elusoft ®

Instructions

eluCad Software

elusoft GmbH
Breitwasenring 4
D 72135 Dettenhausen

Phone +49(0)7157 526-6500
Fax   +49(0)7157 526-6526

info@elusoft.de
www.elusoft.de

©
12.09.2012

Table of contents (TOC)

1 Quick Guide - step by step guide to machining

The following chapters provide an easy and quick guide that will enable you to quickly carry out a machining operation.

This quick guide is subdivided into the following chapters:

1.1 Tools

1.1.1 Tool selection, step 1

In order to be able to carry out a machining operation correctly, the tool being used must be selected and assigned

To select a tool, follow the steps below:

1. In the menu bar under **VIEW**, open the **TOOLS** menu.
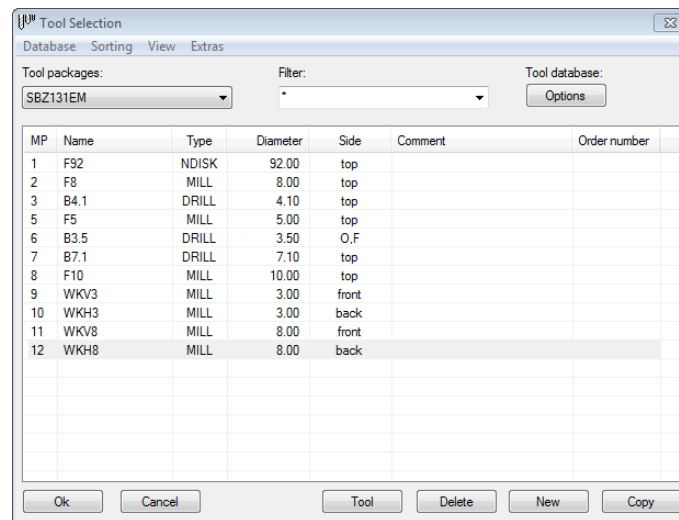
2. The **TOOL SELECTION** menu will open.

3. Press the **NEW** button to open the **NEW TOOL** menu.

4. Press the **ASSISTANT** button to open the **TOOLS** menu.

5. Press the **NEXT** button to open the **SELECTION** menu.
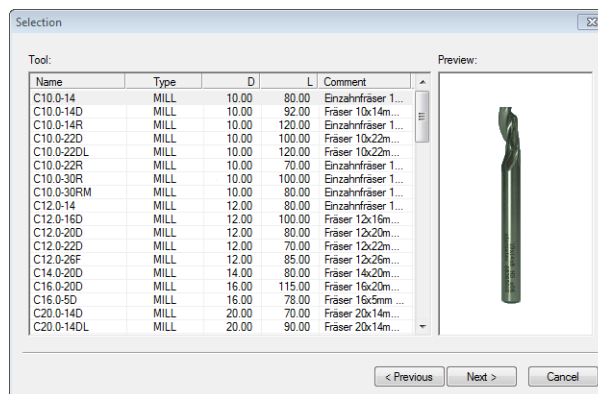
6. Select the tool you want from the list.
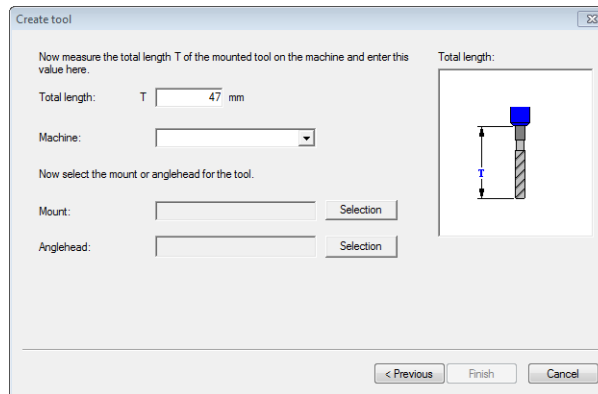
7. Press the **NEXT** button to go to the following step.

1.1.2 Setting up a tool, step 2

In order to be able to use a tool correctly, the holder that will be used needs to be selected and assigned.

In order to set up the tool that you want to use, follow the steps below:

1. The **CREATE TOOL** menu will open. This menu is used to assign the tool's length, the machine being used, and the tool holder being used.

2. The tool length must be measured with the depth gauge. The measured total length comprises the tool holder and the clamped tool!

⚠ CAUTION
⚠	When performing a measurement on the machining tool, please note that sharp tool blades pose an increased risk of injury!

⚠ INFORMATION
ℹ	The tool should should not be to unclamped too widely in order to prevent vibrations during machining. Vibrations can give rise to imprecise machining or damage the tool or the machine.
	The total length has to be measured each time a tool is clamped into the tool holder!
	This value must always be entered or it will be impossible to perform the following steps!

3. Enter the measured length into the **TOTAL LENGTH** input field.

4. In the **MACHINE** drop-down menu, press the ▼ button to select the machine model you will be using.

5. Select the holder or angle head you will be using:

   - **Holder:**   Press the **SELECTION** button to open the **FITTINGS** menu.

   - **Angle head:**   Press the **SELECTION** button to open the **ANGLE HEADS** menu.

6. In the menu that opens, select the holder or angle head you will be using.
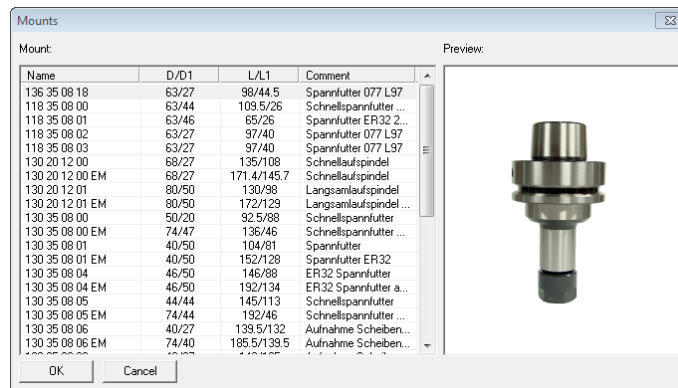
7. Press the **OK** button to apply your selection and close the menu.

8. Press the **FINISH** button to apply your selection. The **NEW TOOL** menu will be shown again.
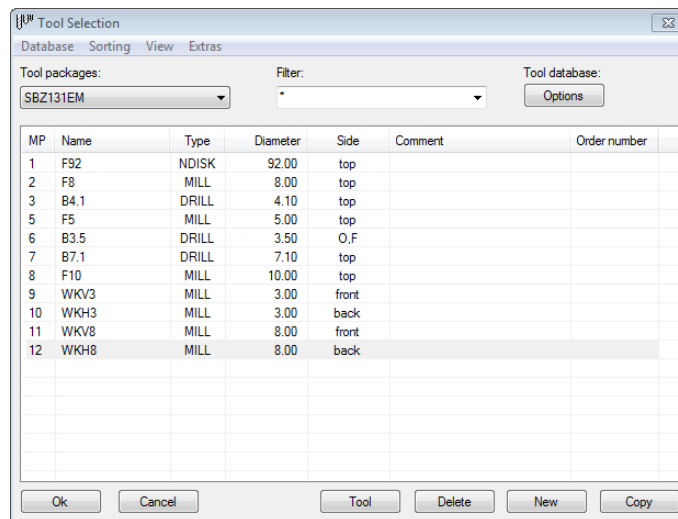
9. You can check the information for the new tool once again in the **NEW TOOL** menu.

1.1.3 Tool changer, step 3

In order to be able to perform the machining operation, the appropriate tools must be activated in the machine's tool changer. Depending on the machine type and features, there may be multiple magazines in the **TOOL CHANGER**.

In order to activate the tools that you want to use, follow the steps below:

1. The **NEW TOOL** menu shows the relevant input and selection fields.
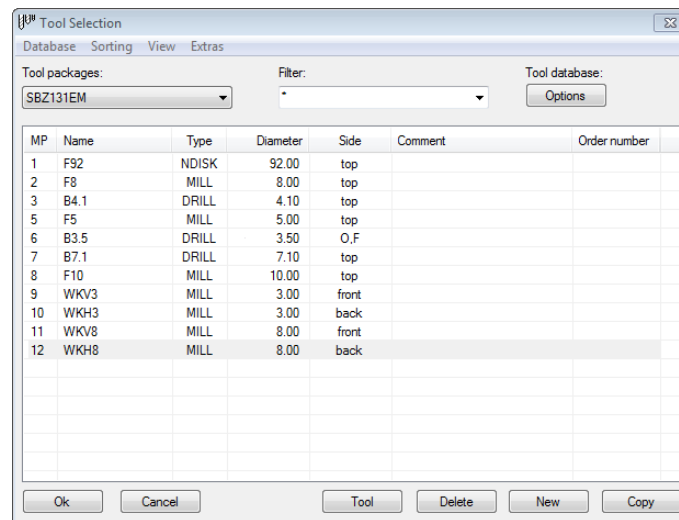
2. Select the **GENERAL** tab on the menu.

3. Enter the position in the machine's tool changer into the **MAGAZINE POSITION** input field.

4. In the **SIDE** field, enable the appropriate selection fields in order to define the tool machining procedure for the sides.

5. You can check the remaining information for the tool once again under the **GENERAL** tab.

6. You can check the tool's information once again under the **GEOMETRY** tab.

7. You can check the tool's information once again under the **TECHNOLOGY** tab.

8. You can check the tool's information once again under the **FITTING** tab.

9. Press the **OK** button to apply the information and close the menu.

10. The new tool will be shown on the **TOOL SELECTION** menu.

11. Press the **OK** button to close the menu.

1.2 Creating a job

An appropriate job must be selected in the job management dialog or a new one created to carry out a machining operation.

In order to create a new job, follow the steps below:

1. A job needs to be selected or a new job needs to be created (in the program's user interface).

   a) Press the **Öffnen** button to open the directory selection dialog box that is used to open already existing jobs.

   b) Press the **Neu** button to start creating a new job. Make sure the job has a descriptive and unique name.
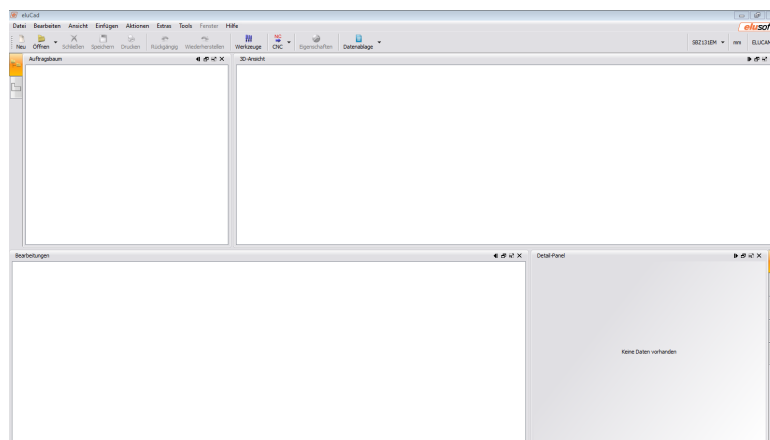
1.2.1 Open existing job

You can access and use previously stored jobs at a later point in time.

To open a job that already exists, follow the steps below:

1. Press the ![Öffnen] button to open the directory selection dialog box.
2. In the directory, select the location where the job you want is stored.
3. Press the **OPEN**button to open the selected job. The job will be shown in eluCAD.
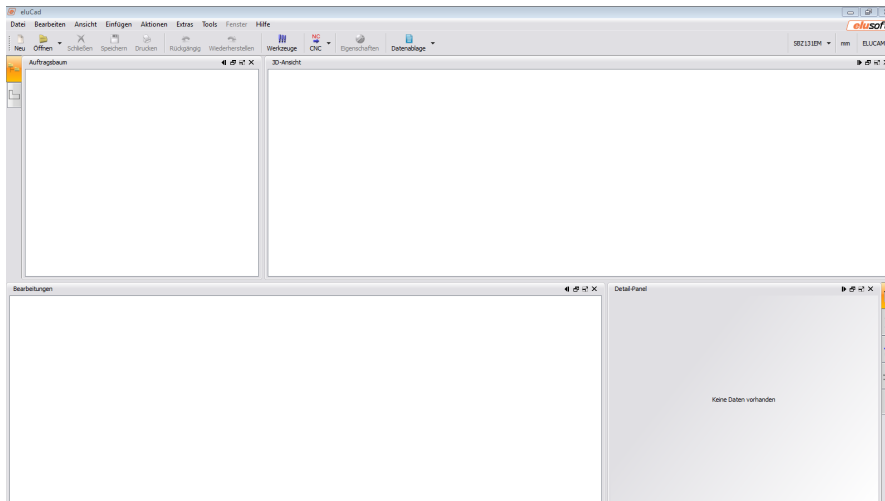
1.2.2 Create new job

You can add the programs you need when creating a new job.

To create a new job, you must perform the following steps:

1. Press the **Neu** button to open an input field used to enter the job's name.

2. Enter the name you want or the number for the job into the **JOB NUMBER** field. Make sure the job has a descriptive and unique name.

3. You can use the **REMARK** field to add additional information concerning the job.

4. Press the **NEXT** button to accept the entry. The **PROFILE DATABASE** menu will open.

5. Select an appropriate profile from the **PROFILE DATABASE** menu.

6. Press the **OK** button to apply your selection and close the **PROFILE DATABASE** menu.

7. The new job you just created will be shown in the job tree.

1.2.2.1 Create a new profile

To create a new profile, use the **PROFILE DATABASE** menu.

To create a new profile, perform the following steps: In the menu bar under **VIEW,** open the **PROFILE DATABASE** menu.

1. The **PROFILE DATABASE** menu can be used to create a profile manually [9] or with a DXF import [12].
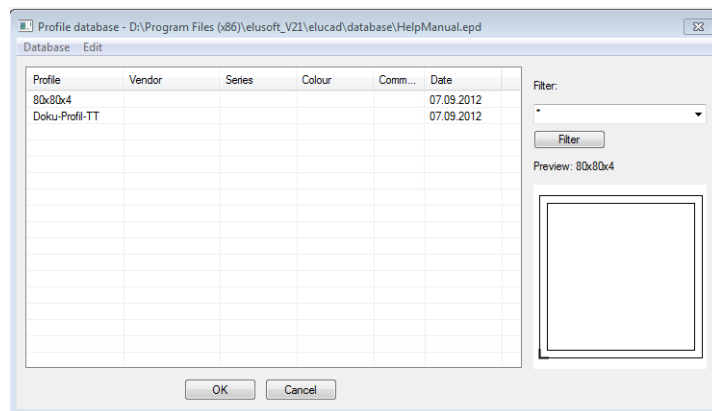
The further procedure is described in the respective chapters.

1.2.2.1.1 Creating a profile manually

Profiles can be created manually in the **PROFILE DATABASE** menu.

Carry out the following steps to manually create a new profile:

1. Open the **PROFILE DATABASE** as described in the [Creating a profile](#) 8 chapter.

2. In the menu bar under **EDIT | NEW ENTRY**, open the **PROFILE** menu.

3. In the **PROFILE NUMBER** input field, enter the designation of the new profile. Make sure the profile has a descriptive and unique name.

4. Enter or select all further basic information about the profile in the remaining input and selection fields.
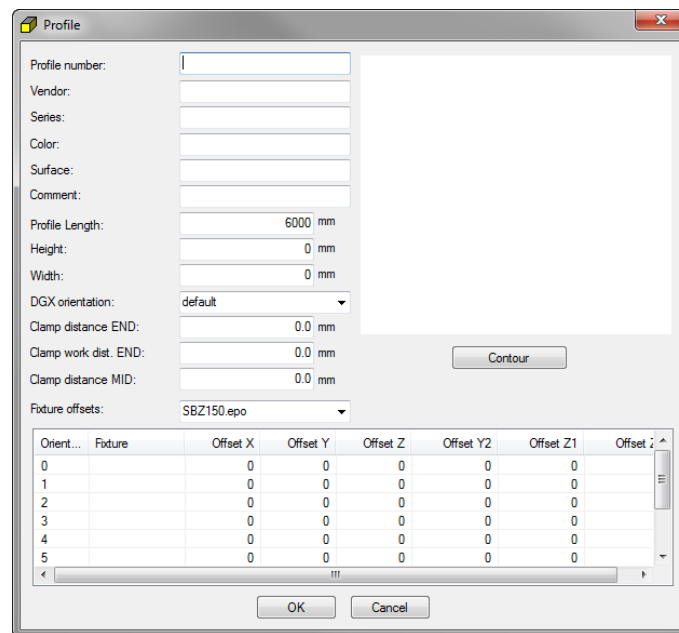
5. Press the **CONTOUR** button to open the **CAD EDITOR** menu so that you can create the profile you want
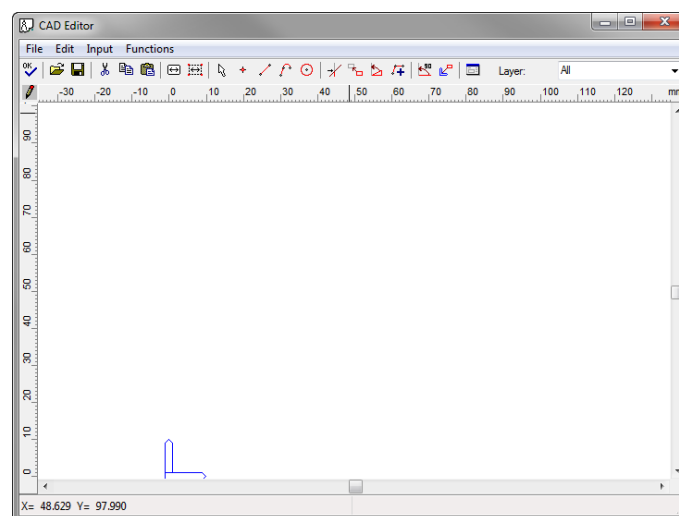
6. In the menu bar under **FUNCTIONS | STANDARD PROFILES,** open the **STANDARD PROFILE** menu.

7. Select the desired basic type via the **PROFILE TYPE** selection field.

8. Enter the appropriate dimensions for the profile into the input fields.

9. Press the **OK** button to apply the information in the **CAD EDITOR** menu and close the menu.

10. The new profile is displayed purely as a DXF profile in the **CAD EDITOR** menu.

11. Press the [OK] button to accept the new profile and close the menu.
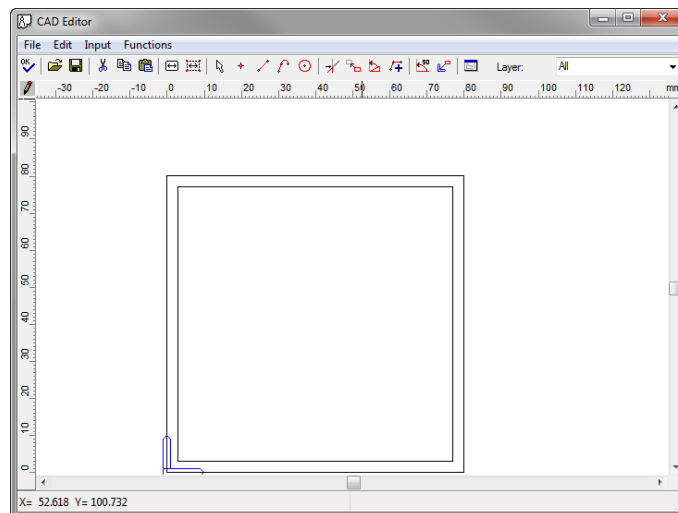
12. The **PROFILE** menu reopens with the data of the newly created profile.
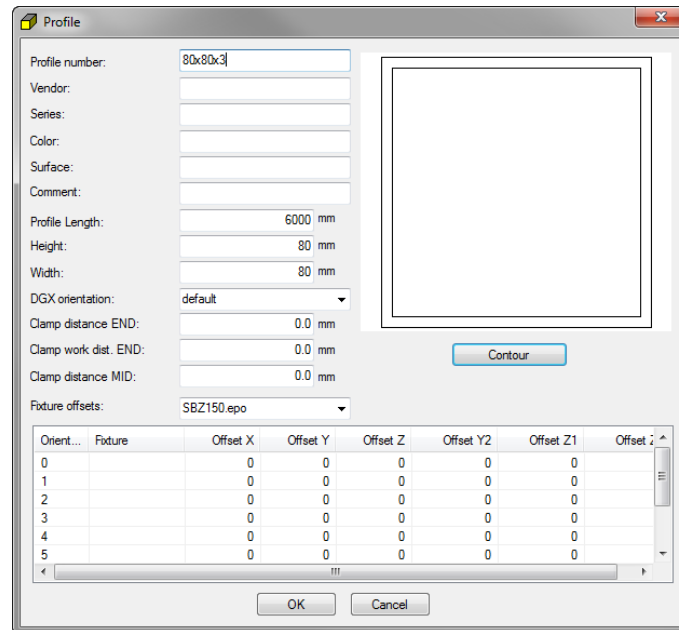
13. The row in the support blocks table with the desired profile position must be selected.

14. Double-click on the row in order to open the **SUPPORT BLOCK** menu.

15. Enter the name for the new support block into the **NAME** input field. Make sure the support block has a descriptive and unique name.

16. Enter the appropriate values for the support blocks into the remaining input fields.
    If the **DISPLAY PROFILE VIEW** field is enabled, the offset of the profile due to the support blocks can be displayed in the profile view.

    Use the button to select previously stored support blocks from the **SUPPORT BLOCK DATABASE** menu. Select the support block you need and apply your selection with **OK**.
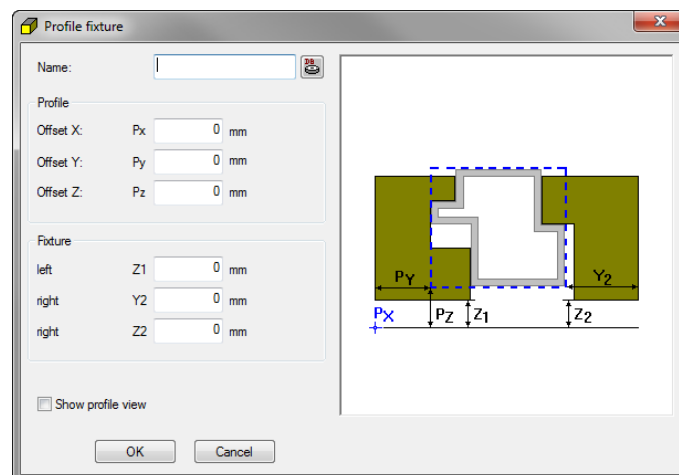
17. A Z-axis offset can also be specified in the **SUPPORT BLOCK** menu.

18. Press the **OK** button to apply the support block information in the **PROFILE** menu and close the menu.

19. If you want to create additional support blocks, repeat steps 13 through 16.

20. Once you have entered all the information you want, press the **OK** button to accept the new profile in the **PROFILE DATABASE** menu and close the menu.

1.2.2.1.2 Importing a profile in DXF format

Profiles in the DXF format can be imported in the **PROFILE DATABASE** menu.

The following steps are required to import a new profile in DXF format:

1. Open the **PROFILE DATABASE** as described in the [Creating a profile](#) ⌐8⌐ chapter.

2. In the menu bar under **EDIT | NEW ENTRY**, open the **PROFILE** menu.
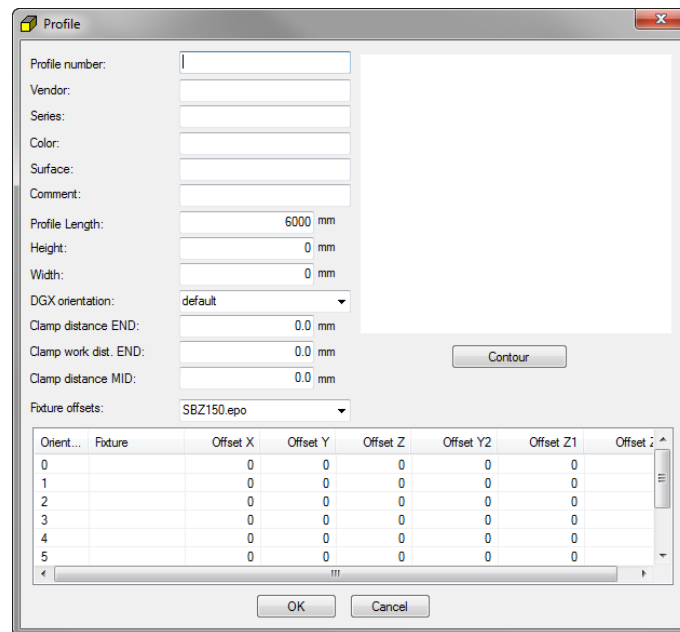
3. Press the **CONTOUR** button to open the **CAD EDITOR** menu so that you can create the profile you want
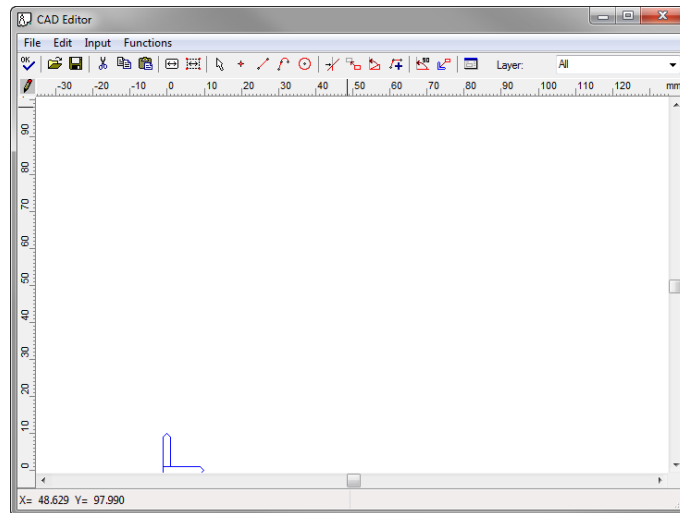
4. In the menu bar under **FILE | OPEN DXF FILE,** open the **OPEN** submenu.

5. Use the directory selection item to select the storage location for the desired DXF file.

6. Use the **OPEN** button to use the selected file and close the submenu.

7. The opened drawing is displayed in the scale in which it was created in the **CAD EDITOR**.

⚠ INFORMATION	
i	The machine uses the actual dimensions of the drawing. If the drawing was not created in 1:1, scaling is required in the **CAD EDITOR** menu.

8. Select the drawing with the keys **CTRL + A**.

9. Use the button to shift the drawing to the origin.

10. Press the button to accept the newly created profile.

11. The **PROFILE** menu reopens with the data of the newly created profile.
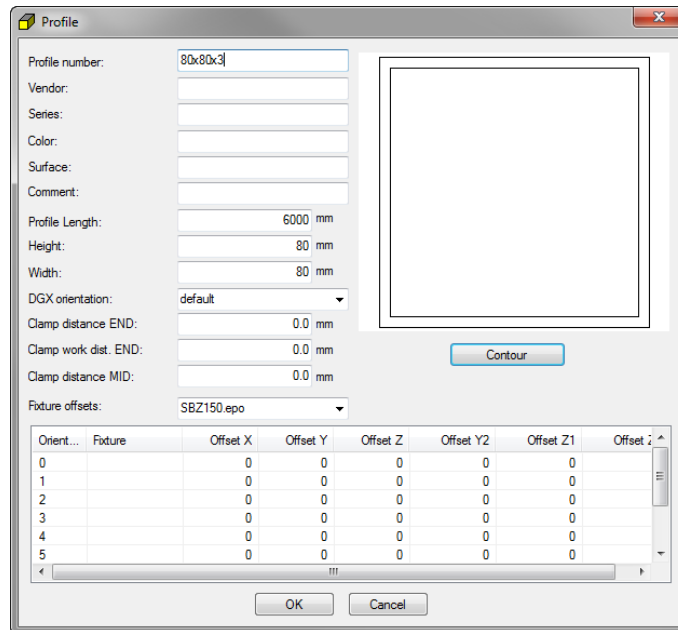
12. In the **PROFILE NUMBER** input field, enter the designation of the new profile. Make sure the profile has a descriptive and unique name.

13. Enter or select all further basic information about the profile in the remaining input and selection fields.
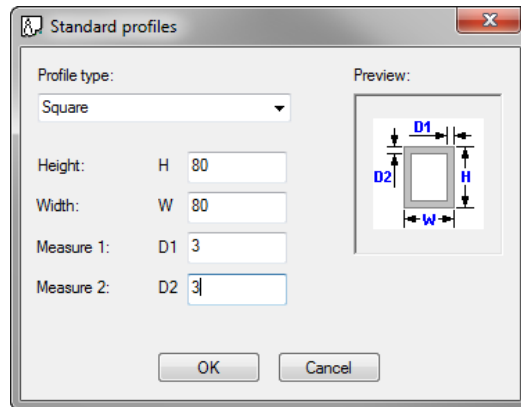
14. The row in the support blocks table with the desired profile position must be selected.

15. Double-click on the row in order to open the **SUPPORT BLOCK** menu.

16. Enter the name for the new support block into the **NAME** input field. Make sure the support block has a descriptive and unique name.

    Use the button to select previously stored support blocks from the **SUPPORT BLOCK DATABASE**. Select the support block you need and apply your selection with **OK**.

17. Enter the appropriate values for the support blocks into the remaining input fields.
    If the **DISPLAY PROFILE VIEW** field is enabled, the offset of the profile due to the support blocks can be displayed in the profile view.
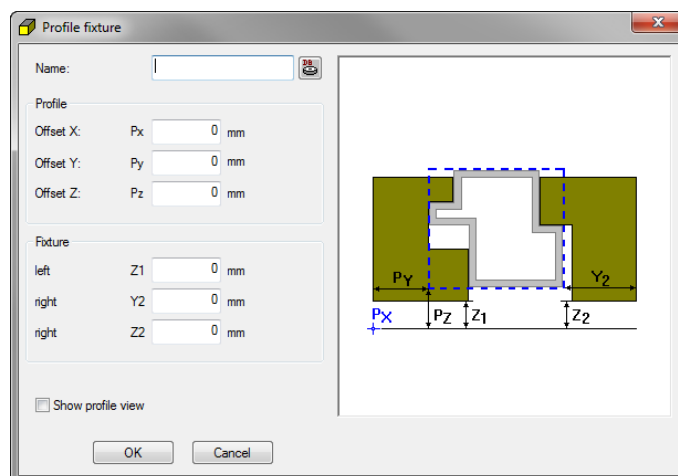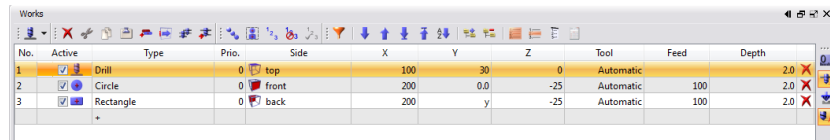
18. A Z-axis offset can also be specified in the **SUPPORT BLOCK** menu.

19. Press the **OK** button to apply the support block information in the **PROFILE** menu and close the menu.

20. If you want to create additional support blocks, repeat steps 14 through 19.

21. Once you have entered all the information you want, press the **OK** button to add the new profile to the profile database and close the menu.

22. Press the **OK** button to close the **PROFILE DATABASE** menu.

1.3 Machining program - View

1.3.1 Creating a machining operation

The individual machining tasks can be created through manual input of the data and values in the Machining tasks table.

To create a new machining operation, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks .If a line containing a machining operation is selected, it can be overwritten!

2. Use a double click on the **TYPE** cell to select the basic type in the selection window.

3. The machining task is activated automatically but can be deactivated at any time.

4. Use a double click on the **SIDE** cell to select the desired machining side via the selection window.

5. Enter the X-position of the machining operation in the **X** cell.

6. Enter the Y-position of the machining operation in the **Y** cell.

7. Enter the Z-position of the machining operation in the **Z** cell.

8. The tool is defined automatically by default in the **Tool** cell.
Double-click on the cell in order to enable the **AUTOMATIC** button. Using this button, a tool can be defined via the **TOOL SELECTION** menu.

9. The value of 100% is already entered in the **Feed** cell.
The desired value can be entered by selecting the cell.

10. Enter the maximum depth of the machining task in the **Depth** cell.
When a value is entered, further buttons will be displayed.

11. The button triggers automatic wall detection. The value determined is displayed in the **DEPTH** cell and is added to the Depth table.

12. Open the **DETAIL** ![icon] tab and enter the respective values for the machining task or select them.
Different input boxes can be active depending on which type is selected.

13. Open the **PARAMETERS** ![icon] tab and enter the respective data or select them.
Different input boxes can be active depending on which type is selected.

14. Open the **DEPTH TABLE** tab and enter the respective data or select them.

	D	M	F	RPM
1	30.00 mm	☐	100%	100%
2	31.80 mm	☑	80 %	100 %

Depth table

☑ Automatic security distance

If automatic wall detection was performed in the **MACHINING TASKS** table, the values are already present in the depth table.

15. Repeat steps 1-14 to create additional machining tasks.

2 Using the CAM software

The use of the CAM software is described in the following chapters. They exemplify procedures and options available for use with software.

The solutions proposed are examples only and it may also be possible to achieve the results using other solutions.

2.1 Profile data

The use of the profile data is described in the following chapters.

2.1.1 Creating a profile

The data of the profile to be machined are required to be able to perform a machining task. The **PROFILE DATABASE** menu manages all the profiles used and saves the respective data from the different profile positions. To achieve a better overview, the **PROFILE DATABASE** menu should always be maintained in an orderly fashion and meaningful names should be given.

⚠ INFORMATION
i   Incorrect profile data can result in collisions and damages to the machine!

To create new profiles, perform the following steps:

1. In the menu bar under **VIEW,** open the **PROFILE DATABASE** menu.
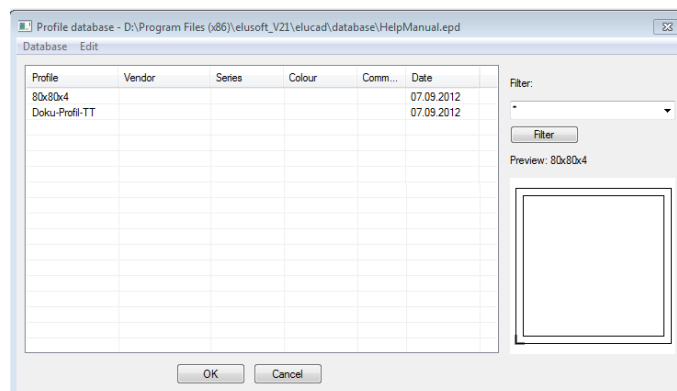
2. In the **PROFILE DATABASE** menu, a profile can be <u>created manually</u> [20] or <u>imported in DXF format</u> [23]. The further procedure is described in the respective chapters.

2.1.1.1 Creating a profile manually

Profiles can be created manually in the **PROFILE DATABASE** menu.

Carry out the following steps to manually create a new profile:

1. Open the **PROFILE DATABASE** as described in the <u>Creating a profile</u> 19 chapter.

2. In the menu bar under **EDIT | NEW ENTRY**, open the **PROFILE** menu.

3. In the **PROFILE NUMBER** input field, enter the designation of the new profile.

4. Enter or select all further basic information about the profile in the remaining input and selection fields.

5. Press the **CONTOUR** button to open the **CAD EDITOR** menu so that the desired profile can be created.

6. In the menu bar under **FUNCTIONS | STANDARD PROFILES,** open the **STANDARD PROFILE** menu.

7. Select the desired basic type via the **PROFILE TYPE** selection field.

8. Enter the respective dimensions in the input fields.

9. Pressing the **OK** button applies the data in the **CAD EDITOR** menu and closes the **STAN-DARD PROFILE** menu.

10. The new profile is displayed purely as a DXF profile in the **CAD EDITOR** menu.

11. Press the [OK] button to accept the newly created profile and close the **CAD EDITOR**.
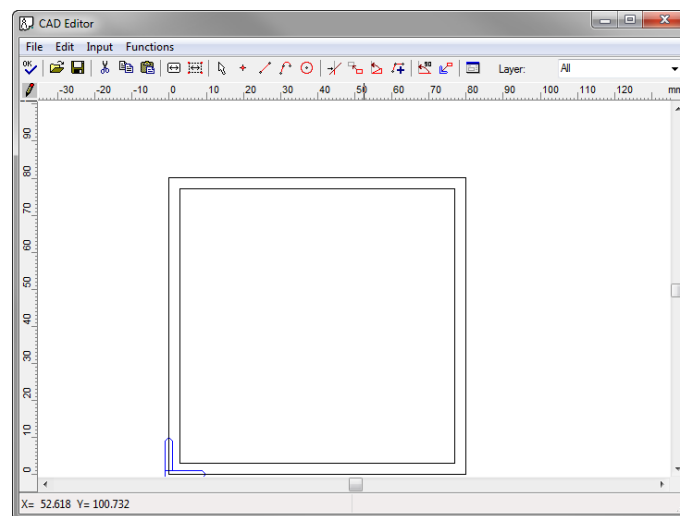
12. The **PROFILE** menu reopens with the data of the newly created profile.
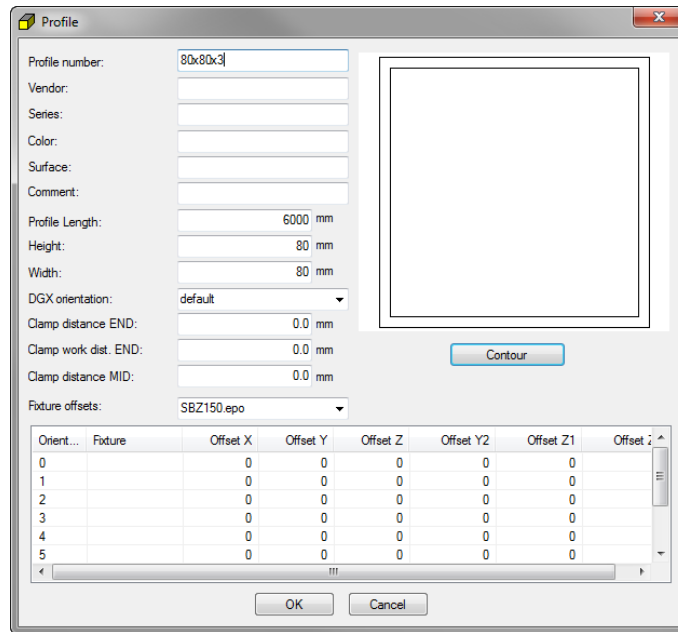
13. The profile-specific support blocks used can be defined in the support blocks table.
    If support blocks need to be created for the new profile, proceed as described in the chapter on .

14. Once all of the data has been entered, press **OK** to accept the new profile and close the **PRO-FILE** menu.

2.1.1.2 Importing a profile in DXF format

Profiles in the DXF format can be imported in the **PROFILE DATABASE** menu.

The following steps are required to import a new profile in DXF format:

1. Open the **PROFILE DATABASE** as described in the [Creating a profile](#) 19 chapter.

2. In the menu bar under **EDIT | NEW ENTRY**, open the **PROFILE** menu.

3. In the **PROFILE NUMBER** input field, enter the designation of the new profile.

4. Enter or select all further basic information about the profile in the remaining input and selection fields.

5. Press the **CONTOUR** button to open the **CAD EDITOR** menu so that the desired profile can be created.
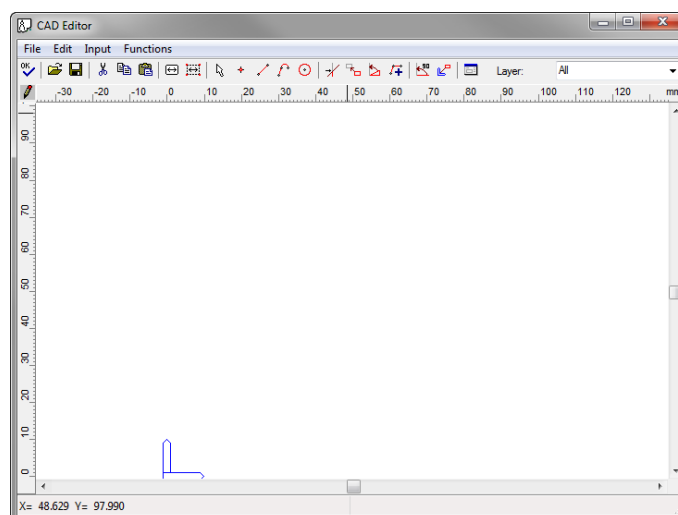
6. In the menu bar under **FILE | OPEN DXF FILE,** open the **OPEN** submenu.

7. Use the directory selection item to select the storage location for the desired DXF file.

8. Use the **OPEN** button to use the selected file and close the submenu.

9. The opened drawing is displayed in the scale in which it was created in the **ELUCAD EDITOR**.

⚠ **INFORMATION**
The machine uses the actual dimensions of the drawing. If the drawing was not created in 1:1, scaling is required in the **CAD EDITOR** menu.

10. Select the drawing with the keys **CTRL + A**.

11. Use the [icon] button to shift the drawing to the origin.

12. Press the [OK icon] button to accept the newly created profile.

13. The **PROFILE** menu reopens with the data of the newly created profile.
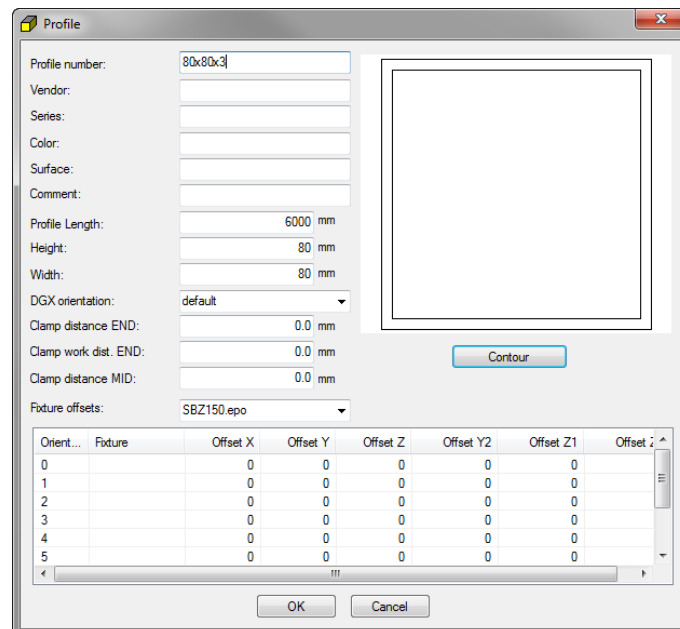
14. The profile-specific support blocks used can be defined in the support blocks table.
If support blocks need to be created for the new profile, proceed as described in the chapter on .

15. Once all of the data has been entered, press **OK** to accept the new profile and close the **PROFILE** menu.

2.1.1.3   Copying a profile

The data of previously existing profiles can be copied in the **PROFILE DATABASE** menu.

To copy a profile, perform the following steps:

1. Open the **PROFILE DATABASE** as described in the <u>Creating a profile</u> [19] chapter.

2. Select the desired profile in the opened menu.
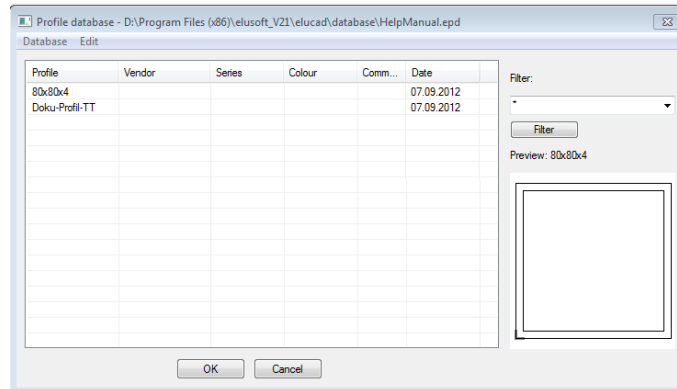
3. In the menu bar under **EDIT | COPY ENTRY**, open the **PROFILE** menu.

4. All basic information for the profile is displayed in the opened menu.

5. A new designation for the profile is entered in the **PROFILE NUMBER** input field automatically. The designation can be changed accordingly at any time.
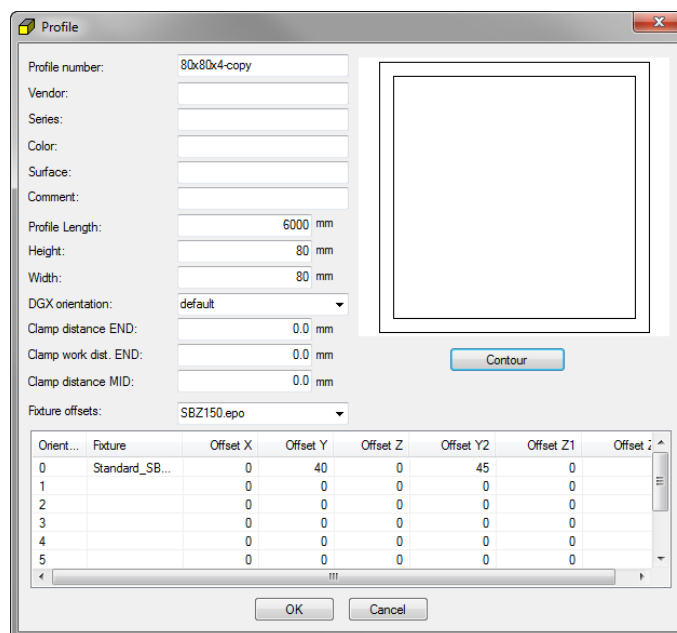
6. The data in the further selection and input fields can be modified or adapted accordingly.

7. Pressing the **OK** button applies the data and closes the **PROFILE** menu.

2.1.1.4   Modifying a profile

The data of previously existing profiles can be modified in the **PROFILE DATABASE** menu.

To edit a profile, perform the following steps:

1. Open the **PROFILE DATABASE** as described in the chapter.

2. Select the desired profile in the opened menu.

3. In the menu bar under **EDIT | EDIT ENTRY**, open the **PROFILE** menu.

4. All basic information for the profile is displayed in the opened menu.

5. The data in the selection and input fields can be modified or adapted accordingly.
   Pressing the **OK** button applies the modified data and closes the **PROFILE** menu.

2.1.1.5 Support blocks

Support blocks aid in the machining of the profile. The right selection is very important to achieve sound and precise machining results. Depending on the machining operation, it may be necessary to swap the support blocks when changing the clamping position.

Support blocks have an offset in the X, Y and Z directions. Two different types of support blocks are used and displayed on the machine.

⚠ **INFORMATION**	
ℹ	When entering data, always make sure to enter the correct sign!
	Before starting a machining operation, make sure to check that the profile support blocks are correctly configured!

- **Profile-specific support blocks:**

    The profile-specific **SUPPORT BLOCKS** can differ depending on the profile and clamping position. The support blocks add an additional offset to the profile.

    To set the profile support blocks in the eluCad software, the following steps must be performed:

    1. Open the **PROFILE DATABASE** in the menu bar under **VIEW | PROFILE DATABASE**.

    2. In the menu bar under **EDIT | EDIT ENTRY**, open the **PROFILE** menu.
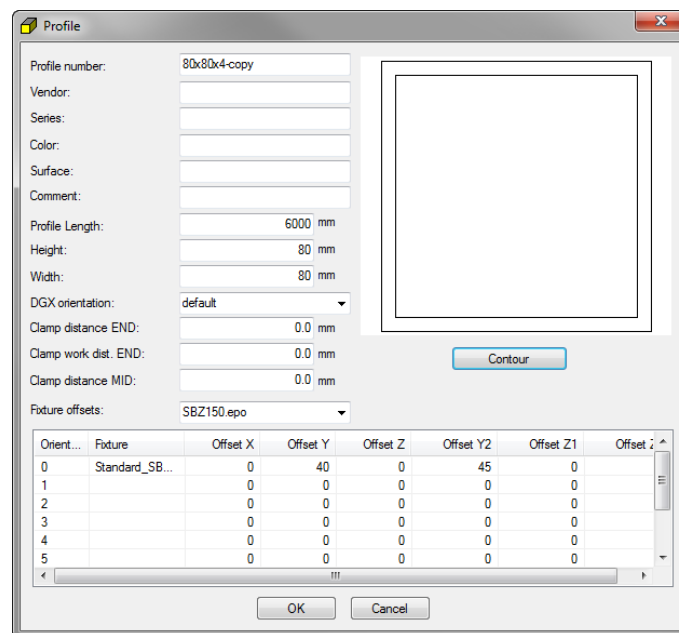
    3. The row in the support blocks table with the desired profile position must be selected.

4. Use a double click on the **S.BLOCK** cell to open the **SUPPORT BLOCK** menu.
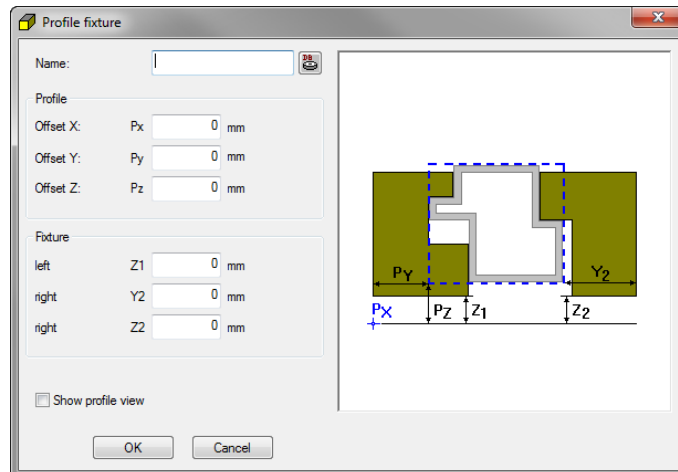
5. The corresponding data for the support blocks must be entered in the input fields of the **SUPPORT BLOCK** menu.
   If the **DISPLAY PROFILE VIEW** field is enabled, the offset of the profile due to the support blocks can be displayed in the profile view.
   The profile support blocks are stored as 3D solids and are available for collision-related considerations.

6. Use the button to select previously stored support blocks from the **SUPPORT BLOCK DATABASE**

7. Pressing **OK** applies the data of the support blocks in the **PROFILE** menu and closes the **PROFILE SUPPORT BLOCKS** menu.
   The support blocks are set to 0 by default.

⚠ **INFORMATION**	
i	It is recommended that the profile be clamped in the machine before machining is performed and that the correct and proper use of the profile support blocks is tested.
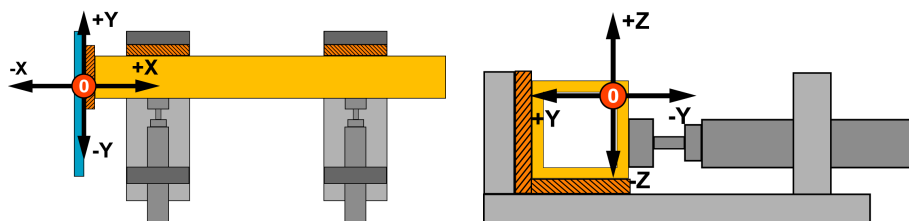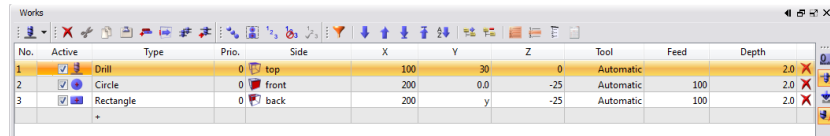
8. If additional profile support blocks are to be created, repeat items 3 and 7 at the desired profile position.

9. Pressing the **OK** button applies the modified data to the selected profile and closes the **PROFILE** menu.

10. Pressing the **OK** button applies the data and closes the **PROFILE DATABASE** menu.

2.1.1.5.1 Editing the support block geometry data

If it is necessary to modify the support block geometry data, this can be accomplished in the **SUP-PORT BLOCKS DATABASE** menu.

The following steps must be performed to modify the support block geometry data:

1. In the **PROFILE SUPPORT BLOCKS** menu, use the button to open the **SUPPORT BLOCKS DATABASE** menu.

2. Select the desired support block in the **SUPPORT BLOCK DATABASE** menu.

3. In the menu bar under **EDIT | EDIT ENTRY,** the **SUPPORT BLOCK** menu opens.

4. All basic information for the support block is displayed in the **SUPPORT BLOCK** menu.

5. Use the **CONTOUR** button to open the **CAD EDITOR** menu.

6. The contour of the profile support block can be edited in the opened **CAD EDITOR** menu.

7. Open the segment input window by double clicking on the contour.

8. The data in the selection and input fields can be modified or adapted accordingly. The data that is entered is used later for collision-related considerations. It is therefore important that the values match the actual profile support blocks.

9. Once all of the data has been entered, confirm by pressing **OK** twice to add the support block to the support block database.

10. Pressing the **OK** button closes the **PROFILE DATABASE** and assigns the support blocks to the profile.

2.2 Machining task data

The use of the machining task data is described in the following chapters.

2.2.1 Creating a new machining task manually

The individual machining tasks can be created through manual input of the data and values in the Machining tasks table.

To create a new machining operation, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks .If a line containing a machining operation is selected, it can be overwritten!

2. Use a double click on the **TYPE** cell to select the basic type in the selection window.

3. The machining task is activated automatically but can be deactivated at any time.

4. Use a double click on the **SIDE** cell to select the desired machining side via the selection window.

5. Enter the X-position of the machining operation in the **X** cell.

6. Enter the Y-position of the machining operation in the **Y** cell.

7. Enter the Z-position of the machining operation in the **Z** cell.

8. The tool is defined automatically by default in the **Tool** cell.
   Use a double click on the cell to activate the **AUTOMATIC** button. Using this button, a tool can be defined via the **TOOL SELECTION** menu.

9. The value of 100% is already entered in the **Feed** cell.
   The desired value can be entered by selecting the cell.

10. Enter the maximum depth of the machining task in the **Depth** cell.
    When a value is entered, further buttons will be displayed.

11. The button triggers automatic wall detection. The value determined is displayed in the **DEPTH** cell and is added to the Depth table.

12. Open the **DETAIL** tab and enter the respective values for the machining task or select them.
Different input boxes can be active depending on which type is selected.

Details: Rectangle		▶ ⊡ ⊡ ✕
Length:	30	mm
Width:	20	mm
Corner radius:	4	mm
Angle:	0.0	°
A-Achse:	90.0	
C axis::	0.0	
Tool diameter:	0.0	mm
Comment:		
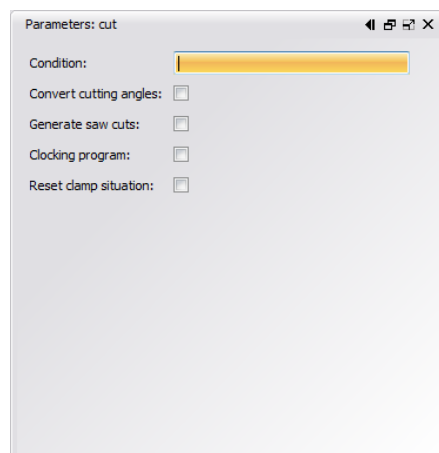
13. Open the **PARAMETERS** tab and enter the respective data or select them.
Different input boxes can be active depending on which type is selected.

Parameters: cut		◀ ⊡ ⊡ ✕
Condition:		
Convert cutting angles:	☐	
Generate saw cuts:	☐	
Clocking program:	☐	
Reset clamp situation:	☐	

14. Open the **DEPTH TABLE** tab and enter the respective data or select them.
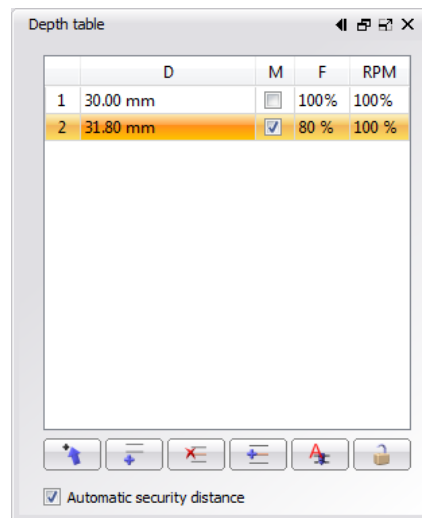
If automatic wall detection was performed in the **MACHINING TASKS** table, the values are already present in the depth table.

15. Repeat steps 1-14 to create additional machining tasks.

2.2.2 Machining task examples - eluCAM coordinate system

The following are examples of simple standard machining operations with the eluCAM coordinate system. They exemplify procedures and options available for use with eluCAD Software.

The solutions proposed are examples only and it may also be possible to achieve the results using other solutions. The individual solutions may also be expanded or combined.

2.2.2.1 Creating a new machining task at TOP

This example shows manual compilation of a machining operation from the top.
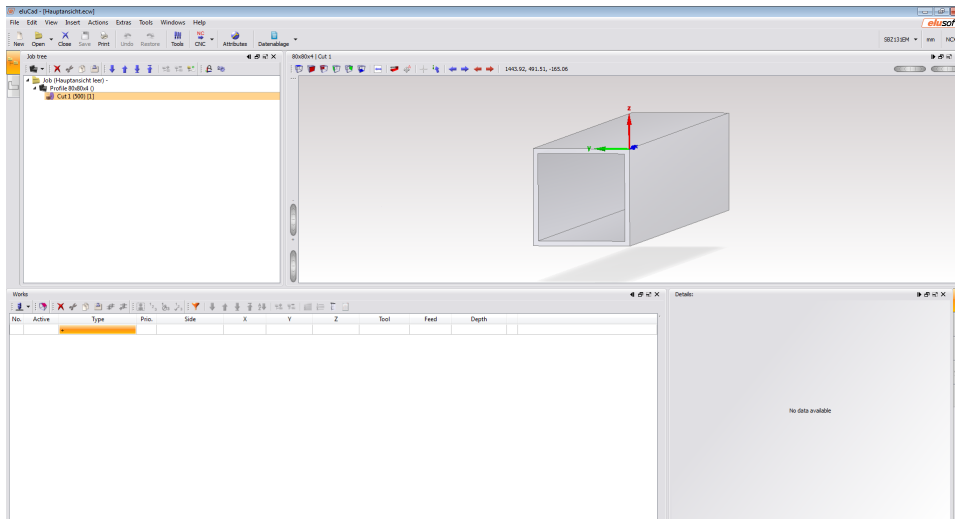
Default machining task:
- Circle pocket with 12 mm diameter
- Position top; X = 50 mm; Y = 30 mm
- Work feed rate 80%
- Depth 10 mm

To create the machining operation from the top, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **CIRCLE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **TOP** machining side is selected automatically.

5. Enter 50 mm for the X-position in the **X** cell.

6. Enter 30 mm for the Y-position in the **Y** cell.

7. The **Z** cell already contains 0 for the Z-position.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. Enter the value of 80% in the **FEED RATE** cell.

10. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

11. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    -   **Diameter:**   Enter 12 mm for the diameter of the machining task.

12. Check all inputs in the profile view.

2.2.2.2 Creating a new machining task at FRONT

This example shows manual compilation of a machining operation from the front.
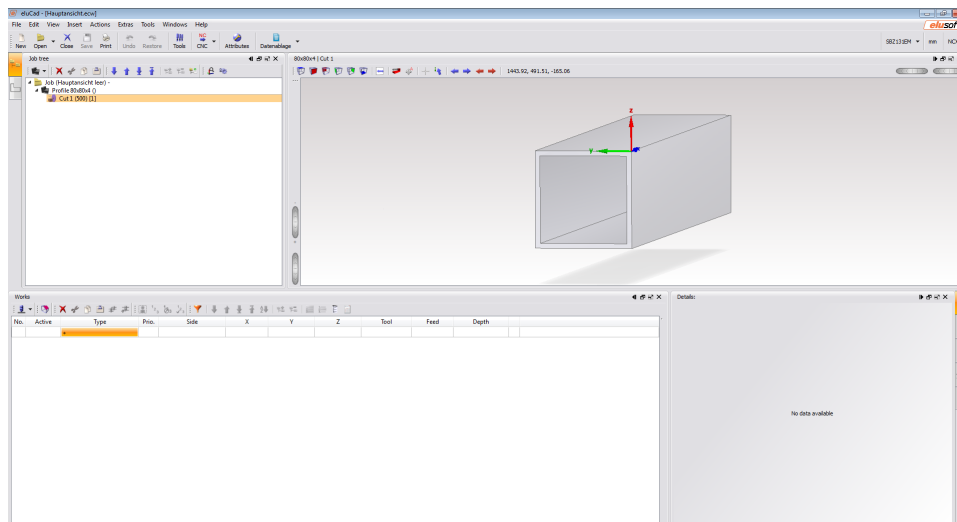
Default machining task:
- Long hole of length 40 mm; width 10 mm
- Position front; X = 30 mm; Z = -15 mm
- Machining angle 15°
- Work feed rate 100%
- Depth 10 mm

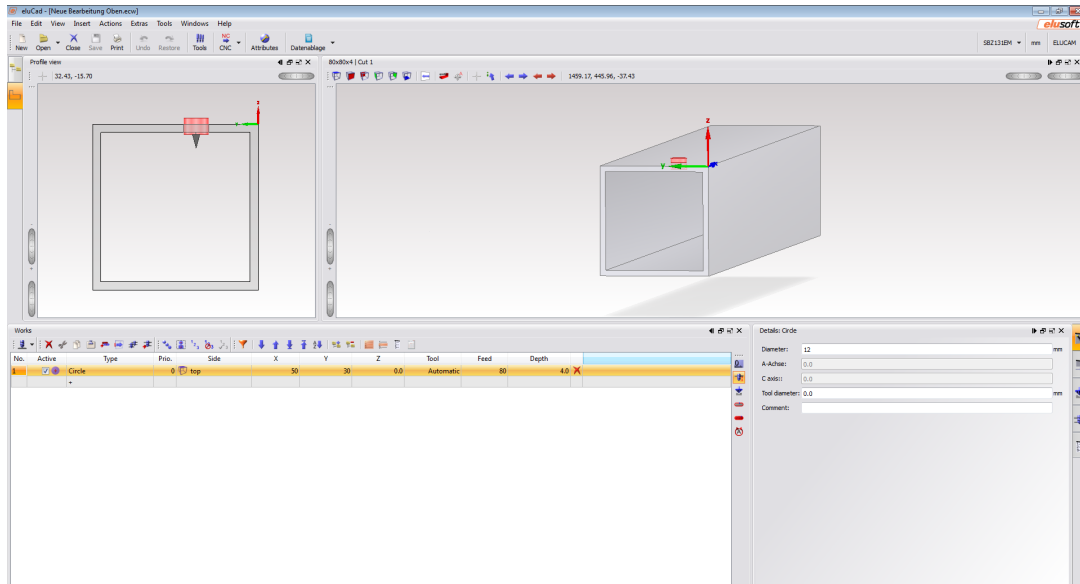To create the machining operation from the front, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **LONG HOLE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. Use a double click on the **SIDE** cell to select the **FRONT** machining side in the selection window.

5. Enter 30 mm for the X-position in the **X** cell.

6. The **Y** cell already contains 0 mm for the Y-position.

7. Enter -15 mm for the Z-position in the **Z** cell.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. The value of 100% is already entered in the **FEED** cell.

10. Enter the value of 10 mm in the **DEPTH** cell. Use the ![button] button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the ![button] button to manually create or change the depth table.

---

11. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

- **Length:**   Enter 30 mm for the length of the machining task.

- **Width:**   Enter 10 mm for the width of the machining task.

- **Angle:**   Enter 15° for the machining angle.

- **Tool diameter:**   Diameter of the tool used.
  Only required if no automatic assignment is to be made!

12. Check all inputs in the profile view.
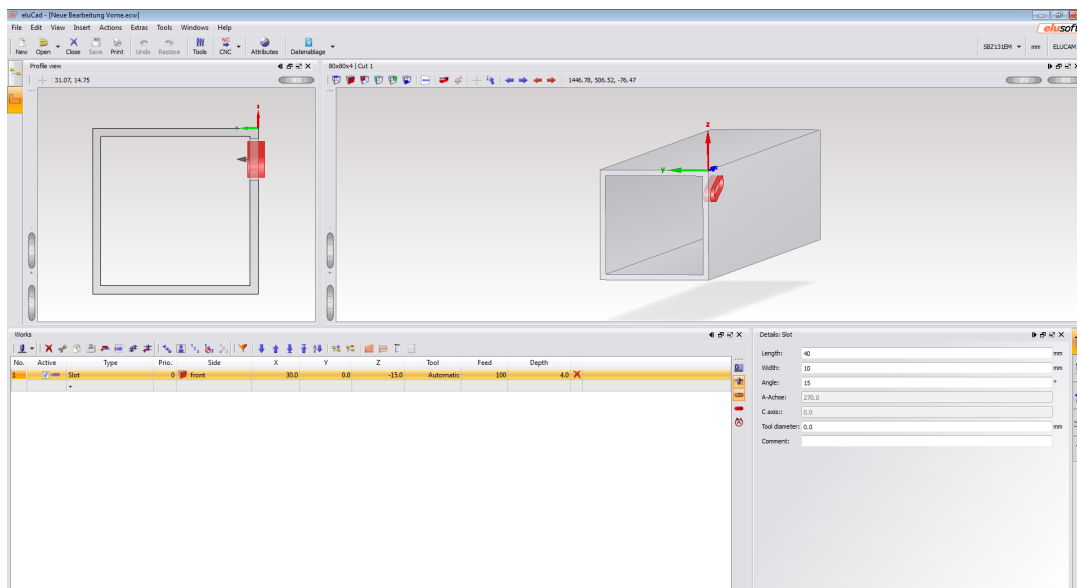
2.2.2.3   Generate new machining task at BACK

This example shows manual compilation of a machining operation from the rear.
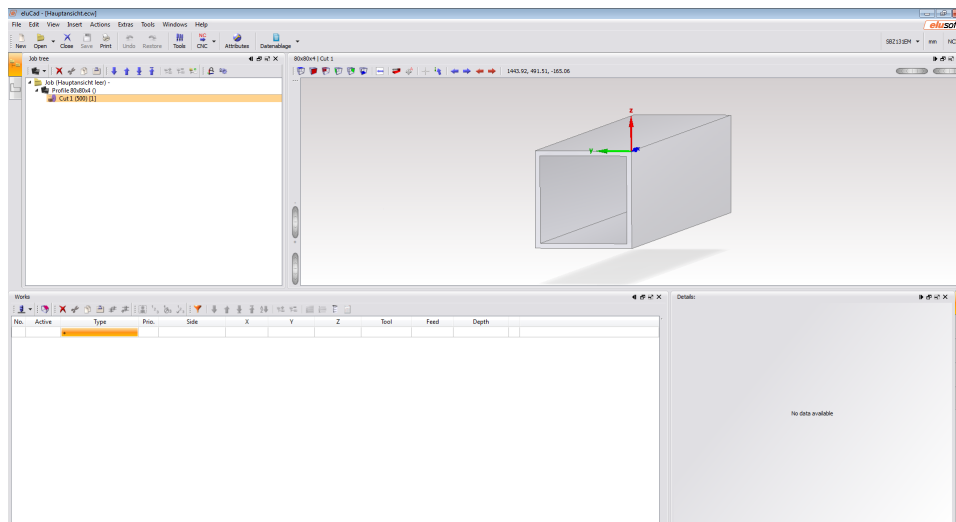
Default machining task:
- Rectangle of length 60 mm; width 20 mm
- Position at back; X = 50 mm; Y = variable *PW* or *Y* (profile width); Z = -10 mm
- Corner radius 5 mm; machining angle 0°
- Work feed rate 100%
- Depth 10 mm

To create the machining operation from the rear, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **RECTAN-GLE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. Use a double click on the **SIDE** cell to select the **REAR** machining side in the selection window.

5. Enter 50 mm for the X-position in the **X** cell.

6. Enter the variable *PW* or *Y* (profile width) for the Y-position in the **Y** cell.

7. Enter -10 mm for the Z-position in the **Z** cell.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. The value of 100% is already entered in the **FEED** cell.

10. Enter the value of 10 mm in the **DEPTH** cell. Use the ⬚ button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the ⬚ button to manually create or change the depth table.

11. Use the ⬚ button to deactivate the additional depth because the depths correspond directly to the entries.

12. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

- **Length:**   Enter 60 mm for the length of the machining task.

- **Width:**   Enter 20 mm for the width of the machining task.

- **Corner radius:**   Enter 5 mm for the maximum corner radius of the machining task.
                       The software may only automatically assign a milling cutter with a maximum of 10 mm.

- **Angle:**   Enter 0° for the machining angle.

- **Tool diameter:**   Diameter of the tool used.
                       Only required if no automatic assignment is to be made!

13. Check all inputs in the profile view.

2.2.2.4   Generating new LEFT and RIGHT machining tasks

This example shows manual compilation of a machining operation from the left and the right.

Default machining task:

Left:
- Line of length 15 mm; width of 8 mm
- Position left; X = 0 mm; Y = 30 mm (start point); Z = 5 mm (start point)
- Machining angle 270°
- Direction, centre
- Work feed rate 100%
- Depth 5 mm

Right:
- Line of length 30 mm; width 10 mm
- Right position; X = variable *PL* or *H* (profile length); Y= 15 mm (start point); Z = -2 mm (start point: wall thickness centre)
- Machining angle 0°
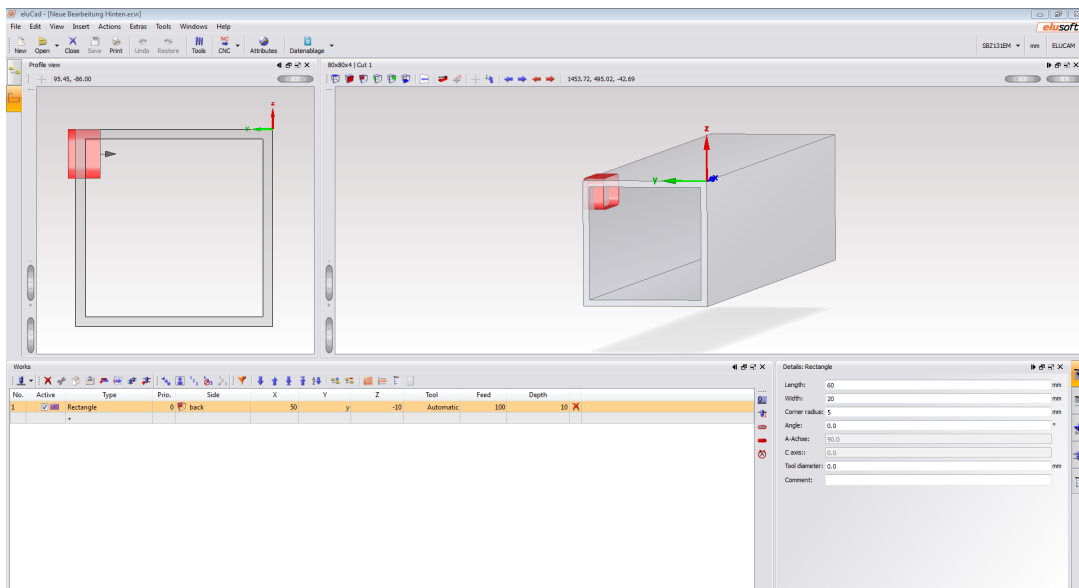- Direction, centre
- Work feed rate 100%
- Depth 12 mm

To create the machining operation from the left and the right, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks. If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **LINE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. Use a double click on the **SIDE** cell to select the **LEFT** machining side in the selection window.

5. Enter 0 mm for the X-position in the **X** cell.

6. Enter 30 mm for the Y-position in the **Y** cell.

7. Enter 5 mm for the Z-position in the **Z** cell.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. The value of 100% is already entered in the **FEED** cell.

10. Enter the value of 5 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

11. Use the ![] button to deactivate the additional depth because the depths correspond directly to the entries.

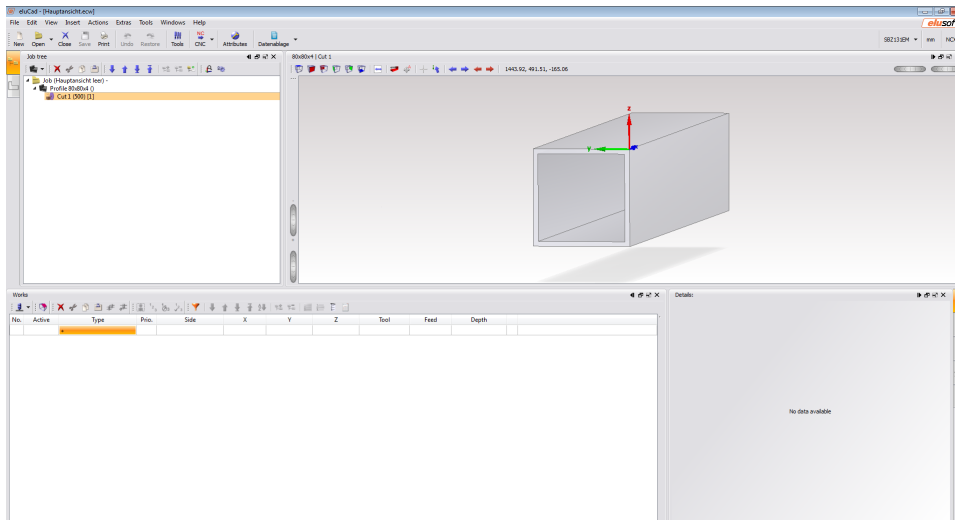12. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

-	**Length:**	Enter 15 mm for the length of the machining task.
-	**Width:**	Enter 8 mm for the width of the machining task.
-	**Angle:**	Enter 270° for the machining angle.
-	**Direction:**	Select **CENTRE FOR THE DIRECTION.**

13. In the **MACHINING TASKS** table, select the next blank line in the list of profile machining tasks .If a line containing a machining operation is selected, it can be overwritten!

14. Open the selection window in the **TYPE** cell with a double click and select the type **LINE**.

15. The machining task is activated automatically but can be deactivated at any time.

16. Use a double click on the **SIDE** cell to select the **RIGHT** machining side in the selection window.

17. For the X-position in the **X** cell, enter either of the variables *PL* or *H* (profile length).

18. Enter 15 mm for the Y-position in the **Y** cell.

19. Enter -2 mm for the Z-position in the **Z** cell.

20. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

21. The value of 100% is already entered in the **FEED** cell.

22. Enter the value of 12 mm in the **DEPTH** cell. Use the ![] button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the ![] button to manually create or change the depth table.

23. Use the ![] button to deactivate the additional depth because the depths correspond directly to the entries.

24. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

-	**Length:**	Enter 30 mm for the length of the machining task.
-	**Width:**	Enter 10 mm for the width of the machining task.
-	**Angle:**	Enter 0° for the machining angle.
-	**Direction:**	Select **CENTRE FOR THE DIRECTION.**

25. Check all inputs in the profile view.
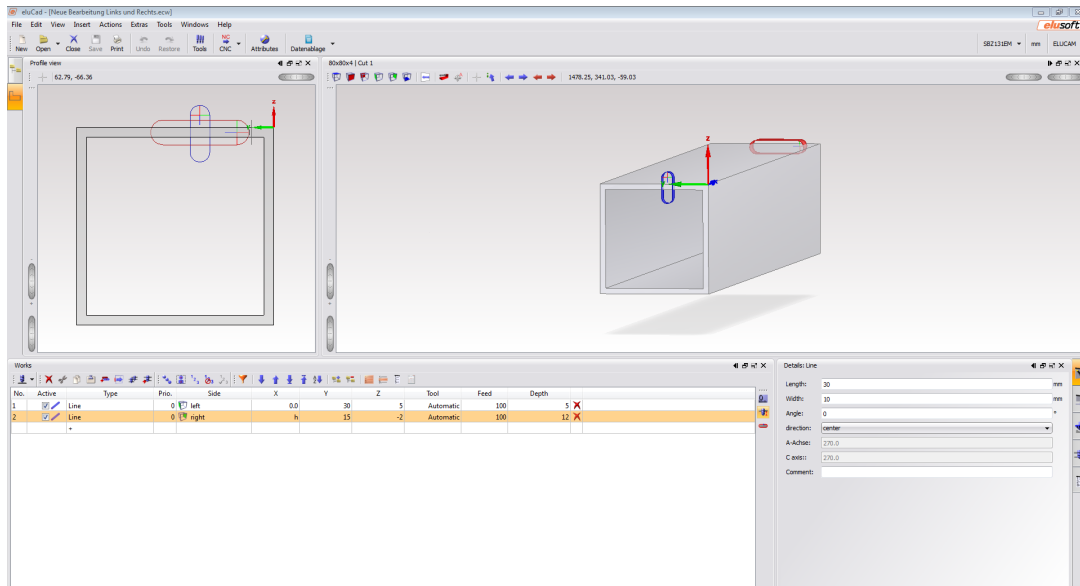
2.2.2.5 Creating a new machining task at BOTTOM

This example shows manual compilation of a machining operation from below.

Default machining task:
-   Bore with 8 mm diameter
-   Position bottom; X = 20 mm; Y = 35 mm; Z = variable *-PH* or *-Z (profile height)*
-   Depth 10 mm
-   Machining can only be performed from the top

To create the machining operation from below, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **DRILLING**.

3. The machining task is activated automatically but can be deactivated at any time.

4. Use a double click on the **SIDE** cell to select the **BOTTOM** machining side in the selection window.

5. Enter 20 mm for the X-position in the **X** cell.

6. Enter 35 mm for the Y-position in the **Y** cell.

7. For the Z-position in the **Z** cell, enter either of the variables *PH* or *-Z* (profile height) .

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. Enter the value of 10 mm in the **DEPTH** cell. Use the [button icon] button to trigger the automatic wall detection, which is transferred to the depth table.

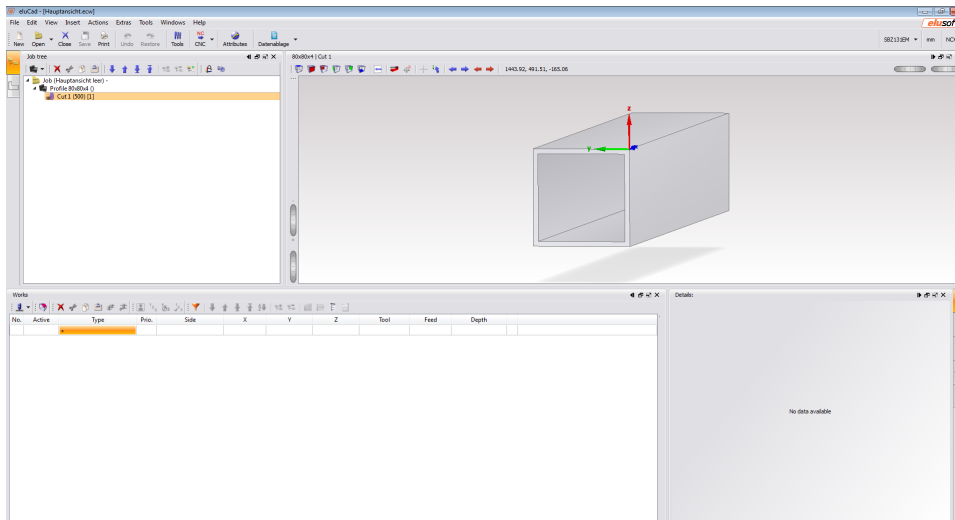   Use the [button icon] button to manually create or change the depth table.

10. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    -   **Diameter:**   Enter 8 mm for the diameter of the machining task.

11. Enter the data and values for the machining task in the input fields of the **PARAMETERS** tab.

    -   **Change of orientation:**   Enter working orientation no. 2 (180° not mirrored) for the orientation of the profile during machining.

   -   **Condition:**   No input is required in the input field.

- **Machine condition:**   No selection is required in the selection field.

- **Relative position:**   No selection is possible in the selection field.

12. Check all inputs in the profile view.
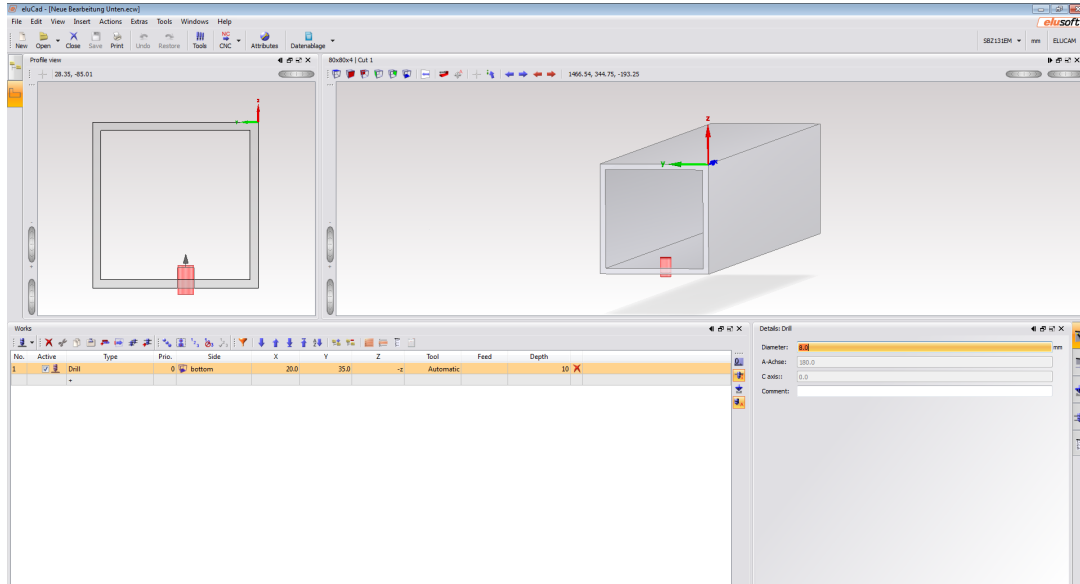
2.2.2.6 New machining task from TOP with Y-value picking

This example shows the manual creation of a machining task from the top whereby the Y-position must yet be determined.
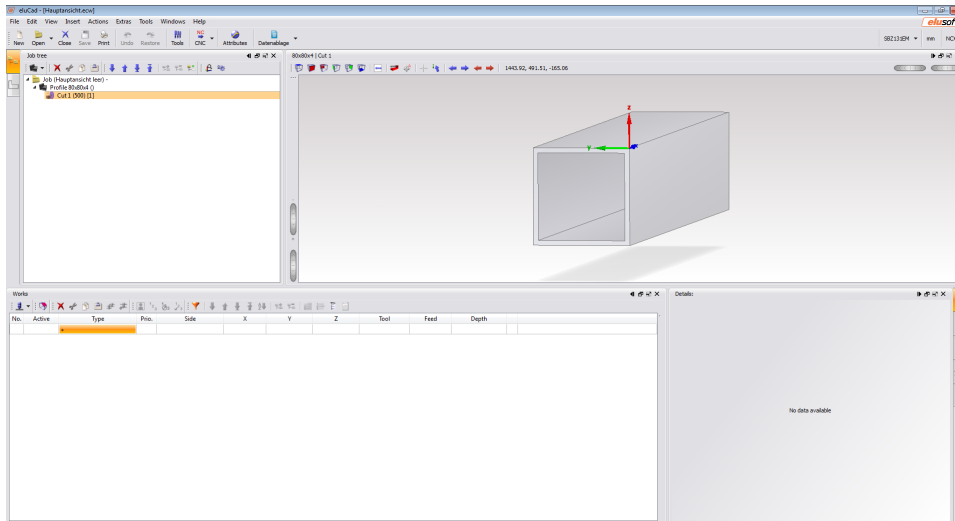
Default machining task:
- Circle pocket with 12 mm diameter
- Position top; X = 50 mm; Y = profile centre
- Work feed rate 100%
- Depth 5 mm

To create the machining operation from the top, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **CIRCLE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **TOP** machining side is selected automatically.

5. Enter 50 mm for the X-position in the **X** cell.

6. Use a double click on the **Y** cell to activate the [button icon] button.

7. Use the [button icon] button to automatically activate the **PICKING** function in the profile cross-section.

8. Use the ▼ button to open the Selection dialog in the **PICKING** function.
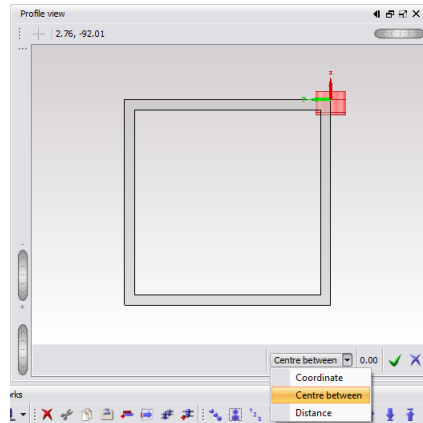
9. Select the **CENTRE POINT** option in the opened selection dialog to determine the coordinates.

10. Select the first coordinate point on the outer edge of the front profile wall using the mouse pointer.

⚠ **INFORMATION**	
ℹ	To improve orientation, the mouse pointer is shown with cross hairs. When the edge of the profile wall has been reached, a red dot appears in the cross hairs.

11. Select the second coordinate point on the outer edge of the rear profile wall using the mouse pointer.

12. The centre dimension determined is displayed in the **VALUE** field.

13. Use the ✔ button to apply the determined centre point dimension in the **Y** cell.
    The **PICKING** function will close automatically.

14. The **Z** cell already contains 0 for the Z-position.

15. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

- **Diameter:**   Enter 12 mm for the diameter of the machining task.

- **Tool diameter:**   Diameter of the tool used.
                               Only required if no automatic assignment is to be made!

16. Check all inputs in the profile view.
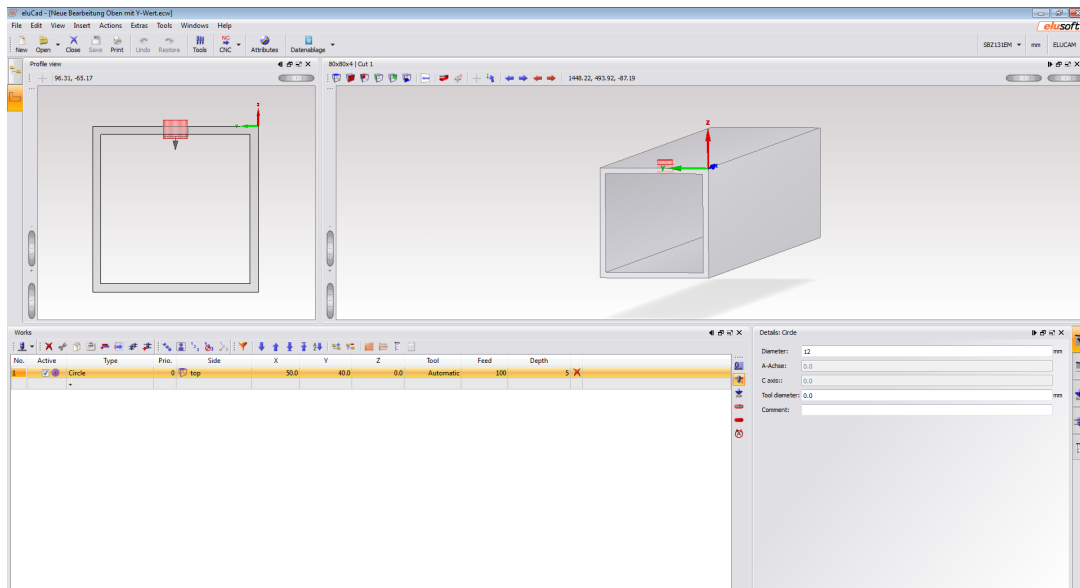
2.2.2.7   New machining task from FRONT with Z-value picking

This example shows the manual creation of a machining task from the front whereby the Z-position must yet be determined.


Default machining task:
- Line of length 30 mm; width 12 mm
- Position front; X = 50 mm; Z = determine dimension; the machining task is to be positioned at the lower edge of the upper profile wall.
- Direction right
- Work feed rate 100%
- Depth 10 mm

To create the machining operation from the top, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **LINE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. Use a double click on the **SIDE** cell to select the **FRONT** machining side in the selection window.

5. Enter 50 mm for the X-position in the **X** cell.

6. The **Y** cell already contains 0 mm for the Y-position.

7. Use a double click on the **Z** cell to activate the [button icon] button.

8. Use the [button icon] button to automatically activate the **PICKING** function in the profile cross-section.

9. Use the ▼ button to open the Selection dialog in the **PICKING** function.
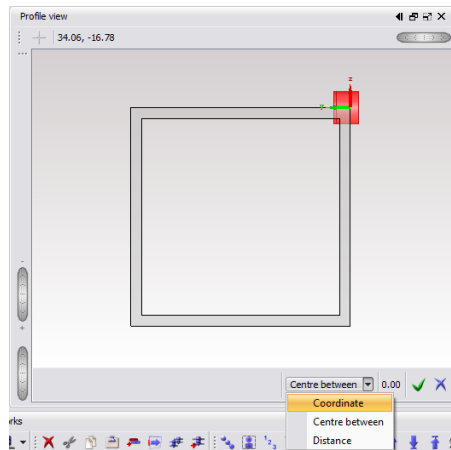
10. Select the **COORDINATES** option in the opened selection dialog to determine the coordinates.

11. Select the coordinate point on the inner edge of the upper profile wall using the mouse pointer.

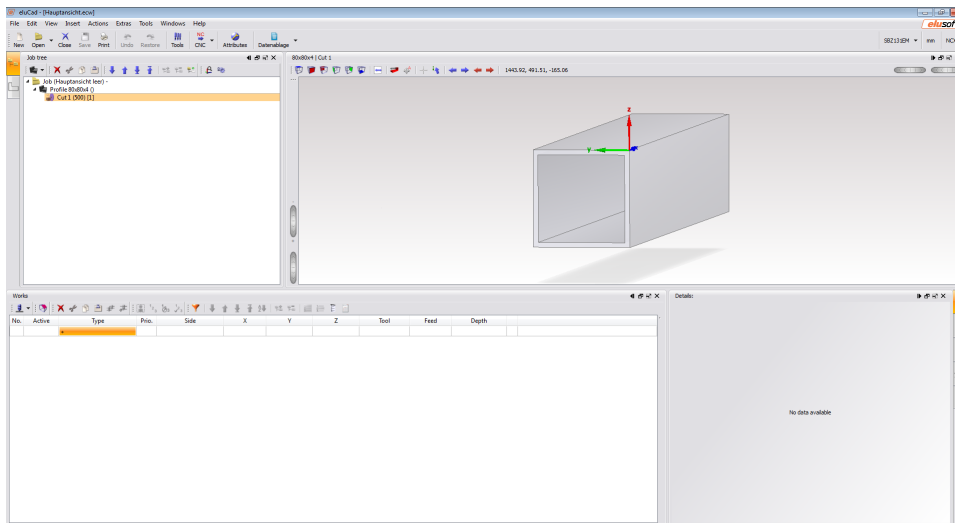| ⚠ **INFORMATION**	
ℹ	To improve orientation, the mouse pointer is shown with cross hairs. When the edge of the profile wall has been reached, a red dot appears in the cross hairs.

12. The coordinate dimension determined is displayed in the **VALUE** field.

13. Use the ✔ button to apply the determined centre point dimension in the **Z** cell. The **PICKING** function will close automatically.

14. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

- **Length:** Enter 30 mm for the length of the machining task.

- **Width:** Enter 12 mm for the width of the machining task.

- **Angle:** The value of 0 is already entered for the angle.

- **Direction:** Select Right for the direction of the machining task.

15. Check all inputs in the profile view.
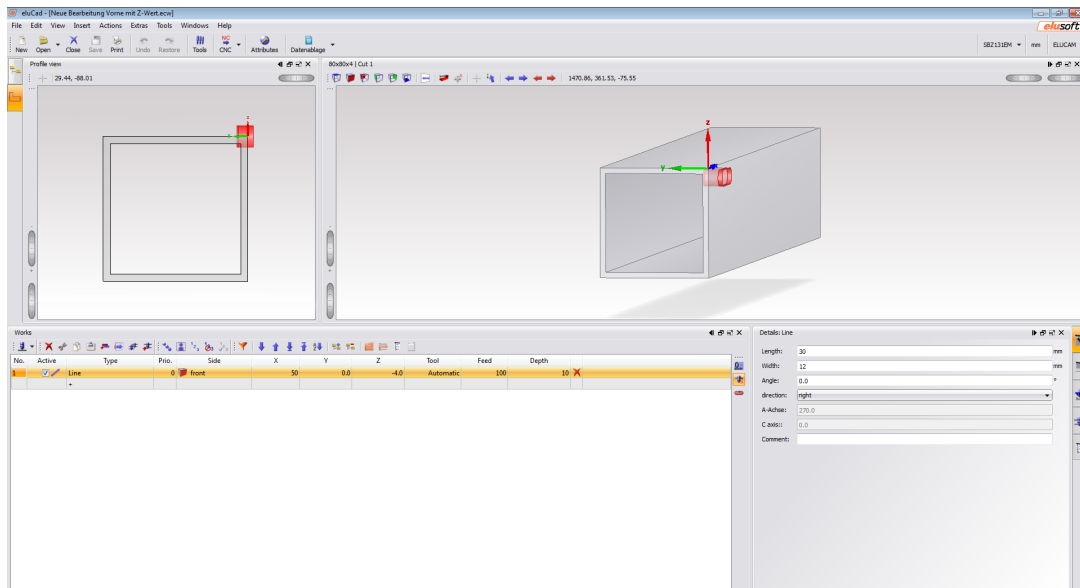
2.2.2.8 Create a new machining task with disk milling cutter

This example shows manual compilation of a machining operation using a side and face milling cutter.

⚠ INFORMATION

i	Upmilling should be employed to ensure proper machining using the disk milling cutter! The machining procedure must be adapted correspondingly.

Default machining task:
- Slot at beginning of profile, from above with distance between 8 and 22 mm
- Disk milling cutter with 92 mm diameter and a 6 mm blade
- Safety distance in front of and behind machining (outside of profile) 10 mm
- Machining angle -90°
- Work feed rate 100%
- Depth 20 mm

To create the machining operation using side and face milling cutter, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks. If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **LINE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **TOP** machining side is selected automatically.

5. For the X-position in the **X** cell, enter either of the variables TR or $D/2$ (tool radius) + 20 mm (depth).

6. For the Y-position in the **Y** cell, enter either of the variables PW or Y (profile width) + 10 mm (safety distance) + variables TR or $D/2$ (tool radius).

7. The **Z** cell already contains 0 mm for the Z-position.

8. The tool is defined automatically by default in the Tool cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. The value of 100% is already entered in the **FEED** cell.

10. The plunge depth must be saved manually in the depth table.

Pressing the ⬚ button allows the depth table to be created manually.

11. Enter the data and values for the machining task in the input fields of the **DEPTH TABLE** tab.

- **1. Row:**   Enter 8 mm for the depth in the **D** cell.

  Deactivate the selection window in the **M** cell.

  Enter 100% for the feed rate in the **F** cell.

  Enter 100% for the spindle speed in the **RPM** cell.

- **2. Row:**   Enter 14 mm for the depth in the **D** cell (depth of the first machining task, 8 mm + 6 mm for the cutter of the disk milling cutter).

  Activate the selection window in the **M** cell.

  Enter 100% for the feed rate in the **F** cell.

  Enter 100% for the spindle speed in the **RPM** cell.

- **3. Row:**   Enter 19 mm for the depth in the **D** cell (depth of the second Machining task, 14 mm + 6 mm for the cutter of the disk milling cutter - 1 mm overlap).

  Activate the selection window in the **M** cell.

  Enter 100% for the feed rate in the **F** cell.

  Enter 100% for the spindle speed in the **RPM** cell.

- **4. Row:**   Enter 22 mm for the depth in the **D** cell (overall depth of the machining task).

  Activate the selection window in the **M** cell.

  Enter 100% for the feed rate in the **F** cell.

  Enter 100% for the spindle speed in the **RPM** cell.

Depth table				
	D	M	F	RPM
1	8.00 mm	☐	100%	100%
2	14.00 mm	☑	100%	100%
3	19.00 mm	☑	100%	100%
4	22.00 mm	☑	100%	100%

☑ Automatic security distance

12. Use the ⬚ button to deactivate the additional depth because the depths correspond directly to the entries.

13. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

-	**Length:**	Enter 10 mm for the length of the machining task (safety distance) + variable *TD* or *D* (tool diameter) + variable *Y* or *PW* (profile width).
-	**Width:**	Enter 92 mm for the width of the machining task (tool diameter).
-	**Angle:**	Enter -90° for the machining angle.

14. Check all inputs in the profile view.
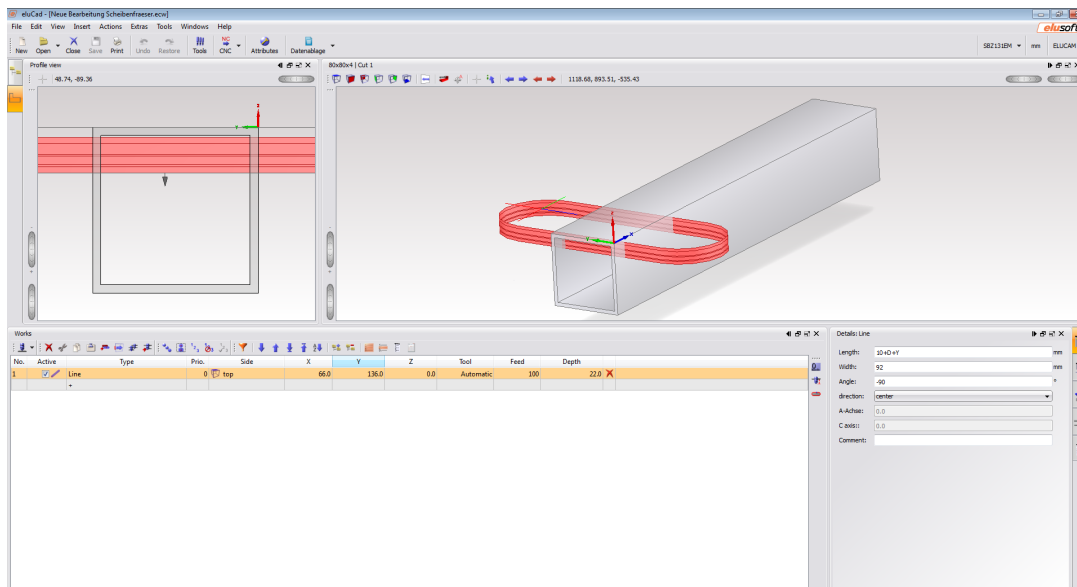
2.2.2.8.1 Creating a new machining task using a disk milling cutter left/rear

This example shows manual compilation of a machining operation using a side and face milling cutter from the left/rear.

⚠ **INFORMATION**

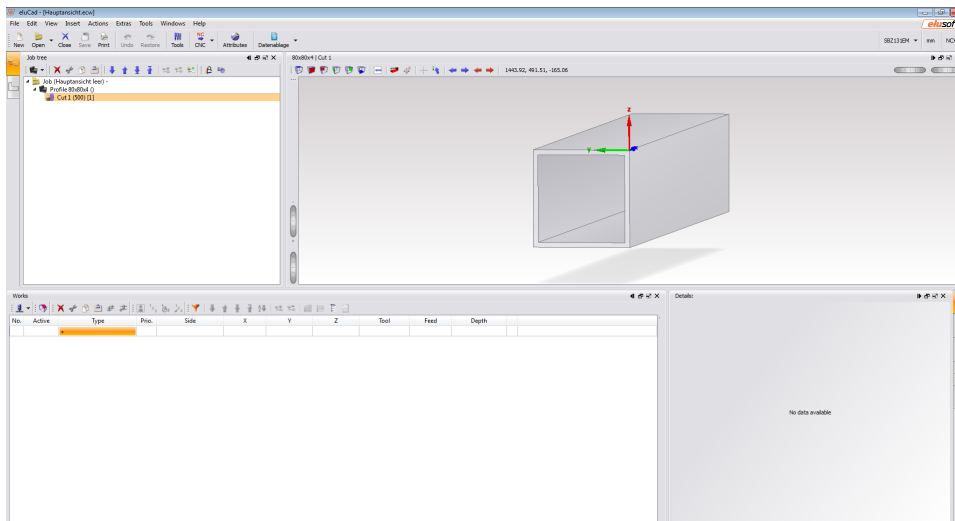| ℹ	Upmilling should be employed to ensure proper machining using the disk milling cutter! The machining procedure must be adapted correspondingly. The profile must be appropriately rotated for machining, because it is only possible to use the disk milling cutter from above. The creation of the machining task is performed in the normal profile view.

Default machining task:
- Slot of 6 mm at the rear start of the profile
- Disk milling cutter with 92 mm diameter and a 6 mm blade
- Safety distance in front of and behind machining (outside of profile) 10 mm
- Angle -90°
- Direction, centre
- Work feed rate 100%
- Depth 20 mm

To create the machining operation using side and face milling cutter, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **LINE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. Use a double click on the **SIDE** cell to select the **REAR** machining side in the selection window.

5. For the X-position in the **X** cell, enter either of the variables *TR* or *-D/2* (tool radius) + 20 mm (depth).

6. Enter the variable *PW* or *Y* (profile width) for the Y-position in the **Y** cell.

7. For the Z-position in the **Z** cell, enter either of the variables *-PH* or *-Z* (profile height) - 10 mm (safety distance) - variable *TR* or *D/2* (tool radius).

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. The value of 100% is already entered in the **FEED** cell.

10. The plunge depth must be saved manually in the depth table.

    Pressing the [button icon] button allows the depth table to be created manually.

11. Enter the data and values for the machining task in the input fields of the **DEPTH TABLE** tab.

   - **1. Row:**   Enter 6 mm for the depth in the **D** cell.

                                 Activate the selection window in the **M** cell.

                                 Enter 100% for the feed rate in the **F** cell.

                                 Enter 100% for the spindle speed in the **RPM** cell.

12. Use the [button icon] button to deactivate the additional depth because the depths correspond directly to the entries.

13. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.
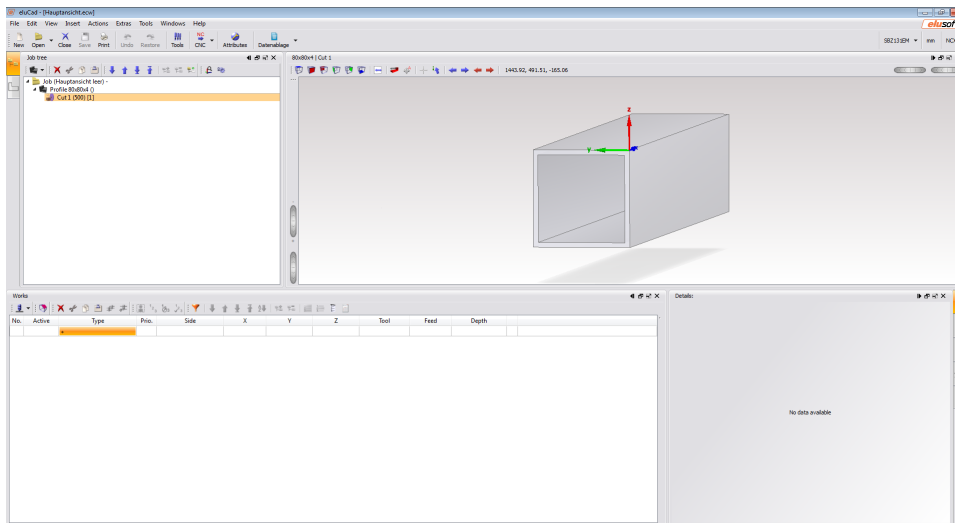
    - **Length:**   Enter 10 mm for the length of the machining task (safety distance) + variable *TD* or *D* (tool diameter) + variable *PH* or *Z* (profile height) + 10 mm (safety distance).

    - **Width:**   Enter 92 mm for the width of the machining task (tool diameter).

    - **Angle:**   Enter -90° for the machining angle.

    - **Direction:**   Select **CENTRE FOR THE DIRECTION.**

14. Check all inputs in the profile view.
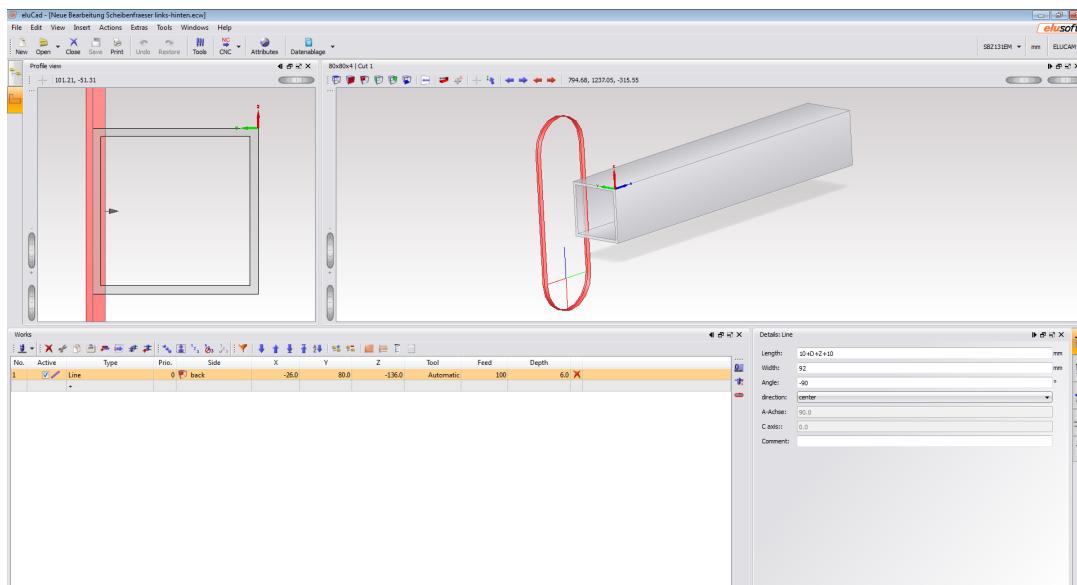
2.2.2.8.2 Create a new machining operation with disk milling cutter right/front

This example shows manual compilation of a machining operation using a side and face milling cutter from the right/front.

⚠ INFORMATION

	Upmilling should be employed to ensure proper machining using the disk milling cutter! The machining procedure must be adapted correspondingly. The profile must be appropriately rotated for machining, because it is only possible to use the disk milling cutter from above. The creation of the machining task is performed in the normal profile view.

Default machining task:
- Slot of 6 mm at the front end of the profile
- Disk milling cutter with 92 mm diameter and a 6 mm blade
- Safety distance in front of and behind machining (outside of profile) 10 mm
- Angle -90°
- Direction, centre
- Work feed rate 100%
- Depth 20 mm

To create the machining operation using side and face milling cutter, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **LINE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. Use a double click on the **SIDE** cell to select the **FRONT** machining side in the selection window.

5. For the X-position in the **X** cell, enter either of the variables *PL* or *H* (profile length) + variable *TR* or *D/2* (tool diameter) - 20 mm (depth).

6. The **Y** cell already contains 0 mm for the Y-position.

7. For the Z-position in the **Z** cell, enter either of the variables *-PH* or *-Z* (profile height) - 10 mm

(safety distance) - variable *TR* or *D/2* (tool radius).

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. The value of 100% is already entered in the **FEED** cell.

10. Enter the data and values for the machining task in the input fields of the **DEPTH TABLE** tab.

- **1. Row:**   Enter 20 mm for the depth in the **D** cell.

Activate the selection window in the **M** cell.

Enter 100% for the feed rate in the **F** cell.

Enter 100% for the spindle speed in the **RPM** cell.

11. Use the button to deactivate the additional depth because the depths correspond directly to the entries.

12. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

- **Length:**   Enter 10 mm for the length of the machining task (safety distance) + variable *TD* or *D* (tool diameter) + variable *PH* or *Z* (profile height) + 10 mm (safety distance).

- **Width:**   Enter 92 mm for the width of the machining task (tool diameter).

- **Angle:**   Enter +90° for the machining angle.

- **Direction:**   Select **CENTRE** for the direction.

13. Enter the data and values for the machining task in the input fields of the **PARAMETERS** tab.
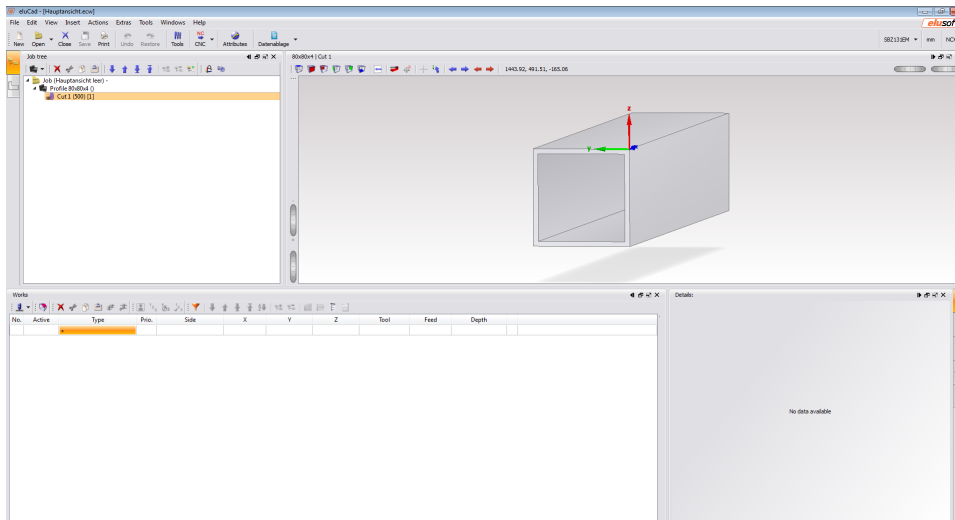
- **Change of orientation:**   Enter working orientation no. 1 (90° not mirrored) for the orientation of the profile during machining.

Info: Machining can only be performed from the top!

- **Condition:**   No input is required.

- **Machine condition:**   No input is required.

- **Relative position:**   No selection can be made.

- **Radius compensation:** No input is required.

14. Check all inputs in the profile view.
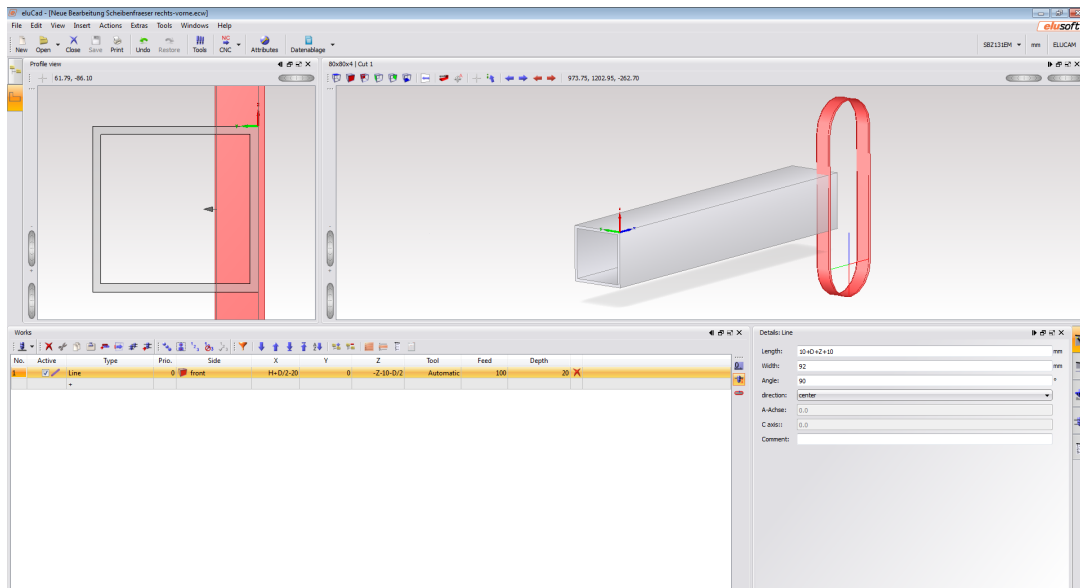
2.2.2.9 Creating a new machining task on a free side

This example shows the manual creation of a machining task on a freely defined side.
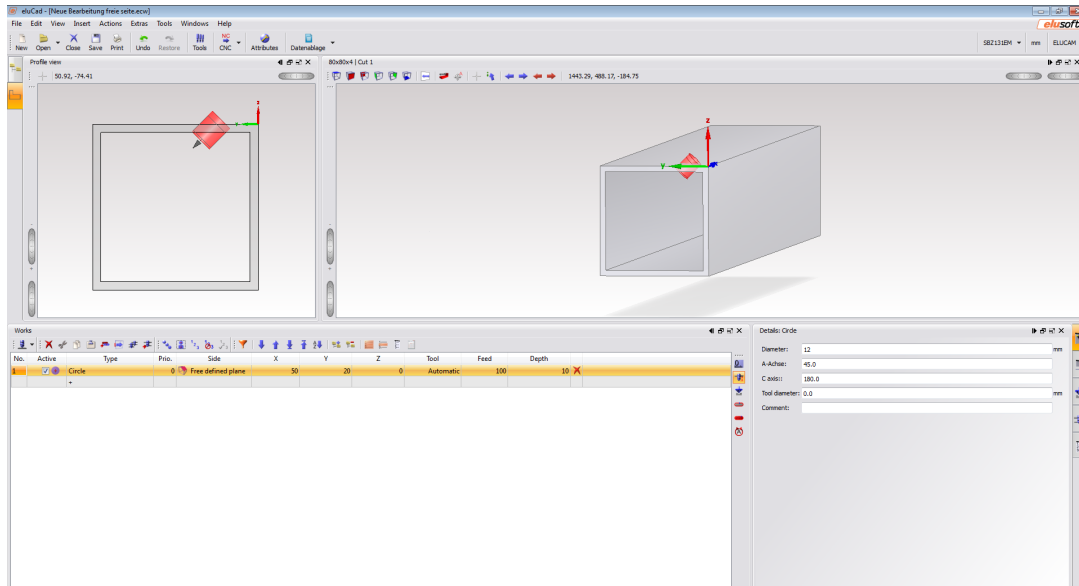

Default machining task:
- Circle pocket with 12 mm diameter
- Position top; X = 50 mm; Y = 20 mm
- Work feed rate 100%
- Depth 10 mm
- A-axis 45°
- C-axis 180°
- Pre pos. dist.   10 mm

In order to create a machining task on a free side, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **CIRCLE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. Double click on the **SIDE** cell to select the **FREE SIDE** machining side in the selection window.

5. Enter 50 mm for the X-position in the **X** cell.

6. Enter 20 mm for the Y-position in the **Y** cell.

7. The **Z** cell already contains 0 for the Z-position.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. The value of 100% is already entered in the **FEED** cell.

10. Enter the value of 10 mm in the **DEPTH** cell. Use the ![button] button to trigger the automatic wall detection, which is transferred to the depth table.

   Use the ![button] button to manually create or change the depth table.

11. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

   - **Diameter:**   Enter 12 mm for the diameter of the machining task.

   - **A-axis:**   Enter 45° for the angle setting of the A-axis.

   - **C-axis:**   Enter 180° for the machining angle of the C-axis.
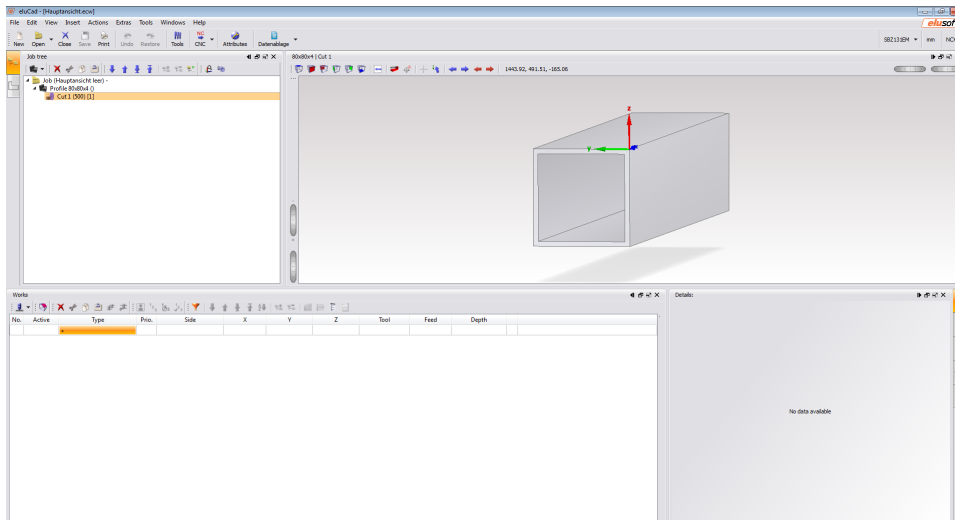
- **Tool diameter:** Diameter of the tool used.
    Only required if no automatic assignment is to be made!

12. Check all inputs in the profile view.

2.2.2.10   Creating a new machining task for a free side using picking

This example shows the manual creation of a machining task on a freely defined side.

Default machining task:
- Line of length 30 mm; width 15 mm
- Position front; X = 100 mm
- Determine Y and Z-dimensions
- Determine A and C-angles
- Desired position, "lower corner" on the profile
- Work feed rate 100%
- Depth 2 mm

In order to create a machining operation for a free side using picking, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks. If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **LINE**.

3. The machining task is activated automatically but can be deactivated at any time.

4. Double click on the **SIDE** cell to select the **FREE SIDE** machining side in the selection window.

5. The **FREE SIDE** selection window appears in the 3D view.

6. Pressing the [button] button displays the input menu of the new side.

7. Pressing the [button] button activates picking.

8. Move the mouse over the plane which is to be machined.

9. Click on the centre point of the plane with the left mouse button.

10. Enter the data and values for the machining task in the input fields of the **FREE SIDE** menu.

- **Free side:**   Standard is already entered.

- **Name:**   Enter line, 45° front.

- **Angle:**   Enter 0° for the machining angle.

- **A-axis:**   The value of -45° is already entered for the A-axis angle setting.

- **C-axis:**   The value of 0° is already entered for the C-axis angle setting.

- **X offset:**   Enter 100 mm for the X-position.

- **Y offset:**   The value of 0.25 mm is already entered for the Y-position.

- **Z offset:**   The value of -60.25 mm is already entered for the Z-position.

11. Use the ![button] button to add the free side just created to the table.

12. Press the ![button] button to add the free side just created to the **SIDE** cell.

13. Enter 100 mm for the X-position in the **X** cell.

14. The **Y** cell already contains 0 mm for the Y-position.

15. The **Z** cell already contains 0 mm for the Z-position.

16. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

17. The value of 100% is already entered in the **FEED** cell.

18. Enter the value of 2 mm in the **DEPTH** cell. Use the ![button] button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the ![button] button to manually create or change the depth table.

19. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.
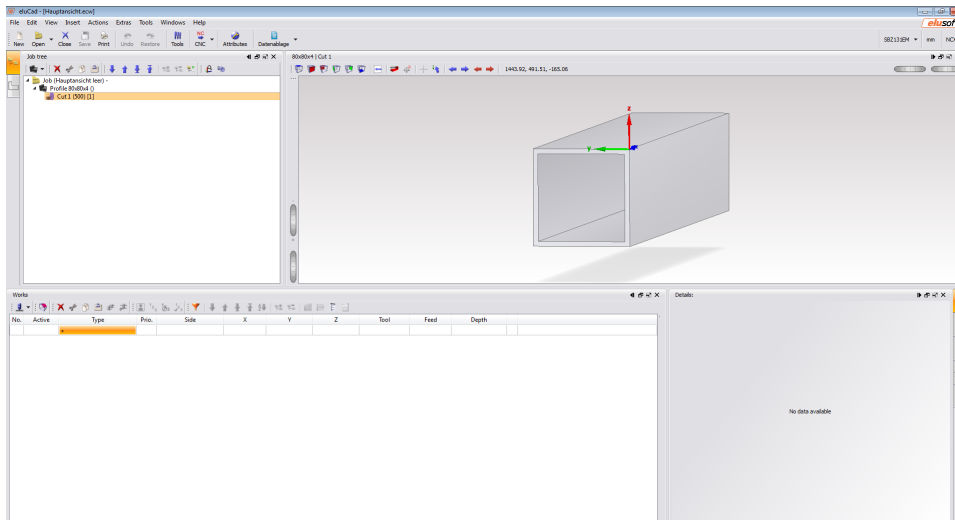
    - **Length:**   Enter 30 mm for the length of the machining task.

    - **Width:**   Enter 15 mm for the width of the machining task.

    - **Angle:**   The value of 0° for the machining angle is already entered.

    - **Direction:**   The direction of the machining task is already entered as Centre.

20. Check all inputs in the profile view.
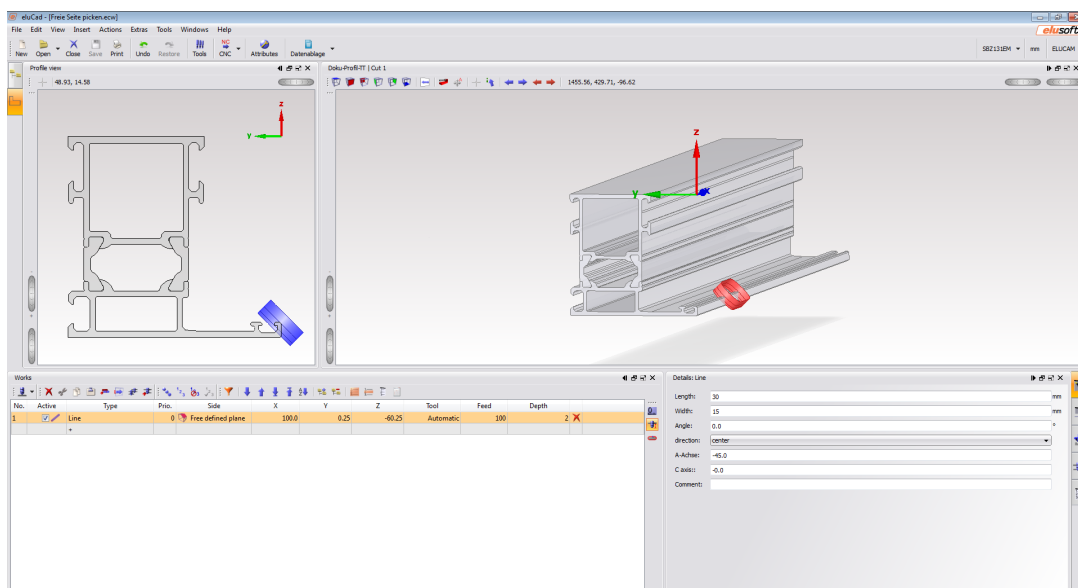
2.2.2.11 Creating a new machining task as a machining series

This example shows manual compilation of a machining operation as a machining chain.

Default machining task:
- Bore with 15 mm diameter
- Position top; X = 50 mm; Y = 20 mm
- The machining task is to be repeated 5 times horizontally at a spacing of 30 mm.
- Work feed rate 100%
- Depth 10 mm

To create the machining tasks as a series of drilled holes, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **DRILLING**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **TOP** machining side is selected automatically.

5. Enter 50 mm for the X-position in the **X** cell.

6. Enter 20 mm for the Y-position in the **Y** cell.

7. The **Z** cell already contains 0 for the Z-position.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

   Use the button to manually create or change the depth table.

10. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    - **Diameter:**   Enter 15 mm for the diameter of the machining task.

11. Use the ![icon] button to change the selected machining task into a machining series.

No.	Active	Type	Prio.	Side	X	Y	Z		
1	☑	Work chain						✗	
		Drill	0	top	50.000	20.000	0.000	✗	
		+							

12. In the machining task list, a machining series row is automatically inserted before the selected row and is opened.

13. Enter the data and values for the machining series in the input fields of the **DETAIL** tab.

- **Series type:** Select the **HORIZONTAL** series type.

- **Division:** Select **FIXED** for the division.

- **Quantity X:** Enter 5 for the quantity of machining tasks.

- **Distance X:** Enter 30 mm for the spacing of the machining task.
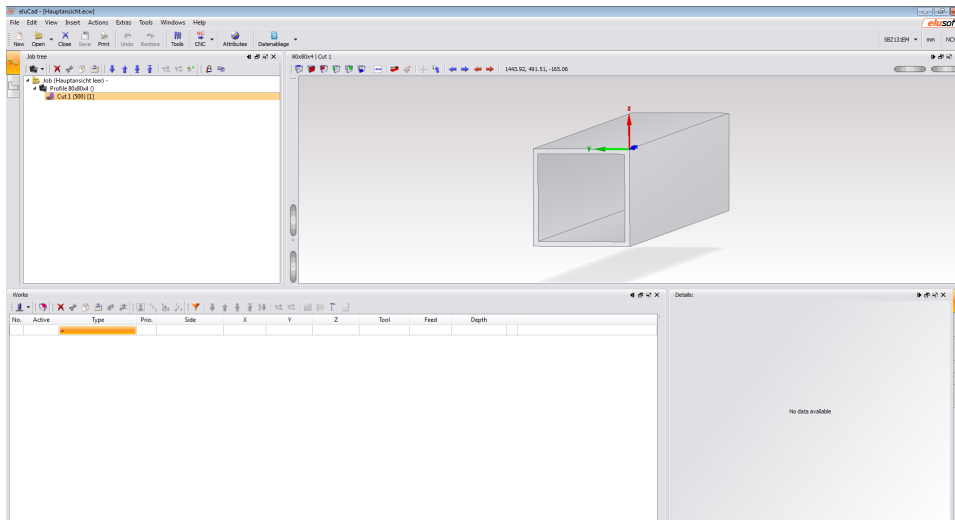
⚠ **INFORMATION**	
ℹ	In the profile view, the individual machining tasks in the series are displayed in a different colour to make their association easier to recognise.

14. Check all inputs in the profile view.
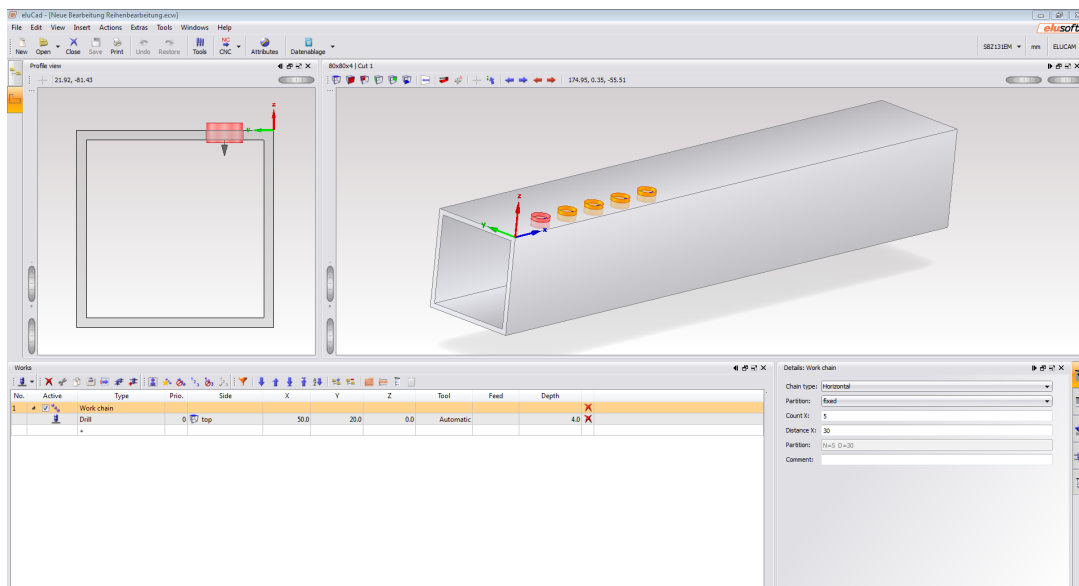
2.2.2.12   Creating a new machining task with combo-thread

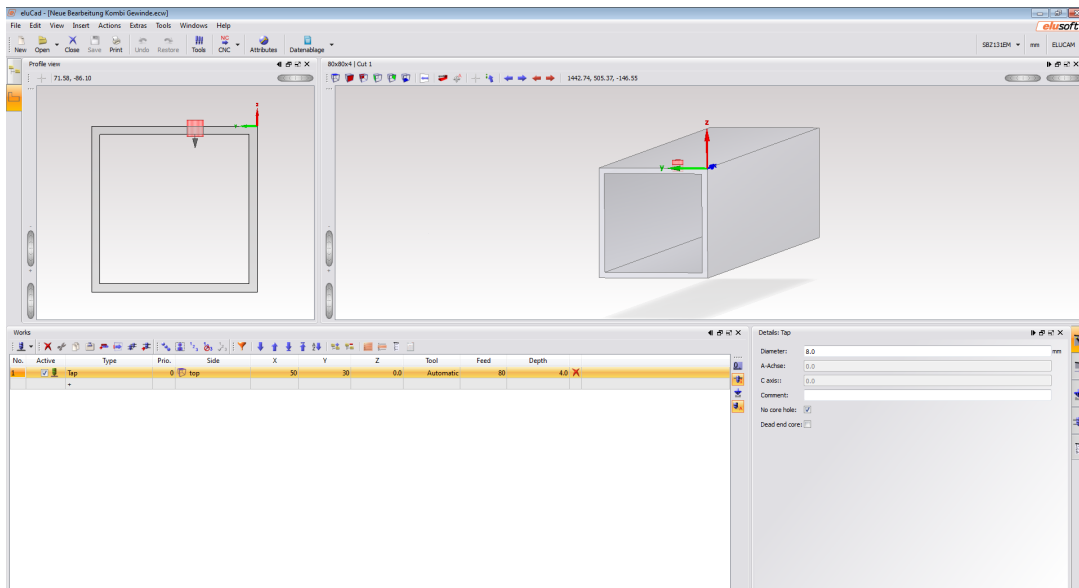This example shows the manual creation of a tapping machining task.

Default machining task:
-   M8 tapping machining task
-   Position top; X = 50 mm; Y = 30 mm
-   Work feed rate 80%
-   Depth corresponds to tapping tool selected

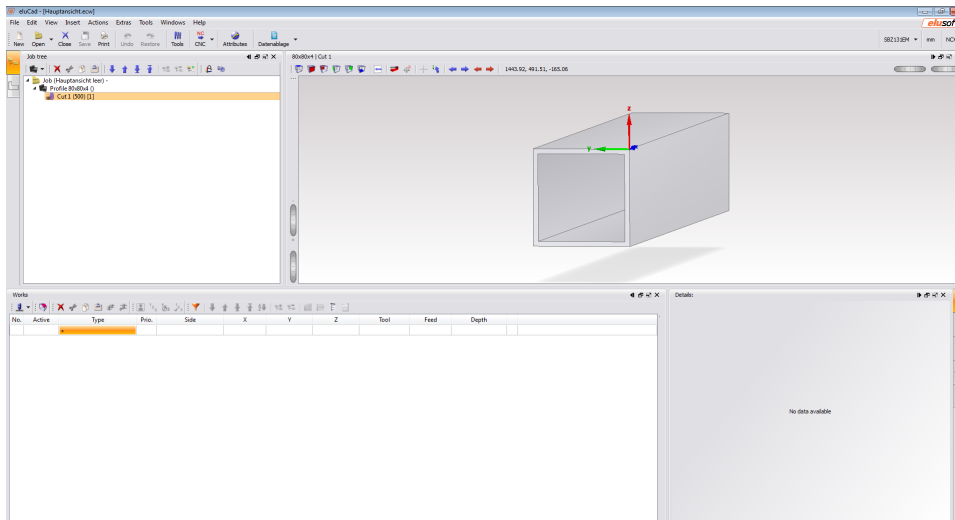To create the tapping machining task, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **TAP**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **TOP** machining side is selected automatically.

5. Enter 50 mm for the X-position in the **X** cell.

6. Enter 30 mm for the Y-position in the **Y** cell.

7. The **Z** cell already contains 0 for the Z-position.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. Enter the value of 80% in the Feed rate cell.

10. Enter the value of 10 mm in the **DEPTH** cell. Use the [button icon] button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the [button icon] button to manually create or change the depth table.

11. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    -   **Diameter:**   Enter 8 mm for the diameter of the machining task.

    -   **Without core hole:**   Activate field

12. Check all inputs in the profile view.

2.2.2.13 Creating a new machining task with a countersink

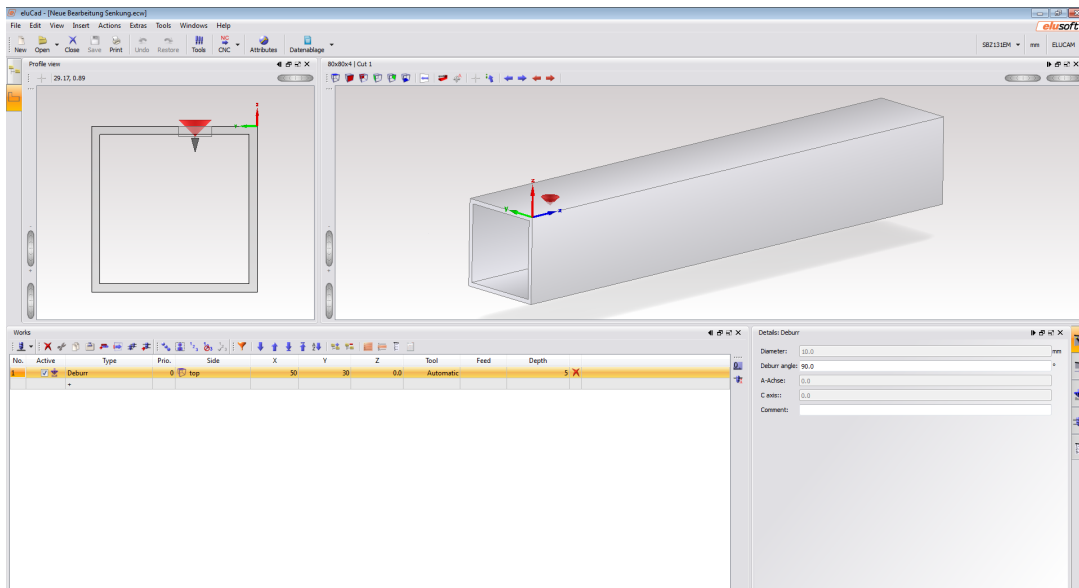This example shows the manual creation of a countersink.

Default machining task:
- Countersink with a diameter of 10 mm at the outside edge of the part
- Deburr angle 90°
- Position top; X = 50 mm; Y = 30 mm
- Work feed rate 100%
- Determine depth

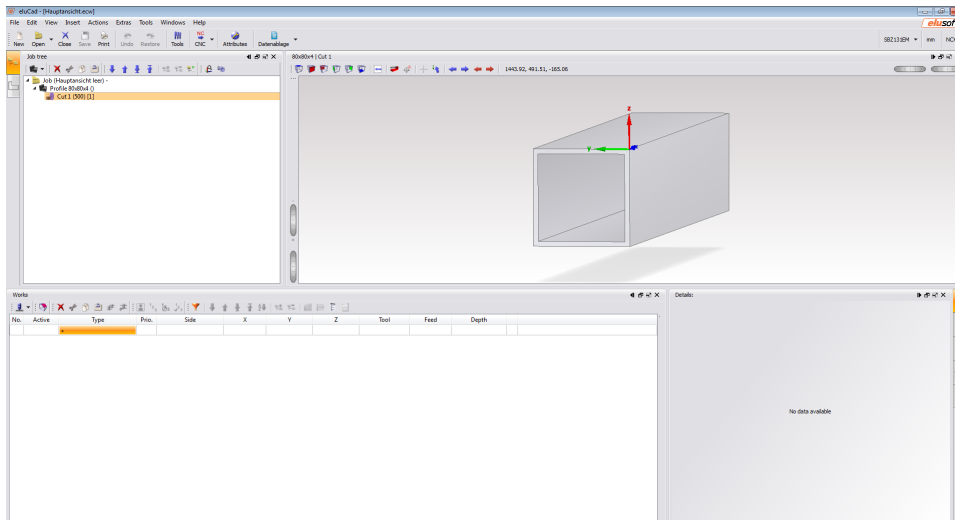To create the countersink, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **COUNTER-SINK**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **TOP** machining side is selected automatically.

5. Enter 50 mm for the X-position in the **X** cell.

6. Enter 30 mm for the Y-position in the **Y** cell.

7. The **Z** cell already contains 0 for the Z-position.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. Enter the value of 5 mm in the **DEPTH** cell. Use the [button] button to manually create or change the depth table.
   Info: For some counter sinks, no tool tip is present. The software nevertheless always references the point of the tip and calculates the corresponding values based on this!

10. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

- **Diameter:**   The diameter is displayed automatically.
  The value is calculated from the deburr angle and the depth.

- **Deburr angle:**   Enter 90° for the deburr angle.

11. Check all inputs in the profile view.

2.2.2.14 New machining task, drilled hole with countersink

This example shows the counter sinking of a drilled hole.

Default machining task:
- Countersink with a diameter of 16 mm for a drilled hole with a diameter of 10
- Deburr angle 90°
- Position top; X = 50 mm; Y = 30 mm
- Work feed rate 100%
- Determine depth
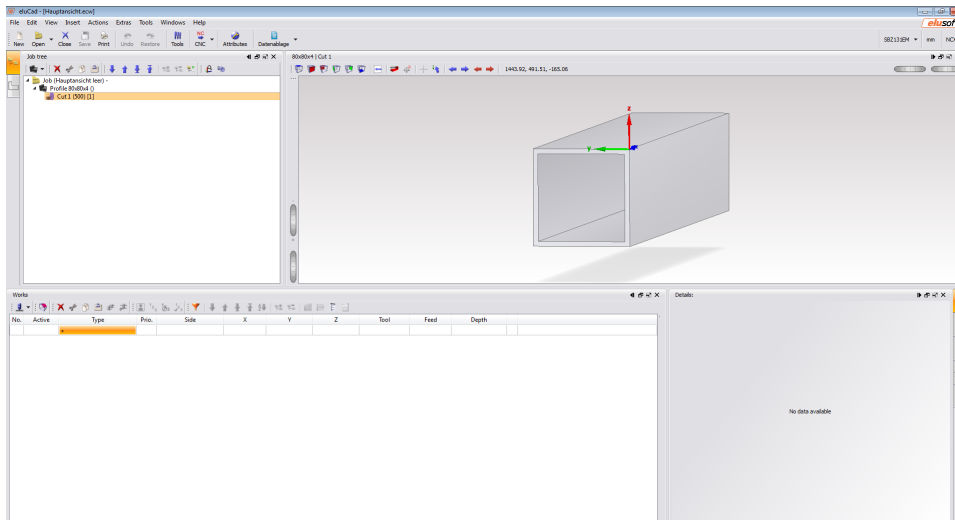
To create the countersink, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **DRILLING**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **TOP** machining side is selected automatically.

5. Enter 50 mm for the X-position in the **X** cell.

6. Enter 30 mm for the Y-position in the **Y** cell.

7. The **Z** cell already contains 0 for the Z-position.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. Enter the value of 10 mm in the **DEPTH** cell. Use the [button] button to manually create or change the depth table.

10. Pressing the [button] button opens the **COUNTERSINK PARAMETERS** tab.

11. Enter 3 mm in the **COUNTERSINK DEPTH** cell.
    Info: Calculation (D1 screw head - D2 drilled hole) / 2
    For some counter sinks, no tool tip is present. The software nevertheless always references the point of the tip and calculates the corresponding values based on this!

12. Check all inputs in the profile view.
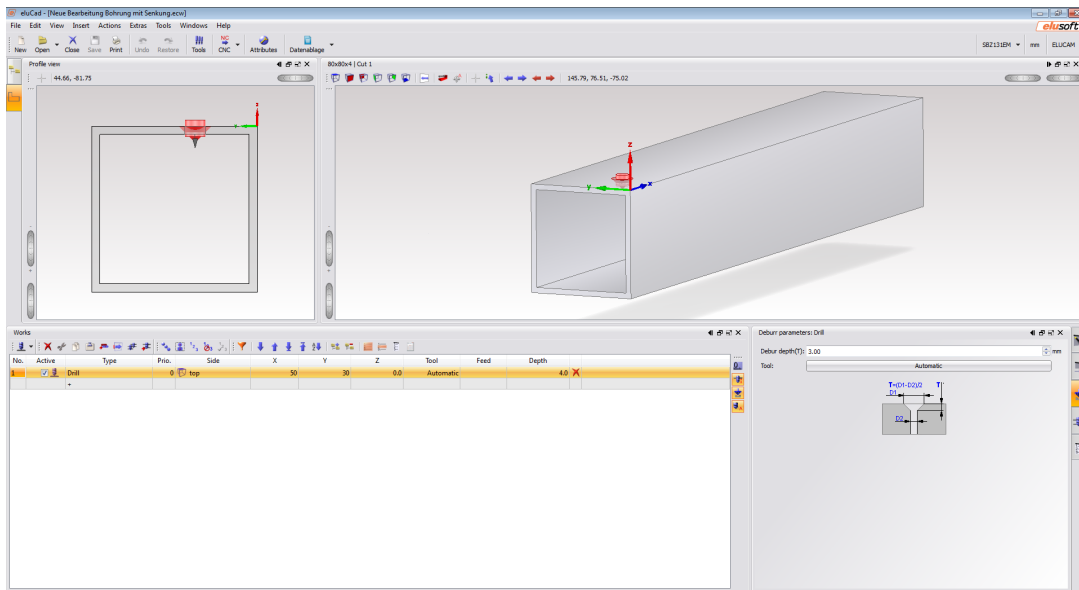
2.2.2.15 Creating a new machining task with a saw cut at the beginning of the part

This example shows the manual creation of a saw-cut machining task.


Default machining task:
- Position in X = 10 mm
- The cutoff is performed from back to front
- Saw blade tilting angle A equals 90°
- Saw blade tilting angle C equals -90°
- Additional depth of 1 mm

To create the saw cut machining task, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **SAW ASSISTANT**.

3. Selecting the **PROFILE VIEW** field automatically opens the saw assistant.
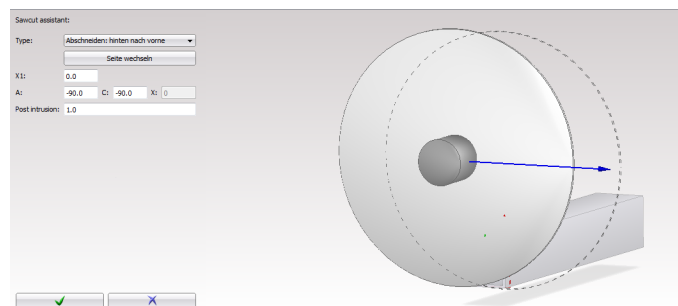
4. In the **TYPE** selection field, select the *Cut off: from back to front* option.

⚠ INFORMATION

| ℹ | At the beginning of the part, the saw cut should always be performed from back to front. |

---

5. Use the **CHANGE SIDE** button to define the side of the spindle to be used.
   This selection is made to avoid collisions with the spindle and to achieve optimal machining.

6. Enter 10 mm in the **X1** field for the start position in the X-direction.

7. In the **A** field, enter 90° for the tilting angle of the saw blade.

8. In the **C** field, enter 90° for the pivoting angle of the saw blade.

9. Enter the value of 1 mm in the **ADDITIONAL DEPTH** field.

10. Check all saw cut assistant inputs in the profile view.

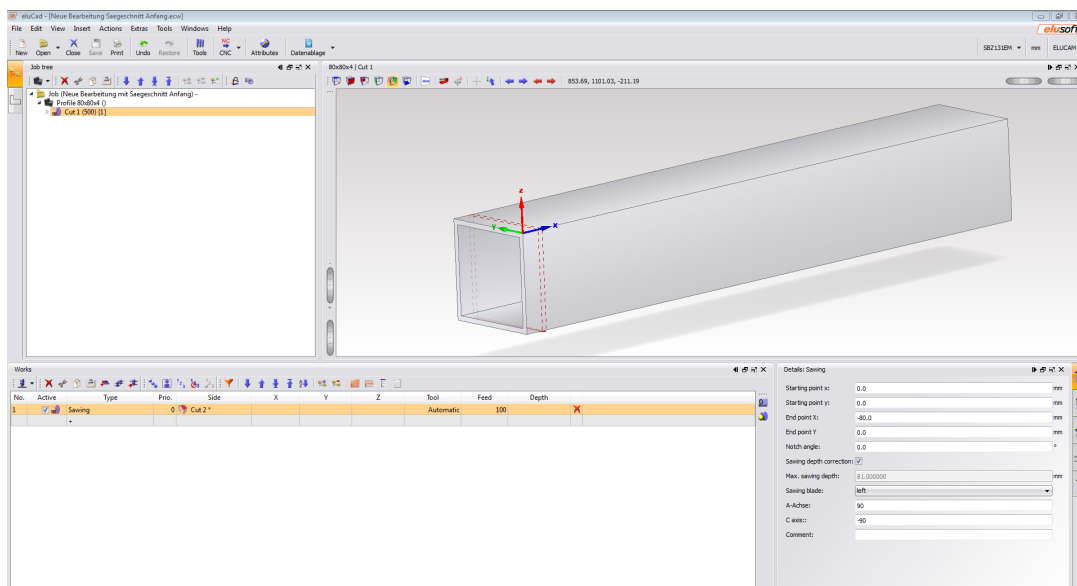11. Press the ✔ button to apply the values to the **SAW CUT** machining task.

⚠ **INFORMATION**	
i	Press the 👥 button to call up the saw assistant once again to make any changes. Press the ✔ button to apply the changes to the **SAW CUT** machining task.

12. The machining task is activated automatically but can be deactivated at any time.

13. No entries are required on the **DETAIL** tab.

14. No entries are required on the **PARAMETERS** tab.

15. Check all inputs in the profile view.

2.2.2.16 Creating a new machining task with a saw cut at the end of the part

This example shows the manual creation of a saw-cut machining task.

Default machining task:
- Position in X = PL-10 mm
- The cutoff is performed from front to back
- Saw blade tilting angle A equals -90°
- Saw blade tilting angle C equals -135°
- Additional depth of 1 mm

To create the saw cut machining task, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **SAW ASSISTANT**.

3. Selecting the **PROFILE VIEW** field automatically opens the saw assistant.
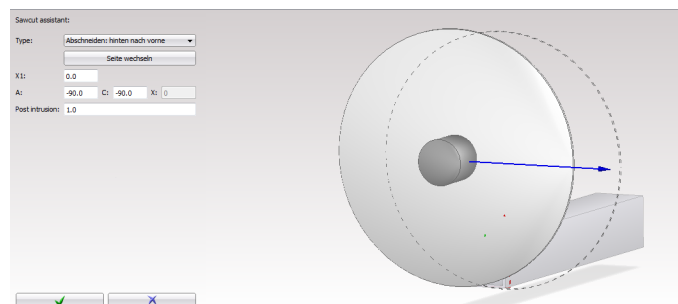
4. In the **TYPE** selection field, select the *Cut off: from front to back* option.

⚠ **INFORMATION**

ℹ At the part end, the saw cut should always be performed from front to back.

5. Use the **CHANGE SIDE** button to define the side of the spindle to be used.
This selection is made to avoid collisions with the spindle and to achieve optimal machining.

6. In the **X1** field, enter either of the variables *PL* or *H* (profile length) - 10 mm for the start position in the X-direction.

7. In the **A** field, enter -90° for the tilting angle of the saw blade.

8. In the **C** field, enter -135° for the pivoting angle of the saw blade.

9. Enter the value of 1 mm in the **ADDITIONAL DEPTH** field.

10. Check all saw cut assistant inputs in the profile view.

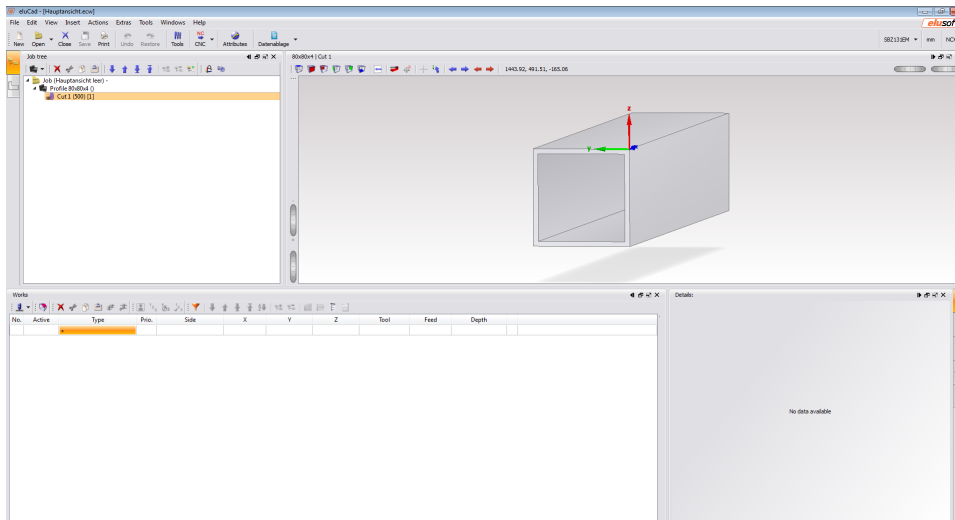11. Press the ✔ button to apply the values to the **SAW CUT** machining task.

⚠ **INFORMATION**	
ℹ	Press the 🔧 button to call up the saw assistant once again to make any changes. Press the ✔ button to apply the changes to the **SAW CUT** machining task.

12. The machining task is activated automatically but can be deactivated at any time.

13. No entries are required on the **DETAIL** tab.

14. No entries are required on the **PARAMETERS** tab.

15. Check all inputs in the profile view.
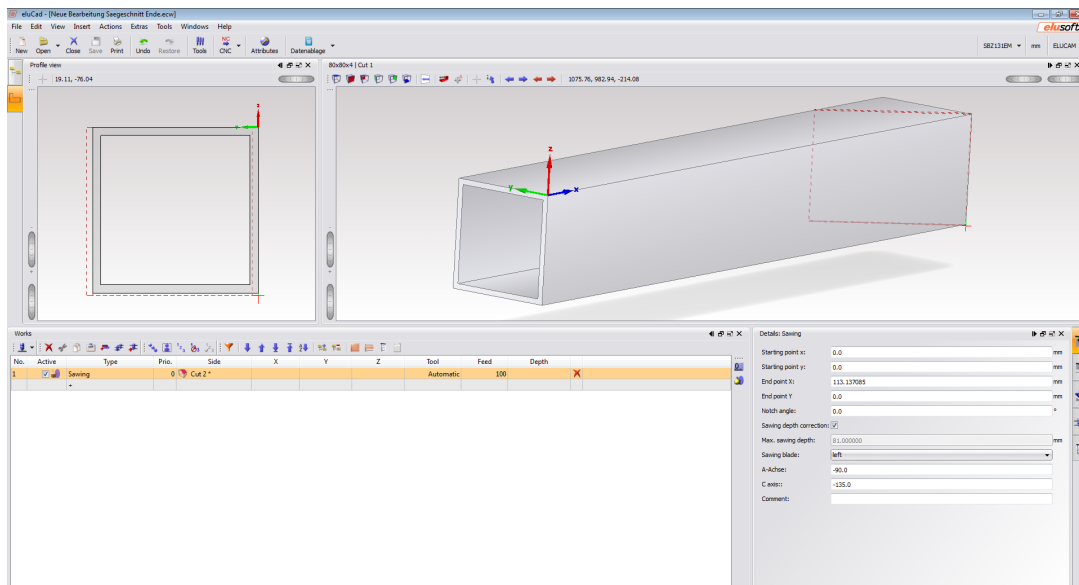
2.2.2.17 Creating a new machining task with a saw cut

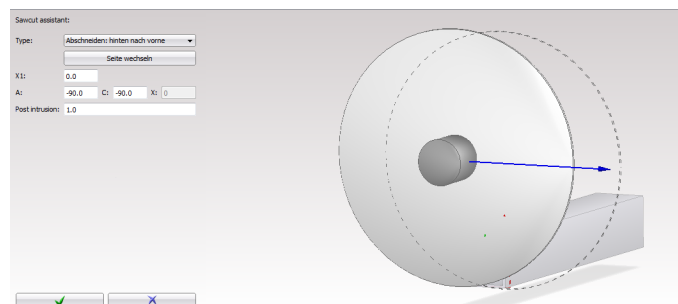This example shows the manual creation of a saw cut.

Default machining task:
- Position in X = 50 mm
- The cut is performed from back to front with a depth of 10 mm
- Saw blade tilting angle A equals -90°
- Saw blade tilting angle C equals -90°

To create the saw cut machining task, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **SAW ASSISTANT**.

3. Selecting the **PROFILE VIEW** field automatically opens the saw assistant.

4. In the **TYPE** selection field, select the *Cut in: from back to front* option.

5. Use the **CHANGE SIDE** button to define the side of the spindle to be used.
   This selection is made to avoid collisions with the spindle and to achieve optimal machining.

6. Enter 50 mm in the **X1** field for the start position in the X-direction.

7. Enter -10 mm in the **Z1** field for the position in the Z-direction.

8. In the **A** field, enter -90° for the tilting angle of the saw blade.

9. In the **C** field, enter -90° for the pivoting angle of the saw blade.

10. Check all saw cut assistant inputs in the profile view.

11. Press the ✔ button to apply the values to the **SAW CUT** machining task.

> ⚠️ **INFORMATION**
>
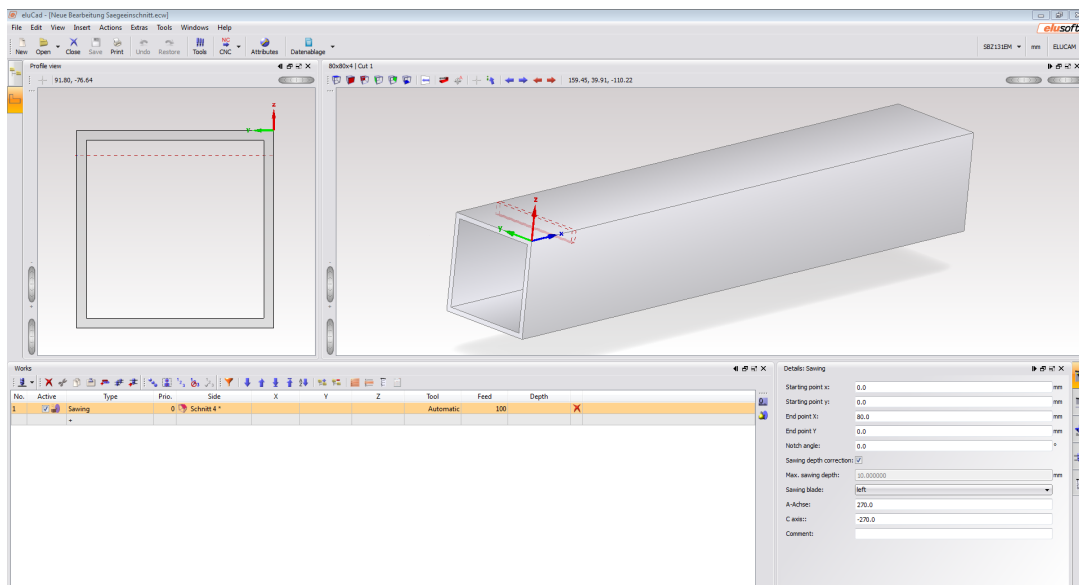> ℹ️ Press the 🐞 button to call up the saw assistant once again to make any changes.
>
> Press the ✔ button to apply the changes to the **SAW CUT** machining task.

12. The machining task is activated automatically but can be deactivated at any time.
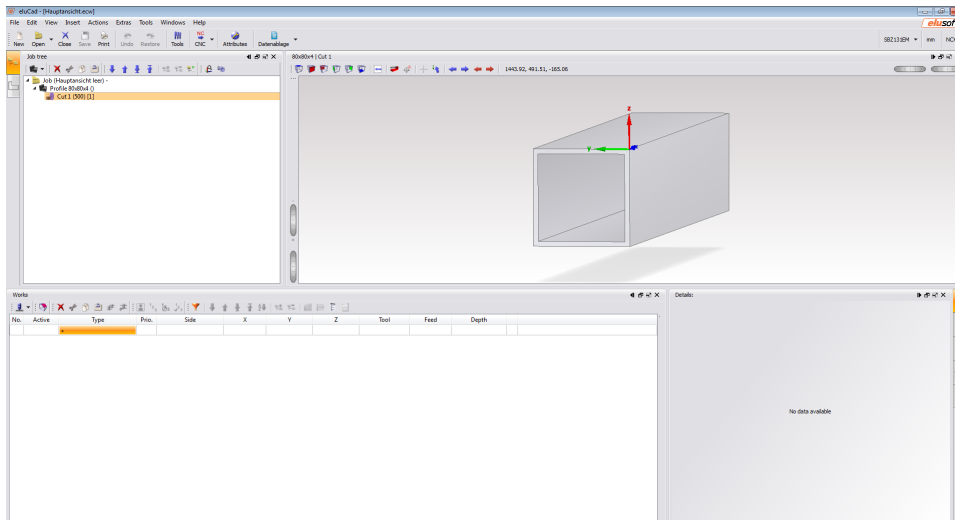
13. No entries are required on the **DETAIL** tab.

14. No entries are required on the **PARAMETERS** tab.

15. Check all inputs in the profile view.

2.2.2.18   Creating a new machining task with a saw cut and an additional notch

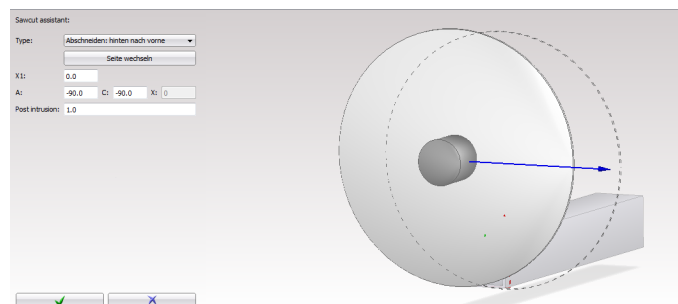This example shows the manual creation of a saw cut with an additional notch.

Default machining task:
  - Position in X = 50 mm
  - The cut is performed from back to front with a depth of 20 mm
  - Saw blade tilting angle A equals -90°
  - Saw blade tilting angle C equals -90°
  - Additional notch with an angle of 45°

To create the saw cut machining task, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **SAW ASSI-STANT**.

3. Selecting the **PROFILE VIEW** field automatically opens the saw assistant.

4. In the **TYPE** selection field, select the *Cut in: from back to front* option.

5. Use the **CHANGE SIDE** button to define the side of the spindle to be used.
   This selection is made to avoid collisions with the spindle and to achieve optimal machining.

6. Enter 50 mm in the **X1** field for the start position in the X-direction.

7. Enter -20 mm in the **Z1** field for the position in the Z-direction.

8. In the **A** field, enter -90° for the tilting angle of the saw blade.

9. In the **C** field, enter -90° for the pivoting angle of the saw blade.

10. Check all saw cut assistant inputs in the profile view.

11. Press the ✓ button to apply the values to the **SAW CUT** machining task.

⚠ **INFORMATION**	
ⓘ	Press the 🟡 button to call up the saw assistant once again to make any changes. Press the ✓ button to apply the changes to the **SAW CUT** machining task.

12. The machining task is activated automatically but can be deactivated at any time.

13. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

14. The value of 100% is already entered in the **FEED** cell.

15. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

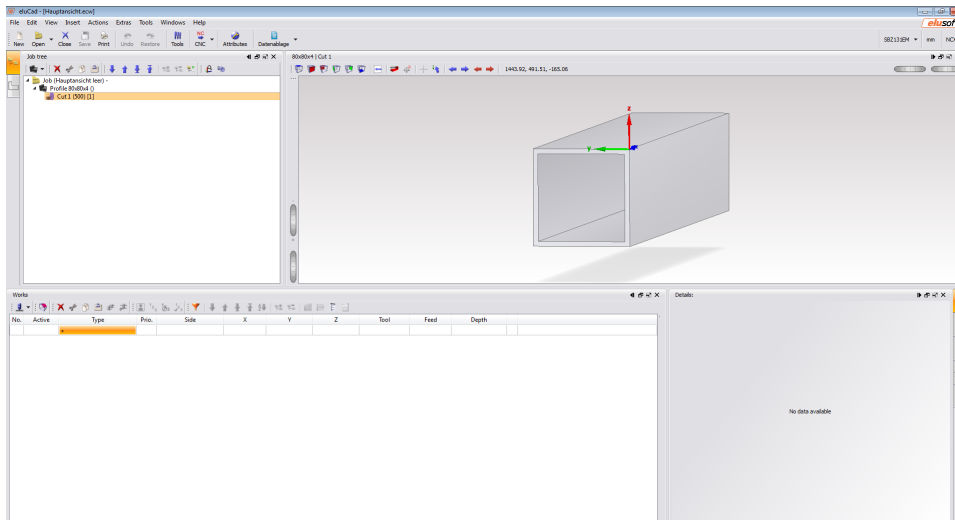|		
-	**Notch angle:**	Enter 45° for the notch angle.
-	**Correct saw depth:**	Activate the correction of the saw depth.
-	**Max. saw depth:**	The display is automatic.
-	**Saw blade:**	Set the selection to **LEFT**.
-	**A-axis:**	The value is generated automatically by the saw assistant.
-	**C-axis:**	The value is generated automatically by the saw assistant.

16. No entries are required on the **PARAMETERS** tab.

17. Check all inputs in the profile view.
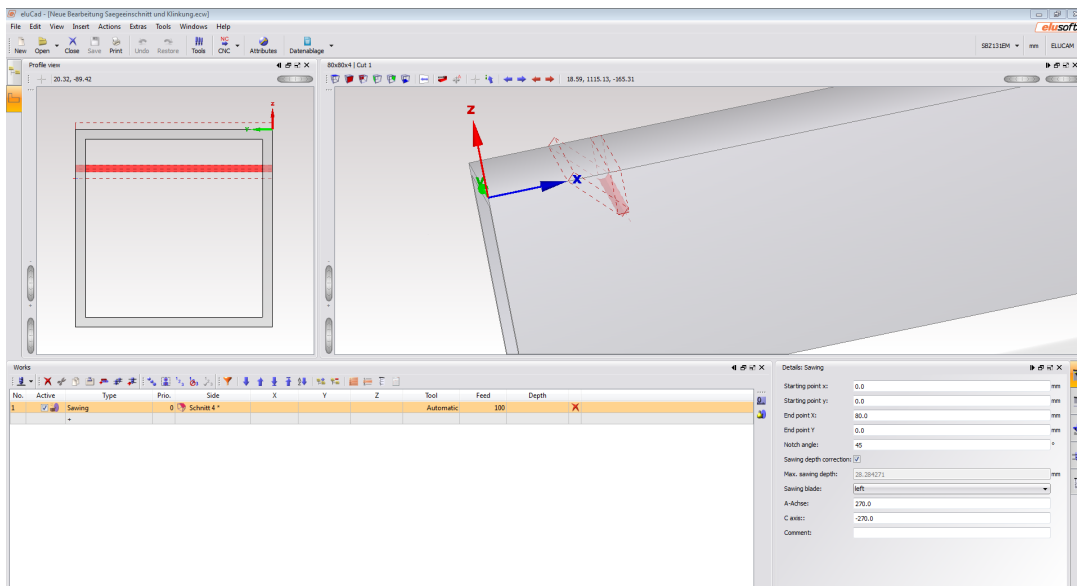
2.2.2.19   Creating a machining task with notching

This example shows the manual creation of a notch.

Default machining task:
-   Notch: Position left; X = 50 mm; Z = 20 mm
-   Type 1 notch

To create the notch, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **NOTCH**.

3. By selecting the next cell, the notch automatically opens.

4. The machining task is activated automatically but can be deactivated at any time.

5. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

6. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    -   **Type:**   Selection of the type and side of the notch

        Select type no. 1 via the selection dialog.

        Select notch at the beginning of the part.

    -   **1:**   Enter 50 mm for the X-position of the notch.

    -   **2:**   Enter 20 mm for the Y-position of the notch.

    -   **Waste material plane:**   The desired plane must be defined via the selection field. The default waste material plane presetting is top.
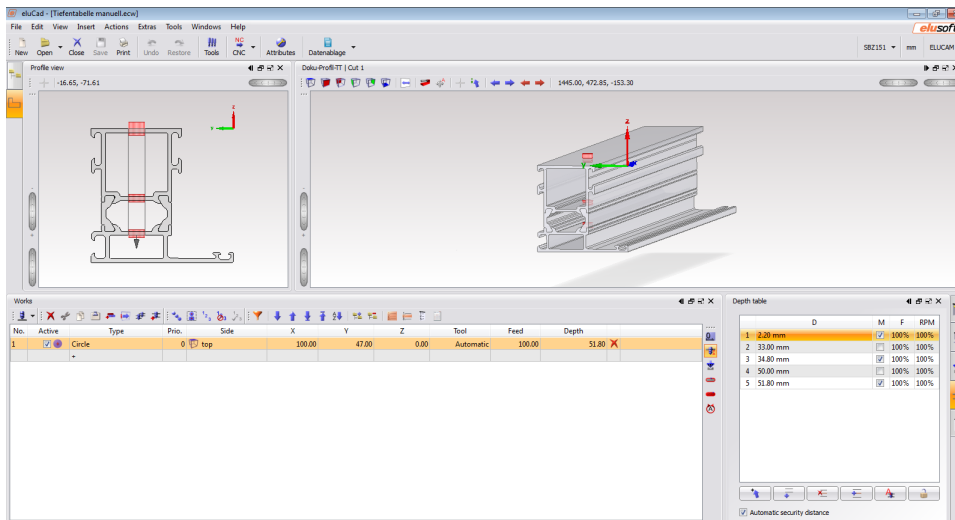
7. No entries are required on the **PARAMETERS** tab.

8. Check all inputs in the profile view.
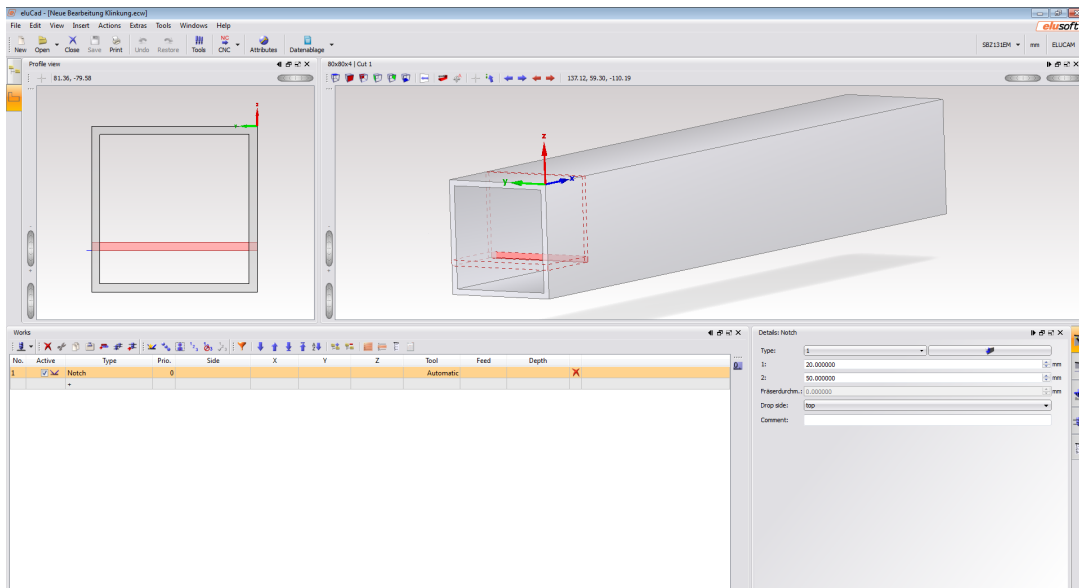
2.2.3 Depth table

The **DEPTH TABLE** describes tool immersion in the material. It is primarily important when machining through the individual chambers.

⚠ INFORMATION

i	The starting point is the same as the retraction point and therefore the point of movement to the next machining operation.

Only the individual walls, the safety distances and the machining speeds on the workpiece are described in the **DEPTH TABLE**. The actual values of the workpiece must always be entered. The safety distances and the additional depth are added automatically.
Depending on the requirements, the feed rate and the spindle speed can be set for the respective machining task and the depth step.

The **DEPTH TABLE** is created manually or automatically .

2.2.3.1 Creating a depth table manually

This example shows the manual creation of a depth table.
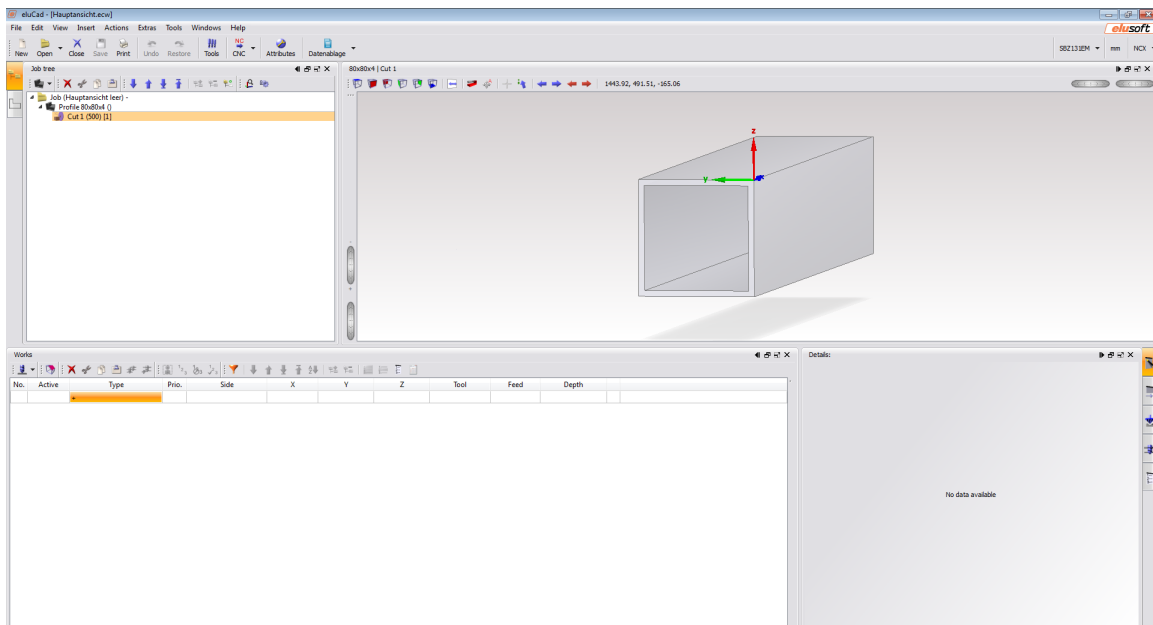
The following steps are necessary to create a **DEPTH TABLE** manually:

1. Open the **PROGRAM EDITOR** and enter the desired machining task.

2. Enter the depth of the machining task in the **PLUNGE DEPTH** input field on the **DETAIL** tab.

3. The ⬛ button can be used to have the depth calculated automatically. The values can then be used as a basis and can be edited.

4. Open the depth table using the ⬛ button on the **DETAIL** tab or using the **DEPTH TABLE** tab.

⚠ INFORMATION

ℹ	The actual values of the profile must always be entered. Safety distances and additional depth are added automatically.

---

5. Select the first line.

   - Enter the dimension of the material thickness (wall) in cell **D**.

   - Activate the selection field in cell **M** to mark the existing material (wall) in the size range.

   - Enter the feed rate in percent for the material (wall) in cell **F**.

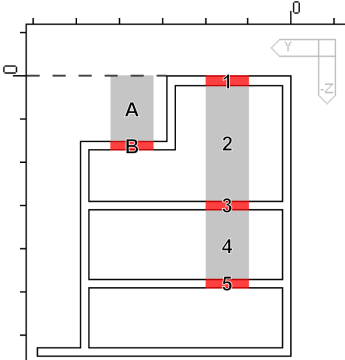   - Enter the spindle speed in percent for the material (wall) in the **RPM** cell.

   Note: If there is a spacing between the origin and the 1st material (wall), an additional row must be added beforehand. Here, the dimension between the origin and the 1st material (wall) must be entered in cell **D**.

6. For additional walls, you must perform the following steps:

   a) Use the ⬛ button to insert an additional row.

   - Enter the dimension for the space between the end of the 1st and the beginning of the 2nd material in cell **D** .

- Deactivate the selection field in cell **M** to mark the dimensional range as a space.

- Enter the feed rate in percent for the space in cell **F**.

- Enter the spindle speed in percent for the space in the **RPM** cell.

b) Use the ⊡ button to insert an additional row.

- Enter the dimension of the material thickness (wall) in cell **D**.

- Activate the selection field in cell **M** to mark the existing material (wall) in the size range.

- Enter the feed rate in percent for the material (wall) in cell **F**.

- Enter the spindle speed in percent for the material (wall) in the **RPM** cell.

c) For additional walls, repeat the input for the next lines as described in a) and b).

7. Once all walls and spaces have been entered, close the **DEPTH TABLE** by selecting a different tab.
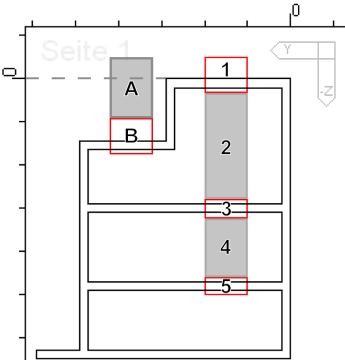
Example:

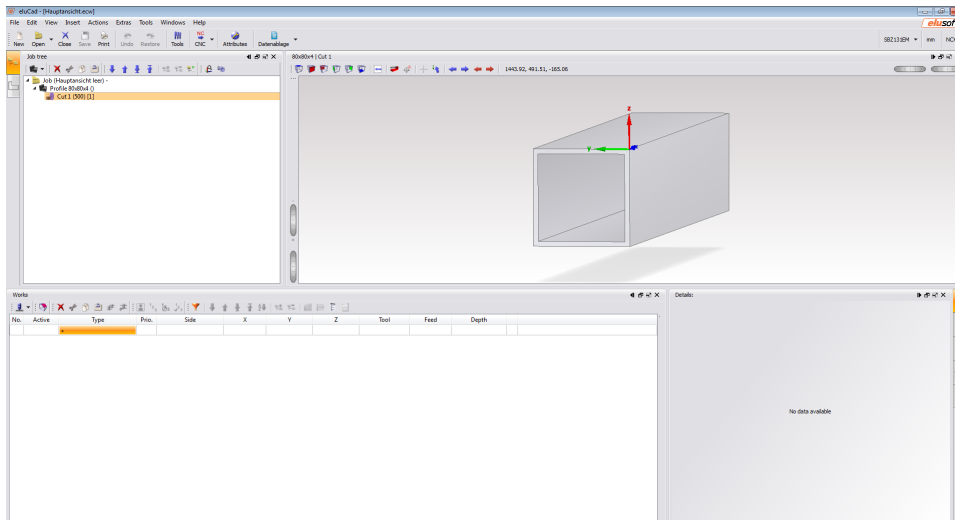| Machining through a wall	Schematic sketch:	Machining through several walls
1. Profile wall is 30 mm away from the origin (A) and contains no material. Maximum feed until next wall. 2. Profile wall with 1.8 mm material thickness (B). Feed in the wall with 80 %.		1. Profile wall with 2.2 mm material thickness (1). Feed in the wall with 80 %. 2. Second profile wall is 33 mm away (2) and does not contain any material. Maximum feed until next wall. 3. Profile wall with 1.8 mm material thickness (3). Feed in the wall with 80 %. 4. Third profile wall is 15.2 mm away (4) and does not contain any material. Maximum feed until next wall. 5. Profile wall with 1.8 mm material thickness (5). Feed in the wall with 80 %.

Depth table (left)

	D	M	F	RPM
1	30.00 mm	☐	100%	100%
2	31.80 mm	☑	80 %	100 %

☑ Automatic security distance

Depth table (right)

	D	M	F	RPM
1	2.20 mm	☑	100%	100%
2	33.00 mm	☐	100%	100%
3	34.80 mm	☑	100%	100%
4	50.00 mm	☐	100%	100%
5	51.80 mm	☑	100%	100%

☑ Automatic security distance

Schematic sketch: Depth table with safety distances and additional depth

2.2.3.2   Creating a depth table automatically

This example shows the automatic creation of a depth table.

The following steps are necessary to create a **DEPTH TABLE** automatically:

1. Open the **PROGRAM EDITOR** and enter the desired machining task in the list.

2. Enter the depth of the machining task in the **PLUNGE DEPTH** input field on the **DETAIL** tab.

3. There are 2 options available for creating the **DEPTH TABLE** automatically:

    1) Use the [button icon] button to create the **DEPTH TABLE** automatically.

    2) Select depth using mouse:

       - Use the mouse pointer to select the **PROFILE CROSS-SECTION** field.

       - An arrow is displayed on the machining task automatically.

       - Select the tip of the arrow and drag it to the desired target point by holding down the left mouse button.

       - After the mouse button is released, the **DEPTH TABLE** is generated automatically up to the selected target point.
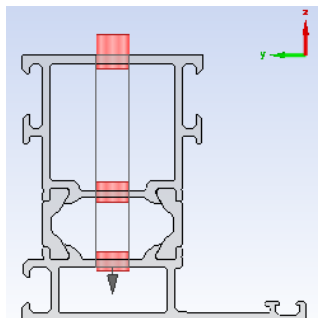

4. When creating the **DEPTH TABLE,** the software automatically calculates all intersection points of the machining task with the existing material (walls). The result is applied automatically in the **DEPTH TABLE** along with the defined safety distances. The graphic display of the **DEPTH TABLE** is shown in the **PROFILE CROSS-SECTION** field and in the **PROFILE VIEW** field.
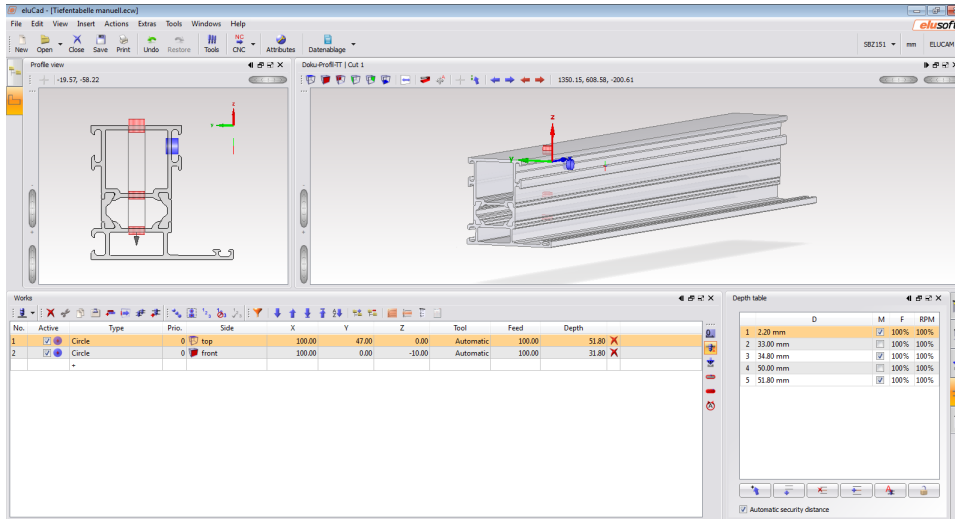
2.2.4   Groups / Macros

A **GROUP** describes several individual machining tasks which are grouped into a single machining task for the program. If a group can be used in several programs, the data are saved as a macro.
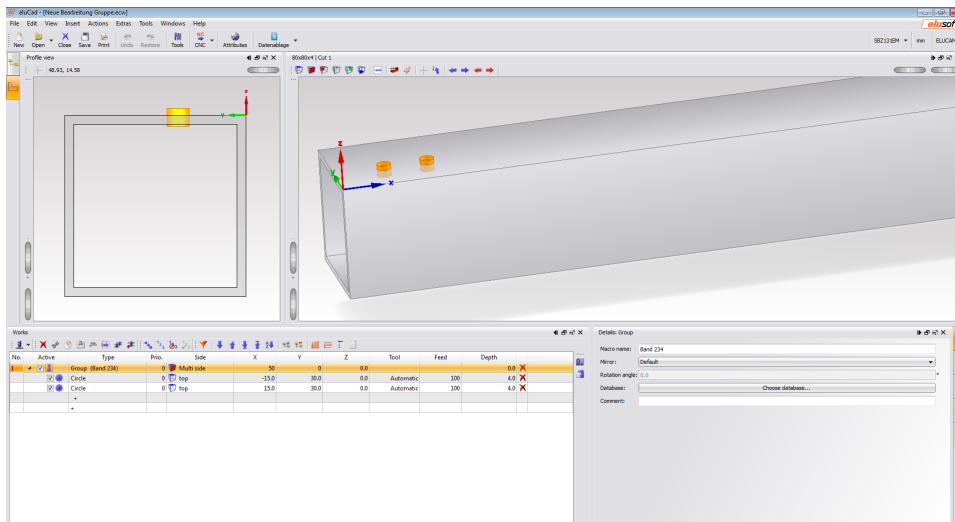
> ⚠ **INFORMATION**
>
> Groups and macros are displayed in a different colour in the **PROFILE VIEW** field so that the association of the individual machining tasks is apparent.
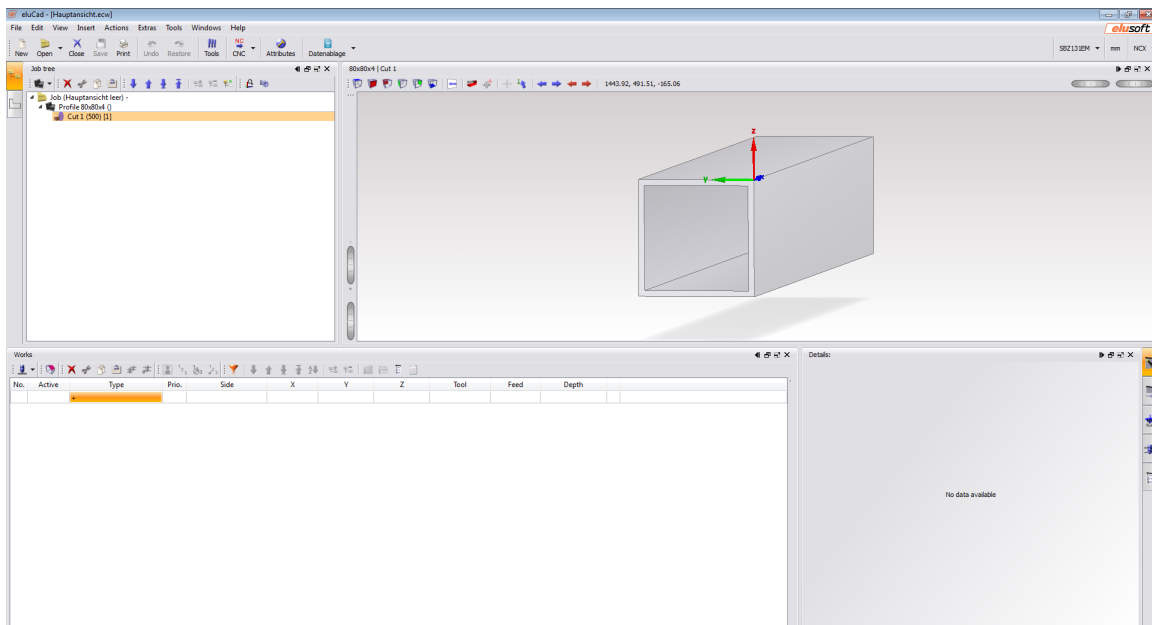
If several machining tasks are required together, these can be grouped together in a **GROUP**. Within the **GROUP**, the individual machining tasks all refer to a common point of reference. This point is also used for the positioning of the entire **GROUP**.
In the **GROUP,** the individual machining tasks are always listed as subpoints in the **MACHINING TASK LIST**.

In order to be able to use a **GROUP** in other programs, it must be saved in a database as a **MACRO**. The **MACRO** is inserted into the **MACHINING TASK LIST** in the same way as a machining task. The subpoints with the individual machining tasks are not listed.

2.2.4.1   Creating a new machining task as a group

This example shows the manual creation of a machining task as a group.
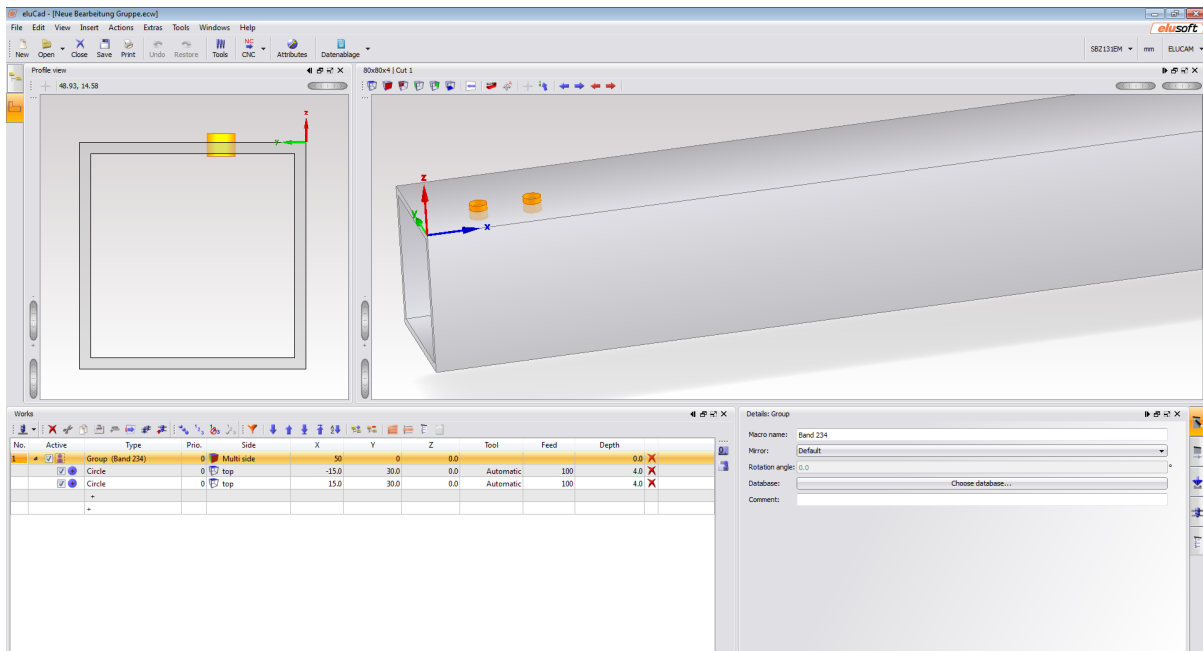

Default machining task:
- Home position of the group; X = 50 mm; Y = 0 mm
- 2 circle pockets each with a 10 mm diameter
- Circle pocket position 1: top; X = -15 mm; Y = 30mm
- Circle pocket position 2: top; X = 15 mm; Y = 30mm
- Work feed rate 100%
- Depth 10 mm

To create the machining task as a machining chain, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **GROUP**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **MULTI-SIDED** machining side is selected automatically.

5. Enter 50 mm for the X-home position of the group in the **X** cell.

6. The **Y** cell already contains 0 for the Y-home position of the group.

7. The **Z** cell already contains 0 for the Z-position.

8. Enter the data and values for the group in the input fields of the **DETAIL** tab.

   - **Macro name:**   Enter the name of the machining task group (macro).

   - **Mirror:**   Select the setting *Normal*.

9. Select the first blank subline in the list of profile machining tasks.
   Info: Sublines are displayed with an offset **+** symbol.

10. Open the selection window in the **TYPE** cell with a double click and select the type **CIRCLE**.

11. The machining task is activated automatically but can be deactivated at any time.

12. In the **SIDE** cell, the **TOP** machining side is selected automatically.

13. Enter -15 mm for the X-position in the **X** cell.

14. Enter 30 mm for the Y-position in the **Y** cell.

15. The **Z** cell already contains 0 for the Z-position.

16. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

17. The value of 100% is already entered in the **FEED** cell.

18. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

19. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    - **Diameter:**   Enter 10 mm for the diameter of the machining task.

    - **Tool diameter:**   If needed, enter the desired tool diameter.

20. Select the second blank subline in the list of profile machining tasks.

21. Open the selection window in the **TYPE** cell with a double click and select the type **CIRCLE**.

22. The machining task is activated automatically but can be deactivated at any time.

23. In the **SIDE** cell, the **TOP** machining side is selected automatically.

24. Enter 15 mm for the X-position in the **X** cell.

25. Enter 30 mm for the Y-position in the **Y** cell.

26. The **Z** cell already contains 0 for the Z-position.

27. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

28. The value of 100% is already entered in the **FEED** cell.

29. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

30. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    - **Diameter:**   Enter 10 mm for the diameter of the machining task.

    - **Tool diameter:**   If needed, enter the desired tool diameter.

31. Check all inputs in the profile view.

2.2.4.2 Saving a new machining task as a macro

This example shows the manual creation of a machining task which is to be saved as a macro.
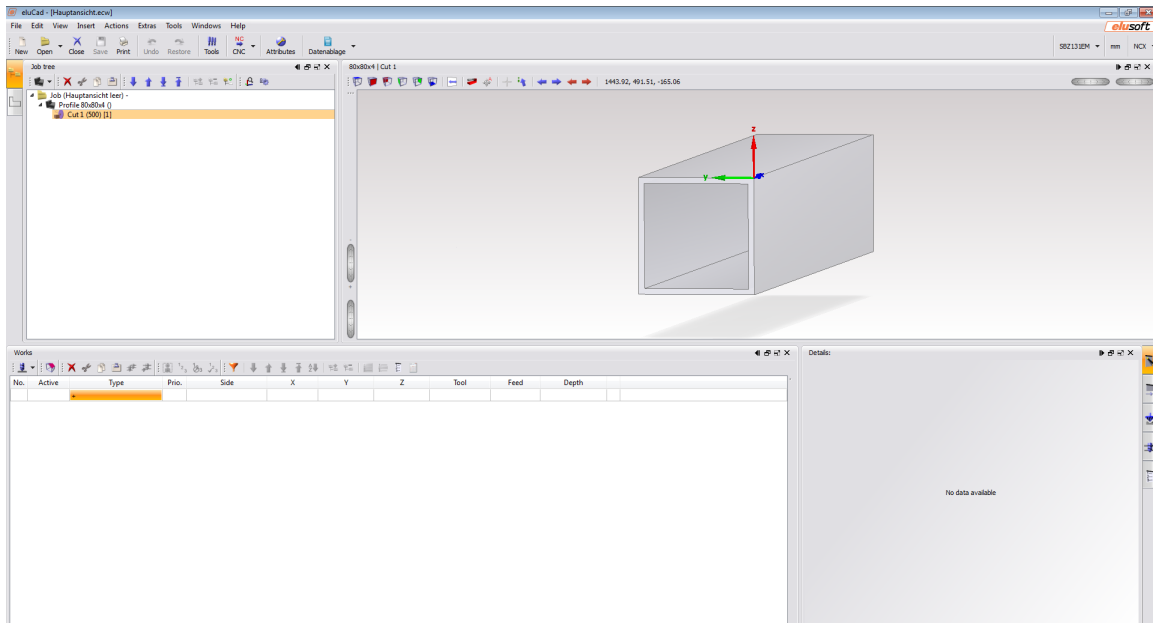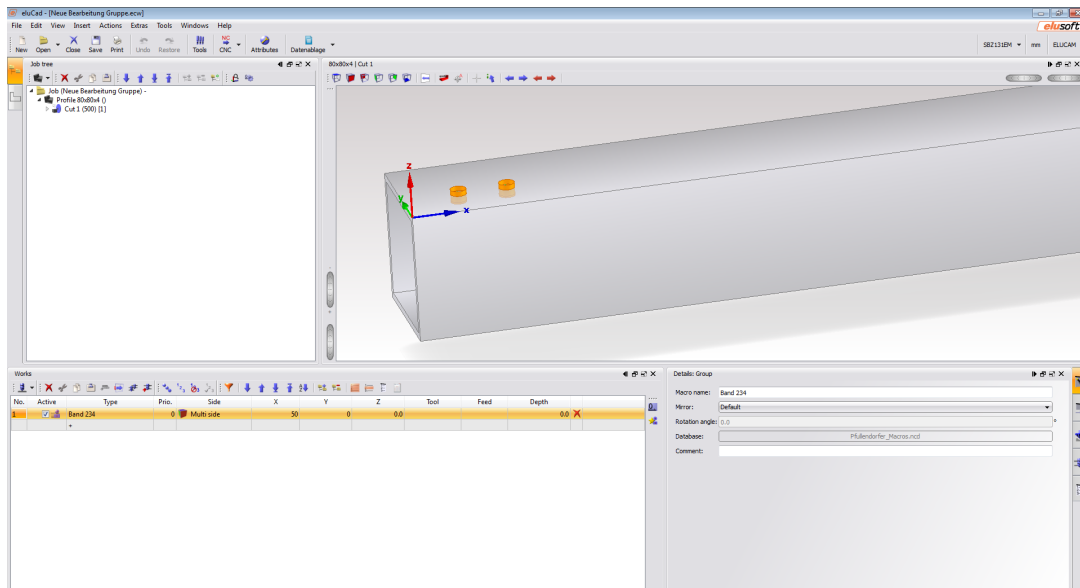

Default machining task:
- Home position of the group; X = 50 mm; Y = 0 mm
- 2 circle pockets each with a 10 mm diameter
- Circle pocket position 1: top; X = -15 mm; Y= 30 mm
- Circle pocket position 2: top; X = 15 mm; Y= 30 mm
- Work feed rate 100%
- Depth 10 mm

To save the machining task as a macro, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **GROUP**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **MULTI-SIDED** machining side is selected automatically.

5. Enter 50 mm for the X-home position of the group in the **X** cell.

6. Enter 30 mm for the Y-home position of the group in the **Y** cell.

7. The **Z** cell already contains 0 for the Z-position.

8. Enter the data and values for the group in the input fields of the **DETAIL** tab.

   - **Macro name:**   Enter the desired designation for the macro

   - **Mirror:**   Select the setting *Normal*.

9. Select the first blank subline in the list of profile machining tasks.
   Info: Sublines are displayed with an offset **+** symbol.

10. Open the selection window in the **TYPE** cell with a double click and select the type **CIRCLE**.

11. The machining task is activated automatically but can be deactivated at any time.

12. In the **SIDE** cell, the **TOP** machining side is selected automatically.

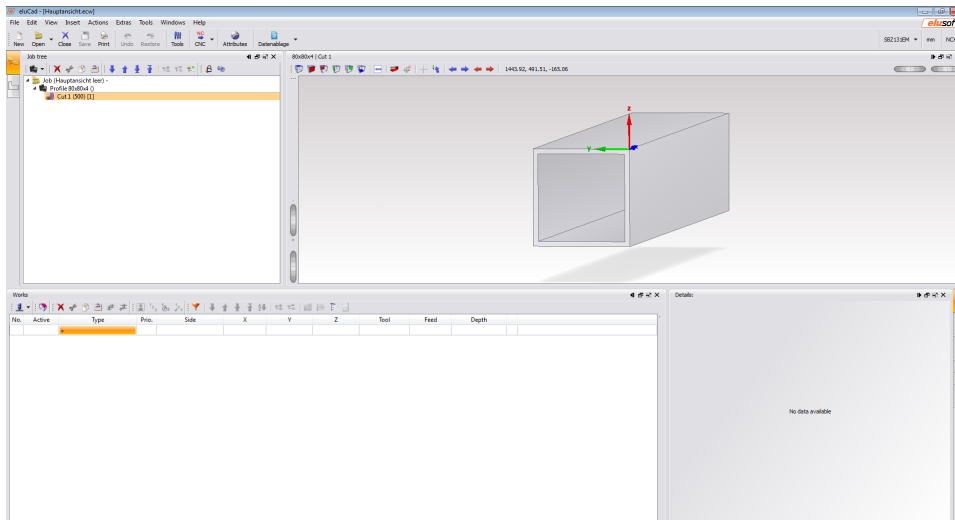13. Enter -15 mm for the X-position in the **X** cell.

14. Enter 10 mm for the Y-position in the **Y** cell.

15. The **Z** cell already contains 0 for the Z-position.

16. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

17. The value of 100% is already entered in the **FEED** cell.

18. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

19. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    - **Diameter:**   Enter 10 mm for the diameter of the machining task.

    - **Tool diameter:**   If needed, enter the desired tool diameter.

20. Select the second blank subline in the list of profile machining tasks.

21. Open the selection window in the **TYPE** cell with a double click and select the type **CIRCLE**.

22. The machining task is activated automatically but can be deactivated at any time.

23. In the **SIDE** cell, the **TOP** machining side is selected automatically.

24. Enter 15 mm for the X-position in the **X** cell.

25. Enter 30 mm for the Y-position in the **Y** cell.

26. The **Z** cell already contains 0 for the Z-position.

27. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

28. The value of 100% is already entered in the **FEED** cell.

29. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

30. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    - **Diameter:**   Enter 10 mm for the diameter of the machining task.

    - **Tool diameter:**   If needed, enter the desired tool diameter.

31. Select the line with the group designation.

32. Use the button to save the group as a macro.

33. An information window which displays the saving of the macro will open automatically.

34. Press the **OK** button to confirm saving and to close the information window.

35. Check all inputs in the profile view.

2.2.4.3　Creating a new machining task with a macro

This example shows the manual creation of a machining task with a macro.
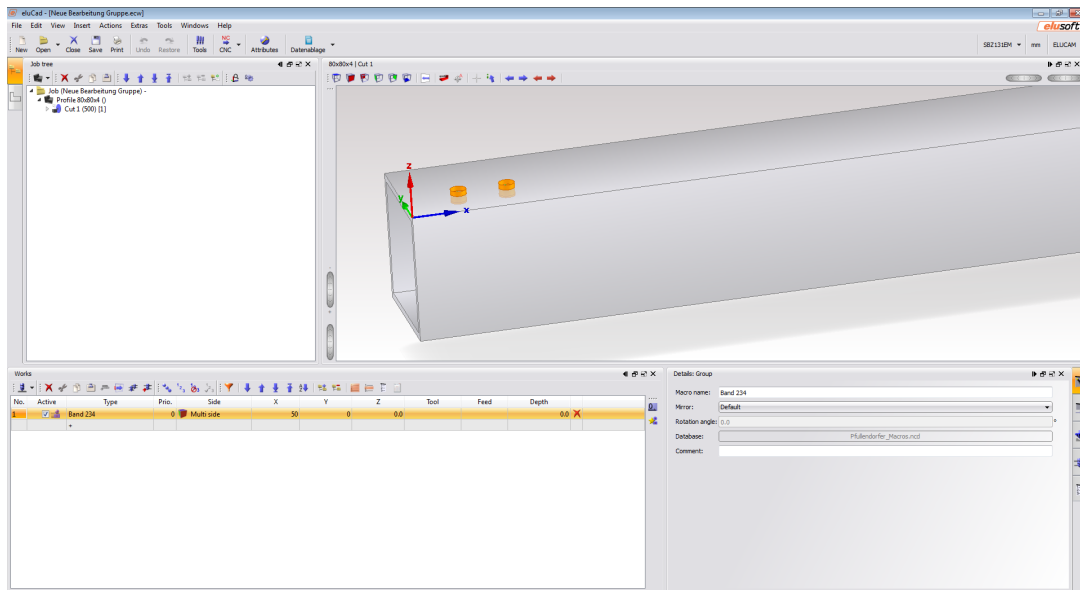

Default machining task:
- Home position of the macro; X = 50 mm; Y = 0 mm
- 2 circle pockets each with a 10 mm diameter
- Macro used: Docu-macro
- Work feed rate 100%
- Depth 10 mm

To create the machining task as a machining chain, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **MACRO**.

3. By selecting the next cell, the **MACRO DATABASE** menu with the previously created macros opens automatically.

4. Select the desired macro in the **MACRO DATABASE**
   The machining task of the selected macro is displayed in the **PREVIEW** field.

5. Pressing the **OK** button applies the macro in the list of the machining and closes the **MACRO DATABASE** menu.

6. The machining task is activated automatically but can be deactivated at any time.

7. In the **SIDE** cell, the **MULTI-SIDED** machining side is selected automatically.

8. Enter 50 mm for the X-position of the macro in the **X** cell.

9. The **Y** cell already contains 0 mm for the Y-position.

10. The **Z** cell already contains 0 for the Z-position.

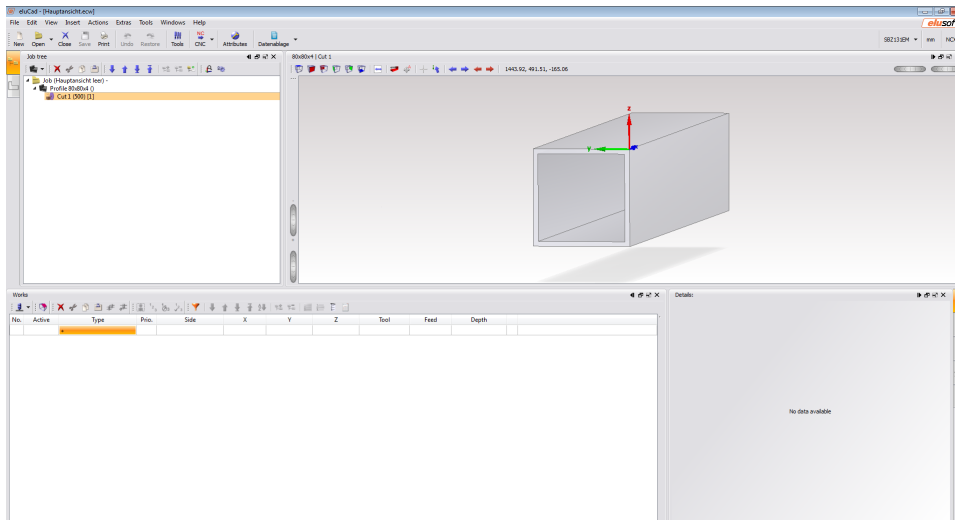11. Enter the data and values for the group in the input fields of the **DETAIL** tab.

   - **Mirror:**　　　　　　　Select the setting *Normal*.

12. Check all inputs in the profile view.

2.2.4.4 Creating a multi-sided group

This example shows the manual creation of a machining task as a machining multi-sided group.

Default machining task:
- Home position of the group; X = 50 mm; Y = 0 mm
- 3 drilled holes with a diameter of 10.5 mm each
- Drilled hole position 1: top; X = 0 mm; Y = PW/2 or Y/2; Z= 0 mm
- Drilled hole position 2: front; X = 0 mm; Y = 0 mm; Z= -PH/2 or -Z/2
- Drilled hole position 3: back; X = 0 mm; Y = PW or Y; Z= -PH/2 or -Z/2
- Depth 10 mm

To create the machining task as a machining chain, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **GROUP**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **MULTI-SIDED** machining side is selected automatically.

5. Enter 50 mm for the X-home position of the group in the **X** cell.

6. The **Y** cell already contains 0 for the Y-home position of the group.

7. The **Z** cell already contains 0 for the Z-position.

8. Enter the data and values for the group in the input fields of the **DETAIL** tab.

   - **Macro name:**   Enter the name of the machining task group (macro).

   - **Mirror:**   Select the setting *Normal*.

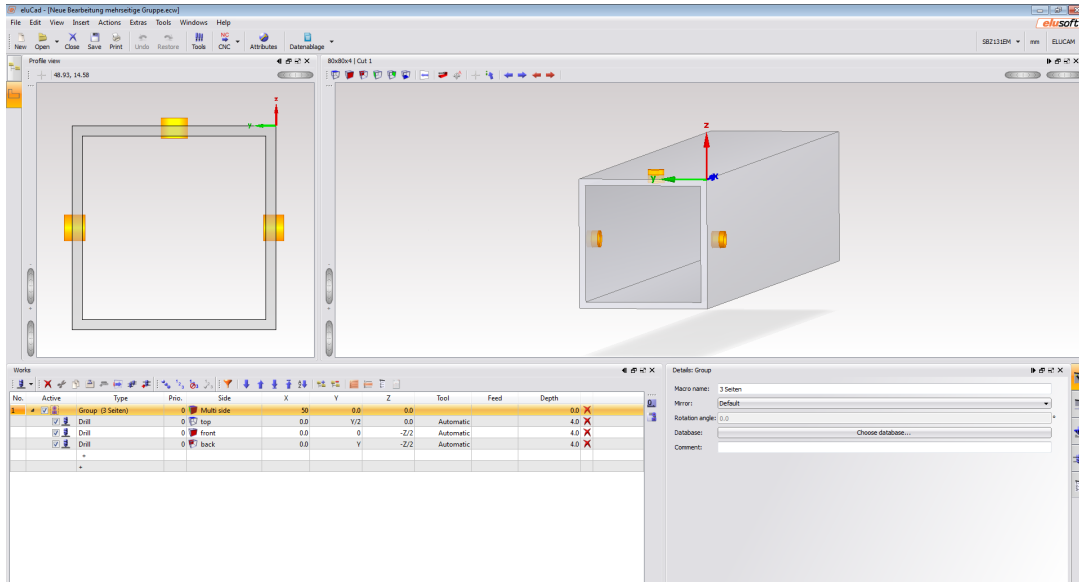9. Select the first blank subline in the list of profile machining tasks.
   Info: Sublines are displayed with an offset **+** symbol.

10. Open the selection window in the **TYPE** cell with a double click and select the type **DRILLING**.

11. The machining task is activated automatically but can be deactivated at any time.

12. In the **SIDE** cell, the **TOP** machining side is selected automatically.

13. The **X** cell already contains 0 mm for the X-position.

14. Enter PW/2 or Y/2 for the Y-position in the **Y** cell.

15. The **Z** cell already contains 0 mm for the Z-position.

16. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

17. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

18. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    - **Diameter:**   Enter 10.5 mm for the diameter of the machining task.

19. Select the second blank subline in the list of profile machining tasks.

20. Open the selection window in the **TYPE** cell with a double click and select the type **DRILLING**.

21. The machining task is activated automatically but can be deactivated at any time.

22. Open the selection window in the **SIDE** cell with a double click and select **FRONT** as the machining side.

23. The **X** cell already contains 0 mm for the X-position.

24. The **Y** cell already contains 0 mm for the Y-position.

25. Enter -PH/2 or -Z/2 for the Z-position in the **Z** cell.

26. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

27. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

28. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.

    - **Diameter:**   Enter 10.5 mm for the diameter of the machining task.

    - **Tool diameter:**   If needed, enter the desired tool diameter.

29. Select the third blank subline in the list of profile machining tasks.

30. Open the selection window in the **TYPE** cell with a double click and select the type **DRILLING**.

31. The machining task is activated automatically but can be deactivated at any time.

32. Open the selection window in the **SIDE** cell with a double click and select **BACK** as the machining side.

33. The **X** cell already contains 0 mm for the X-position.

34. Enter PW or Y for the Y-position in the **Y** cell.

35. Enter -PH/2 or -Z/2 for the Z-position in the **Z** cell.

36. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

37. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

38. Enter the data and values for the machining task in the input fields of the **DETAIL** tab.
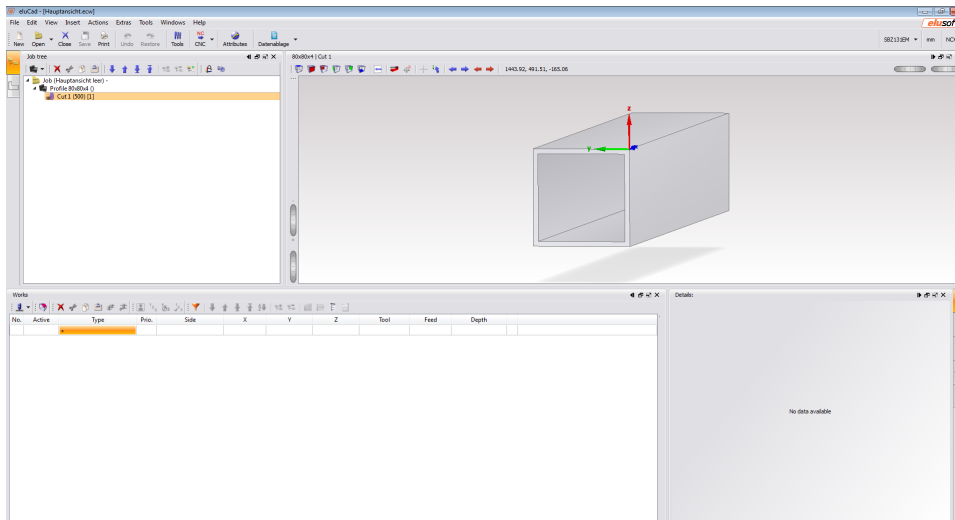
     - **Diameter:**   Enter 10.5 mm for the diameter of the machining task.

     - **Tool diameter:**   If needed, enter the desired tool diameter.

39. Check all inputs in the profile view.

2.2.4.5   Converting a macro to a group

This example shows the conversion of a macro to a group.

Default machining task:
- Home position of the macro; X = 50 mm; Y = 0 mm
- Macro used: Attachment A01

Perform the following steps to convert a macro to a group:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **MACRO**.

3. By selecting the next cell, the **MACRO DATABASE** menu with the previously created macros opens automatically.

4. Select the desired macro, *Attachment A01*, in the **MACRO DATABASE**
   The machining task of the selected macro is displayed in the **PREVIEW** field.

5. Pressing the **OK** button applies the macro in the list of the machining and closes the **MACRO DATABASE** menu.

6. The machining task is activated automatically but can be deactivated at any time.

7. In the **SIDE** cell, the **MULTI-SIDED** machining side is selected automatically.

8. Enter 50 mm for the X-position of the macro in the **X** cell.

9. The **Y** cell already contains 0 mm for the Y-position.

10. The **Z** cell already contains 0 for the Z-position.

11. Select the line with the group designation.

12. Pressing the [button icon] button saves the macro as a group.

13. An information window opens automatically.

14. Press the **YES** button to confirm saving and to close the information window.

15. Check all inputs in the profile view.
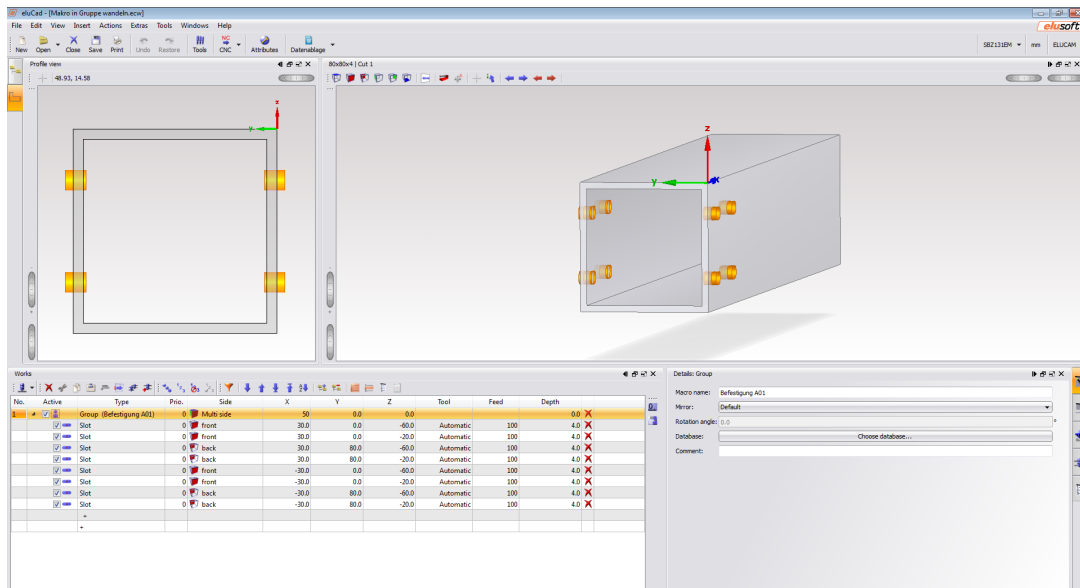
2.2.5 Manual changing of working orientation

In some programs it is not possible to execute the machining task with the profile in its normal working orientation. It is therefore necessary to rotate the profile to the appropriate position before beginning with machining. As the machining tasks are performed, the machine operator is prompted to rotate the profile. When creating a machining task, the programmer must define the position of the profile on the **PARAMETERS** tab under **CHANGE WORKING ORIENTATION**.

Parameters: Drill	
Orientation change:	0
Condition:	
Machine condition:	STOP
Relative position:	310.00 mm

⚠ **INFORMATION**

ℹ️ The machine first processes all machining tasks which can be performed in the standard position. Then the message display will prompt you rotate the profile. After manually rotating the profile, the machining operation is continued by pressing the **START (F12)** button.

The following steps must be performed to specify a manual change of working orientation:

1. In the **MACHINING TASKS** table, select the desired machining task in the list of profile machining tasks.
2. Selecting the **PARAMETERS** tab opens additional settings for the machining task.
3. Enter the desired working orientation directly in the **CHANGE WORKING ORIENTATION** field or select it using the selection buttons.
   Only a value of 0-7 may be entered or selected! The assignments of the working orientations can be found in the chapter Quick help for working orientations.
4. When the next input field is selected, the selected value is applied automatically.

2.2.6 Machining task conditions

Machining tasks can be adapted to various jobs using conditions. A machining operation is only performed when the corresponding condition is met.
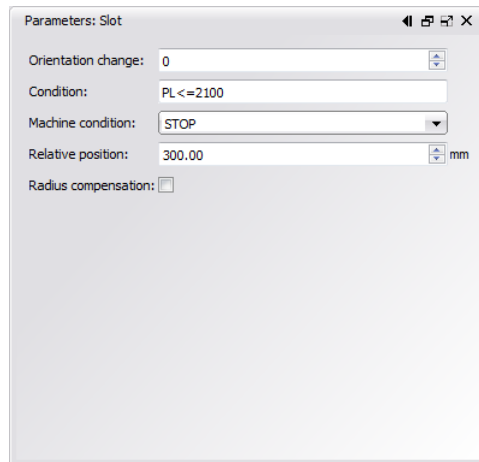
To create a condition, you must perform the following steps:

1. In the **MACHINING TASKS** table, select the desired machining task in the list of profile machining tasks.
2. Selecting the **PARAMETERS** tab opens additional settings for the machining task.
3. Enter the appropriate condition in the **CONDITION** input field.

> ⚠ **INFORMATION**
>
> ℹ️ It is possible to use variables that you have created yourself. These will be requested when the program is started.

Example:
If *PL<=2100* is entered in the **CONDITION** input field, the profile will only be machined if the profile length is 2100 mm or less.

2.2.7 Machine conditions for machining tasks

Machining tasks can be adapted to various jobs by means of the machine conditions. A machining task is only performed if the corresponding machine condition is met. The machine conditions are fixed and can only be selected accordingly.
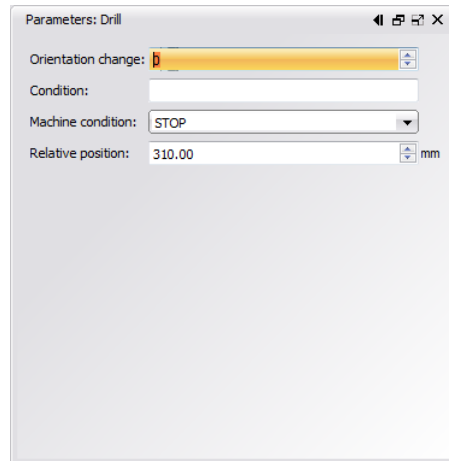
To create a machine condition, the following steps must be performed:

1. Open the **PROGRAM EDITOR** and select the desired machining operation in the list of profile machining operations.

2. Selecting the **PARAMETERS** tab opens additional settings for the machining task.

3. Select the respective condition in the **MACHINE CONDITION** input field. Example: var0>1200

4. A position can be entered in the **RELATIVE POSITION** input field to achieve an offset position.

Example:
If *Stop is entered* in the **MACHINE CONDITION** input field, the machine moves to the home position and waits for a start command.
If a value is also entered in the **RELATIVE POSITION** input field, the machine first moves to the home position. The machine then moves to the X-position entered and waits for a start command.

2.2.8   Using copy and insert in profile machining tasks

This example demonstrates the copying and insertion of values in the individual cells.


To copy a cell, perform the following steps:

+--+
| ⚠ **INFORMATION** |
+------------------+---+
	The copying and insertion of cells can only be done within the same co-
ℹ	lumn. Values from other columns cannot be inserted!
	When an entire row is selected, the copy function is applied to the entire
	row.
+------------------+---+

1. Mark the desired origin cell in the list of profile machining operations.

2. Press the [icon] button to copy the value.

3. Mark the desired target cell in the list of profile machining tasks.

4. Press the [icon] button to insert the value.

5. Check all the entries in the list of profile machining tasks.
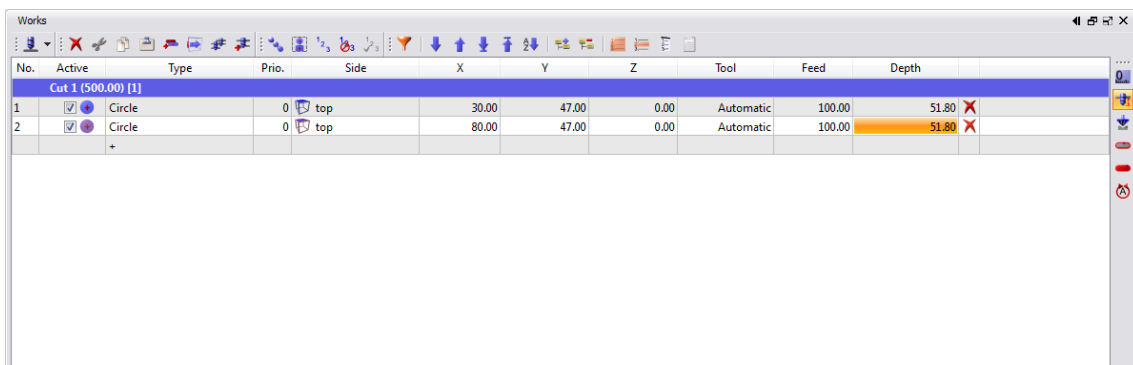
2.2.9   Modifying several values in profile machining tasks

This example shows how to modify several values in the table.
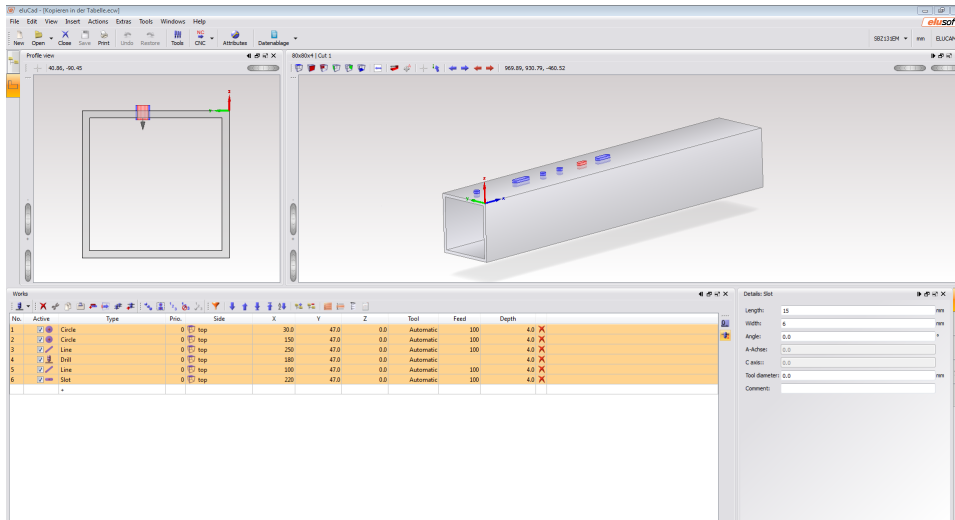
To modify several values at once, perform the following steps:

⚠ **INFORMATION**

> The modification of several values can only be done within the same column. Values from other columns cannot be changed!

---

1. Mark the desired origin cell in the list of profile machining operations.

2. Use the ⇧ key on the keyboard and the left mouse button to mark further cells in the column.

3. Enter the desired value.

4. Press the ↵ key on the keyboard to apply the value in all marked cells.

5. Check all the entries in the list of profile machining tasks.
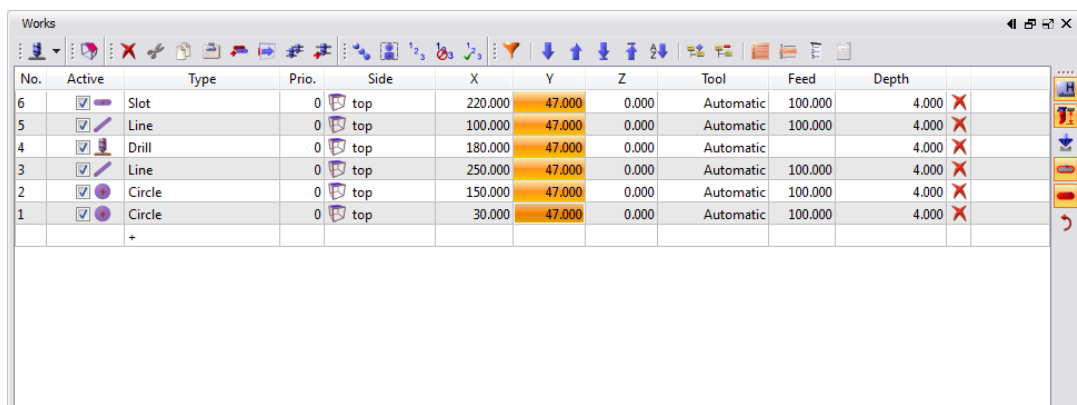
2.2.10 Using drag and drop in profile machining tasks

This example demonstrates the copying and insertion of values in the individual cells using the drag and drop function.
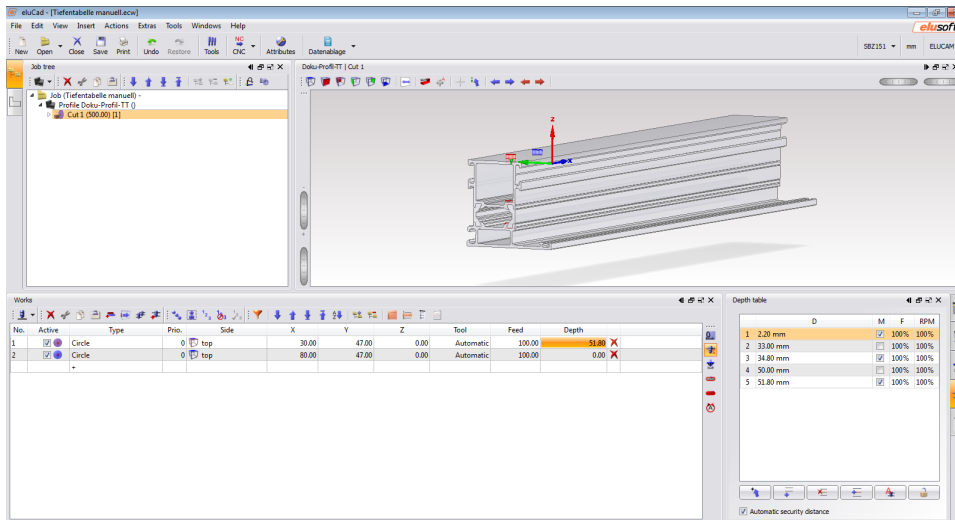
To copy a cell, perform the following steps:

⚠	**INFORMATION**
i	The copying and insertion of cells can only be done within the same column. Values from other columns cannot be inserted! When an entire row is selected, the copy function is applied to the entire row.

1. Mark the desired origin cell in the list of profile machining operations.
2. Select the cell with the left mouse button and hold the button down.
3. While the left mouse button is held down, select the desired target cell.
4. When the left mouse button is released, the value is inserted into the target cell.
5. Check all the entries in the list of profile machining tasks.
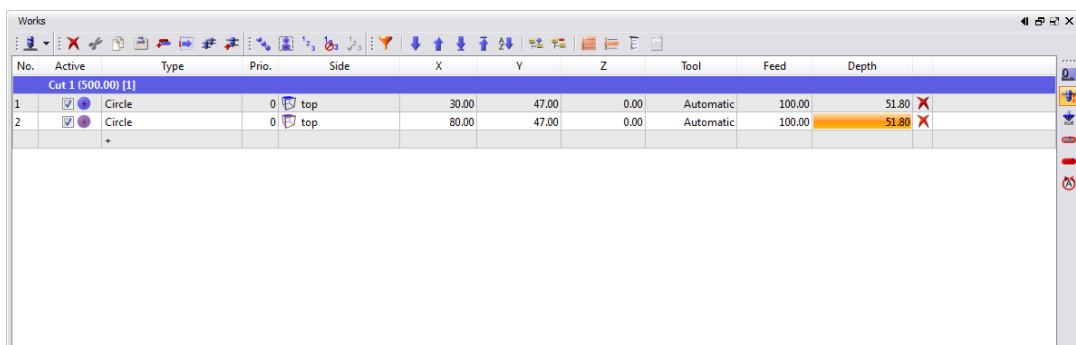
2.2.11 Performing the filter function in profile machining tasks

This example shows the filter function in the list of profile machining tasks.
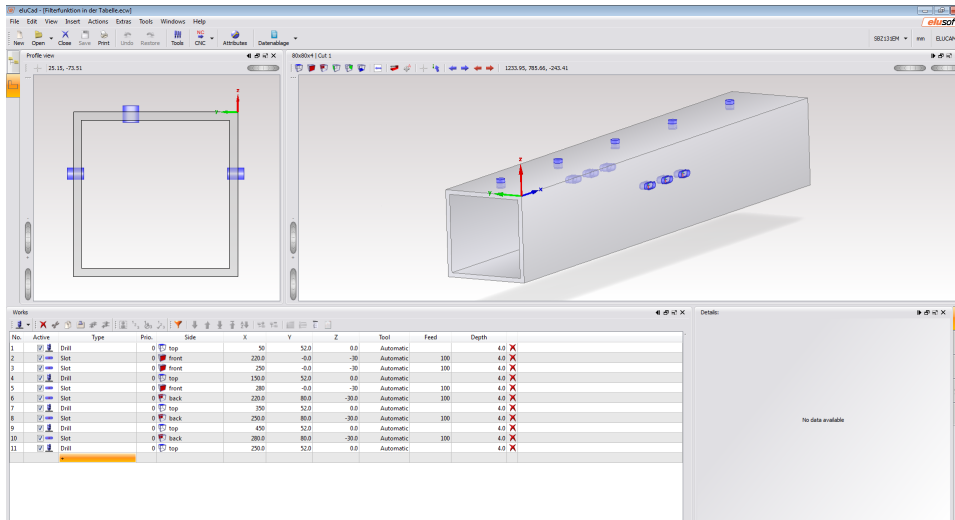
Operation:
- Filter for all drilling operations from the top

The following steps are required to perform the filter function:

1. The [icon] button displays the **FILTER** row.

2. To filter for the bore holes, enter a **B** in the input field of the **TYPE** filter cell.

3. Now, only the machining tasks of the **DRILLING** type are displayed in the list of profile machining tasks.

4. In the input field of the **SIDE** filter cell, enter an **O** to filter for the top side.

5. Now, only the machining tasks of the **DRILLING** type and the side **TOP** are displayed in the list of profile machining tasks.

6. Check all the entries in the list of profile machining tasks.
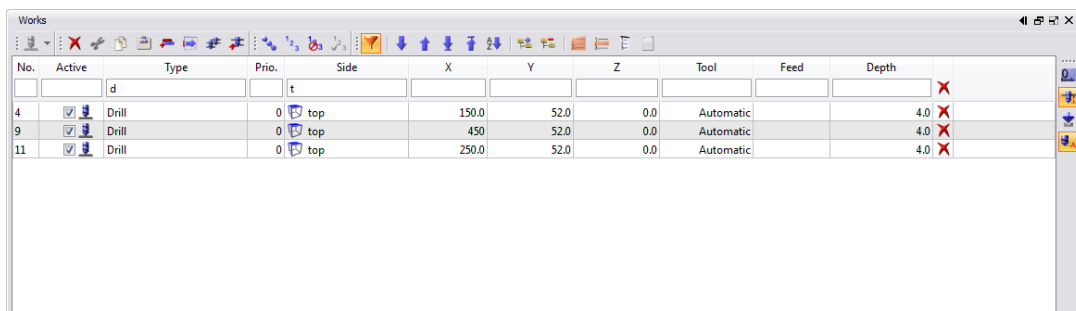
2.2.12 Performing automatic priority assignment in profile machining tasks

This example shows the automatic assignment of priorities in the list of profile machining tasks.
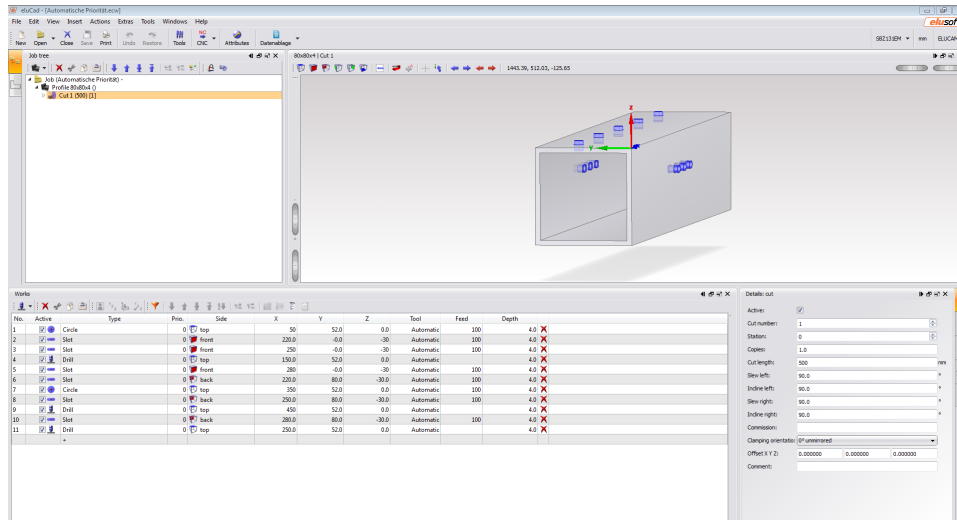
The following steps are required to perform the automatic assignment of priorities:
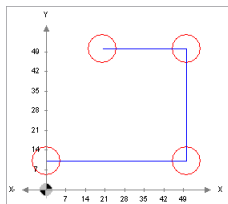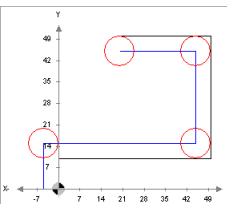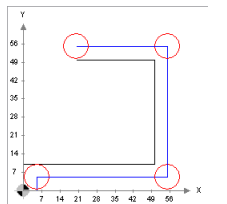
1. Use the button to have incremental priorities assigned to the individual machining tasks automatically.

2. Check all inputs in the profile view.

2.2.13 Free forms / Milling contours

Complex milling paths can be created in the **FREE FORM** menu. A free form, or also a milling contour, is entered in a program as a normal machining task. It is possible to enter them with basic CNC knowledge.

The free form always begins at the insertion point, which is defined in the machining task list. The milling path is created in the free form as a track with individual elements.
The selection of the direction defines the orientation of the milling cutter with respect to the programed milling path.

Centre	Left = Left-sided correction (G41)	Right = Right-sided correction (G42)
The tool machines the specified contour.	The tool performs machining to left of the specified contour.	Tool performs machining to the right of the specified contour.

The feed rate can be defined in the individual free form elements in the **FEED RATE** cell.

In order to be able to insert a free form into the machining task list, it must either be created new or selected from a free form database.
The selection is made by means of the two machining task types:
· **FREE FORM** - Creating new free forms
· **FREE FORM IMPORT** - Opening previously created free forms

2.2.13.1 Creating a free form without polar coordinates

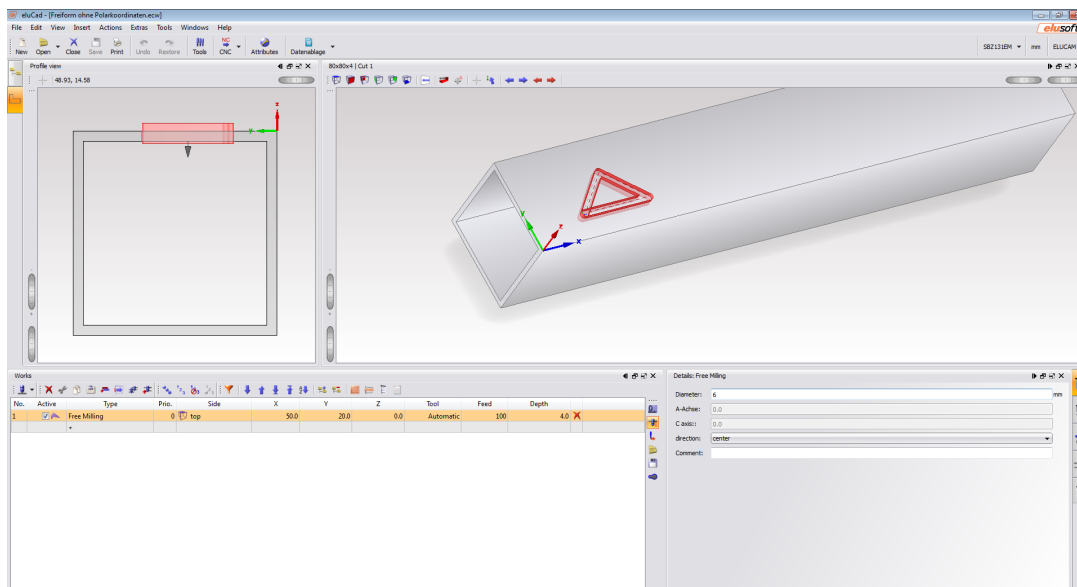This example shows the creation of a new free form without polar coordinates.

Default machining task:
- Home position of the free form: top; X = 50 mm; Y = 20 mm
- Free form with a triangular shape using a 6 mm tool
- 1st free form position X = 30 mm; Y = 30 mm
- 2nd free form position X = 60 mm
- 3rd free form position X = 0 mm; Y = 0 mm
- Work feed rate 100%
- Direction: centre
- Depth 10 mm

To create the free form without polar coordinates, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **FREE FORM**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **TOP** machining side is selected automatically.

5. Enter 50 mm for the X-home position of the free form in the **X** cell.

6. Enter 20 mm for the Y-home position of the free form in the **Y** cell.

7. The **Z** cell already contains 0 for the Z-position.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. The value of 100% is already entered in the **FEED** cell.

10. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

11. Pressing the button closes the **EDIT FREE FORM** menu.

12. Select the **INPUT** menu in the menu bar and deactivate the **POLAR** menu item.

    Info: When is displayed, the input with polar coordinates is activated.

13. Pressing the **NEW** button opens a new free form.

14. In the **POSITION X** input field, enter the value of 30 mm for the 1st machining task.

15. In the **POSITION Y** input field, enter the value of 30 mm for the 1st machining task.

16. In the **ARC** selection field, select the option **NONE**.

17. The value of 100% is already entered in the **FEED RATE** input field.

18. Pressing the **NEW** button opens a new free form element.

19. In the **POSITION X** input field, enter the value of 60 mm for the 2nd machining task.

20. In the **POSITION Y** input field, enter the value of 0 mm for the 2nd machining task.

21. In the **ARC** selection field, select the option **NONE**.

22. The value of 100% is already entered in the **FEED RATE** input field.

23. Pressing the **NEW** button opens a new free form element.

24. In the **POSITION X** input field, enter the value of 0 mm for the 3rd machining task.

25. In the **POSITION Y** input field, enter the value of 0 mm for the 3rd machining task.

26. In the **ARC** selection field, select the option **NONE**.

27. The value of 100% is already entered in the **FEED RATE** input field.

28. When the **ACCEPT** key is pressed, the new free form is added to the machining task list.

29. Enter the data and values for the group in the input fields of the **DETAIL** tab.

-	**Diameter:**	Enter 6 mm for the diameter of the machining task.
-	**Direction:**	Select **CENTRE** for the machining track of the tool.

30. Check all inputs in the profile view.

2.2.13.1.1 Example of a free form circle

This example shows the manual creation of a free form circle.

⚠ **INFORMATION**
i	The example is shown in simplified form. It shows the values for the individual free form entry steps. The exact procedure for creating a free form is explained in the chapter **CREATING A FREE FORM WITHOUT POLAR COORDINATES**[109].

Operation:
- Circle with diameter 20 mm
- Without tool correction
- Starting point is at centre

To create the free form circle, perform the following steps:

1. Free form element: Approach the free form

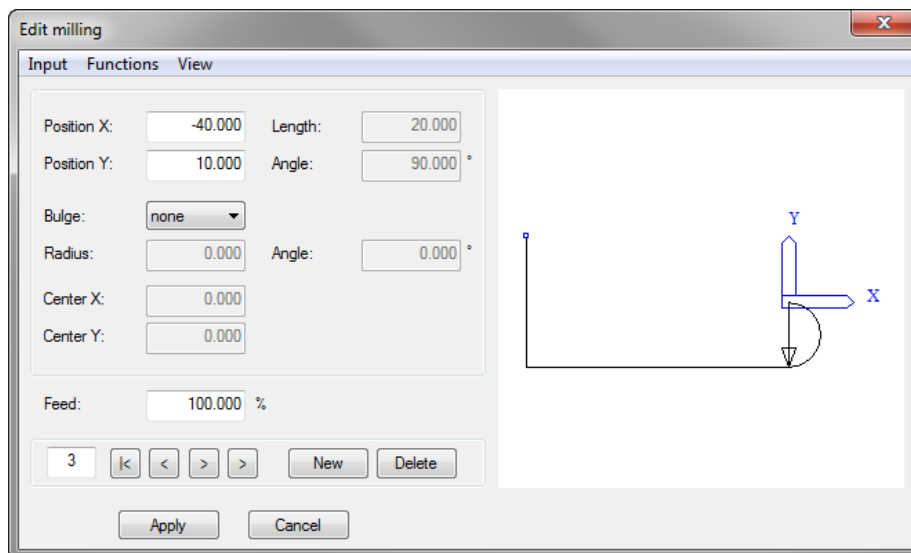| Position X	Position Y	Arc	Radius	Centre point X	Centre point Y
0	-10	Left	5	0	-5
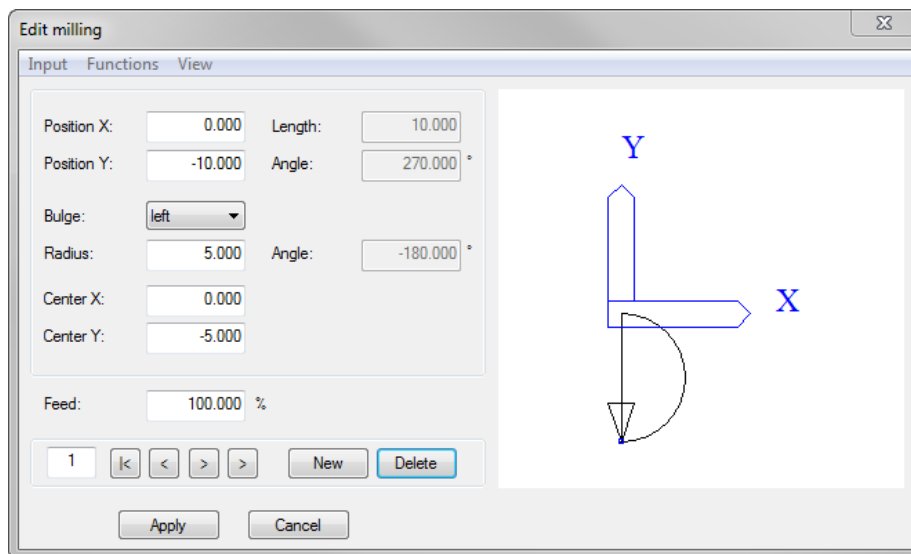
2. Free form element: First semicircle

Position X	Position Y	Arc	Radius	Centre point X	Centre point Y
0	10	Left	10	0	0
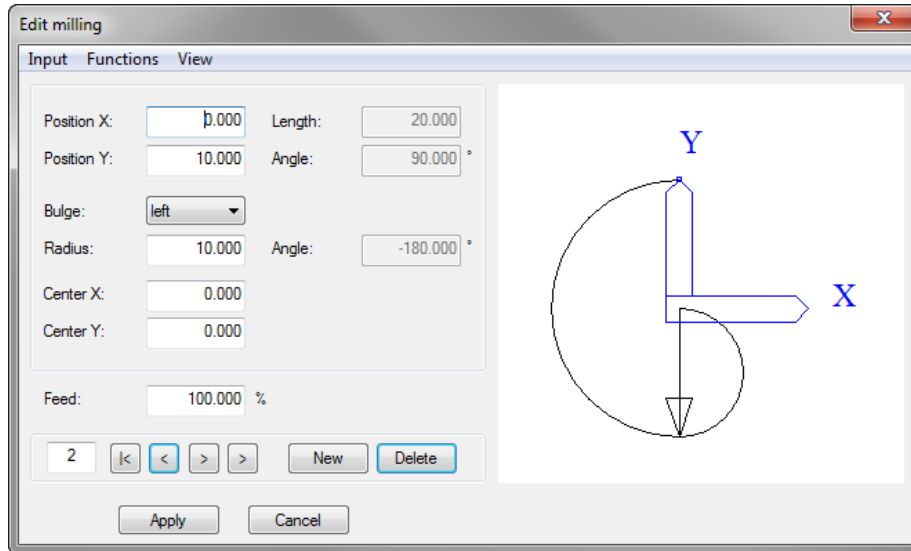
3. Free form element: Second semicircle

Position X	Position Y	Arc	Radius	Centre point X	Centre point Y
0	-10	Left	10	0	0
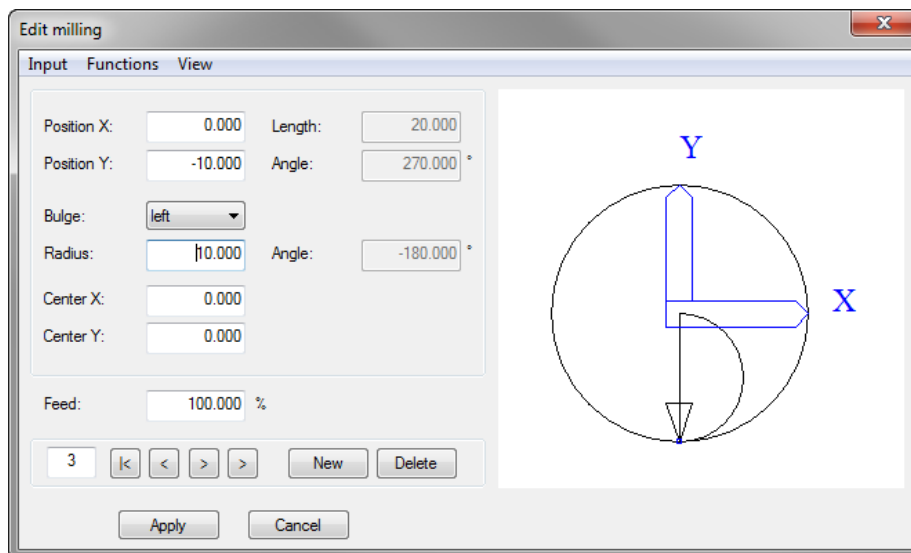
4. Free form element: Tracing the free form

Position X	Position Y	Arc	Radius	Centre point X	Centre point Y
0	0	Left	5	0	-5
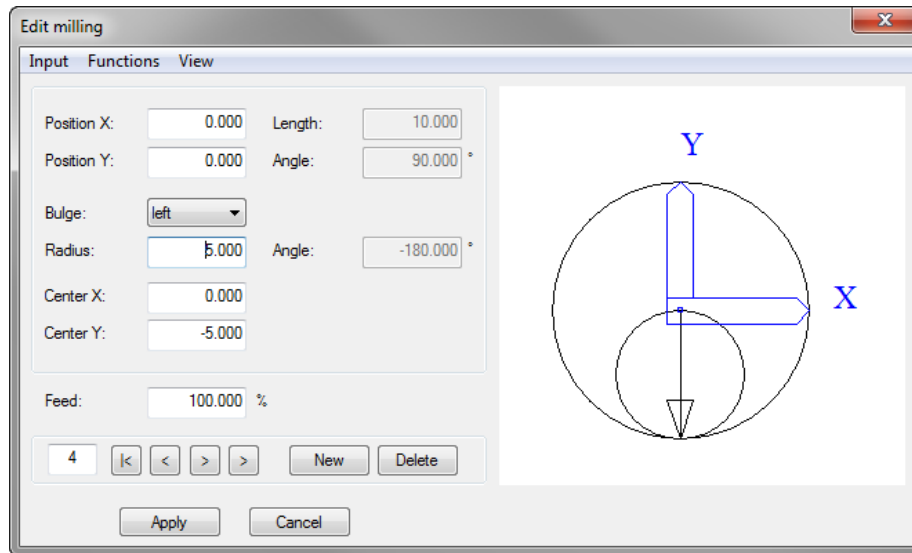
Entire free form in the **PROGRAM EDITOR**:
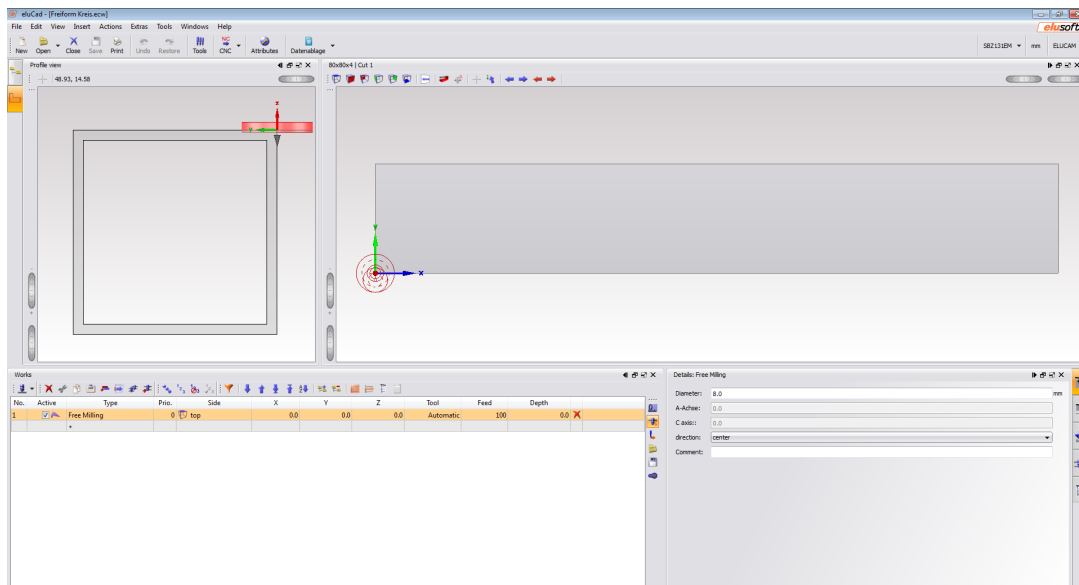
2.2.13.1.2 Example of a free form rectangle

This example shows the manual creation of a free form rectangle.

⚠ INFORMATION
ℹ	The example is shown in simplified form. It shows the values for the individual free form entry steps. The exact procedure for creating a free form is explained in the chapter **CREATING A FREE FORM WITHOUT POLAR COORDINATES**[109].

Operation:
- Rectangle with a length of 80 mm and a width of 20 mm
- Without tool correction
- Starting point is at centre

To create the free form rectangle, perform the following steps:

1. Free form element: Approach the free form

Position X	Position X	Arc	Radius	Centre point X	Centre point X
0	-10	Left	5	0	-5
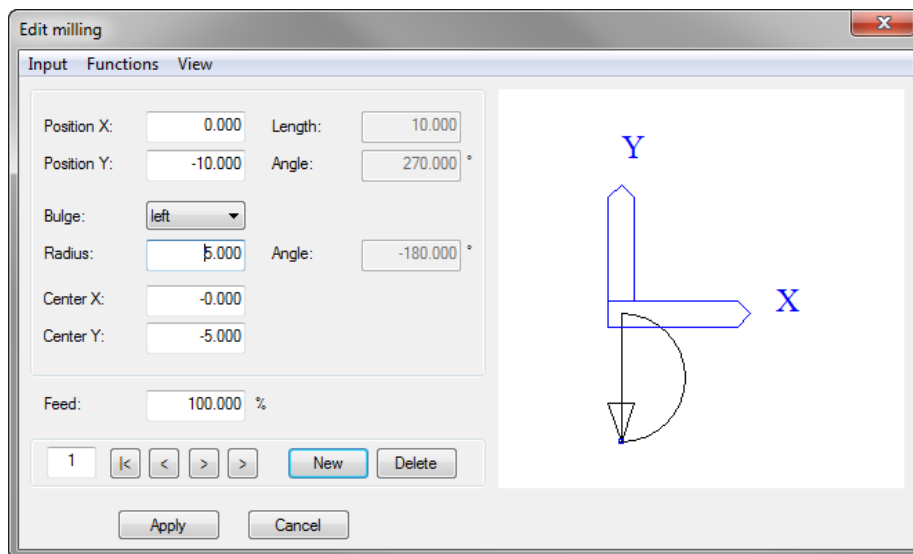
2. Free form element: First line

Position X	Position X	Arc	Radius	Centre point X	Centre point X
-40	-10	None	-	-	-
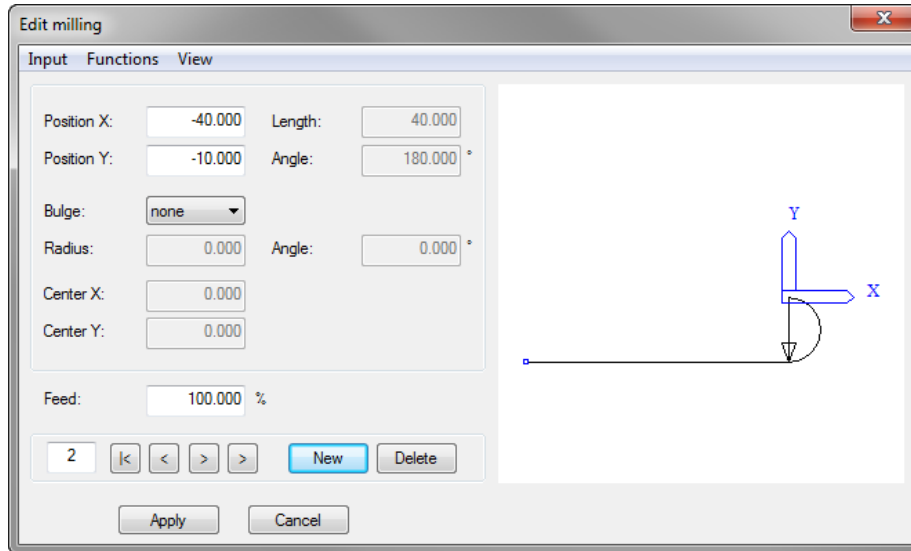
3. Free form element: Second line

Position X	Position X	Arc	Radius	Centre point X	Centre point X
-40	10	None	-	-	-

4. Free form element: Third line

Position X	Position X	Arc	Radius	Centre point X	Centre point X
40	10	None	-	-	-
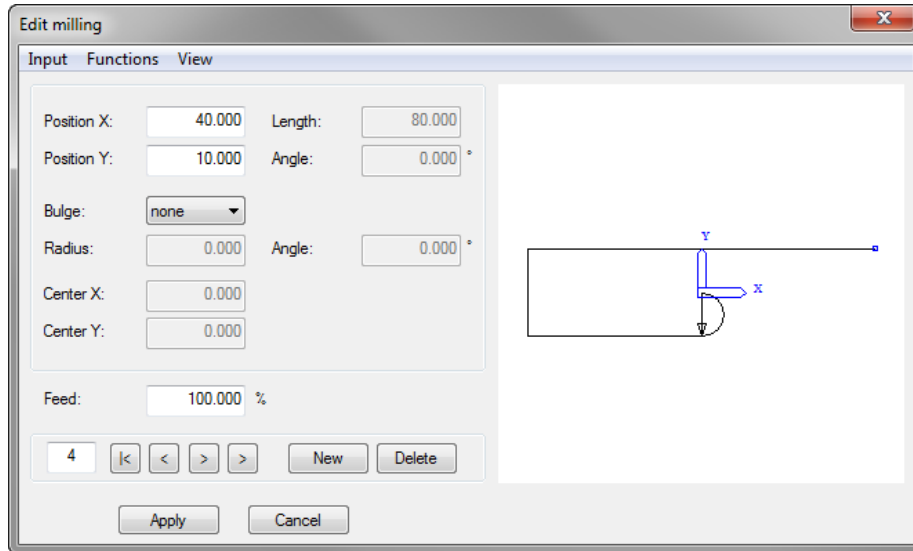
5. Free form element: Fourth line

Position X	Position X	Arc	Radius	Centre point X	Centre point X
40	-10	None	-	-	-
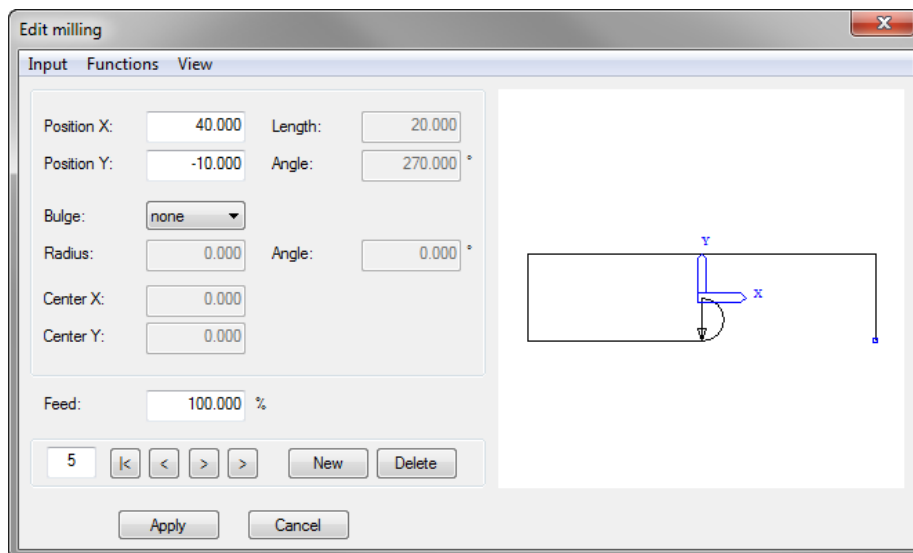
6. Free form element: Fifth line

Position X	Position X	Arc	Radius	Centre point X	Centre point X
0	-10	None	-	-	-
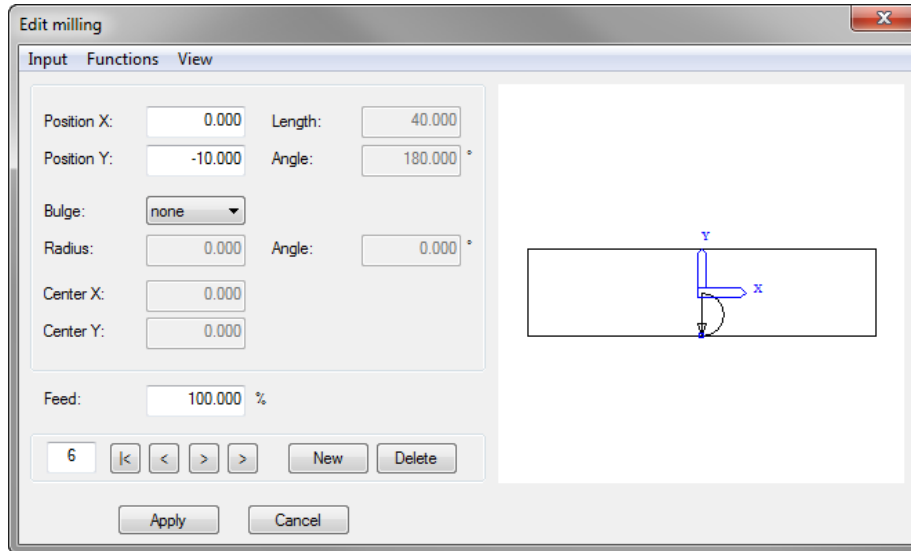
7. Free form element: Tracing the free form

Position X	Position X	Arc	Radius	Centre point X	Centre point X
0	0	Left	5	0	0
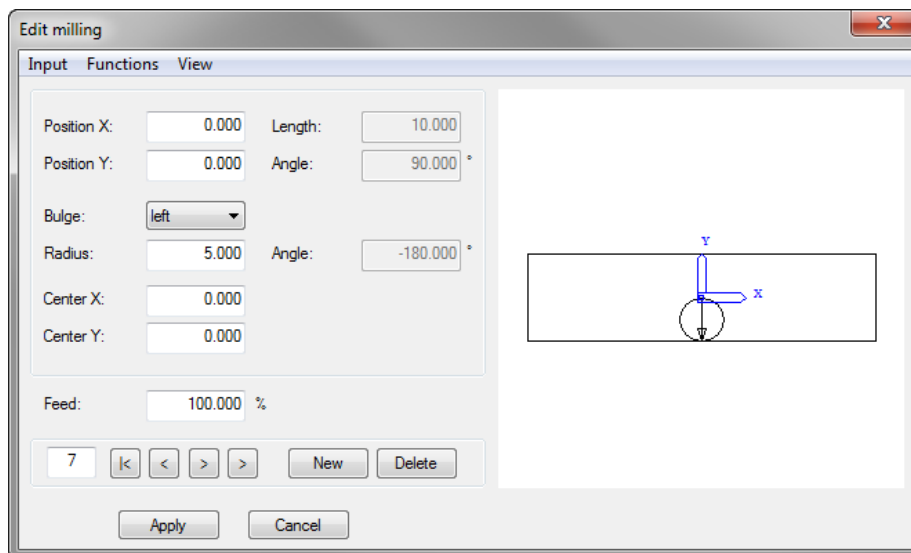
8. Entire free form in the **PROGRAM EDITOR**:
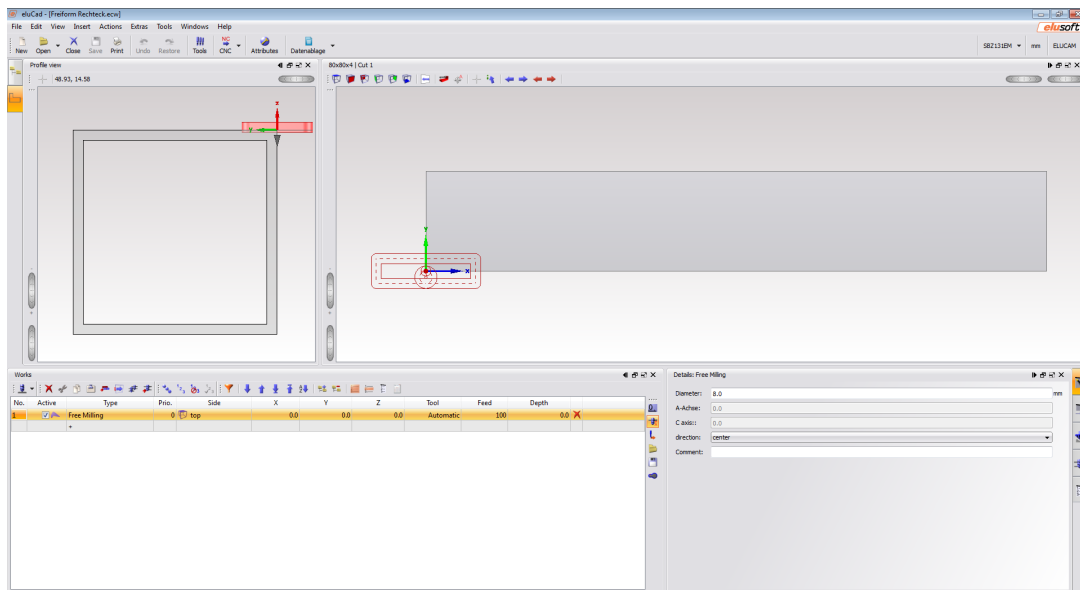
2.2.13.2 Creating a free form with polar coordinates

This example shows the creation of a new free form using polar coordinates.

Default machining task:
- Home position of the free form: top; X = 50 mm; Y = 20 mm
- Free form with a U shape, using a 6 mm tool
- 1st free form position X = 25 mm
- 2nd free form position Y = 15 mm
- 3rd free form position X = -25 mm
- Work feed rate 100%
- Direction: centre
- Depth 10 mm

To create the free form with polar coordinates, perform the following steps:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **FREE FORM**.

3. The machining task is activated automatically but can be deactivated at any time.

4. In the **SIDE** cell, the **TOP** machining side is selected automatically.

5. Enter 50 mm for the X-home position of the free form in the **X** cell.

6. Enter 20 mm for the Y-home position of the free form in the **Y** cell.

7. The **Z** cell already contains 0 for the Z-position.

8. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

9. The value of 100% is already entered in the **FEED** cell.

10. Enter the value of 10 mm in the **DEPTH** cell. Use the ⬛ button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the ⬛ button to manually create or change the depth table.

11. Pressing the ⬛ button closes the **EDIT FREE FORM** menu.

12. Select the **INPUT** menu in the menu bar and activate the **POLAR** menu item.

    Info: When ☑ is displayed, the input with polar coordinates is activated.

13. Pressing the **NEW** button opens a new free form.

14. In the **LENGTH** input field, enter the value of 25 mm for the 1st machining task.

15. In the **ARC** selection field, select the option **NONE**.

16. The value of 100% is already entered in the **FEED RATE** input field.

17. Pressing the **NEW** button opens a new free form element.

18. In the **LENGTH** input field, enter the value of 15 mm for the 2nd machining task.

19. In the **ANGLE** input field, enter the value of 90° for the 2nd machining task.

20. In the **ARC** selection field, select the option **RIGHT**.

21. In the **ANGLE** input field, enter the arc value of 180° for the 2nd machining task.

22. The value of 100% is already entered in the **FEED RATE** input field.

23. Pressing the **NEW** button opens a new free form element.

24. In the **LENGTH** input field, enter the value of -25 mm for the 3rd machining task.

25. In the **ANGLE** input field, enter the arc value of 180° for the 3rd machining task.

26. In the **ARC** selection field, select the option **NONE**.

27. The value of 100% is already entered in the **FEED RATE** input field.

28. When the **ACCEPT** key is pressed, the new free form is added to the machining task list.

29. Enter the data and values for the group in the input fields of the **DETAIL** tab.

- **Diameter:** Enter 6 mm for the diameter of the machining task.

- **Direction:** Select **CENTRE** for the machining track of the tool.

30. Check all inputs in the profile view.
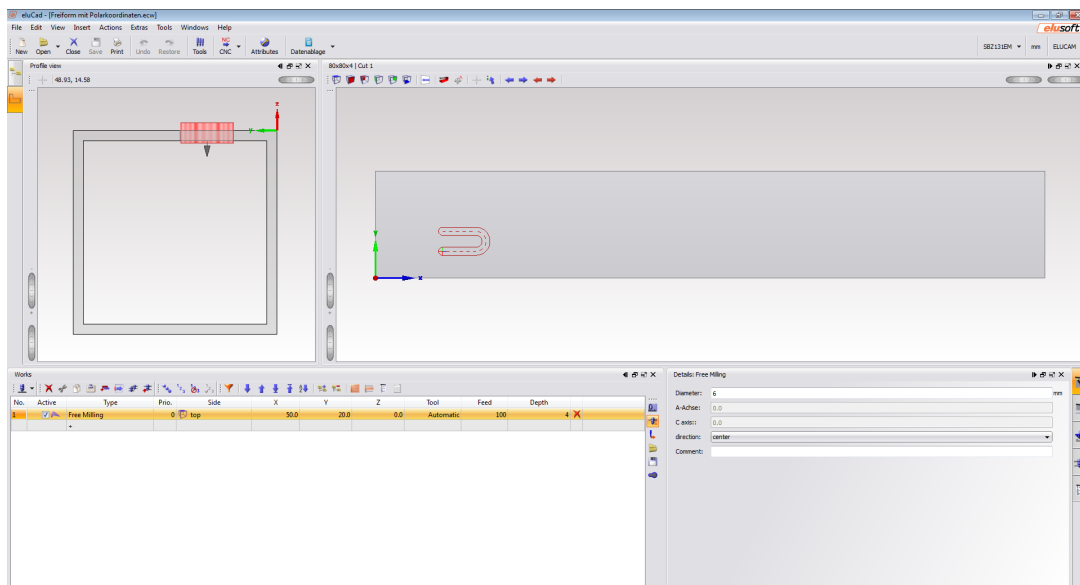
2.2.13.3 Importing a free form

This example shows the import of a previously existing free form.
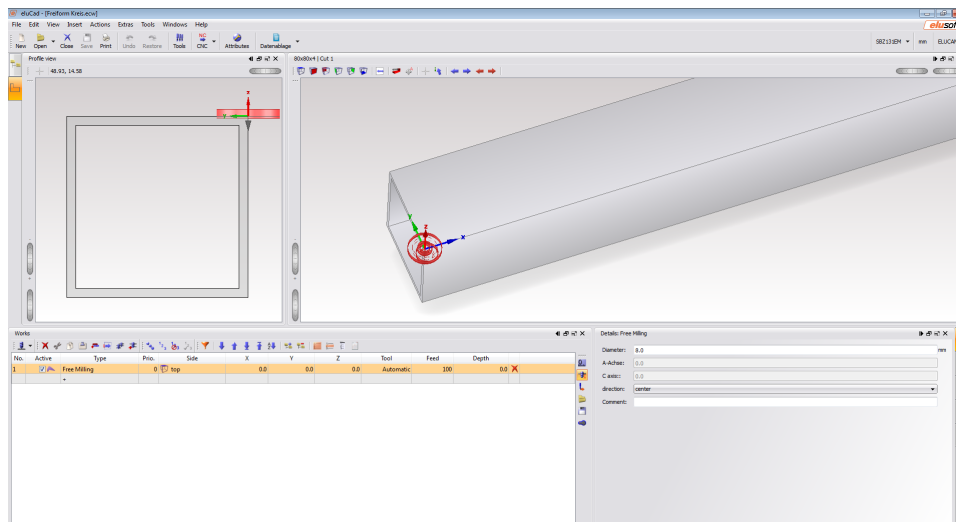
Default machining task:
- Import an existing free form circle
- Tool with a diameter of 6 mm
- Position front; X = 30 mm; Z = -20 mm
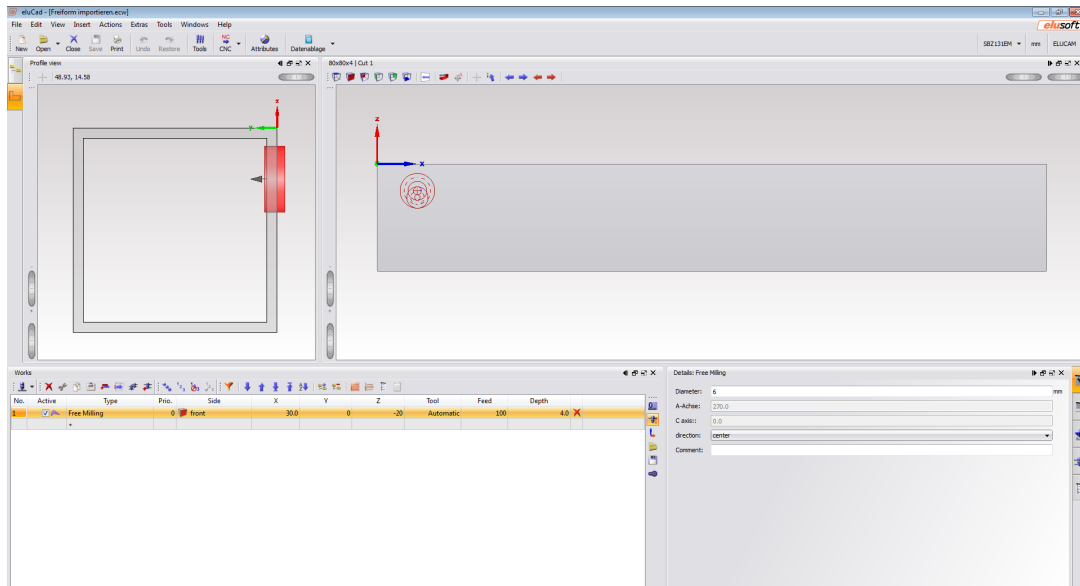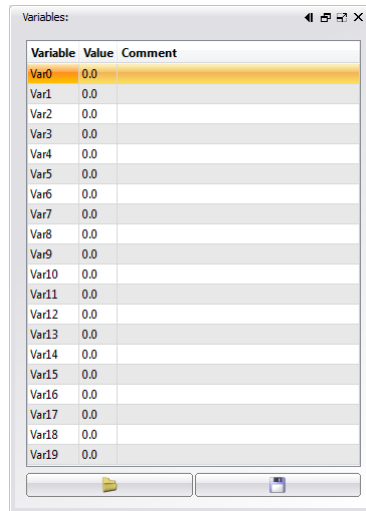- Work feed rate 100%
- Depth 10 mm

The following steps are required to import an existing free form:

1. In the **MACHINING TASKS** table, select the first blank line in the list of profile machining tasks.If a line containing a machining operation is selected, it can be overwritten!

2. Open the selection window in the **TYPE** cell with a double click and select the type **FREE FORM IMPORT**.

3. The machining task is activated automatically but can be deactivated at any time.

4. By selecting the next cell, the **SELECT CONTOUR** menu automatically opens.

5. The list of existing free forms is displayed.

6. Select the desired free form file from the list.
   Info: Use the **FILTER FUNCTION** to reduce the list of files to the filter selection.

7. Use the **OK** button to add the selected free form file to the machining task list.

8. Open the selection window in the **SIDE** cell with a double click and select **FRONT** as the machining side.

9. Enter 30 mm for the X-home position of the free form in the **X** cell.

10. The **Y** cell already contains 0 mm for the Y-position.

11. Enter -20 mm for the Z-home position of the free form in the **Z** cell.

12. The tool is defined automatically by default in the **TOOL** cell. A tool can be defined in the **TOOL SELECTION** menu by pressing the **AUTOMATIC** button.

13. Enter the value of 80% in the **FEED RATE** cell.

14. Enter the value of 10 mm in the **DEPTH** cell. Use the button to trigger the automatic wall detection, which is transferred to the depth table.

    Use the button to manually create or change the depth table.

15. Enter the data and values for the group in the input fields of the **DETAIL** tab.

- **Diameter:** Enter 6 mm for the diameter of the machining task.

- **Direction:** Select **CENTRE** for the machining track of the tool.

16. Check all inputs in the profile view.

2.2.14 Creating a variables table

This example demonstrates the free definition of variables.

⚠	INFORMATION
ℹ	The quantity and names of the variables are predefined in the variable table and cannot be changed. The variables defined are only active in the selected job!

To create your own variable, you must perform the following steps:

1. Pressing the button opens the **VARIABLES WINDOW** menu in the profile machining tasks.

2. Mark the desired row in the variables table.

3. Open the input window in the **VALUE** cell with a double click and enter the desired value.

4. Open the input window in the **COMMENT** cell with a double click and enter the desired comment.

5. If further variables are to be defined, repeat steps 2 through 4.

6. Use the [💾] button to save the variable table to the selected folder.

7. Press the [✕] button to apply the new values in the **VARIABLE TABLE** menu.

Index