number of machining cycles and the cutting-plan waste. The display is refreshed automatically as soon as any change is made to the nested cutting plan. For multihead cutting plans, the cutting plan also displays the number of tools used and the maximum number of tools.

Icons in the workspace

Symbol	Description
	Displays the number of machining cycles of the cutting plan.
	Displays the number of parts in the cutting plan. If the cutting plan has several cycles, the number of parts is displayed for all cycles.
	Displays the maximum number of tools available on the machine as squares, and the tools used in the cutting plan as dots. ☞ **Note:** Only displayed on water jet cutting machines.
	Shows the number of cutting areas for repositioning machines.
	Displays the percentage waste for the cutting plan. If a cost table has been saved with the cutting plan, the icon is colored in the cost table based on the waste target. -  Equal or less than the waste target. -  Greater than the waste target.

Symbol	Description
	Displays the waste costs of the cutting plan in the currency of the selected cost table. **Note:** The icon is only displayed if the cutting plan has a cost table with the appropriate values for the material *(See: ›Cost table‹ Page 912)*. ■  Equal or less than waste costs. ■  Waste costs were exceeded.
	Displays the number of gripper tools used in the sorting plan. **Tip:** Click on the icon to display the **Check gripper tools** dialog with detailed information about the gripper tools.
  	Shows the status of the cutting plan. The status is updated in real time. ■ **Symbol 1:** The cutting plan is OK. The cutting plan can be exported. ■ **Symbol 2:** The cutting plan contains errors. The cutting plan can be exported. ■ **Symbol 3:** The cutting plan contains errors. The cutting plan cannot be exported. **Tip:** Click on this symbol to call up the **Verify plan** dialog with detailed information about the status of the cutting plan.
	Displays the sheet size for the cutting plan.
	Navigation bar for switching between multiple cutting plans.

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Information about the part

Moving the mouse over a part opens a pop-up window displaying various information about the part.

> **Note:** A part must not be selected if you wish to display part information.



Fig. 10-1: Displaying information about parts in the cutting plan

Interactive buttons

Pressing the space bar will display interactive buttons in the Part Nester workspace. The buttons remain visible so long as the space bar is pressed.

Buttons

Button	Description
	This button reduces a sheet to the dimensions of the nested parts.
	This button adds a new cutting plan to the **Cutting plans** tab.
	This button deletes the entire cutting plan.
	Indicates that parts are nested on the cutting plan in the grain direction. The parts are defined on the **Parts** tab with the **Grain direction** rotation allowance.

Button	Description
	This button nests the current cutting plan. If a part is selected in the cutting plan, this button is also displayed to the lower left of the move cross (**A**). This button is used in order to nest only the selected part in the cutting plan. If multiple parts are selected, they are nested as a module *(See: ›Nesting parts as a module‹ Page 1366)*.
	Use this button to set or remove a programmed stop on a part. ☞ **Note:** This button is only available after you set the machining sequence.
	The part can be nested with the **Row** function.
	All changes to the selected parts are reset to the original state. This function is displayed if at least one part in the part selection has subsequent changes. The following changes can be applied to a single part in the cutting plan: ■ Set microjoints ■ Edit lead-ins/lead-outs ■ Change machining direction ■ Delete text on parts and their instances in the current cutting plan.
	Press this button to edit the selected cluster.
	Manual common cut nesting has been disabled for the selected part. To enable it, click on the button.
	Manual common cut nesting has been enabled for the selected part. To disable it, click on the button.
	**Sort Editor:** The number of parts in the stack is defined manually. Click the button to define the number of parts dynamically.
	**Sort Editor:** The number of parts in the stack is defined dynamically. Click the button to define the number of parts manually.
	**Sort Editor:** The parts are stacked with a small offset inside the broken line. To disable it, click on the button.
	**Sort Editor:** The parts are stacked without an offset. Click the button to stack the parts with an offset.

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☞ **Note:** If the **Always display interactive buttons** option is enabled in the Part Nester settings, the buttons will be displayed permanently *(See: ›Always display interactive buttons‹ Page 906)*.

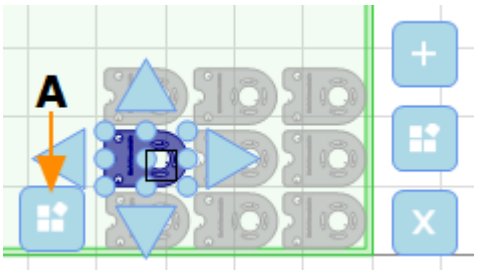


Fig. 10-2: Buttons displayed when space bar is held down and a part is selected

Creating a job automatically

Follow the steps below to create a job from various parts and to export it for a cutting machine:

1. In the Part Nester module, select the **New job** function.
2. In the **Job properties** dialog box, specify the job name, the material, the thickness and the machine that you want to use to cut the job.
3. Confirm your entries with **OK**.
4. In the **Data** menu, select the **Add sheet** function to add at least one sheet to the job.
5. Select the **Add part** function.
6. In the **Add part** dialog box, specify the parts that you wish to create the job from.
7. Press **Open** to confirm the selection. The parts are added to the job.
8. For each part, specify the quantity of parts to be produced in the **Parts** tab of the **Job data** area.
9. Select the **Automatic cutting technology** function to create the job automatically. If the nesting result is not optimum, check the settings using the **Settings** function and re-nest the job.
10. Run the **Verify plan** function from the **Export** menu to check the job. If the job has been created correctly, run the **Export plans** function to export the cutting plans to the Export folder.

Menus in Part Nester

- File functions
- Data
- Nesting
- Cutting technology
- Sort Editor
- Extras
- Machining time
- Export
- Window

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10.1 File functions

The functions in this menu relate to the files and user interface of the module.

Functions for the current job

These functions relate solely to the active job.

- Save
- Save as...
- Print working documents
  - Quick print
  - Preview
- Print labels
  - Preview
  - Quick print
- Export
- Undo
- Restore
- Close

General functions

These functions are available to run irrespective of the active job.

- Open
- Save all
- Import
- Settings
- System settings

10.1.1 New job



How do I open this dialog box?

Select: **Part Nester >** 

This function creates an empty job. Certain details must be specified in the **Job Properties** dialog box before the job is generated.

To create a new job, it is essential to enter the name. All the other details can be entered or modified later in the **Data** menu using the **Properties** function in the **Job** group, and when saving the job.



Note: This dialog is also displayed when using the **Import job** function after you have selected the import file *(See: ›Import job‹ Page 894)*.

Basic steps for creating a new jobs:

1. Create a new job file
2. Add sheets
3. Insert Parts
4. Define nesting settings
5. Nest job
6. Define export settings
7. Export cutting plans
8. Save the job in the database

Settings

This section is only displayed when you create a new job.

from template

You can select a job template from the job database in this field.

The job settings are used from the selected template.



Tip: Use job templates for recurring jobs with the same settings *(See: ›Creating job templates‹ Page 1359)*.

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from Settings Manager

The job settings are used from Settings Manager in Administrator.

General

Name

The name of the job must be entered in this field. If the **Job name generator** is enabled, the name is entered automatically *(See: ›Job name generator‹ Page 904)*.

☞ **Note:** The name is essential for the program to be able to save the record.

Description

You can enter a more detailed description of the current object in this field.

Info 1–8

Extra information about the current object can be entered in these fields.

System of units

The system of units for the current object is defined in this field (mm/inches)

☞ **Note:** The job can only be nested with parts that use the same system of units.

Material

Name

This field displays the name of the material. Clicking on the arrow symbol on the right lets you select the material from a list.

Thickness

This field displays the thickness of the material. Clicking on the arrow symbol on the right lets you select a thickness from a list. All the thicknesses available for the material are displayed. If you require a different thickness, enter it in the field from the keyboard.

Cutting process

Machine

This field displays the machine that shall be used to machine the object. Clicking on the arrow symbol on the right lets you select a different machine from a list. This list only contains those machines that are configured in the system. If you require a new machine, you must create it in the **Administrator** module in the **Cutting machines** form *(See: ›Cutting machine‹ Page 102)*.

Cutting gas

Select the cutting gas used for the job in this box. This information is used for the Cutting Wizard filter definitions in the export settings in the **Export** menu *(See: ›Filter definitions‹ Page 1129)*.

Abrasive (ByJet)

Select the abrasive used for the job in this box. This information is used for the Cutting Wizard filter definitions in the export settings in the **Export** menu *(See: ›Filter definitions‹ Page 1129)*.

Profile (Tube Nester)

In this area you specify the profile type and its dimensions for the tubular-part job. Only those parts that have the same profile type and dimensions can be added to the job.



Note: As soon as there is a part in the tube job, the profile type can no longer be changed.

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10.1.2 New job from template



How do I open this dialog box?

Select: **Modules > Part Nester**

This function creates a job from a job template. This function can be used to create cutting plans immediately from the job template or for nesting extra parts in the template.

You can use a list box to restrict the search according to certain criteria.

- **Entire text**

  The program searches for a text that matches the character string exactly.
- **Start of text**

  The program searches for a text that starts with this character string.
- **Part of text:**

  The program searches for a text that contains this specified character string.

Creating a job from a template

1. Select the **New job from template** function.
2. In the **Open job** dialog box, select the template and press the **Open** button.
3. Enter a name for the job in the **Job properties** dialog box. Entering a name is not mandatory.
4. Close the dialog box with **OK**. A copy of the job template is opened as a new job, which can be edited to suit your requirements.

10.1.3 Save as template



Where do I find the function?

Select: **Part Nester >** 

This function can be used to save the current job as a template in the **Templates** folder. You can create additional child folders below this folder for saving the job templates. Use the **Save as** dialog box to save the template. A default job name is suggested in the **Name** field which is composed of the job settings of machine, material, thickness and system of units.



Note: Job templates for the automatic cutting-time calculation in the Plant Manager module must be saved in the **Plant Manager** template folder for them to be available for use by the Plant Manager application. Job templates are managed in the same way as standard jobs in **Object Manager**.



Tip: Creating job templates can cut the time needed for new jobs, because recurring settings in new jobs do not need to be customized each time.

Examples of uses for templates:

- Manually created nesting plans that cannot be reproduced with the automatic nesting system.
- Recurring cutting jobs as a package.
- For automatic calculation of the machining time for cutting plans in the Plant Manager .

Saving a job template for manually created nesting plans

1. Select the **New job** function.
2. Define the system of units, material, thickness and machine for the job template and confirm your settings with **OK**.
3. Create the manual nesting plan with the relevant export settings.
4. From the File functions, select the **Save as template** function.
5. Press **Save** to save the job as a template.

Saving a job template for recurring cutting jobs

1. Use the **Open job** function to open the job that you wish to save as a job template.

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2. From the File functions, select the **Save as template** function.
3. Press **Save** to save the job as a template.

10.1.4 Save as default template



Where do I find the function?

Select: **Part Nester >** 

This function is used to save the current job as a **Default** template in the **PartJobs > Templates** folder. This job template is used every time new jobs are created using the **New job** function.



Note: Job templates are managed in the same way as standard jobs in **Object Manager**.



Tip: Use the default template to specify those job settings that occur most often.

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10.1.5 Print working documents



Where do I find the function?

Select: **Part Nester >** 

This function can be used to print out the working documents for the current job.

Printer

In this area you can select the printer and set your printer preferences.

Print

In this area you can specify preferences for the print layout.

- **Print template**
  Specifies what layout is used for the printed document.
- **Number of copies**
  Specifies how many copies of the document are printed.

Preview

This button previews the print document. The document can also be printed or emailed from the Preview dialog box.

10.1.6 Print labels



Where do I find the function?

Select: **Part Nester >** 

Printer

In this area you can select the printer and set your printer preferences.

Print

In this area you can specify preferences for the print layout.

- **Print template**
  Specifies what layout is used for the printed document.
- **Number of copies**
  Specifies how many copies of the document are printed.

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Labels

In this area you can set your preferences for printing labels. If the option **By cutting sequence** is not enabled, the labels for the parts are printed in the order of the parts list.

- **Mode**
  - **One label per part**: One label is printed for each part in the job
  - **One label per stack**: One label is printed for each stack. The number of stacks is calculated from the total quantity of a part divided by the number of parts for a stack, which is specified under the **Number of parts per stack** option.
- **Number of parts per stack**

  Specifies the number of parts per stack, with one label printed per stack. This option can only be used when the **One label per stack** attribute has been selected for the **Mode** option.
- **Number in reserve**

  Specifies how many extra labels of each label are printed as reserve stock. When a quantity of 1 is specified, twice as many labels are printed as are required.
- **Print filler parts**

  If this option is enabled, the filler parts are included in label printing.
- **By cutting sequence**

  Prints the labels for the parts according to the sequence of machining in the cutting plan.

Preview

This button previews the print document. The document can also be printed or emailed from the Preview dialog box.

10.1.7 Both



Where do I find the function?

Select: **Part Nester >** 

This function automatically prints working documents and labels for the current job based on the respective print settings.

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10.1.8 Import job



Where do I find the function?

Select: **Part Nester >** 

This function can be used to import a job in another format into BySoft CAM. All parts in an imported job are embedded in the job and are not saved in the parts database.

The following job formats are supported:

- **Bysoft NCP/LCC (*.lcc/ncp):** *This is the format for importing a cutting plan from an NCP or LCC file (See: ›Importing parts from LCC/NCP files‹ Page 1391).*
- **Bysoft BVC/CCP (*.bvc;*.ccP;):** *This is the format for importing a cutting plan from an BVC or CCP file*
- **Bysoft 7 PartJob (*.partjob):** *This is the format for re-importing a job exported from BySoft CAM.*
- **Bysoft6 Bywork (*.zip):** *This format imports a job from Bysoft 6 (See: ›Importing a job from Bysoft 6‹ Page 1358).*



Tip: Use the **Migration Wizard** tool to import several jobs at once.

Import job

This example describes the basic procedure for importing jobs.

1. Select the **Import job** function.
2. In the **Import job** dialog box, select the relevant file and confirm the selection with **Open**.
3. In the **Job properties** dialog box, confirm the job details with **OK**. The job is opened in Part Nester.
4. Use the **Save** function to save the job in the job database. The job is now available for further editing.



Tip: The function can also be launched from the quick access toolbar *(See: ›Quick access toolbar‹ Page 11)*.

10.1.9 Export job



Where do I find the function?

Select: **Part Nester >** 

This function exports the current job. This file can be re-imported into BySoft CAM using the **Import job** function. The following export formats are supported:

- BySoft CAM (*.partjob)



Tip: Use this function to transfer jobs between different BySoft CAM workstations that do not have access to the same database.

Export job

1. In the **Export job** dialog box, select the folder in which you wish to save the job.
2. In the **File name** field, enter the name that you wish to save the job under (optional). The job name is used by default.
3. Confirm the selection with **OK**. The job is saved in the folder under the specified file name.

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10.1.10 Copy screenshot to clipboard

This function copies the displayed cutting plan onto the Windows clipboard as a screenshot. The screenshot can be retrieved from the clipboard in Word or in an image editor.



Note: This function is available in every menu except the **Machining time** menu.

10.1.11 Settings



Where do I find the function?

Select: **Part Nester >** 

In this dialog box you can define the view settings for the **Part Nester** user interface and the options for logging data about nesting operations.

Use the **Reset** button to restore the default settings for the tab.

Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

Tabs in the Settings dialog box:

- View
- Options
- Commands
- Log

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10.1.11.1 View

How do I open this tab?

Select: **Part Nester >** **> Settings**

This tab contains options for customizing the user interface of the **Part Nester** module. Personal settings are saved separately for each user.

Use the **Reset** button to restore the default settings for the tab.

What each option does:

- **Show grid**

  Displays a square grid in the workspace. The grid helps with orientation.

- **Show axes**

  Displays the origin of the object as axes crosshairs.

- **Show crosshairs**

  Displays the cursor additionally as crosshairs. The crosshairs are displayed over the full workspace.

- **Grid size**

  Specifies the minimum grid size for the optical grid When zooming in, the grid is displayed down to this minimum size.

- **Snap grid**

  If the **Align with grid** snap function is enabled, the value specified here defines the snap grid pitch, i.e. the gap between snap points. The default value is 1mm. The grid can be used for measurement functions or placing text, for example.

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Setting the color

Colored parts

This option can be used to enable/disable the colored display of parts in the cutting plan and in the **Parts** tab. If this option is enabled, all parts are displayed in different colors to distinguish them more easily. If this option is disabled, all parts are displayed in a gray tone.

Attention! Do not use luminous colors in the color preferences because these cause eye strain after a prolonged period.

Note: When customizing the color options, some elements will no longer be visible if you select identical or similar colors for different color options.

The color preferences are used to set the color display of the background, axes and grid in the Part Nester module. The elements in a job can also be displayed in particular colors.

10.1.11.2 Options

How do I open this tab?

Select: **Part Nester >** **> Settings**

This tab contains additional options for displaying the cutting time and the interactive buttons.

Verify plan

In the background

This option automatically performs a background check on the job whenever it is changed. The result is displayed with an icon in the workspace *(See: ›Icons in the workspace‹ Page 876)*.

Tip: Disable the option to increase performance on weak computers.

Waste calculation

With inner contours

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This option calculates cutting plan waste with the inner contours of the parts. If the option is disabled, cutting plan waste is calculated without inner contours.

Cutting time

Display cutting time in minutes

This option displays the cutting time in decimal format in minutes. If this option is not enabled, the cutting time is displayed in the «HH:MM:SS» format.

- Option enabled: **343.9min**
- Option not enabled: **05:43:54**



Tip: Use this option for calculating the total of several cutting times by hand.

Automatic cutting time calculation

When this option is used, the machining time for a new or modified job will be automatically calculated in the background. Job machining times are displayed in the **Machining time** menu. After enabling this option and pressing **OK** to quit the dialog, the program immediately calculates machining time.

The cutting parameter is selected automatically. If multiple cutting parameters exist in the same combination of machine, laser, material, thickness, etc., it is not possible to calculate cutting time. The parameter must be selected manually in the **Machining time** menu.



Tip:
Add a **$** character to the end of a parameter to set it as default. If there are multiple identical parameter values with different machining attributes, it is automatically selected for calculating cutting time.

Example:

Correct

- ByStarFiber3015_L605_6000_AW5754_1.5_N2_**$**.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2_**$**Quality.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2_Quality**$**.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2_Quality_**$**.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2_Quality_**$**_XY.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2_Quality_**$**Z.PAR

Wrong

- ByStarFiber3015_L605_6000_AW5754_1.5_N2**$**.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_N2**$**_Quality.PAR
- ByStarFiber3015_L605_6000_AW5754_1.5_**$**_N2.PAR



Note: If this option is enabled, the **Calculate** and **Remove** functions are disabled in the **Machining time** menu. Machining time can only be calculated automatically if the correct cutting parameter exists!

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Export to MS Excel

When experting a job to a job (.partjob) or BVC file (.bvc), these options are used to export information about parts and cutting plans to two Excel worksheets each.

All of the columns configured in the **Job Data** section of the **Parts** or **Cutting plans** tabs are exported to the Excel worksheet.

Parts

This option exports the information about all existing parts to an Excel worksheet.

Cutting plans

This option exports the information about all existing cutting plans to an Excel work-sheet.

Output path

The Excel files are exported to the path specified here. If no path is specified, the Excel files are exported to the directory in which the cutting plans are exported *(See: ›NC output path 1‹ Page 105)*.

Residual sheet management

Residual sheet management manages residual sheets from cutting plans based on the options configured in the appropriate folder.

Tip: You can display and edit the available residual sheets in Object Manager.

Active

Activates residual sheet management. If this option is disabled, residual sheets are not saved in jobs.

Minimum residual sheet width

Only residual sheets larger than the specified width are saved. A value of **0** stores all widths of residual sheets in the system.

Minimum residual sheet length

Only residual sheets larger than the specified length are saved. A value of **0** stores all lengths of residual sheets in the system.

Subfolder for material

Saves the residual sheet in the folder with the material designation.

Subfolder for thickness

Saves the residual sheet in the folder with the thickness designation.

> **Note:** Can only be activated if the **Subfolder for thickness** option is enabled.

Manage serial number

If this option is enabled, a serial number is assigned to each residual sheet and engraved on the residual sheet.

The serial number is appended to the residual sheet file name. The serial number for used residual sheets can also be displayed on the **Sheets** tab in the **Job Data** dialog.

Continue to use serial number

If this option is enabled, the residual sheets created from an existing residual sheet will have the same serial number as the original residual sheet.

> **Note:** Can only be activated if the **Manage serial number** option is enabled.

Current serial number

Displays the current number of the serial number used for residual sheets. The value is incremented by one for each residual sheet created.

> **Note:** The value can be customized.

Residual sheet management supports the following functions:

Save residual sheet

If residual sheet management is enabled and you export a job with one or several residual sheets, the **Update residual sheets?** prompt is displayed. Select **Yes** to save the residual sheets.

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Remove residual sheet

If residual sheet management is enabled and you create a job from a residual sheet, the **Update residual sheets?** prompt is displayed when you export the job. Select **Yes** to remove the residual sheet from residual sheet management.

Job name generator

Use the job name generator to automatically generate names for new jobs. If this option is enabled, a new job name is automatically input in the **Name** box in the **Job properties** dialog.



Tip: The use of a variable for **Job number** or **Date** prevents duplicate job names from being generated.

Active

Activates the job name generator.

Available variables

This section displays all the variables available for creating a job name. The variables can be entered in the **Name** field.

Name

Input the name and variables for the job name in this box.

Example

Displays the job name based on the definition in **Name** and **Current job number**.

Current job number

Lets you change the counter value. The number to be assigned next is always shown. If the counter value is changed, job numbering continues consecutively starting from the new number.



Attention! If you change the counter value manually, there is a risk that duplicate job names will be generated. This means that cutting plans may be overwritten when exported to the machine.

Press **Save** to apply the changed counter.

Other functions

Clustering parts

Enables clustering of parts. An additional **Cluster** tab is displayed in the job data *(See: ›Cluster‹ Page 1212)*.

Name of cluster (framed) - Prefixed text

When exporting a framed cluster, this defines the default name of the cluster followed by a serial number (e.g. FramedCluster-1).

Name of cluster (frameless) - Prefixed text

When exporting a frameless cluster, this defines the default name of the cluster followed by a serial number (e.g. FramelessCluster-1).

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10.1.11.3 Commands

How do I open this tab?

Select: **Part Nester >** **> Settings**

This tab contains options for logging information about nesting operations. Nesting operations are logged by the Part Nester module and Plant Manager.

General

Always display interactive buttons

If this option is enabled, the interactive buttons are always displayed when selecting one or more parts in the cutting plan. If this option is disabled, the buttons are not displayed for the selected part until the space bar is pressed. The buttons remain visible so long as the space bar is pressed.

Place parts

Without collision by default

If this option is enabled, the program prevents part contours from being positioned on top of each other when manually nesting parts. If you try to nest over the contour of another part, the part to be nested is automatically positioned in the next free area of the cutting plan.

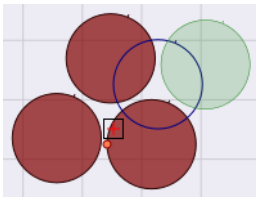


Fig. 10-3: Place part with option enabled

10.1.11.4 Log

How do I open this tab?

Select: **Part Nester >** **> Settings**

This tab contains options for logging information about nesting operations. Nesting operations are logged by the Part Nester module and Plant Manager.

- **Log active**

  Enables logging of the nesting operation in a log file.

  Send this file to Bystronic Customer Service if you notice inexplicable irregularities in the automatic nesting function or if the nesting aborts prematurely.

- **Log file**

  Enter the file name and storage location for the log file.

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10.2 Menu: Data

Where do I find the menu?

Select: **Modules > Part Nester**

In this menu, you can nest plans fully automatically based on the configured settings, and you can edit the sheets, parts and plans in the job.

Functional groups

The function bar contains the following functional groups:

- Job
- Sheet
- Part
- Plan
- View

10.2.1 Job properties



Where do I find the group?

Select: **Modules > Part Nester > Data > Job**

This function displays the properties of the current job in the tabs **General** and **Cutting technology**. The settings affect the cutting technology used, the machining time and the job export.

10.2.1.1 General



How do I open this tab?

Select: **Modules > Part Nester > Data > Job > Properties**

This tab displays the general properties of the current job.

☞ **Note:** If a job name has not been specified yet under **Name**, when you save the job, the **Save as** dialog box will open in which you can enter a name to save the job under.

Settings

- **Apply Settings Manager**

  This button applies the Settings Manager settings to the part based on the options selected in the drop-down box.

General

The general information describes the job and the system of units that are used. When the job is saved, various data is created automatically under the **Created**, **Changed** and **Revision** options, which cannot be changed manually.

- **Name**

  Displays the name of the job. The job name can only be changed subsequently in the **Object Manager**.

- **Path**

  Shows the path to the object.

- **Created**

  Displays the date and time at which the object was created and the name of the user who created it.

- **Changed**

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Displays the date and time at which the object was last saved and the name of the user who saved the object.

- **Revision**

  Displays the current revision number for the object. The revision number is increased by a count value every time the object is saved.

- **Description**

  You can enter a more detailed description of the current object in this field.

- **Info 1–8**

  Extra information about the current object can be entered in these fields.

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

Material

The information about the material is used for calculating the weight and for assigning the cutting technology.

- **Name**

  This field displays the name of the material. Clicking on the arrow symbol on the right lets you select the material from a list.

- **Thickness**

  This field displays the thickness of the material. Clicking on the arrow symbol on the right lets you select a thickness from a list. All the thicknesses available for the material are displayed. If you require a different thickness, enter it in the field from the keyboard.

Cutting process

The information about the machine is needed in order to assign the cutting technology and to calculate the machining time.

Information about the machine and cutting gas are used in the export settings pre-processor in the **Cutting Wizard** section as a filter criterion for selecting the Cutting Wizard *(See: ›Filter definitions‹ Page 1129)*.

Machine

This field displays the machine that shall be used to machine the object. Clicking on the arrow symbol on the right lets you select a different machine from a list. This list only contains those machines that are configured in the system. If you require a new machine, you must create it in the **Administrator** module in the **Cutting machines** form *(See: ›Cutting machine‹ Page 102)*.

Cutting gas

Use this box to select the cutting gas with which to cut the cutting plan.

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Cost table

You can assign a cost table to the job in this section. Based on this table, the program calculates the waste of the nested sheets and displays it as an icon in the workspace.

Profile (Tube Nester)

The profile type and its dimensions for the tube job are displayed in this area. Only those parts that have the same profile type and dimensions can be added to the job.

> **Note:** As soon as there is a contour in the work area, the profile type can no longer be changed.

10.2.1.2 Cutting technology



How do I open this tab?

Select: **Modules > Part Nester > Data > Job > Properties**

This tab displays the cutting technology properties for the current job.

Kerf

This option specifies the width of the kerf of the selected tool, thereby defining the distance between two contours that are cut using a common cut.

For precision common-cut parts, you need to enter the parts separation exactly as 2 x **Tool radius** *(See: › Finding the tool radius‹ Page 1360)*).

10.2.1.3 Big part removal



How do I open this tab?

Select: **Modules > Part Nester > Data > Job > Properties**

On this tab you can configure the properties for removal of big parts.

Default values

- **Big parts**
  Location where big parts are stored in the automation module.
- **Max. stack height**
  Maximum stack height for big parts at the storage site.
- **Tolerance to gripper contour**
  Minimum clearance from contour of part required to activate a suction cup.

Min. part area

Minimum part area for detecting a part as a big part.

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10.2.2 Group: Sheet

Where do I find the group?

Select: **Modules > Part Nester > Data**

This group contains all the functions for creating and editing sheets. At least one sheet must exist in order to be able to nest a job. New jobs do not yet contain any sheets. Therefore in new jobs, create a sheet first before adding parts to the job.

Note: All sheets in a job are displayed in the **Sheets** tab of the **Job data** docking dialog box. If this dialog box is not displayed, it can be shown using the **Job data** function in the **Window** menu.

10.2.2.1 Add



Where do I find the function?

Select: **Modules > Part Nester > Data > Sheet**

This function adds a new sheet format to a job. In the **Add sheet** dialog box, you can define the format using a template or define your own custom format using the length and width.

To add a residual sheet with a rectangular cut-out, select the **L-shape** attribute in the **Shape** selection field.

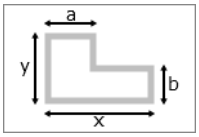


Fig. 10-4: L-shape residual sheet

The new sheet is displayed in the **Sheets** tab of the **Job data** docking dialog box.



Note: New templates for sheet formats can be created in the **Administrator** module under **System data** in the **Sheet format** form.

What each field does:

- **Shape**
  Defines the basic shape of the sheet.
  - **Rectangle**: Standard shape of a sheet
  - **L-shape**: Sheet with rectangular cut-out (residual sheet)
  - **Circle**: Sheet with round shape
- **Sheet format**
  Selection box for preset sheet formats.
- **Length (x)**
  Length of the sheet in the X-direction.
- **Width (y)**
  Width of the sheet in the Y-direction.
- **Distance a**
  Specifies the shortened X dimension of the cut-out for an L-shape sheet.
- **Distance b**
  Specifies the shortened Y dimension of the cut-out for an L-shape sheet.
- **Quantity**
  Number of sheets of this sheet format that you wish to add to the job.
- **Diameter**
  Indicates the diameter of a round sheet.

10.2.2.2 Remove

Where do I find the function?



Select: **Modules > Part Nester > Data > Sheet**

This function deletes all the selected sheets. It is also possible to delete sheets that already contain nested parts. If sheets that already contain nested parts are deleted, all the nested cutting plans of this sheet format are also deleted.



Tip: If you accidentally delete nested sheets, use the **Undo** function to restore the deleted sheets.

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10.2.2.3 Copy



Where do I find the function?

Select: **Modules > Part Nester > Data > Sheet**

This function adds all the sheets selected in the **Sheets** tab as a copy to the job.



Tip: You can use this function to create identical sheets with a different sheet edge.

10.2.2.4 Add residual sheet



Where do I find the function?

Select: **Modules > Part Nester > Data > Sheet**

This function is used to add a residual sheet to the **Sheets** tab.

In the **Open** dialog box you can select the residual sheet and then use the **Open** button to add it to the job. If a filter is selected in the dialog, only those residual sheets are displayed that match the filter criteria in the current job.



Attention! If a residual sheet is added with a different material or thickness, the program displays a warning after the symbol on the **Sheet** tab.



Note: If **Residual sheet management** is enabled in the options, the residual sheet is removed from the list after exporting the job and confirming the **Update residual sheets?** prompt if the quantity is <1.

10.2.2.5 Create sheet from part



Where do I find the function?

Select: **Modules > Part Nester > Data > Sheet**

This function can be used to create a sheet from the outer contour of any part. In the **Add part** dialog box, select the part from which you wish to generate a sheet, and confirm your selection with **OK**. A sheet is generated from the contour of the part.



Note: When creating the sheet from a part, the inner contours of the part are not included.

Application:

- To reduce the waste by cutting smaller parts from a large, incorrectly cut part.
- For subsequent cutting of semi-finished products (e.g. cutting ventilation holes in a pre-fabricated door).

10.2.2.6 Rotate sheet 90°

Where do I find the function?



Select: **Modules > Part Nester > Data > Job data > Sheet**

This function rotates the sheet format selected in the **Sheet** tab by 90°. Only those sheet formats can be rotated on which there are not yet any nested sheets.

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10.2.3 Group: Part

Where do I find the group?

Select: **Modules > Part Nester > Data**

This group contains all the functions for adding parts to a job or editing parts that have already been added.

A new job does not contain any parts yet and therefore parts must first be added to the job before nesting.

☞ **Note:** All parts in a job are displayed in the **Parts** tab of the **Job data** docking dialog box. If this dialog box is not displayed, it can be shown using the **Job data** function in the **Window** menu.

10.2.3.1 Add



Where do I find the function?

Select: **Modules > Part Nester > Data > Part**

This function lets you add one or more parts to a job. In the **Add part** dialog box you can select the parts and then use the **Open** button to add them to the job.

Selecting more than one part

- To select parts in different folders, use the **Add** arrow buttons. The selected parts are added to the **Selected elements** area.
- Within a folder, you can select several parts at once using the **Ctrl** + left mouse button and then add them directly to the job using the **Open** button.
- All parts located using the **Find** function are displayed in the **Find** tab and can be added to the job from there.

☞ **Note:** All the parts selected in the **Parts** folder and all the parts in the **Selected elements** section are added to the job.

Find

You can use the **Search** box to search for parts based on certain part names. Press **Enter** to display the search results on the **Find** tab.

You can use a list box to restrict the search according to certain criteria.

- **Entire text**
  The program searches for a text that matches the character string exactly.
- **Start of text**
  The program searches for a text that starts with this character string.
- **Part of text:**
  The program searches for a text that contains this specified character string.

Selected elements

This section displays all parts to add to the **Parts** tab.

- **Add search result**
  Option enabled: If the search only returns one part, this part is automatically added to the **Selected elements** section.

What each button does:

Button	Description
	Adds the selected parts to the **Selected elements** area.
	Removes the parts selected in the **Selected elements** area.

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10.2.3.2 Remove



Where do I find the function?

Select: **Modules > Part Nester > Data > Part**

This function deletes all the selected parts from the job. If parts are deleted that have already been nested in the job, these parts are also removed from the job concerned.



Tip: If you accidentally delete any nested parts, use the **Undo** function to restore the deleted parts. The deleted parts are placed back in the relevant jobs.

10.2.3.3 Copy



Where do I find the function?

Select: **Modules > Part Nester > Data > Part**

This function adds all selected parts, including all set options, as a copy to the **Parts** tab.



Tip: Use this function to nest the same part with different option settings (e.g. Common Cut or Filler part).

10.2.3.4 Extract



Where do I find the function?

Select: **Modules > Part Nester > Data > Part**

This function can be used to extract a part that is embedded in a job in order to save it in the database for parts. This situation arises when a job has been imported that contains parts which are not held in the parts database. The revision number stored on the part is used.



Note:

- This function is only available for embedded parts.
- If the selected folder contains parts with the same name, a dialog allows you to choose whether to overwrite of skip these parts.



Tip: You can select multiple parts to extract several embedded parts together.

Extracting one or more embedded parts

1. Select the embedded part on the **Parts** tab.
2. Click the **Extract** function in the context menu.
3. In the dialog, select the folder in which you wish to save the part.
4. Press the **Open** button. The part is saved to the specified folder.

10.2.3.5 Integrate



Where do I find the function?

Select: **Modules > Part Nester > Data > Part**

This function embeds the selected part in the cutting plan. It removes the link to the part in the Parts database.

All changes to the contour and cutting technology of the part only affect the nested instances of the embedded part in the cutting plan.

 **Note:** Embedded parts are flagged with the symbol for this function. 

 **Tip:** Use the **Extract** function to save the part in the Parts database again.

10.2.3.6 Edit



Where do I find the function?

Select: **Modules > Part Nester > Data > Part**

This function lets you edit the selected part in the Part Editor. The job is locked so long as the part is open in the Part Editor. If changes are made to the contour of the part, the change is applied to all contours of the part that have already been nested.

 **Attention!** If the outer contour is modified for nested parts, the part may overlap other nested parts. Therefore re-nest the plan or use the **Verify plan** function in the **Export** menu to check whether the part will overlap other parts in the plan.

 **Note:** Note that changes made to the outer contour have an impact on all the jobs in which the part is nested. When adding new contours to a part, the part must be re-nested and the cutting technology assigned.

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10.2.3.7 Edit as copy



Where do I find the function?

Select: **Modules > Part Nester > Data > Part**

This function lets you save a copy of the selected part in the parts database and edit this copy in Part Editor.

The job remains locked as long as you are editing the part. Any edits made to the part do not affect the part in the current job.

10.2.3.8 Replace



Where do I find the function?

Select: **Modules > Part Nester > Data > Part**

This function replaces the selected part with another part. To do this, in the **Replace part** dialog box, select the part that will replace the previously selected part.

⚠ **Attention!** If the replaced part is already nested in plans, the replacement part may overlap other nested parts. Therefore use the **Verify plan** function in the **Export** menu to check whether the part will overlap other parts in the plan. When the job is exported, all plans in the job are also checked for overlaps.

10.2.3.9 New part



Where do I find the function?

Select: **Modules > Part Nester > Data > Part**

This function includes a number of methods for creating a new part, which can then be added to the job. In the Part Editor, assign the machine and cutting technology to the part before adding the part to the job.

☞ **Note:** The job is locked from further editing so long as the part is open in the Part Editor.

Creating a new part and adding it to the job

1. Select the **New part** function.
2. Select the **New part** creation method.
3. In the **New part** dialog box, specify the name, system of units, material and thickness of the part.
4. Use the appropriate method to create the part.
5. Assign the machine and cutting technology to the part.
6. Close the part and confirm the prompt to save with **Yes**.
7. In the **Save as** dialog box, save the part in the parts database. The part is added to the job.

You can add a new part to the job using the following methods:

- New part
- Import part
- Import parts from clipboard
- Import part from file

Import parts from clipboard



Where do I find the function?

Select: **Modules > Part Nester > Data > Part > New Part**

With this function you can paste parts from an Excel spreadsheet into the job via the clipboard.

The heading in the first row of the table can be freely chosen for each column. However, it should reflect the meaning of the column. Enter the parts with the appropriate details in subsequent lines.

The table columns must be structured as follows:

Key to columns:

1. Part name or full path to the part in the database
2. Number of parts
3. Order information
4. Info 1
5. Info 2
6. Info 3
7. Color of the part in hex code (#FF0000 = red)

Requirements:

- The parts must exist in the parts database.
- The order of columns must be correct.

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Part name	Quantity	Order Information	User info 1	User info 2	User info 3	Color
Sample_01	10	Sample 01 OrderInfo	Only part name			#3333FF
Sample_02	15	Sample 02 OrderInfo	Only part name			#03910A
c:\BystronicData\BySoftCam\Parts\Samples\Sample_03	20	Sample 03 OrderInfo	Full path to part in database			#FF0000
c:\BystronicData\BySoftCam\Parts\Samples\Sample_04	25	Sample 04 OrderInfo	Full path to part in database			#FFC50D

Fig. 10-5: Table structure for importing parts from the clipboard

Import parts from clipboard

1. Open the table with the import details of the parts.
2. Copy the cells with the import details of the parts to the clipboard.
3. Select the **New part** function.
4. Select the **Import parts from clipboard** creation method.
5. The parts are added to the job on the **Parts** tab.

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Import parts from file



Where do I find the function?

Select: **Modules > Part Nester > Data > Part > New Part**

With this function you can paste parts from an Excel spreadsheet into the job.

The heading in the first row of the table can be freely chosen for each column. However, it should reflect the meaning of the column. Enter the parts with the appropriate details in subsequent lines.

The table columns must be structured as follows:

Key to columns:

1. Part name or full path to the part in the database
2. Number of parts
3. Order information
4. Info 1
5. Info 2
6. Info 3
7. Color of the part in hex code (#FF0000 = red)

Requirements:

- The parts must exist in the parts database.
- The order of columns must be correct.

Part name	Quantity	Order Information	User info 1	User info 2	User info 3	Color
Sample_01	10	Sample 01 OrderInfo	Only part name			#3333FF
Sample_02	15	Sample 02 OrderInfo	Only part name			#03910A
c:\BystronicData\BySoftCam\Parts\Samples\Sample_03	20	Sample 03 OrderInfo	Full path to part in database			#FF0000
c:\BystronicData\BySoftCam\Parts\Samples\Sample_04	25	Sample 04 OrderInfo	Full path to part in database			#FFC50D

Fig. 10-6: Table structure for importing parts from an Excel file

Import parts from an Excel file

1. Select the **New part** function.
2. Select the **Import parts from file** creation method.
3. In File Explorer, select the Excel file with the import details of the parts and click the **Open** button to confirm your selection.
4. The parts are added to the job on the **Parts** tab.

10.2.3.10 Part properties

How do I open this dialog box?

Select: **Modules > Part Nester > Data > Parts**

All the properties of the selected part are displayed in this dialog box. The properties correspond to the columns in the **Parts** tab.

General

This section displays general settings and a preview of the part. If a gripper if set on the part for sorting, it is also shown in the preview.

- **Name**

  Shows the name of the part. The name is used for unique identification of a part.

- **Revision**

  Displays the current revision number for the object. The revision number is increased by a count value every time the object is saved.

- **Quantity**

  Shows the available quantity of a part.

- **Extra quantity**

  Shows the number of additionally nested parts.

- **Color**

  Displays the color of the part. The part can be assigned a different color by clicking on the left arrow button.

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Part information and nesting settings

These options can be used to export information and nesting settings from the saved part to Part Nester.

- **Parts**

  Exports the information from the selected part or from all parts.

- **Action**

  Exports the information from the selected area.

  - **Part information**

    Exports the information from the fields Info 1–3.

  - **Nesting settings**

    Exports all settings in the **Nesting Settings** section.

  - **All**

    Exports all information and settings from the two areas.



Note: The changes are only applied if the job is opened after editing the part.

Information

- **Customer**

  Displays the name of the customer. For jobs from Plant Manager, the name is taken from there. If no name is available, it can be entered manually.

- **Order info**

  This field is not intended for any particular purpose. You can use this field to enter any information you wish.

- **Info 1-6**

  Additional information about the part can be entered in these fields.

Nesting setting

- **Parts protective distance**

  Shows the size of the safety margin placed around each part during nesting.

- **Priority**

  You can use this option to set the order in which the different parts are to be nested. Entering a priority is optional. Enter a number between 1 and 99 for the priority, Where 1 stands for highest priority and 99 for lowest priority. No priority is assigned if 0 is entered.

Attention! The use of different priorities does not usually result in optimum nesting of the cutting plan.

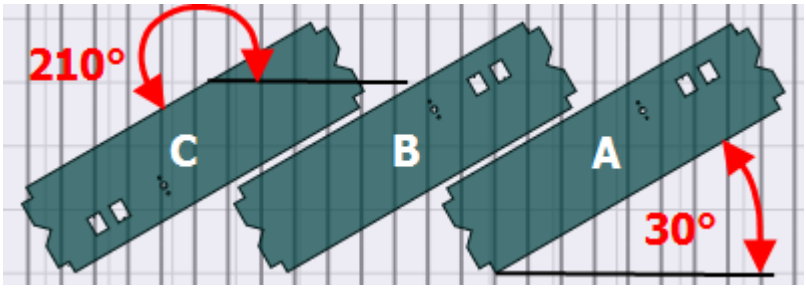
Note: This function is only used for automatic nesting.

- **Same orientation**

  This option nests all instances of the part in the same orientation on the sheet.

Example:

If the part is nested with an angle of **30°** (**A**), the other instances can only be nested with the same angle (**B**) or with an angle of **210°**=180°+30° (**C**).



Application:

- Parts with same direction of grinding
- The correction value only needs to be entered on the bending machine for the first bending part. Because the alignment of the grain direction is the same, it is not necessary to enter a correction for all other parts.

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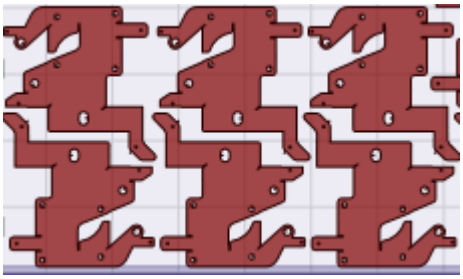


Fig. 10-7: Same orientation of a bending part on the sheet

- **Increment**

  This field defines the increment of the angle of rotation in degrees. The part can be rotated during nesting through this angle.

> **Tip:** A rotation allowance can be entered by clicking on the input field or by selecting a set angle value via the left arrow button.

- **Automatic increment**

  Copies the rotation allowance from the parts in the cluster. The cluster can only be rotated by the angles that are valid for all parts in the cluster.

> **Note:** This option is only displayed for cluster parts!

- **Precise rotation**

  If this option is enabled, the part can be rotated by any angle. The angle of rotation can be specified to two decimal places. The input range < 1° is from 0.01 to 0.99°.

- **Grain direction**

  This option accounts for the grain direction when nesting the part.

  The part is positioned so that the part's grain direction matches the sheet's grain direction. All instances of the part are nested with the same angle of rotation or rotated through 180° *(See: ›Grain direction compensation (whole part)‹ Page 630)*.

> **Note:** The grain direction of the sheet can be specified on the **Sheet** tab in the **Grain direction** column *(See: ›Grain direction‹ Page 1194)*.

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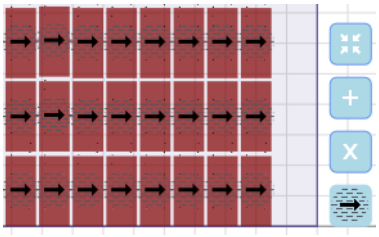


Fig. 10-8: Horizontal grain direction of the sheet from **0°**

- **Discrete angles of rotation**

  You can enter multiple angles of rotation in this box. The part can be rotated during nesting through this angle. The angles of rotation must be separated from each other by semicolons (**;**).

> **Tip:** If this field contains angles of rotation, the angle of rotation in **Rotation allowance** is ignored (not for cluster nesting).

- **Mirroring authorization**

  This option permits the part to be mirrored about the vertical axis during nesting.

- **Filler part**

  Enable this option if you want to use this part to fill the areas leftover from nesting, or if the part is often needed in larger, unspecified quantities.

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- **Test part**

  Flags the part as a test part. In Part Nester all instances of the part are nested as test parts *(See: ›Nesting a test part‹ Page 1375)*.

- **Sheet format**

  This option is for information only and indicates the sheet format on which to cut the part.

- **Part in cluster**

  This option specifies that the part can only be nested in a framed cluster *(See: ›Framed cluster‹ Page 1213)*.

- **Unique in cluster**

  This option specifies the part can only be nested in a cluster. It is not possible to nest it outside a cluster.

- **Take cutouts into account**

  These options determine whether or not to apply cutouts that were set on a part in Part Editor.

  - **Default**

    If the part contains cutouts, they will be taken into account for nesting if the parameter is enabled in the nesting settings.

  - **No**

    No cutouts are taken into account for nesting.

    Attention! The cutouts of a part are changed if the contour of another part is nested in a cutout. Based on the contour, the cutout is divided by the contour.

  - **Yes**

    If the part contains cutouts, they are always taken into account for nesting.

    Note: The cutouts are taken into account even if the parameter is disabled in the nesting settings.

- **Take flexible tandem gripping into account**

  This option can be used to nest large parts to be sorted with tandem grippers with a specific angle of rotation on a cutting plan.

  In order for the parts to be rotated by a certain angle, the option must be enabled in the nesting settings and in the part properties.

- For Dual Bridge sorting units, the option is always enabled by default and cannot be disabled.
- For One Bridge sorting units, the option is enabled by default for new jobs, but can be disabled.



Note: Parts with this option can only be sorted manually on a stack carrier.

Common Cut settings

This function allows you to combine parallel contours during nesting based on certain criteria.

- **Activate**

  Activates Common Cut nesting for the part.
- **Identical parts only**

  Only identical parts will be used when combining contours.
- **By pairs**

  Common cuts will only be placed between two parts.
- **Max. cluster size**

  Defines the side length of a square inside which instances of the part can be nested as a Common Cut cluster. If this value is exceeded, a new Common Cut cluster will be started.

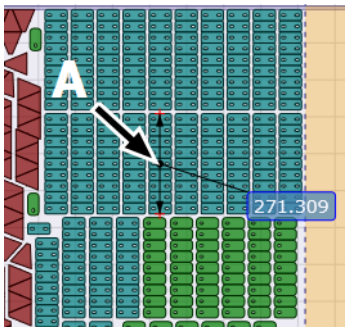


Fig. 10-9: Common Cut nesting with a cluster size of 300 mm (**A**)

- **Max. number of parts**

  Used to specify the maximum number of parts joined by common cuts.
- **Identical edges only**

  Will only join equal-length edges of a part as common cuts.
- **Min. area for new cut-outs**

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Used to set the minimum area for a cut-out (**A**) produced by common-cut nesting in which the part is being nested as a common cut.

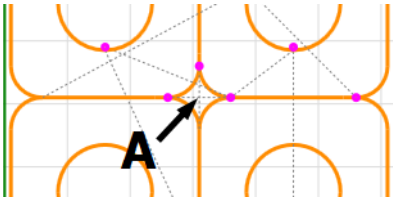


Fig. 10-10: Inside contour of a common-cut nesting arrangement

Max. radius

Used to set the maximum radius after a straight contour at which common-cut nesting will still be performed.

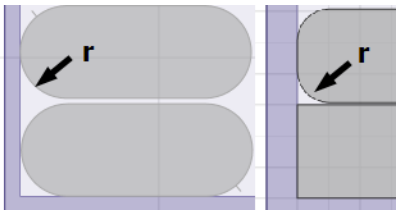


Fig. 10-11: Maximum radius for common cuts

Min. angle

Used to set the minimum angle after a straight contour at which common-cut nesting will still be performed.

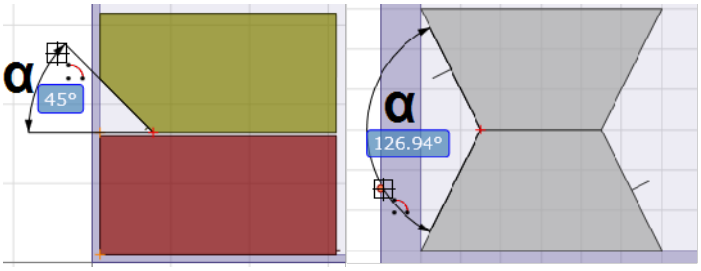


Fig. 10-12: Minimum angle for common cuts

Allow for lead-ins/lead-outs

This option can be used to achieve an optimized nesting result and improved process reliability. If this function is enabled, the lead-ins and lead-outs are handled by the software like a contour. This means that a parts protective distance is also placed around the lead-ins and lead-outs at nesting.

Copy and paste settings

Use these functions to copy the Common Cut settings of one part and assign them to other parts.

- **Copy**

  Copies the Common Cut settings of the selected part to the clipboard.

- **Insert**

  Assigns the copied Common Cut settings to the parts based on the selected part options.

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Export setting

- **Automatic microjoint**
  Microjoints are set automatically by the post-processor.

Serial-number information

Start number

This field displays the number with which the numbering of the selected part will begin the next time the serial-number information is assigned automatically in the **Cutting technology** menu.

Example:

- The start number in the field is 1.
- There are 30 parts that are the same in the cutting plan.
- The **Serial-number information** field is assigned to the part and it has a cutting technology.

1. After the **Automatic** serial function has been started, the values **0001-0030** will be assigned to the number of parts in the cutting plan. The value in the **Start number** field will change to **31**.
2. The next time the **Automatic** function is started, the next part will be assigned the value **0031**. The value in the **Start number** field will change to **62**, etc.

Tip: Serial-number information on a part can be displayed by hovering the mouse over the part.

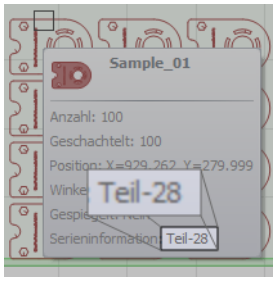


Fig. 10-13: Drop-down display of the serial-number information on a part

- **Increment**

  The counting sequence of the numbers on the parts is specified in this field.

  Example:

Increment	Counting sequence
1	1, 2, 3,…n
2	1, 3, 5,…n
3	1, 4, 7,…n

- **Min. number of digits**

  Determines the minimum number of places of a number. If the number is smaller than the number given here, the missing places are padded with zeros.

  Example:

  Minimum number of places = **4**

  Part counter = **95**

  Number on part = **0095**

- **Preceding text**

  The text is set before the serial-number information.

- **After-text**

  The text is set after the serial-number information.

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10.2.4 Group: Plan

Where do I find the group?

Select: **Modules > Part Nester > Data**

This group contains all the functions for managing the cutting plans in a job and saving residual sheets.

Note: All created cutting plans for the job are displayed in the **Cutting plans** tab of the **Job data** docking dialog box.

10.2.4.1 Add plan



Where do I find the function?

Select: **Modules > Part Nester > Data > Plan**

This function is used to add an empty cutting plan to the **Cutting plans** tab. All available sheets are displayed in the **Add plan** dialog box.

If the dialog box does not display any sheets, the number of sheetsfor the relevant sheet size must be increased by at least one sheet in the **Sheets** tab. The sheet will then be displayed in the **Add sheet** dialog box and can be selected for a cutting plan.

Tip: You can also create a new cutting plan by dragging a sheet into the cutting plan area in the **Sheets** tab.

10.2.4.2 Remove plan



Where do I find the function?

Select: **Modules > Part Nester > Data > Plan**

This function deletes all the selected cutting plans from the job.

Tip: If you accidentally delete any cutting plans, use the **Undo** function to restore the deleted plans.

10.2.4.3 Copy plan



Where do I find the function?

Select: **Modules > Part Nester > Data > Plan**

This function adds all selected plans, including all set options, as a copy to the **Cutting plans** tab.



Note: A cutting plan can only be copied if there are enough parts available for the plan.



Tip: Instead of copying a plan, you also have the option of increasing the number of machining cycles.

10.2.4.4 Save residual sheet

Where do I find the function?



Select: **Modules > Part Nester > Data > Plan**

This function saves the selected residual sheet in Object Manager. All residual sheets are saved under the **ResidualSheets** folder. You can create your own folder structure for residual sheets below this folder.



Tip: Use the default name composed of material, thickness, size and unit for the residual sheet. This helps you to get a quick view of the sheet attributes when adding residual sheets.

Save residual sheet

1. Select the residual sheet in the cutting plan to save.
2. Select the **Save residual sheet** function from the shortcut menu.
3. Enter the name of the residual sheet in the **Save as** dialog. The default name is made up of material, thickness, size and unit.
4. Press **Save** to save the residual sheet in the relevant folder.

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Residual-sheet rotation angle

Where do I find the function?

Select: **Modules > Part Nester > Data > Plan > Save residual sheet**

In this dialog box, you can select which residual sheet to save when there are several residual sheets available, and whether to rotate the sheet through 90° prior to saving.

Rotation can be used to save a portrait-format residual sheet as a landscape format. This allows edge detection to be used for smaller sheets in landscape format, which would no longer be possible in portrait format because of the detection distances.

10.2.4.5 Zoom out plan



Where do I find the function?

Select: **Modules > Part Nester > Data > Plan**

This function reduces the size of the cutting plan to the nested section. A new sheet is created for the reduced cutting plan.



Note: The sheet previously used is added to the number of unused sheets again.

10.3 Menu: Nesting

Where do I find the menu?

Select: **Modules > Part Nester**

In this menu, you can nest plans fully automatically based on the configured settings, and you can edit the sheets, parts and plans in the job.

Functional groups

The function bar contains the following functional groups:

- Selection
- Technology
- Nesting
- Nesting window
- Sectors
- Cycle

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10.3.1 Group: Selection

Where do I find the group?

Select: **Modules > Part Nester > Nesting**

This group displays all the available selection functions. These functions can be applied to all elements of an object.

10.3.1.1 Same part



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Selection**

This function is used to select all references of the selected parts on the cutting plan.



Tip: Use the function to remove all references of a part from the cutting plan. By subsequently selecting the **Invert** function it is possible to remove all references of the unselected parts from the cutting plan.

10.3.2 Group: Technology

Where do I find the group?

Select: **Modules > Part Nester > Nesting**

This group contains all the functions for automatic nesting of the parts.

10.3.2.1 Automatic



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Technology**

This function starts the fully automated cutting-plan creation process. The fully automatic procedure for creating the cutting technology involves the following process steps:

- All available parts are nested on the available sheets.
- All nested parts are assigned a cutting sequence (if enabled).
- Skeleton cuts and residual-sheet cuts are set, based on the relevant settings (if option enabled).
- Sets a tilt-optimized lead-in on all parts (if enabled).



Tip: Automatic nesting can be started from all menus by pressing **F4**.

Other technology functions

Clicking on the **Automatic** button offers the following additional technology functions:

- **Automatic (leave existing clusters)**

  This function deletes all existing plans and creates new plans. Existing clusters are retained and new clusters are nested on existing cluster frames.

  First, all clusters are nested. Then all parts that are not nested in clusters.

  Note: An automatic microjoint is placed on all parts in the cluster.

- **Complete**

  With this function, all parts which have not yet been used in the job are nested in existing or new cutting plans and automatically have the cutting technology assigned to them. This does not affect parts that have already been nested.

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First, the program tries to nest the parts in free areas on existing plans. If it is not possible to nest all parts in existing cutting plans, new cutting plans are generated.

If the jobs uses clusters, parts in the cluster are not changed. If the number of parts in the clusters is greater than specified in the **Quantity** column on the **Parts** tab, additional parts are automatically entered in the **Extra quantity** column.

☞ **Note:** With regard to the machining sequence of existing plans, new parts are appended after existing parts.

- **Complete (leave existing plans)**
  This function is used to nest all parts of the job that have not yet been nested. **No** parts are nested in free areas of existing plans.
- **Settings**
  This function opens the **Cutting technology settings** dialog in which you can edit technology options.

10.3.2.2 Automatic nesting



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function starts automatic nesting of cutting plans to sort after cutting. Selecting the  button will nest the entire job.

Note: This nesting mode includes special settings for sorting parts.

Attention! Parts that are already nested in the cutting plan are re-nested.

Clicking on the **Automatic** button offers the following additional nesting modes:

Note: Parts that are already nested in the cutting plan are retained.

- **Complete**
  This mode is used to nest all parts of the job that have not yet been nested. Parts can be nested in free areas of existing plans.
- **Complete (leave existing plans)**
  This mode is used to nest all parts of the job that have not yet been nested. **No** parts are nested in free areas of existing plans.
- **Settings**
  This button opens the **Nesting settings** dialog box in which the nesting options can be modified.

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10.3.2.3 Settings



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Technology**

You can use this function in the relevant docking dialog box to set the options for fully automated assignment of the cutting technology. Running the **Automatic** function will apply the settings to the entire job.

Nesting

Where do I find the function?

Select: **Modules > Part Nester > Nesting > Technology > Settings**

All settings used for nesting the parts are defined in this section.

General

The options in this area can be used to control the form and efficiency of nesting.

Automatic nesting time

- **enabled:** The computed nesting time will be used for nesting.
- **disabled:** The nesting time set in **Nesting time** will be used for nesting.

Nesting time

Used to set the time, in seconds, that the module is allowed to take when nesting cutting plans.

Computed nesting time

Shows the optimum nesting time computed by BySoft CAM. This nesting time will be affected by the number of contour elements per part and the number of parts that need to be nested.

☞ **Note:** If the **Automatic nesting time** option is enabled, this is the value used for nesting time.

Nesting contours

Defines the resolution for arc contours in segments during nesting. This setting affects the nesting time and the nesting accuracy.

- **Coarse** An arc contour will be resolved into a few segments. The parts in the cutting plan will be nested inaccurately, but very quickly.
- **Average** An arc contour will be resolved into an average number of segments. The parts in the cutting plan will be nested accurately and sufficiently quickly. This is the default setting.
- **Fine** An arc contour will be resolved into a large number of segments. The parts in the cutting plan will be nested very accurately, but relatively slowly.

Parts protective distance

This option prevents collisions when cut parts protrude. During nesting, the specified parts protective distance (**A**) is placed around the outer contour of the part. Hence the minimum distance between two parts equals at least twice the parts protective distance (**B**). The Parts protective distance is not applied to the sheet edge. The parts are nested right up to the edge of the cutting plan.

Attention! If the **Include lead-ins and lead-outs** option is not enabled, the Parts protective distance must be at least as large as the longest lead-in or lead-out. Otherwise the lead-ins or lead-outs for a part may lie within the contour of an adjacent part.

Note: The lead-ins are only taken into account in nesting if the **Include lead-ins and lead-outs** option is enabled.

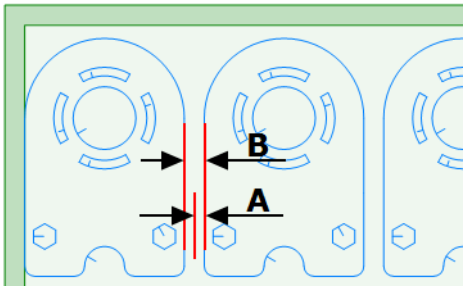


Fig. 10-14: Minimum distance between two parts: A = parts protective distance, B = double parts protective distance

Minimum parts protective distances

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Material	Mild steel				Stainless steel	Aluminum
Assist gas	Oxygen				Nitrogen	
Thickness [mm]	1–5	6–10	12–15	16–25	1–25	1–15
Minimum parts protective distance	5	5	7.5	12.5	5	5

Include lead-ins and lead-outs

This option can be used to achieve an optimized nesting result and improved process reliability. If this function is enabled, the lead-ins and lead-outs are handled by the software like a contour. This means that a parts protective distance is also placed around the lead-ins and lead-outs at nesting.

Attention! This option is not taken into account for Common Cut nesting.

Note: When this option is enabled, the **Parts protective distance** can be set to the minimum spacing required. This achieves better utilization of the sheet by nesting the parts closer together.

To achieve optimum nesting with this function, the following additional option settings are required:

- Nesting type = Default
- Strip width = 1mm

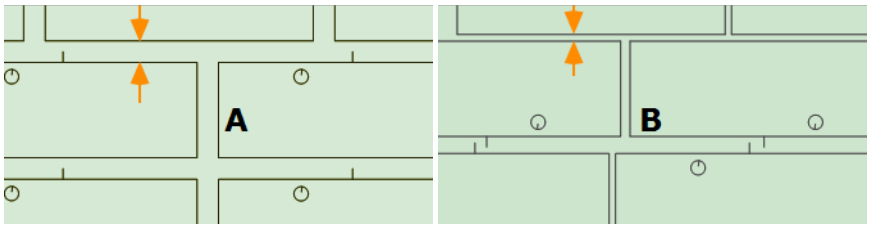


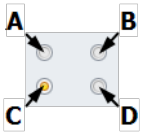
Fig. 10-15: Nesting with option disabled (**A**) and enabled (**B**)

Take cutouts into account

This option takes into account the cutouts defined on a part in Part Editor.

Nesting start

This option defines the starting point of nesting on a sheet. Depending on the selection of the **Nesting direction** option, nesting is performed horizontally or vertically starting from the nesting start.



Key:

A	Starting from top left
B	Starting from top right
C	Starting from bottom left
D	Starting from bottom right

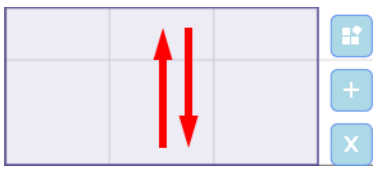
Nesting direction

This option defines the preferred direction for nesting parts on a sheet. You can use this function for partially nested sheets to set the position of the parts on the sheet.

☞ **Note:** The nesting direction is additionally influenced by the **Nesting start** option.

- **Vertical**

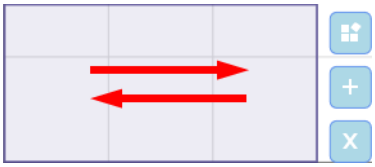
  Nests the parts vertically on the sheet (default).



- **Horizontal**

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Nests the parts horizontally on the sheet.



Priority

This option specifies the priority that will be used to nest parts on a sheet.

- **Parts priority**

  If different priorities have been assigned to the parts, the parts are nested in the order of priority 1, 2, 3, n.

- **Small parts first**

  The part nesting sequence starts with the smallest and ends with the largest parts.

- **Large parts first**

  The part nesting sequence starts with the largest and ends with the smallest parts.



Tip: These attributes can be used in combination with the **Nesting direction** option to facilitate removal of parts on the machine.

Filler parts strategy

This option specifies the way filler parts are used on a sheet.

- **Only nesting area**

  Filler parts are only nested in that area of a sheet used by normal parts. Any residual sheets available are no longer considered for nesting filler parts.

- **Only nesting area on last sheet**

  Filler parts are only nested in that area of the last sheet used by normal parts.

- **On entire sheet**

  If a sheet cannot be fully nested with standard parts, the unused area of the sheet is nested with filler parts. A residual sheet is not produced. Enable this function in order to nest filler parts in spare areas of a sheet.

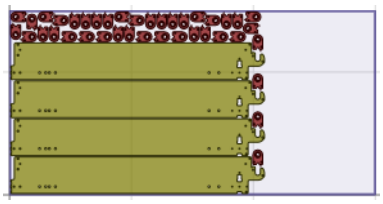


Fig. 10-16: Nested filler parts in nesting area of a sheet

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Min. area for inner-contour nesting

This option defines the minimum surface area of the inner contour of a part into which a part can still be nested. The larger the value, the fewer the parts that can be nested in inner contours.

☞ **Note:** The parts protective distance is included in the calculated value of the surface area. **Calculation:** *Area of inner contour - Parts protective distance ≥ Area of outer contour + Parts protective distance*

In order for a part to be nested inside the inner contour of another part, the following conditions must be met:

- The area of the inner contour must be greater than/equal to the value specified under **Minimum area for inner-contour nesting**.
- The area of the part to be nested inside the inner contour of another part must be less than the value specified under **Minimum area for inner-contour nesting**.

More than one machining cycle allowed per plan

This option allows a cutting plan to be cut several times if a sufficient number of parts are present.

Final optimization

This option can be used to apply the selected attribute additionally to the cutting plan after nesting. The **Compress**, **Expand** and **Shake** attributes are equivalent to the functions with the same names in the **Nesting** group. The attribute **None** means that no optimization is performed after the nesting process.

Take flexible tandem gripping into account

This option can be used to nest large parts to be sorted with tandem grippers with a specific angle of rotation on a cutting plan.

In order for the parts to be rotated by a certain angle, the option must be enabled in the nesting settings and in the part properties.

- For Dual Bridge sorting units, the option is always enabled by default and cannot be disabled.
- For One Bridge sorting units, the option is enabled by default for new jobs, but can be disabled.

☞ **Note:** Parts with this option can only be sorted manually on a stack carrier.

Reduce number of plans

If this option is enabled, the program tries to reduce the number of different nestings of cutting plans to cutting plans with multiple passes.

Sort

If the **Sort** option is enabled, the program attempts to position parts of the same type parallel to each other in the Y direction. Parts protective distance for default nesting is ignored.

Settings taken into account when nesting sorting plans:

- Parts protective distance
- Nesting direction

Nesting settings ignored when nesting sorting plans:

- Final optimization
- Common Cut
- Filler parts
- Small parts mode
- Cluster

Active

Enables nesting of sorting plans.

Parts protective distance

Specifies the spacing between parts for nesting sorting plans.



Note: The parts protective distance for sorting plans should be **4-5** times the parts protective distance for nesting normal cutting plans.

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Group nesting

Nests parts in groups on sheets to facilitate sorting of orders.

All parts with the same attributes in fields 1 thru 3 can be nested as groups together in one or more cutting plans.

Application:

- Enables job-based sorting of parts on sheets.
- Create automatically grouped jobs in Plant Manager based on the setting in **Settings Manager**.

Max. commenced groups per sheet

This option defines the maximum number of commenced sheets that may be on a sheet.

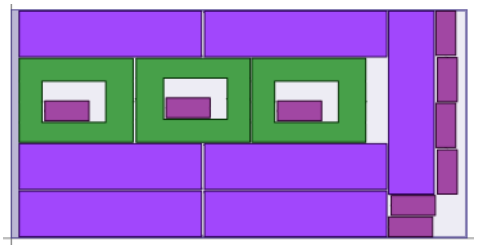


Fig. 10-17: Cutting plan with three groups per sheet.

Attention! The smaller the number of groups per sheet, the greater sheet waste will be.

Field 1–3

These fields define the criteria for forming subgroups.

In a typical application, the attribute **Order info** is used in field **1**. This attribute nests all parts with the same **Order info** as a group.

Another application is grouping by start and end date. The **Start date** attribute in Field **1** is used to nest all parts with the earliest date first and the latest date last on the sheets.

If an attribute is also entered in field **2** and **3**, all parts with the same attributes are nested as a group.

Parts nested on multiple sheets without group nesting

The parts are partially nested across all sheets.

Parts	Sheet 1	Sheet 2	Sheet 3	Sheet 4	Sheet 5	Sheet 6
Part 1						
Part 2						
Part 3						
Part 4						
Part 5						
Part 6						
Part 7						
Part 8						
Part 9						
Part 10						
Part 11						
Part 12						

Parts nested on multiple sheets without group nesting

The parts are nested in groups on a maximum of four sheets.

Parts	Sheet 1	Sheet 2	Sheet 3	Sheet 4	Sheet 5	Sheet 6
Part 1						
Part 2						
Part 3						
Part 4						
Part 5						
Part 6						
Part 7						
Part 8						
Part 9						
Part 10						
Part 11						
Part 12						

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Common Cut

Kerf

This value is used to define the kerf taken into account when combining common cuts.

Min. length

This option is used to set the minimum length of a cut that can still be made as a common cut.



Note: Minimum length must not be less than **10 mm**.

Material costs/cutting costs

Common Cut can be optimized using this option to reduce material consumption or cutting costs. The corresponding slider can be moved to the left or right in order to give greater priority to the corresponding option.

Small parts mode

The options in this section can be used to nest small parts in specific peripheral zones on a sheet.

If there are a lot of small parts, it is possible that parts will only be nested in the peripheral zone on a new sheet. Accordingly, make sure to set a sufficiently large maximum edge distance so that there will be a balance between the parts nested in the peripheral zone and the parts nested in the remaining sheet.



Note: This option requires additional computing time on top of the time specified in **Computed nesting time**.



Tip: The small parts area on a sheet can be shown using the special view by using the **Small parts area** option (**A**).

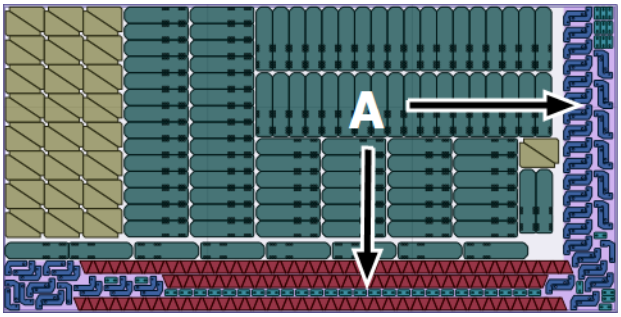


Fig. 10-18: Small parts nested in right and lower peripheral zones

Active

Enables small parts nesting in specific zones on the sheet.

Zones

Used to define the zones on the sheet on which the parts can be nested. Zones can be combined with each other.

Max. edge distance

Used to define the width of the zone, along the perimeter of the sheet, in which small parts can be nested.

Small parts only in zone

All small parts will be nested exclusively in the specified zone.

If there is not enough space inside the zone, the parts will be nested on a new sheet in the same zone.

Max. small part area

Used to specify the maximum size of the parts that can be used as small parts.

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Sector nesting

You can set the options for nesting the parts in sectors of ByVention cutting plans in this area.

☞ **Note:** This area is only displayed for ByVention jobs.

Max. sector width

The sectors will not be created any wider than the value specified here. The maximum sector width which can be specified is determined by the working area of the machine. The maximum sector width of the machine is specified in the **Cutting machine -> Properties** form for the administrator.

Sector clearance

This option specifies the minimum distance of the parts to the separating cut.

Optimize sector width

When this option is enabled, the sector width in automatic nesting is adapted to fit the nested parts. This achieves better utilization of the sheet.

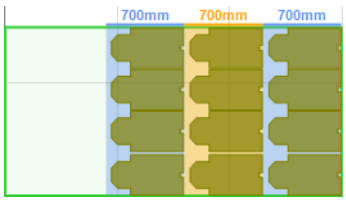


Fig. 10-19: Sector width without optimization

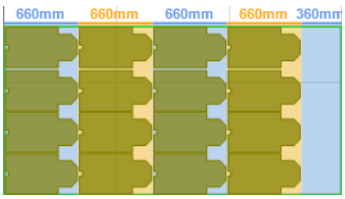


Fig. 10-20: Sector width with optimization

Overlapping sectors

Use this option to specify the nesting strategy for the parts. Regardless of the strategy, the maximum part size must lie within the cutting area. If no part fits into such a sector during automatic nesting, a sector will be added without overlapping.

- **Without overlapping (option not enabled)**
  The sectors are arranged without overlapping.

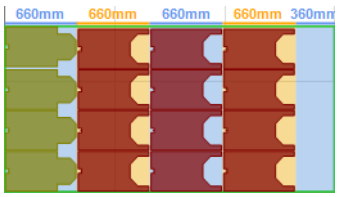


Fig. 10-21: Sectors without overlapping

- **With overlapping (option enabled)**
  The sectors are arranged with overlapping (**A**).

> **Note:** When the option is enabled, the process reliability could be reduced due to separating cuts that are not continuous. For this reason, enable the **Detour** separating cut option in the **Cutting technology settings**.

> **Tip:** Use this option if the parts cannot be nested optimally within one sector. Under certain circumstances, this can make it possible to nest more parts on one sheet.

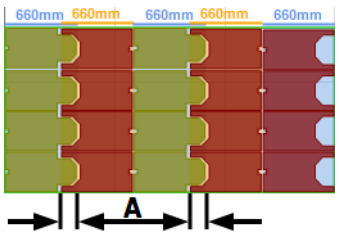


Fig. 10-22: Sectors with overlapping

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Fill with empty sectors

This option fills blank areas of a sheet with sectors.

Application:

- To cut blank areas of a sheet into individual parts.
- To create sectors for manual nesting.

Multihead



Note: These functions are only available if a waterjet cutting machine with multiple cutting heads is assigned to the job.

Tools

This option can be used to enable the number of tools you wish to use on multi-tool machines.

If more than one tool is enabled, the software tries to nest the parts such that the parts can be cut simultaneously with the maximum number of available tools. Up to 4 tools can be enabled, depending on the machine and configuration.

Fixed tool separation enabled

When this option is enabled, nesting plans with multiple tools will be nested using a fixed tool separation.

Application:

Use this option for machines that do not control the tool separation automatically.

Fixed tool separation

This option is used to set the carriage distance between the tools.



Attention! The tool separation entered here must match the tool separation set on the machine! If these separations do not match, this may result in undesirable cutting results.

Machining sequence

Where do I find the function?

Select: **Modules > Part Nester > Nesting > Technology > Settings**

In this area you specify the options that shall be used by the software when computing the machining sequence of the parts in the cutting plan. The machining sequence impacts on the machining time and can affect the cut quality.

Active

Clicking on this option determines whether the fully automated process for setting the cutting technology includes/excludes computing the machining sequence.

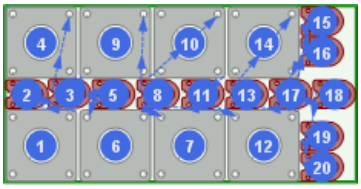
Mode

- **Constructing sequence**

  The machining sequence is set by the order in which the parts are added to the cutting plan.

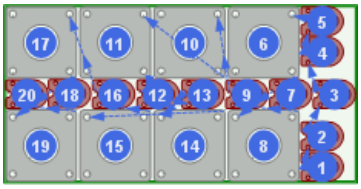
- **Vertical - From left to right**

  The parts are processed vertically (up and down) starting from the lower left and progressing to the top right.



- **Vertical - From right to left**

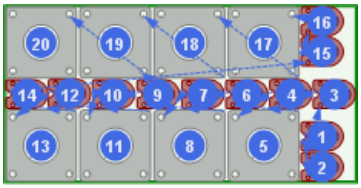
  The parts are processed vertically (up and down) starting from the lower right and progressing to the top left.



- **Horizontal - From bottom to top**

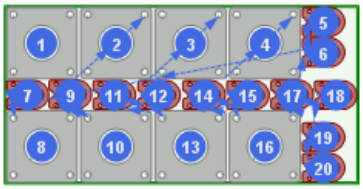
  The parts are processed horizontally (from right to left) starting from the bottom right and progressing to the top left.

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Horizontal - From top to bottom

The parts are processed horizontally (from left to right) starting from the top left and progressing to the bottom right.



Random order

The machining sequence is defined by a random sequence generator. Select **Random order** to avoid too much heat entering the material when laser-cutting small parts or heavy-gage sheets.

Note: This attribute usually results in the longest machining time.

Avoid cut out parts

An attempt is made to set the machining sequence so that the cutting head no longer moves over parts that have already been cut. This reduces the possibility of the cutting head colliding with parts that have already been cut.

Tip: To prevent parts from standing up after cutting, it is recommended to use the **Tilt optimization** function in the **Cutting Technology** menu. This function additionally moves the part lead-ins to an optimal position, taking the cutting grate into account. This also increases the probability of avoiding cut parts when calculating the machining sequence.

Zigzag

The machining sequence always starts the parts in each row (horizontal) or column (vertical) from the same side as the previous one was started.

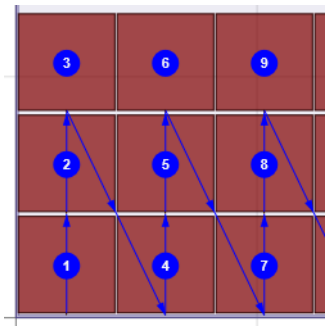


Fig. 10-23: Zigzag mode **Vertical - From left to right**

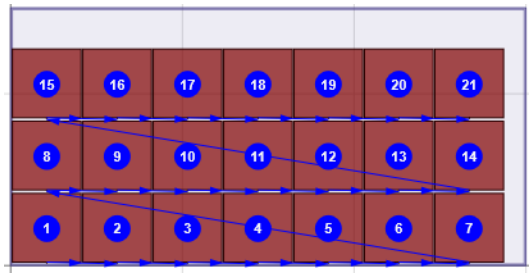


Fig. 10-24: Zigzag mode **Horizontal - From bottom to top**

☞ **Note:** The machining sequence can only be applied to the horizontal and vertical modes.

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Increment

This option can be used to set the increment between machining sequence numbers when the parts are actually cut.

- **Increment 1:** All parts are machined in accordance with the computed machining sequence (default).
- **Increment >1:** The actual machining sequence is applied to every nth part of the computed machining sequence.

Using the increment option for small parts

When the increment is set to 2, only every second part in the cutting sequence is cut. The remaining parts are cut in the second pass in the given sequence.



Fig. 10-25: Machining sequence for small parts with an increment of 2

Tip: Use an increment >1 to distribute the heat generated more evenly when cutting numerous small parts.

Number of parts in bottom corner

The specified number of parts in the lower left corner of the cutting plan are cut first in the machining sequence.

Group by sectors

Use this option to divide the total area of the sheet into sectors. Inside each sector, the parts are processed as a group using the machining sequence set under **Mode**.

Grid X/Y

This option divides the cutting plan into the specified grid and sets the machining sequence to all parts inside this grid.

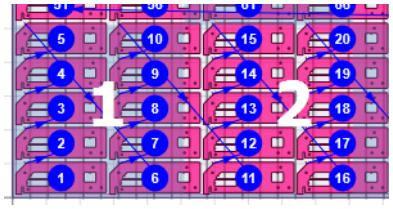


Fig. 10-26: Machining sequence in grid 1 and 2

Note: The sectors are displayed in a preview as blue and white areas defined by the dimensions entered in the **Grid X/Y** fields.

This option can be used for the following applications:

- When there are numerous small parts, in order to distribute the heat input to the material more evenly. In this case, the **Random order** mode is recommended in order to distribute the sectors randomly over the part.

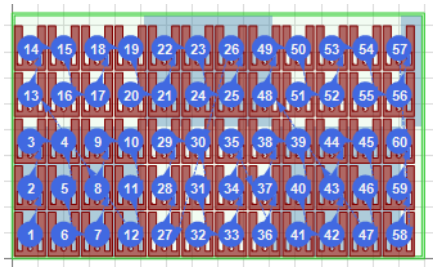


Fig. 10-27: Machining a cutting plan by sectors

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Group by size

This option is used to define the machining sequence of parts based on their size.

- **Small parts first**

  The part machining sequence starts with the smallest and ends with the largest parts.

- **Large parts first**

  The part machining sequence starts with the largest and ends with the smallest parts.

Skeleton / residual sheet

Where do I find the function?

Select: **Modules > Part Nester > Nesting > Technology Settings**

In this area you specify the options that shall be used by the software when computing the skeleton cuts and residual sheets.

Clicking on the **Active** option determines whether the fully automated process for setting the cutting technology includes/excludes skeleton and residual cuts.

Skeleton cuts

These options can be used to divide up cut sheets into individual segments. Dividing up the skeleton sheet makes it easier to remove the leftover material.

Attention! Do not use this function when making common cuts. Cutting up skeleton sheets can increase the cutting time significantly.

X/Y distance

This option specifies the spacing of skeleton cuts in the X and Y direction. A value of **0** means that no skeleton cuts are made in the direction concerned. The skeleton cuts are made from left to right and from bottom to top.

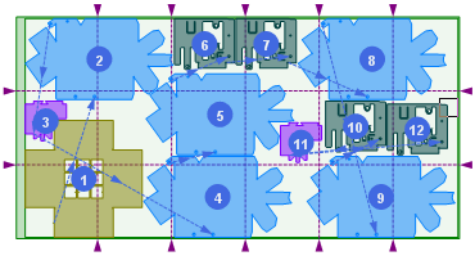
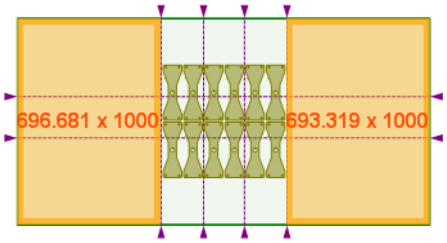


Fig. 10-28: Cutting plan containing horizontal and vertical skeleton cuts

Skeleton cuts are only set on nested areas of a sheet. Skeleton cuts are not used on residual sheets.



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Fig. 10-29: Skeleton cuts on a partially nested sheet

Over entire sheet

This option creates skeleton cuts over the entire area of a sheet. Skeleton cuts are also used on residual sheets (**A**).

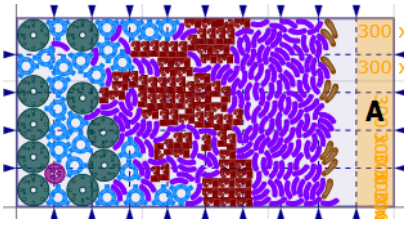


Fig. 10-30: Skeleton cuts over entire sheet

Optimize

When this option is enabled, the software tries to distribute the skeleton cuts evenly over the cutting area. If a skeleton cut lies very close to the end of the cutting area or the end of a residual sheet, the skeleton cut is moved if possible or removed.

If a skeleton cut runs along a contour of a part, the software tries to move the skeleton cut away from the contour by a certain distance. This avoids cutting the contour twice.

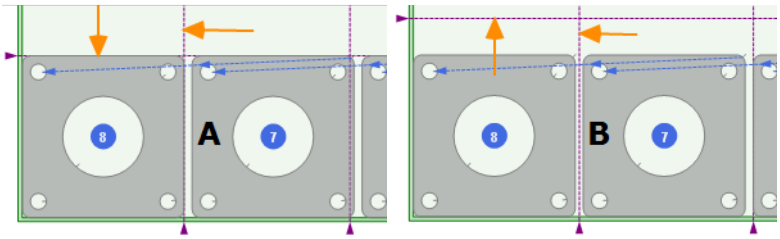


Fig. 10-31: Unoptimized (**A**) and optimized (**B**) skeleton cuts

☞ **Note:** When specifying the distances between cuts, take into account the flow of heat in the material during cutting and the size of waste container for removing the scrap material.

Detour

This option attempts to route the skeleton cuts of cutting plans around the adjacent parts to form a single cut. For the cutting plans of ByVention machines, this option is used for the sector separating cuts when there are overlapping sectors.

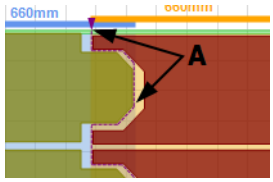


Fig. 10-32: Detour separating cut for overlapping sectors

Sector width

Use this option to set the width of the sector in which the software attempts to set a skeleton cut in the horizontal or vertical direction.



Fig. 10-33: Sector with detour skeleton cut

Residual sheet

This area is used to make the settings for the generation of residual-sheet cuts.

Residual-sheet cuts are used to separate an area of the sheet which you wish to use again later and to which you do not want to assign parts.

Min. length (X)

Use this option to specify the minimum size in the X direction for which a residual sheet will still be produced. A residual sheet will be created if an area that is greater than or equal to the size set here does not contain any more nested parts.

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Min. width (Y)

Use this option to specify the minimum size in the Y direction for which a residual sheet will still be produced. A residual sheet will be created if an area that is greater than or equal to the size set here does not contain any more nested parts.

Divide into segments

Using this option, large horizontal and vertical residual sheets can be divided into several smaller sheets.

- **Segment length (X)**

  Use this option to specify the length in the X direction after which a residual sheet cut is to be made. As many residual sheet cuts will be made as the overall length of the residual sheet allows for.

- **Segment width (Y)**

  Use this option to specify the width in the Y direction after which a residual sheet cut is to be made. As many residual sheet cuts will be made as the overall width of the residual sheet allows for.

- **Residual sheet cut without segment division**

  If the option **Divide into segments** is not enabled, a residual-sheet cut will only be made in the X or Y-direction based on the specifications in the **Min. width (X/Y)** field.

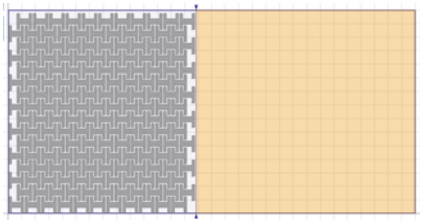


Fig. 10-34: Cutting plan with a vertical residual sheet cut

- **Residual sheet cut with segment division in the X-direction**

  If the option **Divide into segments** in the X or Y-direction is enabled, as many residual-sheet cuts will be made as the dimension in the field **Segment width (X)** or **Segment height (Y)** allows for.

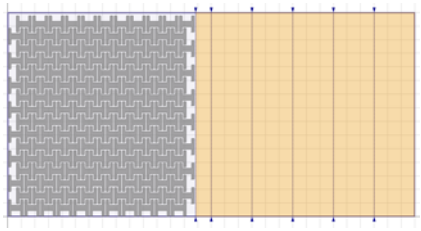


Fig. 10-35: Cutting plan with several vertical residual sheet cuts

Shorten cuts

Use this option to specify for overlapping residual sheet cuts whether the horizontal or the vertical cuts should be shortened.

- **Residual sheet cut shortened horizontally**

  With the **Horizontal** attribute, the horizontal residual sheet cut is only made up to the intersection point with the vertical residual sheet cut.

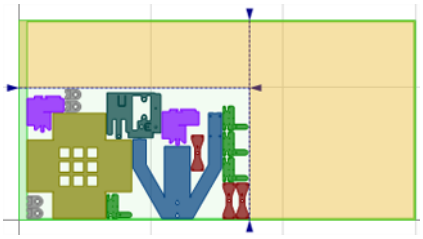


Fig. 10-36: Residual sheet cut shortened horizontally

- **Residual-sheet cuts shortened both horizontally and vertically**

  With the **Horizontal and vertical** attribute, horizontal and vertical residual sheet cuts are only made up to the point where they intersect.

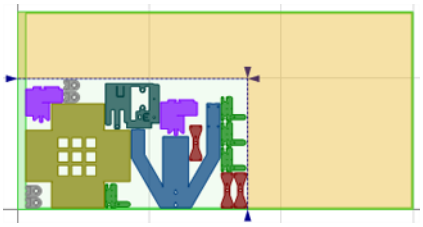


Fig. 10-37: Residual-sheet cuts shortened both horizontally and vertically

Take the following points into account when setting the minimum dimensions:

- Possibility of using the residual sheet later
- Use of filler parts
- Material costs
- Material management (standard sizes, type of storage, storage costs, type of loading system)

☞ **Note:** Nesting is performed from left to right. Consequently, the maximum Y-dimensions usually equal the sheet width and are often larger than the X-dimensions.

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Cuts along contours

This option generates the residual sheet cuts along the bounding rectangles of the parts. This improves usage of the residual sheet when parts of different widths are nested.

☞ **Note:** Gaps between parts narrower than the setting **Min. width Y** are ignored.

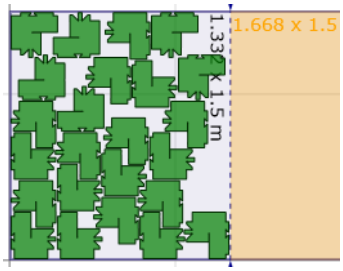


Fig. 10-38: Residual sheet with function disabled

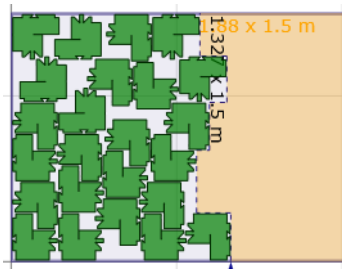


Fig. 10-39: Residual sheet with function enabled

Inscription

This section is used to engrave details about the residual sheet as text on the residual sheet.



Note: If it is not possible to place the text details on any edge owing to size, no text is engraved on the residual sheet.

Font

This option is used to define the font for the inscription text. You can use all fonts installed in the operating system.

Alignment

This option defines the alignment of text relative to the edge of the sheet.

Font size

This option is used to define the font size.

Separator

This option defines a separator to separate the **Dimension** and **Material** texts.

Edge distance

This option defines the distance of the text to the edge of the sheet. Minimum edge distance is 1 mm.

Position

This option is used to define text alignment relative to the edge of the sheet. If the texts do not fit along the specified edge, the text is automatically placed along the longest edge of the residual sheet.

The **Longest edge** option places the text at the top or on the right.

Orthogonal

This option sets the text at right angles to the selected sheet edge.

Contents

This option defines the details to be engraved on the residual sheet.

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Residual sheet sorting

In this section you can define the dimension of a skeleton sheet or residual sheet as of which to set the sorting attribute on it.

Min. length (X)

Defines the minimum length of a residual sheet in the X direction above which the attribute for sorting is set on a skeleton sheet or residual sheet.

Min. length (Y)

Defines the minimum length of a residual sheet in the Y direction above which the attribute for sorting is set on a skeleton sheet or residual sheet.

Cutouts

Where do I find the function?

Select: **Modules > Part Nester > Nesting > Technology Settings**

These settings are used to automatically place cutouts at all inside corners of the outer contour according to the options configured.

Cutout

Max width

This option defines the maximum width starting from which a cutout will be set (**A**).

Min depth

This option defines the minimum depth starting from which a cutout will be set (**B**).

Distance

This option defines the distance between the cutout and the contour (**C**). By default, the distance is equal to the kerf width.

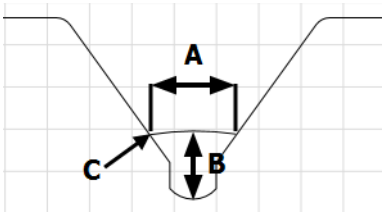


Fig. 10-40: Cutout on an outer contour

Geometry

Arc

This option performs the cutout as an arc.

Line

This option performs the cutout as a line.

Adjust to contour vertices

An attempt is made to match the end of the cutout (**A**) to the vertices of the contour (**B**). All previous parameters are taken into account.

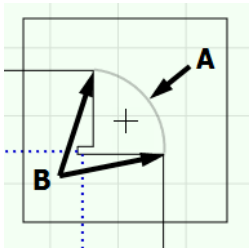


Fig. 10-41: Cut-out between two contour vertices

Tilt optimization

Where do I find the function?

Select: **Modules > Part Nester > Nesting > Technology Settings**

Determines the tilt optimization of parts taking the position of the sheet on the cutting grate into account.

General

Mode

- **Manual loading**
  The sheet is positioned manually into the backgauges on the shuttle table.

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> ☞ **Note:** These options for the additional sheet edge are hidden as they are not needed for manual loading to backgauge position.

- **Automation**

  An automation system places the sheet automatically on the shuttle table without backgauges.

  In order to ensure that the sheet is cut with tilt optimization as calculated in BySoft CAM, it is necessary to compensate for the slightly different placement positions of the sheet caused by the automation system.

  Compensation is performed by adding an additional sheet edge.

Sheet edge allowance

The added sheet edge is calculated by means of statistical normal distribution. The statistical data come from the machine. The position of every sheet is recorded on the machine. These data are taken to calculate means and standard deviations.

> ☞ **Note:** This area is only used for **Automation** mode.

Mode

- **Manual**

  The additional edges are calculated by manually entering the standard deviation.

- **Automatic**

  The additional edges are calculated based on the calculated standard deviation from the machine's XML file.

Expected success rate

Defines the expected rate of success in percent. The higher the value, the larger the shifts between mean and effective sheet position on the shuttle table will be accepted.

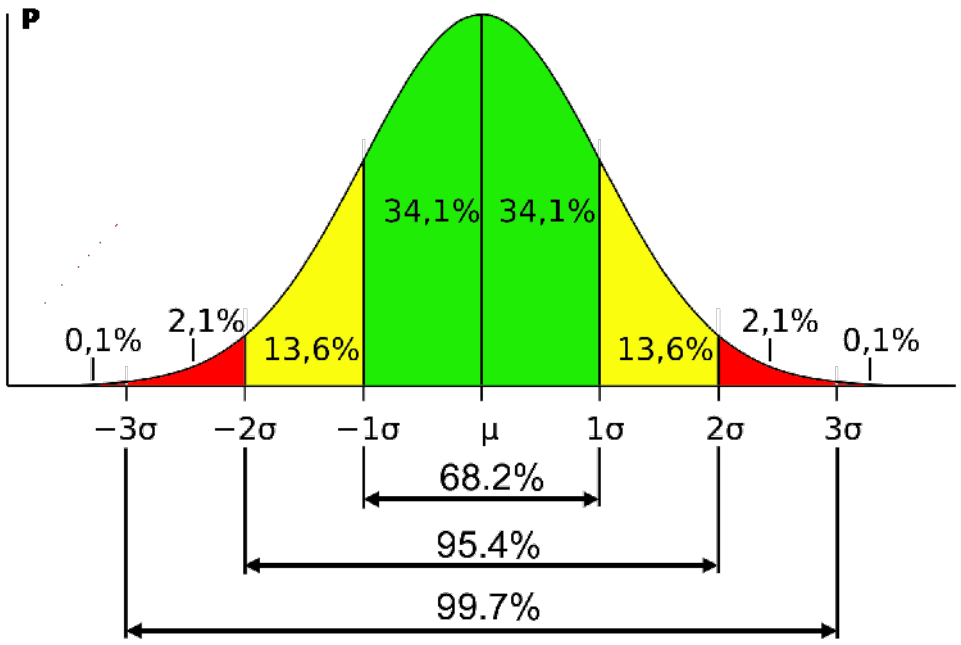


Fig. 10-42: Statistical normal distribution

Example:

- Success rate = 99.7%
- Standard deviation σ_X = **10 mm**
- Standard deviation σ_Y = **5 mm**
- Alpha = 0

☞ **Note:** Safety factor to achieve expected success rate of 99.7% = **3**

Additional edges left/right = factor • σ_X • 2* = 3 • 10 mm • 2 ≈ **60 mm**

Additional edges top/bottom = factor • σ_Y • 2* = 3 • 5 mm • 2 ≈ **30 mm**

* As the sheet can be moved in the positive and negative direction, the added sheet edge must be doubled.

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Standard deviation

Standard deviation determines the possible shift of sheet position on the shuttle table in the in X and Y direction and a possible skew.

- **X/Y**

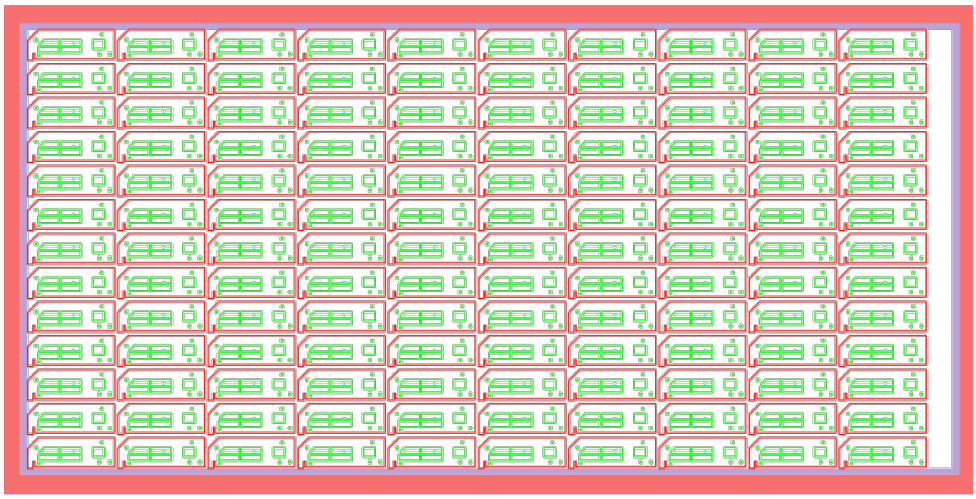
  Defines the maximum sheet offset in the X and Y direction.

- **Angle**

  Defines the maximum angle by which the sheet may be rotated.

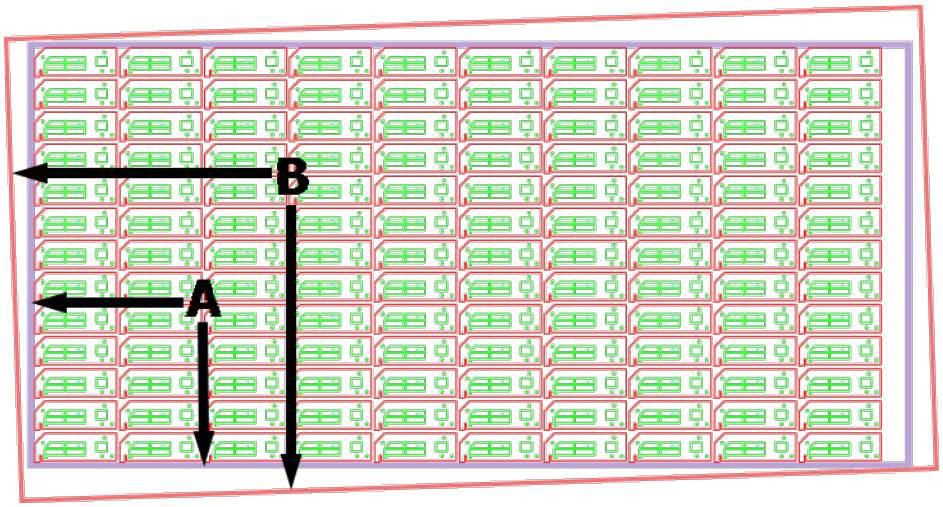
Sheet edge calculated with standard deviation

The red sheet edge is determined by standard deviation. It is added to the standard sheet edge.



Sheet shift on the shuttle table

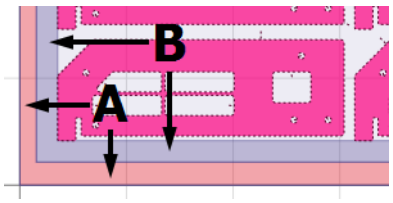
The area in which the parts are nested (**A**) including the standard sheet edge must be inside the sheet area (**B**).



Sheet edge preview

Based on the standard deviations entered and the sheet format selected, the additional sheet edge is indicated for all sides. This sheet edge is added to the standard sheet edge at the outer edge and shown in color in the cutting plan.

☞ **Note:** As soon as a sheet skew is indicated (angle), the added sheet edge is dependent on the sheet format selected.



Key:

A	Additional sheet edge
B	Standard sheet edge

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Colors for part edge zones

If you want the part color and a part edge zone to have the same color, you can edit the color for the appropriate zone.

Parameters

Use the options in this section to customize calculation of the part mechanics.

Safety

Defines the correction coefficient applied to the estimated tilt force.

- Positive values improve tilt safety by increasing the safety factor by up to 25% when calculating zones. Fewer green and more yellow and red zones are shown.
- Negative values reduce tilt safety by reducing the safety factor by up to 50% when calculating zones. More green and fewer yellow and red zones are shown.

Delay after cutting

In certain machine controller versions there is a delay of **40** ms after cutting a part. During this time, the cutting gas is blown onto the end cut of the contour. This time is included for tilt safety.

In machine controllers with a delay of **0**, it may make sense to set this value to **0** as well. This value can reduce the number of microjoints on the contour.

☞ **Note:** It does not make sense to increase the value as the machine only uses delays of **0** or **40** ms.

Apply positioning error simulation

Enable this option to make part tilt behavior more tolerant with regard to shifts of the support points of the cutting grate tips (**A**).

Note: In practice, the tips of the cutting grate may be slightly bent or slightly offset as a result of wear. This option takes this into account.

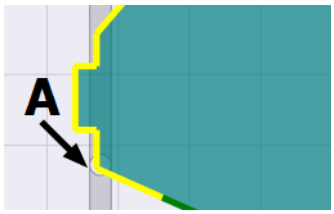


Fig. 10-43: Tilt zones without option enabled

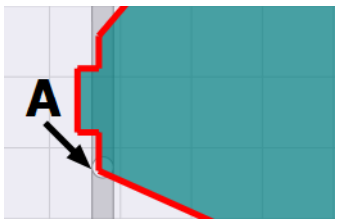


Fig. 10-44: Tilt zones with option enabled

Variance

This option determines the dimensional range used to simulate positioning error.

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Optimization

These settings apply globally to all nested cutting plans.

Automatic

Optimization is performed automatically with the functions **Automatic cutting technology** in the **Nesting** menu and **Tilt optimization** -> **Automatic** in the **Cutting technology** menu.

Keep inside contour machining sequence

The machining sequence of inside contours is not changed.

Keep outside contour machining sequence

The machining sequence of outside contours is not changed.

Keep lead-ins

Lead-ins are not changed.

Number of parts in bottom corner

The specified number of parts in the lower left corner of the cutting plan are cut first in the machining sequence.

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Microjoint settings

The options in this section determine how many microjoints are set on the contours.

Note: Microjoints are only set on those part instances that the tilt optimization calculation classifies as unstable.

Existing microjoints

This option determines what to do with microjoints already set on the contour.

- **Keep all**
  Tilt optimization does not remove any existing microjoints before calculating.
- **Reset**
  Tilt optimization resets the part to its original. If the original part has a microjoint, the system keeps the microjoint on the reference of the nested part.
- **Remove all**
  Tilt optimization resets the part to its original. If the original part has a microjoint, the system removes the microjoint on the reference of the nested part.

Distributed

Sets the number of microjoints defined in **Number** evenly distributed on the selected contours.

Quantity

Use this option to define the number of microjoints to place automatically. The positions of the microjoints are displayed in a preview as gray crosses on the contour. The number of microjoints is distributed evenly along the contour.

Note: This option is only shown if the **Distributed** option is selected.

End point

Sets a microjoint at the end point of all selected contours.

Sector nesting

Where do I find the function?

Select: **Modules > Part Nester > Nesting > Technology > Settings**

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You can set the options for nesting the parts in sectors of ByVention cutting plans in this area.

Note: This area is only displayed for ByVention jobs.

Max. sector width

The sectors will not be created any wider than the value specified here. The maximum sector width which can be specified is determined by the working area of the machine. The maximum sector width of the machine is specified in the **Cutting machine -> Properties** form for the administrator.

Sector clearance

This option specifies the minimum distance of the parts to the separating cut.

Optimize sector width

When this option is enabled, the sector width in automatic nesting is adapted to fit the nested parts. This achieves better utilization of the sheet.

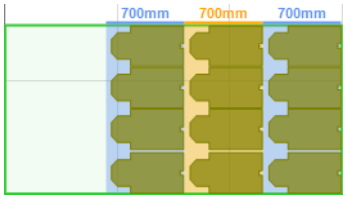


Fig. 10-45: Sector width without optimization

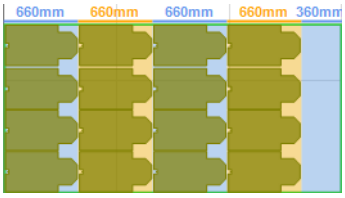


Fig. 10-46: Sector width with optimization

Overlapping sectors

Use this option to specify the nesting strategy for the parts. Regardless of the strategy, the maximum part size must lie within the cutting area. If no part fits into such a sector during automatic nesting, a sector will be added without overlapping.

- **Without overlapping (option not enabled)**
  The sectors are arranged without overlapping.

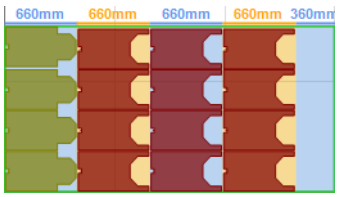


Fig. 10-47: Sectors without overlapping

- **With overlapping (option enabled)**
  The sectors are arranged with overlapping (**A**).

> **Note:** When the option is enabled, the process reliability could be reduced due to separating cuts that are not continuous. For this reason, enable the **Detour** separating cut option in the **Cutting technology settings**.

> **Tip:** Use this option if the parts cannot be nested optimally within one sector. Under certain circumstances, this can make it possible to nest more parts on one sheet.

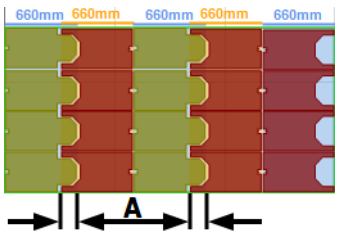


Fig. 10-48: Sectors with overlapping

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10.3.3 Group: Nesting

Where do I find the group?

Select: **Modules > Part Nester > Nesting**

This group contains all the functions that can be used to control the nesting of parts.



Note: All created cutting plans for the job are displayed in the **Cutting plans** tab of the **Job data** docking dialog box. If this dialog box is not displayed, it can be shown using the **Job data** function in the **Window** menu.

10.3.3.1 Automatic



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

These functions start the automated cutting-plan nesting process depending on the selected version.

Selecting the  button will nest the entire job.



Attention! All existing plans are deleted. Parts that are already nested in the cutting plan are re-nested.



Note: No cutting technologies are set on the parts, for example machining sequence or microjoints.

Other nesting functions

Clicking on the **Automatic** button offers the following additional nesting functions:



Note: Parts that are already nested in the cutting plan are retained.

- **Automatic (leave existing clusters)**

  This function deletes all existing plans and creates new plans. Existing clusters are retained and new clusters are nested on existing cluster frames.

  First, all clusters are nested. Then all parts that are not nested in clusters.

- **Complete**

With this function, all parts which have not yet been used in the job are nested in existing or new cutting plans. This does not affect parts that have already been nested.

First, the program tries to nest the parts in free areas on existing plans. If it is not possible to nest all parts in existing cutting plans, new cutting plans are generated.

- **Complete (leave existing plans)**

  This function is used to nest all parts of the job that have not yet been nested. **No** parts are nested in free areas of existing plans.

- **Plan**

  This function is used to nest available parts in the current cutting plan.

- **Part**

  This function is used to nest the parts selected on the **Parts** tab in the current cutting plan or cluster.

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- **Module**

  This function groups all selected parts in a cutting plan into a module, and then nests this module as many times as possible on the cutting plan *(See: ›Nesting parts as a module‹ Page 1366)*.

  Use this version to duplicate a manually created nesting arrangement.

- **Settings**

  This function opens the **Nesting settings** dialog box in which the nesting options can be modified.

10.3.3.2 Settings



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

You can use this function in the relevant docking dialog box to set the options for automatic nesting.

General

The options in this area can be used to control the form and efficiency of nesting.

Automatic nesting time

- **enabled:** The computed nesting time will be used for nesting.
- **disabled:** The nesting time set in **Nesting time** will be used for nesting.

Nesting time

Used to set the time, in seconds, that the module is allowed to take when nesting cutting plans.

Computed nesting time

Shows the optimum nesting time computed by BySoft CAM. This nesting time will be affected by the number of contour elements per part and the number of parts that need to be nested.



Note: If the **Automatic nesting time** option is enabled, this is the value used for nesting time.

Nesting contours

Defines the resolution for arc contours in segments during nesting. This setting affects the nesting time and the nesting accuracy.

- **Coarse** An arc contour will be resolved into a few segments. The parts in the cutting plan will be nested inaccurately, but very quickly.
- **Average** An arc contour will be resolved into an average number of segments. The parts in the cutting plan will be nested accurately and sufficiently quickly. This is the default setting.
- **Fine** An arc contour will be resolved into a large number of segments. The parts in the cutting plan will be nested very accurately, but relatively slowly.

Parts protective distance

This option prevents collisions when cut parts protrude. During nesting, the specified parts protective distance (**A**) is placed around the outer contour of the part. Hence the minimum distance between two parts equals at least twice the parts protective distance (**B**). The Parts protective distance is not applied to the sheet edge. The parts are nested right up to the edge of the cutting plan.



Attention! If the **Include lead-ins and lead-outs** option is not enabled, the Parts protective distance must be at least as large as the longest lead-in or lead-out. Otherwise the lead-ins or lead-outs for a part may lie within the contour of an adjacent part.



Note: The lead-ins are only taken into account in nesting if the **Include lead-ins and lead-outs** option is enabled.

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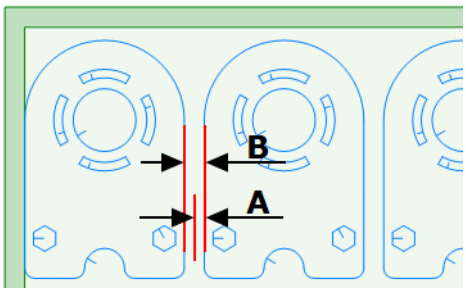


Fig. 10-49: Minimum distance between two parts: A = parts protective distance, B = double parts protective distance

Minimum parts protective distances

Material	Mild steel				Stainless steel	Aluminum
Assist gas	Oxygen				Nitrogen	
Thickness [mm]	1–5	6–10	12–15	16–25	1–25	1–15
Minimum parts protective distance	5	5	7.5	12.5	5	5

Include lead-ins and lead-outs

This option can be used to achieve an optimized nesting result and improved process reliability. If this function is enabled, the lead-ins and lead-outs are handled by the software like a contour. This means that a parts protective distance is also placed around the lead-ins and lead-outs at nesting.

Attention! This option is not taken into account for Common Cut nesting.

Note: When this option is enabled, the **Parts protective distance** can be set to the minimum spacing required. This achieves better utilization of the sheet by nesting the parts closer together.

To achieve optimum nesting with this function, the following additional option settings are required:

- Nesting type = Default
- Strip width = 1mm

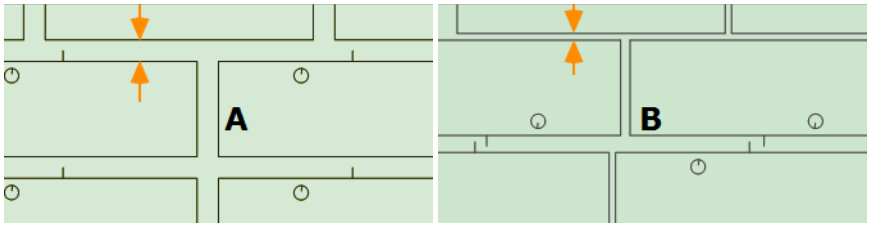


Fig. 10-50: Nesting with option disabled (**A**) and enabled (**B**)

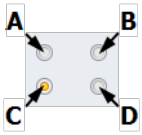
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Take cutouts into account

This option takes into account the cutouts defined on a part in Part Editor.

Nesting start

This option defines the starting point of nesting on a sheet. Depending on the selection of the **Nesting direction** option, nesting is performed horizontally or vertically starting from the nesting start.



Key:

A	Starting from top left
B	Starting from top right
C	Starting from bottom left
D	Starting from bottom right

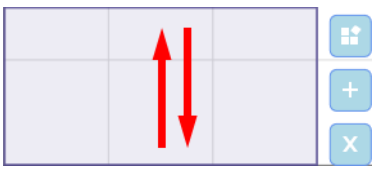
Nesting direction

This option defines the preferred direction for nesting parts on a sheet. You can use this function for partially nested sheets to set the position of the parts on the sheet.

Note: The nesting direction is additionally influenced by the **Nesting start** option.

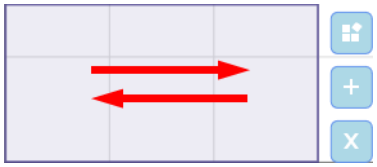
- **Vertical**

  Nests the parts vertically on the sheet (default).



- **Horizontal**

Nests the parts horizontally on the sheet.



Priority

This option specifies the priority that will be used to nest parts on a sheet.

- **Parts priority**

  If different priorities have been assigned to the parts, the parts are nested in the order of priority 1, 2, 3, n.

- **Small parts first**

  The part nesting sequence starts with the smallest and ends with the largest parts.

- **Large parts first**

  The part nesting sequence starts with the largest and ends with the smallest parts.



Tip: These attributes can be used in combination with the **Nesting direction** option to facilitate removal of parts on the machine.

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Filler parts strategy

This option specifies the way filler parts are used on a sheet.

- **Only nesting area**

  Filler parts are only nested in that area of a sheet used by normal parts. Any residual sheets available are no longer considered for nesting filler parts.

- **Only nesting area on last sheet**

  Filler parts are only nested in that area of the last sheet used by normal parts.

- **On entire sheet**

  If a sheet cannot be fully nested with standard parts, the unused area of the sheet is nested with filler parts. A residual sheet is not produced. Enable this function in order to nest filler parts in spare areas of a sheet.

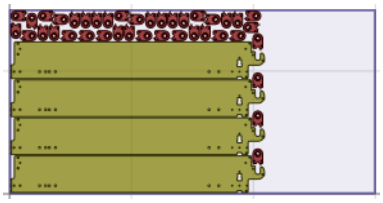


Fig. 10-51: Nested filler parts in nesting area of a sheet

Min. area for inner-contour nesting

This option defines the minimum surface area of the inner contour of a part into which a part can still be nested. The larger the value, the fewer the parts that can be nested in inner contours.

☞ **Note:** The parts protective distance is included in the calculated value of the surface area. **Calculation:** *Area of inner contour - Parts protective distance ≥ Area of outer contour + Parts protective distance*

In order for a part to be nested inside the inner contour of another part, the following conditions must be met:

- The area of the inner contour must be greater than/equal to the value specified under **Minimum area for inner-contour nesting**.
- The area of the part to be nested inside the inner contour of another part must be less than the value specified under **Minimum area for inner-contour nesting**.

More than one machining cycle allowed per plan

This option allows a cutting plan to be cut several times if a sufficient number of parts are present.

Final optimization

This option can be used to apply the selected attribute additionally to the cutting plan after nesting. The **Compress**, **Expand** and **Shake** attributes are equivalent to the functions with the same names in the **Nesting** group. The attribute **None** means that no optimization is performed after the nesting process.

Common Cut

Kerf

This value is used to define the kerf taken into account when combining common cuts.

Min. length

This option is used to set the minimum length of a cut that can still be made as a common cut.

Note: Minimum length must not be less than **10 mm**.

Material costs/cutting costs

Common Cut can be optimized using this option to reduce material consumption or cutting costs. The corresponding slider can be moved to the left or right in order to give greater priority to the corresponding option.

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Small parts mode

The options in this section can be used to nest small parts in specific peripheral zones on a sheet.

If there are a lot of small parts, it is possible that parts will only be nested in the peripheral zone on a new sheet. Accordingly, make sure to set a sufficiently large maximum edge distance so that there will be a balance between the parts nested in the peripheral zone and the parts nested in the remaining sheet.

Note: This option requires additional computing time on top of the time specified in **Computed nesting time**.

Tip: The small parts area on a sheet can be shown using the special view by using the **Small parts area** option (**A**).

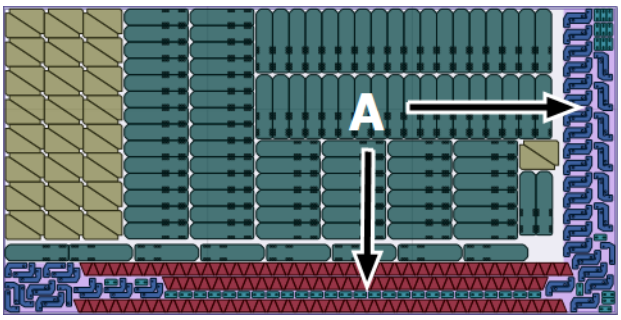


Fig. 10-52: Small parts nested in right and lower peripheral zones

Active

Enables small parts nesting in specific zones on the sheet.

Zones

Used to define the zones on the sheet on which the parts can be nested. Zones can be combined with each other.

Max. edge distance

Used to define the width of the zone, along the perimeter of the sheet, in which small parts can be nested.

Small parts only in zone

All small parts will be nested exclusively in the specified zone.

If there is not enough space inside the zone, the parts will be nested on a new sheet in the same zone.

Max. small part area

Used to specify the maximum size of the parts that can be used as small parts.

Reduce number of plans

If this option is enabled, the program tries to reduce the number of different nestings of cutting plans to cutting plans with multiple passes.

Sort

If the **Sort** option is enabled, the program attempts to position parts of the same type parallel to each other in the Y direction. Parts protective distance for default nesting is ignored.

Settings taken into account when nesting sorting plans:

- Parts protective distance
- Nesting direction

Nesting settings ignored when nesting sorting plans:

- Final optimization
- Common Cut
- Filler parts
- Small parts mode
- Cluster

Active

Enables nesting of sorting plans.

Parts protective distance

Specifies the spacing between parts for nesting sorting plans.



Note: The parts protective distance for sorting plans should be **4-5** times the parts protective distance for nesting normal cutting plans.

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Group nesting

Nests parts in groups on sheets to facilitate sorting of orders.

All parts with the same attributes in fields 1 thru 3 can be nested as groups together in one or more cutting plans.

Application:

- Enables job-based sorting of parts on sheets.
- Create automatically grouped jobs in Plant Manager based on the setting in **Settings Manager**.

Max. commenced groups per sheet

This option defines the maximum number of commenced sheets that may be on a sheet.

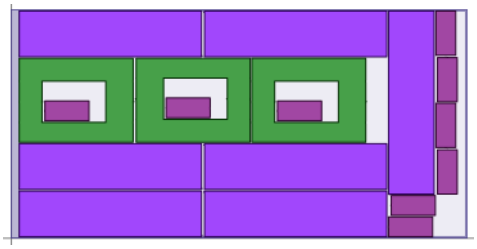


Fig. 10-53: Cutting plan with three groups per sheet.

Attention! The smaller the number of groups per sheet, the greater sheet waste will be.

Field 1–3

These fields define the criteria for forming subgroups.

In a typical application, the attribute **Order info** is used in field **1**. This attribute nests all parts with the same **Order info** as a group.

Another application is grouping by start and end date. The **Start date** attribute in Field **1** is used to nest all parts with the earliest date first and the latest date last on the sheets.

If an attribute is also entered in field **2** and **3**, all parts with the same attributes are nested as a group.

Parts nested on multiple sheets without group nesting

The parts are partially nested across all sheets.

Parts	Sheet 1	Sheet 2	Sheet 3	Sheet 4	Sheet 5	Sheet 6
Part 1						
Part 2						
Part 3						
Part 4						
Part 5						
Part 6						
Part 7						
Part 8						
Part 9						
Part 10						
Part 11						
Part 12						

Parts nested on multiple sheets without group nesting

The parts are nested in groups on a maximum of four sheets.

Parts	Sheet 1	Sheet 2	Sheet 3	Sheet 4	Sheet 5	Sheet 6
Part 1						
Part 2						
Part 3						
Part 4						
Part 5						
Part 6						
Part 7						
Part 8						
Part 9						
Part 10						
Part 11						
Part 12						

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Multihead



Note: These functions are only available if a waterjet cutting machine with multiple cutting heads is assigned to the job.

Tools

This option can be used to enable the number of tools you wish to use on multi-tool machines.

If more than one tool is enabled, the software tries to nest the parts such that the parts can be cut simultaneously with the maximum number of available tools. Up to 4 tools can be enabled, depending on the machine and configuration.

Fixed tool separation enabled

When this option is enabled, nesting plans with multiple tools will be nested using a fixed tool separation.

Application:

Use this option for machines that do not control the tool separation automatically.

Fixed tool separation

This option is used to set the carriage distance between the tools.



Attention! The tool separation entered here must match the tool separation set on the machine! If these separations do not match, this may result in undesirable cutting results.

Sector nesting

You can set the options for nesting the parts in sectors of ByVention cutting plans in this area.

Note: This area is only displayed for ByVention jobs.

Max. sector width

The sectors will not be created any wider than the value specified here. The maximum sector width which can be specified is determined by the working area of the machine. The maximum sector width of the machine is specified in the **Cutting machine -> Properties** form for the administrator.

Sector clearance

This option specifies the minimum distance of the parts to the separating cut.

Optimize sector width

When this option is enabled, the sector width in automatic nesting is adapted to fit the nested parts. This achieves better utilization of the sheet.

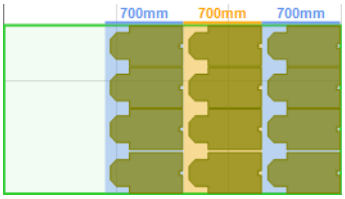


Fig. 10-54: Sector width without optimization

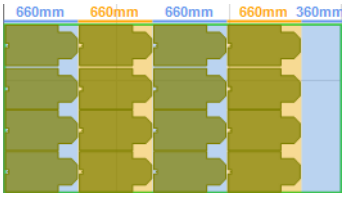


Fig. 10-55: Sector width with optimization

Overlapping sectors

Use this option to specify the nesting strategy for the parts. Regardless of the strategy, the maximum part size must lie within the cutting area. If no part fits into such a sector during automatic nesting, a sector will be added without overlapping.

- **Without overlapping (option not enabled)**
  The sectors are arranged without overlapping.

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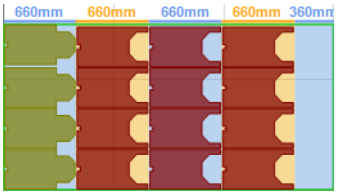


Fig. 10-56: Sectors without overlapping

- **With overlapping (option enabled)**
  The sectors are arranged with overlapping (**A**).

> **Note:** When the option is enabled, the process reliability could be reduced due to separating cuts that are not continuous. For this reason, enable the **Detour** separating cut option in the **Cutting technology settings**.

> **Tip:** Use this option if the parts cannot be nested optimally within one sector. Under certain circumstances, this can make it possible to nest more parts on one sheet.

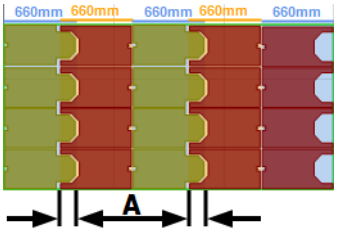


Fig. 10-57: Sectors with overlapping

Fill with empty sectors

This option fills blank areas of a sheet with sectors.

Application:

- To cut blank areas of a sheet into individual parts.
- To create sectors for manual nesting.

10.3.3.3 Compress



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function is used to move parts closer together in a cutting plan where possible. This compaction of parts can be performed automatically in any direction using the direction arrows in the displayed dialog box.

Applying this function to cutting plans does not always produce an optimum result and should be trialed on a case-by-case basis.



Tip: Use this function for fine-optimization of nested cutting plans.

10.3.3.4 Shake



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function re-nests the parts with the aim of packing all the parts together in a certain direction. You can define the direction for packing the parts using the **Nesting direction** option in the nesting settings.



Note: Running this function does not necessarily produce a better nesting result and should be trialed on a case-by-case basis. Nesting may possibly be improved by running the function several times or by changing the nesting direction.

Setting the nesting direction for packing the parts:

Packing direction position	Nesting direction to be used
Right	from right to left
Left	from left to right
Top	from top to bottom
Bottom	from bottom to top

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10.3.3.5 Expand



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function re-nests the parts with the aim of distributing all the parts evenly over the entire sheet area. You can define the direction for expanding the parts using the **Nesting direction** option in the nesting settings.

Expanding parts on a cutting plan

Initial situation:

The cutting plan has an empty area (**A**) on the right that you wish to use for nesting the parts.

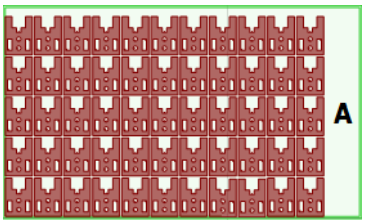


Fig. 10-58: Cutting plan prior to expanding

1. Select the cutting plan.
2. In the nesting settings, select the **Nesting direction** option.
3. Select the **from right to left** nesting direction.
4. Select the **Expand** option. The nested parts are nested over the entire sheet area.

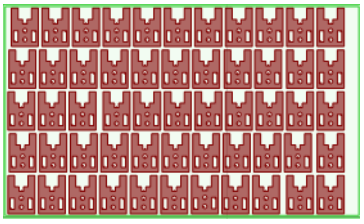


Fig. 10-59: Cutting plan after expanding

☞ **Note:** This function is only applied to the parts present in the cutting plan.



Tip: Use this function to utilize empty areas of a sheet that can no longer be used as a residual sheet. This will increase the spacing between nested parts, ensuring better heat distribution during cutting and hence greater process reliability.

Setting the nesting direction for expanding the parts:

Position of empty sheet area	Nesting direction to be used
Right	from right to left
Left	from left to right
Top	from top to bottom
Bottom	from bottom to top

10.3.3.6 Place part



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function places the selected part in the cutting plan. The part can be placed by clicking with the mouse, entering the X and Y-coordinates or using the arrow key combined with the Ctrl key on the keyboard.

Keys for rotating parts

- ←: Rotates the part counterclockwise by the angle specified for the **Rotation allowance**.
- →: Rotates the part clockwise by the angle specified for the **Rotation allowance**.
- ↑: Rotates the part through 90° counterclockwise.
- ↓: Rotates the part through 90° clockwise.

Nesting parts using the mouse

1. In the **Parts** tab, select the part that you wish to nest in the cutting plan.
2. Select the **Place part** function. The part is positioned in the cutting plan with the part origin at the mouse cursor.
3. Use the arrow keys to rotate the part into the required position.
4. Left-click in the cutting plan at the position where you wish to nest the part.
5. Repeat steps 3 and 4 to nest further parts with the mouse.

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Nesting parts by entering coordinates

1. In the **Parts** tab, select the part that you wish to nest in the cutting plan.
2. Select the **Place part** function. The part is positioned in the cutting plan with the part origin at the mouse cursor.
3. Use the arrow keys to rotate the part into the required position.
4. In the dialog box that opens, use the keyboard to enter the X and Y-coordinates for the nesting position for the part.
5. Confirm the entries with **Enter**. The part is placed with its origin at the entered coordinates.
6. Repeat steps 3 to 5 to nest further parts by entering coordinates.

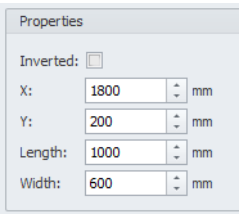


Fig. 10-60: Dialog box for entering coordinates

Inserting parts using the abut facility

In the first step, press the **CTRL** + X-direction arrow keys to place the part on the sheet according to the coordinate system. In the second step, press the **CTRL** + Y-direction arrow keys to nest the part in its final position.

This abut function also works when you place the part first in the Y-direction and then perform final nesting in the X-direction.

Note:

- The parts protective distance is taken into account when placing a part using the abut function. If the **Common Cut** option is selected for a part in the **Parts** tab, then the **Parts protective distance** option is still taken into account when placing using the abut function.
- As with sheet edges, when machining residual sheet the parts are placed at the edge of the residual sheet cut.

Key combinations for placing parts using the abut function

- **Ctrl + ←**

  Moves the part in the X-direction to the furthest possible nesting position on the **left**.

- **Ctrl + →**

  Moves the part in the X-direction to the furthest possible nesting position on the **right**.

- **Ctrl +-↑**

  Moves the part in the Y-direction to the **highest** possible nesting position.

- **Ctrl + ↓**

  Moves the part in the Y-direction to the **lowest** possible nesting position.

10.3.3.7 Move part



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function moves the selected part by a certain distance within the cutting plan. The part can also be rotated simultaneously through 90° by the arrow keys on the PC keyboard. The angle of rotation of the part is displayed in the **Rotation** area.

Base point

The base point of the selected element is displayed as absolute X and Y-coordinates in the **Base point** area.

You have the following options for selecting the base point:

- Clicking on any position.
- Left-clicking on a point in the alignment array.
- Selecting a Snap function.

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Reference point

Defines the reference point for moving the part based on the selected option.

- **Default**
  The reference point is the bottom left corner of the bounding rectangle.
- **Origin**
  The reference point is the origin of the part.
- **Center of gravity**
  The reference point is the center of gravity of the part.

Rotation

- **Angle of rotation**
  Displays the angle by which the part has been rotated on the cutting plan.

Options

- **Mirror**
  This option is used to mirror the part about the vertical axis when the part has been enabled for mirroring in the **Mirror** column in the **Parts** tab If the part is rotated by a certain angle, the angle of rotation is taken into consideration when mirroring.
- **Common Cut**
  This option nests the selected part as a Common Cut part.

> ☞ **Note:** Enable the command **Without collision by default** in the settings for Common Cut nesting *(See: ›Commands‹ Page 906)*. This allows you to easily nest Common Cut parts with each other.

Absolute coordinates / Relative movement

You have the following options for selecting the move point:

- Clicking on any position.
- Entering a new X and Y value in the **Absolute coordinates** or **Relative movement** areas.
- Selecting a sector in the **Absolute coordinates** area (Tube Editor).
- Selecting a tube angle for round tubes (Tube Editor)
- Selecting a Snap function.



Tip: Information about the next step is displayed in the status bar at the lower left.

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Moving a part within a cutting plan

You have the following options for moving a part within a cutting plan:

Moving a part using arrow buttons

Click on an arrow button to move the part to the furthest possible position in the chosen direction.

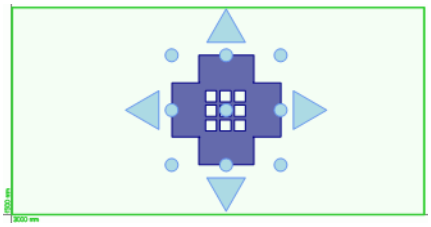


Fig. 10-61: Buttons for moving a part

Note: It is not possible to move a part over another part.

Moving a part by entering coordinates

1. Select the part that you wish to move.
2. Select the **Move part** function.
3. Select the base point of the part.
4. Select the move point of the part.
5. Click on the **Apply** button.

Attention! It is possible to move a part over another part if the command **Without collision by default** is disabled in the settings *(See: ›Commands‹ Page 906)*.

Tip: Pressing the **Ctrl** key while moving the part and clicking on an empty position in the cutting plan will nest the part automatically.

10.3.3.8 Rotate part



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

The part can be rotated through the specified angle in this dialog. To rotate the part, enter in the **Angle of rotation** field the angle through which you want to rotate the part.

- **Mode**

  Shows the increment angle through which the part can be rotated. The angle is configured in the part properties *(See: ›Increment‹ Page 930)*.

- **Angle of rotation**

  Input box for the angle of rotation. You can only enter the increment angle or a multiple thereof.

  Clicking on the arrow buttons in the input field will provide valid values. Click **Apply** to assign the angle of rotation to the part.



Tip: You can also use the mouse thumb wheel to rotate a part. To do this, click in the **Angle of rotation** field and scroll with the mouse wheel until you reach the angle you require.

- **Precise rotation**

  If this option is enabled, you can enter any angles in the **Angle of rotation** field. The angle of rotation can be specified to two decimal places.



Note: **Increment** is disabled.

10.3.3.9 Remove part



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function removes all the selected parts from the list and from all cutting plans.



Tip: If you accidentally delete any parts, use the **Undo** function to restore the deleted parts.

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10.3.3.10 Cut part



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function removes all the parts selected in the cutting plan from the cutting plan and copies them onto the clipboard. The **Insert part** function can be used to re-insert the copied parts into the current cutting plan or into another cutting plan of the same job.



Tip: Use this function to copy nested parts or groups of parts into a different cutting plan.

10.3.3.11 Copy part



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function copies all the parts selected in the cutting plan onto the clipboard. The **Insert part** function can be used to re-insert the copied parts into the current cutting plan or into another cutting plan of the same job.



Tip: Use this function for manual duplication of nested parts or groups of parts.

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10.3.3.12 Row



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

You can use this function to arrange in rows or columns a part or a selection of parts already contained in the plan. The buttons define the direction in which the parts are arranged. This function is particularly useful for manual nesting of rectangular or rotationally symmetric parts.



Attention! This function **does not** notice parts already nested on the sheet. Therefore make sure that there are no other parts in the nesting area.



Tip: To make optimum use of the function, place the selected parts in a corner of the sheet and then create the number of columns and rows from this position outwards. Use the position buttons to set the origin of the nesting.

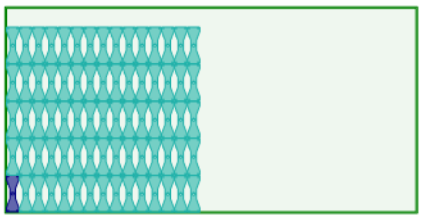


Fig. 10-62: Nesting a part in rows

If more parts are nested in the cutting plan than are available in the job, the missing number of parts are shown in red. If the missing parts are also meant to be nested, enable the option **Book surplus parts to Extra quantity**. If this option is enabled, the color of the parts is changed to green and the parts are used in the cutting plan.

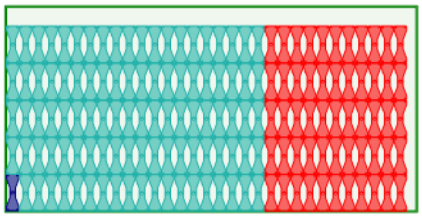


Fig. 10-63: Nesting in rows when the number of parts exceeds the number available in the job

If the cutting plan already contains nested parts, the overlapping parts of the row nesting are colored red. These parts are not included in the row nesting after confirming.

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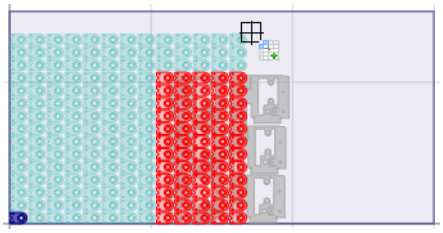


Fig. 10-64: Nesting in rows with overlapping parts before confirming

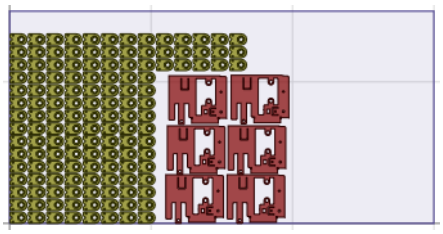


Fig. 10-65: Nesting in rows with overlapping parts after confirming

What each button does:

Button	Description
	The parts selection is arranged from lower left to top right.
	The parts selection is arranged from lower right to top left.
	The parts selection is arranged from top left to lower right.
	The parts selection is arranged from top right to lower left.

What each field does:

Row

- **Number of columns**

  Specifies the number of columns in which the parts selection is duplicated. You can add as many columns as there is space available for nesting the parts selection.

- **Number of rows**

  Specifies the number of rows in which the parts selection is duplicated. You can add as many rows as there is space available for nesting the parts selection.

- **Column separation**

  Specifies the gap between each of the columns.

- **Row separation**

  Specifies the gap between each of the rows.

- **Number of parts**

  Shows the current number of nested parts using the row function.

- **Book surplus parts to Extra quantity.**

  If more parts are nested in the cutting plan than are available in the job, the number of extra parts required are booked into the **Extra quantity** column in the **Parts** tab.

- **Circular parts mode**

  Use this option for circular parts to achieve better nesting of the parts.

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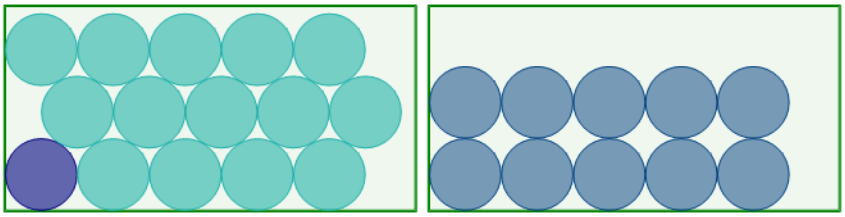


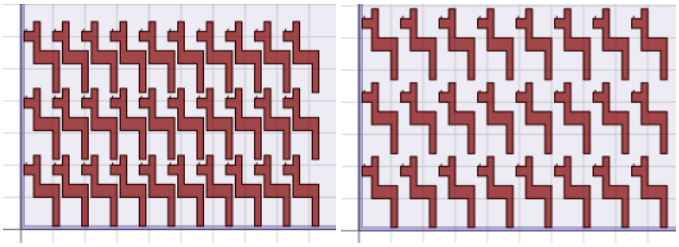
Fig. 10-66: Nesting of a part with/without circular parts mode

Nesting mode

This function includes the contour in row nesting operations. This nests the parts closer together to make better use of the sheet.

- **Enabled**

  Enables nesting mode for the row function.



Row nesting with and without nesting mode enabled.

- **Parts distance**

  Defines the minimum clearance between parts.

- **Nesting direction**

  Defines the direction in which to nest parts next to each other.

- **Nest parts as group**

  - If this option is enabled, all selected parts are nested as a group. This is the default setting.
  - If this function is disabled, if several parts are selected, each part is nested separately.

Attention! Disabling this function makes nesting much slower.

10.3.3.13 Insert part



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function pastes into the current cutting plan all the parts copied onto the clipboard using the **Copy part** function.

10.3.3.14 Edit tools



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function can be used in the **Cutting head settings** docking dialog box to set the number of tools used and the separation distance between the tools.



Note: This function is only available if a water jet cutting machine with multiple cutting heads is assigned to the job.

Creating a cutting plan for multihead machines

The general procedure for creating a multihead cutting plan is described below. The actual procedure may differ from this depending on what parts need to be cut.

1. Place in the lower left corner for the sheet the part that you wish to cut using several tools at once.
2. Select the part and then the **Row** function.
3. In the **Number of columns** field, enter the number of parts that can be nested horizontally. The parts are displayed in a preview.
4. Confirm your settings with **Apply**.
5. Select all the parts on the sheet.
6. Select the **Edit tools** function.
7. In the **Tools** area, select the number of tools that will be used simultaneously to cut the parts. The tool separation is set automatically.
8. Confirm your settings with **Apply**.

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Machining strategies for multihead cutting plans

Multihead cutting plans can employ various machining strategies. The machining strategy for the parts depends on the size of the parts used and the number of tools available.

Cutting plan containing one part

The cutting plan contains just one part, which is cut once per column.

This strategy is particularly suitable for large and medium-sized parts that allow full utilization of the sheet with the available tools.

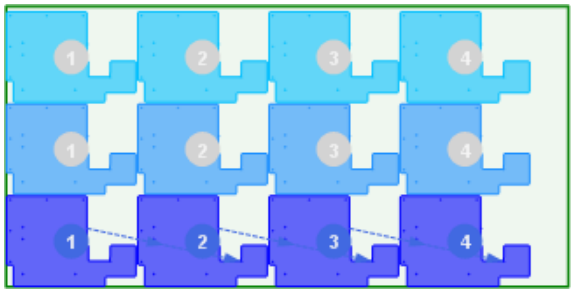


Fig. 10-67: Cutting plan with 3 tools

Cutting plan containing more than one part

The cutting plan contains different parts that are nested in a group in each column.

This strategy is particularly suitable for medium-sized and small parts that can be nested conveniently amongst each other.



Fig. 10-68: Cutting plan containing multiple parts with 4 tools

Cutting plan containing multiple cutting sequences

The cutting plan usually contains just one part, which has been nested in two rows per column. The part in the first row is cut using four tools. The part in the second row is cut using three tools.

This strategy is particularly useful for parts in which the maximum number of tools is not sufficient to utilize the sheet in full in the vertical direction in each column.

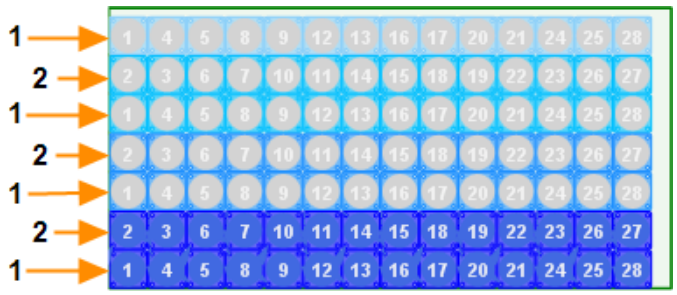


Fig. 10-69: Cutting plan containing two cutting sequences per column

> **Note:** On machines that have just two cutting heads or heads arranged in the Y-direction, a cutting plan can also contain more than two sequences.

Tools

This area displays all the available tools. As soon as a tool is enabled, the respective copy of the selected part or group of parts is displayed in the cutting plan with the appropriate tool separation.

Disable the cutting heads that you do not want to use for cutting the selected parts. The parts are hidden in the cutting plan.

Settings

In this area, the distance between the tools is set in the **Tool separation** field. This setting can therefore be used to increase or decrease the distance between the parts to be cut in parallel by the enabled cutting heads.

The separation distance is the same for all tools. The minimum and maximum tool separation depends on the assigned machine type.

> **Note:** The **Properties** area of the **Cutting machine** form in the **Administrator** module displays the minimum and maximum tool separations for each configured cutting machine.

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10.3.4 Group: Nesting window

Where do I find the group?

Select: **Modules > Part Nester > Nesting**

This group contains all the functions that can be used to create the nesting window or delete it once again.

10.3.4.1 Set window



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

You can use this function to define a sub-region within a cutting plan in which to nest parts. The window area is set by clicking and dragging a window with the mouse or by entering coordinates in the dialog.



Tip: The origin and the dimensions of the nesting window can be precisely determined by input in the dialog.

All automatic nesting modes apart from **Nest job** can be applied to a nesting window. It is also possible to nest parts manually in a nesting window.

Only one window can be set at a time. If a window has already been set, it is deleted. Nesting windows can also be placed over ready nested parts.



Note: Nesting windows are not saved. If the nesting window is no longer displayed when selecting another function, you need to enable display of the nesting window using **F8** and the **Special view**  .

Application:

- For avoiding defects on sheets (rust, hole, warping etc.)
- When using multiple cutting heads on water jet machines.

Setting a window by entering coordinates

1. Select the **Set window** function.
2. Use the keyboard to enter the starting point of the window with the X and Y coordinates.
3. Use the keyboard to enter the length and width of the window.

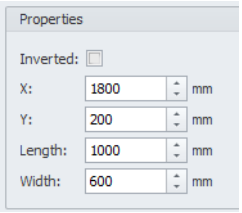


Fig. 10-70: Dialog box for entering coordinates

10.3.4.2 Remove window



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Nesting**

This function removes a set nesting window.

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10.3.5 Group: Sectors

Where do I find the group?

Select: **Modules > Part Nester > Nesting**

This group contains all the functions that can be used to control the nesting of parts in ByVention machines.

10.3.5.1 Compress



Where do I find the function?

Select: **Part Nester > Nesting > Sectors**

This function attempts to increase the size of the sheet's area in which no parts are nested. If there are empty sectors between nested parts, the parts to the left of the empty sectors are pushed to the right edge of the sheet. The edges of the sectors are reduced to a minimum. Empty sectors are deleted.

Under the following conditions, an empty sector cannot be deleted:

- The residual sheet is shorter than 1200mm.
- The **Fill with empty sectors** option is enabled.

☞ **Note:** This function is only displayed for the cutting plans of ByVention machines.

10.3.5.2 Fill



How do I open this dialog box?

Select: **Part Nester > Nesting > Sectors**

Use this function to custom fill a blank or partially nested sheet with sectors. When doing so, you can define the size and the overlap of the sectors.

☞ **Note:** This function is only displayed for the cutting plans of ByVention machines.

Max. sector width

The sectors will not be created any wider than the value specified here. The maximum sector width which can be specified is determined by the working area of the machine. The maximum sector width of the machine is specified in the **Cutting**

machine -> Properties form for the administrator.

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Overlapping sectors

Use this option to specify the nesting strategy for the parts. Regardless of the strategy, the maximum part size must lie within the cutting area. If no part fits into such a sector during automatic nesting, a sector will be added without overlapping.

- **Without overlapping (option not enabled)**
  The sectors are arranged without overlapping.

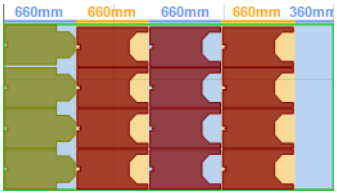


Fig. 10-71: Sectors without overlapping

- **With overlapping (option enabled)**
  The sectors are arranged with overlapping (**A**).

> **Note:** When the option is enabled, the process reliability could be reduced due to separating cuts that are not continuous. For this reason, enable the **Detour** separating cut option in the **Cutting technology settings**.

> **Tip:** Use this option if the parts cannot be nested optimally within one sector. Under certain circumstances, this can make it possible to nest more parts on one sheet.

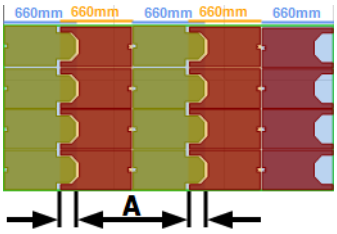


Fig. 10-72: Sectors with overlapping

10.3.6 Group: Cycle

Where do I find the group?

Select: **Modules > Part Nester > Nesting**

This group contains all the functions that can be used to influence the cycles of cutting plans.

10.3.6.1 Maximize number of cycles



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Cycle**

This function can be used to assign the maximum possible number of cycles to a manually nested cutting plan. The number of cycles depends on the number of parts, sheets and raw material available in the job. The number of assigned cycles for the current plan is displayed in the **Cycles** column of the **Cutting plans** tab.

10.3.6.2 Reset number of machining cycles



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Cycle**

This function resets the number of machining cycles for a cutting plan to 1. After resetting the pass number, the number of parts from the reset cutting plans are made available again for nesting.

10.3.6.3 Separate



Where do I find the function?

Select: **Modules > Part Nester > Nesting > Cycle**

This function can be used to divide a job containing several machining cycles into the equivalent number of separate jobs. For instance, a job containing three machining cycles will be split into three single jobs.

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10.4 Menu: Cutting technology

Where do I find the menu?

Select: **Modules > Part Nester**

The cutting technology for the active job is configured in this menu.

The cutting technology is composed of the cutting properties, the machining sequence and the properties for the skeleton sheet and residual sheet.

Functional groups

The function bar contains the following functional groups:

- Selection
- Machining sequence
- Removal sequence
- Tilt optimization
- Skeleton cuts
- Serial-number information
- Text
- Microjoints
- Lead-in/Lead-outs
- Machining direction
- Index mark
- Snap
- View

10.4.1 Group: Machining sequence

Where do I find the group?

Select: **Modules > Part Nester > Cutting technology**

The functions in this group can be used to assign a cutting sequence manually or automatically to the parts in a cutting plan.

10.4.1.1 Set automatically



Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Machining sequence**

This function can be used to assign the machining sequence automatically to a selection of parts, to a plan or to the entire job, based on the options set in the **Automatic machining sequence** docking dialog box.

Settings

In this area you specify the options to be used by the software when computing the machining sequence.

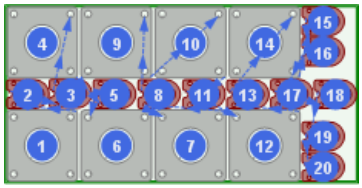
Mode

- **Constructing sequence**

  The machining sequence is set by the order in which the parts are added to the cutting plan.

- **Vertical - From left to right**

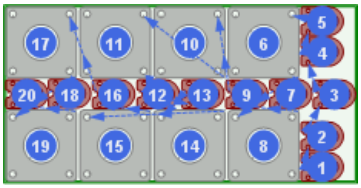
  The parts are processed vertically (up and down) starting from the lower left and progressing to the top right.



- **Vertical - From right to left**

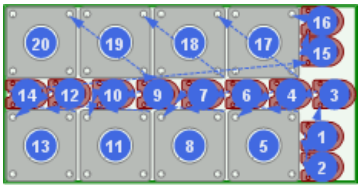
  The parts are processed vertically (up and down) starting from the lower right and progressing to the top left.

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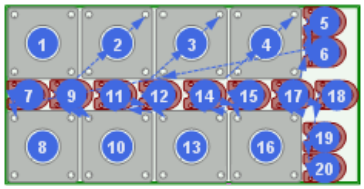
Horizontal - From bottom to top

The parts are processed horizontally (from right to left) starting from the bottom right and progressing to the top left.



Horizontal - From top to bottom

The parts are processed horizontally (from left to right) starting from the top left and progressing to the bottom right.



Random order

The machining sequence is defined by a random sequence generator. Select **Random order** to avoid too much heat entering the material when laser-cutting small parts or heavy-gage sheets.

Note: This attribute usually results in the longest machining time.

Avoid cut out parts

An attempt is made to set the machining sequence so that the cutting head no longer moves over parts that have already been cut. This reduces the possibility of the cutting head colliding with parts that have already been cut.

Tip: To prevent parts from standing up after cutting, it is recommended to use the **Tilt optimization** function in the **Cutting Technology** menu. This function additionally moves the part lead-ins to an optimal position, taking the cutting grate into account. This also increases the probability of avoiding cut parts when calculating the machining sequence.

Zigzag

The machining sequence always starts the parts in each row (horizontal) or column (vertical) from the same side as the previous one was started.

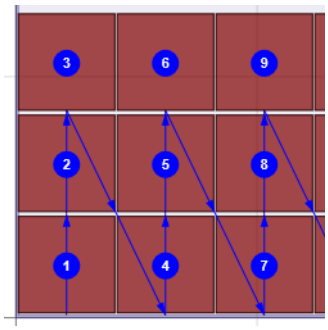


Fig. 10-73: Zigzag mode **Vertical - From left to right**

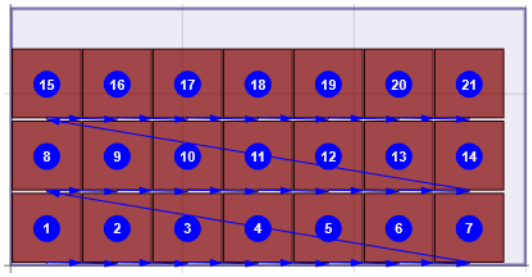


Fig. 10-74: Zigzag mode **Horizontal - From bottom to top**

☞ **Note:** The machining sequence can only be applied to the horizontal and vertical modes.

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Increment

This option can be used to set the increment between machining sequence numbers when the parts are actually cut.

- **Increment 1:** All parts are machined in accordance with the computed machining sequence (default).
- **Increment >1:** The actual machining sequence is applied to every nth part of the computed machining sequence.

Using the increment option for small parts

When the increment is set to 2, only every second part in the cutting sequence is cut. The remaining parts are cut in the second pass in the given sequence.



Fig. 10-75: Machining sequence for small parts with an increment of 2

Tip: Use an increment >1 to distribute the heat generated more evenly when cutting numerous small parts.

Number of parts in bottom corner

The specified number of parts in the lower left corner of the cutting plan are cut first in the machining sequence.

Group by sectors

Use this option to divide the total area of the sheet into sectors. Inside each sector, the parts are processed as a group using the machining sequence set under **Mode**.

Grid X/Y

This option divides the cutting plan into the specified grid and sets the machining sequence to all parts inside this grid.

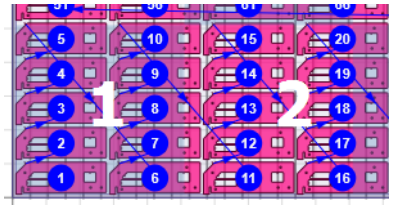


Fig. 10-76: Machining sequence in grid 1 and 2

Note: The sectors are displayed in a preview as blue and white areas defined by the dimensions entered in the **Grid X/Y** fields.

This option can be used for the following applications:

- When there are numerous small parts, in order to distribute the heat input to the material more evenly. In this case, the **Random order** mode is recommended in order to distribute the sectors randomly over the part.

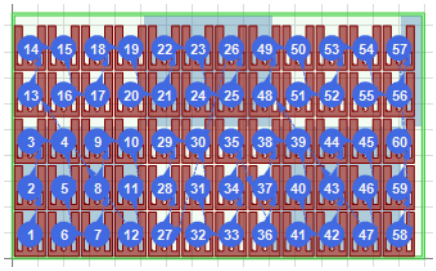


Fig. 10-77: Machining a cutting plan by sectors

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Group by size

This option is used to define the machining sequence of parts based on their size.

- **Small parts first**

  The part machining sequence starts with the smallest and ends with the largest parts.

- **Large parts first**

  The part machining sequence starts with the largest and ends with the smallest parts.

This option divides the cutting plan into the specified grid and sets the machining sequence to all parts inside this grid.

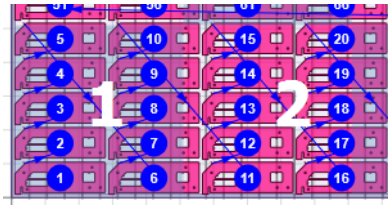


Fig. 10-78: Machining sequence in grid 1 and 2

Buttons

The buttons can be used to set the machining sequence automatically to the parts selection, to the cutting plan or the entire job, depending on the selected function.

- **Automatic - Selection**

  The machining sequence is only set on the selected parts.

- **Automatic - Plan**

  The machining sequence is set on all the parts in the current cutting plan. If the cutting plan already contains parts that have a machining sequence, those parts that have no machining sequence are appended to the machining sequence after these parts.

- **Automatic - Job**

  The machining sequence is set on all the parts of the entire job. Existing machining sequences are removed.

10.4.1.2 Set manually

Where do I find the function?

Select: **Modules > Part Nester > Cutting Technology > Machining Sequence > Manual**

This function can be used to set or edit the machining sequence manually for parts in a cutting plan.

The traversing path between two parts is indicated by a blue dotted arrow. The machining sequence is indicated by a blue number on each part that has been assigned a machining sequence.

Note: Before editing the machining sequence, first select the mode and then the part that you wish to assign the selected mode to.

Tip: The machining sequence can be set by simultaneously pressing the left mouse button and dragging the mouse over the parts.

Mode

You can use the mode in this section to create the machining sequence manually or to modify an existing sequence.



Append

The selected part is added at the end of the machining sequence.



Insert

The selected part is inserted in the machining sequence in front of the reference part.



Set reference part for insertion

The selected part is specified as the reference part for the **Insert** option. The reference part is required for the **Insert** mode and is indicated by a red number.



Delete individually

The selected part is removed from the machining sequence.



Set/remove programmed stop

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A programmed stop is set or deleted on the selected part.

10.4.1.3 Set user-defined



Where do I find the function?

Select: **Modules > Part Nester > Cutting Technology > Machining Sequence > Manual**

This function can be used for user-defined editing of the machining sequence for parts and skeleton cuts in a cutting plan.

User-defined machining sequence, properties

- The machining sequence of parts can be combined with the machining sequence of skeleton cuts.



Tip: Use the function to optimize heat build-up on the sheet when working with thick sheet by combining the machining sequences of Skeleton cuts and parts. This allows you to avoid or reduce sheet tensions during the cutting process.

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Append [I]

The selected part is or skeleton cut is added to the end of the machining sequence.

Insert [A]

The selected part or skeleton cut is inserted in the machining sequence in front of the reference part.

Set reference part for insertion [S]

The selected part or skeleton cut is defined as the reference part for the **Insert** option. The reference part is required for the **Insert** mode and is indicated by a red number.

Delete individually [X]

The selected part or skeleton cut is removed from the machining sequence.

Set / delete programmed stop [E]

A programmed stop is set or deleted on the selected part or skeleton cut.

Buttons

Use the two buttons to remove all skeleton cuts and machining sequences from the current cutting plan.

10.4.1.4 Remove machining sequence

Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Machining sequence**

This function removes the machining sequence from the active cutting plan.

Tip: If you accidentally delete the machining sequence, use the **Undo** function to restore it.

10.4.2 Group: Tilt optimization

Where do I find the group?

Select: **Modules > Part Nester > Cutting technology**

This group contains the functions for tilt optimization of cutting plans.

Tilt optimization prevents cutting gas pressure from causing parts to tilt or rise up after cutting if parts on the cutting grate have an unstable support.

Note:

- Tilt optimization must be enabled in the BySoft CAM system settings *(See: ›Tilt optimization‹ Page 24)*.
- Tilt optimization is useful when sheets are always placed in the same position on the table. This is the only way to ensure that grid contact points are correctly factored into the calculation.

10.4.2.1 Tilt zone calculation

Where do I find the function?

Select: **Modules > Part Nester > Cutting Technology > Tilt Optimization**

This function calculates the zones for tilting parts for the contours of all parts. The calculated tilt zones are shown in color on the contour.

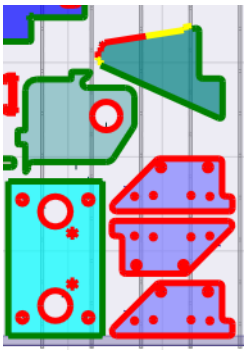


Fig. 10-79: Color-coded tilt zones on the contour

Cause of part tilting

During cutting, the parts are only resting on the tips of the cutting grate. Owing to the cutting gas pressure exerted on the sheet during cutting, parts may tilt after being cut out. Parts are always cut out at the position of the lead-in.

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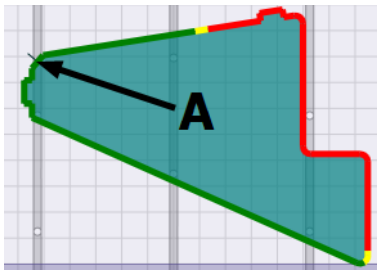
Tilt zone color coding

The colors of the tilt zones indicate the tilt behavior of the part if the lead-in is performed in this area of the contour. The colors can be edited in the tilt optimization settings.

There are three different colors for tilt zones.

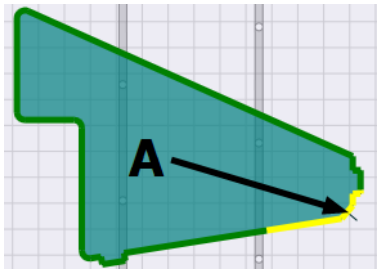
- **Green**

  The part cannot tilt. The lead-in (**A**) is located in the green zone.



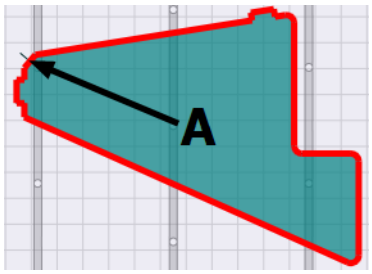
- **Yellow**

  The part may rise up to 10 degree after cutout if the cutting head immediately moves to the next lead-in with the 3-axis positioning system. If this is not the case, it may tilt. The lead-in (**A**) is located in the yellow zone.



- **Red**

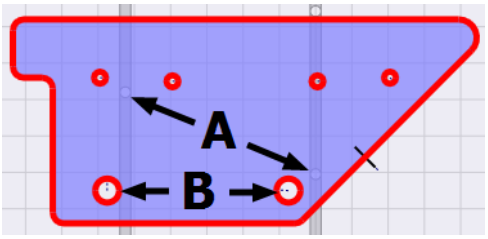
  The part will very probably tilt. The lead-in (**A**) is located in the red zone.



⚠ **Attention!** There is a risk of a collision with the cutting head. This may damage the part or the cutting head.

If the contour is completely red, the contour may not be resting stably on the cutting grate. The part is only resting on two support points (**A**).

Small contours such as holes are always red as they fall through the cutting grate after cutting (**B**).



If a segment has been assigned the **no lead-in** or **microjoint** attribute, the contour segment is always shown in red as no lead-in must be placed on this segment *(See: ›No lead-in or microjoint‹ Page 692)*.



Note: Before you can use this function, you must first configure a cutting machine, cutting gas, and cutting parameters. It is also necessary to configure a cutting grate for the selected cutting machine.

10.4.2.2 Tilt optimization

Where do I find the function?



Select: **Modules > Part Nester > Cutting Technology > Tilt Optimization**

This function performs fully automatic optimization of the tilt zones. During the process, the following changes are made:

- The program defines tilt-proof lead-ins for every contour of the nested parts. No lead-ins are set in segments with the attribute **Do not move lead-in** *(See: ›No lead-in or microjoint‹ Page 692)*.
- Outside contours for which no safe lead-in was found are automatically given microjoints.
- The option to cut waste into smaller sizes is enabled for large inside contours if no safe lead-in was found for them and if there is a great risk of a collision with the cutting head. The option must be enabled in the postprocessor export settings.
- The existing cutting sequence is reset. A new optimized cutting sequence is calculated. This prevents the program from moving over parts that have already been cut. At the same time, the part lead-ins are positioned on tilt-proof areas of the contour based on the tilt zone calculation.

It may still be necessary to move over parts that have already been cut. In this case, the machine controller will reduce cutting gas pressure briefly after the cutout and then perform a **Raise over cut parts**.

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> **Note:** Before you can use this function, you must first configure a cutting machine, cutting gas, and cutting parameters. It is also necessary to configure a cutting grate for the selected cutting machine.

10.4.2.3 Settings

Where do I find the function?



Select: **Modules > Part Nester > Cutting Technology > Tilt Optimization**

This function opens a dialog box with tilt optimization settings.

General

Mode

- **Manual loading**

  The sheet is positioned manually into the backgauges on the shuttle table.

  > **Note:** These options for the additional sheet edge are hidden as they are not needed for manual loading to backgauge position.

- **Automation**

  An automation system places the sheet automatically on the shuttle table without backgauges.

  In order to ensure that the sheet is cut with tilt optimization as calculated in BySoft CAM, it is necessary to compensate for the slightly different placement positions of the sheet caused by the automation system.

  Compensation is performed by adding an additional sheet edge.

Sheet edge allowance

The added sheet edge is calculated by means of statistical normal distribution. The statistical data come from the machine. The position of every sheet is recorded on the machine. These data are taken to calculate means and standard deviations.

> **Note:** This area is only used for **Automation** mode.

Mode

- **Manual**

  The additional edges are calculated by manually entering the standard deviation.

- **Automatic**

  The additional edges are calculated based on the calculated standard deviation from the machine's XML file.

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Expected success rate

Defines the expected rate of success in percent. The higher the value, the larger the shifts between mean and effective sheet position on the shuttle table will be accepted.

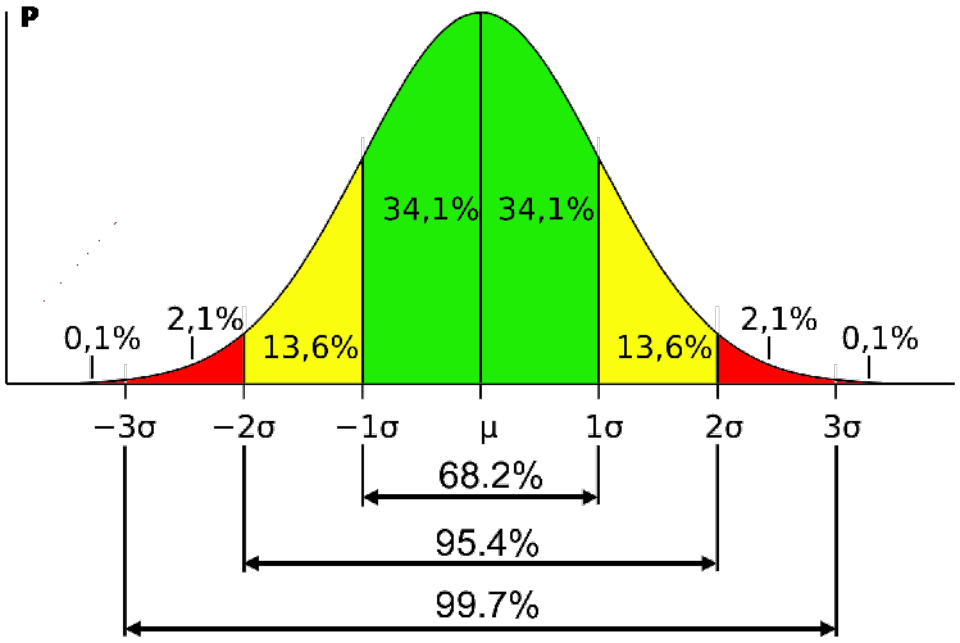


Fig. 10-80: Statistical normal distribution

Example:

- Success rate = 99.7%
- Standard deviation σ_X = **10 mm**
- Standard deviation σ_Y = **5 mm**
- Alpha = 0

☞ **Note:** Safety factor to achieve expected success rate of 99.7% = **3**

Additional edges left/right = factor • σ_X • 2* = 3 • 10 mm • 2 ≈ **60 mm**

Additional edges top/bottom = factor • σ_Y • 2* = 3 • 5 mm • 2 ≈ **30 mm**

* As the sheet can be moved in the positive and negative direction, the added sheet edge must be doubled.

Standard deviation

Standard deviation determines the possible shift of sheet position on the shuttle table in the in X and Y direction and a possible skew.

- **X/Y**

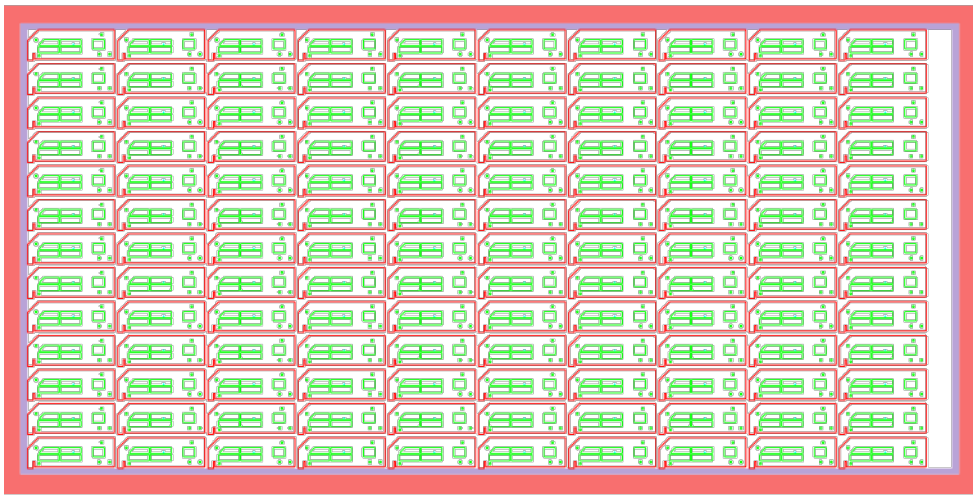
  Defines the maximum sheet offset in the X and Y direction.

- **Angle**

  Defines the maximum angle by which the sheet may be rotated.

Sheet edge calculated with standard deviation

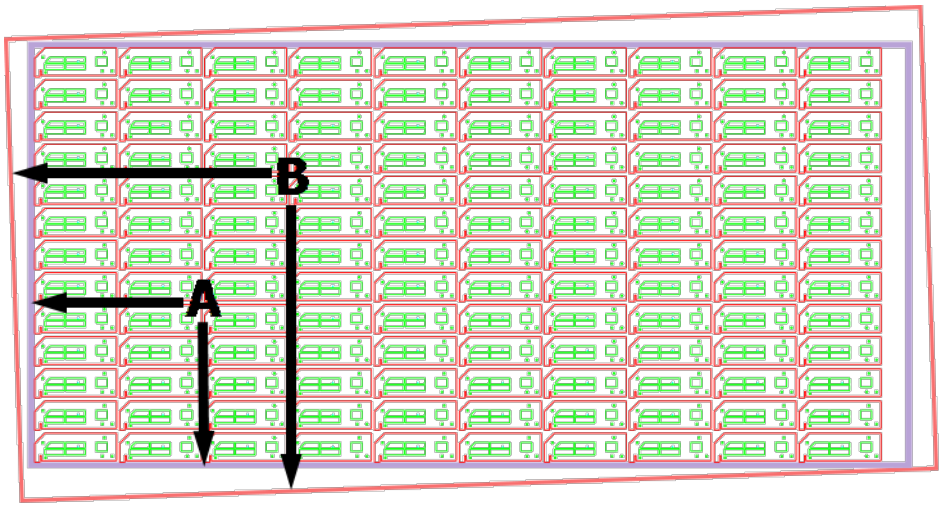
The red sheet edge is determined by standard deviation. It is added to the standard sheet edge.



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Sheet shift on the shuttle table

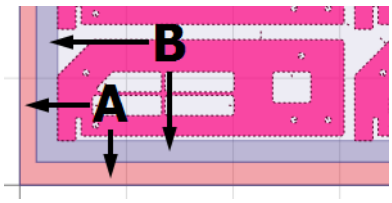
The area in which the parts are nested (**A**) including the standard sheet edge must be inside the sheet area (**B**).



Sheet edge preview

Based on the standard deviations entered and the sheet format selected, the additional sheet edge is indicated for all sides. This sheet edge is added to the standard sheet edge at the outer edge and shown in color in the cutting plan.

☞ **Note:** As soon as a sheet skew is indicated (angle), the added sheet edge is dependent on the sheet format selected.



Key:

A	Additional sheet edge
B	Standard sheet edge

Colors for part edge zones

If you want the part color and a part edge zone to have the same color, you can edit the color for the appropriate zone.

Parameters

Use the options in this section to customize calculation of the part mechanics.

Safety

Defines the correction coefficient applied to the estimated tilt force.

- Positive values improve tilt safety by increasing the safety factor by up to 25% when calculating zones. Fewer green and more yellow and red zones are shown.
- Negative values reduce tilt safety by reducing the safety factor by up to 50% when calculating zones. More green and fewer yellow and red zones are shown.

Delay after cutting

In certain machine controller versions there is a delay of **40** ms after cutting a part. During this time, the cutting gas is blown onto the end cut of the contour. This time is included for tilt safety.

In machine controllers with a delay of **0**, it may make sense to set this value to **0** as well. This value can reduce the number of microjoints on the contour.



Note: It does not make sense to increase the value as the machine only uses delays of **0** or **40** ms.

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Apply positioning error simulation

Enable this option to make part tilt behavior more tolerant with regard to shifts of the support points of the cutting grate tips (**A**).

Note: In practice, the tips of the cutting grate may be slightly bent or slightly offset as a result of wear. This option takes this into account.

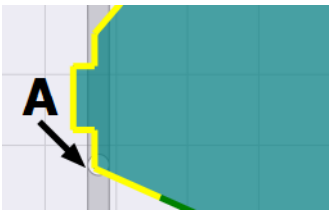


Fig. 10-81: Tilt zones without option enabled

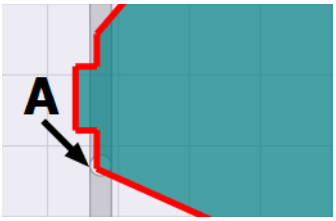


Fig. 10-82: Tilt zones with option enabled

Variance

This option determines the dimensional range used to simulate positioning error.

Optimization

These settings apply globally to all nested cutting plans.

Automatic

Optimization is performed automatically with the functions **Automatic cutting technology** in the **Nesting** menu and **Tilt optimization** -> **Automatic** in the **Cutting technology** menu.

Keep inside contour machining sequence

The machining sequence of inside contours is not changed.

Keep outside contour machining sequence

The machining sequence of outside contours is not changed.

Keep lead-ins

Lead-ins are not changed.

Number of parts in bottom corner

The specified number of parts in the lower left corner of the cutting plan are cut first in the machining sequence.

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Microjoint settings

The options in this section determine how many microjoints are set on the contours.



Note: Microjoints are only set on those part instances that the tilt optimization calculation classifies as unstable.

Existing microjoints

This option determines what to do with microjoints already set on the contour.

- **Keep all**

  Tilt optimization does not remove any existing microjoints before calculating.

- **Reset**

  Tilt optimization resets the part to its original. If the original part has a microjoint, the system keeps the microjoint on the reference of the nested part.

- **Remove all**

  Tilt optimization resets the part to its original. If the original part has a microjoint, the system removes the microjoint on the reference of the nested part.

Distributed

Sets the number of microjoints defined in **Number** evenly distributed on the selected contours.

Quantity

Use this option to define the number of microjoints to place automatically. The positions of the microjoints are displayed in a preview as gray crosses on the contour. The number of microjoints is distributed evenly along the contour.



Note: This option is only shown if the **Distributed** option is selected.

End point

Sets a microjoint at the end point of all selected contours.

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10.4.3 Group: Skeleton cuts

Where do I find the group?

Select: **Modules > Part Nester > Cutting technology**

This group contains all the functions that can be used to set skeleton sheet cuts and residual sheet cuts on cutting plans.

10.4.3.1 Set automatically



Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Skeleton cuts**

You can use this function to assign skeleton cuts and residual-sheet cuts to the current plan or the entire job.

Settings

In this area you specify the options that shall be used by the software when computing the skeleton cuts and residual-sheet cuts.

X/Y distance

This option specifies the spacing of skeleton cuts in the X and Y direction. A value of **0** means that no skeleton cuts are made in the direction concerned. The skeleton cuts are made from left to right and from bottom to top.

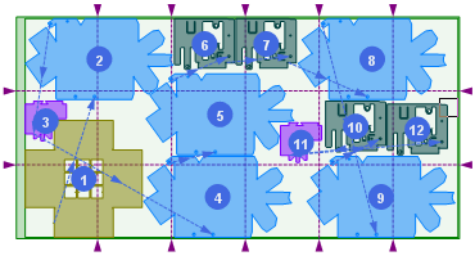


Fig. 10-83: Cutting plan containing horizontal and vertical skeleton cuts

Skeleton cuts are only set on nested areas of a sheet. Skeleton cuts are not used on residual sheets.

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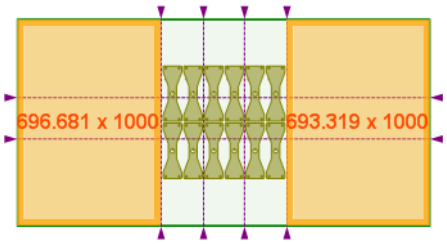


Fig. 10-84: Skeleton cuts on a partially nested sheet

Over entire sheet

This option creates skeleton cuts over the entire area of a sheet. Skeleton cuts are also used on residual sheets (**A**).

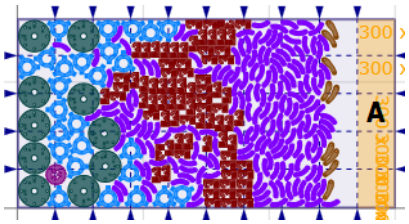


Fig. 10-85: Skeleton cuts over entire sheet

Optimize

When this option is enabled, the software tries to distribute the skeleton cuts evenly over the cutting area. If a skeleton cut lies very close to the end of the cutting area or the end of a residual sheet, the skeleton cut is moved if possible or removed.

If a skeleton cut runs along a contour of a part, the software tries to move the skeleton cut away from the contour by a certain distance. This avoids cutting the contour twice.

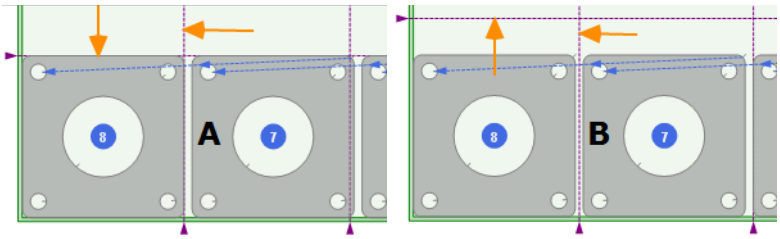


Fig. 10-86: Unoptimized (**A**) and optimized (**B**) skeleton cuts

☞ **Note:** When specifying the distances between cuts, take into account the flow of heat in the material during cutting and the size of waste container for removing the scrap material.

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Detour

This option attempts to route the skeleton cuts of cutting plans around the adjacent parts to form a single cut. For the cutting plans of ByVention machines, this option is used for the sector separating cuts when there are overlapping sectors.

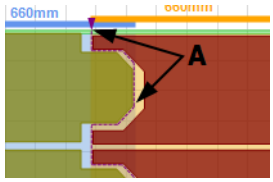


Fig. 10-87: Detour separating cut for overlapping sectors

Sector width

Use this option to set the width of the sector in which the software attempts to set a skeleton cut in the horizontal or vertical direction.



Fig. 10-88: Sector with detour skeleton cut

Residual sheet

Min. length (X)

Use this option to specify the minimum size in the X direction for which a residual sheet will still be produced. A residual sheet will be created if an area that is greater than or equal to the size set here does not contain any more nested parts.

Min. width (Y)

Use this option to specify the minimum size in the Y direction for which a residual sheet will still be produced. A residual sheet will be created if an area that is greater than or equal to the size set here does not contain any more nested parts.

Divide into segments

Using this option, large horizontal and vertical residual sheets can be divided into several smaller sheets.

- **Segment length (X)**

  Use this option to specify the length in the X direction after which a residual sheet cut is to be made. As many residual sheet cuts will be made as the overall length of the residual sheet allows for.

- **Segment width (Y)**

  Use this option to specify the width in the Y direction after which a residual sheet cut is to be made. As many residual sheet cuts will be made as the overall width of the residual sheet allows for.

- **Residual sheet cut without segment division**

  If the option **Divide into segments** is not enabled, a residual-sheet cut will only be made in the X or Y-direction based on the specifications in the **Min. width (X/Y)** field.

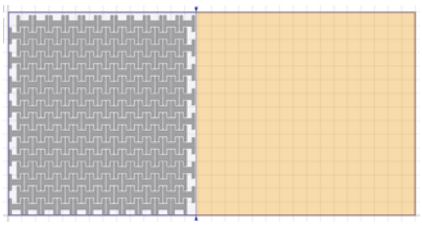


Fig. 10-89: Cutting plan with a vertical residual sheet cut

- **Residual sheet cut with segment division in the X-direction**

  If the option **Divide into segments** in the X or Y-direction is enabled, as many residual-sheet cuts will be made as the dimension in the field **Segment width (X)** or **Segment height (Y)** allows for.

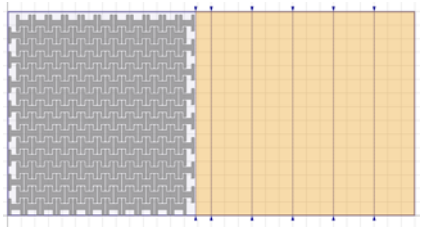


Fig. 10-90: Cutting plan with several vertical residual sheet cuts

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Shorten cuts

Use this option to specify for overlapping residual sheet cuts whether the horizontal or the vertical cuts should be shortened.

- **Residual sheet cut shortened horizontally**

  With the **Horizontal** attribute, the horizontal residual sheet cut is only made up to the intersection point with the vertical residual sheet cut.

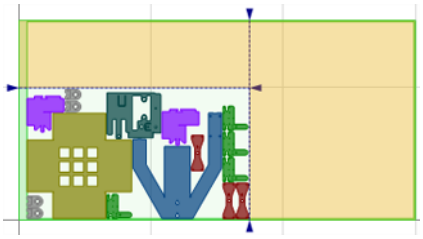


Fig. 10-91: Residual sheet cut shortened horizontally

- **Residual-sheet cuts shortened both horizontally and vertically**

  With the **Horizontal and vertical** attribute, horizontal and vertical residual sheet cuts are only made up to the point where they intersect.

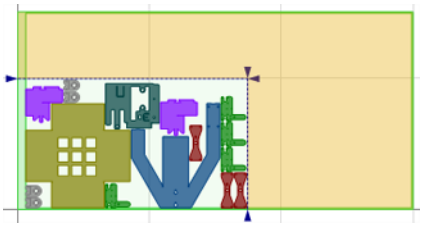


Fig. 10-92: Residual-sheet cuts shortened both horizontally and vertically

Take the following points into account when setting the minimum dimensions:

- Possibility of using the residual sheet later
- Use of filler parts
- Material costs
- Material management (standard sizes, type of storage, storage costs, type of loading system)

☞ **Note:** Nesting is performed from left to right. Consequently, the maximum Y-dimensions usually equal the sheet width and are often larger than the X-dimensions.

Cuts along contours

This option generates the residual sheet cuts along the bounding rectangles of the parts. This improves usage of the residual sheet when parts of different widths are nested.

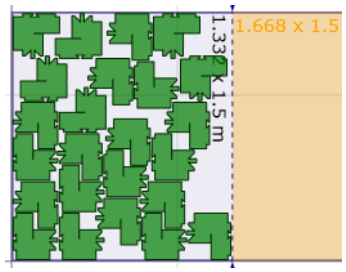


Fig. 10-93: Residual sheet with function disabled

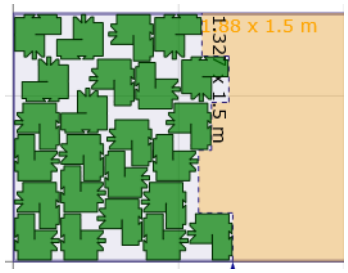


Fig. 10-94: Residual sheet with function enabled

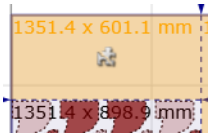
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Round position

This option rounds the size of the residual sheets to a specified number of decimal places.

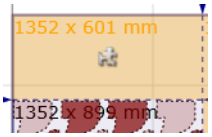
- **Do not round**

  No rounding of the calculated value is performed.



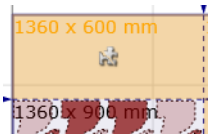
- **Nearest one**

  Rounds to the nearest one.



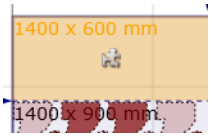
- **Nearest ten**

  Rounds to the nearest ten.



- **Nearest hundred**

  Rounds to the nearest hundred.



Inscription

This section is used to engrave details about the residual sheet as text on the residual sheet.

> **Note:** If it is not possible to place the text details on any edge owing to size, no text is engraved on the residual sheet.

Font

This option is used to define the font for the inscription text. You can use all fonts installed in the operating system.

Alignment

This option defines the alignment of text relative to the edge of the sheet.

Font size

This option is used to define the font size.

Separator

This option defines a separator to separate the **Dimension** and **Material** texts.

Edge distance

This option defines the distance of the text to the edge of the sheet. Minimum edge distance is 1 mm.

Position

This option is used to define text alignment relative to the edge of the sheet. If the texts do not fit along the specified edge, the text is automatically placed along the longest edge of the residual sheet.

The **Longest edge** option places the text at the top or on the right.

Orthogonal

This option sets the text at right angles to the selected sheet edge.

Contents

This option defines the details to be engraved on the residual sheet.

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Residual sheet sorting

In this section you can define the dimension of a skeleton sheet or residual sheet as of which to set the sorting attribute on it.

Min. length (X)

Defines the minimum length of a residual sheet in the X direction above which the attribute for sorting is set on a skeleton sheet or residual sheet.

Min. length (Y)

Defines the minimum length of a residual sheet in the Y direction above which the attribute for sorting is set on a skeleton sheet or residual sheet.

Buttons

The buttons can be used to set the skeleton cuts and residual-sheet cuts automatically on the cutting plan or the entire job, depending on the selected function.

Automatic - Plan

The skeleton cuts are set on the current plan.

Automatic - Job

The skeleton cuts are set on all the plans in the entire job. Existing skeleton cuts are removed.



Note: The button functions also apply to residual sheets.

10.4.3.2 Set manually



Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Skeleton cuts > Manual**

This function can be used to set or move straight skeleton cuts and residual sheet cuts manually on the current cutting plan. It can also be used to set skeleton cuts composed of several straight lines. This facility lets the machine cut around parts in a plan when making a skeleton cut.

Manual skeleton/residual sheet cuts, properties

- The machining sequence of existing skeleton and residual sheet cuts is determined by the postprocessor and cannot be influenced.
- Skeleton cuts are always performed before parts are cut and residual sheet cuts are always performed after parts are cut.



Note: Depending on the settings in the **Automatic** function in the **Residual sheet settings** section, residual sheets are automatically created or deleted when setting or moving skeleton cuts *(See: ›Residual sheet‹ Page 1052)*.

Saving skeleton areas as residual sheets

You can save areas of skeleton cuts (**A**) as a residual sheet (**B**).

1. Select a skeleton area to save as a residual sheet.
2. Select the **Save residual sheet** function from **Data** menu.
3. Enter the name of the residual sheet in the **Name** field.
4. Press Save to save the residual sheet. You can now add the residual sheet to the **Sheets** tab using the **Add residual sheet** function.

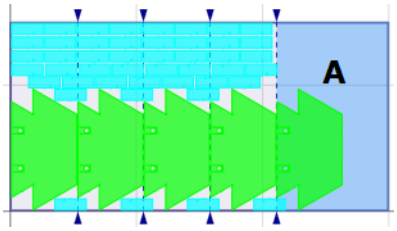


Fig. 10-95: Skeleton area to save as residual sheet

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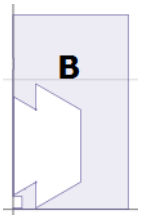


Fig. 10-96: Residual sheet saved from skeleton area

Mode

You can use the modes in this area to set, move or delete skeleton cuts.

Tip: Use detour skeleton cuts to create asymmetric residual sheet cuts. When setting the skeleton cuts, the left mouse button adds a new point while the right mouse button removes the last point that was added.

Insert vertically

Left-clicking on the position you require will insert a vertical, straight skeleton cut in this position.

Insert horizontally

Left-clicking on the position you require will insert a horizontal, straight skeleton cut in this position.

Vertical detour

Left-clicking several times in any position in the cutting plan will insert a vertical skeleton cut composed of several straight lines. To set the last point on the vertical sheet edge, left-click on the appropriate position outside the sheet.

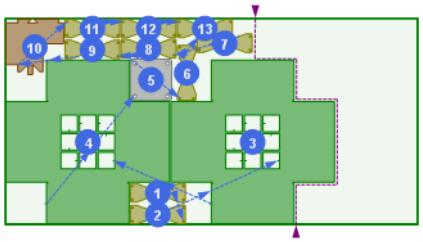


Fig. 10-97: Manually set vertical skeleton cut

Vertical detour (auto)

When the blue sector is positioned over the cutting plan and you click on the left mouse button, the program attempts to set a vertical skeleton cut within the sector.

Tip: If no skeleton cut can be set, enlarge the sector width and execute the option once again.

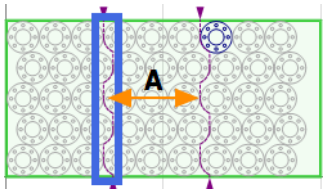


Fig. 10-98: Automatically set skeleton cuts with vertical detour (**A**)

Horizontal detour

Left-clicking several times in any position in the cutting plan will insert a horizontal skeleton cut composed of several straight lines. To set the last point on the horizontal sheet edge, left-click on the appropriate position outside the sheet.

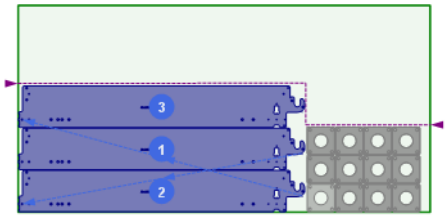


Fig. 10-99: Skeleton cut with horizontal detour

Horizontal detour (auto)

When the blue sector is positioned over the cutting plan and you click on the left mouse button, the program attempts to set a horizontal skeleton cut within the sector.

Tip: If no skeleton cut can be set, enlarge the sector width and execute the option once again.

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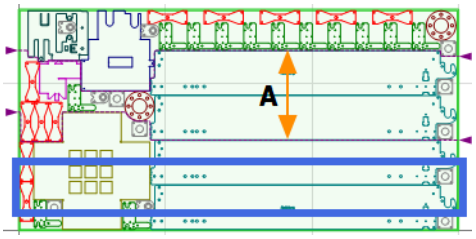


Fig. 10-100: Automatically set skeleton cuts with horizontal detour (**A**)

Move

The mouse moves the selected skeleton cut to any position in the cutting plan.

Note: Use the **Save residual sheet** function from the **Data** menu to save the residual sheets in the database.

Moving a skeleton cut

1. Left-click on the skeleton cut.
2. Use the mouse to drag the skeleton cut to the position you require.
3. Left-click on this position.

Delete

Left-clicking on a skeleton cut will delete it.

Trim

This option is used to shorten intersecting skeleton cuts (**A**) to the common point of intersection (**B**).

Select the skeleton cut from the point of intersection to shorten.

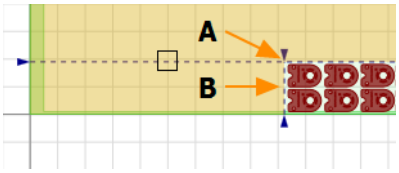


Fig. 10-101: Trimming skeleton cuts on a residual sheet.

 Add inscription

This option adds an inscription on the residual sheet. The inscription is engraved based on the details in the **Inscription** section.

Procedure…

1. Press button.
2. Click on residual sheet. The program adds the inscription.

 Remove inscription

This option removes the inscription on the residual sheet.

Procedure…

1. Press button.
2. Click on residual sheet. The program removes the inscription.

 Add sorting

This option can be used to stack the residual sheet on a stack carrier via a sorting system. Sortable residual sheets are indicated by am icon.

The sortable residual sheets can be stacked on a stacking site via the **Waste parts** tab in the following modules.

- Part Nester -> Sort Editor -> Sort data
- Sort Job Editor -> Job data

.Procedure…

1. Press button.
2. Click on residual sheet. The sorting is added to the residual sheet.

 Remove sorting

This option removes the sorting from the residual sheet.

Procedure…

1. Press button.
2. Click on residual sheet. The sorting is removed.

Settings for detour separation cuts

Sector width

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Use this option to set the width of the sector in which the software attempts to set a skeleton cut in the horizontal or vertical direction.



Fig. 10-102: Sector with detour skeleton cut

Buttons

Skeleton sheet cuts on enclosing rectangle

This function automatically sets a maximum of 4 skeleton cuts on the bounding rectangle of all the parts on a cutting plan.

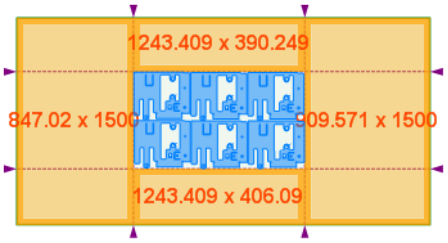


Fig. 10-103: Skeleton sheet cuts on enclosing rectangle

Inscription

This section is used to engrave details about the residual sheet as text on the residual sheet.

Buttons

Set inscriptions

This function automatically sets an inscription on all available residual sheets.

Remove inscriptions

This function automatically deletes the inscription from all available residual sheets.

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☞ **Note:** If it is not possible to place the text details on any edge owing to size, no text is engraved on the residual sheet.

Font

This option is used to define the font for the inscription text. You can use all fonts installed in the operating system.

Alignment

This option defines the alignment of text relative to the edge of the sheet.

Font size

This option is used to define the font size.

Separator

This option defines a separator to separate the **Dimension** and **Material** texts.

Edge distance

This option defines the distance of the text to the edge of the sheet. Minimum edge distance is 1 mm.

Position

This option is used to define text alignment relative to the edge of the sheet. If the texts do not fit along the specified edge, the text is automatically placed along the longest edge of the residual sheet.

The **Longest edge** option places the text at the top or on the right.

Orthogonal

This option sets the text at right angles to the selected sheet edge.

Contents

This option defines the details to be engraved on the residual sheet.

Residual sheet sorting

In this section you can assign the attribute for sorting residual sheets and skeleton sheets to all existing residual sheets and skeleton sheets of the current cutting plan.

The dimension of a skeleton sheet or residual sheet as of which to set the Sort attribute can be specified using the appropriate options.

Note: Skeleton sheets and residual sheets that have the Sort attribute are indicated by an icon ( ).

Set sorting

Sets the attribute for sorting on all skeleton sheets and residual sheets of the current cutting plan.

Remove sorting

Removes the attribute for sorting from all skeleton sheets and residual sheets of the current cutting plan.

Min. length (X)

Defines the minimum length of a residual sheet in the X direction above which the attribute for sorting is set on a skeleton sheet or residual sheet.

Min. length (Y)

Defines the minimum length of a residual sheet in the Y direction above which the attribute for sorting is set on a skeleton sheet or residual sheet.

10.4.3.3 Set user-defined



Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Skeleton cuts > Manual**

This function can be used to set user-defined skeleton cuts on the current cutting plan.

User-defined skeleton cuts, properties

- The machining sequence of skeleton cuts can be defined freely. The machining sequence of parts can be combined with the machining sequence of skeleton cuts.

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Note: Skeleton cuts without a machining sequence are cut at the end of the cutting plan. A warning is displayed when you export the cutting plan.

- Skeleton cuts can be defined in any direction.
- Only skeleton cuts can be defined but no residual sheet cuts.

Tip: Residual sheet cuts can be set in the **Manual** section.

Mode

Use the functions in this section to create or edit user-defined skeleton cuts.

Tip: Use the letters in square brackets to select the appropriate function using the keyboard.

Insert horizontal [H]

Left-click to insert the start point of a horizontal skeleton cut at this position.

Click again to define the end point.

Insert vertical [V]

Left-click to insert the start point of a vertical skeleton cut at this position.

Click again to define the end point.

Insert cut [F]

Left-click to insert the start point of a freely positionable skeleton cut at this position.

Click again to continue the skeleton cut in any direction.

Insert orthogonal [O]

Left-click to insert the start point of an orthogonal skeleton cut at this position.

Click again to continue the skeleton cut in the horizontal or vertical direction.

Move cut [M]

The mouse moves the selected skeleton cut to any position in the cutting plan. The start and end of the skeleton cut is automatic ally adjusted up to the next contour, sheet edge, or next residual sheet cut.

Trim cut [T]

This option is used to shorten user-defined skeleton cuts (**A**) between points of intersection (**B**). Points of intersection can be other skeleton cuts or contours between which the skeleton cuts runs.

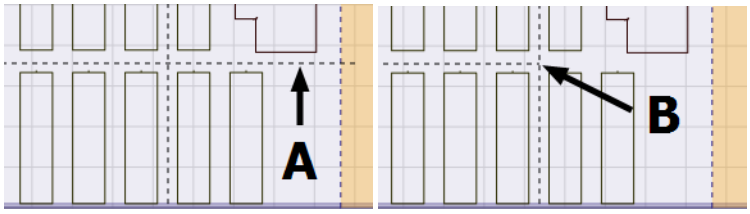


Fig. 10-104: Trimming skeleton cuts on a cutting plan

Delete cut [R]

Left-clicking on a skeleton cut will delete it.

Convert cut [C]

Use this option to convert manual skeleton cuts into user-defined skeleton cuts.

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Change orientation [D]

Use this option to invert the direction of cutting a skeleton cut. The direction of cutting is indicated by a little blue arrow on the skeleton cut.

Machining sequence

This function can be used for user-defined editing of the machining sequence for parts and skeleton cuts in a cutting plan.

Tip: Use the function to optimize heat build-up on the sheet when working with thick sheet by combining the machining sequences of Skeleton cuts and parts. This allows you to avoid or reduce sheet tensions during the cutting process.

Append [I]

The selected part is or skeleton cut is added to the end of the machining sequence.

Insert [A]

The selected part or skeleton cut is inserted in the machining sequence in front of the reference part.

Set reference part for insertion [S]

The selected part or skeleton cut is defined as the reference part for the **Insert** option. The reference part is required for the **Insert** mode and is indicated by a red number.

Delete individually [X]

The selected part or skeleton cut is removed from the machining sequence.

Set / delete programmed stop [E]

A programmed stop is set or deleted on the selected part or skeleton cut.

Buttons

Use the two buttons to remove all skeleton cuts and machining sequences from the current cutting plan.

Inscription

This section is used to engrave details about the residual sheet as text on the residual sheet.

Buttons

Set inscriptions

This function automatically sets an inscription on all available residual sheets.

Remove inscriptions

This function automatically deletes the inscription from all available residual sheets.



Note: If it is not possible to place the text details on any edge owing to size, no text is engraved on the residual sheet.

Font

This option is used to define the font for the inscription text. You can use all fonts installed in the operating system.

Alignment

This option defines the alignment of text relative to the edge of the sheet.

Font size

This option is used to define the font size.

Separator

This option defines a separator to separate the **Dimension** and **Material** texts.

Edge distance

This option defines the distance of the text to the edge of the sheet. Minimum edge distance is 1 mm.

Position

This option is used to define text alignment relative to the edge of the sheet. If the texts do not fit along the specified edge, the text is automatically placed along the

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longest edge of the residual sheet.

The **Longest edge** option places the text at the top or on the right.

Orthogonal

This option sets the text at right angles to the selected sheet edge.

Contents

This option defines the details to be engraved on the residual sheet.

Residual sheet sorting

In this section you can assign the attribute for sorting residual sheets and skeleton sheets to all existing residual sheets and skeleton sheets of the current cutting plan.

The dimension of a skeleton sheet or residual sheet as of which to set the Sort attribute can be specified using the appropriate options.

Note: Skeleton sheets and residual sheets that have the Sort attribute are indicated by an icon ( ).

Set sorting

Sets the attribute for sorting on all skeleton sheets and residual sheets of the current cutting plan.

Remove sorting

Removes the attribute for sorting from all skeleton sheets and residual sheets of the current cutting plan.

Min. length (X)

Defines the minimum length of a residual sheet in the X direction above which the attribute for sorting is set on a skeleton sheet or residual sheet.

Min. length (Y)

Defines the minimum length of a residual sheet in the Y direction above which the attribute for sorting is set on a skeleton sheet or residual sheet.

10.4.3.4 Remove skeleton sheet cuts

Where do I find the function?



Select: **Modules > Part Nester > Data > Skeleton cuts**

This function removes all the skeleton sheet cuts and residual sheet cuts from the active cutting plan.

Tip: If you accidentally delete the skeleton sheet cuts, use the **Undo** function to restore them.

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10.4.4 Group: Serial-number information

Where do I find the group?

Select: **Modules > Part Nester > Cutting technology**

The functions in this group can be used to place serial-number information into parts that have a series field. With serial-number information, you can specifically identify every part with a consecutive number *(See: ›Create serial-number information‹ Page 1338)*.

Application:

- For the unique identification of a part for quality assurance

10.4.4.1 Automatic serial-number information



Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Serial-number information**

This function automatically assigns the serial-number information to all parts with a series field. The assignment of the serial-number information is done for all nested cutting plans of the job.

In order to be able to use the function, the **Serial-number information** data field must be set for the part in Part Editor using the **Insert** function from the **Text menu** *(See: › Insert data field‹ Page 491)*. The default setting is for the serial-number information to be engraved on the part. The start number of the first part is defined in the **Part properties** dialog.

Attention! If parts with serial-number information are nested in a cutting plan with multiple cycles, the cutting plan must be separated again using the **Separate** function in the **Geometry** menu. Otherwise, the same serial-number information will be on the parts in every cycle.

Note: After every assignment of the function, the start number of the part is incremented by the number of parts in the job.

Tip: To reset the start number, enter the appropriate start number in the **Series info - Start number** column on the **Parts** tab. *(See: ›Start number‹ Page 936)*.

10.4.4.2 Manual serial-number information



How do I open this dialog box?

Select: **Modules > Part Nester > Cutting technology > Serial-number information**

Use this function to manually specify a text via the input field that is to be placed on a part in the job which has a series field.



Attention! When the **Automatic** function is once again executed, the manually entered text is overwritten by the serial-number information in the **Part properties** dialog box.

10.4.4.3 Removing serial-number information



Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Serial-number information**

This function removes the serial-number information from all parts. The current start number is retained.

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10.4.5 Group: Text

Where do I find the group?

Select: **Modules > Part Nester > Cutting technology**

This group contains functions that can be used to set any desired texts on a sheet. Using job text information to identify sheets can serve quality assurance purposes in the production process.

Note: When creating engraved text on sheets, be sure that the text does not run onto skeleton cuts or between the contours of parts.

10.4.5.1 Insert text



Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Text**

With this function you can set any text on the sheet or parts in the cutting plan. You can also set predefined texts from various data fields on the sheet or part by pressing the  button. The **Engrave** operating mode is automatically assigned to the text.

Tip: You can position the text quickly and easily by left-clicking on a location inside the sheet.

Setting texts on a sheet or part

1. Select the **Insert text** function.
2. Select the mode with which to set the text on the sheet or parts.
3. Enter the desired text or the data field in the **Text** field.
4. Format the text and then arrange it on the sheet using the relevant options.
5. Assign the text to the sheet or part by pressing the **Apply** button and save the job.

Insert data field

1. Select the **Insert text** function.
2. To do so, click on the  icon at the right end of the input field.
3. Select the data field in the corresponding category.

4. Left-click to set the data field at the appropriate position on the sheet. The contents of the data field are displayed at the insertion point.

Note: Texts are only generated from data fields when the data fields concerned actually contain text.

Mode

In this section you can choose whether to place the text on the sheet or on the parts in the cutting plan.

What each option does:

- **Plan**
  Sets the text at the selected position of the sheet in the cutting plan.
- **Part**
  Sets the text on all instances of the selected part in the cutting plan.

Note: The text can be deleted again using the interactive button in the cutting plan.

Text

You can enter or re-edit the text to engrave in the **Text** field. The added data fields are also entered in this field. These data fields are defined as code and therefore must not be changed.

What each field does:

- **Material**
  All available data fields for the material will be shown.
- **Cutting plan**
  All available data fields for the cutting plan are displayed.
- **Sheet**
  All available data fields of one sheet are displayed.
- **Residual sheet**
  The **Dimension** data field will be shown. This data field can be set on nested sheets and residual sheets.

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- **Miscellaneous**
  Additional data fields will be shown.

Options

- **Only in the selected part**
  Sets the text only on the selected part. If the option is not enabled, the text will be set on all instances of the part in the cutting plan.

> ☞ **Note:** The option is displayed only if the **Part** module is selected.

Formatting

In this area you can format the text entered in the **Text** field and also align the text on the cutting plan.

What each option does:

- **Font**
  Defines the font with which to create the text. For cutting text, use special laser fonts which preserve the font outline.
- **Line text**
  This option only displays fonts that consist only of a line. These fonts minimize inscription time during engraving.
- **Font size**
  This option changes the font size of the entered text.
- **Rotation**
  You can use this option to rotate the text through a specified angle.
- **Mirror**
  Mirrors the text around the vertical axis.
- **Bold**
  This option is used to increase the text contour size slightly.
- **Italic**
  This option is used to italicize the text contour.
- **Alignment**
  This option is used to define the position in which you wish to arrange the text. Click on the particular point in the array that corresponds to the text alignment you require.

Tip: The Snap functions can also be used for aligning the text.

Label text

This option prints the text on the material with an inkjet printer. No cutting technology is assigned to the text. This option can only be used if an inkjet printer has been configured for the machine.

Note: Use this option if you need to remove the text from the material.

Position

In this area, the text can be positioned on the sheet at four different predefined positions using the buttons or via the X and Y-coordinates.

10.4.5.2 Edit text

Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Text**

This function can be used to edit an existing text. This function cannot be used until an existing text has been selected. Otherwise the function works in the same way as the **Insert text** function *(See: ›Insert text‹ Page 1076)*.

10.4.5.3 Remove text

Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Text**

With this function you can remove texts on a sheet from the sheet again. To remove text, first select the **Remove** function and then select the text to be deleted from the sheet.

Note: The function remains enabled until another function is selected or it is canceled with the **Esc** key.

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10.4.6 Group: Microjoints

Where do I find the group?

Select: **Modules > Part Nester > Cutting technology**

This group contains the functions for editing microjoints.

10.4.6.1 Edit



Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Microjoints**

This function can be used to set as many microjoints or FixMasters as you want on contours or to remove them.

 **Tip:** The Snap functions can be used for placing microjoints.

Set microjoint

1. Enable the **Set** function.
2. Left-click on the position on the contour in which you want to set a microjoint. The microjoint is indicated by a black cross on the contour.

Mode

Set

Sets the microjoint (**A**) on the selected contour.

If the **Execute FixMaster** option (**B**) is set on a contour, all microjoints are executed as gluing points (ByJet only).

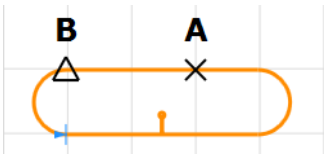


Fig. 10-105: Microjoint and FixMaster on a contour

Remove

Removes the selected microjoint or FixMaster from the contour.

Copy

Copies the microjoint or FixMaster from the contour of the selected part to the instances of the part in the cutting plan or in the entire job.

Run FixMaster (ByJet only)

Specifies the pierce exit point (**B**) for executing the FixMaster and sets a gluing point on all microjoints defined (**A**).

Note: Microjoints situated behind the last FixMaster execution point are ignored and no longer set.

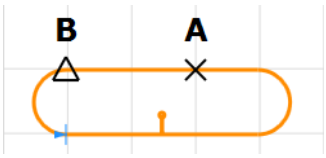


Fig. 10-106: Microjoint and FixMaster on a contour

General

Offset

This option can be used to set a microjoint at a certain distance to a point on the contour. A negative figure sets the microjoint before the selected point on the contour.

Set microjoint with offset

1. Enter the distance to the appropriate point on the contour in the **Offset** box.
2. Select the point on the contour. The microjoint is set at the specified distance to this point.

Tip: Use this option for water jet cutting to set a microjoint before the lead out. This means that the part stays attached to the sheet and is not damaged when the water jet pulls out.

Copy

These options can be used to apply microjoints of the selected part to different instances of the part. Any microjoints that already exist on instances are removed.

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> **Note:** The function can only be used if the **Copy** option is selected in the **Mode** section. Only one part in the plan can be selected.

Apply for same parts

If this option is enabled, movement of microjoints will only be applied to identical parts.

> **Note:** The function can only be used if the **Set** or **Remove** option is selected in the **Mode** section.

Parts

- **Plan**: Copies the microjoints to all instances of the part in the active cutting plan.
- **Job**: Copies the microjoints to all instances of the selected part in all cutting plans of the job.

Alignment

- **All**: Copies the microjoints with any orientation to all instances of the selected part based on the settings in the **Parts** option.
- **Same orientation**: Copies the microjoints to all instances of the selected part with the same orientation based on the settings in the **Parts** option.

Apply

Copies the microjoints to instances of the selected part based on the **Parts** and **Orientation** settings.

> **Note:** The function is only displayed when **Copy** mode is selected.

10.4.7 Group: Lead-in/Lead-outs

Where do I find the group?

Select: **Modules > Part Nester > Cutting technology**

This group contains the functions for editing lead-ins and lead-outs.

10.4.7.1 Editing lead-ins/lead-outs



Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Lead-ins/Lead-outs**

This function can be used to edit the lead-ins and lead-outs of individual contours of parts on the cutting plan.

The following modes are available:

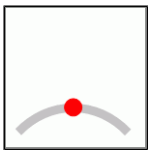
- **Set**

  These options are used to edit the lead-in/lead-out type of the selected part. You can also change the Operating mode for the appropriate lead-in/lead-out type.

Lead-in/lead-out types

- **Direct**

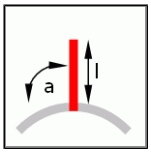
  The initial piercing cut is performed directly on the contour.

  This lead-in is mainly used on thin sheets up to about 1.5mm thick and for open contours, where impairing the contour does not matter too much. Rapid piercing directly on the contour makes this lead-in the choice for high-speed cutting. This is the default lead-out setting for laser machines.

- **Straight**

  A straight lead-in is set on the contour.

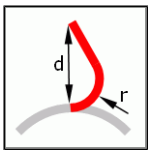
  - **length (l)** = length of the lead-in
  - **angle (a)** = angle of the straight line to the contour

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This lead-in is used for most contours. This lead-in causes the least damage to the contour but is rather slower than the **Hook** lead-in. This lead-in is also the best choice for microjoints and pulsed piercing. This is the default lead-out setting for water jet cutting machines.

- **Hook**

  This lead-in makes a hook shape with the contour.



  - **distance (d)** = perpendicular distance of initial piercing cut from contour
  - **radius (r)** = radius of the hook path taken to the contour.

  This lead-in offers better dynamic performance compared with the **Straight** lead-in and can be used to reduce the heat build-up in thick sheets and difficult contours. The higher speed used for the lead-in means more damage to the contour.

- **Tangential**

  The lead-in is set in an arc tangentially to the contour.

> **Note:** If this type of lead-in is used, the position **Along the cut sequence** must be selected.



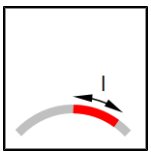
  - **Length (l)** = length of the arc
  - **radius (r)** = radius of the arc for the lead-in

- **Following the contour**

  The lead-in or lead-out is set directly following the contour.

  Application:

  As lead-out for water jet cutting so that the contour is reliably cut completely thanks to the drag. The pierce exit is performed by the specified length of the end cut after the contour lead-in.



- **Length (l)** = length of the lead-in or lead-out on the contour

Ramp

A straight lead-in is set with an arc at a tangent to the contour.

- **Length (l)** = total length of the lead-in including arc
- **Angle (a)** = angle of the straight line to the contour
- **Radius (r)** = radius of the arc

☞ **Note:** Only displayed if lead-ins are set manually.

Application:

Improved influence on heat build-up during piercing.

Automatic

The type and position of the lead-in is set in correspondence to the contour shape. This means that each contour has the optimal lead-in and lead-in position assigned to it.



☞ **Note:** Only displayed in the settings for automatic setting of lead-ins.

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- **Move**

  Moves the selected lead-in along the contour.

  Procedure…

  1. Select the Edit function and then the lead-in on the contour.
  2. Use the mouse to drag the lead-in to the appropriate position.
  3. Left-click on the point to set the lead-in at the selected position.

- **Copy**

  Copies the modified lead-in of a part to all instances of the part in the cutting plan or to the instances in the entire job.

  - **Apply for same parts**

    If this option is enabled, movement of lead-ins will only be applied to identical parts.

    ☞ **Note:** The function can only be used if the **Move** option is selected in the **Mode** section.

  - **Parts**
    - **Plan**: Copies the lead-in to all instances of the selected part in the current cutting plan.
    - **Job**: Copies the lead-in to all instances of the selected part in all cutting plans of the job.
  - **Alignment**
    - **All**: Copies the lead-in to all instances of the selected part.
    - **Same orientation**: Copies the lead-in to all instances of the selected part with the same orientation in the cutting plan.

10.4.8 Group: Machining direction

Where do I find the group?

Select: **Modules > Part Nester > Cutting technology**

This group contains functions for changing the direction of cutting on the contour of a part.

10.4.8.1 Change machining direction



Where do I find the function?

Select: **Modules > Part Nester > Cutting technology > Direction of rotation**

This function can be used to change the direction in which a contour is cut.

- Sets the direction of cutting for the selected parts to clockwise.
- Sets the direction of cutting for the selected parts to counterclockwise.

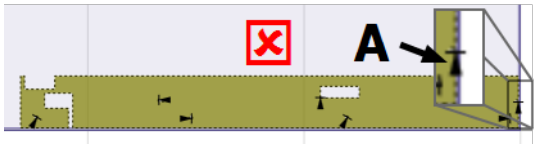
Application:

To improve dimensional accuracy of long parts nested close to the edge of the sheet and whose lead-in must not be changed.

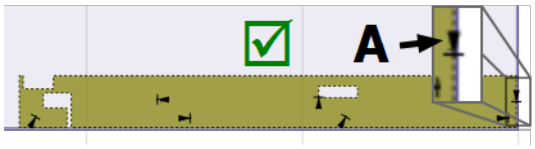
If the first contour cut is the one on the inside of the sheet, heat build-up during cutting can cause tensions on the outside of the contour that may result in dimensional inaccuracies.

By changing the direction of cutting, it is possible to cut the contour of the part on the outside of the sheet first.

- Direction of cutting (**A**) from right lead-in counterclockwise.



- Direction of cutting (**A**) from right lead-in clockwise.



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10.4.9 Group: Index marking

Where do I find the group?

Select: **Modules > Part Nester > Cutting technology**

This group contains the functions for setting markings.

10.4.9.1 Insert markings



Where do I find the function?

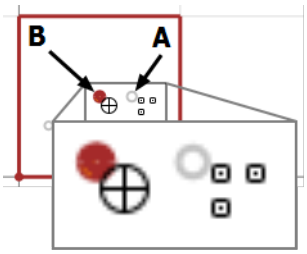
Select: **Modules > Part Nester > Cutting Technology > Index Marking**

This function can be used to set markings on the sheet. Markings are used in order to make it possible to detect the position of a sheet or blank on the shuttle table.



Attention! When inserting index marks, it is always necessary to set two index marks. Otherwise it is not possible to create the cutting plan.

Fig. 10-107: Index and calibration marks on a sheet



A	Index mark
B	Calibration mark

Highlighting

Type

Two types of markings can be set.

Index marking (A)

Defines an existing round contour hole on a sheet as an index marking (**A**). There must be at least two holes per sheet that can be used as index markings.

Before the part contour is cut, the index holes are detected with **Detection Eye** in order to determine the position of the sheet.

> **Note:** Index marks are not cut.

Calibration marking (B)

Defines a round contour on a sheet as a calibration marking (**B**). This contour is cut in order to compensate for the expansion caused by the material between the cutting head and **Detection Eye** being heated up.

Before the part contour is cut, the calibration holes are detected with **Detection Eye** in order to determine the position of the sheet more accurately.

Dimension

Position X/Y

These options can be used to specify the X-axis and Y-axis positions for the marking on the sheet.

Diameter

This option is used to specify the diameter for the marking. The marking diameter must be between **5-45 mm**.

10.4.9.2 Editing markings



Where do I find the function?

Select: **Modules > Part Nester > Cutting Technology > Index Marking**

This function can be used to edit existing marks *(See: ›Insert markings‹ Page 1088)*.

10.4.9.3 Remove marking



Where do I find the function?

Select: **Modules > Part Nester > Cutting Technology > Index Marking**

With this function, previously created markings can be removed from the sheet. To remove markings, first select the **Remove** function and then the marking to be deleted.

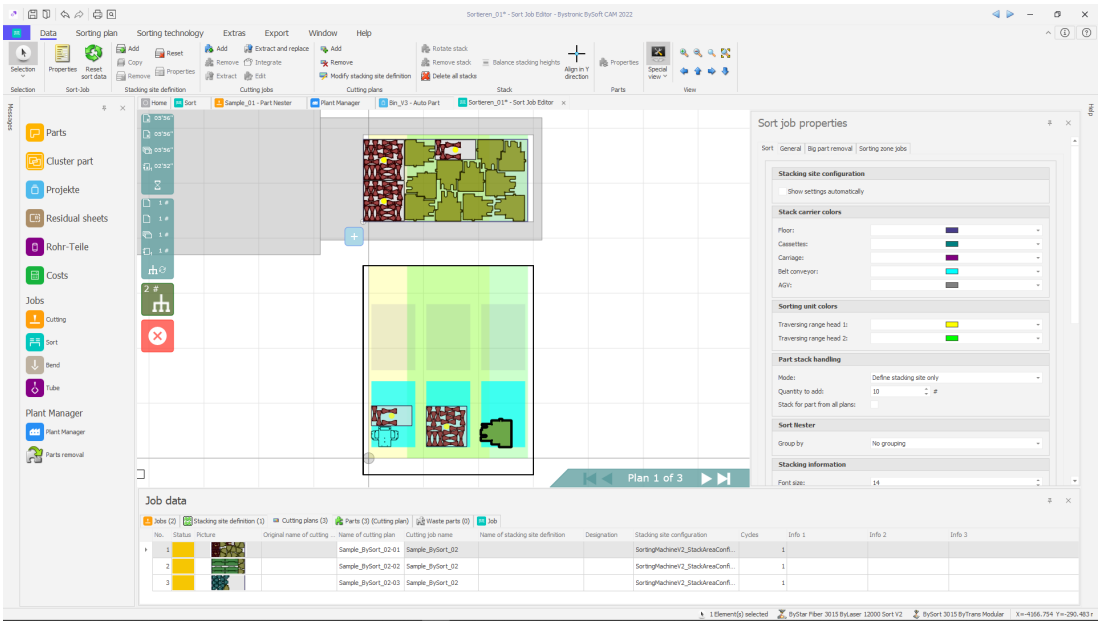
> **Note:** The function remains enabled until another function is selected or it is canceled with the **Esc** key.

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10.5 Menu: Sort Editor

Where do I find the menu?

Select: **Modules > Part Nester**



The Sort Editor menu can be used to generate sorting plans based on jobs. Sorting information is exported together with cutting plans by the postprocessor and imported into the machine.

Ctrl+C copies a stack to the clipboard. **Ctrl+V** pastes the copied stack onto the selected stacking site definition.

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Context menu functions

Creating a sorting plan

1. Select the **Sort Editor** menu.
2. Select the **Add** function in the **Cutting plans** section.
3. In the **Add cutting plans** dialog box, select the **Select all** option to assign all cutting plans of the job to a sorting plan.
4. In the **Configuration** list box, select the stacking site configuration that should be used to stack the parts.
5. Optionally, enter a name for the stacking site into the **Name** field and confirm with OK.
6. The cutting plans will be assigned to the stacking site configuration and shown in the **Sort data** dialog box.
7. Select the **Automatic** function in the **Sort** section. The sorting plan will be automatically created for each cutting plan based on the stacking site definition.

Copy a stack to a different stacking site definition.

1. Select the stack on the stacking site definition.
2. Press **Ctrl+C** to copy the stack to the clipboard.
3. Select the stacking site definition to paste the stack into on the **Stacking site definitions** tab.
4. Press **Ctrl+V** to paste the stack into the stacking site definition. The stack is pasted into the stacking site definition in its original position.

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Icons in the workspace

Various icons to call up information about the sorting plan are displayed on the left in every sorting plan. The display is refreshed automatically as soon as any change is made to the sorting plan.

Symbol	Description
	Icon for sorting times section
	Sorting time for all the stacking site definitions in the job.
	Sorting time for the stacking site definition that is currently selected. If there is only one stacking site definition, the sorting time will be the same as that for the job.
	Sorting time for the cutting plan that is currently selected. If a cutting plan contains multiple cycles, sorting time is only displayed for one cycle.
	Icon for tool change section
	Number of tool changes for all the stacking site definitions in the job.
	Number of tool changes for the stacking site definition that is currently selected. If there is only one stacking site definition, the number of tool changes is the same as that for the job.
	Number of tool changes for the cutting plan that is currently selected. If a cutting plan contains multiple cycles, the number of tool changes is only displayed for one cycle.
	Displays the number of gripper tools used in the sorting plan. **Tip:** Click on the icon to display the **Check gripper tools** dialog with detailed information about the gripper tools.

Symbol	Description
	Displays the status of the stacking site definition. The status is updated in real time. ■ **Symbol 1:** The stacking site definition is OK. The stacking site definition can be exported. ■ **Symbol 2:** The stacking site definition contains errors. The stacking site definition can be exported. ■ **Symbol 3:** The stacking site definition contains errors. The stacking site definition cannot be exported. **Tip:** Click on this symbol to call up the **Check stacking site definition** dialog with detailed information about the status of the stacking plan.
	Navigation bar for switching between multiple cutting plans.

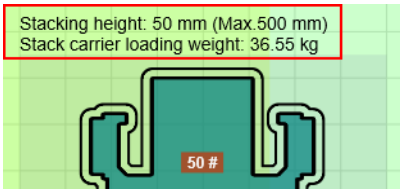
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Stack carrier information

Additional information can be displayed for each stack carrier in the special display.

Information above the stack carrier:

- The height of the stack and the maximum possible stack height on the stack carrier.
- The weight of all stacks on the stack carrier.



Information below the stack carrier:

- Additional information about each part from the **Order information** column.



Interactive buttons

When a part is selected in the stacking area, interactive buttons will be displayed. The buttons will remain visible as long as at least one part is selected.

Button	Description
	The number of parts in the stack is defined manually. Click the button to define the number of parts dynamically.
	The number of parts in the stack is defined dynamically. Click the button to define the number of parts manually.
	The parts are stacked with a small offset inside the broken line. To disable it, click on the button.
	The parts are stacked without an offset. Click the button to stack the parts with an offset.
	Adds new cutting plans to the sorting plan.
	Deletes the cutting plan from the sorting plan. To delete it, select the cutting plan and click on the button.



Note: If the **Always display interactive buttons** option is enabled in the Part Nester settings, the buttons will be displayed permanently *(See: ›Always display interactive buttons‹ Page 906)*.

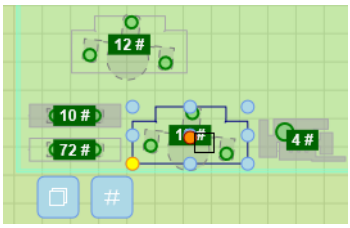


Fig. 10-108: Buttons displayed for a selected stack

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Overlapping stacking areas

If there are overlapping stacking areas on the sorting system, each stacking area has its own button (**A**). Click this button to activate the respective stacking area.

The activated stacking area is indicated by a black line (**B**) and the disabled stacking area is indicated by a broken line (**C**).

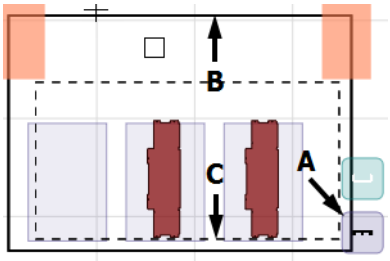


Fig. 10-109: Pallet stacking area

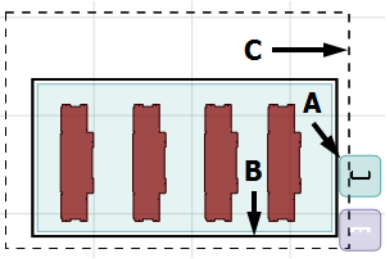


Fig. 10-110: Cassette stacking area (DUH ByTrans Cross)

☞ **Note:** Stacks cannot be moved between stacking areas directly. To move a stack to another stacking area, first delete it and then drag it to another stacking area on the **Parts** tab.

Button	Description
	Activates the cassette stacking area.
	Activates the pallet stacking area.

Overlapping stacks

If multiple stacks in parts groups are arranged on top of each other, you can highlight them with an additional border in the special display. To do so, select the **Display parts groups** option in the special display. The number of all parts in the group is displayed in the center of the parts group.

Hover the mouse over the group to display a pop-up dialog with information about the parts group. A maximum of 3 stacks can be displayed in the pop-up dialog.

Click once to move the entire group and twice to move a single stack out of the group. A single stack selection is indicated by a green dot.

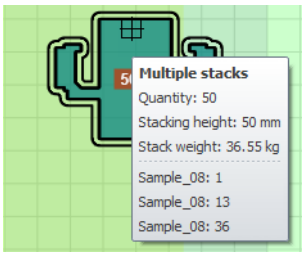


Fig. 10-111: Highlighted stack in a parts group

Functional groups

The function bar contains the following functional groups:

- Selection
- Job
- Automatic
- Parts
- Cutting plans
- Stacking site
- Stack
- View

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10.5.1 Group: Job

Where do I find the group?

Select: **Modules > Part Nester > Sort Editor**

This group contains all the functions for automatic nesting of the parts.

10.5.1.1 Sort job properties



How do I open this dialog box?

Select: **Modules > Part Nester > Sort Editor > Job**

This dialog is used to configure settings for displaying the stacking site, sorting unit, and part stack.

Stacking site configuration

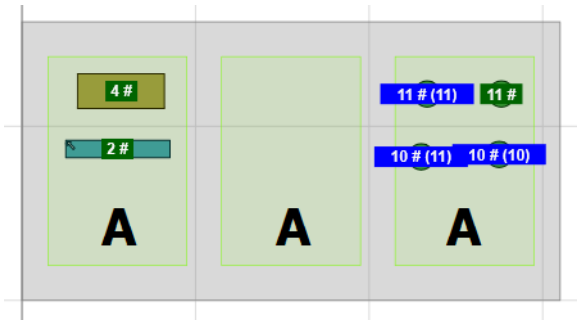
- **Show settings automatically**

  When switching stacking site definitions on the **Stacking site definition** tab, the appropriate settings are displayed in the **Stacking site definition settings** dialog. If this option is disabled, you must select the **Properties** function in the **Stacking site definition** group to display the stacking site definition.

Stack carrier colors

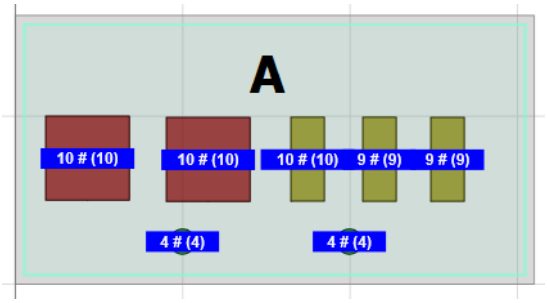
- **Pallets**

  Defines the color of the pallet places (**A**).



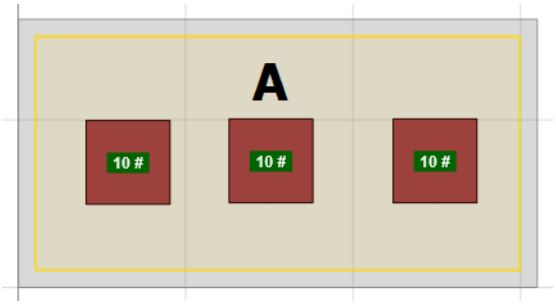
Cassettes

Defines the color of the cassette places (**A**).



Carriage

Defines the color of the carriage places (**A**).



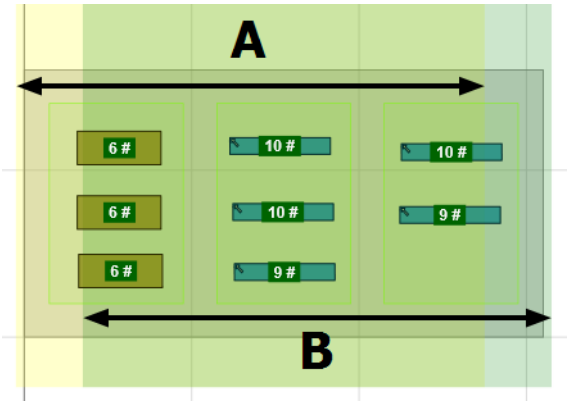
Sorting unit color

Head 1 range of travel

Defines the color of the range of travel of sorting head 1 (**A**)

Head 2 range of travel

Defines the color of the range of travel of sorting head 2 (**A**)



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Part stack handling

- **Mode**
  - **Max. possible stack height**
    The maximum possible stack height is used for the stacks. If the number of parts is less than the maximum possible stack height, only one part stack can be set.
  - **Balanced stack heights**
    All available instances of a part are distributed evenly among all stacks.
  - **Add defined part quantity**
    Only a certain number of parts per stack is stacked, taking maximum stack height into account.
  - **Define stacking site only**
    Only a stacking site is defined for the part. This is the default setting.

☞ **Note:** By default, the option **Stacking site only** should be used as this results in the fastest sorting time. The other options should only be used if you need a certain number of parts per stack.

- **Quantity to add**
  Defines the number of parts per stack when **Add defined part quantity** mode was selected.
- **Stack for part from all plans**
  If a part is manually set as a stack and its instances are spread out across different cutting plans, the instances from the various cutting plans are sorted onto this stack.

☞ **Note:** Maximum stack height is defined in the part properties and for the stack carrier in the **Sort** section in Administrator.

Sort Nester

- **Group by**
  - **No grouping**
    This is the default setting for Sort Nester.
  - **Order information**
    Stacks with the same order information are nested together.
  - **Info 1-3**
    Stacks with the same Info 1–3 are nested together.

Stacking information

The display settings for stacking information are defined in this section. The stacking information is displayed below the stacking site definition in the Sort Editor work-space.

- **Font size**
  Defines the maximum font size in pixels. Font size auto-adjusts up to this size when zooming in the workspace. At larger zoom levels it stays this size.
- **Field 1-3**
  Defines the contents of the display for the respective field. You can select columns **Order information** and **Info 1-3** on the **Parts** tab of the sort data. If the **Display nothing** attribute is selected, no stack information is displayed.

Sorting program

- **Allowance between parts**
  Defines the allowance per part to compensate for the difference in stack height due to burrs on the part and the gap between parts. The allowance corresponds to an empirical value and can be determined as follows using parts that have already been cut:

  Measured height of the part stack divided by the number of stacked parts minus part thickness.

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Sequencing strategy

The sequence strategy determines which automatically determined removal sequence is used to sort cutting plan parts. The automatic removal sequence is defined for the cutting plan in the **Sorting technology** menu *(See: ›Set automatically‹ Page 845)*.

Consecutive

Sorting of the cutting plan starts alternately from the cutting plan side closest to the stacking site to the cutting plan side furthest from the stacking site.



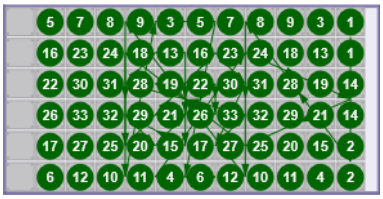
From inside

Sorting of the cutting plan starts in a spiral from the inside out.



From outside

Sorting of the cutting plan starts in an inward spiral from outside.



Save sorting information on export

Activate this option when connecting to a robot system to remove parts from stack carriers by robot.

The information is stored in a separate file that can be read by the robot system. It contains all information about the current stacking situation. By default, the file is saved in the ***SortInfos*** directory under the default path. The default path is defined in **System settings** -> **Data storage**.



Note: During export, a Save dialog appears in which you can copy the file name from the sort job or create a user-defined one.

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Cutting plan loading position

In some cases, especially with rotated machines, the cutting plans are not aligned with the origin of the cutting machine. The limits of the sheet stops may be on the other side of the shuttle table, which may result in a different position for sheets with smaller formats and affect the accessibility of parts for the sorting heads.

With these options it is possible to use a predefined orientation of cutting plans and a user-defined position of cutting plans in the order so as to improve accessibility of parts for the sorting heads.

- **Alignment**

  Defines the alignment of the sheet on the shuttle table. By default, **Bottom Left** is set. Changes to this setting are displayed directly in the machine layout preview.

  When **User-defined** is selected, the bottom two options are enabled. This allows you to set a user-defined sheet origin for the X and Y position.

  Example:



Fig. 10-112: Default setting, Bottom Left

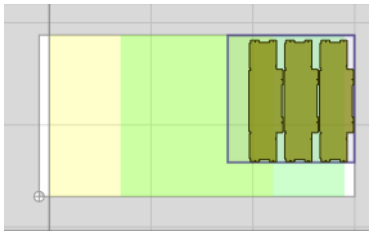


Fig. 10-113: Top Right setting

- **User-defined X position**

  Defines the sheet origin for the X position.

- **User-defined Y position**

  Defines the sheet origin for the Y position.

Gripper tool stations

These options define whether the tools in the gripper tool stations are assigned automatically or manually. Specific settings can be made for each gripper tool station.

- **Gripper tool station**

  This field is used to select the gripper tool station for the particular sorting head to which the settings below apply.

  The name of the gripper tool stations is defined in Administrator under **System data** -> **Sorting** -> **Sorting unit** -> **Manage tool station** *(See: ›Gripper tool stations‹ Page 173)*.

- **Automatic composition**
  - If the option is **enabled**, the program automatically assigns the sorting tools to the gripper tool station.
  - If the option is **disabled**, the tools can be assigned manually to the respective gripper tool station in the **Composition** section.

- **Composition**

  This section is used to assign the tools to the selected gripper tool station. You can only select as many tools as the selected gripper tool station has tool slots.

> **Note:** If no tool is selected for the gripper tool station when this option is enabled, the program automatically assigns the sorting tools to the gripper tool station.

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10.5.2 Menu: Sorting plan

Where do I find the group?

Select: **Modules > Sort Job Editor**

This group contains all the functions for automatic creation of stacking site definitions.

10.5.2.1 Automatic



Where do I find the function?

Select: **Modules > Sort Job Editor > Sort plan > Stacking site definition**

This function automatically creates stacks of all sortable parts in a stacking site definition.

If there the same parts are present in different cutting plans, a separate stack is created for each cutting plan. These stacks are arranged on top of each other in the stacking site definition. As a result, only the topmost stack is visible.

Drag and drop to arrange the stacks individually on the stacking site.



Note: Part stacks are automatically created for parts of cutting plans with the same stacking site definition.

10.5.3 Group: Cutting plans

Where do I find the group?

Select: **Modules > Part Nester > Sort Editor**

This group contains all the functions for editing cutting plans.

10.5.3.1 Split up cycles



Where do I find the function?

Select: **Modules > Part Nester > Sort Editor > Cutting plans**

This function splits the cycles of the selected cutting plan into two cutting plans with the cycles defined for each cutting plan.

Application:

The quantity and maximum height of the stack is not enough to stack all parts of the cutting plan on the stacking site definition. Splitting the cutting plan into two cutting plans makes it possible to stack on one stacking site definition.

Attention! If the copy of the cutting plan is not assigned to any other stacking site definition, the stacking site definition must be cleared after stacking the original stacking site.

Split up plan

- **Total cycles**
  Displays the total number of cycles of the selected cutting plan.

Original plan

- **Number of cycles**
  Defines the number of cycles of the original cutting plan. The maximum number of cycles is automatically entered in the field.
- **Plan name**
  Shows the original cutting plan name with a counter of 1. The cutting plan name can be customized.
- **Stacking site definition**
  Displays the stacking site definition on which the cutting plan is stacked. The comment field below displays information about stacking the cutting plan.

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Copy of plan

- **Active**

  Enables this option.

- **Number of cycles**

  Defines the number of cycles of the copied cutting plan. The necessary number of cycles is automatically entered in the field.

- **Plan name**

  Shows the name of the copied cutting plan with a counter of 1. The cutting plan name can be customized.

- **Add**

  Assigns a different stacking site definition to the copy of the cutting plan.

Copy of plan, stacking site definition, and stacks

- **Active**

  Enables this option.

- **Number of copies (w/o original)**

  Only used to display the number of copies.

- **Number of cycles per copy**

  Defines the number of cycles that can be stacked on a stacking site definition. By default, the maximum number of cycles is automatically set.

> ☞ **Note:** If the number of cycles is decreased, the number of copies increases accordingly.

10.5.4 Special view



How do I open this dialog box?

Select: **Modules > Part Nester > Sort Editor > View**

This function shows/hides the options selected under the text button. The options can be used to display additional views on the object. Various options are available for selection depending on the module.



Tip: Use this function to get a quick overview of the sorting information in the current cutting plan.

The following information can be displayed:

- **Display parts group:** *Displays groups of overlapping parts with an additional border. The number of all parts in the group is displayed in the center of the parts group.*
- **Stack carrier height:** *Shows the height of the stack and the maximum possible stack height on the stack carrier. If there are multiple stacks on a stack carrier, the height of the largest stack is shown.*
- **Stack carrier weight:** *Shows the weight of all stacks on the stack carrier.*
- **Additional stack carrier information:** *Shows additional information about each part from the **Order information** column. The sorting rule is also displayed for each stack carrier.*
- **Stack quantities:** *Displays the number of parts per stack.*
- **Gripper:** *Displays the gripper set on the part.*
- **Show parts of all plans:** *Displays the direction of machining of all parts in a cutting plan*
- **Machine layout:** *Displays the machine layout as a gray area.*
- **Cutting plan:** *Displays the selected cutting plan at loading position on the shuttle table.*
- **Pick-place zones:** *Displays the whole BySort sorting area on the system.*

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10.6 Menu: Export

Where do I find the menu?

Select: **Modules > Part Nester**

This menu contains all the functions for checking and exporting cutting plans.

The cutting plans are exported by a preprocessor, which converts the cutting plans into LCC, NCP, BVC or DXF files depending on the controller version of the machine.

Functional groups

The function bar contains the following functional groups:

- Selection
- Plans
- View
- Cost Calculator

10.6.1 Group: Cutting plans

Where do I find the group?

Select: **Modules > Part Nester > Export**

This group contains all the functions for exporting and testing cutting plans.

10.6.1.1 Automatic Export



Where do I find the function?

Select: **Modules > Part Nester > Export > Plans**

This function exports all the cutting plans in the job automatically to the output folder assigned to the machine. The output folder is displayed in the **Plan Export** tab of the **Export settings**.

The software runs a background check of the cutting plans prior to export. If any cutting plans are found to contain errors, these are displayed in the **Results of plan check** dialog box with a list of the errors. Select **Continue** if you still want to export the cutting plans or **Cancel** if you do not want to export the plans.



Note:

- If any residual sheets are defined on a cutting plan, the information about them is written to the NCP file.
- If plans with the same name have already been exported, a security prompt is displayed asking whether you wish to overwrite the existing plans.

Export selected plans

This sub-function is used to export to the output folder only those plans that are selected in the **Cutting plans** tab.

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10.6.1.2 Manual export



Where do I find the function?

Select: **Modules > Part Nester > Export > Plans**

This function exports all the cutting plans in the job to a user-defined output folder. Running the function opens a dialog box in which you must specify the output folder.

After confirming with **OK**, all cutting plans in the job are exported to this folder.

The software runs a background check of the cutting plans prior to export. If any cutting plans are found to contain errors, these are displayed in the **Results of plan check** dialog box with a list of the errors. Select **Continue** if you still want to export the cutting plans or **Cancel** if you do not want to export the plans.



Note:

- If any residual sheets are defined on a cutting plan, the information about them is written to the NCP file.
- If plans with the same name have already been exported, a security prompt is displayed asking whether you wish to overwrite the existing plans.

Export selected plans

This sub-function is used to export to the output folder only those plans that are selected in the **Cutting plans** tab.

10.6.1.3 Export selected plans

Where do I find the function?

Select: **Modules > Part Nester > Export > Plans > Export / Manual export**

This function is used to export only those plans that are selected in the Cutting plans tab. The function can be used for both automatic and manual exports.

10.6.1.4 Results of plan check

This dialog box displays any errors identified in the respective cutting plans prior to export.

- Select **Continue** if you still want to export the cutting plans.
- Select **Cancel** if you wish to terminate the export of the cutting plans.

This dialog box only appears if some of the parts in the job have not been nested yet, or if errors have been found in the cutting plans.

Note: Use the **Verify plan** function to check the cutting plans for possible errors before export.

Tip: To generate any missing machining sequences and skeleton cuts automatically prior to export, enable the relevant options in the **Extras** tab of the **Export settings**.

10.6.1.5 Create plan names



Where do I find the function?

Select: **Modules > Part Nester > Export > Plans**

This function automatically creates a cutting plan name for unnamed cutting plans. The names of the cutting plans are displayed in the **Name** column of the **Cutting plans** tab.

Tip: Use this function when cutting plans have been created manually and have not yet been assigned names.

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10.6.1.6 Change plan name



Where do I find the function?

Select: **Modules > Part Nester > Export > Plans**

This function changes the cutting plan name for all cutting plans in the job. Enter the new name for the cutting plans in the **Insert Text** box.

☞ **Note:** The names of the cutting plans are displayed in the **Name** column of the **Cutting plans** tab.

10.6.1.7 Verify plan



Where do I find the function?

Select: **Modules > Part Nester > Export > Plans**

This function checks the current plan against various criteria. The **Verify plan** docking dialog box displays the criteria used in the check.

☞ **Note:** Only those criteria are displayed that apply to the selected cutting plan.

Plan information

This area contains information about the current cutting plan and the parts contained in the cutting plan.

Verify all plans

Verifies all plans in the job and shows the **Status** on the **Cutting plans** tab *(See: ›Status of verification criteria‹ Page 1121)*.

Status for laser cutting machines

This area lists all the criteria that are used for checking the cutting plan for laser cutting machines. If you do not want certain criteria to be checked, you can disable them with the checkboxes on the right.

The following examples show the basic procedure to follow for cutting-plan parts that contain errors.

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Parts

A part referenced in the cutting plan no longer exists in the parts database.

- Import the part into the parts database again in Part Editor.
- Delete the part from the cutting plan.

Part release

The cluster contains parts that have not been released for production.

Machine compatibility

An attempt has been made to export a cutting plan that is larger than the cutting area of the assigned machine.

1. Remove all invalid sheet formats from the **Sheets** tab.
2. Use the **Add sheet** function to add a new sheet format size that is less than or equal to the maximum cutting area of the machine.
3. Re-nest the plans.

Parts exist

An attempt has been made to export an empty cutting plan.

- Add at least one part from the **Parts** tab to the cutting plan.
- Increase the quantity of parts in the **Parts** tab and then re-nest the cutting plan.
- Add at least one new part to the Parts tab for nesting in the cutting plan.

Part machining sequence

One or more parts do not have a machining sequence.

1. Select the **Cutting technology** menu.
2. Select the **Automatic** function from the **Machining sequence** group.
3. Select the **Automatic - Plan** button.

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Serial-number information

A cutting plan with serial-number information on parts is cut in several passes. If a counter is used, there will be several parts with the same number.

1. Disable the option **More than one machining cycle allowed per plan** in the **Cutting technology settings** dialog.
2. Nest the plan again.

Machining time

Machining time could not be calculated due to a missing cutting parameter or a change in geometry or cutting technology.

1. Assign a cutting parameter to the job in the **Machining time** menu.
2. Run the **Calculate** function.

Cutting parameter set

No cutting parameter is assigned to the job.

- Assign a cutting parameter to the job in the **Job Properties** dialog.

Grain direction

One or more parts are not nested in the grain direction.

- Nest the parts again.
- Remove the **Grain direction** option in the part's properties.

Parts with split-up cutouts

One or more parts are nested in a cutout.

- Move the part out of the area of the cutout or nest the plan again.

Overlapping parts

Contours of parts overlap or touch each other.

1. Select the part that overlaps another part.
2. Delete the part from the cutting plan or select the **Move part** function from the **Cutting technology** menu in order to shift the part into an empty place in the cutting plan.

Parts in edge region

Contours of parts lie within the edge region of a sheet.

1. Select the part that is lying in the sheet edge region.
2. Delete the part from the cutting plan or select the **Move part** function from the **Cutting technology** menu in order to move the part out of the sheet edge region.

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Position/size of texts

The programs checks whether the text in a textbox goes outside any part contours.

1. Select the part from the **Parts** tab in the Job data.
2. Select the **Edit** function.
3. Move the textbox in Part Editor so that it is inside the part contour.
4. Save the part.

Parts outside plan

Contours of parts lie outside the cutting plan.

1. Select the part that is lying outside the cutting plan.
2. Delete the part from the cutting plan or select the **Move part** function from the **Cutting technology** menu in order to shift the part to an empty place within the cutting plan.

Risk of collision

Cut contours are flagged with the **Collision risk** status if the nozzle passes over them again after cutting.

Only those contours are flagged as dangerous that are between the minimum and maximum X/Y dimension. The size range is defined in the Cutting machine dialog in Administrator *(See: ›Collision calculation‹ Page 108)*.

☞ **Note:** Only parts smaller than twice the distance between two cutting grates can cause a collision.

Common Cut segments

Found Common Cut nesting with overlapping segments.

- Re-abut Common Cut parts.
- Re-nest Common Cut parts.
- Re-nest overlapping segments with parts protective distance.

Common Cut parts

Found parts without Common Cut nesting.

- Re-abut Common Cut parts.
- Re-nest Common Cut parts.

Export setting – cutting wizard

The settings in the cutting plan do not match the Cutting Wizard.

- Edit the settings in the cutting plan.
- Choose a different Cutting Wizard.

Separation cuts when clamping enabled

Horizontal separation cuts are not permitted when clamping is enabled.

- Remove all horizontal separation cuts from the cutting plan.
- Disable the **Clamping active** option in the **Extras** tab of the export settings.

Sector separation cuts present

A separation cut is missing between sectors.

- Insert separation cuts between the sectors.

Clearance to sector separation cuts

There are parts too close to the separation cuts.

- Increase the clearance of the appropriate parts to the separation cuts.

Position of microjoints

One or more microjoints are in an invalid position.

- Place microjoints in a valid position on the part.

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Status for water jet cutting machines

This area lists all the criteria that are used for checking the cutting plan for water jet cutting machines. The following examples show the basic procedure to follow for cutting-plan parts that contain errors and are intended for cutting on water jet machines.

FixMaster overlap

The pierce exit point is too close to the FixMaster gluing point.

- Delete all the cutting plans and re-nest the job.
- In the Properties in the **Data** menu, assign a 4-head machine to the job or modify the cutting plan for your 2-head machine.

Multihead setting

An attempt was made to export a 4-head cutting plan to a 2-head machine.

- Delete all the cutting plans and re-nest the job.
- In the Properties in the **Data** menu, assign a 4-head machine to the job or modify the cutting plan for your 2-head machine.

Tool configuration

An attempt was made to export cutting plans having a vertical tool configuration for a machine with a horizontal tool configuration, or vice versa.

- Delete all the cutting plans concerned and re-nest the job.
- In the Properties in the **Data** menu, assign a machine with the correct tool configuration to the job.

Tool separation

The minimum or maximum tool separation in the cutting plan lies outside the valid range for the machine.

- Delete all the cutting plans concerned and re-nest the job.
- In the Properties in the **Data** menu, assign to the job a machine with the applicable minimum and maximum tool separations for the job.

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Reach of tool

The assigned tool cannot reach one or more parts.

- Delete all the cutting plans concerned and re-nest the job.
- Move the parts to a position in the cutting plan that can be machined by the tool.

Elements in the dialog

Links to verification criteria

If a criterion is associated with an error or warning, this is indicated by a red or yellow dot respectively. The text **select** and **edit** appears on the right beside the verification criterion.

- **select:** *Selects all elements or objects in the working area that are valid for the verification criterion.*
- **edit:** *Opens the appropriate edit function to remedy the problem.*

Status of verification criteria

-  The cutting plan is OK. The cutting plan can be exported.
-  The cutting plan contains errors. The cutting plan can be exported.
-  The cutting plan contains errors. The cutting plan cannot be exported.



Note: The overall status of the cutting plan is shown as a color icon under the verification criteria and in the workspace. Only highlighted verification criteria are applied for the status.

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10.6.1.8 Verify cluster



Where do I find the function?

Select: **Modules > Part Nester > Export > Plans**

This function checks the current cluster against various criteria. The **Verify Cluster** docking dialog box displays the criteria used in the check.

☞ **Note:** Only those criteria are displayed that apply to the selected cluster.

Plan information

This section contains information about the current cluster and the parts contained in the cluster.

Verify all plans

Verifies all plans in the job and shows the **Status** on the **Cluster** tab *(See: ›Status of verification criteria‹ Page 1125)*.

Status for laser cutting machines

This area lists all the criteria that are used for checking the cutting plan for laser cutting machines. If you do not want certain criteria to be checked, you can disable them with the checkboxes on the right.

The following examples show the basic procedure to follow for cutting-plan parts that contain errors.

Parts

A part referenced in the cutting plan no longer exists in the parts database.

- Import the part into the parts database again in Part Editor.
- Delete the part from the cutting plan.

Part release

The cluster contains parts that have not been released for production.

Parts exist

An attempt has been made to export an empty cutting plan.

- Add at least one part from the **Parts** tab to the cutting plan.
- Increase the quantity of parts in the **Parts** tab and then re-nest the cutting plan.
- Add at least one new part to the Parts tab for nesting in the cutting plan.

Part machining sequence

One or more parts do not have a machining sequence.

1. Select the **Cutting technology** menu.
2. Select the **Automatic** function from the **Machining sequence** group.
3. Select the **Automatic - Plan** button.

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Securing parts in the cluster with microjoints

There are parts without microjoints in the cluster.

- Secure parts sufficiently with microjoints to prevent them from falling out.

Parts overlapping with gripper zones

One or more parts are inside the gripper zone. The outline of the gripper zone is shown in color.

- Position the parts outside the gripper zone.

Overlapping parts

Contours of parts overlap or touch each other.

1. Select the part that overlaps another part.
2. Delete the part from the cutting plan or select the **Move part** function from the **Cutting technology** menu in order to shift the part into an empty place in the cutting plan.

Parts in edge region

Contours of parts lie within the edge region of a sheet.

1. Select the part that is lying in the sheet edge region.
2. Delete the part from the cutting plan or select the **Move part** function from the **Cutting technology** menu in order to move the part out of the sheet edge region.

Position/size of texts

The programs checks whether the text in a textbox goes outside any part contours.

1. Select the part from the **Parts** tab in the Job data.
2. Select the **Edit** function.
3. Move the textbox in Part Editor so that it is inside the part contour.
4. Save the part.

Parts outside plan

Contours of parts lie outside the cutting plan.

1. Select the part that is lying outside the cutting plan.
2. Delete the part from the cutting plan or select the **Move part** function from the **Cutting technology** menu in order to shift the part to an empty place within the cutting plan.

Risk of collision

Cut contours are flagged with the **Collision risk** status if the nozzle passes over them again after cutting.

Only those contours are flagged as dangerous that are between the minimum and maximum X/Y dimension. The size range is defined in the Cutting machine dialog in Administrator *(See: ›Collision calculation‹ Page 108)*.

☞ **Note:** Only parts smaller than twice the distance between two cutting grates can cause a collision.

Elements in the dialog

Links to verification criteria

If a criterion is associated with an error or warning, this is indicated by a red or yellow dot respectively. The text **select** and **edit** appears on the right beside the verification criterion.

- **select:** *Selects all elements or objects in the working area that are valid for the verification criterion.*
- **edit:** *Opens the appropriate edit function to remedy the problem.*

Status of verification criteria

- The cluster is OK. The cluster can be exported.
- The cluster contains errors. The cluster can be exported.
- The cluster contains errors. The cluster cannot be exported.

☞ **Note:** The overall status of the cluster is shown as a color icon under the verification criteria and in the workspace. Only highlighted verification criteria are applied for the status.

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10.6.1.9 Run simulator



Where do I find the function?

Select: **Modules > Part Nester > Export > Plans**

This function opens the cutting plan in the **Cutting Simulator** module. The simulator can be used to check the cutting processes of cutting plans prior to export.

10.6.2 Export settings



Where do I find the function?

Select: **Modules > Part Nester > Export > Plans**

In this dialog box you can define the settings for exporting cutting plans. These settings are applied for both automatic and manual cutting-plan exports.

Info

This section displays information about the selected machine, the cutting processor. and the output folder for cutting plans.

10.6.2.1 Preprocessor

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings**

This tab is where all the settings are made for the technology options in the cutting plans. Click on the circular icon in the header of the technology concerned to enable/disable the technology. The arrow buttons can be used to show/hide the technology options.

The settings in this tab are used only temporarily when exporting the cutting plans. The contours of the parts and their attributes are not changed.



Note: Technology options without an activation button are always used if they are present on the plan.



Tip: If you wish to reuse the preprocessor settings for other jobs, create a new job with the relevant settings for the preprocessor and save this job as a job template *(See: ›Save as template‹ Page 887)*.

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What each button does:

Icon	Meaning
/ Active	The technology option is enabled and will be applied to the cutting plans at export.
/ Active	The export setting is not enabled.
▼	Closes the technology option view.
▶	Opens the technology option view.

Cutting Wizard

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

This option applies the selected Cutting Wizard to all lead-ins and contours in the cutting plan that have the **Default** attribute defined for their operating mode and piercing mode. This does not affect the lead-in type or the lead-in position.

Name

The selected Cutting Wizard is displayed in the **Name** field. If there are several Cutting Wizards available, you can select another Cutting Wizard by clicking in the field.

☞ **Note:** When selecting a cutting wizard, the system remembers the cutting wizard that was last selected. If no cutting wizard is found by setting filters, and if you disable the filters again, this box displays the cutting wizard that was last selected.

Automatic

Enabling this box adopts the settings from Settings Manager. The filter definitions are hidden.

If no valid table is found, an error message is displayed to this effect.

Filter definitions

These filters can be used to restrict the choice of Cutting Wizards. The properties of the Cutting Wizard can be displayed and edited in the Administrator module in the **Cutting Wizard** form *(See: ›Cutting Wizard‹ Page 116)*.

If a number of Cutting Wizards are found that match the filter criteria, the user must manually select the data record that has the appropriate cutting gas.

☞ **Note:** The filter definitions is defined in the **Job properties** > **General** in the **Material** and **Cutting process** sections.

Machining mode

This option applies the settings in the **Cutting Wizard** area to the cutting plans.

- **Replace standard technology only**: Replaces the technology on all contours to which the **Default** operating mode is assigned. All other modes are not replaced.
- **Reset whole technology**: Replaces the technology on all contours.

Tip: Use this attribute to export parts in cutting plans that have been assigned a different machine or a different material in Part Editor.

Overwrite lead-ins

This option also overwrites the lead-ins on the contours based on the settings in the **Cutting Wizard** section in Administrator.

☞ **Note:** This setting is only active if machining mode is set to **Reset whole technology**.

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Corner mode

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

This area can be used to control the cutting of corners. The settings made in these fields determine whether corners are cut using a looping function or, above a certain value, with a radius.

Even when cutting corners with small radii the machine moves under dynamic control; it no longer decelerates to a standstill or accelerates out of a standstill.



Note: With dynamic machining of corners you speed up machining and reduce the wear on the axes.

Advantages of radii:

- The axes move dynamically
- Smaller heat affected zone
- Less burring

Minimum radii as a function of the material, assist gas and thickness

Material	Mild steel			
Assist gas	Oxygen			
Thickness (mm)	1-5	6-10	12-15	16-20
Minimum rounding-off radius (mm)	0.8	1	1.5	3

Material	Stainless steel			Aluminum		
Assist gas	Nitrogen					
Thickness (mm)	1-3	4-6	8-15	1-3	4-5	6-8
Minimum rounding-off radius (mm)	0.8	1	2	0.8	1	2

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Selection mode

Use this option to select the corner mode. There are various attributes available for cutting corners.

- **Use selection:** The specified corner mode is only applied to those corners whose contours have the **Round off** or **Looping** tag All other corners are cut normally *(See: ›Tag‹ Page 684)*.
- **Round off:** Cuts those corners that lie in the range 5°- 170° with the radius specified here, provided they have no radius in the cutting plan.
- **Looping:** Cuts those corners that lie in the range 15°- 135° with a loop, provided they have no radius in the cutting plan.

Note: If Looping is enabled, corners in inner contours that have no radius are also cut with a loop. To avoid this, assign the inner contour corners a radius of half the kerf width *(See: › Finding the tool radius‹ Page 1360)*.

Tip: This option is particularly useful for very sharp corners such as saw blades.

Use values from Cutting Wizard

If this option is selected, the values for cutting corners are used from the Cutting Wizard. If this option is disabled, the values indicated here are used to cut corners.

Note: This option can only be used with the Cutting Wizard. If a Cutting Wizard is selected, the name of the selected Cutting Wizard is displayed next to the selection box.

Rounding radius for outside corners / inside corners

These options are used for specifying the rounding radii for outside corners (**A**) and inside corners (**B**).

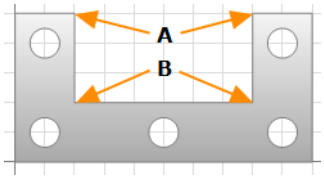


Fig. 10-114: Rounding radii for outside and inside corners

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Looping length

This option specifies the extent of the loop (**L**) from the corner of the contour.

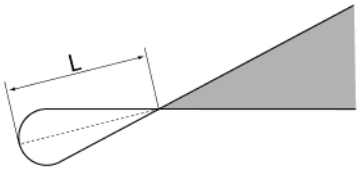


Fig. 10-115: Looping length when cutting sharp corners

Pre-piercing

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this area you can set your preferences for pre-piercing at the piercing points on the parts contours. Pre-piercing can be applied per part or per plan. When the parts are subsequently cut, piercing is performed directly without piercing time.

Note:
Pre-piercing increases the machining time by approximately twice the traversing time for the cutting plan.

If this option is enabled on cutting plans for **water jet cutting**, all pierces are **pre-drilled** by default.

Usage for laser cutting

Select this technology for parts containing numerous holes or when producing many small parts, and in steel of sheet thickness greater than 12mm.

- Pre-piercing helps to distribute more evenly the heat that is generated when cutting numerous small holes or parts.

Usage for water jet cutting

- When machining glass, pre-piercing avoids the frequent pressure change between piercing and cutting, helping to protect the pressure pipelines.
- Pilot drilling prevents material being washed out during piercing.

Machining mode

These attributes are used to define the machining process for pre-piercing.

- **Per part**

  All contours of a part are first pre-pierced and then cut, in both cases according to the set machining sequence.

- **Per plan**

  The contours of all the parts in the cutting plan are first pre-pierced and then cut, in both cases according to the set machining sequence.

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Selection mode

These attributes can be used to select the contours for pre-piercing.

- **Selected contours**

  Only those contours are pre-pierced that have the **Pre-piercing** tag *(See: ›Tag‹ Page 684)*.

- **All contours**

  All contours in the cutting plan are pre-pierced.

Tools

These attributes can be used on water jet cutting machines to select the tools to be used for pre-piercing or pilot drilling.

- **Active tools**

  All active tools are used for pre-piercing or pilot drilling.

- **Tool 1-4**

  Only the selected tool is used for pre-piercing or pilot drilling.

Travel path

This attribute specifies the distance between two points between which the machine will travel during piercing. A default distance of **0.05mm** is used.

Optimized positioning path

The machining sequence for pre-piercing is re-computed to give the shortest overall distance for all traversing paths together. On water jet cutting machines, optimization can only be performed amongst parts with the same carriage separation.

Pre-piercing instead of predrill

All pierces are pre-drilled with the water jet.

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Scanning

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this area you can set your preferences for scanning contours of parts. Scanning can be used to process rectangular or square contours (**A**), diamond-shaped contours (**B**), and contours of circular ring cut-outs (**C**).

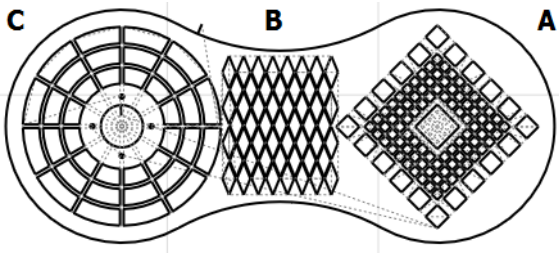


Fig. 10-116: Contours that can be processed with scanning

Note: In scanning mode, piercing takes place directly on the contour. If this is not permissible, scanning cannot be used. With scanning, only circular ring cut-outs that have the same circle center point can be cut.

Application:

- The application range includes sheets up to 2mm thick.
- Ideal for repeating rectangular or circular hole templates (e.g. for covers, cooling slits, mountings for filter pads).

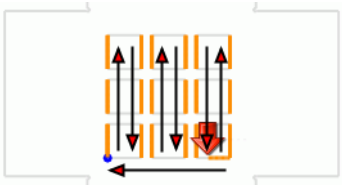


Fig. 10-117: Vertical and horizontal cutting process in scanning mode

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Selection mode

These attributes define the contours for the scanning machining process.

- **Marked contours**

  Only those contours are used for scanning for which the **Scanning** tag has previously been set in the Part Editor *(See: ›Tag‹ Page 684)*.

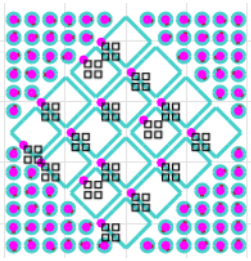


Fig. 10-118: Squares marked with the **Scanning** tag

- **All contours**

  All contours are used for scanning which meet the criteria for **Contour spacing** and **Number of contours**.

Max. contour separation

This option specifies the maximum contour separation rectangles can have and still belong to a scanning group.

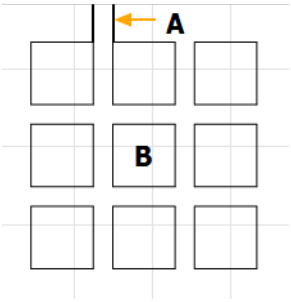


Fig. 10-119: The contour separation of rectangles (A) which belong to a scanning group (B)

Min. number of contours

This option specifies the minimum number of rectangles which must be in a group so that they can still be cut using scanning. The rectangles for this case may also be in a row.

Start angle

This option defines the direction in which the program starts cutting scanning groups at this angle.

- A value of **0°** cuts all scanning contours horizontally first.
- A value of **90°** cuts all scanning contours vertically first.



Note: For rotated contours, you can specify the appropriate angle of rotation as the start angle to define the start direction.

Cutting waste into smaller sizes

With this option, cut-outs from contours are automatically cut into smaller sizes by separating cuts made using scanning. Separating cuts can be created running horizontally or vertically.



Note: The function which cuts waste into smaller sizes only works with rectangular parts, however, **not** with circular ring cut-outs which are cut using scanning. The separating cuts are made until they reach the contour.

Minimum contour length

This option specifies the minimum contour length for which a separating cut is generated. If it is possible to generate several separating cuts in a cut-out, they will be distributed uniformly along the horizontal or vertical contour length (**A**).

Example:

For a minimum edge length of 50mm and a contour length (**L**) of 100mm, two separating cuts (**A**) are set in the cut-out of the contour (100/50=2).

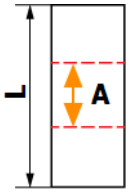


Fig. 10-120: Separating cut in an inner contour cut with scanning

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Laser/Water operating mode

This option is used to specify for all scanning cuts which operating mode is to be used to machine these cuts. If the **Default** operating mode is selected, the operating mode for the scanning contour of the part is used.

Laser/Water piercing mode

This option is used to specify the piercing mode for all scanning cuts. If the **Default** piercing mode is selected, the piercing mode for the scanning contour of the part is used.

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Vaporization

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this area you can set your preferences for film vaporization prior to the actual cutting of a sheet. Before cutting a film-coated sheet, the film must be removed by vaporization from all the contours or just from the lead-ins, depending on the quality of the film.

Contours can be cut using optimized cutting parameters for film-coated sheets. These parameters are identified by the words «plastic foil» appended to the parameter file name.

Example:

SPEED3015_8103_5200_DC01_2_1_N2_**plastic foil**.par

Note: Center-punch marks are not included in vaporization.

Selection mode

This option defines which contours of a part are vaporized.

Complete part

The film is vaporized from all contours of the parts prior to cutting. Machining is performed in two steps.

1. Vaporization of the film from all contours of the part.
2. Cutting of the contours of the part.

Tip: This attribute is particularly suitable for cutting film-coated sheets from which the film tends to peel off during cutting. This attribute can also be used for removing unwanted coatings, such as rust or paint, from the sheet before cutting.

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- **Lead-in only**

  The film is vaporized only from the lead-ins of parts before cutting. The contours are cut without prior vaporization.

  > **Tip:** This attribute is particularly suitable for cutting film-coated sheets to which the film is firmly bonded, allowing the sheet to be cut in one step.

- **Selected contours**

  Only those contours of a part are vaporized that have the **Vaporization** tag set *(See: ›Tag‹ Page 684)*.

Laser/Water operating mode

This option is used to specify the operating mode for vaporizing the contours. Depending on the controller version of the machine, the **Engrave** or **Process macro 5** operating mode is used.

Operating mode for CNC Version P8104 onwards.

The **Process macro 5** operating mode is used for vaporization.

Operating mode for CNC versions earlier than P8104 and CNC96

The **Engrave** operating mode is used for vaporization.

Cutting waste into smaller sizes

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this area you can set your preferences for cutting up the scrap from inner contours into smaller pieces. The settings apply to all contours for which the **Scrap cut-down** tag has been set in Part Editor *(See: ›Tag‹ Page 684)*.

When scrap cut-down is applied, a grid of separation cuts is placed within the cut-out prior to making the cut-out. The grid is always oriented in the X and Y-directions, irrespective of the orientation of the cut-out. The cut up material can then drop easily through the support grating.

Note: Using this technology increases the machining time significantly, so it is important to check the machining time.

Tip: Use these parameters to increase process reliability when cutting inner contours that might cause a collision if cut parts protrude from the sheet.

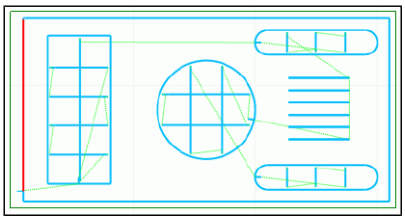


Fig. 10-121: Cutting waste from inner contours into smaller pieces

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Distance from contours

The grid cuts (**B**) are cut up to this distance (**A**) away from the inner contour of the parts.

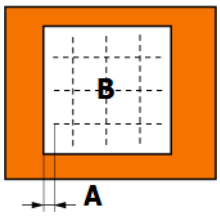


Fig. 10-122: Distance of grid cuts from the contour

Grid spacing X/Y

These options specify the pitch of the grid divisions in the X-direction (**A**) and Y-direction (**B**).

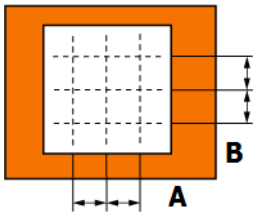


Fig. 10-123: Grid spacing in the X- and Y-direction.

Minimum cut length

This option sets the minimum length of cut, at and above which a grid cut is set within an inner contour. Grid cuts that are shorter than this length (**A**) are no longer set.

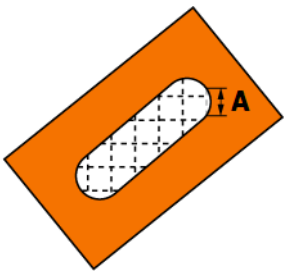


Fig. 10-124: Grid cuts in a slot lying at an angle.

Length of lead-in

This option specifies the length of the lead-in (**A**) on a separating cut.

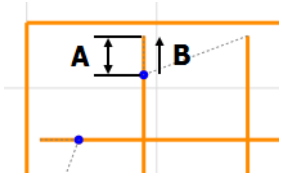


Fig. 10-125: Length of lead-in on separating cut

Distance from contours parallel

This option defines the minimum distance between a separating cut and the contour. If a separating cut is closer than this setting, it is performed at this distance from the contour (**A**) or deleted.

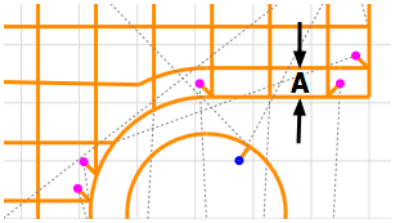


Fig. 10-126: Distance of separating cut from contour

Rotation allowed

This option allows rotation of the grid (**A**) if the part is rotated through a certain angle during nesting. If this option is disabled, the grid (**B**) remains at right angles to the support grate (**C**).

Attention! The option may lead to problems with waste parts falling out as the cuts do not run at right angles to the support grate.

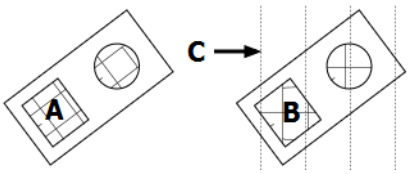


Fig. 10-127: Rotated grid in inside contours

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Sequence

This option determines which separation cuts to cut first. The reference point for separation cuts is always the origin of the part (**A**). It does not matter how the part is nested in the cutting plan.

- **Horizontal first**

  Cuts the horizontal separation cuts of the inside contour first.

- **Vertical first**

  Cuts the horizontal separation cuts of the inside contour first.

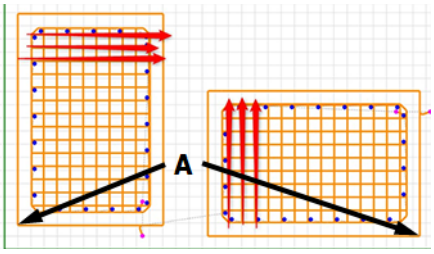


Fig. 10-128: Horizontal separation cuts on a normal and a rotated part

Lead-in type

This option determines the lead-in type cuts that cut waste into smaller sizes. This allows you to prevent damage to the cutting part caused by excessive heat input while cutting the lead-in.

- **Hook**

  This lead-in type is used by default. Only one pierce is performed per cut. The contour (**B**) may, however, get damaged by excessive heat input on the tip of the cutting contour (**A**).

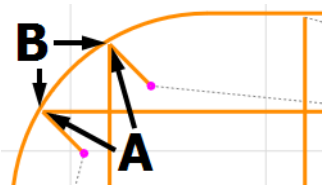


Fig. 10-129: Hook lead-in for cuts for cutting waste into smaller sizes

- **Direct**

  This lead-in type pierces directly on the contour of the cut for cutting up waste (**A**) and cuts up to just before the contour (**B**). From here, the machine moves back to the pierce without a laser beam and continues to cut from this position.



Tip: This lead-in type causes less input on the contour of the part and should be preferred when cutting thin sheets.

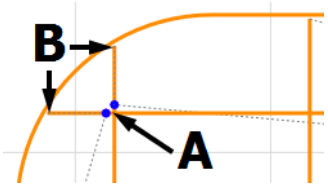


Fig. 10-130: Direct lead-in for cuts for cutting waste into smaller sizes

Laser/Water operating mode

This option is used to specify the operating mode for all scrap cut-down cuts.

The operating mode depends on the type and thickness of material used. If the Cutting wizard is enabled and the **Default** operating mode selected, the **Cutting** operating mode is used.

Laser/Water piercing mode

This option is used to specify the piercing mode for all scrap cut-down cuts.

The piercing mode depends on the type and thickness of material used. If the Cutting wizard is enabled and the **Default** piercing mode selected, the **Normal** operating mode is used.

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Raise cutting head

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this section you can define the options for raising the cutting head when moving over contours that have already been cut.

Raising the cutting head reduces the risk of a collision with protruding parts after they have been cut.

☞ **Note:** Positioning paths with raised cutting head are displayed **blue** in the simulator.

Contours filter

Raising the cutting head is evaluated and applied separately for the minimum and maximum size in the X and Y direction. The collision calculation settings are in the Administrator **Cutting machine** form *(See: ›Collision calculation‹ Page 108)*.

Operating mode filter

This option specifies in which operating modes cutting-head raising is applied.

☞ **Note:** If no operating mode is selected, the cutting head is not raised.

Contours with microjoints

If this option is enabled, the cutting head is also raised over parts with microjoints.

Outside contour direction of rotation

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this section you can define the options for the direction of machining the outside contour of parts. This function first cuts the contour with the least material on its outside.

This option reduces warping due to thermal expansion and thus improves part accuracy. The dimensions include the lead-in.

☞ **Note:** The function is also used in repositioning plans.

Thermal expansion when changing the direction of machining a part



Fig. 10-131: Thermal expansion (**A**) warps the part (**B**)



Fig. 10-132: Thermal expansion (**A**) warps the edge of the sheet (**B**)

☞ **Note:** The function is only performed if all specified criteria are met.

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Ratio X/Y

This option defines the ratio of length to width of the bounding rectangle of a part.

Example:

- Length = **1000**
- Width = **500**
- Ratio X/Y = **2**

The function is performed if the ratio **≥** the configured setting.

Delta Y

This option is used to calculate the direction of rotation and should not be changed.

Min. length of longer side

This option is used to set the minimum side length as of which to apply the function.

Point spacing

This option is used to calculate the direction of rotation and should not be changed.

Alignment

This option defines which part orientation to include. In the following example, those parts are listed for which the direction of rotation is adjusted so that the contour at the edge of the sheet is cut first.

- **X direction**: 2
- **Y direction**: 3
- **X and Y direction**: 2 and 3

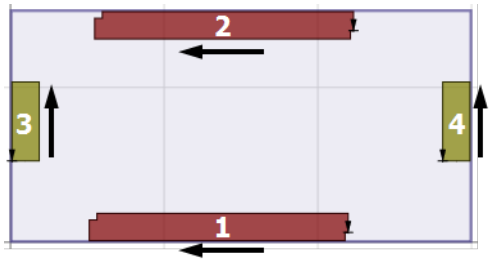


Fig. 10-133: Original direction of rotation, Part Nester

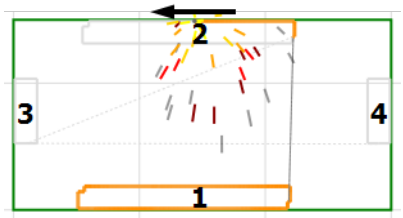


Fig. 10-134: Changed direction of rotation in the cutting plan of part 2

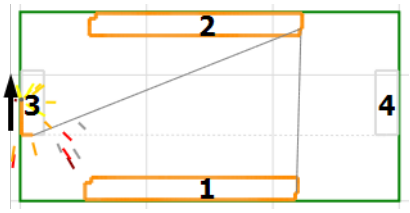


Fig. 10-135: Changed direction of rotation in the cutting plan of part 3

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Sheet edge inscription

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

The settings for automatically inscribing the sheet edge are configured in this section. The inscription is engraved.
The sheet edge inscription along with the cutting plan name allows the operator to correctly identify the cutting plan, for example.



Attention! In order to be able to engrave the sheet edge inscription on the sheet edge, the sheet edge must be at least **9mm**.

- **Text**

  In this field you can enter or re-edit the text. The added data fields can also be entered in this field. These data fields are defined as code and therefore must not be changed.

- **Data field**

  This option inserts predefined data fields in the text input area.

  Note: You can place text and data fields together in any combination. Any change to a text is displayed in the workspace as a preview.

- **max. font size**

  This option defines the maximum font size with which to engrave the text on the sheet. Maximum text size depends on the size of the sheet edge. If the value is **< 3mm**, no sheet edge inscription is created.

- **Position**

  Defines the corner(s) of a sheet in which to engrave a sheet edge inscription. Multiple corners may be selected.

Text

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

This section can be used to configure how texts on parts should be engraved.

Output separately

This option enables separate text output on parts.

Machining mode

These attributes are used to define the machining mode for texts on parts.

- **Per part**

  The text will be engraved on the first part and the part will then be cut, then the text will be engraved on the second part (if any) and the part will then be cut, and so on, all while following the machining sequence.

- **Per plan**

  The texts for all the parts in the cutting plan will first be engraved and all the parts will then be cut, all while following the machining sequence.

Interval

This option defines the parts on which text will be engraved.

A value of 1 means that the text will be engraved on every part. A value of 2 means that the text will be engraved on every other part (etc.).

Center punch mark

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this area you can set your preferences for making center-punch marks. The settings apply to all center-punch marks that are set on a contour as a tag in Part Editor *(See: ›Tag‹ Page 684)*.

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Center-punch shape

This option defines the form of mark made by the center punch.

- **Point**

  The center-punch mark is made as a small dot. The dot is displayed in the simulator as a dash. The dash is only visible in the simulator at high zoom levels. Select this attribute, for example, to cut a small hole. In order to cut the center-punch mark, the **Cutting** operating mode must be assigned to the contour in Part Editor *(See: ›Contour‹ Page 682)*.

- **Circle**

  The center-punch mark is made as a circle. Select this attribute, for instance, to mark drill holes in steel. In order to engrave the center-punch mark, the **Engraving** operating mode must be assigned to the contour in Part Editor *(See: ›Contour‹ Page 682)*.

- **Cross**

  The center-punch mark is made as a cross. Select this attribute, for instance, to mark drill holes in stainless steel. In order to engrave the center-punch mark, the **Engraving** operating mode must be assigned to the contour in Part Editor *(See: ›Contour‹ Page 682)*.

Note: To check the form of the center-punch mark on a contour, you need to open the job in the **simulator**.

Center-punch size

This option lets you specify the size of the cross or the diameter of the circle of a center-punch mark, depending on which form you have selected. This value does not affect the **Point** center-punch form.

Common Cut

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this area you define the options that are used for cutting common cut edges in the cutting plan *(See: ›Creating Common Cut cutting plans‹ Page 1360)*.

☞ **Note:** If a part has microjoints on a contour that is being used as a Common Cut, these microjoints are taken into account in the common cut.

Group-wise mode

Optimized cut for parts pair

This option sets the lead-in (**A**) on a straight line for the entire cut (**B**). This allows you to cut the common cut and the outer contour without an interruption. The length of the lead-in is defined by the **Length of overcut** option.

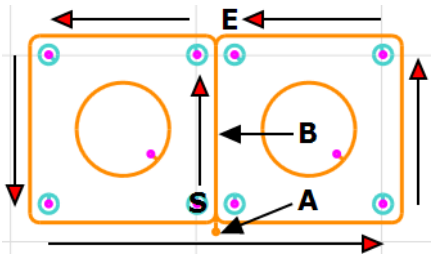


Fig. 10-136: Cutting sequence for optimized cut for parts pair

S	Start of common cut with outer contour
E	End of common cut with outer contour

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Length of overcut

The common cut (**A**) is cut beyond the outer contour (**C**) by the length of overcut (**B**). The common cut is extended by this distance at the start and end of the cut. This avoids any impairment to the outer contour from piercing at the start and end of common cuts. A value of 1 mm is sufficient in most cases.

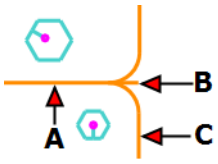


Fig. 10-137: Length of overcut for a common cut

Max. bridging distance

The purpose of this option is to avoid switching the laser off and back on again when the distances between two common cuts are short.

Common cuts (**B**) can lie up to a certain distance apart (**A**) for them to be combined into a common cut (**C**). The cuts must lie on the same axis. When cutting the common cuts, the cutting head re-traverses (**C**) the already cut inner contour (**D**) with cutting power on.

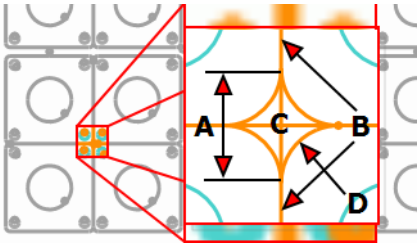


Fig. 10-138: Bridging distance between common cuts

Attention! Set the maximum bridging distance to be no greater than the outer diameter of the cutting nozzle. Otherwise a descent of the cutting head into a pre-cut inner contour (**D**) can cause a collision.

Sensing OFF for bridging distance

This option disables sensing if the cutting head moves over a common cutout (**C**) *(See: ›Max. bridging distance‹ Page 1154)*.

Length of lead-in for outer contour

This option specifies the length of the lead-in (**A**) of the outer contour. All lead-ins on the combined outer contours are removed (**B**).

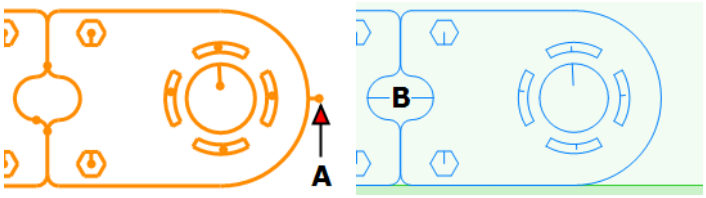


Fig. 10-139: Lead-ins on outer contours

Min. microjoint distance

This option defines the minimum distance between two microjoints on a common cut. If the distance between two microjoints is less than the value specified here, the second microjoint is deleted.

Application:

If one or more microjoints are set on the contour of a CommonCut cut of two parts.

Sort inside contours

This option attempts to position the inside contour machining sequence on connected Common Cut parts so as to achieve the shortest positioning path.

Attention! This option must not be enabled when using the **Scanning** machining mode, otherwise the machining sequence will be changed for scanning.

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Partial mode

Pulsed piercing

When this option is enabled, the machine will use piercing with individual pulses instead of with a continuous laser output. Pulsed piercing produces smaller piercing holes than piercing with a continuous laser output. However, the piercing time is longer than with a continuous laser output.

Piercing without time

If this option is enabled, the contour will be cut immediately after piercing. There will be no waiting time before cutting.

Tip: When using common cut plans, use this option as the default setting.

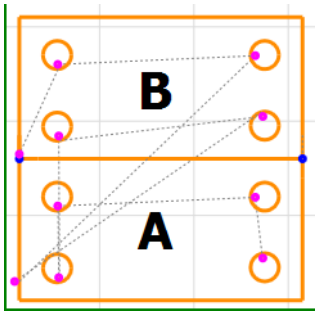
Note: This option is not suitable for plans for which large thermal stresses should be expected. Thermal stresses can be produced when cutting small contours in thick sheets, for example. The sheet can warp so heavily due to these thermal stresses that the new piercing cut may not match the existing safety cut.

Safety distance

This option is used to define the length of the safety cut made in an adjacent part. With this cut, the outer contour – and possibly the inside contours – of the adjacent parts in a common cut will be cut to a length that ensures that the cutting head will not collide with the current part later on when cutting the adjacent parts.

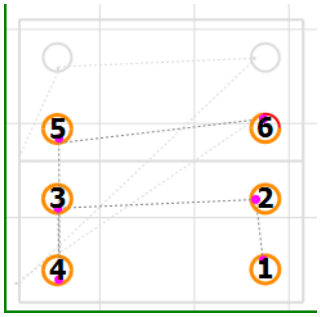
Cutting sequence for two common cut parts with a safety distance

- Initial situation



Part 1 (**A**) and part 2 (**B**) will be cut with a shared common cut.

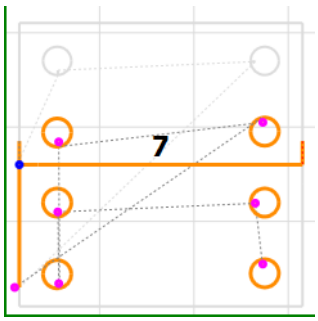
- Cutting sequence step 1



Holes **1** through **4** in part 1 are cut in advance.

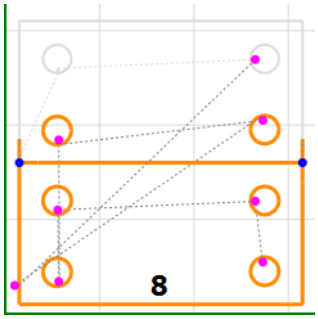
Holes **5** and **6** in part 2 are cut in advance (only if the safety distance extends beyond the distance of the next part's inside contour in the X/Y-axis direction).

- Cutting sequence step 2



The safety cut (**7**) is cut in part 2.

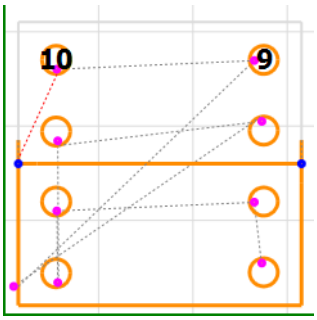
- Cutting sequence step 3



The contour (**8**) for part 1 is cut completely.

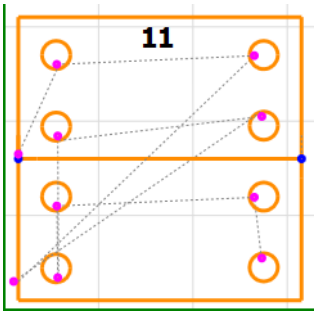
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- Cutting sequence step 4



Holes (**9**) and (**10**) in part 2 are cut.

- Cutting sequence step 5



The contour for part 2 is cut completely.

Laser/Water operating mode

This option is used to specify for all common cuts the operating mode to be used to machine the common cuts. If the **Default** operating mode is selected, the operating mode for the outer contour of the common cut part is used.

Laser/Water piercing mode

This option is used to specify the piercing mode for all common cuts. If the **Default** piercing mode is selected, the piercing mode for the outer contour of the common cut part is used.

Microjoint

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this area you can set your preferences for making microjoints with a lead-in and lead-out. The settings apply to all the microjoints that are set on the outer contour of a part using the **Set microjoint** function in the **Cutting technology** menu of the Part Editor.

Note: This option is only applied to outer contours of a part and results in a slightly longer machining time. How much longer it takes to machine the parts depends on the number of parts in the cutting plans.

Machining mode

These attributes are used to define the machining process for microjoints.

- **Execute on machine**

  Microjoints are defined by the parameter on the machine.

- **Separate contours**

  All microjoints on the outer contour of a part will be produced with a lead-in and lead-out. In addition, you can specify the length of the lead-in (**B**) and the lead-out (**A**), as well as the width of the micro joint (**C**). The break in the contour means that the lead-out lies at the start of the microjoint and the lead-in at the end of the microjoint.

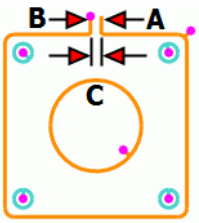


Fig. 10-140: View of microjoints with lead-in and lead-out in the Simulator.

Application:

This technology is particularly suitable for water jet cutting. Piercing is no longer performed on the contour so there is practically no damage to the contour, which would otherwise be the case with water jet cutting.

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What each field does:

These options are only active for the **Separate contour** machining mode.

- **Length of lead-in**
  Specifies the length of the lead-in of a microjoint.
- **Length of lead-out**
  Specifies the length of the lead-out of a microjoint.
- **Microjoint width**
  Specifies the width of a microjoint.

Set on texts

This option places a microjoint on the end cut of all cut text contours.

Skeleton cuts

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this area you can set your preferences for cutting skeleton cuts. If this option is enabled, all skeleton cuts set on the sheet are applied to the cutting plan.

☞ **Note:** This option is enabled by default for new jobs.

Machining mode

This option defines the order in which the skeleton cuts are made with respect to the parts.

- **Before parts**
  The skeleton cuts are made before the parts.
- **After parts**
  The skeleton cuts are made after the parts.
- **Before the parts (with residual sheet cuts)**
  The skeleton cuts and residual sheet cuts are done before the parts.

  Application:

- For simulating residual sheet cuts and skeleton cuts.
- For tilt optimization so that after cutting the parts, the head does not have to reposition over the parts to perform the residual sheet cuts.



Note: The default selection in **Before the parts**.

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Sequence

This option determines which skeleton cuts to cut first, depending on their orientation.

- **Horizontal and vertical**

  Starts cutting the residual sheet closest to the sheet origin (**A**) (**1**). At the end of each residual sheet cut, cutting continues with the nearest residual sheet (**2**).

- **Horizontal first**

  Starts cutting the horizontal residual sheet closest to the sheet origin (**A**) (**1**). At the end of each residual sheet cut, cutting continues with the nearest horizontal residual sheet (**2**).

- **Vertical first**

  Starts cutting the vertical residual sheet closest to the sheet origin (**A**) (**3**). At the end of each residual sheet cut, cutting continues with the nearest vertical residual sheet (**4**).

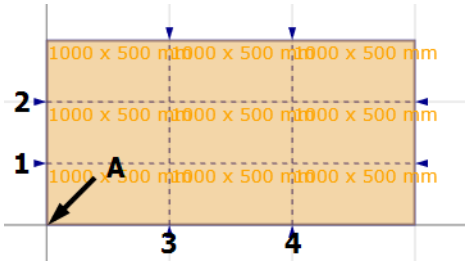


Fig. 10-141: Cutting sequence for skeleton cuts

Distance from contours

The skeleton cuts (**B**) are cut up to this distance (**A**) away from the contours of the parts.

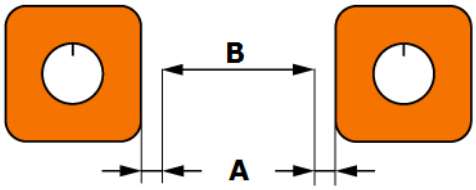


Fig. 10-142: Distance of skeleton cuts from the contour

Distance from sheet edge without detection

From this distance (**A**) onwards, detection is switched off for safety reasons for cutting beyond the sheet edge (**B**).

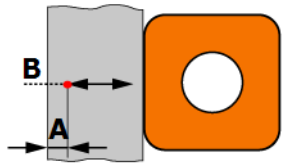


Fig. 10-143: Distance from the sheet edge at which detection is switched off

Length of overcut

Specifies the distance (**A**) beyond the sheet edge (**B**) for which cutting is performed. This distance can be defined for top, bottom, left and right sheet edge.

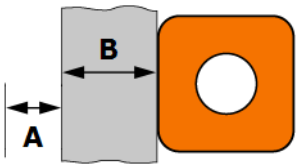


Fig. 10-144: Distance outside the sheet

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Take inner contours into account

If this option is enabled, skeleton cuts are also performed in inner contours (**A**). Parts nested inside inner contours are also taken into account.

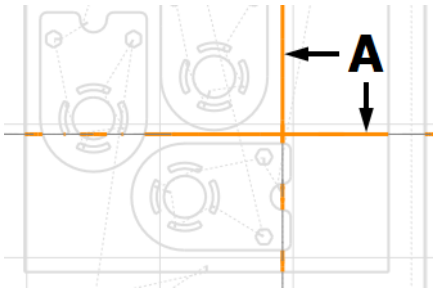


Fig. 10-145: Skeleton cuts inside inner contours

Number of microjoints

This option defines the number of microjoints on a skeleton cut. The microjoints are distributed evenly over the entire length of the skeleton cut.

Attention! This option must not be used together with the **Microjoints** option, otherwise no microjoints will be placed on the skeleton cuts separated using this option.

Laser/Water operating mode

This option is used to specify the operating mode for all skeleton sheet cuts.

The operating mode depends on the type and thickness of material used. If the Cutting wizard is enabled and the **Default** operating mode selected, the **Cutting** operating mode is used.

Laser/Water piercing mode

This option is used to specify the piercing mode for all skeleton sheet cuts.

The piercing mode depends on the type and thickness of material used. If the Cutting wizard is enabled and the **Default** piercing mode selected, the **Normal** operating mode is used.

Microjoints

If this option is enabled, when vertical and horizontal residual grid sections intersect, each skeleton cut that was cut first is separated at the intersection (**A**). This prevents sheets from standing up during cutting.

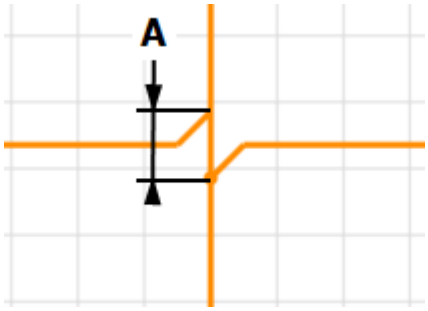


Fig. 10-146: Separated horizontal skeleton cut

Attention! This option must not be used together with the **Number of microjoints** option, otherwise no microjoints will be placed on the skeleton cuts separated using this option.

Microjoint width

This option defines the width of the microjoint (**A**). This option can only be set if the **Microjoints** option is enabled.

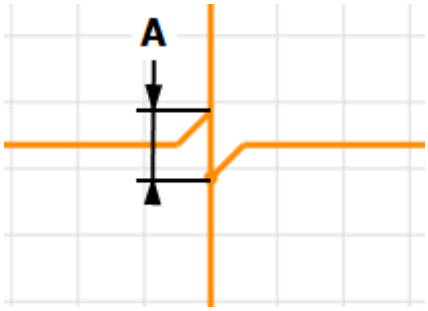


Fig. 10-147: Width of the separated horizontal skeleton cut as microjoint

Attention! The value must not be **0**, otherwise a warning message will be displayed.

Residual sheet serial number

This option sets a serial number on each residual sheet.

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Reposition

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this section you can set repositioning options for machines with multiple cutting areas. If repositioning is enabled, the cutting plan is divided into up to three cutting areas *(See: ›Cutting plans for repositioning machines‹ Page 1370)*.

Alternate

This option divides the cutting areas into different height section for repositioning. The height sections are machined in turn when repositioning.

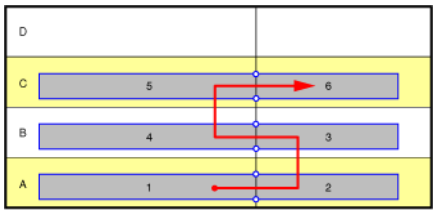


Fig. 10-148: Alternating machining sequence for a repositioning plan.

A	Height level 1
B	Height level 2
C	Height level 3
D	Height level 4

Note: In order to ensure a logical machining sequence, the machining sequence must be set to **Horizontal - From bottom to top** mode *(See: ›Set automatically‹ Page 1027)*.

Number of height sections

This option allows you to divide the sheet into multiple height sections. The maximum number of height sections is limited to 10.

Set lead-in

Sets the lead-in of a contour between two cutting areas exactly to the separation position. This lead-in is cut perpendicular to the contour.

These attributes are used to define the machining process for pre-piercing.

Length of lead-in

Specifies the length of the lead-in. Length of lead-Do not set length of lead-in too long in order to avoid a collision of the lead-in and another contour (approx. 5mm).

These attributes are used to define the machining process for pre-piercing.

Laser/Water piercing mode

This option is used to specify the piercing mode for all repositioning cuts. If the **Default** piercing mode is selected, the piercing mode for the outer contour of the repositioning part is used.

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Parts ID

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this section you can define settings for the parts ID point matrix.

The point matrix consists of modules consisting of several points arranged in a square (**A**). The size of a point matrix is calculated from the number of points in a module (**A**) and point spacing (**B**).

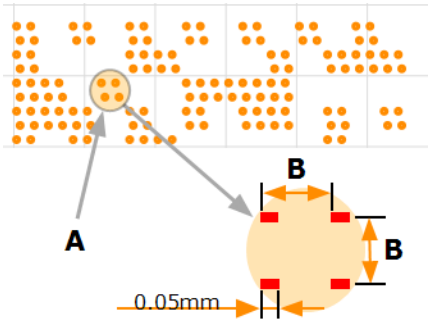


Fig. 10-149: Point matrix with module size 2



Note: The parts ID is set on the part in the **Geometry** menu of the Part Editor (*(See: ›Insert‹ Page 502)*).

Machining mode

These attributes are used to set the machining mode for parts ID.

- **Per part**

  The point matrix of the parts ID of a part is first engraved and then cut according to the machining sequence.

- **Per plan**

  The point matrix of the parts ID of all parts in the cutting plan is first engraved and then cut according to the machining sequence.

Module size

This option determines the number of points which make up a module. The number of points is calculated from the square of the number input. A value of 2 therefore means a module with 4 pierces (**A**).

☞ **Note:** Only change the default value (=2) if there are problems detecting the point matrix of the part.

Point spacing

This option defines the spacing of a module's points from pierce to pierce (**B**).

☞ **Note:** Only change the default value (=0.2) if there are problems detecting the point matrix of the part.

Interval

This option determines which parts to set the engraving on.

A value of 1 means that the part ID is engraved on every part. A value of 2 means that the part ID is engraved on every second part (etc.).

Only top part on stack

If a stacking plan is created in the job, this option sets the parts ID only on the topmost part of a stack. The part that receives the parts ID is displayed in the Simulator.

☞ **Note:** The **Interval** option must be set to 1 in this case.

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Bend lines

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

This section can be used to configure the settings for engraving bend lines. Engraving ensures that all the bends on a part can be identified more easily.



Attention! Engraving bend lines can result in significantly longer machining times. Recalculate to check the machining time.

- **Positive bend angle**
  a continuous line is engraved.
- **Negative bend angle:**
  a dashed line is engraved.

Selection mode

These options are used to define which bend lines will be engraved.

- **None**

  None of the bend lines will be engraved.

- **All**

  All bend lines will be engraved.

- **All with cutting technology**

  All bend lines for which a cutting technology has been set will be engraved. The cutting technology for bend lines can be set in the Part Editor's **Cutting technology** menu by using the **Contour** function *(See: ›Contour‹ Page 681)*.

Output entire line

If this option is enabled, the bend line's entire length will be engraved.

Length at both ends

This option is used to define the engraving length on a bend line. The specified length will be engraved on both sides of the bend line (**A**).

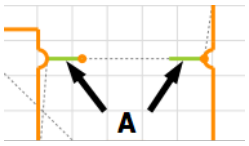


Fig. 10-150: Engraved bend line ends

Overwrite existing operating mode

If an operating mode other than the one selected here is set on the bend line, the operating mode specified here will be used to overwrite it.

Laser/Water operating mode

This option is used to specify for all common cuts the operating mode to be used to machine the common cuts. If the **Default** operating mode is selected, the operating mode for the outer contour of the common cut part is used.

Open profiles

How do I open this tab?

Select: **Modules > Tube Nester > Export > Plans > Export settings > Preprocessor**

In this section you can define settings for separating cuts for L and U profiles.

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Distance from profile edge without detection

Defines the position (**A**) before the edge of the profile as of which cutting head scanning is disabled in order to prevent the cutting head from being lowered after moving over the edge of the profile.

☞ **Note:** The switch-off position applies to both edges of the profile.

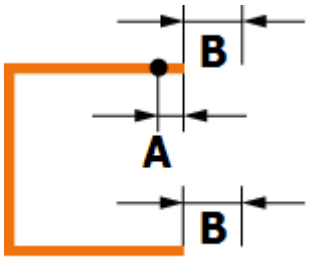


Fig. 10-151: Scanning switch-off position at the edge of the profile

Top overcut

Specifies the distance (**B**) to cut over the top edge of the profile.

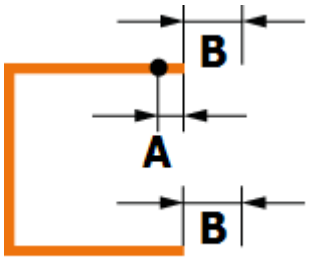


Fig. 10-152: Top distance outside profile

Bottom overcut

Specifies the distance (**B**) to cur over the bottom edge of the profile.

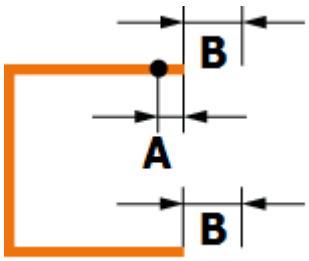


Fig. 10-153: Bottom distance outside profile

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Special

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings > Pre-processor**

In this section you can configure special settings for the preprocessor.

Complete all engravings first

If this option is enabled, all the engravings will be engraved before the contours are cut.

- **Per part**
  All engravings are performed first for each part.
- **Per plan**
  All engravings are performed first for each cutting plan.

Always pulsed piercing

When this option is enabled, all piercing cuts will be made in pulsed mode, regardless of whether or not the Cutting Wizard is used.

Disable sheet clamping

This option disables automatic sheet clamping on the **BySpeed Pro** machine. This setting overrides the setting on the machine.

Plan code

This option engraves the name of the plan as a data matrix code at the bottom left edge of the sheet (**A**). When you read the code with a hand scanner, the appropriate cutting plan is displayed automatically in the **Parts removal** module.

☞ **Note:** When using this option, the bottom left edge of the sheet should be at least **15 mm** (**B**). The size of the code depends on the length of the plan name.

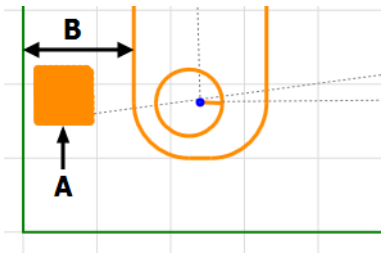


Fig. 10-154: Data matrix code

Print part labels

If this option is enabled, the labels in the cutting plan are output as a print job for Part Marker in the cutting plan. The print job is saved as a separate file in the BVC cutting plan.

☞ **Note:** If the **Part Marker** option is disabled for the current machine in **Administrator** -> **Cutting machines** -> **Options**, the labels are output on the part as an engraving based on the **Parts ID** settings.

Sensing off on lead-out

This option turns off scanning on an end cut. The option can prevent a collision of the cutting head at a tangential end cut if the cut-out has already fallen out and the cutting head would be lowered into the hole in the material due to scanning.

Do all inside contours first

This option cuts all inside contours on the cutting plan first with the specified interval. An interval of **1** means that every second part is cut in the machining sequence. An interval of **0** means that the machining sequence is executed as defined in the cutting plan.

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10.6.2.2 Extras

How do I open this tab?

Select: **Modules > Part Nester > Export > Plans > Export settings**

General settings for export options can be made in this tab.

General settings

Integrate cutting parameters

If this option is enabled, the path entered for the cutting parameter file in the Cutting parameters menu is written to the cutting program. The effect of this entry is that the cutting parameter file is automatically assigned to the cutting plan when the cutting plan is loaded on the machine.



Note: This parameter path must match the parameter path of the machine. If the paths are not the same, the CNC controller on the machine will not find the cutting parameter file.

Application:

- Enable this option if the same cutting plans are always machined with the same material on the same machine.
- Disable this option if the same cutting plans are machined with different material or on different machines.

Mirror cutting plans

This option mirrors the cutting plan around the X axis.

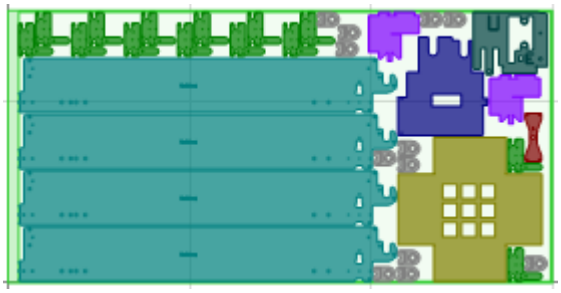


Fig. 10-155: Cutting plan prior to exporting

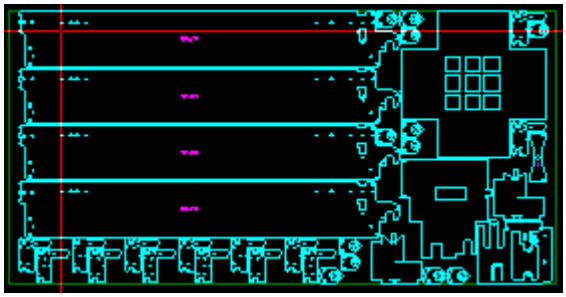


Fig. 10-156: Cutting plan after exporting

Application:

- Cutting tears or checkered plate when the flat underside faces upwards on the cutting table for cutting.

Pulse contour in radius area of section edges (Tube Nester)

This option prevents burning of corners in rectangular or polygonal tubes when cutting with oxygen.

☞ **Note:** Always enable this option when cutting rectangular or polygonal tubes with oxygen.

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Add missing cutting technology

If a cutting plan is exported for which the **Machining sequence** or **Skeleton cuts** have not been set yet, they can be set automatically prior to export if you enable the relevant option.

Large format processing

The options for cutting with sheet formats > 4000 mm in the X-axis direction can be configured in this area.

Number of height sections

This option allows you to divide the sheet into multiple height sections. The maximum number of height sections is limited to 20.

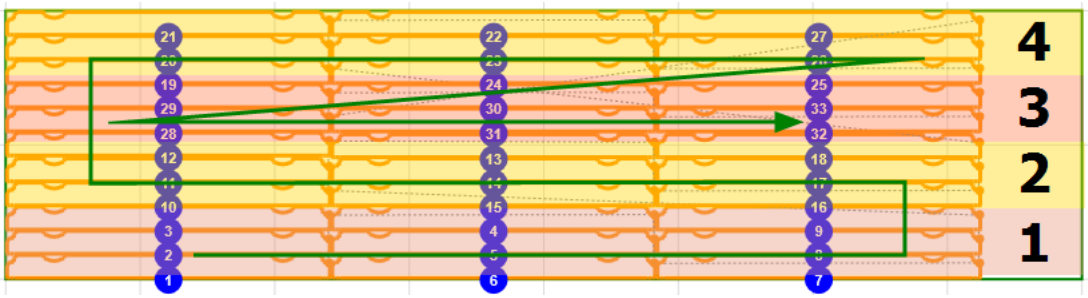


Fig. 10-157: Processing areas for a large format sheet with 4 height sections

☞ **Note:** This option ensures improved heat distribution on the sheet, as well as the resulting improved dimensional accuracy for the cut parts.

Multiple engraving

These options can be used to perform engraving in a certain thickness with the **Multiple engraving** tag. The thickness of the engraving is defined by a specified radius (**r**). The contrast of engraving is defined by the number of contour repetitions (**A**).

Attention! Using this option may increase machining time significantly.

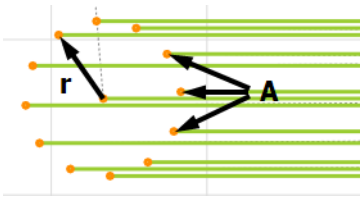


Fig. 10-158: Multiple engraving on a contour

Tip: To ensure that the distance between contour repetitions is as uniform as possible, enter an odd number of repetitions.

Number of repetitions

This option is used to define the number of engraving repetitions for a contour.

Note: The larger the number of repetitions, the longer it will take to process a contour.

Radius

This option defines the radius of a circle around which the number of contour repetitions are arranged.

Print

These options can be used to automatically perform print jobs for working documents or labels when exporting cutting plans or jobs.

Depending on the printer output setting in the print dialog, the print job is sent to the selected printer or saved in the export folder as a PDF. The printer output can be configured by pressing the **Settings** button.

Show dialog for PDF printer

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If a PDF printer is selected in the printer settings, a dialog is displayed before the working papers are generated, in which a path for the PDF must be selected.

Tip: Use this option if the path changes frequently.

PDF output folder

If a PDF printer is selected in the printer settings, the PDF will be saved to the path specified here.

Note: This option can only be used if the **Show dialog for PDF printer** option is not enabled.

Tip: Use this option if the path always remains the same.

ICP scanning off

This option can be used to activate ICP scanning and define the radius of scanning.

10.6.3 Group: View

Where do I find the group?

Select: **Modules > Part Nester > Extras**

This group contains all the functions that can be used to change the current view of the job. This group is also displayed in the **Data**, **Cutting technology** and **Export** menus.

The various functions and views are classified as follows:

Zoom functions

The zoom functions can be used to magnify or reduce objects in the graphics window.

Tip: You can also perform the zoom functions using the mouse thumb wheel.

Moving the view

The arrow symbols are provided in this group for moving the view of an object.

Tip: You can use the center mouse button to move the object freely in the current view. The view is moved by dragging and dropping with the mouse.

Special view

The **Special view** function is used to display various view options. The special-functions display can be specified by clicking on the arrow symbol and enabled/disabled by clicking on the symbol above it.

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10.6.3.1 Special view



This function shows/hides the options selected under the text button. The options can be used to display additional views on the object. Various options are available for selection depending on the module.



Tip: Use this function to get a quick overview of the technology options set on the current cutting plan.

The following information can be displayed:

- **Machining sequence:** *Displays the machining-sequence numbers of all the parts in the cutting plan.*
- **Removal sequence:** *Shows the removal sequences for the sorting process on the parts.*
- **Bounding rectangles:** *Displays the bounding rectangles for the parts in the cutting plan.*
- **Machining direction:** *Displays the direction of machining of all parts in a cutting plan*
- **Insertion points:** *Displays the origins of the parts in the cutting plan.*
- **Tools:** *For the water jet cutting machine, displays the cutting heads used for each part as a graded blue color.*
- **Machining sequence (color):** *Displays the machining sequence for the parts as a green color gradient.*
- **Common Cut:** *Highlights all parts with Common Cuts in color.*
- **Mirrored:** *Shows all parts in the cutting plan that are mirrored.*
- **Index holes:** *Shows all index holes on the cutting plan (See: ›Insert markings‹ Page 1088).*
- **Microjoints:** *Displays microjoints on contours.*
- **Geometry without cutting technology:** *Also displays contours of parts from an imported CAD drawing that do not have a cutting technology (See: ›Set cutting technology on part in CAD drawing‹ Page 1387).*
- **Support grate:** *Displays the shuttle table support grate if it has been assigned to the machine in Administrator. Use this display to nest parts manually taking the support grate into account (See: ›Aligning parts on the support grate‹ Page 1372).*
- **Sheet clamping:** *Displays the sheet clamping area in the cutting plan if it has been assigned to the machine in Administrator. Use this display to nest parts manually taking sheet clamping into account (See: ›Aligning parts on sheet clamping‹ Page 1374).*

- **Suction-cup frame:** *Displays the suction cup layout of the suction cup frame on the automation system if it is assigned to the machine in Administrator. Use this display to nest parts manually taking the suction-cup frame into account (See: ›Aligning parts on the ByTrans suction-cup frame‹ Page 1373).*
- **Small parts area:** *Shows the defined zones for nesting small parts on a sheet (See: ›Small parts mode‹ Page 956).*
- **Head-place zones:** *Shows the sorting area of the sorting heads as a colored area.*
- **Tilt optimization zones:***Shows the calculated risk zones for part tilting around the part's contour in different colors.*

10.6.3.2 Shade parts



This function is used to improve the appearance of the cutting plan by shading the surfaces of parts in different colors. You cannot edit the colors of the parts. The parts are displayed in the colors used to represent them in the **Picture** column of the **Parts** tab. The surfaces of the parts are semi-transparent so that overlying or underlying parts are still visible.

Note: Above a certain zoom level, the parts are only displayed as a contour.



Fig. 10-159: Cutting plan containing color-shaded parts

If the function is disabled, only the contour of the part is shown in the relevant color. The surfaces of the parts are transparent.



Fig. 10-160: Cutting plan containing parts with transparent surfaces

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10.6.4 Group: Cost Calculator

Where do I find the group?

Select: **Modules > Part Nester > Extras**

This group contains all the functions for calculating the job.

10.6.4.1 Create calculation

This function is used to create a calculation for the job in Cost Calculator.



After specifying the cost table in the **Cost calculator properties** dialog, he parts of the job are opened as a new cost calculation in Cost Calculator.

10.7 Menu: Extras

Where do I find the menu?

Select: **Modules > Part Nester**

This menu contains the functions for measuring coordinates, distances, and angles, plus various view functions.

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10.7.1 Group: Special functions

Where do I find the group?

Select: **Modules > Part Nester > Extras**

This group contains all special functions for nesting parts.

10.7.1.1 Original part



Where do I find the function?

Select: **Modules > Part Nester > Extras > Special functions**

This function is used to reset all contour element changes made in the cutting plan for the selected part back to their original state.

Affected contour elements:

- Lead-ins
- Microjoints

10.7.1.2 Separate parts



Where do I find the function?

Select: **Modules > Part Nester > Extras > Special functions**

This function embeds a copy of the selected part in the cutting plan. The embedded copy is no longer linked to the part in the Parts database.

All changes to the contour and cutting technology of the embedded part only affect the nested instances of the embedded part in the cutting plan.



Note: Separated parts are flagged with the **Integrate part** function symbol *(See: ›Integrate‹ Page 921)*.



Tip: Use the **Extract** function in the **Data** menu to save the separated part in the Parts database.

10.8 Menu: Machining time

Where do I find the menu?

Select: **Modules > Part Editor**

This menu displays all the machining times for a job. The machining times are displayed for the following elements of a job:

- Entire job
- All the plans in the job
- All parts used in the job

Click on the **Calculate** button to calculate the machining time. The machining time is calculated from the assigned cutting parameters and the machine setup values.

If one of the values is not displayed, the machining time cannot be calculated. The setup value is assigned to the displayed machine in the **Cutting machine** form of the Administrator *(See: ›Creating a new cutting machine‹ Page 1342)*.

The machining time is composed of the times for the following operating modes:

- Positioning
- Cutting
- Pulse
- Engraving
- Start/end of cuts
- Process macros

If there is not a unique set of cutting parameters that can be assigned to the part, the specific file required must be defined in the **Cutting parameters** field. To do this click on the ··· button and select the parameter file you require from File Explorer.

Clicking on the black arrow opens a filter selection (**A**), which can be used to enlarge or reduce the number of files displayed.

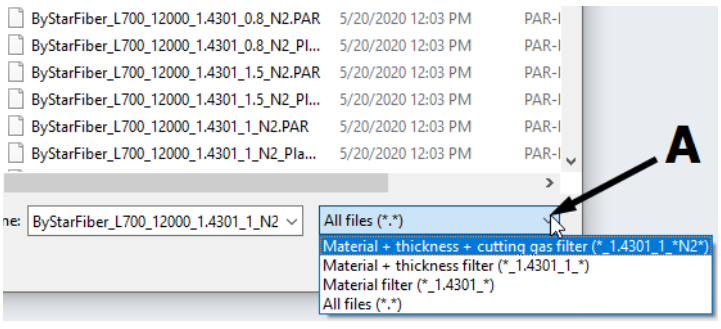


Fig. 10-161: Filter selection in File Explorer for cutting time calculation

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Note: If the **Automatic cutting time calculation** is enabled in the module settings, the **Calculate** and **Remove** functions are disabled.

Applications for calculating the machining time

- **Optimizing the job machining time**

  to check the machining time when the cutting technology is assigned manually, in order to optimize the machining time. In principle, an optimum balance between machining time and quality is achieved using the default parameters and the cutting rule sets.

- **Producing a quote**

  to provide a quotation for the customer.

- **Production planning**

  for optimum scheduling and utilization of the machine.

- **Recalculation**

  for checking the calculated and actual values for the machining time.



Tip: Each table can be selected individually and copied onto the clipboard with the **Ctrl + C** shortcut. This lets you reuse the calculated values in other documents.

10.8.1 Calculate machining time



Where do I find the menu?

Select: **Modules > Part Nester > Machining time**

This function calculates the machining time for the current job and shows the result in a table.

Machining times are calculated for the entire job and for each cutting plan and part present in the job. If a calculation has not been performed yet, just the table header is displayed.

The machining time can only be calculated if the **Cutting parameters**, **Setup values** and **Machine** fields each contain an entry.

If the cutting plan is changed after a calculation, machining time is automatically deleted again.



Note: If the Cutting Wizard is used, you must specify a Cutting Wizard in the export settings for calculating machining time. If this is not the case, an error message is displayed.



Attention! If the selected parameter file does not match the current combination of material, thickness and cutting gas, the program displays a warning before the calculation. Select the appropriate parameter file for correct calculation of machining time.

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10.9 Job data

How do I open this dialog box?

Select: **Modules > Part Nester**

This dialog box contains all the objects that are being used in the current job.

Each job must contain at least one element from the following objects in order to be able to create and export a cutting plan:

- A sheet or residual sheet
- A part with the quantity 1.
- A cutting plan containing at least one nested part

The objects are displayed in a table in the corresponding tabs. The number of objects in the tab is displayed in brackets. The **Job** tab displays a summary of all the objects being used.



Note: You can customize how the columns are displayed in each of the tabs *(See: ›Customizing table columns‹ Page 1334)*.

10.9.1 Sheets

How do I open this tab?



Select: **Modules > Part Nester > Job data**

This tab contains a table showing all the sheets in a job. All the functions in the **Sheet** group of the **Data** menu relate to this tab and can be run directly from within the group or via the shortcut menu.

Normally one sheet format is required per job. If you wish to use different formats of sheets or sheets with different edge distances in one job, you can also add more than one sheet format to the job.



Note:
The automatic nesting function can process multiple sheet sizes in order to achieve an optimal nesting result.

Residual sheets are used for nesting first. Nesting clearances can be defined around existing cutouts on residual sheets.

The use of multiple sheet sizes will not work:

- With multihead jobs.
- With ByVention jobs.
- If the **small parts mode** is enabled.
- If the parts in the job have various priorities.



Tip: If more sheets of one format are required for cutting plans than are available in the **Quantity** column, increase the number of sheets entered.

Adding sheets to a job

You have the following options for adding sheets to the job:

- Use the **Add sheet** function to add a sheet from a template or a user-defined format *(See: ›Add‹ Page 914)*.
- Use the **Add residual sheet** function to add a residual sheet *(See: ›Add residual sheet‹ Page 916)*.
- Use the **Create sheet from part** function to create a sheet from a part *(See: ›Create sheet from part‹ Page 917)*.
- Use the **Copy sheet** function to create a copy from an existing sheet *(See: ›Copy‹ Page 916)*.



Tip: You can use Drag & Drop with the mouse to copy sheets into another job.

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The entries in the following columns can be modified by hand:

- Quantity
- Length
- Width
- Edge distances

Categories in the status display

The status display uses different colors and symbols to indicate the status of the sheets. Each color is assigned to a specific category.

Icon	Category	Description
	Green	The whole number of sheets is used.
	Yellow	The number of sheets is used partially.
	Red	The sheet is not used yet.
		The sheet is a residual sheet.

What each column means:

No.

Shows the number of the object.

Picture

Previews the object in a diagram.

Status

Displays the status of the sheets in the nesting process.

Quantity

Shows the number of sheets of the same material that are available for nesting.

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Used

Displays the number of sheets used.

> **Note:** If the quantity shown here is greater than the number of cutting plans displayed in the **Cutting plans** tab, one or more cutting plans are being cut in multiple machining cycles.

Length

Shows the length of a sheet format.

Width

Shows the width of a sheet format.

Grain direction

Defines the grain direction of the sheet. It can be selected for **0°** and **90°**. The grain direction is used for grain direction compensation on a bending part *(See: ›Grain direction compensation (whole part)‹ Page 630)*.

If the **Grain direction** option is enabled in the part properties, all parts are nested so that the grain direction of the parts matches the grain direction specified on the sheet *(See: ›Grain direction‹ Page 930)*.

Left edge

Shows the left-hand edge distance for the sheet; parts can be nested up to this distance.

Top edge

Shows the top edge distance for the sheet; parts can be nested up to this distance.

Right edge

Shows the right-hand edge distance for the sheet; parts can be nested up to this distance.

Bottom edge

Shows the bottom edge distance for the sheet; parts can be nested up to this distance.

Info 1–8

Extra information about the current object can be entered in these fields.

Edge around cutouts

Sets a clearance around cutouts on residual sheets up to which parts can be nested.

Article information

Displays additional information about the product.

Batch information

Displays additional information about the product batch.

Area

Shows the total area of the sheet.

Weight

Shows the total weight of the sheet.

Available

Shows the number of sheets on which no parts have been nested yet.

Spacer plate

Shows the product number of the spacer plate. If there isn't an entry, you can enter the product number for a spacer plate manually.

The product number of the spacer plate is sent to ByVision Cutting in the *.bvc file and is then sent on to the store system if the **Spacer plate** option is enabled.

Priority

Defines the priority of sheets for nesting parts. The higher the number, the lower the priority of the sheet.

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10.9.2 Parts



How do I open this tab?

Select: **Modules > Part Nester > Job data**

This tab contains a table showing all the parts in a job. All the functions in the **Part** group of the **Data** menu relate to this tab and can be run directly from within the group or via the shortcut menu.

The colors of the parts are assigned automatically by the software. You can change the color of a part manually to improve the view in the cutting plan using the **Color** item in the context menu.



Note: If a part was selected that is nested in the current cutting plan display, all instances of the part are highlighted in the cutting plan.



Tip: You can use Drag&Drop to copy parts into another job.

Manual changes to the color of a part

1. Select the part from the **Parts** tab.
2. In the shortcut menu, click in the list box under the **Color** function.
3. Define the new color for the part in the color selection dialog box.

Copy part into a new job using drag & drop

1. Select the part to be copied from the **Parts** tab.
2. Click on the part with the left mouse button and drag it to the tab of the new job into which the part is to be copied. The job display switches to the selected job.
3. Drag the part into the workspace of the new job and then release the left mouse button. The part is copied into the job and is displayed in the Parts tab.

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Adding parts using a barcode scanner

A barcode scanner can be used to add parts to the job. To do this, connect the barcode scanner to the PC and scan in the barcode. The parts database is searched for the name of the part.

- If the part exists, it is entered in the **Parts** tab.
- If several parts are found with the same name, a dialog box is displayed in which you can select the part before adding it.

☞ **Note:** When scanning in using the barcode, there must be no parts selected in the input box of the **Parts** tab.

Adding parts to a job

- Use the **Add** function from the parts database *(See: ›Add‹ Page 918)*.
- Add a part to the cutting plan by dragging and dropping with the mouse. If you also press the **Ctrl** key when adding a part, the active part is automatically nested. Filler parts cannot be added using Drag&Drop.
- Copy an existing part using the **Copy part** function *(See: ›Copy‹ Page 920)*.
- Create or import a new part using the **New part** function *(See: ›New part‹ Page 922)*.
- Replace an existing by a new part from the parts database using the **Replace part** function *(See: ›Replace‹ Page 922)*.
- Scan in a part using a barcode scanner.

Nest parts as test parts

You can flag parts as test parts for test cuts.

To flag a part as a test part, select the part and enable the option in the **Test part** column *(See: ›Nesting a test part‹ Page 1375)*.

Test part properties

- Automatic nesting does not remove the nested part from the cutting plan.
- If a machining sequence is automatically assigned, the part and all of its instances are set at the start of the machining sequence.
- After cutting each part, a **programmed stop** is performed on the machine.

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Batch number quantity

This function multiplies the value entered by the number of selected parts in the parts list. This function is located under **Quantity** in the context menu of parts in the parts list.

Example:

1. Quantity of part = **10**
2. Batch number quantity = **2**
3. New quantity of part = **20**

Categories in the status display

The status display uses different colors and symbols to indicate the status of the parts. Each color and symbol is assigned to a specific category. The categories have the following relevance for nesting and exporting cutting plans:

Icon	Category	Description
	Color code	There are no restrictions on nesting or exporting the part. ☐ **Green** The full quantity of the part has been nested. ☐ **Yellow** Some of the specified quantity of the part has been nested. ☐ **Red** The part has not been nested yet.
	Note	The part appears more than once in the table. The part can be nested and exported in the cutting plan.
	Warning	The part can be nested and exported in the cutting plan. ☐ Contours in the cutting plan may overlap because of revision differences. In this case, check whether the nested plans containing the part need to be re-nested. ☐ The part has contours without a cutting technology. Set the cutting technology on all contours in Part Editor. ☐ The name of the part is changed.
	Error	Automatic nesting cannot be carried out. If the part is already nested in a cutting plan, the part is not exported with the cutting plan at export.

Icon	Category	Description
	Information	Displays important status information about the part.
	Embedded	The part is embedded in the job. It is not saved in the parts database.

☞ **Note:** The color of the symbol in the tab indicates the lowest nesting status of all the parts.

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What each column means:

No.

Shows the number of the object.

Status

Displays the nesting status of parts.

Picture

Previews the object in a diagram.

Name

Shows the name of the part. The name is used for unique identification of a part.

Quantity

Shows the available quantity of a part.

Nested

Displays the nested quantity of a part.

Extra quantity

Specifies the additional nested quantity of a part.

Available

Shows the number of a part not yet nested.

Priority

You can use this option to set the order in which the different parts are to be nested. Entering a priority is optional. Enter a number between 1 and 99 for the priority, Where 1 stands for highest priority and 99 for lowest priority. No priority is assigned if 0 is entered.



Attention! The use of different priorities does not usually result in optimum nesting of the cutting plan.



Note: This function is only used for automatic nesting.

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Size

Shows the size of the bounding rectangle of a part.

Rotation allowance

If rotation allowance is set to **Increment** in the part properties, you can select a preset angle in the selection dialog.

If rotation allowance is set to **Grain direction** or **Discrete angle of rotation** in the part properties, the **Rotation allowance** dialog opens for you to configure the various options *(See: ›Rotation allowance‹ Page 1208)*.

Filler part

Enable this option if you want to use this part to fill the areas leftover from nesting, or if the part is often needed in larger, unspecified quantities.

Auto Microjoint

When exporting, this option automatically sets a microjoint at the part's lead-in on every outer contour. Microjoints already set manually or by tilt optimization are ignored.

Mirroring authorization

This option permits the part to be mirrored about the vertical axis during nesting.

Description

Displays a more precise description of the part.

Area including eff. waste

Shows the area of the number of nested parts including waste. Waste of the entire sheet is distributed proportionately among the number of nested parts. Residual sheets are not included in the waste calculation.

Surface area including waste

Shows the area of the number of nested parts including waste. Waste of the entire sheet is distributed proportionately among the number of nested parts. Residual sheets are included in the waste calculation.

Order information

This field is not intended for any particular purpose. You can use this field to enter any information you wish.

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Common Cut

This function allows you to combine parallel contours during nesting based on certain criteria.

- **Activate**

  Activates Common Cut nesting for the part.

- **Identical parts only**

  Only identical parts will be used when combining contours.

- **By pairs**

  Common cuts will only be placed between two parts.

- **Max. cluster size**

  Defines the side length of a square inside which instances of the part can be nested as a Common Cut cluster. If this value is exceeded, a new Common Cut cluster will be started.

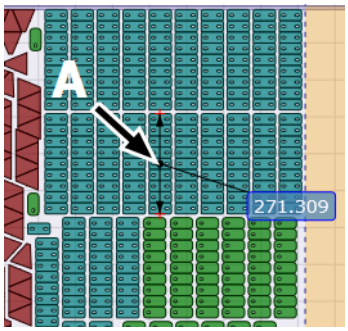


Fig. 10-162: Common Cut nesting with a cluster size of 300 mm (**A**)

- **Max. number of parts**

  Used to specify the maximum number of parts joined by common cuts.

- **Identical edges only**

  Will only join equal-length edges of a part as common cuts.

- **Min. area for new cut-outs**

  Used to set the minimum area for a cut-out (**A**) produced by common-cut nesting in which the part is being nested as a common cut.

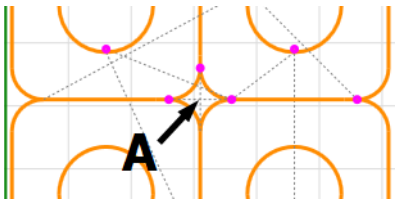


Fig. 10-163: Inside contour of a common-cut nesting arrangement

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Max. radius

Used to set the maximum radius after a straight contour at which common-cut nesting will still be performed.

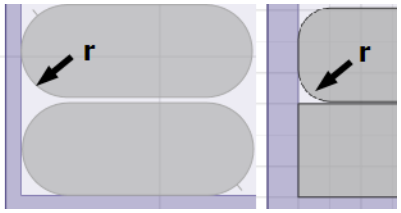


Fig. 10-164: Maximum radius for common cuts

Min. angle

Used to set the minimum angle after a straight contour at which common-cut nesting will still be performed.

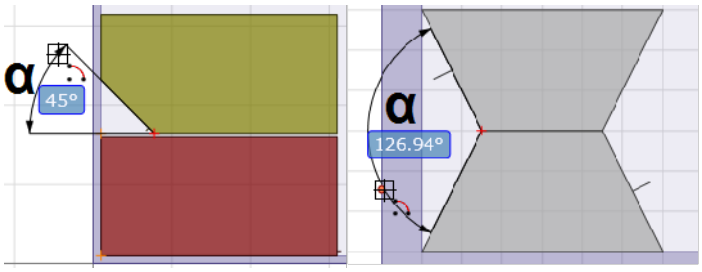


Fig. 10-165: Minimum angle for common cuts

Allow for lead-ins/lead-outs

This option can be used to achieve an optimized nesting result and improved process reliability. If this function is enabled, the lead-ins and lead-outs are handled by the software like a contour. This means that a parts protective distance is also placed around the lead-ins and lead-outs at nesting.

Copy and paste settings

Use these functions to copy the Common Cut settings of one part and assign them to other parts.

Copy

Copies the Common Cut settings of the selected part to the clipboard.

Insert

Assigns the copied Common Cut settings to the parts based on the selected part options.

Weight, outer contour

Displays the weight of the part without taking inner contours into account.

Effective weight

Shows the effective weight of the part. Inner contours are taken into account. The density of the material must be specified in the Administrator in order to be able to display the weight of the part.

Weight, rectangle

Displays the weight of the bounding rectangle of a part without taking inner contours into account.

Series info - Start number

Displays the number with which the next part begins when serial-number information is assigned automatically again in the **Cutting technology** menu.

Series info - Increment

The counting sequence of the numbers on the parts is specified in this field.

Example:

Increment	Counting sequence
1	1, 2, 3,…n
2	1, 3, 5,…n
3	1, 4, 7,…n

Series info - Min. number of places

Determines the minimum number of places of a number. If the number is smaller than the number given here, the missing places are padded with zeros.

Example:

Minimum number of places = **4**

Part counter = **95**

Number on part = **0095**

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Series info - Prefix

The text is set before the serial-number information.

Series info - Suffix

The text is set after the serial-number information.

Test part

Flags the part as a test part. In Part Nester all instances of the part are nested as test parts *(See: ›Nesting a test part‹ Page 1375)*.

Info 1–8

Extra information about the current object can be entered in these fields.

Cutting time

Shows the cutting time of the part in the format **h/min/sec**.

Cutting time [min]

Shows the cutting time of the part in the format **min** as a decimal.

Part status

Displays the status of the part in Part Editor. This allows you to quickly identify any errors or missing information on the part and use the **Edit** function to remedy these problems *(See: ›Icons in the workspace‹ Page 325)*.

Part in cluster

This option specifies that the part can only be nested in a framed cluster *(See: ›Framed cluster‹ Page 1213)*.

Unique in cluster

This option specifies the part can only be nested in a cluster. It is not possible to nest it outside a cluster.

Same orientation

Nests all instances of the part in the same orientation *(See: ›Same orientation‹ Page 929)*.

Bending part

If the field is checked, the part contains bending information. If the bending information is deleted from the part, the mark is automatically removed.

Take cutouts into account

Displays the setting for cutouts on the part. The setting is defined in the properties of the part *(See: ›Take cutouts into account‹ Page 932)*.

Customer

Displays the name of the customer. For jobs from Plant Manager, the name is taken from there. If no name is available, it can be entered manually.

Path

Displays the path for the part in the database.

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10.9.2.1 Rotation allowance

How do I open this dialog box?

Select: **Modules > Part Nester > Job data > Parts**

This dialog is used to configure the rotation allowance for the part based on various options.

☞ **Note:** If tandem grippers are set on a part, the rotation allowance is automatically restricted on machines with a sorting unit.

- **Navigate parts list**

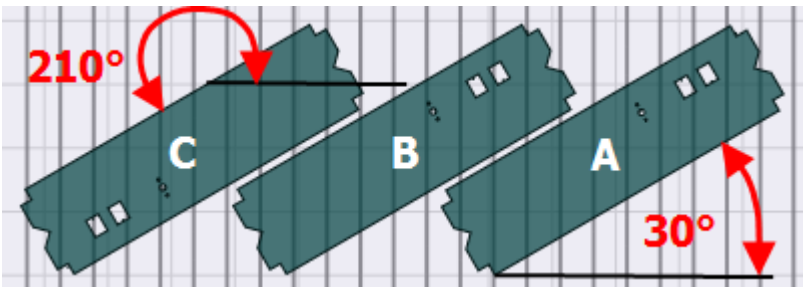
  Use the arrow keys to select the next or previous part in the list.

- **Same orientation**

  This option nests all instances of the part in the same orientation on the sheet.

  Example:

  If the part is nested with an angle of **30°** (**A**), the other instances can only be nested with the same angle (**B**) or with an angle of **210°**=180°+30° (**C**).



  Application:

  - Parts with same direction of grinding
  - The correction value only needs to be entered on the bending machine for the first bending part. Because the alignment of the grain direction is the same, it is not necessary to enter a correction for all other parts.

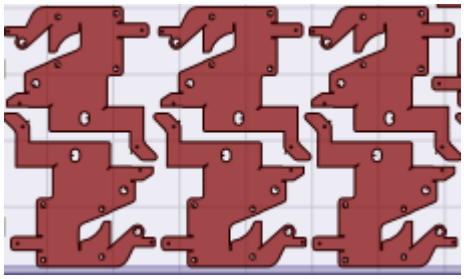


Fig. 10-166: Same orientation of a bending part on the sheet

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Increment

This field defines the increment of the angle of rotation in degrees. The part can be rotated during nesting through this angle.

Tip: A rotation allowance can be entered by clicking on the input field or by selecting a set angle value via the left arrow button.

Automatic increment

Copies the rotation allowance from the parts in the cluster. The cluster can only be rotated by the angles that are valid for all parts in the cluster.

Note: This option is only displayed for cluster parts!

Precise rotation

If this option is enabled, the part can be rotated by any angle. The angle of rotation can be specified to two decimal places. The input range < 1° is from 0.01 to 0.99°.

Grain direction

This option accounts for the grain direction when nesting the part.

The part is positioned so that the part's grain direction matches the sheet's grain direction. All instances of the part are nested with the same angle of rotation or rotated through 180° *(See: ›Grain direction compensation (whole part)‹ Page 630)*.

Note: The grain direction of the sheet can be specified on the **Sheet** tab in the **Grain direction** column *(See: ›Grain direction‹ Page 1194)*.

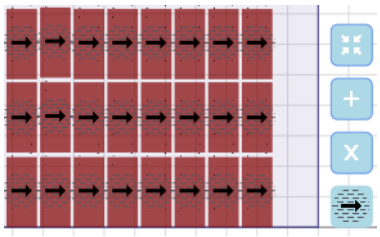


Fig. 10-167: Horizontal grain direction of the sheet from **0°**

Discrete angles of rotation

You can enter multiple angles of rotation in this box. The part can be rotated during nesting through this angle. The angles of rotation must be separated from each other by semicolons (**;**).



Tip: If this field contains angles of rotation, the angle of rotation in **Rotation allowance** is ignored (not for cluster nesting).

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10.9.3 Cluster

How do I open this tab?

Select: **Modules > Part Nester > Job data**

This tab contains a table showing all the clusters. Clusters with and without frame can be created. The type of cluster depends on the intended use.

Each cluster can be activated or deactivated by means of the **Active** column. If a cluster is activated, the maximum possible number of clusters and the required instances of the parts are reserved. As soon as the selected cluster is deactivated, the part instances are available again for normal nesting.

☞ **Note:** This tab is displayed if the **Clustering of parts** option is enabled in the Part Nester options or if a job already contains clusters *(See: ›Clustering parts‹ Page 905)*.

Framed cluster

A framed cluster consists of a closed part contour in which parts can be nested. In order to sort clusters, a gripper must also be set on the area of the cluster frame *(See: ›Add / Edit‹ Page 743)*.

You can create cluster frames in Part Editor and save them as **cluster frames**. You can use cluster frames, for example, for parts from an assembly with the same material and the same thickness *(See: ›Create cluster‹ Page 1394)*.

In order to sort clusters as a whole, the parts inside the cluster must have microjoints. The following options are available:

- Set the microjoints manually using the **Edit Microjoints** function from the **Cutting Technology** menu.
- Check the checkbox in the **Auto. microjoint** column on the **Parts** tab.

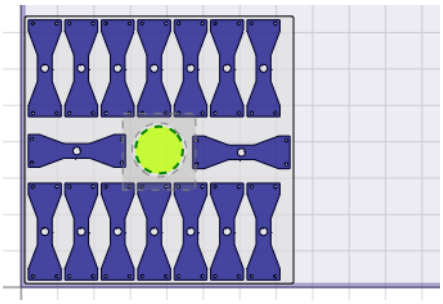


Fig. 10-168: Nested parts in a framed cluster

Note:

- In order for all instances of a part to be nested only in clusters, the **Part in cluster** attribute must be enabled on the part on the **Parts** tab *(See: ›Part in cluster‹ Page 1206)*.
- The cutting technology is defined within the cluster and then applies to all instances in the cutting plans.

Tip: Filler parts can also be stored in framed clusters. Once all normal parts have been nested in a cluster, the program attempts to nest the appropriate filler parts in the vacant area.

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Frameless cluster

A frameless cluster consists of two or more parts that can be nested as a single unit on a cutting plan.

If a frameless cluster is to be sorted off a cutting plan, a gripper must be set on the parts in the cluster by selecting the **Manual** gripper function from the **Sorting technology** menu. The grippers can be set either on the template of the frameless cluster or directly in the plan after being inserted on a plan.

A frameless cluster is indicated by a dashed rectangle around the nested parts.

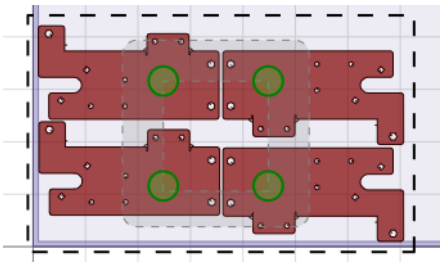


Fig. 10-169: Nested parts in a frameless cluster

Note:

- The parts in the cluster can be nested together with normal parts and can have different cutting technologies (cutting sequence, microjoints, etc.).
- With regard to cutting technology, the parts of frameless clusters can be used like normal individual parts.
- In a cluster template, only the arrangement of parts and grippers (optional) is defined. All other definitions refer to the entire cutting plan.

Frameless clusters can be created in the following ways:

- For parts that have already been nested, using the **Add** function in the **Frameless Clusters** section of the **Data** menu.
- With the context function **Add (Without Frame)** on the **Cluster** tab.

Attention! If the cluster is to be sorted, the parts in the frameless cluster must not contain any microjoints.

Note:

- If frameless clusters are nested in a cutting plan, they can no longer be edited.
- The **Part in cluster** option has no meaning for frameless clusters.



- If no multi-part gripper is assigned to the cluster, the clusters cannot be sorted.

Advantages:

- There is less waste because nesting does not require space for a frame part, as is the case with a framed cluster.
- More parts can be nested on a plan than with frameless clusters.
- Allows multiple parts to be picked up with one gripper tool.
- Allows you to create part modules as an assembly for use in different cutting plans or cutting jobs.

Disadvantages:

- The parts must be large enough to be picked up by a gripper.
- Suitable multi-part grippers must be available for sorting frameless clusters.
- Parts of frameless clusters must be placed on the stack carrier in the same lay-out as they are taken off the cutting plan.

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Nesting clusters

Clusters can be nested in the following ways.

- **Automatic**

  The function automatically nests the cluster part into all existing frameless clusters. New framed clusters are nested based on the frame parts previously contained. Cluster frames with different nestings are created as new clusters on the **Cluster** tab. Unused clusters are deleted.

  You can also use the automatic nesting functions **Automatic**, **Complete**, **Plan** and **Module** to nest clusters (). These functions do not set a machining sequence or any skeleton cuts, however.

  > **Note:** The function can be used only for framed clusters. Frameless clusters are deleted by this nesting strategy.

- **Automatic (leave existing clusters)**

  The function nests all cluster parts into all available clusters. Existing clusters are left in the job and new clusters are additionally nested based on available parts with the **Part in cluster** attribute.

- **Complete**

  With this function, all clusters which have not yet been used in the job are nested in existing or new cutting plans. This does not change any clusters and parts already nested. All available parts and clusters are nested on existing or new plans.

- **Manual**

  By dragging clusters into the cutting plan.

> **Tip:** The **Row** function can also be used to nest clusters.

Using clusters in Plant Manager

- **Job wizard**

  If the Job Wizard contains cluster parts, the **Automatic** function automatically nests the parts in clusters.

  You can use the **Manual** nesting function to nest clusters manually.

- **Scheduler**

  When the system adds parts to an order, it checks whether there are any cluster parts. If so, the system adds all PMC cluster frames to the new order.



Note: The cluster frames for Plant Manager must be saved in the directory **..\Parts\ClusterFrames\PMCClusterFrames**.

Context functions

Add (without frame)

This function replaces existing individual parts on a cutting plan with a frameless cluster. After adding, you can add parts to the cluster using the **Edit** function.

Add

This function adds a cluster frame to the tab. After adding, you can add parts to the cluster using the **Edit** function.

Remove

This function removes the selected cluster from the tab. If the cluster is nested in the cutting plan, the nested instances of are automatically deleted from the cutting plan.

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Edit framed cluster

This function opens the cluster for editing in the workspace. Only parts on the **Parts** tab that have the **Part in cluster** attribute set can be nested in a framed cluster.

Note:

- All functions that are unavailable are grayed out in the **Data**, **Nesting** and **Cutting Technology** menu.
- You can use the appropriate functions in the **Technology** and **Nesting** sections to nest parts in a cluster.
- In the nesting settings you can only configure settings in the sections **General**, **Sort**, and **Small-parts mode**. All other sections are hidden.
- Parts in the cluster can only be nested up to twice the parts protective distance from the edge of the cluster.

 This icon quits cluster editing mode.

Edit frameless cluster

This function opens the cluster for editing in the workspace.

From the **Parts** tab, add the parts you want to use in the cluster. A black frame is automatically drawn around all parts in the cluster.

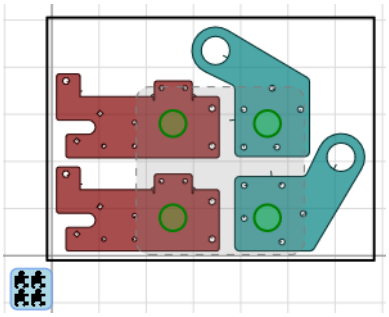


Fig. 10-170: Edit mode in frameless cluster

 This icon quits cluster editing mode.

Note: Frameless clusters can only be edited if no instance of the cluster has been nested in a cutting plan. If instances of the cluster have already been nested, the function is disabled.

Properties

This function calls up the **Cluster part properties** dialog to edit the properties of the selected cluster. This dialog is mostly identical to the **Part properties** dialog *(See: ›Part properties‹ Page 927)*.

Exceptions:

- You cannot edit general properties.
- Part information and nesting settings are hidden.

Edit frame

This function lets you edit the frame or the gripper of the cluster part in Part Editor. Parts nested in the cluster are shown with dashed construction lines. Construction lines are used to assist you when editing the frame so that the parts are always inside the frame.

Attention! If the edited frame does not enclose existing parts or is closed, the change is discarded.

Note:

- Changes are only stored in the frame of the selected cluster part. The original template is not changed.
- The function is only enabled for framed clusters.

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Import)

This function can be used to import a stored cluster template into the job. You can import several clusters at the same time.

Select a template in the **Import cluster** dialog and press the **Open** button to confirm.

☞ **Note:** If the parts are in the parts database but not yet in the job, enough parts are added to the job to nest an instance of the cluster. Therefore, the cluster can only be added to the job once after importing. To create additional instances of the cluster, increase the number of parts on the **Parts** tab or the number of clusters in the **Quantity** column. The parts are then entered as **Extra quantity** on the **Parts** tab.

Export)

This function can be used to export the selected clusters as a template to the cluster database.

☞ **Note:** When exporting multiple clusters, no clusters may have the same name. If you try to export clusters with the same name, an error message is displayed.

What each column means:

No.

Shows the number of the object.

Picture

Previews the object in a diagram.

Status

Shows the nesting status of the cluster with icons in two rows.



- **Row 1:** Displays warnings or errors in the cluster or cluster frame.
  - Cluster or cluster frame contains errors. If there are errors and warnings, the error icon is displayed.
  - Cluster or cluster frame contains warnings.
  - No icon: Cluster and cluster frame are OK.
- **Row 2:** Shows whether the cluster contains errors or warnings.
  - Cluster contains errors
  - Cluster contains warnings
  - No icon: Cluster is OK.

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Name

Displays the name of the cluster. The name is used for unique identification of the cluster. The name can be changed. When adding clusters, the name is used that is given as the default name in the Part Nester settings on the **Options** tab *(See: ›Other functions‹ Page 904)*.

Name of cluster frame

Displays the name of the cluster frame.

Active

Activates the cluster for nesting. If the checkbox is disabled, all clusters that have already been nested are removed from the cutting plan and the part instances that were used are available for normal nesting once again.

Quantity

Displays the number of clusters to be nested. If more parts are required for the number of clusters than are specified in the **Quantity** column on the **Parts** tab, the additional parts are automatically entered in the **Extra quantity** column.

Nested

Displays the number of nested clusters.

Available

Shows the number of cluster not yet nested.

Size

Shows the size of the bounding rectangle of a part.

Description

Displays a more precise description of the part.

Info 1-8

Extra information about the current object can be entered in these fields.

If a cluster contains different parts, the information texts of the respective parts are shown in the field separated by semicolons.

Area including eff. waste

Shows the area of the number of nested parts including waste. Waste of the entire sheet is distributed proportionately among the number of nested parts. Residual sheets are not included in the waste calculation.

Surface area including waste

Shows the area of the number of nested parts including waste. Waste of the entire sheet is distributed proportionately among the number of nested parts. Residual sheets are included in the waste calculation.

Auto Microjoint

When exporting, this option automatically sets a microjoint at the part's lead-in on every outer contour. Microjoints already set manually or by tilt optimization are ignored.

This option is required when you want to automatically set microjoints on all parts in the cluster. This is useful in connection with a sorting system or the **Tilt Optimization** function.

Test part

Flags the part as a test part. In Part Nester all instances of the part are nested as test parts *(See: ›Nesting a test part‹ Page 1375)*.

Rotation allowance

Opens the **Rotation allowance settings** dialog to configure the rotation allowance for the selected part based on various options.

- **Rotation allowance**
  This field defines the increment of the angle of rotation in degrees. The part can be rotated during nesting through this angle.

  Tip: A rotation allowance can be entered by clicking on the input field or by selecting a set angle value via the left arrow button.

- **Automatic rotation allowance**
  Copies the rotation allowance from the parts in the cluster. The cluster can only be rotated by the angles that are valid for all parts in the cluster.

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Grain direction

This option accounts for the grain direction when nesting the part.

The part is positioned so that the part's grain direction matches the sheet's grain direction. All instances of the part are nested with the same angle of rotation or rotated through 180° *(See: ›Grain direction compensation (whole part)‹ Page 630)*.

Note: The grain direction of the sheet can be specified on the **Sheet** tab in the **Grain direction** column *(See: ›Grain direction‹ Page 1194)*.

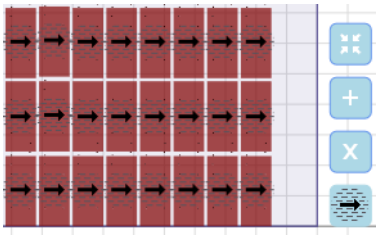


Fig. 10-171: Horizontal grain direction of the sheet from **0°**

Discrete angles of rotation

You can enter multiple angles of rotation in this box. The part can be rotated during nesting through this angle. The angles of rotation must be separated from each other by semicolons (**;**).

Tip: If this field contains angles of rotation, the angle of rotation in **Rotation allowance** is ignored (not for cluster nesting).

10.9.4 Cutting plans



How do I open this tab?

Select: **Modules > Part Nester > Job data**

This tab contains a table showing all the cutting plans in a job. All the functions in the **Plan** group of the **Data** menu relate to this tab and can be run directly from within the group or via the shortcut menu. Clicking on a cutting plan will open the cutting plan in the cutting plan view.



Tip: You can use Drag&Drop with the mouse to change the order of cutting plans within a job or copy cutting plans to another job. When copying to another job, all the parts in the cutting plan being copied are copied along with it.

Creating cutting plans

Cutting plans are created automatically by running the **Automatic** function in the **Cutting technology** menu. The number of cutting plans depends on the number of parts waiting to be nested and the nesting settings. A new cutting plan can be created in the **Sheets** tab by dragging a sheet into the cutting plan view of the Part Nester.

Creating names for cutting plans

The names for cutting plans can be created automatically or manually. The name is used for the LCC or NCP file at export.

- **Automatic**

  Cutting plan names are created automatically by running the **Automatic** function. The name of the cutting plan is composed of the job name and a serial number.

- **Manual**

  For manually created cutting plans, the name of the cutting plan must be entered by hand in the **Name** field.



Note: If a name is not specified for the cutting plan, it is created automatically when the job or cutting plan is exported.

The entries in the following columns can be modified by hand:

- Name
- Name

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- Cycles
- Edge distances

What each column means:

No.

Shows the number of the object.

Status

Displays the check status of cutting plans. The status of each cutting plan is saved as soon as you select the save function.

- **Gray** = The cutting plan has not been checked.
- **Green** = The cutting plan gets the green light (OK). The cutting plan can be exported.
- **Yellow** = The cutting plan contains errors. The cutting plan can be exported.
- **Red** = The cutting plan contains errors. The cutting plan cannot be exported.

Picture

Previews the object in a diagram.

Name

Shows the name of the cutting plan.

Name

You can enter a more detailed description of the cutting plan in this field. This text is written into the LCC or NCP file at export and displayed on the machine in the **Plan view** window in the WORK menu.

Plan dimension

Displays the cutting plan dimension used.

Cycles

Displays the number of repetitions with which the cutting plan is cut.

Waste

Displays the percentage of total material wasted from a sheet.

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Effective waste

Displays the percentage of effective material wasted from a sheet. This waste figure does not include residual sheets.

Left edge

Shows the left-hand edge distance for the sheet; parts can be nested up to this distance.

Top edge

Shows the top edge distance for the sheet; parts can be nested up to this distance.

Right edge

Shows the right-hand edge distance for the sheet; parts can be nested up to this distance.

Bottom edge

Shows the bottom edge distance for the sheet; parts can be nested up to this distance.

Plan dimension

Shows the size of the bounding rectangle for all nested parts.

Total parts

Displays the total number of parts in a job.

Sheet area

Shows the total area of the sheet.

Sheet dimension (consumption)

Shows the dimension of a sheet on which parts are nested.

Sheet area (consumption)

Shows the area of a sheet in **m^2** on which parts are nested.

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Info 1–8

Extra information about the current object can be entered in these fields.

Area of all parts

Shows the total area of all parts nested on the sheet.

Weight

Shows the weight of parts on the sheet.

Total weight

Shows the weight of parts on the sheet including all cycles.

Waste weight

Shows the weight of the skeleton.

Total waste weight

Shows the weight of the skeleton including all cycles.

Repositioning dividing position 1

Defines the position in X direction at which repositioning is performed. For 6.5 m repositioning machines, the default separation position is 4000 mm.

Application:

The optimal separation position is at that point in the cutting plan where the fewest contours need to be separated.

Big parts

Location where big parts are stored in the automation module.

Max. stack height, big parts

Maximum stack height for big parts at the storage site.

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10.9.5 Job

How do I open this tab?

Select: **Modules > Part Nester > Job data**

This tab summarizes information about the various objects in the job. The value shown in the **Total waste** field is the average value of all waste figures from the cutting plans in the job.

Note: The displayed values are for information purposes only and cannot be changed.

10.9.6 General functions

These functions are available in all the tabs of the **Job data** docking dialog box. They can be used to change the way the relevant objects are displayed in the tabs.

10.9.6.1 Change View

This function can be used to switch between the standard view and the custom view.

10.9.6.2 Reset layout

This function can be used to restore the default settings for the layout of the tab view.

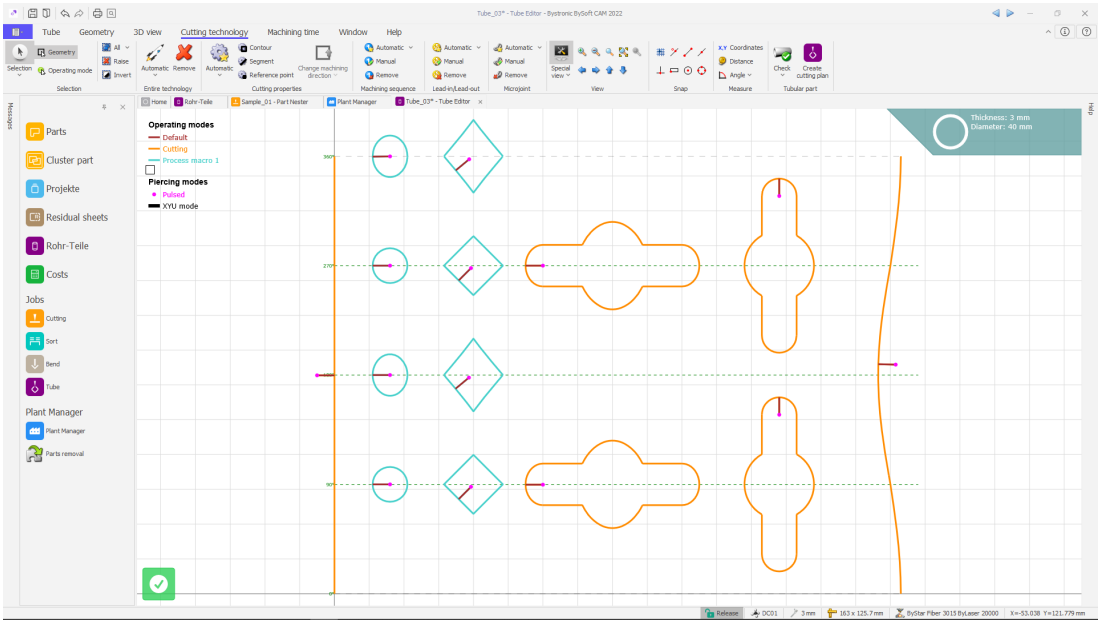
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11 Module: Tube Editor

Where can I find the module?

Select: **Modules >**



The Tube Editor is a module in which existing tubular parts can be edited, new tubular parts created, or tubular parts can be imported from a different format, in order to be able to assign a cutting technology to them. The editing process for parts basically follows the sequence of menus from left to right.

In Tube Editor, you can only make limited changes to the geometry of a part. More extensive changes to the geometry of a part can be made by running the **Start CAD** function from the **Part** menu.

Note: Most of the Tube Editor functions are identical to those in the Part Editor. Functions that are unavailable are shown as disabled and cannot be selected.

Tip: Click on an area of the header to call up the **Shortcuts** function in the context menu. The **Shortcuts** dialog can be used to define keyboard shortcuts for all functions of the current module.

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Preparing a tubular part for the nesting process

This procedure is recommended for importing a new tubular part into Tube Editor and then preparing it for the nesting process:

1. In the **Geometry** menu, set the origin in the lower left corner of the part, if possible.
2. Use the **Check** function to check whether the part contains any open contours.
3. In the **Cutting technology** menu, use the **Automatic** function to assign the cutting technologies to the part.
4. Use the **Check** function to check that all the contours have been assigned a cutting technology.
5. Save the tubular part in the Object Manager.

Tubular parts are displayed in the developed view in Tube Editor. The developed view varies depending on the type of tubular section.

Developed views of closed tubular sections

Circular tube

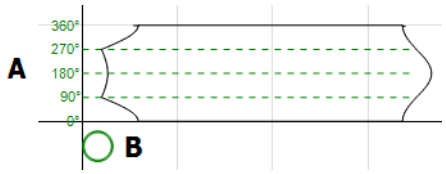


Fig. 11-1: Developed view of a circular tube containing angle details (**A**) and symbol for section type (**B**)

Rectangular tube

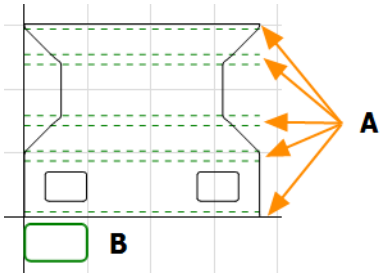


Fig. 11-2: Developed view of a rectangular tube showing radius region for corners (**A**) and symbol for section type (**B**)

Polygonal tube

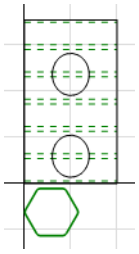


Fig. 11-3: Developed view of a 6-sided tube

Developed views of open sections

U-section tube

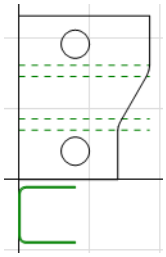


Fig. 11-4: Developed view of a U-section

L-section tube

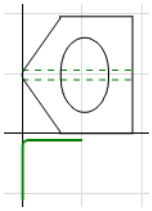


Fig. 11-5: Developed view of an L-section

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11.1 Menu: 3D-View

Where do I find the menu?

Select: **Modules > Tube Editor**

The tubular part is displayed in a 3D view in this menu.

11.1.1 Unfold



Where do I find the function?

Select: **Modules > Tube Editor > 3D view > View**

This function displays the tubular part as a developed view in the 3D view.

Use this function to display the tubular part as a developed view. Select the **Unfold** function again to display the part as a tube again.

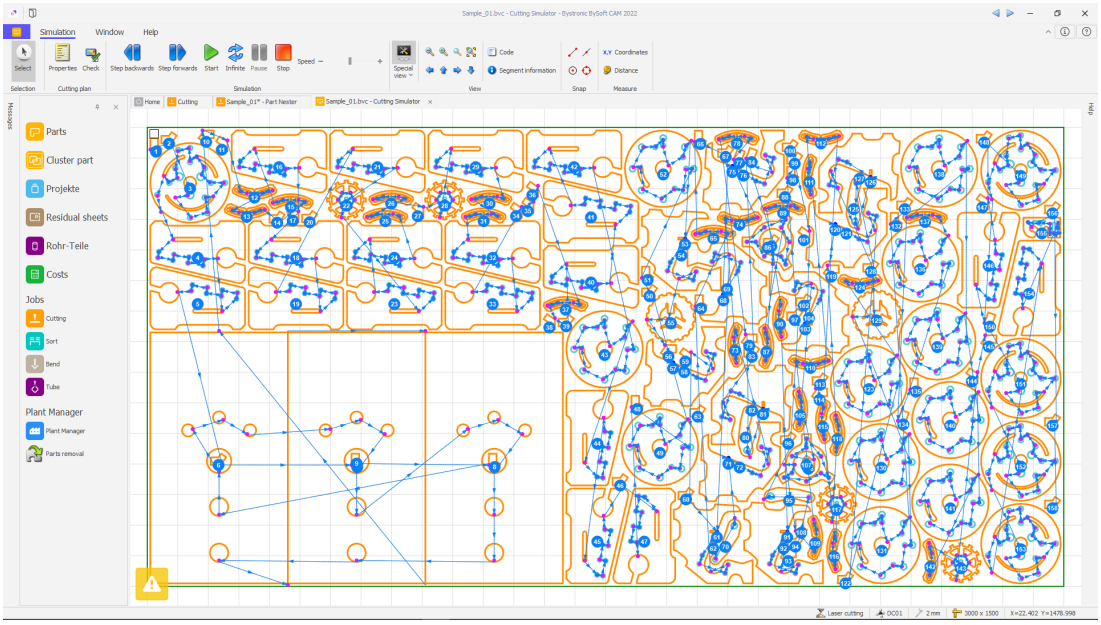
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12 Module: Cutting Simulator

How do I open this dialog box?

Select: **Modules > Cutting Simulator**



This module can be used to simulate the machining process for cutting plans.

Use the simulator to run a check in the preprocessor on the effect of all the activated parameters before exporting the cutting plan. The machining process in the simulator parallels the process carried out on the machine.

Note:
Positioning paths with raised cutting head are displayed **blue** in the simulator *(See: ›Raise cutting head‹ Page 1146)*.

For multihead cutting plans, the machining process is only simulated for the first cutting head.

Tip: Click on an area of the header to call up the **Shortcuts** function in the context menu. The **Shortcuts** dialog can be used to define keyboard shortcuts for all functions of the current module.

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12.1 File functions

The functions in this menu relate to the contents and user interface of the module.

Functions

- Open
- Settings

12.1.1 Open cutting plan

How do I open this dialog box?

Select: **Simulator >** 

This function opens a dialog box for selecting an LCC or NCP cutting plan. Select the part in the view window and then click the **Open** button.

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12.1.2 Settings

Where do I find the function?

Select: **Cutting Simulator >** 

You can use this dialog to configure view settings for the user interface in the **Cutting Simulator** module as well as options for logging nestings.

Use the **Reset** button to restore the default settings for the tab.

Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

12.2 Simulation

Where do I find the group?

Select: **Modules > Cutting Simulator > Simulation**

The buttons in the navigation bar are used to control the machining simulation. Parallel functions, such as changing the simulation speed or zooming in on a specific part, can be performed during the simulation.

What each button does:

Button	Description	Hotkey
	Moves to previous segment or positioning path	Space bar
	Moves to next segment or positioning path	Shift + space bar
	Starts the simulation	Ctrl + S
	The simulation runs in an endless loop. When the simulation ends, it starts again at the beginning.	Ctrl + L
	The simulation is paused at the current position. Press the button again to continue the simulation from the current position.	Ctrl + P
	The simulation is stopped. The start point is set to the first segment of the cutting plan.	Ctrl + Esc
	The simulation speed can be set with the slide control.	Page Up / Page Down

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12.2.1 Properties



Where do I find the function?

Select: **Simulator > Cutting plan**

This dialog box displays all the important information about the current cutting plan. The information is grouped into tabs under plan data, material data and process data.

12.2.2 Group: View

Where do I find the group?

Select: **Modules > Cutting Simulator > Simulation**

This group contains all the functions that can be used to change the current view of the cutting plan in the simulator.

The various functions and views are classified as follows:

Zoom functions

The zoom functions can be used to magnify or reduce objects in the graphics window.

Tip: You can also perform the zoom functions using the mouse thumb wheel.

Moving the view

The arrow symbols are provided in this group for moving the view of an object.

Tip: You can use the center mouse button to move the object freely in the current view. The view is moved by dragging and dropping with the mouse.

Special view

The **Special view** function is used to display various view options. The special-functions display can be specified by clicking on the arrow symbol and enabled/disabled by clicking on the symbol above it.

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12.2.2.1 Special view



Where do I find the function?

Select: **Modules > Cutting Simulator > View**

This function can be used to display various information on the technology for the cutting plan. The information is shown/hidden by clicking on the relevant option.

The following information can be displayed:

- **Operating mode:** *Displays contours in different colors depending on operating mode. Enable the **Key** view to display the operating modes used.*
- **Key:** *Displays in the top left of the workspace the operating and piercing modes used and the point attributes.*
- **Bounding rectangles:** *Displays the bounding rectangle for all contours of a part.*
- **Program number:** *Displays the program number for each part in the cutting plan. Identical parts have the same program number (except parts that have data fields). For parts with the **Pre-piercing** parameter, two numbers are displayed per part for the main program and subprogram.*
- **Positioning path:** *Displays the traversing path taken by the tool to the next part.*
- **Machining sequence:** *Displays the machining-sequence numbers of all the parts in the cutting plan.*
- **Point attributes:** *Shows the point attributes set on a contour.*
- **Tool correction:** *Shows the direction of tool correction as a black arrow.*
- **Particle:** *Displays splashing particles during the cutting simulation.*

Display code



Where do I find the function?

Select the menu: **Window > Cutting Simulator**

This dialog box displays the NC code in the cutting plan. The active code is highlighted during the cutting process or by clicking on a segment. The dialog box displays in-depth cutting process information that relates to the graphics view.

☞ **Note:** This dialog box is only intended for experienced users with a good knowledge of NC code. The NC code can be useful for debugging problems with the cutting plan.

Display segment information

Where do I find the function?

Select the menu: **Window > Cutting Simulator**

This dialog box displays information about the geometry segments during simulation. The information always relates to the currently active segment.

The following information is displayed:

- **Operating mode** *(machining mode for the segment)*
- **Piercing mode** *(piercing mode on the contour)*
- **Tool correction** *(side on which the tool correction lies)*
- **Straight-line segment** *(X/Y coordinates of start point and end point)*

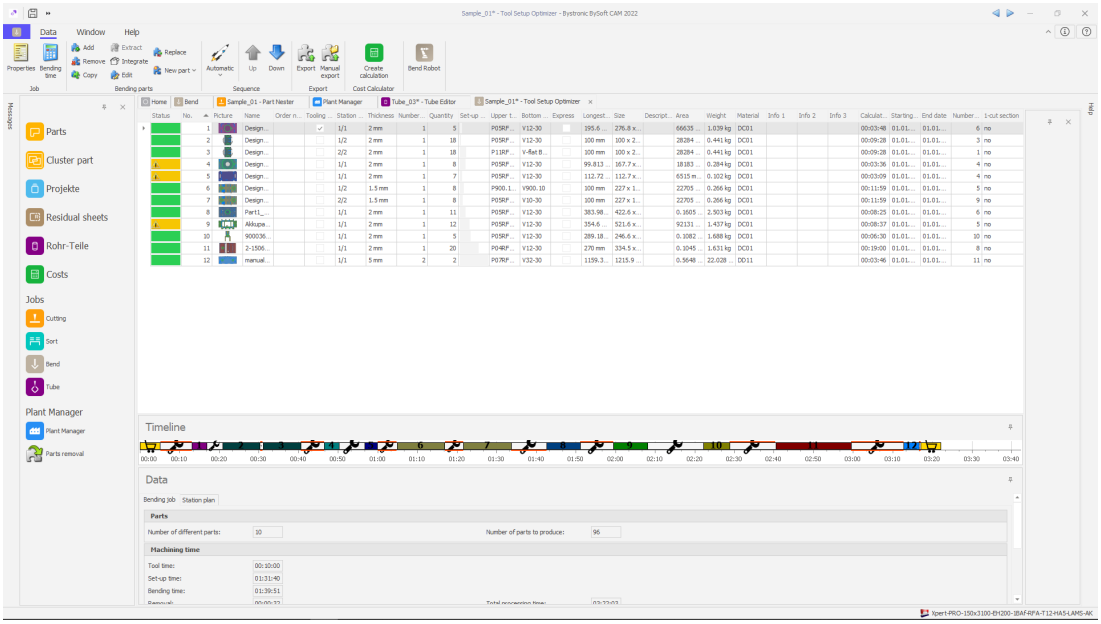
To obtain information on any segment, left-click on the relevant segment.

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12 Module: Tool Setup Optimizer

Where can I find the module?

Select: **Modules >**



The Tool Setup Optimizer is a module used to optimize the machining sequence of bending jobs based on certain criteria. By analyzing station plans and their sequence in the machining process, the module tries to reduce tooling and set-up time.

Tip: The module's greatest benefit is achieved when producing numerous different bending parts.

Tip: Click on an area of the header to call up the **Shortcuts** function in the context menu. The **Shortcuts** dialog can be used to define keyboard shortcuts for all functions of the current module.

Station plan list

Defines the sequence of station plans in the machining process. This results in the sequence of machining the bending parts.

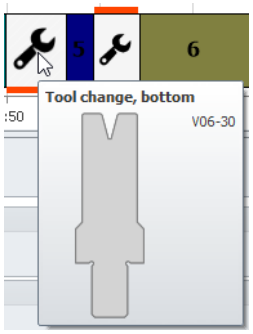
If there are multiple station plans per part, you cannot change the sequence of these station plans. Depending on the setting, however, you can position station plans in between.

Timeline

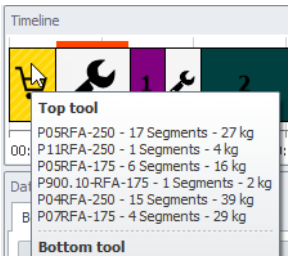
The timeline displays all bending processes with related tooling and set-up times on a time scale.



Fig. 12-1: Production process of bending parts on the timeline

Symbol	Meaning
	Time to bend the part on the station plan.
	Time to set up the station plan. The change of top and/or bottom tool is indicated by a red mark above or below the symbol. Hover the mouse over the set-up time symbol to display the tool changes required.  **Fig. 12-2:** Tool change drop-down display

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Symbol	Meaning
	Time for the tooling process. The tool time is the same for all station plans. Hover the mouse over the tooling process symbol to display all tools to be installed before the next tooling process along with their weights.  **Fig. 12-3:** Tooling process drop-down display

Creating a new job from bending parts

1. Select the **New job** function in the Tool Part Optimizer module.
2. In the **Job properties** dialog box, specify the name of the bending job, the system of units, and the bending machine that you want to use to bend the part.
3. Confirm your entries with **OK**.
4. In the **Data** menu, select the **Add** function and add at least three bending parts to the job.
5. Specify the number of parts to bend on each station plan in the **Number** box in the list of station plans.
6. Select the **Automatic** function in the **Sequence** group.
7. Select the **Print working documents** function in the main menu of the module.
8. Specify the printer, print template and number of copies in the Print dialog.
9. Click **OK** to confirm your selection. The working documents are printed on the specified printer.



Note: The working documents are required to set up the bending jobs on the machine.

What each column means:

Status

Shows the status of the bending part.

No.

Shows the number of the object.

Picture

Previews the object in a diagram.

Name

Shows the name of the part. The name is used for unique identification of a part.

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Number of operators

Shows the number of operators required to bend the part. The number of operators is specified in the **Part properties** in Part Nester

Order number

Shows the unique order number for identifying the order.

Tooling process

Includes tool time for the bending part in the timeline. The time for the tooling process is the same for all stations *(See: ›Tooling process‹ Page 1261)*.

Station plan

Shows the number of station plans for the bending part.

Thickness

Shows the thickness of the bending part.

Quantity

Enter the number of parts to bend in this box. If there are multiple station plans per part, you only need to enter the number for one station.

Set-up time

Shows the time to set up the part as a bar relative to the longest time in the list.

Top tool

Shows the top tool being used.

Bottom tool

Shows the bottom tool being used.

Express

If this field is enabled, the part is given priority over normal parts in the production process.

Longest bend line

Shows the length of the longest bend line on the part.

Size

Shows the size of the bounding rectangle of a part.

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Area

Shows the area of the part. Inner contours are taken into account.

Weight

Shows the weight of the part. Inner contours are taken into account.

Material

Displays the name of the material used.

Bending time calculated

Shows the bending time calculated for the part.

Start date

Displays the earliest date for producing the part. Click in the field to change the Start date.

End date

Displays the latest date for producing the part. If the **End date** is earlier than today's date, the date is highlighted in red.

Bends

Shows the number of bends for the part.

Info 1-8

Extra information about the current object can be entered in these fields.

Description

Displays a more precise description of the part.

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Menus in Tool Setup Optimizer

- File functions
- Data
- Window
- Help

12.3 New job



Where do I find the function?

Select: **Modules > Tool Setup Optimizer**

This function creates an empty job. Certain details must be specified in the **Job Properties** dialog box before the job is generated.

To create a new job, it is essential to enter the name. All the other details can be entered or modified later in the **Data** menu using the **Properties** function in the **Job** group, and when saving the job.

Settings

This section is only displayed when you create a new job.

from template

You can select a job template from the job database in this field.

The job settings are used from the selected template.



Tip: Use job templates for recurring jobs with the same settings *(See: ›Creating job templates‹ Page 1359)*.

from Settings Manager

The job settings are used from Settings Manager in Administrator.

General

Name

The name of the job must be entered in this field. If the **Job name generator** is enabled, the name is entered automatically *(See: ›Job name generator‹ Page 904)*.



Note: The name is essential for the program to be able to save the record.

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Description

You can enter a more detailed description of the current object in this field.

Info 1–8

Extra information about the current object can be entered in these fields.

System of units

The system of units for the current object is defined in this field (mm/inches)

☞ **Note:** The job can only be nested with parts that use the same system of units.

Bending process

Machine

This box is where you specify the machine with which you wish to machine the bending job. Clicking on the arrow symbol on the right lets you select a different machine from a list. This list only contains those machines that are configured in the system. If you require a new machine, you must create it in the **Administrator** module in the **Bending machines** form *(See: ›Bending machine‹ Page 98)*.

Bending-time table

Enter the bending time table for calculating bending time in this box.

12.4 File functions

The functions in this menu relate to the files and user interface of the module.



- Open
- Save
- Save as...
- Save all
- Save as template
- Save as default template
- Print working documents
  - Quick print
  - Preview
- Import
- Export
- System settings
- Close

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12.5 Menu: Data

Where do I find the menu?

Select: **Modules > Tool Setup Optimizer**

This menu includes all processing functions to create the sequence of bending plans.

Functional groups

The function bar contains the following functional groups:

- Job
- Bending parts
- Sequence
- Export

12.5.1 Group: Job

Where do I find the group?

Select: **Modules > Tool Setup Optimizer -> Data**

The functions in this group can be used to configure various job settings.

12.5.1.1 Job properties



How do I open this dialog box?

Select: **Modules > Tool Setup Optimizer > Data > Job**

In this dialog you can configure settings for bending machine, tooling process and tool times.



Note: Custom settings are saved for each user. If the software is run on a different PC with the same user login, the software uses the custom settings for that user.

Tabs in the Settings dialog box:

- General
- Settings

General

How do I open this tab?

Select: **Tool Setup Optimizer> Job > Properties**

This tab displays the basic properties of the bending job.

General

The general information describes the bending jobs and the system of units that are used. When the job is saved, various data is created automatically under the **Created**, **Changed** and **Revision** options, which cannot be changed manually.

- **Name**
  Shows the name of the bending job.
- **Path**
  Shows the path to the object.

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- **Created**

  Displays the date and time at which the object was created and the name of the user who created it.

- **Changed**

  Displays the current revision number for the object. The revision number is increased by a count value every time the object is saved.

- **Revision**

  Displays the current revision number for the object. The revision number is increased by a count value every time the object is saved.

- **Description**

  You can enter a more detailed description of the current object in this field.

- **Info 1–8**

  Extra information about the current object can be entered in these fields.

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

Bending process

- **Machine**

  This list box is where you specify the bending machine on which you wish to machine the bending part. Clicking on the arrow symbol on the right lets you select a different bending machine from a list. This list only contains those bending machines that are available in the system.

Settings

How do I open this tab?

Select: **Tool Setup Optimizer> Job > Properties**

This tab is used to define the times for set-up and the various set-up times. You can enter times in minutes and seconds.

The times should correspond to the mean time required for tooling or setup. You can find them in each case by running tests them on the machine.

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Tooling process

- **Tool time**
  Time for a tooling process.

Set-up times

This section i used to define the times for the various setup tasks.

- **Add / remove segments**
  Time needed to add or remove a segment for a station.
- **Align station**
  Time needed to align a bending station on the bar.
- **Measuring, correcting, rebending**
  Time required for subsequent measurement and correction of bends.
- **General tasks**
  Time needed for general setup tasks.

12.5.1.2 Bending time



How do I open this dialog box?

Select: **Modules > Tool Setup Optimizer > Data > Job**

In this dialog you can configure settings for bending time calculation. By default, bending time is calculated automatically.



Note: Bending time is displayed on the **Bending Jobs** tab.

- **Table**
  In this dialog you can select the bending table for calculation. Bending time is calculated on the basis of this table.

  The bending tables contain the bending times of the various axes of the bending machine. The tables are located in Administrator on the **Bending time tables** tab.
- **Calculate bending time**
  This button calculates bending time.

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12.5.2 Group: Bending parts

Where do I find the group?

Select: **Modules > Tool Setup Optimizer > Data**

This group contains all the functions for adding bending parts to a job or editing bending parts that have already been added.

12.5.2.1 Add



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Bending parts**

With this function you can insert one or more bending parts into bending jobs. In the **Add Part** dialog, you can select the bending parts and add them to the bending job by pressing the **Open** button.

Selecting more than one part

- To select parts in different folders, use the **Add** arrow buttons. The selected parts are added to the **Selected elements** area.
- Within a folder, you can select several parts at once using the **Ctrl** + left mouse button and then add them directly to the job using the **Open** button.
- All parts located using the **Find** function are displayed in the **Find** tab and can be added to the job from there.

☞ **Note:** All the parts selected in the **Parts** folder and all the parts in the **Selected elements** section are added to the job.

Find

You can use the **Search** box to search for parts based on certain part names. Press **Enter** to display the search results on the **Find** tab.

You can use a list box to restrict the search according to certain criteria.

- **Entire text**
  The program searches for a text that matches the character string exactly.
- **Start of text**
  The program searches for a text that starts with this character string.
- **Part of text:**
  The program searches for a text that contains this specified character string.

Selected elements

This section displays all parts to add to the **Parts** tab.

- **Add search result**
  Option enabled: If the search only returns one part, this part is automatically added to the **Selected elements** section.

What each button does:

Button	Description
	Adds the selected parts to the **Selected elements** area.
	Removes the parts selected in the **Selected elements** area.

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12.5.2.2 Remove



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Bending parts**

This function removes all selected bending parts from the bending job.



Tip: If you have deleted bending parts by mistake, use the **Undo** function to restore the deleted parts.

12.5.2.3 Copy



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Bending parts**

This function adds all selected bending parts, including all options, to the bending job as a copy.



Tip: Use this function to nest the same bending part with different option settings in the bending job.

12.5.2.4 Extract



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Bending parts**

This function extracts a bending part embedded in a bending job and stores it in the database for parts. This is the case when a bending job has been imported whose parts do not exist in the parts database. The revision number stored on the bending part is adopted.



Note:

- This function is only available for embedded parts.
- If the selected folder contains parts with the same name, a dialog allows you to choose whether to overwrite of skip these parts.

Extract embedded parts

1. Select the embedded part in the list of parts.
2. Click the **Extract** function in the context menu.
3. In the dialog, select the folder in which you wish to save the part.
4. Press the **Save** button. The part is saved to the specified folder.

12.5.2.5 Integrate



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Bending parts**

This function embeds the selected bending part in the bending job. It removes the link to the part in the Parts database.

All changes to the contour of the part's cutting and bending technology will now only affect the nested instances of the embedded part in the bending job.



Note: Embedded parts are flagged with the symbol for this function. 



Tip: Use the **Extract** function to save the part in the Parts database again.

12.5.2.6 Edit



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Bending parts**

This function can be used to edit the selected bending part in Part Editor. As long as the part is open in Part Editor, the bending job is locked.



Note: If any changes are made to the contour, the properties or the bending technology of the part in Part Editor, these changes are also applied to copied instances of the bending part in the bending job.

12.5.2.7 Replace



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Bending parts**

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This function replaces the selected bending part with another bending part. To do this, in the **Replace part** dialog box, select the part that will replace the previously selected part.

12.5.2.8 New part



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Bending parts**

This function can be used to add bending parts to the bending job in several ways. Assign the bending machine and bending technology to the bending part in Part Editor before adding the part to the bending job.

 **Note:** The job is locked from further editing so long as the part is open in the Part Editor.

Add a new part to the bending job

1. Select the **New part** function.
2. Select the **New part** creation method.
3. In the **New part** dialog box, specify the name, system of units, material and thickness of the part.
4. Use the appropriate method to create the part.
5. Assign the bending machine and bending technology to the part.
6. Close the part and confirm the prompt to save with **Yes**.
7. In the **Save as** dialog box, save the part in the parts database. The part is added to the bending job.

You can add a new part to the job using the following methods:

- New part
- Import part
- Import parts from clipboard
- Import part from file
- Import parts from job

Import bending parts from job



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Bending parts > New part**

This function adds all bending parts of a job to the bending job in Tool Setup Optimizer.

To do this, select the job containing the bending parts to be imported in the **Add Job** dialog and press the **Open** button to confirm your selection. All bending parts of the cutting job are imported into the bending job in Tool Setup Optimizer.

12.5.2.9 Multiple change



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Bending parts**

With this function you can change the values of different options for multiple bending parts at the same time. The options are displayed in a submenu of the function. As soon as you select a different option or exit the function, the specified values are adopted.



Note: The function can only be called via the context menu in the list of parts.



Tip: Bending parts for which you want to change certain values can be selected with **Ctrl + mouse click**.

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12.5.3 Group: Sequence

Where do I find the group?

Select: **Modules > Tool Setup Optimizer -> Data**

The functions in this group are used to specify the sequence of machining bending jobs.

12.5.3.1 Set sequence automatically



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Sequence**

This function is used to determine the sequence of bending jobs automatically based on the optimization settings *(See: ›Optimization settings‹ Page 1268)*.

12.5.3.2 Optimization settings

How do I open this dialog box?

Select: **Modules > Tool Setup Optimizer > Data > Sequence**

This dialog box is used to configure settings for optimizing tool times and set-up times.

Express orders

Prioritize express orders

If this option is enabled, the part's stations are moved to the beginning of the machining sequence.

Multi-person operation

Group

If this option is enabled, stations with the same number of operators are grouped together if there are different numbers of operators.

Sorting direction

- **Descending**
  Stations with highest number of operators first.
- **Ascending**

Stations with lowest number of operators first.

1-cut sections

Keep together

If this option is enabled, station plans with 1-cut profiles are placed successively.

Sort by bend line length

- **Longest first**
  Station with longest bend line first.
- **Shortest first**
  Station with shortest bend line first.

Parts with same tools

Optimization mode

- **Minimum segment change**
  All tools are taken into account to minimize segment changes.
- **Minimum segment change top**
  Only the top tool is taken into account to minimize segment changes.
- **Minimum segment change bottom**
  Only the bottom tool is taken into account to minimize segment changes.

Assessment criterion

- **Segment weights**
  Tries to use as little weight as possible from station to the next.
- **Number of segments**
  Tries to use as few segment changes as possible from station to the next.

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Parts with multiple station plans

Mode

- **Keep station plans together**
  The part is bent successively on the station plans.
- **Add station plans separately**
  You can insert station plans of other parts between the station plans. This does not change the sequence of station plans.

> ☞ **Note:** This option means that part bends cannot be performed successively and therefore that the part must be put down several times.

Tooling processes

Set tooling processes

These options set tooling processes in the timeline. The time for the tooling process is the same for all stations *(See: ›Tooling process‹ Page 1261)*.

12.5.3.3 Up



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Sequence**

This function moves the selected station plan one place up. Press the keyboard shortcut Shift + left mouse button to move multiple station plans at once.

12.5.3.4 Down



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Sequence**

This function moves the selected station plan one place down. Press the keyboard shortcut Shift + left mouse button to move multiple station plans at once.

12.5.4 Group: Export

Where do I find the group?

Select: **Modules > Tool Setup Optimizer -> Data**

The functions in this group can be used to export bending jobs automatically or manually.

12.5.4.1 Export bending parts



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Export**

This function exports all bending parts to the NC output folder for the active bending machine. The **NC output folder** for the bending machine is defined in the **Administrator** -> **Bending Machine** module *(See: ›Bending machine‹ Page 98)*.

12.5.4.2 Export bending parts manually



Where do I find the function?

Select: **Modules > Tool Setup Optimizer > Data > Export**

This function exports all bending parts to a user-defined directory.

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12.5.5 Bending job

How do I open this tab?

Select: **Modules > Tool Setup Optimizer > Data**

This tab contains information about all bending jobs listed in Tool Setup Optimizer.

☞ **Note:** The information applies to the sum of all bending jobs.

12.5.6 Station plan

How do I open this tab?

Select: **Modules > Tool Setup Optimizer > Data**

This tab displays the station plan of the selected bending part with the related information.

If more than one station plan is being used, you can switch between the various station plans using the two arrow keys.

For each station, the station number is displayed between the top and bottom tool.

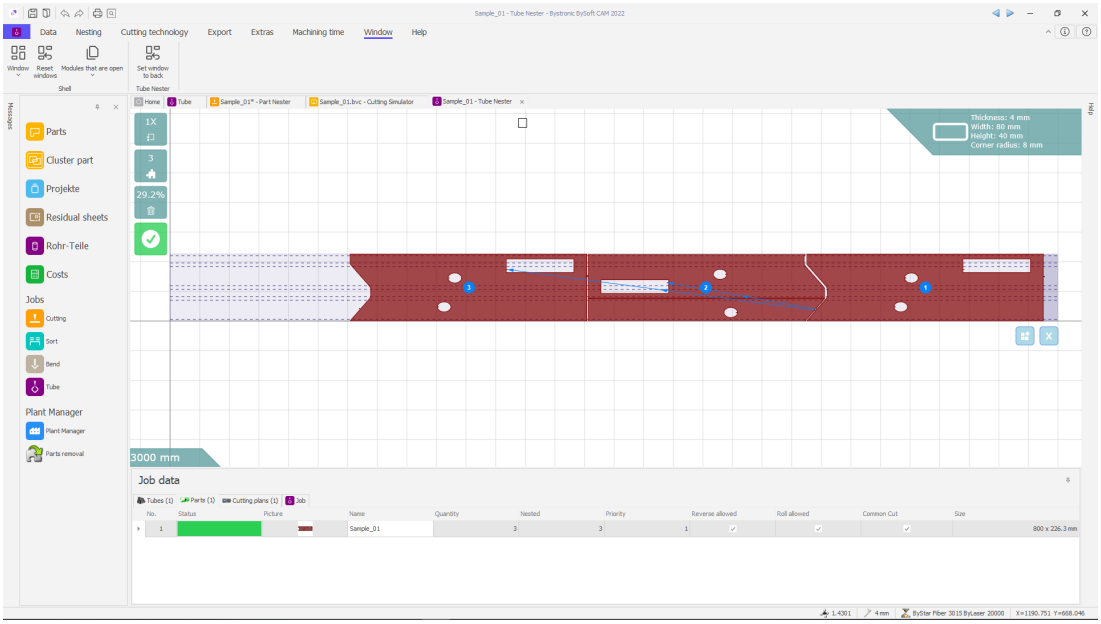
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13 Module: Tube Nester

Where can I find the module?

Select: **Modules >**



The Tube Nester is a module in which jobs can be produced from tubular parts and can be exported as a cutting plan. The generation process for jobs basically follows the sequence of menus from left to right. The parts on a tube are always nested from right to left.

All the parts in a cutting plan are always nested against the edge of the nearest part on the right. This prevents any overlap of contours or gaps between parts when placing parts manually.

Note: For cutting plans for tubes, the machining sequence is always from right to left.

Tip: Click on an area of the header to call up the **Shortcuts** function in the context menu. The **Shortcuts** dialog can be used to define keyboard shortcuts for all functions of the current module.

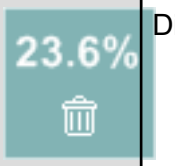
If the job contains parts with the **Warning** or **Error** category, this is indicated by a suitable message which is displayed for a certain period of time on opening the job.

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Cutting plan information

Each cutting plan displays information about the current cutting plan in the form of symbols. The symbols indicating the waste and number of cycles is refreshed automatically as soon as any change is made to the nested cutting plan. A navigation bar lets you move between several cutting plans.

Icons in the workspace

Symbol	Description
	Displays the number of machining cycles of the cutting plan.
	Displays the number of parts in the cutting plan. If the cutting plan has several cycles, the number of parts is displayed for all cycles.
	Displays the percentage waste for the cutting plan.
	Shows the status of the cutting plan. The status is updated in real time. ■ **Symbol 1:** The cutting plan is OK. The cutting plan can be exported. ■ **Symbol 2:** The cutting plan contains errors. The cutting plan can be exported. ■ **Symbol 3:** The cutting plan contains errors. The cutting plan cannot be exported.  **Tip:** Click on this symbol to call up the **Verify plan** dialog with detailed information about the status of the cutting plan.
	Displays the section dimensions for the cutting plan.
	Displays the length of the cutting plan.
	Displays the number of machining cycles of the cutting plan.

Information about the part

Moving the mouse over a part opens a pop-up window displaying various information about the part.

Note: A part must not be selected if you wish to display part information.

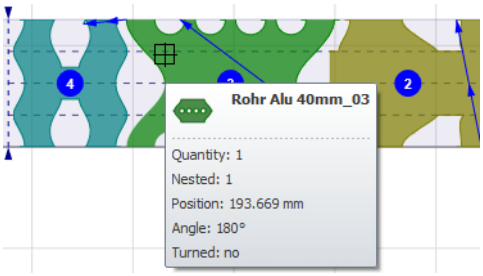


Fig. 13-1: Displaying information about parts in the cutting plan

Interactive buttons

Pressing the space bar will display interactive buttons in the Part Nester workspace. The buttons remain visible so long as the space bar is pressed.

Note: If the option **Always display interactive buttons.** is enabled in the Tube Nester settings, the buttons are displayed permanently.

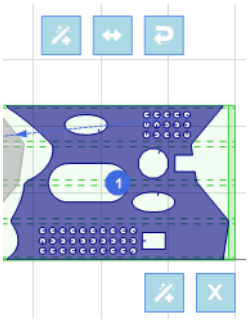


Fig. 13-2: Interactive buttons on a tube cutting plan

Buttons

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Button	Description
	This button deletes the entire cutting plan.
	This button nests the current cutting plan. If a part is selected in the cutting plan, this button is also displayed above the selected part (**A**). This button is used in order to nest only the selected part in the cutting plan. If multiple parts are selected, these are nested as a module.
	This button can be used to move the part to a different position on the tube, and is equivalent to the **Move part** function.
	This button rotates the selected part about the vertical axis and is equivalent to the **Reverse part** function.

Creating a tube job automatically

Follow the steps below to create a tube job from various parts and to export it for a cutting machine:

1. In the Tube Nester module, select the **New job** function.
2. In the **Job properties** dialog box, specify the job name, the material, the thickness, the machine and the section type.
3. Confirm your entries with **OK**.
4. In the **Tubes** menu, select the **Add** function to add at least one tubular section to the job.
5. Select the **Add part** function.
6. In the **Add part** dialog box, specify the parts that you wish to create the job from.
7. Press **Open** to confirm the selection. The parts are added to the job.
8. For each part, specify the quantity of parts to be produced in the **Parts** tab of the **Job data** area.
9. Select the **Automatic** function from the **Cutting technology** menu to create the job automatically. If the nesting result is not optimum, check the settings using the **Settings** function and re-nest the job.
10. Run the **Verify plan** function from the **Export** menu to check the job. If the job has been created correctly, run the **Export plans** function to export the cutting plans to the Export folder.

13.1 Menu: Data

Where do I find the menu?

Select: **Modules > Tube Nester**

In this menu you can nest tube plans fully automatically based on the settings and you can edit the raw material, parts and plans in the job.

Functional groups

The function bar contains the following functional groups:

- Tube

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13.1.1 Group: Tube

Where do I find the group?

Select: **Modules > Tube Nester > Data**

This group contains all the functions for editing tubes.

13.1.1.1 Add tube

Where do I find the function?

Select: **Modules > Tube Nester > Data > Tube**

This function adds a new tube format to the job. The format and quantity for the tube are defined in the **Add tube** dialog box. The new tube is displayed in the **Tubes** tab of the **Job data** docking dialog box.

What each field does:

- **Length**

  Length of the tube

- **Left edge**

  Length of the left-hand edge distance, within which parts are not allowed to be nested.

- **Right edge**

  Length of the right-hand edge distance, within which parts are not allowed to be nested.

- **Quantity**

  Number of tubes of this tube format that you wish to add to the job.

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13.1.1.2 Remove tube

Where do I find the function?

Select: **Modules > Tube Nester > Data > Tube**

This function deletes all marked tubes. It is also possible to delete tubes that already contain nested parts. If tubes that already contain nested parts are deleted, all the nested cutting plans of this tube format are also deleted.



Tip: If you accidentally delete nested tubes, use the **Undo** function to restore the deleted tubes.

13.1.1.3 Copy tube

Where do I find the function?

Select: **Modules > Tube Nester > Data > Tube**

This function adds all the tubes selected in the **Tubes** tab to the job as a copy.



Tip: You can use this function to create identical tubes with different tube edges.

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13.2 Menu: Nesting

Where do I find the menu?

Select: **Modules > Tube Nester**

In this menu, you can nest plans fully automatically based on the configured settings, and you can edit the tubes, parts and plans in the job.

Functional groups

The function bar contains the following functional groups:

- Entire technology
- Nesting

13.2.1 Group: Entire technology

Where do I find the group?

Select: **Modules > Tube Nester > Nesting**

This group contains all the functions for tube cutting technology.

13.2.1.1 Cutting technology settings



Where do I find the function?

Select: **Modules > Tube Nester > Nesting > Technology**

You can use this function in the relevant docking dialog box to set the options for fully automated assignment of the cutting technology and for nesting. Running the **Automatic** function will apply the settings to the entire job.

Nesting

The options in this area can be used to control the form and efficiency of nesting.

Resolution

This option specifies the angular increment in degrees through which the part can be rotated on the tube during nesting.

The resolution can be set between 1° and 360°. 1° gives the most precise nesting because the software considers every angle for nesting. A Value of 360° means that the part can no longer be rolled on the tube during nesting.

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Parts protective distance

This option prevents collisions when cut parts protrude. During nesting, the specified parts protective distance (**A**) is placed around the outer contour of the part. Hence the minimum distance between two parts equals at least twice the parts protective distance (**B**). The Parts protective distance is not applied to the sheet edge. The parts are nested right up to the edge of the cutting plan.

⚠ **Attention!** If the **Include lead-ins and lead-outs** option is not enabled, the Parts protective distance must be at least as large as the longest lead-in or lead-out. Otherwise the lead-ins or lead-outs for a part may lie within the contour of an adjacent part.

☞ **Note:** The lead-ins are only taken into account in nesting if the **Include lead-ins and lead-outs** option is enabled.

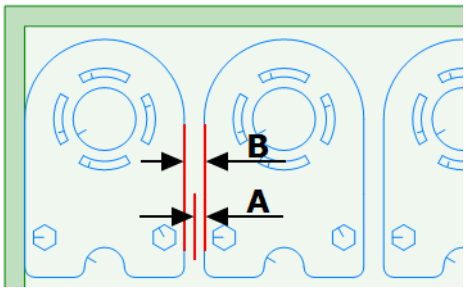


Fig. 13-3: Minimum distance between two parts: A = parts protective distance, B = double parts protective distance

Minimum parts protective distances

Material	Mild steel				Stainless steel	Aluminum
Assist gas	Oxygen				Nitrogen	
Thickness [mm]	1–5	6–10	12–15	16–25	1–25	1–15
Minimum parts protective distance	5	5	7.5	12.5	5	5

Max. permissible deviation

This option specifies the maximum tolerance up to which value the parallel outer contours of two parts are still cut with a common cut. A value of 0.5mm results in a common cut being made when the parallel contours have a maximum separation of 0.7mm.



Note: The maximum separation is calculated as the maximum permissible deviation plus the kerf width of 0.2mm (width of cutting gap) (e.g. 0.5 + 0.2 = 0.7mm).

Common Cut

Kerf

This value is used to define the kerf taken into account when combining common cuts.

Cuts

- **Intermediate cuts**

  If this option is enabled, a straight intermediate cut will be performed after every part.

- **End Cut**

  If this option is enabled, a lead-out is set at the end of a tube cutting plan based on the details.

Min. residual tube length

This option defines the minimum length of a residual tube on which a lead-out is still set.

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13.2.2 Group: Nesting

Where do I find the group?

Select: **Modules > Tube Nester > Nesting**

This group contains all the functions for nesting tube parts.

13.2.2.1 Place part

Where do I find the function?

Select: **Modules > Tube Nester > Cutting technology > Edit plan**

This function places the selected part in the cutting plan. The insertion position is indicated by two blue triangles (**A**). Left-clicking will place the part in the current position of the mouse cursor. The cutting sequence is automatically adjusted to suit this new position.

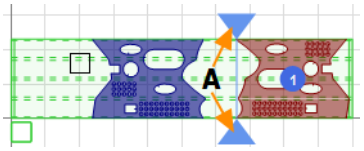


Fig. 13-4: Insertion position of a part on a rectangular tube

Placing parts using the mouse

1. In the **Parts** tab, select the part that you wish to place in the cutting plan.
2. Use the mouse to drag the part into the cutting plan. The part is positioned in the cutting plan at the mouse cursor.
3. Move the part into the required insertion position.
4. Left-click in the cutting plan at the new position where you wish to place the part.
5. Repeat steps 3 and 4 to place further parts with the mouse.

13.2.2.2 Move part

Where do I find the function?

Select: **Modules > Tube Nester > Cutting technology > Edit plan**

This function moves the selected part to a new position within the cutting plan. The position it will be moved to is indicated by two blue triangles. Left-clicking will place the part in the current position of the mouse cursor. The cutting sequence is automatically adjusted to suit this new position.

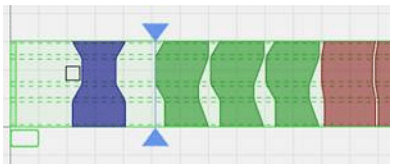


Fig. 13-5: Position to which a part shall be moved in the cutting plan

Moving a part using the mouse

1. Select from the cutting plan the part that you wish to move.
2. Select the **Move part** function. The part is positioned in the cutting plan at the mouse cursor.
3. Move the part into the required position. The insertion position is indicated by two triangles on the cutting plan.
4. Left-click in the cutting plan at the position where you wish to place the part.

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13.2.2.3 Reverse part

Where do I find the function?

Select: **Modules > Tube Nester > Cutting technology > Edit plan**

This function inverts the selected part through 180° in the cutting plan.

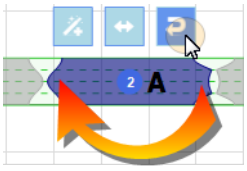


Fig. 13-6: Reversed part in the cutting plan of a circular tube

Reversing parts on a rectangular tube

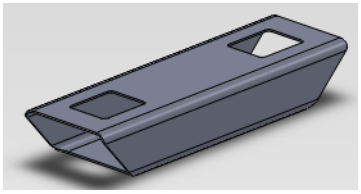


Fig. 13-7: 3D drawing of a rectangular tube

The reversal of parts on a rectangular tube constitutes a special case, because rectangular tubes have two different sector sizes: S 0+2 and S 1+3.

In order to ensure that the unfolded areas are still located in the correct sector sizes after reversing the part, the unfolded area **A** is moved into sector 3 after reversing the part.

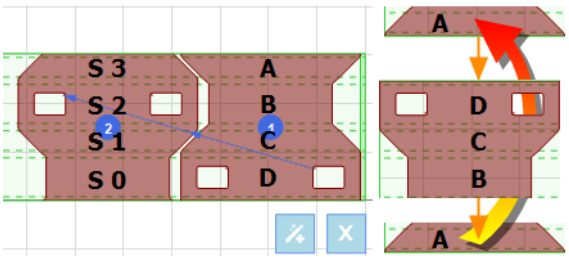


Fig. 13-8: Reversed part in the cutting plan of a rectangular tube

13.3 Menu: Cutting technology

Where do I find the menu?

Select: **Modules > Tube Nester**

The cutting technology for the current tube job is configured in this menu.

Functional groups

The function bar contains the following functional groups:

- End Cut

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13.3.1 Group: End Cut

Where do I find the group?

Select: **Modules > Tube Nester > Cutting technology**

This group contains all the functions for the tube end cuts.

13.3.1.1 Automatically set lead-out



How do I open this dialog box?

Select: **Modules > Tube Nester > Cutting technology > Lead-out**

This function sets a lead-out on tube cutting plans. The lead-out can be set on the current cutting plan or all cutting plans in the job.

Settings

- **Intermediate cuts**

  If this option is enabled, a straight intermediate cut will be performed after every part.

- **End Cut**

  If this option is enabled, a lead-out is set at the end of a tube cutting plan based on the details.

Min. residual tube length

This option defines the minimum length of a residual tube on which a lead-out is still set.

13.3.1.2 Remove lead-out



Where do I find the function?

Select: **Modules > Tube Nester > Cutting technology > Lead-out**

This function removes the lead-out from the current tube cutting plan.

13.4 Job data

How do I open this dialog box?

Select: **Modules > Tube Nester**

This dialog box contains all the objects used in the current tube job.

Each job must contain at least one element from the following objects in order to be able to create and export a cutting plan:

- A rectangular or round tube
- A tube part with the quantity 1.
- A cutting plan containing at least one nested part

The objects are displayed in a table in the corresponding tabs. The number of objects in the tab is displayed in brackets. The **Job** tab displays a summary of all the objects being used.



Note: You can customize how the columns are displayed in each of the tabs *(See: ›Customizing table columns‹ Page 1334)*.

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13.4.1 Tubes

How do I open this tab?

Select: **Modules > Tube Nester > Job data**

This tab contains a table showing all the tubes in a job. All the functions in the **Tubes** group of the **Data** menu relate to this tab and can be run directly from within the group or via the shortcut menu.

Normally one tube format is required per job. If you wish to use tubes with different edge distances in one job, you can use the **Add** function to add more than one tube format to the job.

Note: You can add new tubes to the job using the **Add** function.

Tip: If more tubes of one format are required for cutting plans than are available in the **Quantity** column, increase the number of tubes using the **Quantity -> New quantity** function from the shortcut menu.

The entries in the following columns can be modified by hand:

- Quantity
- Used
- Length
- Left edge
- Right edge

What each column means:

No.

Shows the number of the object.

Picture

Previews the object in a diagram.

Quantity

Shows the number of tubes that are available for nesting.

Used

Displays the number of tubes used.

> **Note:** If the quantity shown here is greater than the number of cutting plans displayed in the **Cutting plans** tab, one or more cutting plans are being cut in multiple machining cycles.

Length

Shows the length of the tube.

Left edge

Shows the left-hand edge distance for the tube; parts can be nested up to this distance.

Right edge

Shows the right-hand edge distance for the tube; parts can be nested up to this distance.

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13.4.2 Parts

How do I open this tab?

Select: **Modules > Tube Nester > Job data**

This tab contains a table showing all the parts in a job. All the functions in the **Part** group of the **Data** menu relate to this tab and can be run directly from within the group or via the shortcut menu.

The colors of the parts are assigned automatically by the software. You can change the color of a part manually to improve the view in the cutting plan using the **Color** item in the context menu.



Tip: You can use Drag&Drop to copy parts into another job, if this job has the same tube format.

Manual changes to the color of a part

1. Select the part from the **Parts** tab.
2. In the shortcut menu, click in the list box under the **Color** function.
3. Define the new color for the part in the color selection dialog box.

Copy part into a new job using drag & drop

1. Select the part to be copied from the **Parts** tab.
2. Click on the part with the left mouse button and drag it to the tab of the new job into which the part is to be copied. The job display switches to the selected job.
3. Drag the part into the workspace of the new job and then release the left mouse button. The part is copied into the job and is displayed in the Parts tab.

Adding parts using a barcode scanner

A barcode scanner can be used to add parts to the job. To do this, connect the bar-code scanner to the PC and scan in the barcode. The parts database is searched for the name of the part.

- If the part exists, it is entered in the **Parts** tab.
- If several parts are found with the same name, a dialog box is displayed in which you can select the part before adding it.



Note: When scanning in using the barcode, there must be no parts selected in the input box of the **Parts** tab.

Adding parts to a job

- Use the **Add** function from the parts database *(See: ›Add‹ Page 918)*.
- Add a part to the cutting plan by dragging and dropping with the mouse. If you also press the **Ctrl** key when adding a part, the active part is automatically nested. Filler parts cannot be added using Drag&Drop.
- Copy an existing part using the **Copy part** function *(See: ›Copy‹ Page 920)*.
- Create or import a new part using the **New part** function *(See: ›New part‹ Page 922)*.
- Replace an existing by a new part from the parts database using the **Replace part** function *(See: ›Replace‹ Page 922)*.
- Scan in a part using a barcode scanner.

Categories in the status display

The status display uses different colors and symbols to indicate the status of the parts. Each color and symbol is assigned to a specific category. The categories have the following relevance for nesting and exporting cutting plans:

Icon	Category	Description
	Color code	There are no restrictions on nesting or exporting the part. - **Green** The full quantity of the part has been nested. - **Yellow** Some of the specified quantity of the part has been nested. - **Red** The part has not been nested yet.
	Note	The part appears more than once in the table. The part can be nested and exported in the cutting plan.
	Warning	The part can be nested and exported in the cutting plan. - Contours in the cutting plan may overlap because of revision differences. In this case, check whether the nested plans containing the part need to be re-nested. - The part has contours without a cutting technology. Set the cutting technology on all contours in Part Editor. - The name of the part is changed.

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Icon	Category	Description
	Error	Automatic nesting cannot be carried out. If the part is already nested in a cutting plan, the part is not exported with the cutting plan at export.
	Information	Displays important status information about the part.
	Embedded	The part is embedded in the job. It is not saved in the parts database.

 **Note:** The color of the symbol in the tab indicates the lowest nesting status of all the parts.

Column entries that can be edited

- Quantity
- Priority
- Reverse allowed
- Roll allowed
- CommonCut

What each column means:

No.

Shows the number of the object.

Status

Displays the status of the parts in the nesting process.

Picture

Previews the object in a diagram.

Name

Shows the name of the part. The name is used for unique identification of a part.

Quantity

Shows the available quantity of a part.

Nested

Displays the nested quantity of a part.

Priority

You can use this option to set the order in which the different parts are to be nested. Entering a priority is optional. Enter a number between 1 and 99 for the priority, Where 1 stands for highest priority and 99 for lowest priority. No priority is assigned if 0 is entered.

Attention! The use of different priorities does not usually result in optimum nesting of the cutting plan.

Note: This function is only used for automatic nesting.

Reverse allowed

When nesting the part, it can be turned through 180° about the vertical axis *(See: ›Reverse part‹ Page 1288)*.

Roll allowed

When nesting the part, it can be rotated through a certain angle about the tubular axis. For circular tubes, every angle is possible for rolling the part. For rectangular or polygonal tubes, only certain roll angles are possible (e.g. rectangular tube = 180°, hexagonal tube = 60°).

Common Cut

This option allows the combining of parallel contours during nesting based on certain criteria. These criteria are defined in the export settings for the pre-processor in the Part Nester module *(See: ›Common Cut‹ Page 1153)*.

Requirements for common cuts:

- At least two segment curves run parallel to each other.
- The deviation in shape of the parallel curves remains below the maximum permitted deviation *(See: ›Max. permissible deviation‹ Page 1285)*.
- The **Common Cut** option is enabled for the parts in question.

Size

Displays the size dimensions of the tubular part.

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- The length and outside diameter are also displayed for round tubes.
- The length and outside diameter of the edges are displayed for rectangular tubes.

13.4.3 Cutting plans

How do I open this tab?

Select: **Modules > Tube Nester > Job data**

This tab contains a table showing all the cutting plans in a job. All the functions in the **Plan** group of the **Data** menu relate to this tab and can be run directly from within the group or via the shortcut menu. Clicking on a cutting plan will open the cutting plan in the plan view.

Tip: You can use Drag&Drop with the mouse to change the order of cutting plans within a job or copy cutting plans to another job, if this job has the same tube format. When copying to another job, all the parts in the cutting plan being copied are copied along with it.

Creating cutting plans

Cutting plans are created automatically by running the **Automatic** function in the **Cutting technology** menu. The number of cutting plans depends on the number of parts waiting to be nested and the nesting settings. A new cutting plan can be created in the **Tubes** tab by dragging a tube format into the cutting plan view of the Tube Nester.

Creating names for cutting plans

The names for cutting plans can be created automatically or manually. The name is used for the LCC or NCP file at export.

- **Automatic**

  Cutting plan names are created automatically by running the **Automatic** function. The name of the cutting plan is composed of the job name and a serial number.

- **Manual**

  For manually created cutting plans, the name of the cutting plan must be entered by hand in the **Name** field.

Note: If a name is not specified for the cutting plan, it is created automatically when the job or cutting plan is exported.

The entries in the following columns can be modified by hand:

- Name
- Name

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- Cycles
- Edge distances

What each column means:

No.

Shows the number of the object.

Picture

Previews the object in a diagram.

Name

Shows the name of the cutting plan.

Name

You can enter a more detailed description of the cutting plan in this field. This text is written into the LCC or NCP file at export and displayed on the machine in the **Plan view** window in the WORK menu.

Cycles

Displays the number of repetitions with which the cutting plan is cut.

Waste

Displays the percentage of total material wasted from a tube.

Left edge

Shows the left-hand edge distance for the tube; parts can be nested up to this distance.

Right edge

Shows the right-hand edge distance for the tube; parts can be nested up to this distance.

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13.4.4 Job

How do I open this tab?

Select: **Modules > Tube Nester > Job data**

This tab summarizes information about the various objects in the job. The value shown in the **Total waste** field is the average value of all waste figures from the cutting plans in the job.

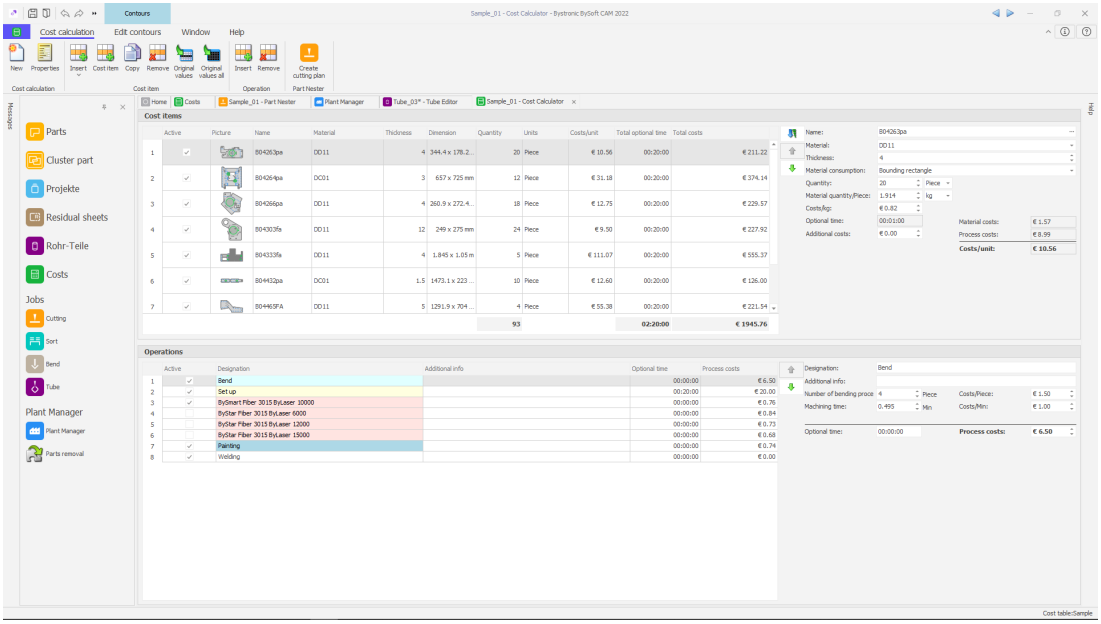


Note: The displayed values are for information purposes only and cannot be changed.

14 Module: Cost Calculator

Where can I find the module?

Select: **Modules > Cost Calculator**



The Cost Calculator is a module in which you can perform a cost calculation for one or several objects. The costs can be calculated in the Cost Calculator for the following objects:

- All parts in BySoft CAM
- Parts that exist only as a sketch or drawing
- User-defined cost items for use as required (e.g., programming costs, transport costs, etc.)

The cost calculation is divided into the **Cost item** and **Operation** areas. Each area contains the relevant input fields and selection boxes for the cost calculation.

Note: Any entries in the white boxes can be customized by the user. Once an option has been changed, all fields are automatically recalculated.

Tip: Click on an area of the header to call up the **Shortcuts** function in the context menu. The **Shortcuts** dialog can be used to define keyboard shortcuts for all functions of the current module.

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Tip:

- All the functions in the groups can also be launched via the shortcut menu in the relevant area.
- For recurring cost calculations using different machines, materials, thicknesses or currencies, save the cost calculation as a template *(See: ›Save as template‹ Page 1317)*.

Buttons

- = Moves the selected row up one place.
- = Moves the selected row down one place.

Example:

- *(See: ›Creating a cost calculation‹ Page 1351)*
- *(See: ›Adding multiple operations to a part‹ Page 1353)*
- *(See: ›Export cost calculation‹ Page 1352)*
- *(See: ›Creating user-defined contours‹ Page 1354)*

Cost items

This is the area where you add all the objects for which a cost calculation must be created.

Automatically calculate machining time

Machining time is calculated automatically when you insert a new cost item. If the properties of the cost item for machine, setup value, and parameter do not match the cutting operation, configure and calculate the settings for calculating machining time manually by pressing the **Machining time** button in the **Operations** section. In this case, an exclamation mark is shown above the image.

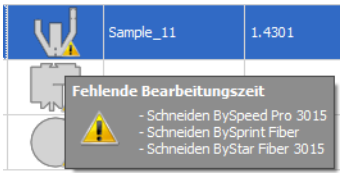


Fig. 14-1: Display of operations for which no automatic machining time could be calculated

Button

- = Enables column sorting.

  Click on the column header to sort the column alphabetically or numerically. The buttons used to move lines are hidden.

Calculating costs for a part from the Parts database

1. Select the **Insert part** function from the **Cost item** group.
2. In the **Add part** dialog box, select the part and confirm your selection with **OK**.
3. Enter in the **Quantity** field the number of parts for which you want to perform a cost calculation.
4. Insert the operation for cutting in order to calculate the process costs.
5. Select the **Calculate machining time** button.
6. In the **Machining time** dialog box, specify the **Cutting parameters** and the **Machine**, and confirm your selection with the **Calculate** button.
7. Add further operations if required, for instance for programming or bending.

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Calculating costs for a part from a drawing or sketch

1. Select the **Insert contours** function from the **Cost item** group.
2. In the **New cost item** dialog box, select the material and thickness to be used for the contours and confirm your selection with **OK**.
3. Select the **User-defined contours** menu.
4. Use the **Insert contour** function to add the necessary contours to the part to create the part in the drawing. Optionally, bends can also be added to the part.
5. Select the **Cost calculation** menu.
6. Enter a name for the part in the **Name** field.
7. Enter the thickness of the part in the **Thickness** field.
8. Enter in the **Quantity** field the number of parts for which you want to perform a cost calculation.
9. Insert the operation for cutting in order to calculate the process costs.
10. Select the **Calculate machining time** button.
11. In the **Machining time** dialog box, specify the **Cutting parameters** and the **Machine** and confirm your selection with the **Calculate** button.
12. Add further operations if required, for instance for programming or bending.

Calculating a user-defined cost item

1. Select the **User-defined** function from the **Cost item** group.
2. Enter a name for the user-defined cost item in the **Name** field.
3. Enter the quantity required in the **Quantity** field.
4. Enter the costs per unit for the cost item in the **Costs/unit** field.



Note: It is not possible to define operations for user-defined cost items.

What each field does:

- **Name**

  Name used to denote the cost items. The part name is used by default for parts from the Parts database.

  For a part from the Parts database, the part can be replaced by another by clicking the right-hand icon.

- **Material**

  Specifies the material from which the part is made. Any of the materials recorded in the Administrator can be selected.

- **Thickness**

  Specifies the thickness of the part. When parts have been added from the Parts database, the thickness of the part is adopted automatically.

- **Material consumption**

  Defines the method used to calculate the material consumption per part. This setting determines the weight of the part in the **Material quantity/unit** field and the resultant material costs.

  - **Effective**

    Calculates the effective area of the part excluding inside contours.

  - **Outer contour**

    Calculates the effective area of the part including inside contours.

  - **Bounding rectangle**

    Calculates the area of the bounding rectangle of a part. This is the default method.

  - **Including waste**

    Adds percentage sheet waste from a cutting plan to the part. This factors in actual material costs for the job. This option increases the value in the **Material quantity/unit** field and thus the costs per part.

    ☞ **Note:** This option is only available for parts of an imported job. Imported jobs must be contain nested cutting plans with a cutting technology in order to calculate waste and cutting time.

- **Quantity**

  Specifies the number of parts or user-defined cost items. These settings affect the costs in the **Total costs** column.

- **Material quantity/unit**

  States the calculated weight of the part.

- **Cost/kg**

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Indicates the cost per kg of material. This value is defined in the Cost table in the Administrator. It is multiplies by the **Material quantity** to give the **Material costs** for the part.

- **Optional time**

  Indicates the optional time per part. The optional time is taken from the sum of all activated operations in the **Operations** section.

- **Additional costs**

  Indicates additional costs for the calculation. These costs are added to the total sum of the cost calculation in the **Total Costs** column.

- **Material costs**

  States the material costs per part. The material costs are calculated from the product of the **Material quantity** and **Costs** fields.

- **Process costs**

  States the process costs per part. The process costs are obtained from the sum of all enabled operations in the **Operations** area.

- **Costs/unit**

  States the total costs for one part or one cost item.

What each column means:

Active

If this option is enabled, the associated cost item is included in the calculation. These operations can be enabled or disabled for each cost item selected.

Picture

Previews the object in a diagram.

Name

Name used to denote the cost items. The part name is used by default for parts from the Parts database.

Material

Displays the name of the material used.

Thickness

Shows the thickness of the part.

Dimensions

Shows the X/Y dimensions of the bounding rectangle for the part.

Quantity

The number of parts or user-defined cost items.

Unit

Units that are used for the quantity.

Costs/unit

Costs of the item in the relevant currency.

Total optional time

Specifies the total optional time of all instances of the part.

Total costs

Sum of the **Material costs** and **Process costs** fields.

Order

Shows the unique part number for identifying the order.

Description

Displays a more precise description of the part.

Weight

Shows the weight of the part.

Info 1-8

Displays further information about the part.

Operations

In this area you can add operations for each cost item. The operations are used for calculating the process costs.

For each new cost item, all operations entered are automatically inserted in the **Operations** section.

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☞ **Note:** To be able to calculate the process costs for a part from the Parts database, a cutting technology must be assigned to the part.

Calculating process costs for cutting

1. Select the **Insert** function from the **Operation** group.
2. In the **New operation** dialog box, select the operation for cutting and confirm your selection with **OK**.
3. Select the **Calculate machining time** button.
4. In the **Machining time** dialog box, specify the **Cutting parameters** and the **Machine** and confirm your selection with the **Calculate** button.
5. Add further operations if required, for instance for programming or bending.

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What each field does:

- **Name**
  States the name describing the operation.
- **Additional info**
  Field for entering additional information.
- **Cutting length**
  States the calculated cutting length based on the set cutting technology.
- **Number of piercings**
  States the calculated number of piercings based on the set cutting technology.
- **Machining time**
  States the calculated machining time based on the set cutting technology.
- **Costs/unit 1-3**
  States the costs per unit. The designation and values can be defined in the Cost table in the Administrator.
- **Number of bending processes**
  States the number of bending processes.
- **Optional time**
  Specifies an optional time for the operation.
- **Process costs**
  States the process costs calculated for the operation.

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What each column means:

Active

If this option is enabled, the associated operation is included in the calculation.

Name

Shows the name describing the operation. The operation types are distinguished by colors.

Color	Operation type
	Cutting
	Bending
	Set-up
	User-defined
	Area/Weight

Additional info

Shows the additional information for the operation.

Process time

Shows the process time for the operation.

Process costs

Shows the process costs for the operation.

Menus in Cost Calculator

- File functions
- Cost calculation
- Window
- Help
- Editing contours

14.1 File functions

The functions in this menu relate to the files and user interface of the module.



Functions for current cost calculation

These functions relate solely to the current cost calculation.

- Open
- Save
- Save as...
- **Close cost calculation:** *closes the current cost calculation. Before closing, a prompt dialog is displayed allowing you to save any changes not yet saved.*
- Save as template

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14.1.1 Cost Calculator properties



How do I open this dialog box?

Select: **Cost Calculator >** 

The fundamental properties forming the basis of the cost calculator are defined in this dialog box. This dialog is displayed when the following functions are performed:

- **New cost calculation** in the Cost Calculator module
- **New** in the **Edit** group
- **Properties** in the **Cost calculation** group
- **Create calculation** in the **Export** menu of the **Part Nester** module
- **Create calculation** in the **Auto Part** menu of the **Auto Part** module
- **Create calculation** on the **Orders** tab of the **Plant Manager** module
- **Create calculation** in the **Data** menu of the **Tool Setup Optimizer** module

A **Cost table** must be defined before a cost calculation can be performed. All the other details can be entered or modified later using the **Properties** function in the **Cost calculation** group.

What each field does:

General

- **Cost table**

  This field is used to specify the cost table to use for the calculation. You can create and edit cost tables in Administrator.

- **Name**

  The name of the cost calculation must be entered in this field.

- **Description**

  You can enter a more detailed description of the current object in this field.

- **Info 1–8**

  Extra information about the current object can be entered in these fields.

- **System of units**

  The system of units for the current object is defined in this field (mm/inches)

- **Currency symbol**

  Enter the currency symbol in this box. This symbol is used for all currencies in the cost calculation.

- **Positioning path**

The option takes the positioning paths into account for the calculation of costs. The option is enabled by default.

- **Machining time**

  This option displays the machining time in hours, minutes and seconds (e.g. 00:01:11). If the option is not activated, the time is displayed as a decimal in minutes (1.199).



Note: This setting only affects the machining time display in the **Operations** section.

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Template

Name

A template for the cost calculation can be selected in this field. All cost items and template settings are applied to the new cost calculation *(See: ›Save as template‹ Page 1317)*.

> **Note:** This option is only displayed when creating a new cost calculation.

14.1.2 Save as template



Where do I find the function?

Select: **Cost Calculator >** 

This function can be used to save the current cost calculation as a copy in the **Templates** folder. You can select the template in the **Cost calculator properties** dialog box when you create a new cost calculation.



Tip: Use templates for recurring cost calculations using different machines, materials, thicknesses or currencies.

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14.1.3 Print working documents



Where do I find the function?

Select: **Cost Calculator>** 

This function can be used to print out the working documents for the current calculation.

Printer

In this area you can select the printer and set your printer preferences.

Print

In this area you can specify preferences for the print layout.

- **Print template**
  Specifies what layout is used for the printed document.
- **Number of copies**
  Specifies how many copies of the document are printed.

Preview

This button previews the print document. The document can also be printed or emailed from the Preview dialog box.

14.2 Menu: Cost calculation

Where do I find the menu?

Select: **Modules > Cost Calculator**

This menu contains all the functions for creating or editing cost items and operations.

Functional groups

The function bar contains the following functional groups:

- Cost calculation
- Cost item
- Operation
- Part Nester

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14.2.1 Group: Cost calculation

Where do I find the group?

Select: **Modules > Cost Calculator > Cost calculation**

This group contains the functions for the Cost calculation preview and properties.

14.2.2 Group: Cost item

Where do I find the group?

Select: **Modules > Cost Calculator > Cost calculation**

This group contains the functions that can be used to create and edit cost items.

14.2.2.1 Insert



Where do I find the function?

Select: **Modules > Cost Calculator > Calculation > Cost item**

This function can be used to add different cost items to the cost calculation. The following types are available:

- **Insert part:** *Inserts a flat part from the Parts database.*
- **Insert tube:** *Inserts a tubular part from the Parts database.*
- **Insert contour:** *Inserts an empty parts template, which then must be assigned contours from the **User-defined contours** menu (See: ›Menu: Editing contours‹ Page 1325).*
- **Insert job:***Inserts all parts of a job from the job database. Use the **Including waste** option in the **Material consumption** section to calculate total material costs (See: ›Including waste‹ Page 1307).*
- **Import part:** *Inserts a part from a CAD drawing (See: ›Import part from CAD drawing‹ Page 1356).*

14.2.2.2 User-defined



Where do I find the function?

Select: **Modules > Cost Calculator > Calculation > Cost item**

You can use this function to create your own custom cost items for the cost calculation. Custom cost items might be administrative costs, transport, dispatch etc.

14.2.2.3 Copy



Where do I find the function?

Select: **Modules > Cost Calculator > Calculation > Cost item**

This function creates a copy of the selected cost item and its operations.

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14.2.2.4 Remove



Where do I find the function?

Select: **Modules > Cost Calculator > Calculation > Cost item**

Removes the selected cost item.

14.2.2.5 Original values



Where do I find the function?

Select: **Modules > Cost Calculator > Cost calculation > Cost item**

Restores all the values defined in the Administrator for the selected cost item and the associated operations.



Tip: Use this function to reset any manually modified values to the values defined in the Administrator.

14.2.2.6 Original values all



Where do I find the function?

Select: **Modules > Cost Calculator > Cost calculation > Cost item**

Restores all the values defined in the Administrator for all the cost items and the associated operations.



Tip: Use this function to reset any manually modified values to the values defined in the Administrator.

14.2.3 Group: Operation

Where do I find the group?

Select: **Modules > Cost Calculator > Cost calculation**

This group contains the functions that can be used to add or remove operations relating to the cost items.

14.2.3.1 Insert



How do I open this dialog box?

Select: **Modules > Cost Calculator > Calculation > Operation**

This function adds a new operation to the selected cost item. Operations are needed for calculating the process costs of a cost item.

14.2.3.2 Remove



Where do I find the function?

Select: **Modules > Cost Calculator > Calculation > Operation**

Removes the selected operation.

14.2.3.3 Machining time

How do I open this dialog box?

Select: **Modules > Cost Calculator > Cost calculation > Operations**

This function calculates the machining time for the current part and shows the result in various tables. Machining times are calculated for the entire job and for each cutting plan and part present in the job. If a calculation has not been performed yet, just the table header is displayed.

☞ **Note:** The machining time can only be calculated if the **Cutting parameters**, **Setup values** and **Machine** fields each contain an entry.

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14.2.4 Group: Part Nester

Where do I find the group?

Select: **Modules > Cost Calculator > Cost calculation**

This group contains the function that can be used to open cost items in Part Nester.

14.2.4.1 Create cutting plan



How do I open this dialog box?

Select: **Modules > Cost Calculator > Calculation > Operation**

This function opens the selected cost items in Part Nester so that cutting plans can be created.

Note: The cost items must have the same material and thickness.

14.3 Menu: Editing contours

Where do I find the menu?

Select: **Modules > Cost Calculator**

In this menu you can add contours or bend lines to a user-defined cost item.

Use the contours to reproduce in the cost item the dimensions and contours of a sketch of the part. The largest contour is used as the outside contour.

The Cutting length, Number of piercings and the Machining time are calculated from the individual contours. For bending parts, the number of bending processes required can be added to the cost item.



Note: The accuracy of the calculation depends on how accurately the sketch can be reproduced using contours.

Functional groups

The function bar contains the following functional groups:

- Contour

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14.3.1 Group: Contour

Where do I find the group?

Select: **Modules > Cost Calculator > User-defined contours**

This group contains the functions that can be used to add or remove user-defined contours or bends.

14.3.1.1 Insert contour



Where do I find the function?

Select: **Modules > Cost Calculator > User-defined contours > Contour**

This function adds to the cost item the contour selected in the **Shape** area together with the associated parameters.

14.3.1.2 Insert bending process



Where do I find the function?

Select: **Modules > Cost Calculator > User-defined contours > Contour**

This function adds a bending process as a contour element to the part. The number of bends for the part can be specified in the **Shape** area.

14.3.1.3 Remove



Where do I find the function?

Select: **Modules > Cost Calculator > User-defined contours > Contour**

Removes the selected contour or bend.

15 Examples

The examples described here are intended to help you work with the software and learn application scenarios.



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15.1 General

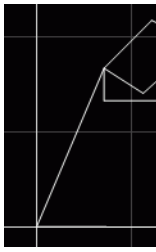
These are application examples which are not specific to only one module.

15.1.1 Resolving a branch point during DXF import

If a contour branches into different segments, this can create a situation which the program cannot resolve when computing the contour. In this case, the user must show the program the correct route to take.

In this example, a part is imported that contains a branch point between several geometry elements.

1. After selecting the function **Create part without selection**, the following message appears: ***Branch encountered. Please select the path to continue along.***



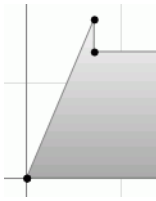
2. Confirm the message with **OK**. The display zooms onto the branch point. The possible paths for continuing the contour are shown as blue lines. Clicking on a blue line tells the program which path to follow for contour recognition.



4. Once all the branch points have been defined, the part is imported into Part Editor. Afterwards, any geometry segments that are no longer needed can be deleted. If this is not possible because the contours are joined, the geometry segments that are no longer needed must be deleted in the external CAD program.



5. The part can now be assigned a technology and saved in the database.



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15.1.2 Laser process macros

Process macros refer to machining modes of the laser beam for machining materials.

Name	No.	Application
CW large contours	0	This macro is used by default for large contours.
Pulsed cutting	1	This macro is used for pulsed cutting in sharp corners and around small radii.
Engraving	2	This macro is used for engraving contours.
CW small contours	3	This macro is used for cutting at reduced feed rate and power for small inner contours.
Lead-in	4	This macro is used for all lead-in cuts in thick sheets. It increases process reliability. ☞ **Note:** On Byspeed machines, this macro is used for the Scanning machining mode.
ATS	5	This macro reduces the feed rate in corners and tight radii (**BySprint Pro only**).
Scanning	6	This macro is used for the Scanning machining mode (**Byspeed and BySprint Pro only**).
Vaporization	7	This macro is used for vaporization of films.

15.1.3 Water process macros

Process macros refer to machining modes of the water jet for machining materials.

Name	Num.	Application
Extra quality cut (X)	1	This macro produces a high-quality cut edge that exhibits practically no drag or corrugation down to the lower edge. Since the feed rate is very slow, this machining mode is only suitable for cutting edges of particularly high quality.
Quality cut (Q)	2	This macro produces a high quality cut edge. A cut made at optimum feed rate is referred to as a quality cut. Apart from a small drag and a slight waviness of the cut edge that may become more distinct towards the bottom edge, a quality cut does not display any irregularities.
Production cut, fine (M)	3	This macro uses the parameters in macros P and Q and takes the arithmetic mean to calculate the feed rate and of the quantity of abrasive.
Production cut (P)	4	This macro is the ideal compromise between cutting quality and cutting speed. A production cut produces moderate drag and corrugation along the cut edge, which may become slightly more pronounced towards the lower edge.
Separation cut (S)	5	This quality grade is primarily used for quick cutting of leftover sheets and for those parts that do not need high quality cut edges. A separation cut has a distinct drag and severe irregularities along the cut edge. It is therefore only fit for limited use.

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15.1.4 Customizing the software user interface

The software user interface can be customized by a range of options, which let you modify views and dialog boxes in the software to suit your personal preferences.

The display preferences are saved individually for each user. When a new user logs in, the following user-specific display preferences are loaded:

- Column appearance in tables
- Arrangement of docking dialog boxes

Adjusting the size of views

The size of different window panes within a view can be changed using split bars. These split bars may run vertically or horizontally.

To change the size of a pane, move the mouse to the edge of the pane until the cursor changes into a double arrow  . With the left mouse button pressed, drag the pane to the required size. The size of the pane is saved.

Adjusting the size of dialog boxes

The size of some dialog boxes can be changed by dragging their edges with the mouse. The horizontal and vertical size can be changed simultaneously by dragging a corner of a dialog box. The size of the dialog boxes is saved.

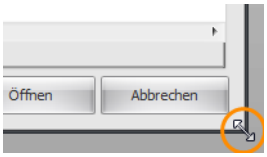


Fig. 15-1: Changing the size of a dialog box using the mouse

Setting the language version

The language version of BySoft CAM used is determined by the language setting in the Regional and Language Options of the Windows operating system. If the selected language is not supported by BySoft CAM, the program runs in English.

Note: When BySoft CAM is installed, the language versions installed by default are German, English and the language set for the operating system.

Tip: If you wish to install another language later, run Setup again and select the language you require during Setup under the **User defined** option.

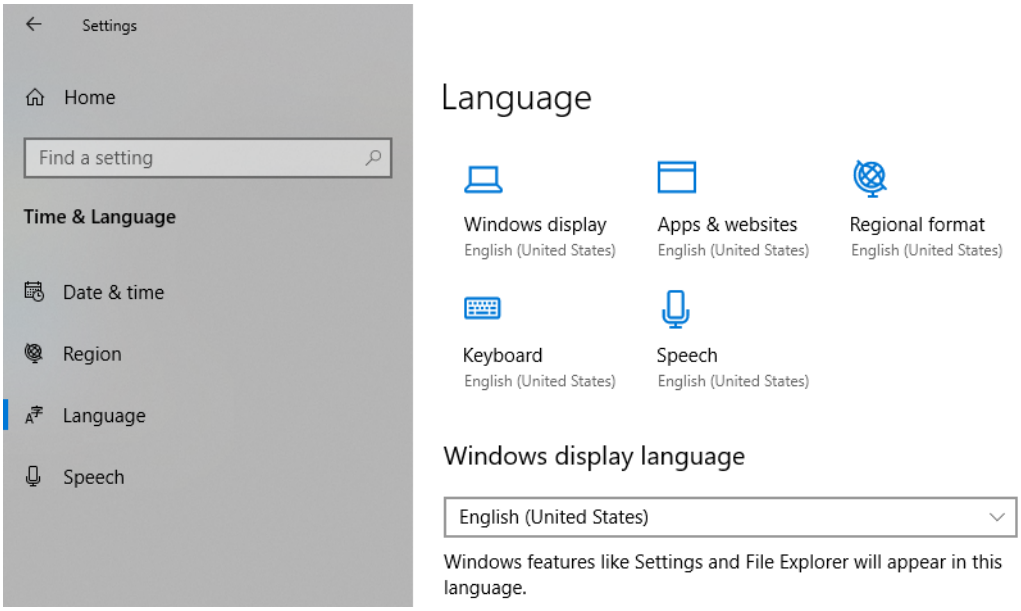


Fig. 15-2: Language setting on the “Language” tab

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15.1.5 Customizing table columns

You can customize the appearance of table columns in the various tabs or dialog boxes to suit your own requirements. To change the table layout, right-click in the table header and select the relevant function from the shortcut menu.

Note: Some functions are not available depending on the type of table.

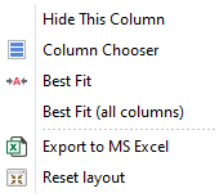


Fig. 15-3: Functions in the shortcut menu for tables

Hiding a table column

You can use this option to hide columns that are not needed in a table.

1. Right-click in the header of the column that you wish to hide.
2. Select the **Remove this column** option from the shortcut menu.

Showing a hidden table column

Use this option to reveal hidden columns in the table.

1. Right-click in the column header of the table and select the **Column chooser** option from the shortcut menu.
2. In the **Customize** dialog box, select the column and, holding down the left mouse button, drag the column to the required position in the table.

Adjusting the width of a table column

This option adjusts the width of the selected column to fit the contents.

1. Right-click in the column header of the column whose width you wish to adjust.
2. Select the **Optimum column width** option from the shortcut menu.

Adjusting the width of all table columns

This option adjusts the width of all the columns in a table at once to fit their contents.

1. Right-click in the column header of the table.
2. Select the **Optimum width (all columns)** option from the shortcut menu.



Note: To restore the default table layout, select the **Reset layout** option from the shortcut menu.



Tip: Hide unwanted columns if the contents of columns cannot be displayed in full because of the number of columns being shown.

Exporting tables to MS Excel

This option can be used to export the entire table to MS Excel.

1. Right-click on the column header.
2. Select the **Export to MS Excel** option from the shortcut menu.
3. Enter the directory and file name in the Export dialog.
4. Press **Save** to confirm.

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15.1.6 Arranging docking dialog boxes

All docking dialog boxes can be positioned wherever you want around the edges of the software view. Multiple docking dialog boxes can also be joined together. Tabs are then used to select the relevant dialog box.

☞ **Note:** The size and display settings for the dialog boxes are retained after closing the software. You can only move docking dialog boxes when they are permanently displayed.

Moving dialog boxes

1. Click in the header of the dialog box and hold down the left mouse button.
2. With the mouse button still pressed, drag the dialog box into another area of the software view. Various position buttons appear in the middle and at the edges of the software view.
3. Drag the dialog box over the position button of the position you require. A blue frame is displayed as a preview at the selected position.
4. Release the mouse button. The dialog box is docked in this position.

Position buttons

Various position buttons are provided for positioning docking dialog boxes in specific locations of the software view or for combining them.

Icon	Description
	Button cross in the area of the left-hand docking position. The dialog box is docked at the position of the current dialog box indicated by the arrow position. The button in the center lets you combine the docking dialog with the underlying dialog box as tabs. **Note:** At least one docking dialog box must be permanently displayed for the button cross to appear. **Tip:** Combine several dialog boxes as tabs as a space-saving way of integrating several dialog boxes in the user interface. **Attention!** The Help dialog box cannot be combined with other dialog boxes because then the context-sensitive help will not work.
	Button cross in the area of the module view. The dialog box is docked within the module view at the position indicated by the arrow position.
	Docks the dialog box above the module view and the side dialog boxes. If a dialog box is already displayed here, the dialog box is docked above the existing dialog box.
	Docks the dialog box at the bottom of the software view. If a dialog box is already displayed here, the dialog box is docked below the existing dialog box.
	Docks the dialog box at the left edge of the software view. If a dialog box is already displayed here, the dialog box is docked to the left of the existing dialog box.
	Docks the dialog box at the right edge of the software view. If a dialog box is already displayed here, the dialog box is docked to the right of the existing dialog box.

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15.1.7 Create serial-number information

This example explains how the total quantity of a single part in a cutting plan can be marked with a unique serial number.

Placing serial-number information on a part within a cutting plan

1. Open the part in the Part Editor and select the **Insert text** function from the **Geometry** menu.
2. Select the **Serial-number information** in the **Text** field.
3. Adjust the size and position of the serial-number information in the **Formatting** section.
4. Position the data field at the insertion point over the part using the mouse.
5. Click on the insertion point with the left mouse button to place the data field onto the part.
6. Place the cutting technology onto the part from the **Cutting technology** menu.
7. Open the job in which the part is to be nested.
8. Specify the desired quantity for the part in the **Quantity** column of the cutting plan.
9. Use the mouse to select the part.
10. Select the **Automatic** function from the **Nesting** menu in the **Part** group.
11. In the **Parts** tab, click on the part and select the **Properties** context-menu function.
12. You can enter a text to be prefixed to the number in the **Preceding text** field. In order for the numbering to begin with **1**, the **Start number** field must contain the value of **1**.
13. To place the serial-number information on all parts, select the **Automatic** function from the **Cutting technology** menu in the **Serial-number information** group.
14. The serial-number information of one or more parts can be changed manually to mark certain parts - for example as test parts for quality control inspection. To manually change the serial-number information of certain parts, first select the part and then the **Manual** function.
15. Enter the text to identify the part in the **Serial-number information** box (e.g. “Test part”).
16. The cutting plan can now be assigned a cutting technology and then be exported.



Note: The serial-number information texts are always executed as engravings.



Tip: Serial-number information can be used to uniquely mark a series of parts.

15.2 Administrator

These examples show different procedures having to do with the creation of elements in Administrator.

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15.2.1 Creating a new material

New materials can be created in the **Materials** form in the Administrator. You have two options for creating materials.

- Via the button. The settings for the new material are adopted from the selected material; any necessary changes to the settings can then be made.
- Via the button. The settings for the new material must be created from scratch.

Copying an existing material

1. In the Administrator module, select the **Material** form.
2. In the **Available entries** pane, select the material that will be saved under a new name.
3. Select the button.
4. Enter the name of the new material in the **Name** field.
5. Change any options that you need to modify for the new material.
6. Press **OK** to confirm the selection. The new material is created under **Available entries**.

Creating a new material

1. In the Administrator module, select the **Material** form.
2. Use the button to add a new object to the **Available entries** pane.
3. Enter the name of the new material in the **Name** field. You have the option to enter extra information about the material in the **Description** and **Info** fields (e.g. type of material, surface finish, etc.).
4. Select the system of units being used.
5. Enter the density of the material in the **Density** field. This information is needed for calculating the weight of the parts.
6. In the Cutting tables selection field, assign the cutting table that applies to the material. If the material will also be used for bending parts, assign the appropriate bending table to this material in the **Bending table** field.
7. Set the output folder for the cutting plans for the machine.

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15.2.2 Creating a new cutting table

You have two options for creating new cutting tables.

- Via the button. The settings for the new cutting table are adopted from the selected cutting table; any necessary changes to the settings can then be made.
- Via the button. The settings for the new cutting table must be created from scratch.

Copying an existing cutting table

1. In the Administrator module, select the **Cutting tables** form.
2. In the **Available entries** pane, select the cutting table that will be saved under a new name.
3. Select the button.
4. Enter the name of the new cutting table in the **Name** field.
5. Change any options that you need to modify for the new cutting table.
6. Press **OK** to confirm the selection. The new cutting table is created under **Available entries**.

Creating a cutting table for a material

1. In the Administrator module, select the **Cutting tables** form.
2. Use the button to add a new thickness to the **Available entries** pane.
3. Enter a name for the new cutting table in the **Name** field.
4. You have the option to enter extra information about the cutting table in the **Description** and **Info** fields.
5. In the **Filter for cutting parameters** field, enter the filter for the relevant cutting parameters.
6. In the **Cutting Wizard filter** field, enter the filter for the appropriate Cutting Wizard.
7. Enter the system of units for the material in the **System of units** field.
8. Click the button in the **Cutting tables** area.
9. Enter a thickness for the material and confirm with **OK**.
10. For the thickness, enter in the relevant fields the parts protective distance, the distance of nested parts from the sheet edge and the filter for the cutting parameters.
11. Repeat steps **8** to **10** until you have recorded all the thicknesses.

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15.2.3 Creating a new cutting machine

You have two options for creating a new cutting machine.

- Via the button. The settings for the new cutting machine are adopted from the selected cutting machine; any necessary changes to the settings can then be made.
- Via the button. The settings for the new machine must be created from scratch.

Requirements:

- It must be possible to access the Setup values and Cutting parameters for the machine concerned via a network.

Creating a new cutting machine

1. In the Administrator module, select the **Cutting machine** form.
2. Use the button to add a new object to the **Available entries** pane.
3. Enter the name of the new machine in the **Name** field. You have the option to enter extra information about the machine in the **Description** and **Info** fields (e.g. site, production line, etc.).
4. Select the correct machine type from the **Machine type** field. The associated cutting process and sheet dimensions are shown in the Properties area.
5. After specifying the system of units, select the form of sheet clamping. If the machine has no sheet clamping, select the attribute **None**.
6. In the **Max. thickness sheet clamping**, specify the sheet thickness up to which sheet clamping shall be used.
7. Select the applicable cutting processor for the machine.
8. Set the output directory for the cutting plans for the machine. The directory should be located on a network so that the machine controller can access it.
9. Set the path to the file in which the machine setup values are saved (*.stp).
10. Set the path to the directory containing the machine cutting parameters.
11. Enter as filters for the Cutting Wizard the name of the machine and the laser power. The filter is needed to limit the choice of Cutting Wizard in the Part Nester module.
12. Select the **Cutting wizard** form.
13. Save the new machine via the prompt dialog box.
14. In the **Available entries** pane, check the name of the filter specified for the cutting wizard.

Copying an existing cutting machine

1. In the Administrator module, select the **Cutting machine** form.
2. In the **Available entries** pane, select the machine that will be saved under a new name.
3. Select the button.
4. Enter the name of the new machine in the **Name** field.
5. Change any options that you need to modify for the new cutting machine.
6. Press **OK** to confirm the selection. The new cutting machine is created under **Available entries**.

Creating a cutting machine fitted with CNC96 controller

The procedure for creating a machine fitted with CNC96 controller is identical to the procedure described under **Creating a new machine** except for two pieces of information in the **Cutting processor** and **Conversion file CNC96** fields.

Note: The **Setup.cfg** file in the machine **Setup** folder must be selected for the setup values.

Specifying the cutting processor

In the **Cutting processor** field, select the **Bystronic CNC96** option. This enables the **Conversion file CNC96** field.

Selecting the CNC96 conversion file

1. Click on the icon in the **Conversion file CNC96** field.
2. In File Explorer, select the **Cnc96Mapfiles** folder.
3. Select the appropriate map file for the machine concerned and confirm the selection with **OK**.

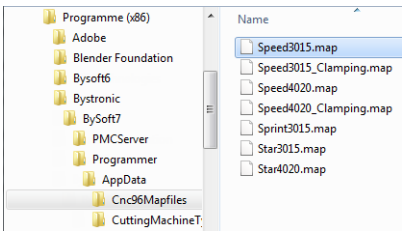


Fig. 15-4: Path to the **Cnc96Mapfiles** folder in Windows Vista / 7

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Calculating the machining time

Various information must be entered in the **Cutting machine** form so that the software can calculate the machining or cutting time for the parts and cutting plans.

Entering the details required to calculate the machining time

1. In the **Setup values** field, select the setup file that contains the setup values for the machine.
2. In the **Parameter path** field, select the path to the cutting parameters for the machine.
3. In the **Filter for Cutting Wizard** field, enter the first two terms denoting the machine and laser power that appear in the name of the Cutting Wizard. The available Cutting Wizards are listed in the **Cutting wizard** form in the **Available entries** pane.

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15.2.4 Creating a new machine part

This example shows how to create a drawing template for a machine part.

Creating a drawing template for a support grate

1. In the **Administrator** module, select the **Machine parts** form.
2. Use the  button to add a new object to the **Available entries** pane.
3. Enter the name of the support grate in the **Name** field. You have the option to enter additional information about the support grate in the **Description** and **Info** fields (e.g. size, machine, production line, etc.).
4. Enter the valid system of units.
5. Press **Import geometry** to import the geometry data of the support grate. The imported geometry is displayed below the button.
6. Select the **Save** function to save the new machine part.

☞ **Note:** Import drawing templates for suction-cup frames and sheet clamping in the same way as described in this example.

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15.3 Cost calculation

These examples show different procedures relating to the Cost Calculator module.

15.3.1 Creating a cost table

Cost tables are needed for the cost calculation in the Cost Calculator module. They can be created and edited in the Administrator.

In this example you will learn:

- How to create a cost table.
- How to add an operation to the cost table.
- How to automatically create for an operation a material including all thicknesses.
- How to assign material and operation costs to the material.



Tip: In the **Available entries** area create a separate cost table for each currency required. This allows you to create cost calculations in your own currency or a foreign currency.

Creating a cost table

1. In the **Available entries** area, select the button for creating a new cost table.
2. Enter the name of the cost table in the **Name** field.
3. In the **System of units** field, specify the system of units used.
4. In the **Currency symbol** field, specify the currency symbol for the currency used.
5. In the **Operation costs** area, select the button for creating a new operation.
6. Select the **Cutting** operation from the **Select operation type** dialog box.
7. Press **OK** to confirm the selection.
8. Select the **Automatic** function from the menu bar to insert the materials for the operation and material costs.
9. Select the material and associated thicknesses from the **Cost table** dialog box.
10. Assign the material selection to the **Cutting** operation.
11. In the **Material costs** area, specify for the material the costs for the selected unit.
12. Press **OK** to confirm the selection.
13. Enter in the **Name** field the name of the machine that applies to the **Cutting** operation.
14. In the **Operation costs** area, specify the costs per unit 1-3 for each thickness of the material.
15. Use **Ctrl+S** to save the cost table.
16. Continue to import materials and assign costs until you have recorded all the materials needed for the **Cutting** operation.

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15.3.2 Generating operation for programming

This example shows you how easy it is to represent the costs for programming in a cost table in the Administrator.



Note: Since programming costs are incurred only once per part, the programming costs must be divided by the number of parts produced. Therefore use the **Set up** operation for programming.

Generating operation for programming

1. Select the  button.
2. Select the **Set up** operation from the **Select operation type** dialog box.
3. Press **OK** to confirm the selection.
4. Enter the designation “Programming” in the **Name** field.

The operation for programming is now available for the calculation in the Cost Calculator. Programming costs must be input in the cost calculation in Cost Calculator.

15.3.3 Generating operation for drilling

This example shows you how easy it is to represent the costs for drilling in a cost table in the Administrator.

The costs for drilling are specified for the drilled depth, the drilled diameter and the basic cost per drilled hole.

Description of costs:

- **Depth:** extra cost for the drilled depth in mm.
- **Diameter:** extra cost for the drilled diameter in mm.
- **Quantity:** basic cost per drilled hole for the relevant material thickness.



Note: The costs for depth and diameter are added to the basic cost of the drilled hole.

Step 1: Creating the operation

1. Select the  button.
2. Select the **User-defined** operation from the **Select operation type** dialog box
3. Press **OK** to confirm the selection.
4. Enter the designation “Drilling” in the **Name** field.
5. Enter the term “Depth” in the **Label 1** field.
6. Enter the units for the depth in the **Unit 1** field.
7. Enter the term “Diameter” in the **Label 2** field.
8. Enter the units for the diameter in the **Unit 2** field.
9. Enter the term “Quantity” in the **Label 3** field.

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Step 2: Generating the operation costs

By generating the operation costs for drilling, the user then only needs to enter the number of drilled holes in the cost calculation.

1. Select the “Drilling” operation.
2. Select the **Automatic** function from the **Edit** group.
3. Select the materials and associated thicknesses in the **Cost table** dialog box.
4. Assign your material selection to the “Drilling” operation.
5. Press **OK** to confirm the selection.
6. In the **Operation costs** area, specify the costs per unit 1-3 for each thickness of the material.
7. Save the cost table.

The operation for drilling is now available for the calculation in the Cost Calculator.

Note: To calculate the drilling costs according to the material thickness, the value **1** must be entered in the **Depth** and **Diameter** fields for the **Drilling** operation in the Cost Calculator.

Tip: Export the operation costs into Excel as a table so that you can edit the values more easily *(See: ›Editing table values in Excel‹ Page 227)*.

Note: Since the operation costs can also be entered manually in the Cost Calculator, the second step of generating the operation costs is not essential.

15.3.4 Creating a cost calculation

This example shows how to easily create a cost calculation for a part.

In this example you will learn:

- How to create a new cost calculation.
- How to add a cost item to the cost calculation.
- How to calculate the process costs for the cutting operation.

Creating a cost calculation for a part

1. Select the **New cost calculation** function in the **Cost Calculator** module.
2. In the **Cost calculation properties** dialog, select the cost table and enter the name of the cost calculation in **Name**.
3. Confirm your entries with **OK**.
4. Add a new part of the cost calculation in the **Cost item** section using the context function **Insert part**.
5. Enter in the **Quantity** field the number of parts you need to produce.
6. In the **Operations** section, use the **Insert** context function to add the operation for cutting to the cost calculation.
7. If part machining time has not been calculated in Part Editor yet, press the **Calculate machining time** button.
8. In the **Machining time** dialog, enter the parameter and the machine with which to cut the part.
9. Press **Calculate** to perform the calculation.
10. The process costs of the operation are added to the material costs and equal total costs per part.



Note: Add additional parts to the cost calculation if a job consists of multiple parts.

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15.3.5 Export cost calculation

This example shows how to export a cost calculation to MS Excel for use in a quotation.

Add operation

1. Right-click in the column header of the table in the **Cost items** section.
2. Select the **Export to MS Excel** function from the shortcut menu.
3. In the dialog, enter the name of the Excel file and save the file in the appropriate folder.



Tip: If you create your quotations in Excel too, you can link the quotation with the total costs field. This allows you to calculate several versions without having to copy the costs to the quotation every time.

15.3.6 Adding multiple operations to a part

This example shows how to easily create multiple operations for different process costs in a cost calculation.

In this example you will learn how to calculate the process costs of the following operations:

- Bending
- Programming
- Set-up

Add operation

1. In the **Operations** section, use the **Insert** context function to add the operation for bending to the cost calculation.
2. If the bending technology is assigned to the part and the costs per bend have been entered in the operation, the number of bending processes and the costs per bend are automatically filled in.
3. In the **Operations** section, use the **Insert** context function to add the operation for programming to the cost calculation.
4. Enter the programming costs for the part in the **Process costs** box.
5. In the **Operations** section, use the **Insert** context function to add the operation for set-up to the cost calculation.
6. Enter the costs for set-up of the part in the **Process costs** box.

The process costs of the part are calculated using the process costs of all operations.

- The process costs for cutting and bending are added directly to the process costs per part.
- Because the process costs for programming and set-up only arise once, they are divided by the number in **Quantity** and the result is added to the process costs per part.



Note: Add additional parts with the appropriate operations to the cost calculation if a job consists of multiple parts.



Attention! Because the set-up costs only arise once for all cost items, use the **User-defined** cost item in the **Cost item** section for set-up costs for multiple parts.

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15.3.7 Creating user-defined contours

This example shows how to easily create a rough cost calculation for a part which exists only as a drawing.

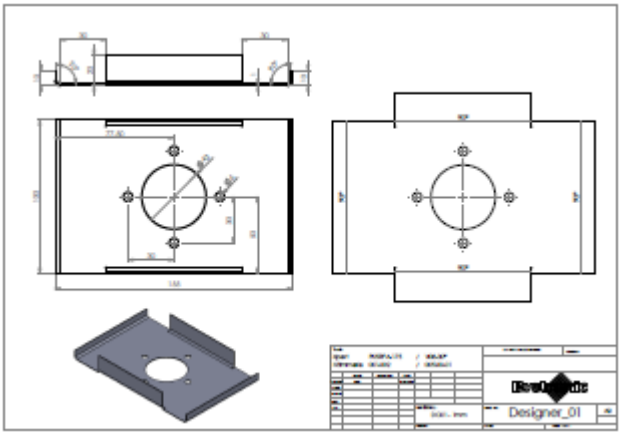


Fig. 15-5: Technical drawing of a bending part

Creating a cost calculation for a part from a drawing

1. Select the **New cost calculation** function in the **Cost Calculator** module.
2. In the **Cost calculation properties** dialog, select the cost table and enter the name of the cost calculation in **Name**.
3. Confirm your entries with **OK**.
4. Add a new part of the cost calculation in the **Cost item** section using the context function **Insert contours**.
5. Enter the material and thickness of the new part in the **New cost item** dialog.
6. Confirm your entries with **OK**.
7. Enter the drawing name of the new part in the **Name** field.
8. In the **User-defined contours** menu, select the **Insert contour** function to insert the **Rectangle** shape.
9. Enter the length and width of the rectangle surrounding the part from the drawing.
10. If the part contains any inside contours or bend lines, add them using the **Insert contour** or **Inset bending process** function.
11. Select the **Cost calculation** menu.
12. In the **Operations** section, use the **Insert** context function to add the operation for cutting to the cost calculation.
13. Press the **Calculate machining time** button to calculate the process costs for cutting.
14. In the **Machining time** dialog, enter the machine and cutting parameters with which to cut the part.

15. Press **Calculate** to perform the calculation. The **Costs/unit** box displays the costs of one part.
16. Add additional operations to the part.

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15.3.8 Import part from CAD drawing

This example shows how to import a part from a CAD drawing into Cost Calculator for calculation.

Importing a part from a CAD drawing into Cost Calculator

1. Select the **Import part** function.
2. In the **Import part** dialog box, select the CAD drawing and press the **Open** button to confirm import.
3. In the **Part properties** dialog box, set the unit and material properties of the part.
4. In the drawing, use the selection and view functions to select the contour of the part that you wish to import. You can use the **Shift key + left mouse-click** to add additional contours to the selection.
5. Use the **From selection** function to import the part.
6. Use the **Save** function to save the part in the Parts database. The part is entered in Cost Calculator as a cost item.



Note: In order to calculate cutting time for the imported part, the cutting technology must be assigned to the part.

15.4 Jobs

These examples show different procedures having to do with the creation of jobs.

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15.4.1 Importing a job from Bysoft 6

This example shows you how to import a job from Bysoft 6 into BySoft CAM. In order to be able to import the job, you must first export the job from Bysoft6 as a ZIP file via Job Administration.

The material and parts for the job are embedded in the job. If the job cannot be produced using the embedded material, you must assign a valid material to the job using the **Properties** function in the **Data** menu.

☞ **Note:** The cutting technology for the job is imported at the same time.

Importing a job from Bysoft6

1. In the **Modules** dialog box, select the **Import job** function.
2. In the **Import job** dialog box, select the ZIP file of the exported Bysoft 6 file and confirm your selection with **Open**.
3. Confirm the information given in the **Job properties** dialog box with **OK**. The job is imported into the Part Nester module.
4. Run the **Verify plan** function from the **Export** menu to check the cutting technology for the part.
5. Use the **Save** function to save the job in the job database.

Saving embedded parts in the parts database

Parts embedded in an imported job can be saved in the BySoft CAM parts database using the **Extract** function.

☞ **Note:** The procedure shown in the example must be carried out for each part that you wish to save in the parts database.

Saving in the parts database parts from an imported job

1. In the **Parts** tab, select the part that you wish to save in the parts database.
2. Select the **Edit** function from the shortcut menu. The part is opened in the Part Editor module.
3. Select the **Cutting technology** menu.
4. Confirm the information in the **New cutting technology** dialog box with **OK**.
5. Use the **Check** function to check the cutting technology for the part.
6. If the part does not contain any errors, save it in the Parts database using the **Save as** function. The part can now be used for creating jobs.

15.4.2 Creating job templates

You can create job templates for recurring jobs with the same settings. Job templates allow you to create a new job without having to modify settings every time.

Creating job templates

1. Select the **New job** function.
2. In the **Job properties** dialog box, specify the machine, material and thickness for the job template.
3. Confirm your entries with **OK**.
4. In the **Cutting technology** menu, specify the job-nesting settings.
5. In the **Export** menu, specify the job-export settings.
6. In the **Machining time** menu, specify the cutting parameters for automatic calculation of the machining time.
7. From the File functions, select the **Save as template** function.
8. Save the template in the appropriate folder under the name shown in the **Name** field.

Next time you create a new job you can choose the template in the **Template** section of the **Job properties** dialog.



Tip: You can create job templates for all thickness and materials.

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15.4.3 Creating Common Cut cutting plans

Common Cut lets you machine two edges of separate parts in one cut. Common cuts are generated by the preprocessor during export of the cutting plan, but not before this stage.

The Common Cut function searches a cutting plan for those parts that are nested at a kerf-width apart. A new part is then generated from these close-lying parts. The common cuts produced in this process become inner contours of the new part. The new part also contains all the inner contours of the individual parts.

☞ **Note:** The Common Cut function only works for straight cuts. To obtain optimum results, identical parts should always be grouped together. Common cuts have the advantage of a shorter machining time and less waste.

Nesting settings for Common Cuts

In order to be able to use the Common Cut function, outside contours must be nested against each other at a separation equal to the kerf width. Kerf width is configured in the job properties on the **Cutting technology** tab.

Ranges for parts protective distance

- CO_2 laser = 0.2-0.5 mm
- Fiber laser = 0.1-0.6 mm
- Water = 0.8 mm

To obtain precision parts when cutting common cuts, the **Parts protective distance** must equal twice the tool radius for the material and thickness being used. The tool radius is specified in each cutting parameter file and can be found on the machine's Panel PC in the **Parameter Editor** software or via the network using a standard Editor program.

Finding the tool radius

Finding the tool radius in the Parameter Editor

1. On the machine Panel PC, run the **Parameter Editor** software.
2. In the Parameter Editor, open the parameter file for the relevant material and thickness.
3. Search for the **ToolRadius** entry in the **WorkSectionDynamic** section.

☞ **Note:** The operating mode must be set to **CW** in this case.

Finding the tool radius via a network

If you have the facility to access the cutting parameters directly from the workstation via a network, you can open the parameter file in a standard Editor and search for the text **ToolRadius** (see figure) The value highlighted in the box is the tool radius in millimeters.

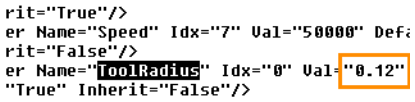


Fig. 15-6: Displaying the tool radius in Windows editor

Nesting Common Cut parts

The nesting of Common Cuts takes into account the options **Min. length Common Cut** and **Parts protective distance** in the nesting settings. Parts can be nested automatically or manually.

Requirements:

- The outer contour of the part has a straight side whose length equals at least the value given in the field **min. Length Common Cut**.
- The part has been assigned the **Common Cut** attribute in the **Parts** tab of the **Job file** docking dialog box.

Automatic nesting

1. In the **Parts** tab of the Part Nester, select the part that you wish to be cut with Common Cut.
2. Assign the **Common Cut** option to the part in the appropriate column.
3. In the **Cutting plans** tab, use the **Add plan** function to create an empty cutting plan.
4. In the **Cutting technology** menu, run the **Automatic** function with the **Part** sub-function. The part is nested in the cutting plan.
5. Use the **Simulator** function from the **Export** menu to check that the common cuts have been applied correctly.

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Manual nesting

1. Select the **Settings** function from the **Cutting technology** menu.
2. In the **Cutting plans** tab, use the **Add plan** function to create an empty cutting plan.
3. In the **Parts** tab of the Part Nester, select the part that you wish to be cut with Common Cut.
4. Using the mouse, drag the first part into the empty cutting plan.
5. Using the **Ctrl** + arrow keys, move the part to the sheet edge.
6. Using the **Ctrl** + arrow keys, move the Common Cut outer contour of the second part up to the relevant contour of the first part.
7. Depending on the size of the parts, create a Common Cut group.
8. Select the Common Cut parts that you have grouped together.
9. Select the **Row** function.
10. Copy the group until the cutting plan is full.
11. Use the **Simulator** function from the **Export** menu to check that the common cuts have been applied correctly.

Exporting Common Cut cutting plans

In order to export the nested Common Cut parts as a Common Cut cutting pan, the **Common Cut** parameter must be enabled in the Export settings.



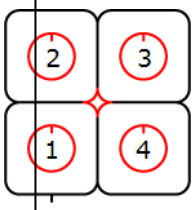
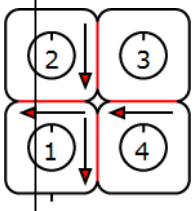
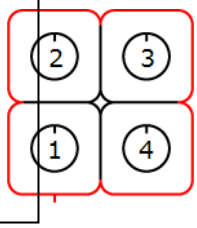
Note: Before exporting a Common Cut cutting plan, use the Simulator to check that the common cuts have been applied correctly.

Exporting Common Cut cutting plans

1. Select the **Export settings** function from the **Export** menu.
2. Select the **Common Cut** technology settings on the **Preprocessor** tab.
3. Enable the option **Optimized cut for parts pair**.
4. Use the **Simulator** function to check that the common cuts have been set correctly in the cutting plan.
5. Use the export functions in the **Plans** group to export the cutting plan.

Cutting process for Common Cut parts

This example shows the cutting process for a group of Common Cut parts.

1. Step Cut the cut-outs. These include the cut-out in the center of the group of parts, which was created by joining the parts together.	
2. Step Cut the common cuts. The cutting direction is the same for common cuts.	
3. Step Cut the outer contour.	

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15.4.4 Cutting clamped sheets

Machining cutting plans with sheet clamping is only possible for Byspeed machine. The following options are available for machining cutting plans with clamped sheets:

- Sheet clamping with automatic retraction of the clamping jaws (ByVision P8103 onwards)
- Sheet clamping without automatic retraction of the clamping jaws (up to ByVision P8102)

Sheet clamping with automatic retraction of the clamping jaws

Parts can be nested over the entire area of the sheet, achieving better utilization of raw material. To avoid the cutting head hitting the clamping jaws (**B**) when cutting parts in the clamping area, the sheet clamping mechanism is automatically retracted before machining these parts.

Sequence of the cutting process

1. All parts in the sheet clamping no-go area (**B**) are moved to the end of the cutting plan in the cutting sequence.
2. On first contact with the no-go area, the cutting process is interrupted with a Stop. The clamping jaws are retracted.
3. After retracting the clamping jaws, the cutting process continues with an automatic restart **Continue where cutting left off**. The clamping jaws cannot be activated any more.
4. The cutting process continues at reduced speed.

☞ **Note:** When assigning the cutting technology to cutting plans containing sheet clamping, make sure that the contour machining sequence always progresses from right to left (**A**). This will significantly reduce the risk of the sheet vibrating during machining. To do so, use the machining sequence in the Cutting technology settings *(See: ›Machining sequence‹ Page 961)*.

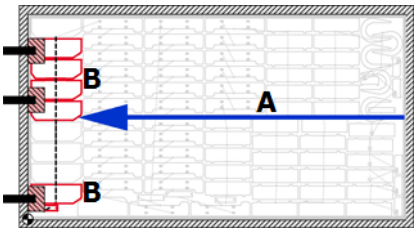


Fig. 15-7: Cutting plan without protective edge region

Tip: Once clamping has been disabled, all contours are cut at reduced dynamic speed. This can result in far longer cutting times. Therefore do not nest parts with a long X-dimension in the no-go area.

Sheet clamping without retraction of the clamping jaws

In the sheet-clamping area, a protective edge region needs to be created before nesting the parts (**A**). No parts are nested in this protective region. The jaws remain clamped closed throughout the cutting process. The axes can be moved with maximum dynamic speed. There are the following options for creating a protective edge region:

- **Automatic:** *The width of the protective region is specified using the **Sheet clamping edge distance** option in the **Cutting machine** form of the Administrator module. Each new sheet for this cutting machine is automatically assigned this protective edge region and can be used for nesting.*
- **Manual:** *The protective edge distance must be set manually for each sheet in the **Left edge** column of the **Sheets** tab before the sheet can be used for nesting.*

Note: This procedure is only required when clamping cutting plans up to **ByVision P8102**, because these early ByVision versions do not retract the clamping jaws automatically for parts in the no-go area. The size of the no-go area can be defined on the machine.

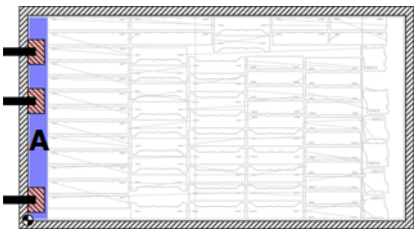


Fig. 15-8: Cutting plan with protective edge region

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15.4.5 Nesting parts as a module

When nesting modules, the software tries to nest the combined parts as a module as many times as possible in the current cutting plan.



Note: A module can be composed of a certain number of identical or different parts.

Requirements:

Create a new job and add the **Sample_2** part to the job in the **Parts** tab. Then create an empty cutting plan.

Nesting parts as a module in a cutting plan

1. In the **Parts** tab, select the **Sample_2** part from which you wish to create a module. A module can also be created from different parts.
2. Holding down the left mouse button, drag the part to the top end of the sheet.
3. Press and hold the **Ctrl key** and the left and down arrow keys to nest a row of parts along the left-hand sheet edge.
4. Use the **Ctrl + A** keys to select the parts.
5. In the selection field of the **Automatic** function, select the **Modules** option. All the selected parts are used as a module in the nesting process.
6. You can now continue to edit the cutting plan in the Part Nester.

15.4.6 Pilot drilling all initial piercing cuts using a drill head

This example shows the procedure on a water jet cutting machine for pilot drilling all parts in a multihead nesting plan using a single drill head prior to cutting.



Note: Pilot drilling involves the drill tool on the first cutting head.

Requirements:

- In the Administrator module, the **Drill** option is enabled in the **Cutting machine** area.
- A multihead nesting plan has been created and assigned the machining sequence.

Pilot drilling all initial piercing cuts in a multihead nesting plan

1. Select the **Export settings** function from the **Export** menu.
2. Enable the **Pre-piercing** area in the **Preprocessor** tab.
3. Select the **Per plan** attribute for the **Mode** option.
4. Select the **All contours** attribute for the **Selection mode** option.
5. Select the **Tool 1** attribute for the **Tools** option.
6. Run the **Simulator** function in order to check the pilot drilling prior to export.
7. Select a view option from the Special view menu. All initial piercing cuts are identified by a colored circle.
8. Export the cutting plan.

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15.4.7 Creating a machining sequence using horizontal or vertical sectors

For certain applications it is useful to distribute the machining sequence for parts in a cutting plan over a number of sector strips (**A**) in turn. The sectors can be created to run in the horizontal or vertical direction.

Note: The sectors are represented as blue and white areas on the sheet.

Application:

When there are numerous small parts in a heavy-gage sheet, to distribute the heat input to the material evenly over the entire sheet.

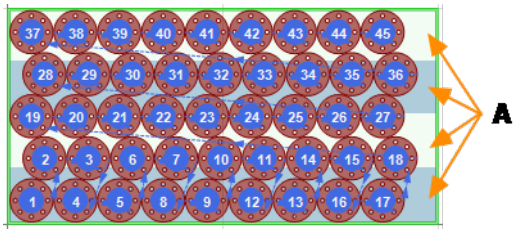


Fig. 15-9: Division of a cutting plan horizontally into sectors

Using sectors to create a machining sequence that runs horizontally

1. Select the **Settings** function from the **Cutting technology** menu.
2. In the **Machining sequence** area, select the **Vertical - From left to right** mode.
3. Enable the **Group by sectors** option.
4. In the **Grid X** field, enter the total length of the sheet.
5. In the **Grid Y** field, enter the width of the sector in which the machining sequence shall be performed.
6. Run the automatic cutting plan creation process.

Note: To create sectors running vertically, you must enter the total width of the sheet in the **Grid Y** field and select the **Horizontal - From bottom to top** mode.

15.4.8 Nesting small parts along the bottom edge of the sheet

In order to make it easier to remove cut sheets, it helps to nest small parts in the lower area of a sheet. For this to be done automatically in the automatic nesting process, the following nesting preferences must be set:

1. Select the **Settings** function from the **Fully automatic** or **Nesting** group.
2. Select the **From bottom to top** attribute for the **Nesting direction** option.
3. Select the **Small parts first** attribute for the **Priority** option.
4. Nest the job or the cutting plan.

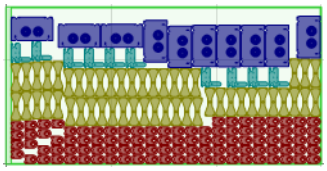


Fig. 15-10: Cutting plan containing small parts nested along the bottom edge of the sheet

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15.4.9 Cutting plans for repositioning machines

A repositioning cutting plan is always programmed as a single plan. The raw material must be selected to suit the plan size. However, because the cutting area is only 4 meters maximum, all plans that are longer than 4m must be divided up. The cutting plan is divided into a maximum of 3 cutting areas (12m machine).

Separating contours:

- If a contour is not separated or only touched by a repositioning line, the contour is not changed.
- If a contour lies on a repositioning line, it is separated and the lead-ins removed. The new lead-in points lie on the repositioning line.

☞ **Note:** The settings for repositioning are defined in the preprocessor export settings *(See: ›Reposition‹ Page 1166)*.

Nesting long parts on 6.5 m machines

For nested parts in cutting area 2 that are shorter than the maximum cutting area (4m), the machine does not reposition into cutting area 1 when crossing the repositioning line.

Example:

While cutting part **A**, the machine repositions on repositioning line **C** into cutting area **2**. After completing part **A**, part **B** is cut without repositioning on repositioning line **C** into cutting area **1**.

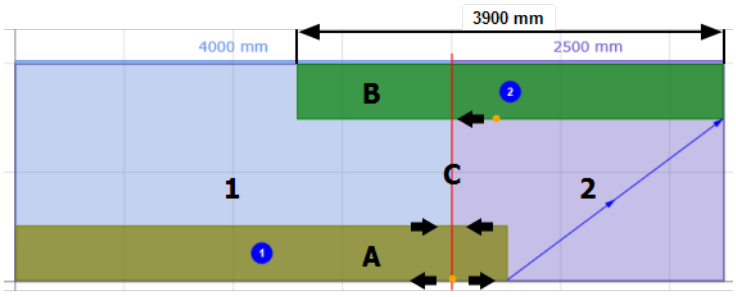


Fig. 15-11: Cutting long parts in cutting area 2

Tips for nesting repositioning cutting plans

- When nesting on large sheets, note that the quality is poorer near the edges than in the center. Tightly toleranced parts should therefore be placed in the center. Nest simple parts where quality is not that important near the edges.
- A sufficient distance from the edge must be allowed at the lower sheet edge and also specifically along the upper sheet edge.
- Used pulsed piercing on the contour if it is interrupted by a separating position. Note that material being used.
- Residual grid cuts must be placed so that they cut across the whole sheet. This prevents heat propagating in the sheet (thermal break).

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15.4.10 Nesting with machine parts

These examples show how to optimize cutting plans for certain applications on the machine in the **Nesting** menu. Optimization is based on the geometries of machine parts that can be displayed in the special view.

Note: To display the appropriate machine part, enable the appropriate option in special view.

Attention! To make sure that the nested parts match the geometry of the particular machine part on the machine, the sheet must be exactly at the origin of the cutting plan. This is the customer's responsibility and can be accomplished with the aid of an alignment gage mounted on the shuttle table. An alignment gage is available as an option from Bystronic Laser AG.

Aligning parts on the support grate

This example shows manually nested parts where the parts are laid out so that the tips of the support grate **A** do not touch the contours of the parts.

Tip: Use this option for high-quality parts that must not have any burn marks on the cutting surfaces.

Tip: Nest the parts so that the tips of the support grate prevent the part from tilting over.

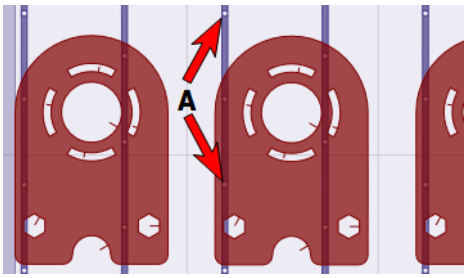


Fig. 15-12: Part aligned on support grate

Aligning parts on the ByTrans suction-cup frame

This example shows how to align big parts on a displayed suction-cup frame for the BigPart remover in the **Nesting** menu.

1. Delete all parts in current cutting plan.
2. Manually align the big parts on the suction cups in the cutting plan.
3. Use the **Automatic** > **Plan** function to add unnested parts to the cutting plan.

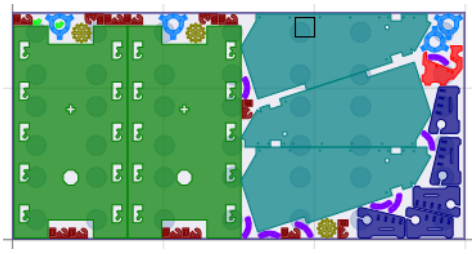


Fig. 15-13: Cutting plan with big parts and suction cups displayed

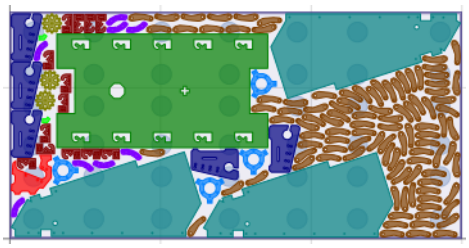


Fig. 15-14: Cutting plan with big parts aligned on suction cups

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Aligning parts on sheet clamping

In this example, the part collides with the displayed sheet clamping after automatic nesting (**A**). By mirroring the part, it can be nested without it colliding with the sheet clamping (**B**).

For information about nesting cutting plans with sheet clamping, refer to:

- *(See: ›Cutting clamped sheets‹ Page 1364)*
- *(See: ›Sheet clamping‹ Page 104)*

☞ **Note:** This option is only available for BySpeed machines.

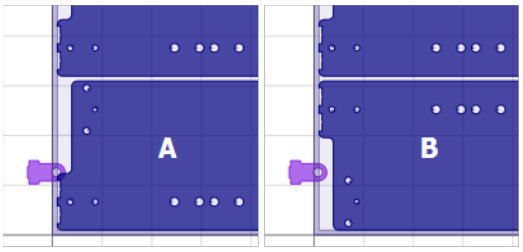


Fig. 15-15: Aligning part on sheet clamping

15.4.11 Nesting a test part

Test parts are used to check cutting quality on a sheet. All nested instances of a part defined as a test part are used as test parts.

Icons on a test part

Symbol	Meaning
	Test part
	Programmed stop

Nesting a special test part

This example describes how to nest a special test part for different operating modes in a cutting plan.

1. Select the **New job** function.
2. Add the test part to the list of parts in the **Job data** using the **Add** function.
3. Specify the number of test parts to cut.
4. Enable the option in the **Test part** column.
5. Manually nest the specified number of test parts at an easily accessible location on the sheet.
6. Select the **Automatic machining sequence** function from the **Cutting technology** menu.
7. Save the job as a template to use it again.

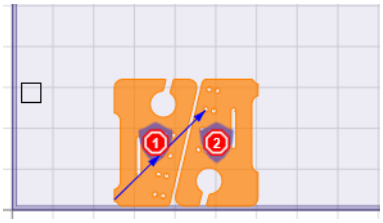


Fig. 15-16: Test parts on a sheet

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Nesting a production part as a test part

This example describes how to test cutting quality with a production part. The production part is nested as a test part in all four corners of the sheet.



Note: The example can be used on laser machines with CO_2 resonators on which cutting quality may differ across the whole sheet.

1. Select the part that you wish to cut as a test part.
2. Using the **Add** context function, insert a second copy of the part to the list of parts in the **Job data**.
3. Specify the number of test parts to cut.
4. Enable the option in the **Test part** column.
5. Manually nest the test parts in all four corners of the sheet (**A**).
6. Automatically nest all other parts on the sheet.
7. Select the **Automatic machining sequence** function from the **Cutting technology** menu.
8. Use the mouse to select parts 1–3 in the machining sequence.
9. Use the interactive button  to remove the **programmed stop** attribute from the test parts.
10. Export the cutting plan. All test parts are cut one after another before the machine stops at the programmed stop.

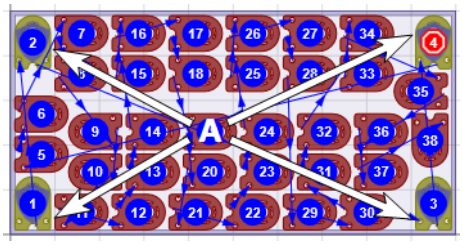


Fig. 15-17: Test parts nested in the corners of the sheet

15.5 Parts

These examples show how to import parts from various file formats into BySoft CAM.

- Apply autoform
- Importing a part from Bysoft6
- Extracting parts from a DXF file
- Importing parts from LCC/NCP files

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15.5.1 Apply autoform

The shapes can be used on the contours and areas of a part in the following manners:



Note: The position of the shape can also be changed later using the **Move** function.

Create an inner contour from a shape

The shape is inserted into a part as an inner contour (**A**).

1. In the **Shape** area, select the shape which is to be inserted as an inner contour.
2. In the **Parameters** area, enter the values required for the shape.
3. In the **Position**area, make an entry to define the insertion point for the shape.
4. Click **Insert** to accept the shape.

Note: The settings made in the **Mode** area have no influence on the insertion of an inside contour as long as it does not overlap with another contour.

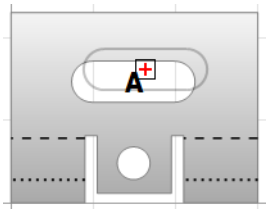


Fig. 15-18: A slot was inserted as an inner contour.

Change an inner contour with a shape

Depending on the mode, the shape is either subtracted from or added to the inner contour.

1. In the **Mode** area, select how the shape is to be applied to the inner contour.
2. In the **Shape** area, select the shape which is to be used to change the inner contour.
3. In the **Parameters** area, enter the values required for the shape.
4. In the **Position**area, make an entry to define the insertion point for the shape.
5. Click **Insert** to accept the shape.

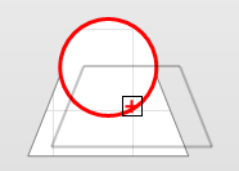


Fig. 15-19: Trapezoidal cut-out added to a circular cut-out

Cut out a shape from an outside contour

The contour of the selected shape which overlaps with the inner area of the part contour is subtracted from the part contour.

1. In the **Mode** area, select the **Cut** option.
2. In the **Shape** area, select the shape which is to be subtracted from the outside contour.
3. In the **Parameters** area, enter the values required for the shape.
4. In the **Position**area, make an entry to define the insertion point for the shape.
5. Click **Insert** to accept the shape.

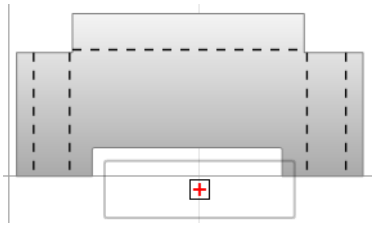


Fig. 15-20: Rectangular cut-out on a contour

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Add a shape to an outer contour

The contour of the selected shape which overlaps with the outer area of the part contour is added to the part contour.

1. In the **Mode** area, select the **Add** option.
2. In the **Shape** area, select the shape which is to be added to the outside contour.
3. In the **Parameters** area, enter the values required for the shape.
4. In the **Position**area, make an entry to define the insertion point for the shape.
5. Click **Insert** to accept the shape.

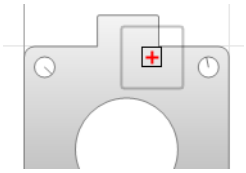


Fig. 15-21: A tongue added to a contour

Creating cut-outs on bend lines

A cut-out is subtracted from the contour of the part at the end of a bend line.

1. In the **Mode** area, select the **Cut** option.
2. In the **Shape** area, select the **Groove** option.
3. In the **Parameters** area, enter the values required for the shape.
4. Select the **End point** snap function.
5. Click on the bend line near the end point of the bend line (**A**). The cut-out is subtracted from the contour at the end of the bend line (**B**).

Tip: Use this option for bends whose sides need to fit into an inside contour. Adding cut-outs at the end of bend lines prevents compressed material from protruding over the contour after bending.

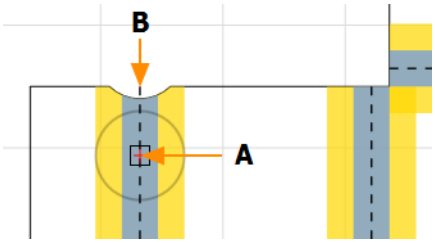


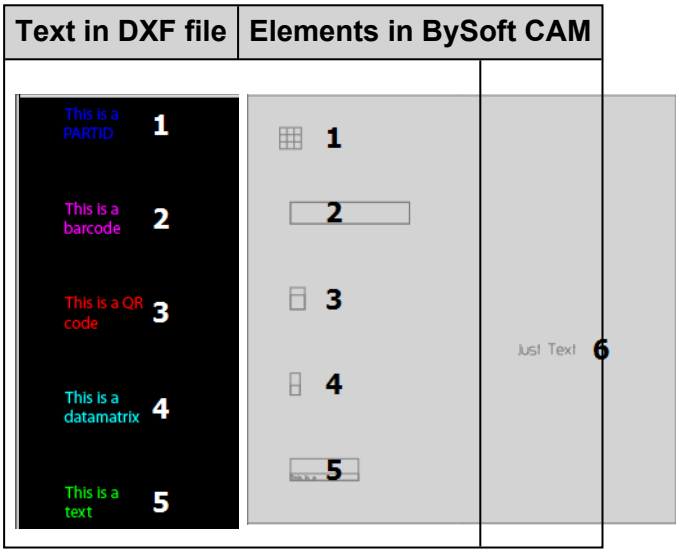
Fig. 15-22: Cut-out at the end of a free bend line.

15.5.2 Define layer ID for labels in DXF file

This example shows how to define text information for labels, part identifiers, or normal text in a DXF file. The texts are automatically converted into the respective element when imported into Part Editor.

Requirements:

Each element is defined with a unique ID. The ID must be created as a separate layer with the corresponding designation in the drawing (see key).



Key:

	ID	Element
1	DATA MATRIX	**Parts ID**
2	LABEL_BARCODE	**Part Marker barcode**
3	LABEL_QRCODE	**Part Marker QR Code**
4	LABEL_DATAMATRIX	**Part Marker Data Matrix**
5	LABEL_TEXT	**Part Marker text**
6	0	**Normal text element**

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15.5.3 Multi-head cutting in a single part

The example shows the basic procedure when creating a cutting plan for multihead cutting in a part.

By cutting long straight or parallel contours with multiple cutting heads it is possible to shorten machining time. This machining mode is particularly suited to large single parts which cannot be multiply nested.

Note: Multi-head cutting can only be used on water jet machines with multiple cutting heads.

Initial situation

- A rectangular part is used as an example with two rows of holes and outside dimensions of 2050 x 310mm.
  - For this part, the two long sides of the outer contour and the drill holes are to be cut simultaneously with 4 heads.

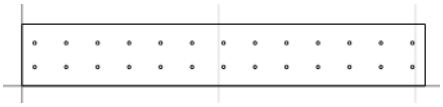


Fig. 15-23: Longitudinal support

Requirements for multi-head cutting

- The inside contours to be cut simultaneously must be congruent.
- Cutting head distance must be the same over the entire contour cutting area (see Fig.).
- Minimum cutting head distance of **220mm** must be observed.
- The **Precision Aligner** must be present and configured correctly on all cutting heads.

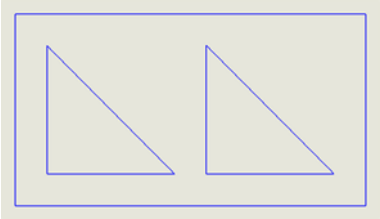


Fig. 15-24: Multi-head cutting possible

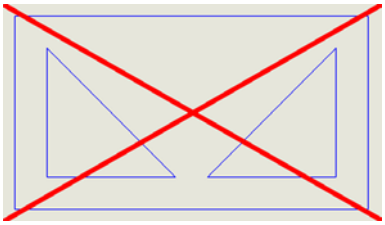
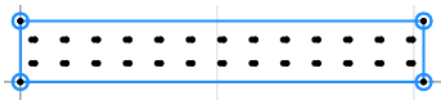


Fig. 15-25: Multi-head cutting not possible

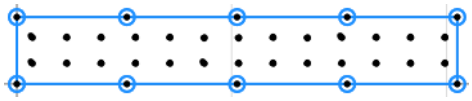
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Separating outer contours

1. Activate the special display **Open Contours** in the **Geometry** menu.
2. Activate the **End point** snap function.
3. Select the **Separate** function in the **Edit** selection dialog.
4. Select the four corner points of the rectangle using the mouse.



5. Select the **Separate multiple** function in the **Edit** selection dialog.
6. Enter a value of 4 in the **Number** field.
7. Select the first contour to subdivide.
8. Press the **Apply** button to confirm the selection.
9. Select the second contour to subdivide.
10. Press the **Apply** button to confirm the selection.



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Assigning cutting heads to the contours

1. Select the **Cutting technology** menu.
2. Select the **Automatic** function in the **Multi-head** group.
3. Check cutting head assignment by activating the special display **Tool** and **Groups**.

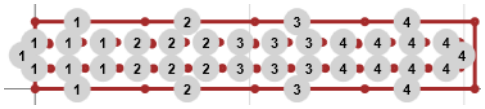


Fig. 15-26: Tool special view

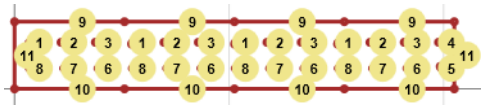


Fig. 15-27: Groups special display

> **Note:** If automatic tool and group assignment was not set ideally, you can modify the assignment to suit your own requirements using the **Manual** function.

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Assign cutting technology

> **Caution!** The machining sequence must **not** be set to the whole part using the **Automatic** function. Because the outer contour has been separated, the program no longer recognizes it. As a result, parts of the outer contour may get cut before the inside contour.

1. Select all inside contours using the mouse.
2. Select the **Automatic** function from the **Machining sequence** group.

> **Note:** Only the machining sequence of head 1 is displayed.

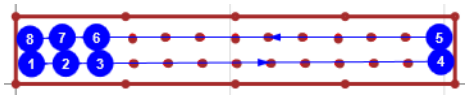


Fig. 15-28: Machining sequence of inside contours

3. Cancel the selection of the selected contours.
4. Select the **Complete** function from the **Machining sequence** group.

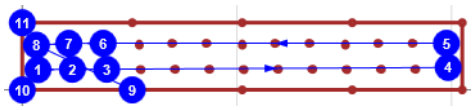


Fig. 15-29: Machining sequence with outer contour

5. Select the **Automatic** function in the **Lead-in/Lead-out** group.

Create cutting plan

1. Select the **Create cutting plan** function.
2. Enter a job name in the **Job properties** dialog and click **OK** to accept the part name.
3. Export the part as a cutting plan using the **Export** menu. If the number of parts >1, nest the part on a normal sheet before exporting the cutting plan.

15.5.4 Set cutting technology on part in CAD drawing

This example shows how to use a part straight from an imported CAD drawing in Part Editor without having to delete the elements.

> **Note:** You can display the part's drawing elements in Part Nester by calling up the special view **Geometry without cutting technology**.

Assigning a cutting technology to a part from an imported CAD drawing

1. Select the **Import part** function.
2. In the **Import part** dialog box, select the CAD drawing and press the **Open** button to confirm import.
3. In the **Part properties** dialog box, set the unit and material properties of the part.
4. Use the **No selection** function to import the part.
5. Select the part contours in the drawing.
6. Use the **Automatic** function to assign the cutting technology to the contours.
7. Select the **Invert** selection function.
8. Select the **Contour** function from the **Geometry** menu.
9. Select the **Engraving** option in the **Contour properties** dialog.
10. Use the **Save** function to save the part in the Parts database. You can now nest the part in Part Nester.

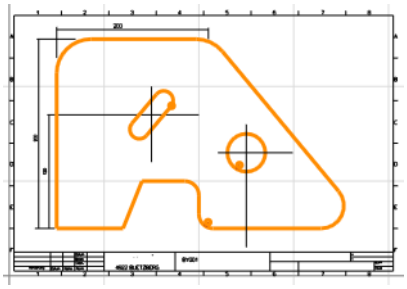


Fig. 15-30: Part with cutting technology in a CAD drawing

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15.5.5 Importing a part from Bysoft6

This example shows how you can easily import a part from the Bysoft6 database folder into BySoft CAM.



Note: The cutting technology for the part is imported at the same time.

Importing a part from Bysoft6 database folder (*.bpd)

The Bysoft6 files are normally located in the default Bysoft database folder **partroot** of the Bysoft installation. You can find out the actual storage location from the «bysoft6.ini» file in `[Directories]`.

Storage location of the bysoft6.ini file

- **Windows XP:** *C:\Windows*
- **Windows Vista / 7:** *c:\Users\lbeu\AppData\Local\VirtualStore\ProgramData\bystronic\Bysoft6*

1. Select the **Import part** function.
2. In the **Import part** dialog box, select the Bysoft6 file that you wish to import and confirm your selection with **OK**. The part is imported into BySoft CAM and opened in Part Editor.
3. Switch to the **Cutting technology** menu. In the **New cutting technology** dialog box, specify the machine that you wish to use for producing the part.
4. Confirm your entries with **OK**.
5. Use the **Save** function to save the imported part in the Parts folder.

15.5.6 Extracting parts from a DXF file

This example shows how you can easily extract a part from a DXF drawing and import it into BySoft CAM.

Importing a part into BySoft CAM from a DXF file

1. Select the **Import part** function.
2. In the **Import part** dialog box, select the DXF drawing and confirm the import with the **Open** button.
3. In the **Part properties** dialog box, set the unit and material properties of the part.
4. In the drawing, use the selection and view functions to select the contour of the part that you wish to import. You can use the **Shift key + left mouse-click** to add additional contours to the selection.
5. Use the **From selection** function to import the part.
6. Use the **Close** geometry function to correct any open contours.
7. Use the **Save** function to save the part in the Parts database. You can now continue to edit the part in the Part Editor.

Manually extracting multiple parts from a DXF file

1. Select the **Import part** function.
2. Open the DXF file that contains the parts.
3. Use the **Class** function and the View functions to select the geometries of the parts.
4. Use the **Create** function to import the geometries. The geometries are opened in Part Editor.
5. Select the geometry that you wish to extract as a part.
6. In the **Extract** group select the **Manual** function.
7. Select **Save parts** mode.
8. Enter the path to the part in **Directory**.
9. In the **Name** field, enter the name that you wish to save the part under.
10. Confirm the entry with **Apply**. All selected parts are extracted as individual parts and automatically saved to the path specified.

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Assigning the cutting technology to extracted parts

1. Select the first extracted part.
2. Select the Cutting technology menu. This opens the **New cutting technology** dialog box.
3. Specify the machine and the nesting settings.
4. Confirm your entries with **OK**. The cutting technology is assigned to the part.
5. Select the **Save** function.
6. In the **Save as** dialog box, specify where you wish to save the file and then click the **Save** button to save the part in the specified folder.
7. Repeat the procedure for each extracted part.

15.5.7 Importing parts from LCC/NCP files

If you require parts that are only available in an LCC or NCP file, the part can be imported again into BySoft CAM using the Import function in the Part Nester module. The cutting technology for the part is retained unchanged. Before saving the part, however, it is worth checking the part for errors using the Check function.

Importing a single part into BySoft CAM from an NCP file

1. In the **Modules** dialog box, select the **Import job** function.
2. In the **Import job** dialog box, select the NCP file that contains the part you require and confirm your selection with **Open**.
3. Confirm the information given in the **Job properties** dialog box with **OK**. The job is imported into the Part Nester module.
4. In the **Parts** tab, select the part that you wish to import.
5. Select the **Extract** function from the context menu. The part is opened in the Part Editor module.
6. Call up the **Save as** dialog to save the part in the appropriate folder.
7. Select the **Open part** function in the **Modules** dialog to edit the part in Part Editor. This opens the **New cutting technology** dialog box.
8. Specify the machine and the nesting settings.
9. Confirm your entries with **OK**. The cutting technology is assigned to the part.
10. Select the **Save** function.
11. In the **Save as** dialog box, specify where you wish to save the file and then click the **Save** button to save the part in the specified folder.
12. The part can now be used for creating jobs.

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15.5.8 Update part contour

This example shows how to easily import an updated part from a DXF drawing without redefining the cutting technology in the existing part.

Attention! This procedure only works with the same parts in which an inside contour was added or deleted.

Note: If a DXF files consists only of part contours, they are imported immediately into the existing part. There is no intermediate step to select the contour from the **Import** menu.

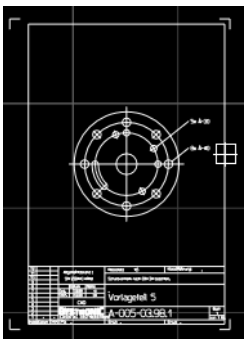


Fig. 15-31: DXF drawing in **Import** menu

Update part contour from a DXF file

1. Open the part to update in Part Editor. Then select the **Import** function from the **Parts** menu.
2. In the Import dialog, select the DXF drawing that you want to use to update the part from the Parts database. Then press **Open** to open the drawing in Part Editor.
3. Enter the material of the part in the **Job properties** dialog and press **OK** to confirm.
4. Select the part contours using the view function in the **Import** menu.
5. Use the **From selection** function to create the part.
6. Close the new part and save it in the prompt dialog. All new contours are imported into the existing part. The technology on identical contours is retained.
7. Select the new contours in the **Cutting technology** menu and assign the appropriate technology to them.

15.6 Sort

These are examples of sorting parts which are not specific to only one module.

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15.6.1 Create cluster

You can use a cluster in the Part Nester module to nest multiple parts that belong together in a single cluster. The next step is to nest the cluster in a cutting plan.



Attention! The parts in a cluster must always have microjoints. Otherwise, the cluster cannot be sorted as whole.

Requirements:

- The parts must have the same thickness and consist of the same material.
- At least one microjoint must be set on the outside contour.

Part Editor

The first step is to create a cluster frame in Part Editor. Later, the parts are nested inside this frame. The frame can be used again for other clusters.

Creating a cluster frame

1. Create a new part using **New** function in the file functions.
2. In the **Part Properties** dialog box, specify the name, system of units, material and thickness of the cluster frame.
3. Assign the technology to the cluster frame in the **Cutting Technology** menu.
4. Select the **Add** function in the **Stack** menu.
5. In the **Add Gripper** dialog, set the appropriate gripper tool on the area inside the frame. Note that it is not possible to nest parts in the gripper area.



Note: If multiple grippers have been assigned to apart, only the first gripper is used.

6. From the File functions, select the **Save as cluster frame** function.
7. Enter the name of the new cluster frame in the **Name** field or accept the suggested name.
8. Press **Save** to save the part as a cluster frame.

Part Nester

Creating a cutting plan with clusters

1. Add an empty cutting plan to the workspace on the **Sheet** tab.
2. Add the appropriate quantity of individual parts on the **Parts** tab.
3. Flag all parts to nest in a cluster in the **Part in cluster** column. If you only want to nest instances of apart in a cluster, additionally enable the option **Unique in cluster**.
4. Add a cluster frame using the **Add** context menu function on the Cluster tab.
5. Select the **Edit** context menu function to begin editing the cluster and nest the appropriate parts in the cluster. Select the  button to stop editing the cluster.
6. You can now nest the clusters with the individual parts in cutting plans.

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15.6.2 Stack-carrier-oriented sorting

In contrast to the machine-layout-oriented programming of sorting, stack-carrier-oriented programming of sorting is created independently of the physical sorting unit.

In stack-carrier-oriented sorting, it is first determined how parts are placed on sorting carriers (e.g. pallets). At the end of the process, it is determined how the plans will be cut and how pallets or other sorting carriers will be arranged in the sorting unit.

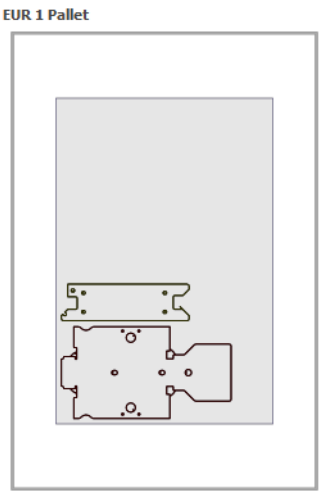
Advantages of stack-carrier-oriented sorting process

- Machine-independent design of the sorting processes
- Production-oriented sorting processes
- Unlimited order size and number of sorting carriers per order
- Optimization of production by minimizing total production time and changing of stack carriers

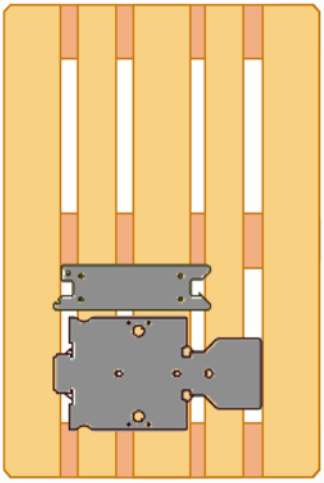
Concept of a sorting process

The principle of a sorting plan for stack carriers (nesting) is similar to that of a cutting plan. They both have a number of cycles or passes. At the end of the process, physical stack carriers are created from them. A stack carrier consists of nested stacks of parts. It can be used multiple times for different sorting steps.

- Stack carriers in the sorting process



- Stack carriers in the sorting plan



Use of stack-carrier-oriented sorting process

This process can save time when programming sort jobs with a typical carrier layout is highly repetitive. This may be the case in mass production, where a cutting plan has a large number of cycles and therefore requires a large number of pallets or cassettes that all have a similar stack layout.

Creating cutting-plan-based sorting

1. In Sort Job Editor, select the **New job** function.
2. In the **Job properties** dialog, enter a name for the sort job.
3. Select the machine with the sorting system.
4. Select the **...ByTrans Modular** sorting unit.
5. Activate the option **Stack carrier oriented** and press **OK** to confirm.
6. Use the “Sort plan settings” function to select the stacking areas to use. Optionally, you can define different sorting zone jobs in the sort job properties on the **Sorting zone jobs** tab.
7. Use the **Add** function to add the cutting jobs to be sorted. The selected jobs are displayed on the **Jobs** tab in the **Job data** dialog.
8. Use the **Add** function to add the cutting plans of the jobs to be sorted. The selected cutting plans are displayed on the **Cutting Plans** tab in the **Job data** dialog. Optionally, you can now allocate the cutting plan parts to the defined sorting zones in the sort job properties on the **Sorting zone jobs** tab.
9. On the **Stack carrier sorting plans** tab, right-click to call up the context menu.
10. Select the **Edit stack carrier sorting plans** function from the menu.
11. In the **Define stacking area sorting plans** dialog, select the type of stacking area and the stack carrier.

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12. Press the **Add stack carrier** button. An empty stack carrier is added to the stacking area.
13. In the **Parts** section, use the **Add** column to add the parts to the stack carrier that are to be stacked on the stack carrier. The system automatically nests the stacks of parts on the carrier.
14. Create additional stack carriers as needed and assign parts to them.
15. Once all stack carriers have been created, press **OK** to confirm the dialog box.
16. Select the **Automatic** function in the **Sorting plan** section. The sorting process is created automatically and displayed in the **Manage sorting process** dialog.



Tip: The sorting process can be customized in the dialog.

17. Select the **Generate sorting plans** function in the **Sorting plan** section. The program creates the exportable sorting plans with the physical stack carriers and displays them in the Sort Job Editor workspace. The stacks of parts and stack carriers are displayed accordingly for each sorting phase (cutting plan).



Tip: The stacks of parts on the stack carriers can be customized in the view.

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16 FAQs

FAQ stands for frequently asked question, so FAQs are questions that people often ask about the software or which are of general interest.



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16.1 FAQs about BySoft CAM

General questions

Can I still use Bysoft 6 data?

Yes. BySoft CAM includes a Migration wizard for importing Bysoft 6 data into BySoft CAM.

Geometry	Version	Data import	Comment
Flat	Bysoft < *6.0	no	
	Bysoft >= 6.0	yes	Automatic data import
Bending	Bysoft < *6.0	no	
	Bysoft >= 6.0	Requirement	Backgauge positions are lost
Tube	Bysoft < *6.0	no	
	Bysoft >= 6.0	Not implemented yet	

* Data from Bysoft 5 must be imported first into Bysoft 6.

Can I still use Bysoft 5 data?

No. Bysoft 5 data is no longer supported. Data from Bysoft 5 must be imported first into Bysoft 6.

Can I import existing cutting plans (LCC/NCP)?

Yes. BySoft CAM can import existing cutting plans in the LCC/NCP format and save them as new jobs.

Can I import existing CAD files (DXF/DWG)?

Yes. You can import existing DXF/DWG files individually into BySoft CAM.

Can I also import existing CAD files (DXF/DWG) automatically?

Yes. BySoft CAM includes a separate module called Part Importer, which can automatically import multiple CAD files into BySoft CAM.

Can two users work on the same geometry?

The user who opens a geometry file first has exclusive rights and therefore can also save any changes. Any other users who open this geometry file get to use a copy. The edited copy can only be saved under a different name.

What languages are supported by BySoft CAM?

Language	BySoft CAM
German	x[1]
English	x[1]
French	x[1]
Italian	x[1]
Spanish	x
Portuguese[2]	x[1]
Dutch	x
Danish	x
Swedish	x
Chinese[3]	x[1]
Korean	x[1]
Russian	x
Czech	x
Polish	x
Turkish	x
Hungarian	x
Japanese	x
Thai	x
Vietnamese	x
Taiwanese	x

[1] Online Help

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[2] Brazilian

[3] Simplified

What technologies are supported by BySoft CAM?

- Laser cutting
- Water jet cutting
- Air bending
- 3-point bending
- Drilling, thread tapping, recessing

What exchange formats can be imported?

Programmer	DXF, DWG

Are older Bystronic cutting machines also supported?

BySoft CAM supports all machines running CNC96 and CNC02 controllers. Older machines using CNC90 controllers are no longer supported.

Are older press brakes also supported?

BySoft CAM now only supports Hämmerle or Xpert press brakes that use the MMC-Bybend controller. Older Beyeler PR (Cybelec controller) and Hämmerle (Infinitron controller) machines are no longer supported.

Where can I find the software version information?

The software version can be displayed as follows

- With the **System information** function in the **Help** menu on the **System information** tab, in the first line.
- On the splash screen when you launch the software.

What Bystronic controllers can BySoft CAM be used with?

Technology	Machine	Controller
Laser	Byspeed / BySpeed Pro	CNC96 / ByVision
	Bysprint / BySprint / BySprint Pro	CNC96 / ByVision
	BySprint Fiber	ByVision
	Bystar	CNC96 / ByVision
	Bystar with rotary axis	CNC96 / ByVision
	ByVention	ByVention
Bending	Hämmerle 3P	MMC 7.x.x
	Hämmerle P21	MMC 2.x.x / VCN Pmac
	Xpert	MMC 7.x.x
	Beyeler	Cybelec
Water jet	Byjet	CNC 96 / ByVision
	ByJet (Pro, Classic, Smart)	ByVision

Bending

Where can I define common bends?

Bends can be combined into a common bend in the **Tools** menu using the **Connect** function.

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System

What are the system requirements for BySoft CAM?

BySoft CAM has the following system requirements:

Operating system	
Windows 10	64-bit
Windows Server 2012 - 2019	64-bit

Hardware requirements	
Processor	Intel i5 with 4 cores, recommended Intel i7 6 cores
RAM	8 GB, recommended ≥ 16 GB
Graphics card	SolidWorks certified graphics card Driver: http://www.solid-works.com/sw/support/videocardtesting.html
Installation medium	USB2, broadband internet connection
Screen resolution	1280 × 1024, recommended 1920 × 1080

Will BySoft CAM also run on older PC systems?

See system requirements for BySoft CAM

Does BySoft CAM support 3D input devices?

BySoft CAM supports the 3D Mouse Space Navigator. In order for the input device to work, it is additionally necessary to install an driver from 3Dconnexion.

- Driver download

16.2 Bystronic Customer Service

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Germany Gotha

Greece

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Poland

Romania

Russia

Singapore

South Africa

South Korea

South Korea Busan

South Korea Daegu

Spain/Portugal

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Switzerland Service

Switzerland Sales

Switzerland Automation

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17 Glossary

1

1-cut sections

This type of section is used for station plans in which the part can be bent on a single station without any special tools.

3

3-point bending

3-point bending achieves a repeatable and highly accurate angle setting using the V-shaped system. The bend angle is determined by the two entry edges, the bottom tool and the channel floor of the bottom tool. The 3 points lie on the same side of the sheet. The angle is set using height-adjustable stops in the bottom tool. The hydro cushion compensates for deformation by the ram. Each of the segmented top tools works autonomously against the hydro cushion and is protected against overload. The bending process is not complete until the material lies on the channel floor of the bottom tool, and the top tool is submerged in the hydro cushion.

7

75% rule

If «3-point» is selected as the bending method, the load on a tool segment must equal 75% minimum. If this minimum value is not met, the hydro cushion on the 3-point bending machine can sustain permanent damage. If the load cannot be increased to at least 75% owing to the use of another segment, the bending method must be set to Air bending.

A

Administrator

“Administrator” is a software module and is used to manage the material, cutting, bending, and sorting data held in the system. It can also be used to modify or create report templates for working documents.

AGV

An AGV (Automated Guided Vehicle) is a driverless transport vehicle with its own drive that is controlled automatically and guided without physical contact. AGVs are used for automatic transport of material between different production sites in a production process.

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Air bending

With this bending method, the bending part is pressed into the bottom tool by the top tool. The work piece shape is essentially determined by the relative position of the tools. Different bend angles are obtained by setting different penetration depths of the top tool into the bottom tool.

Angle measurement system

The angle measurement system (LAMS) minimizes the material-dependent variations in the bend angle during the bending process. This ensures that excellent angle precision is maintained over the entire length of the part.

Auto Part

Auto Part is a module that you can use to separate and unfold parts of 3D assemblies in a process and assign a technology to them.

Auxiliary bend

The auxiliary bend is temporarily inserted for certain bend shapes, and then bent back again, to allow the final shape of the bending part to be achieved.

B

Batch number

A batch means a set of products that were manufactured in one production run and hence have identical properties as regards the time at which they were made and product quality. The batch number is a unique serial number that the manufacturer can assign to the production run of a product at a later date.

Bend

The bend is an element of the profile programming and designates a change in shape of the work piece. A number of attributes can be assigned to a bend: side length, bend angle, bend radius (optional), initial-bend angle (optional), additional attributes (optional). A bend can consist of several bending processes.

Bend allowance

The reduction in the side length for bends made on the bending part is called a bend allowance. Bending causes the part to be stretched in a longitudinal direction. Therefore the length of the side of a bend must be reduced again by this amount in the cutting part. The bend allowance, i.e. shortening of the side length, compensates for this stretching.

Bend line

A bend line refers to a line along which the side is bent.

Bend Robot

“Bend Robot” is a program for offline programming of robots which automatically perform part bending processes on a bending machine.

Bend zone

Area around a bend line that is deformed by bending a radius.

Bending method

3-point bending, air bending and coining are referred to as bending methods.

Bending process

All actions or procedures that are needed for bending one or more bends are referred to as the bending process.

Bending sequence

The bending sequence refers to the specific order in which all the bending edges on a part are implemented. A bending part can usually be bent using a number of different bending sequences. When the bending sequence is set automatically, the software works out the optimum bending sequence and then assigns it to the bending part.

Bending station

A bending station always comprises a top bending tool and bottom bending tool and is included in a station plan.

Bending time

Time required to bend the part.

Blank segment

An empty segment is an area within a bending station that is defined as vacant. Empty segments are used for connected bend lines between two sides when there is a contour between the sides that must not be bent. An empty segment must always be bounded by two normal segments.

BTC

ByTrans Cross automation module

ByLink

Program for establishing links to a range of services between the machine and a higher-level system.

C

Cluster part

A cluster part consists of a usually rectangular part contour in which multiple parts can be nested.

Coining

Open-die and closed-die coining is the most precise bending method. The shape and the bend angle are set by the geometry of the top and bottom tools. The extremely high pressing forces that are required increase the wear on the press brake.

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Contact edge

The edge on the bending part that is pressed against a stop point on the backgauge finger for precise positioning.

Corner reliefs

Corner reliefs are notches made cut in the material at the ends of bend lines. They are needed to reduce deformation at the ends of the bends as a result of stretching and compression of material.

Cost Calculator

“Cost Calculator” is a software module in which you can perform cost calculations for flat and tubular parts.

Crowning

A system for compensating for deviations that can be traced to deflection of the top and bottom tool during the bending process.

Cutting Simulator

“Cutting Simulator” is a software module in which you can simulate the machining process of cutting plans.

Cutting technology

Cutting technology refers to the way in which material is machined by the laser or water jet tool. The cutting technology differs for different machining modes (e.g. laser: cutting, engraving, pulsed and macro1–5).

Cutting Wizard

Cutting Wizards are technology tables that can be used to set a cutting technology automatically on a part or a cutting plan.

D

DataMatrix code

The DataMatrix code is a 2D barcode used for permanent labeling of various materials. Thanks to its square data matrix, it is regarded as very reliable with regard to reading and is used above all in industrial applications.

Detection Eye

Optical system for detecting the origin in the X and Y direction and for calculating the angle of rotation of a sheet.

Docking dialog box

Docking dialog boxes are located at the sides of the software window and can be displayed if required by moving the mouse over them or by enabling the Dock icon.

Domain

A domain is a local security realm in which resources are managed centrally and which constitutes an enclosed administrative unit.

DUG

Removal unit (gripper)

DUH

Removal unit, horizontal

DUV

Removal unit, vertical

E

Element

An element denotes part of an object. Elements may be contours, segments, reference points, positioning paths or even texts on a part.

Entry zone

Shows the area around a bend line that is equal to the size of the aperture of the bottom tool. If an inner contour lies within this zone, it may be deformed. This deformation can be avoided by making a strain relief cut in front of the inner contour.

ERP

The term Enterprise Resource Planning (ERP for short) describes the business function of planning the resources within an enterprise as efficiently as possible for business operation. In Plant Manager, the term ERP refers to information on the cutting and bending process provided by an existing ERP system.

F

FixMaster

FixMaster is a method which places a gluing point as a microjoint on a contour which has already been cut. This option is only used on ByJet machines.

Flange length

The side length is the perpendicular length measured from the bend line to the outer contour of the bending part.

Functional groups

Function groups are where similar or associated functions are grouped in the ribbon.

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G

Geometry elements

Lines, arcs, circles, rectangles, slots and polygons, or user-defined combinations of these elements, are all referred to as geometry elements.

Gripper

A gripper is an element that can grip material by means of a vacuum or magnetically.

Gripper hand

This defines the different parts of the gripper tool. A gripper hand is a combination of different groups and defines the area in which the groups are placed.

Gripper tool

Gripper tools can be used to pick up cut parts from a cutting plan and place them on a stack carrier. The gripper tool is attached to a sorting head that can move around four axes.

Groups

Groups consist of a list of grippers with coordinates relative to the origin of the group. Groups all move together along defined axes in the X or Y direction and can be activated or deactivated by valves.

H

Hemming

This bend type bends the sheet over flat with a bend angle of 180°. The bending action is completed in two consecutive steps of «Pre-bending» and «Hemming». These steps may also be performed separately from one another.

Horn

A horn is a top tool whose tool holder on the left or right side is shorter than the length of the tool tip that is used for bending.

Host

In information technology, a host is a network computer on which one or more servers are operating.

Hotkey

A “Hotkey” refers to a combination of keys on a PC keyboard used to run a function.

Hydro cushion

The hydro cushion compensates for deformation of the top and bottom beams during the bending process, thus enabling repeatable bend angles that are constant over the entire bend length. The hydro cushion dispenses with the need for crowning. The hydro cushion is only used on 3-point press brakes.

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I

Inlet radius

The radius of the curve running from the upper edge to the inner contour of the bottom tool.

J

Job

A job is a cutting plan containing a number of nested parts, which is cut once or repeatedly using the specified raw material.

Job database

The location in which all the jobs recorded in the program are stored. The job database is managed in the Object Manager.

L

LAMS

See Angle measurement system

Load carriers

A load carrier can be a pallet, a mobile table, or AGV with which cut material can be transported to another production site.

LUH

Loading unit, horizontal

LUV

Loading unit, vertical

M

Machining process

The production process includes scheduling, nesting and manufacturing the parts of one or more orders.

Material

Combination of material and sheet thickness (without sheet format).

Metadata

Metadata contains information and data about various objects in the software. For instance details about the material, the machine or even settings for an object.

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Module

A module refers to a component, element or generally a part of a program.

Multi-part gripper

Multi-part grippers have several gripper hands (suction cups/magnets) that can be moved along axes and controlled separately by valves.

N

NC code

The NC code is a set of control commands that are used to control the working operations and movements in CNC-controlled machines.

Nesting

The process of arranging the required parts on the given raw material. When arranging the parts, the ideal mutual position is sought, taking the given criteria into account (minimal waste material, optimum cutting time, etc.).

Normal bending

Normal bending is the standard method for creating a bend. Bending is defined by the bend angle and an optional initial-bend angle.

O

Object

Objects are files which are saved in different directories in the program. Parts, cutting plans and even configuration files are referred to as objects.

Object Manager

“Object Manager” is a software module and is used for managing and structuring the object data used in the software.

Order

An order contains at least one part, which is assigned various production process data such as order number, start date, end date, quantity, thickness, material etc.

P

Part

Also referred to as construction part or work piece. The product to be manufactured for the customer. These may be cut-out shapes and contours or even the machined raw material.

Part Editor

"Part Editor" is a software module in which cutting and bending parts are prepared for machining.

Part Marker

Part Marker is a module on the laser cutting machine used to print information such as text, barcode, QR code or Data Matrix code on parts.

Part Nester

"Part Nester" is a software module in which cutting parts can be nested to create jobs and exported to the relevant machine as cutting plans.

Parts database

The location in which all the parts recorded in the program are stored. The parts database is managed in the Object Manager.

Parts removal

"Part removal" is a sub module of "Plant Manager" with which you can assign cut parts to the appropriate jobs when collecting parts. In order to be able to use the module meaningfully, the software must be installed on a PC terminal nearby the collection site.

Peripheral axes

Peripheral axes are automatically movable backgauges, bending aids, and angle measurement systems of a bending machine.

Plant Manager

"Plant Manager" is a software module. You can use this module to generate production orders for cutting and bending parts from an ERP system and assign them to your machine fully automatically or semi-automatically.

Popup window

A popup window is a dialog which is displayed when the mouse pointer is moved over a certain area in a program. As soon as the mouse pointer leaves this area, the dialog disappears once again.

Pre-bending

This bend type creates a bend in two steps. The bend is first made with an initial-bend angle and then the finished bend is made later with a final angle. Use pre-bending for complex bending parts to avoid collisions during the bending process.

Process costs

Process costs denote all costs that arise in the production of a part or product.

Process macro

Process macros can be used to modify parameters within a parameter file without needing to load a new parameter file at the machine.

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Programmed stop

A programmed stop is a machining stop set on a part without interrupting processing of the cutting plan. Machining is stopped after the part was cut with a programmed stop.

PRU

Cassette carriage

R

Raw material

Combination of material, sheet thickness and sheet format (length and width dimensions).

RBG

Storage/retrieval unit for transporting cassettes in a store system.

REGEX

A regular expression (REGEX) is a string used to describe sets or subsets of strings using certain syntax rules.

Report Designer

Program with which report templates for reports can be created or changed.

Retraction

Retraction refers to the backwards movement of the backgauge arm that is required to avoid the backgauge finger colliding with the bending part. Retraction is performed once the bending part has been clamped between top and bottom tools.

Ribbon

The Ribbon is composed of one or more functional groups and the associated functions.

S

Scanning

Scanning refers to the process of machining rectangular and square contours in a linear sequence. The segments of the rectangular contours are cut first in the vertical direction and then in the horizontal direction. Linear processing has the advantage of requiring fewer changes in direction. This enables a substantially shorter machining time. Compared to conventional machining this can be up to 50% faster for scanning.

Scheduler

The Scheduler is a service that sorts orders and parts according to various criteria and creates jobs from them.

Segment

A segment is a bending tool that is used as a bending station or part of a bending station.

Server

The term server refers either to a piece of software within the client-server system or a piece of hardware on which this software runs as part of this system. Server software is a program that communicates with another program, the client, in order for the client to gain access to special services. Server hardware is a computer on which one or more server software programs run.

Set-up time

Time to set up station plans including measuring and correcting the first bending parts of a series. It does not include the time for fetching tools from the cabinet (see Tool time).

Shortcut menu

Function menu displayed by right-clicking with the mouse.

Skins

These are attribute files for computer programs for customizing the user interface.

Snap on

To snap on means to set the backgauge finger against a contact edge or a corner.

Sort Job Editor

Sort Job Editor is a module in which sorting plans can be created from several jobs. Sorting information is exported together with cutting plans by the postprocessor and imported into the machine.

Sorting head

Is a mobile transport module with a gripper system that moves along a bridge and carries cut parts from the shuttle table to a stacking position where it puts them on a stack.

Sorting zone jobs

Sorting zone jobs can be used to stack parts that belong together for certain reasons (e.g. same downstream process or same assembly) on the same stack carrier.

Split bar

This is a bar between split panes of the software that can be dragged using the mouse to change the window size of the panes.

Stack carrier

The stack carrier is the area on which parts can be stacked. Multiple stack carriers can be defined per stacking area.

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Stacking area

BySort has pallet, cassette and carriage stacking areas. These are the basic modules for stacking parts.

Stacking site

The stacking site consists of a combination of pallet, cassette and carriage stack carriers.

Stacking site configuration

See stacking site

Stacking site definition

The stacking site definition consists of the selected stacking site configuration, the allocated cutting plans, and the stacks specified on them. It is used to create sorting plans.

Station

A station consists of one or more segments of a top tool and a bottom tool.

Station plan

A station plan consists of at least one station on which the bending part is bent.

Stepped radius bend

A stepped radius bend is made when the angle and radius of a bent side is achieved by making a number of individual bends.

Stop point

Refers to a point or surface on the backgauge finger that can be used as a stop for a contact edge.

T

TCP-IP

TCP/IP (Transmission Control Protocol/Internet Protocol) is a network protocol and is known simply as the Internet protocol for short because of its huge relevance to the Internet. Computers are identified by IP addresses.

Tongue

A tongue refers to a bend whose bend line, when extended, runs along a contour.

Tool Setup Optimizer

The “Tool Setup Optimizer” puts bending jobs in the optimum order to reduce tool changes. This reduces tool times in the bending process. The module is particularly useful for bending a lot of different bending parts.

Tool setup plan

The tool setup plan can consist of one or more station plans. All the tools for the whole bending process are defined in one tool setup plan.

Tool time

Time for a tooling process. Tool time includes fetching and fitting tools. It includes: - Going from the machine to the tool cabinet - Putting away tools no longer needed - Maintaining the tools to be put away (cleaning, oiling) - Taking out the tools for the next stations plan(s) - Going from the tool cabinet back to the machine.

Tube Editor

“Tube Editor” is a software module in which tubular parts are prepared for machining.

Tube Nester

“Tube Nester” is a software module in which tubular parts can be nested to create jobs and exported to the relevant machine as cutting plans.

U

Update

An order update overwrites an existing order in the Plant Manager with an order from an ERP system.

V

View

In the “Modules” docking dialog box, a view refers to a subordinate entry in the module tree.

W

Wildcards

Wildcards are placeholders for characters in a character string. The most frequently used placeholders are the question mark (?) for any single character or the asterisk (*) for any number of characters.

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