TopSolid Cam

Tutorial

TopSolid'Cam 7

MASTER YOUR MANUFACTURING PROCESS

7, Rue du Bois Sauvage
F-91055 Évry, FRANCE
Web: www.topsolid.com
Email: contact@topsolid.com

Version 7.14 Rev.01

Note: If you are experiencing any problems with this tutorial, please feel free to send your feedback and comments at edition@topsolid.com.

Contents

Creating a Machining Part

Introduction

The purpose of this chapter is to create a machining document which can be used to perform various machining operations that can be accessed from the Cam section of **TopSolid**.

We will explain in detail five different methods that will produce a similar result.

We will start with the easiest method (**TopSolid** will automatically respond to some questions for the users), then we will discuss the most detailed method (where we must answer all of the questions).

Working with the PDM

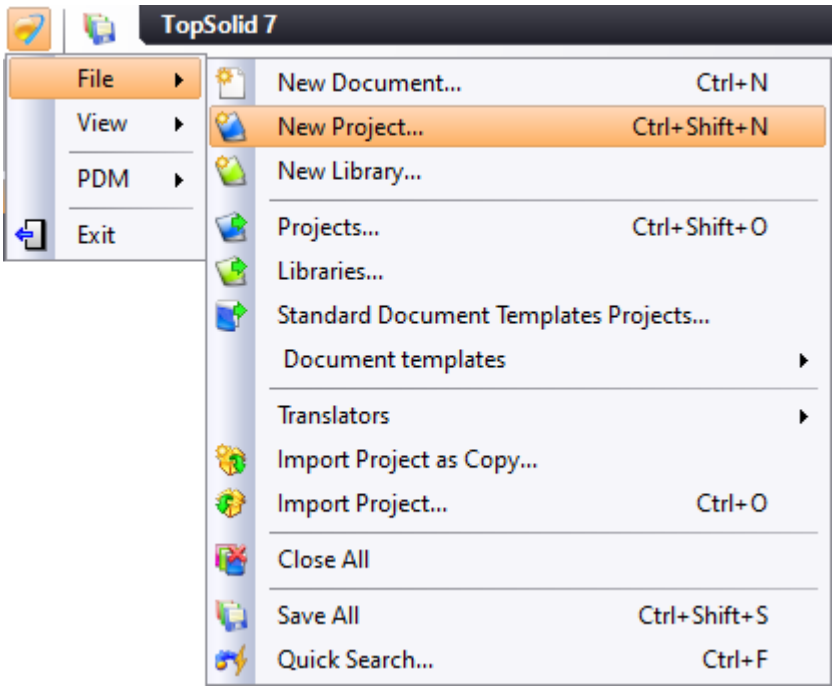
As explained in the online help and the Design support, the PDM allows you to store and monitor changes made to any files created in **TopSolid 7**.

In addition, in order to isolate all the tutorial files, we will create a new project containing all the required files.

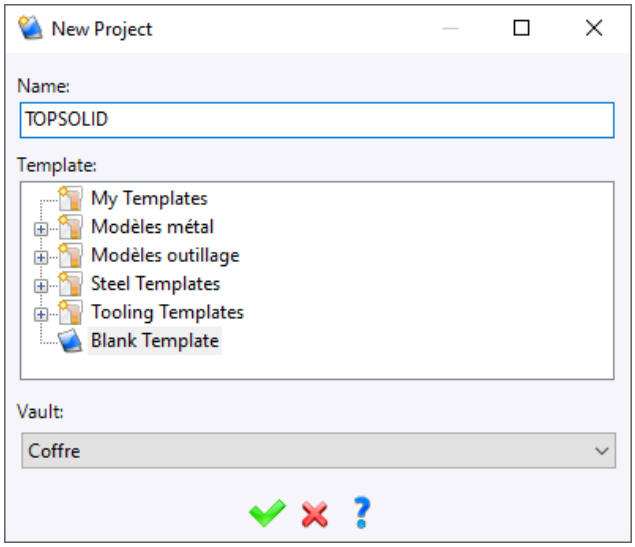
To create this new project, proceed as follows:

- Click on the icon of the **TopSolid 7** menu.
- Select **File** from the drop-down menu.
- Select the **New Project** command.

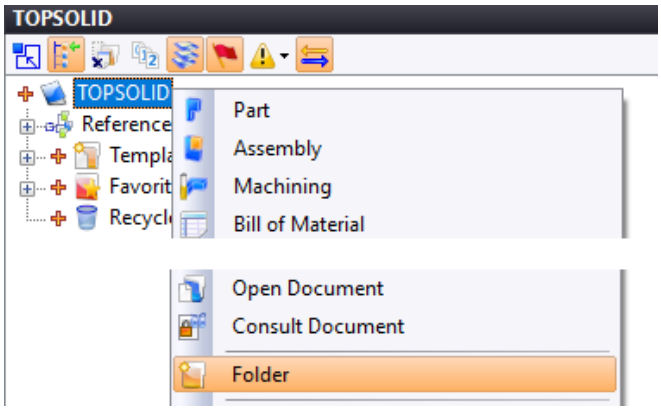
You can also click on the icon from the **Home** tab or use the **Ctrl** + **Shift** + **N** keyboard shortcut to create a new project.



- The following dialog box is displayed. Rename the project *TOPSOLID*, select **Blank Template** and confirm with the green checkmark .



- From the Project tree, right-click on the project name and select the **Folder** command from the contextual menu.

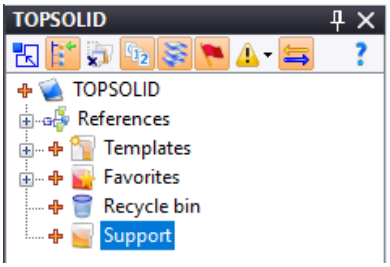


By default, the created folder will be named *Folder*.

- Rename the folder *Support*.

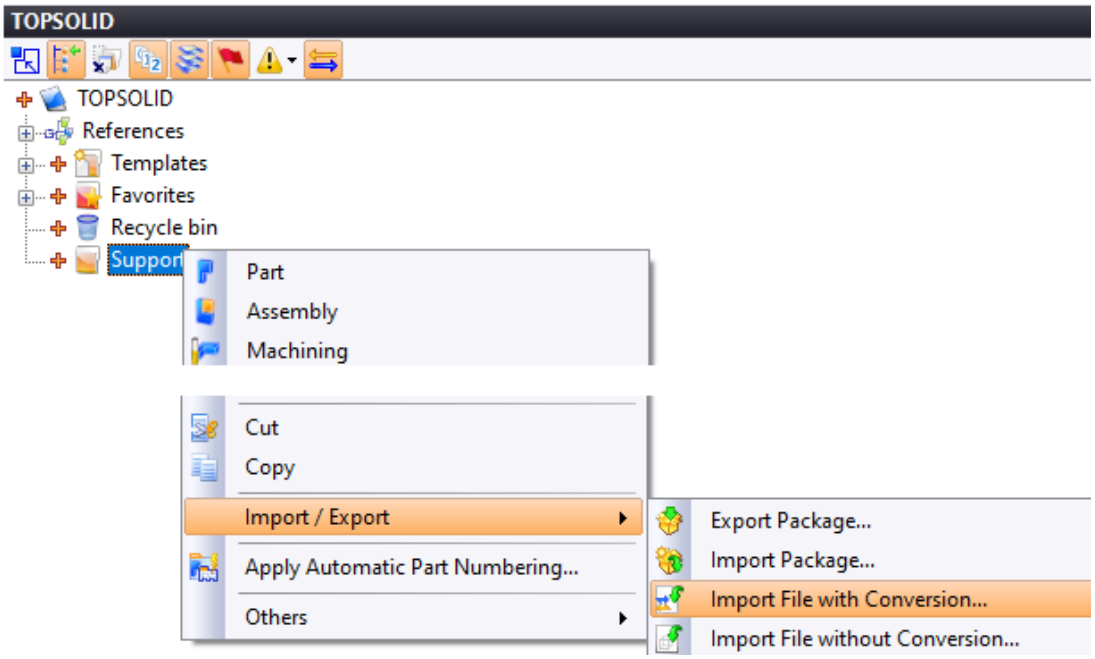
Note: To rename a file, a document or a folder, you simply have to press the F2 key or slowly double-click on the left mouse button (such as for Microsoft Windows Explorer). Examples of folder names are provided in order to easily locate them when creating the machining.

When changing the name of a file, it will be preceded by the icon and followed by an asterisk * indicating that the document has been changed or created. Saving the document will remove the asterisk.

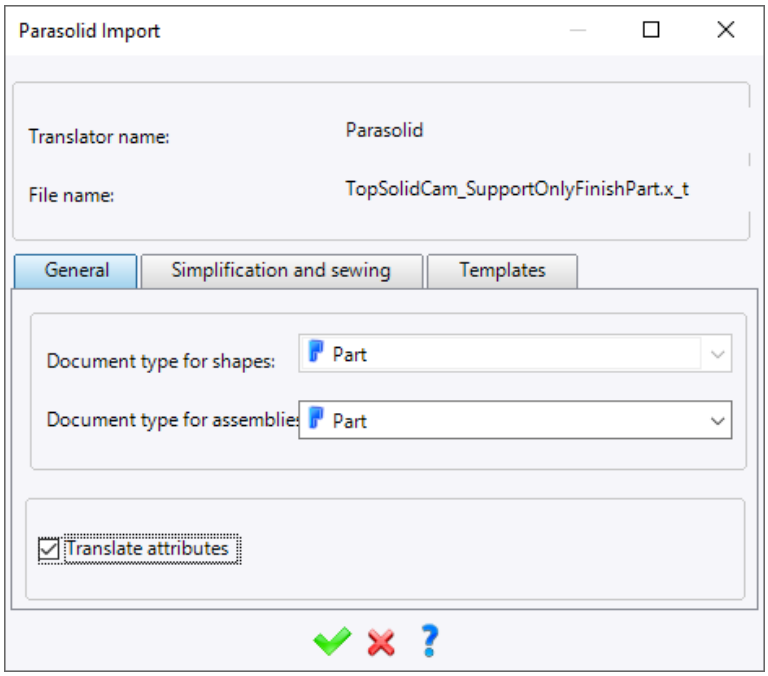


In the *Support* folder, you will now be able to create and machine a new part.
Given that the purpose of this chapter is not to geometrically define the part (see *TopSolid'Design Tutorial*), we will import a part. In this case, since this part was not designed in **TopSolid**, we will lose all the benefits of the integrated solution (interactive modification, use of machining information such as H7 hole, etc.).

- To import the part, right-click on the *Support* folder and select the **Import/Export** > **Import File with Conversion** command.



- Open the *TopSolidCam_SupportOnlyFinishPart.x_t* file (during installation, this file is installed in the "Programs Files\TOPSOLID\TopSolid 7.14\Samples" folder).
- In the dialog box that appears, select the following parameters.

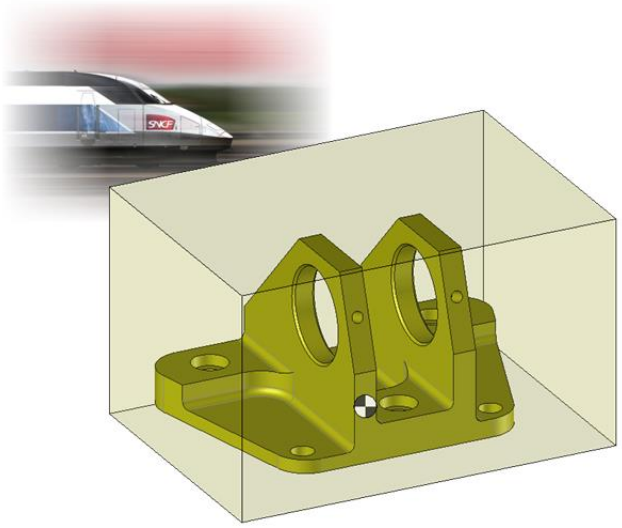


- Click on the icon to **confirm** the import.
- Using the F2 key, rename the imported file *Support (part only)*.

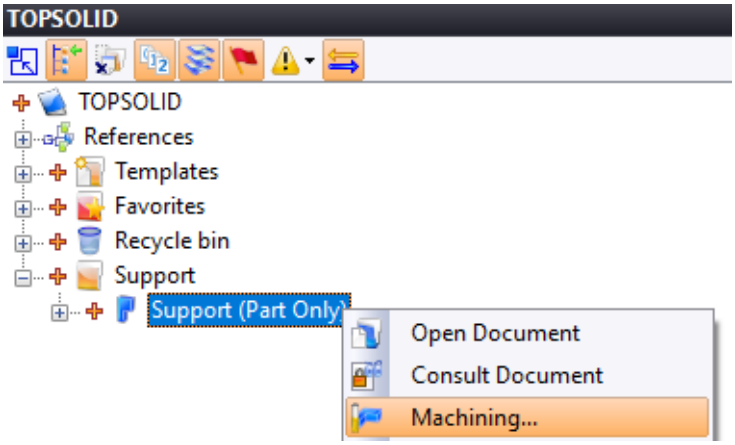
Note: Right-clicking on a folder allows you to create a new entity in it, import one or more documents into it, or export, edit, or copy it.

Step 1 – Quick machining

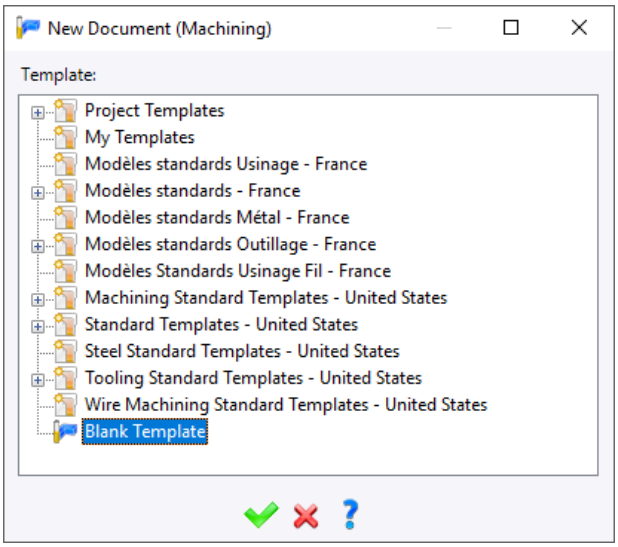
In this section, we will explain how to quickly start machining the part that we just imported.



- From the Project tree, right-click on the part document and select a **Machining** document.



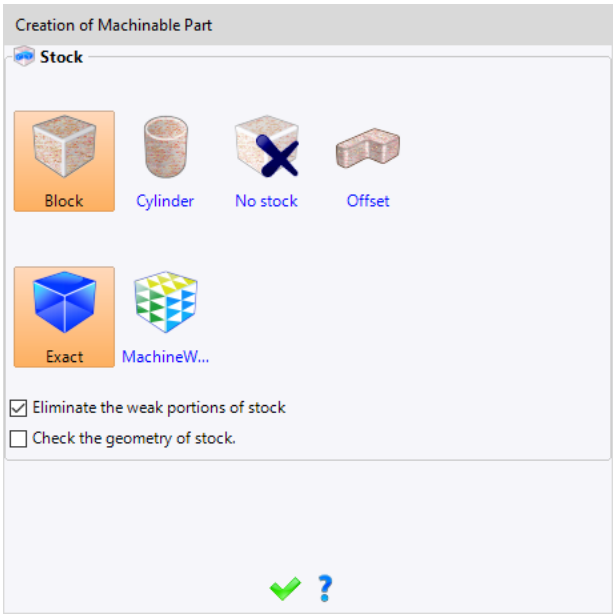
The following dialog box appears.



You can select a predefined template. In our case, we will select **Blank Template** and click on the icon to **confirm**.

The following dialog box appears, prompting you to choose some settings related to the machinable part.

- Choose a prismatic-shaped stock by selecting **Block**.
- Check the **Eliminate the weak portions of stock** box.



- Click on to **confirm**.
- Click on the Positioning 1 button to exit the positioning context.

The machining document is automatically created and placed in the same directory as its related part.

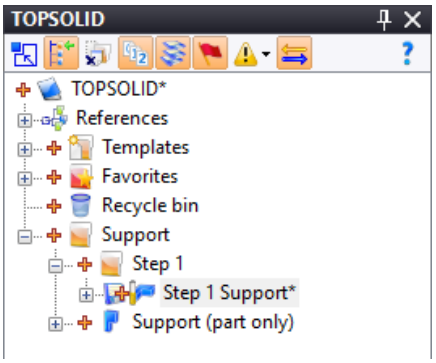
- **Save** the document by clicking on the icon or by using the **Ctrl** + **S** keyboard shortcut.

For clarity purposes and to help you become familiar with the Cam section of **TopSolid**, we will create a new folder. By doing this for each of the five methods that we are explaining, we will be able to come back more quickly to any of these methods later, if necessary.

- Right-click on the *Support* folder and select the **Folder** command. Rename the folder *Step 1*.

We will then move the machining document into the *Step 1* folder and rename it.

- Hold down the left mouse button and drag and drop the machining document into the *Step 1* folder.
- Rename the machining document *Step 1 Support.*

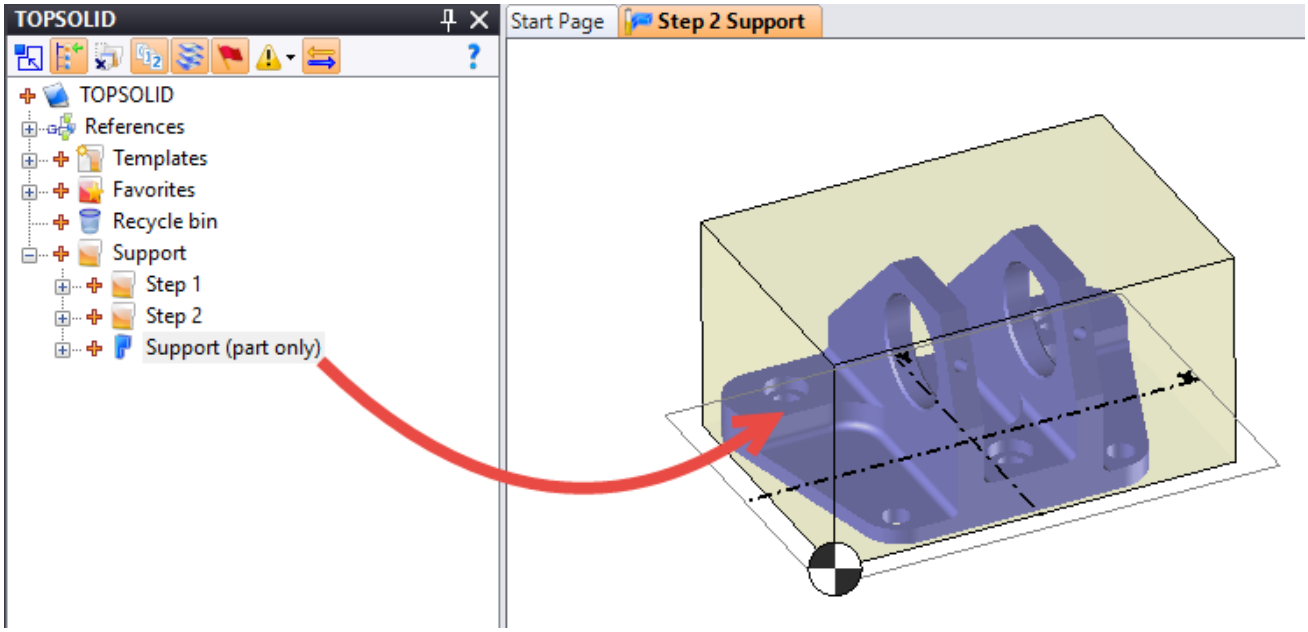


- **Save** and close all open documents.

Note: You can see that the part is enclosed in a prismatic stock and its margins have been defined by default. We will see later in the *Step 6 – Modifying a stock* section how to change this stock.

You can now start machining the part.

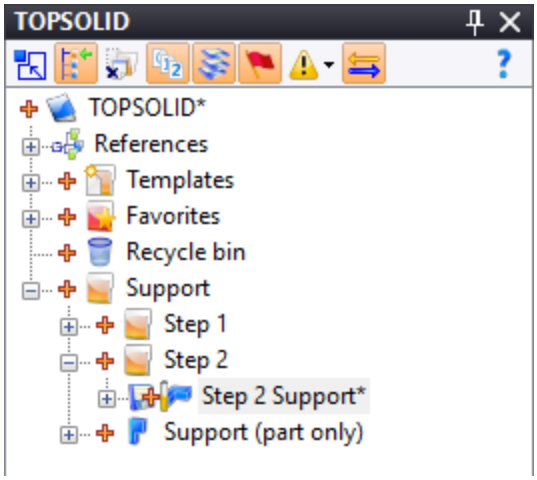
Step 2 – Inserting a part in a blank machining



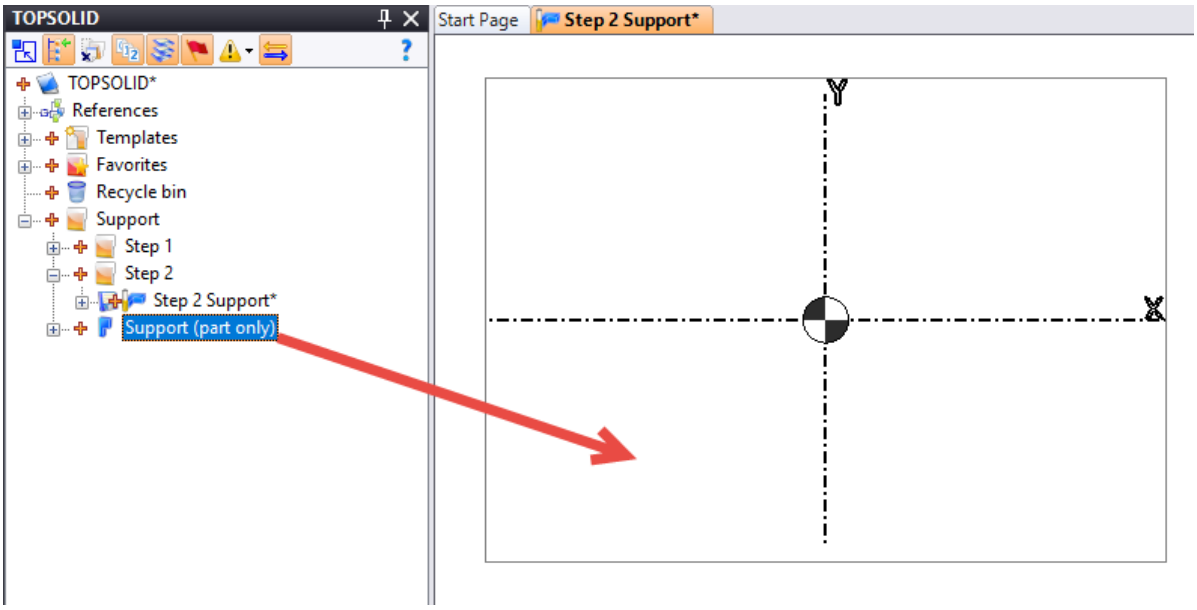
This step produces the same result as Step 1, but introduces another feature of the software. In fact, we will create a blank machining document and then insert the machinable part.

Using this document would make it possible, for example, to use a template directly afterwards with a machine and fixed tools.

- In the *Support* folder, create a new **folder** and rename it *Step 2*.
- As before, right-click on this folder and select the **Machining** command.
- Select a **blank template** and click on to **confirm**.
- Rename the machining document *Step 2 Support.*

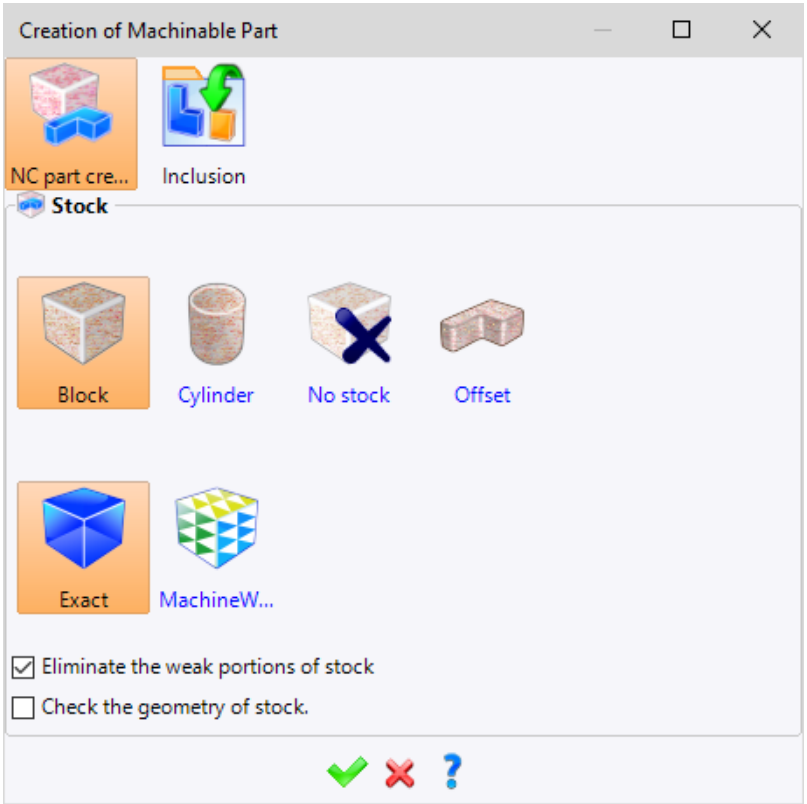


A machining document has been created. This document is blank for now. There are no machinable parts. We will now indicate to **TopSolid** that we want to machine the imported part.
To do this, we will place the part document in the machining document using the drag and drop technique. This technique is easily done by simply holding down the left mouse button while dragging the *Support (part only)* part document into the open machining document.



The following dialog box appears, prompting you to choose some settings related to the machinable part.

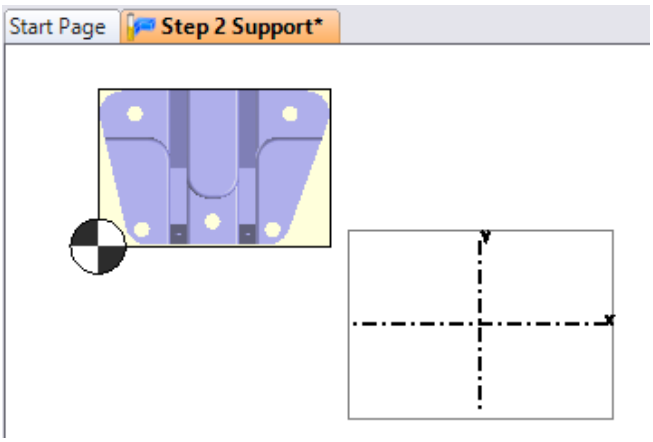
- Select the **NC part creation** option.
- Select a **block** prismatic-shaped stock.
- Check the **Eliminate the weak portions of stock** box.



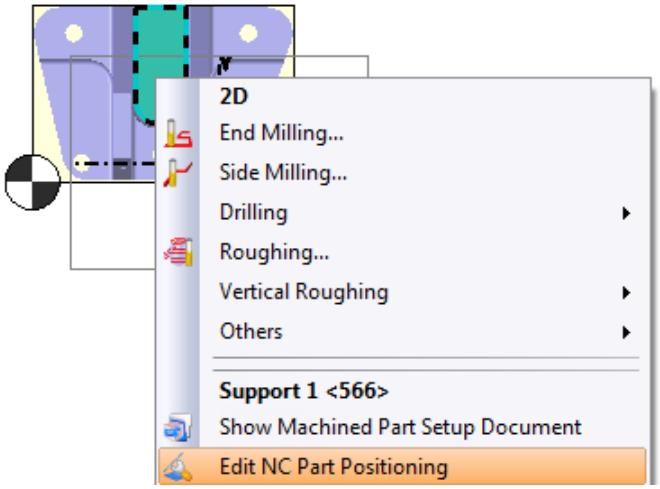
- Click on ✔ to **confirm**.
- Click on the Positioning 2 button to exit the positioning context.

The part is ready to be machined but it is positioned where you released the mouse button. This position might not be the one you require.

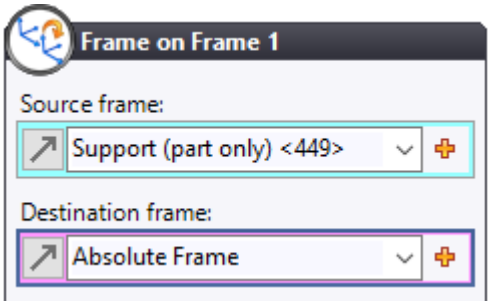
Example:



- If you want to modify the position of the part, right-click on the part and select the **Edit NC Part Positioning** command.

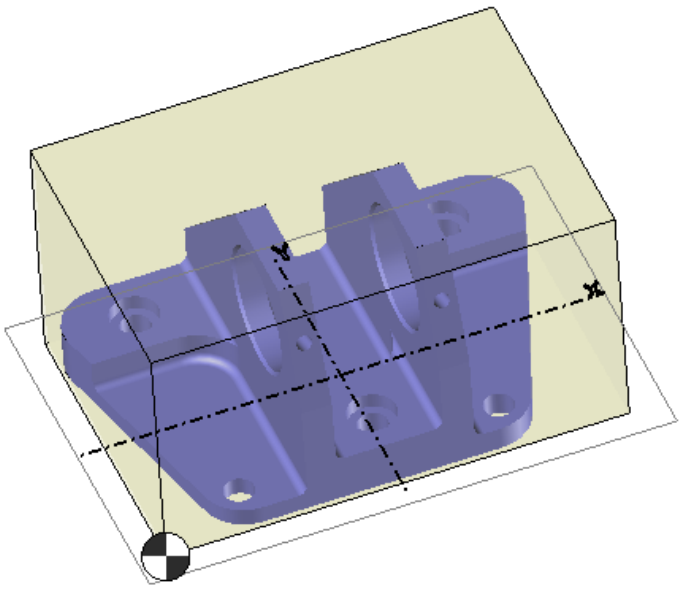


- In the positioning context, select the **Frame on Frame** command from the **Assembly** tab. In the **Source frame** field, select the **Support (part only)** frame from the drop-down list. In the **Destination frame** field, select **Absolute Frame**.



- Click on the Positioning 2 button to **confirm** the positioning.

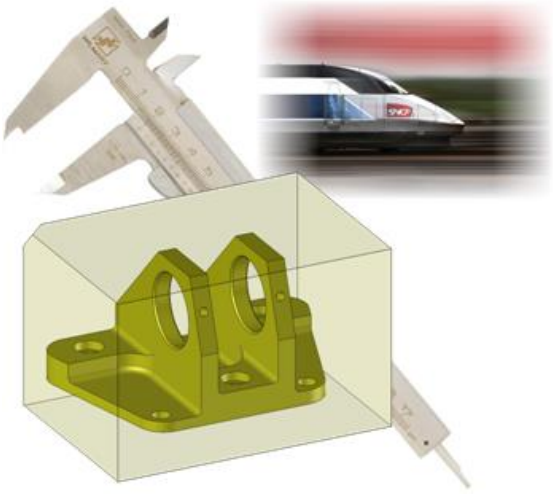
The part is now positioned in the center of the table.



- **Save** and close all open documents.

You can now start machining the part.

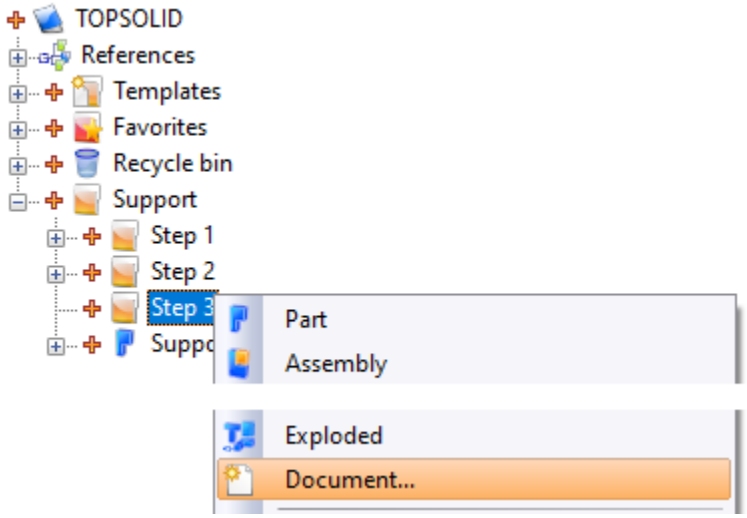
Step 3 – Quick machining with setup



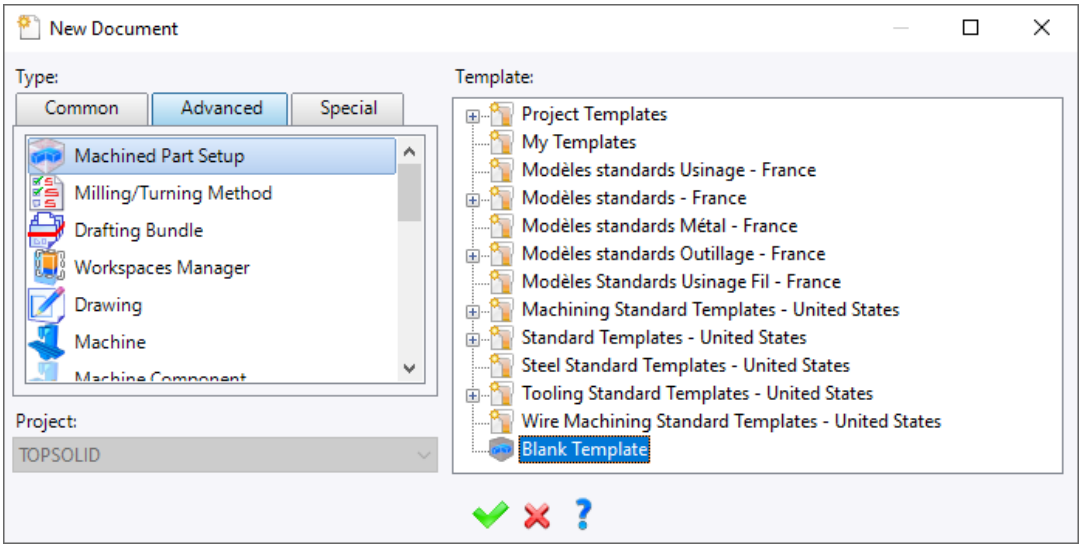
In this section, we will manually set up the machining, which was automatically performed in the previous steps.

There are two possible ways to create this machining setup, as seen in Step 1 (creating a setup document from an existing file) and Step 2 (creating an empty setup document and inserting the machinable part). Here, we will explain how to achieve the machining by following the method in Step 2.

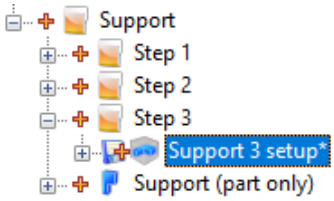
- In the *Support* folder, create a new **folder** and rename it *Step 3*.
- Right-click on this folder and select the **Document** command.



- From the **Advanced** tab, select **Machined Part Setup**, select **Blank Template**, then click on to **confirm**.



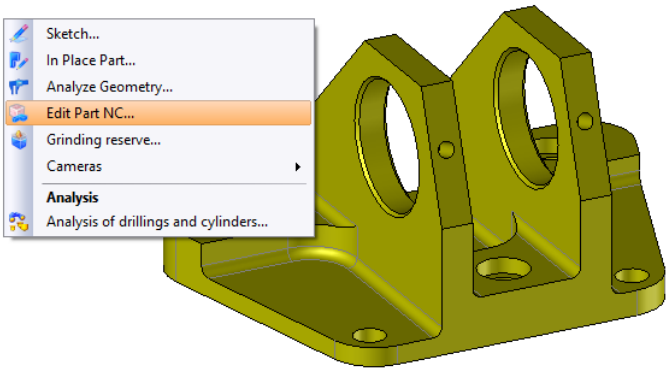
- Rename the machining setup document *Support 3 setup*.



A blank machining setup document has been created. To insert the part to set up, we will use the drag-and-drop technique discussed in Step 2.

Once the part has been inserted into the machining setup document, you can see that it does not include a stock.

- To edit the machining setup, right-click in the graphics area.
- Select the **Edit Part NC** command.

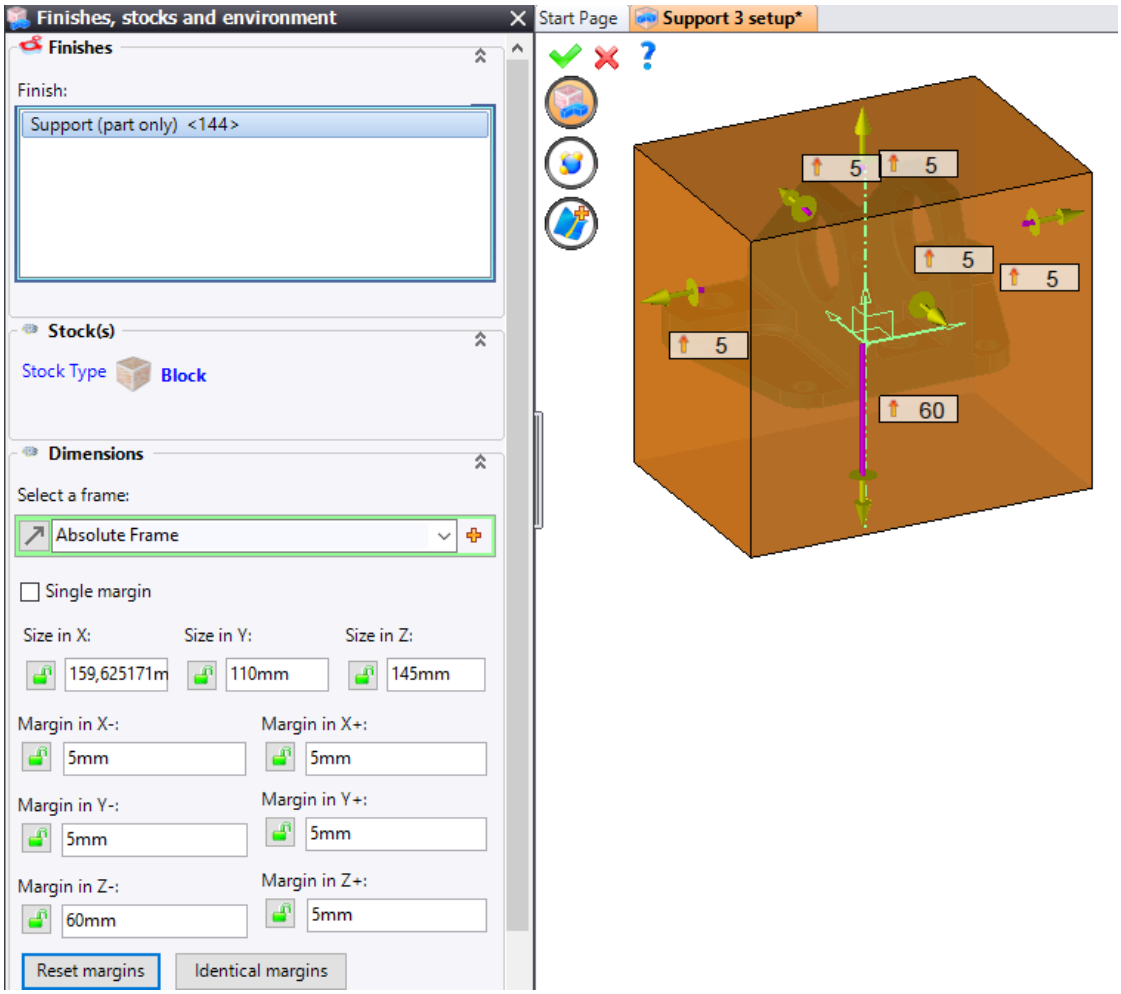


You must now define the part and its stock.

- Click on the **Finishes, stocks and environment** icon.

Note: **TopSolid** has automatically selected the finish and defined an enclosing stock that can be modified.
You can define unique margins (identical margins in all directions) or customize the margins. Modifications can be assessed in real time in the 3D view.

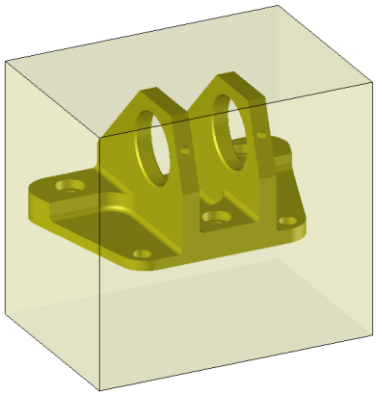
We will choose 5mm margins in all the absolute frame's directions, except for the Z- margin that will be 60mm.



- Click on ✔ to **confirm**.

The part and its adjusted stock are now defined.

Your screen should now look like the following image:



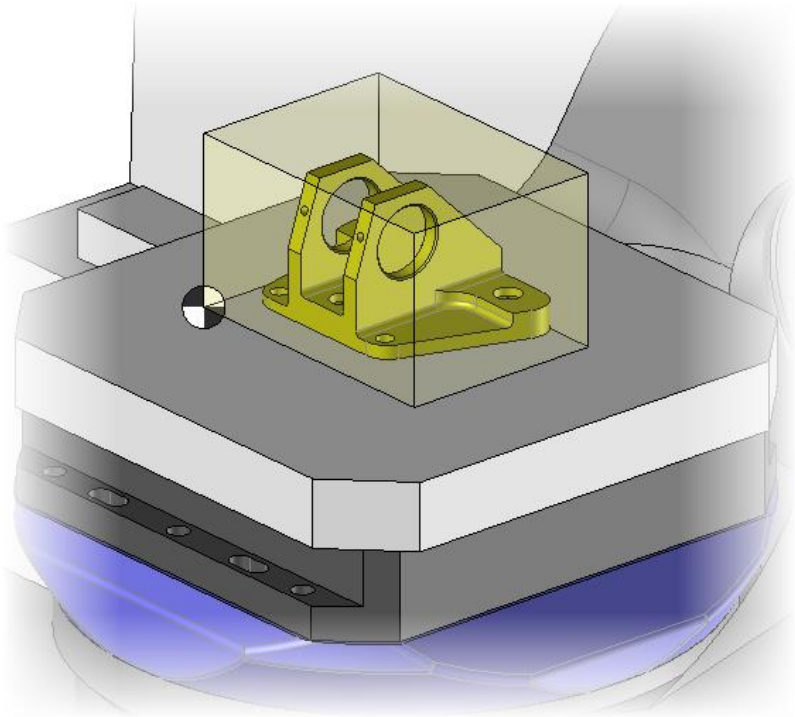
- **Save** the document.

There are two possible ways to create this machining, as seen in Step 1 (creating a machining document from the existing setup) and Step 2 (creating an empty machining document and inserting the existing setup).

- **Save** and close all open documents.

You can now start machining the part.

Step 4 – Machining with a selected machine and inserting a part or setup

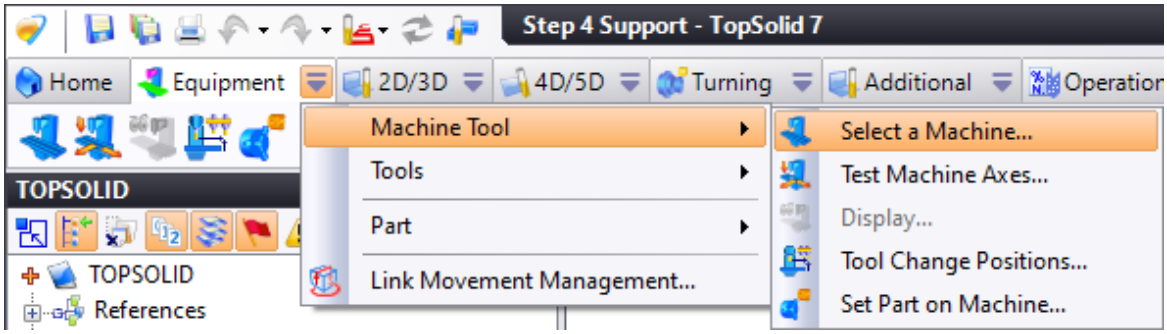


This method is similar to the method in Step 2, with the exception that it includes a choice of machine. We will therefore only discuss how to include a NC machine.

- In the *Support* folder, create a new **folder** and rename it *Step 4*.
- Create a new **Machining** document using a **blank template**, then click on to **confirm.**
- Rename this document *Step 4 Support*.

We now have a blank machining document in which we will insert a part.

- From the **Equipment** tab, click on the button to display the drop-down menu.
- From the **Machine Tool** menu, select the **Select a Machine** command.

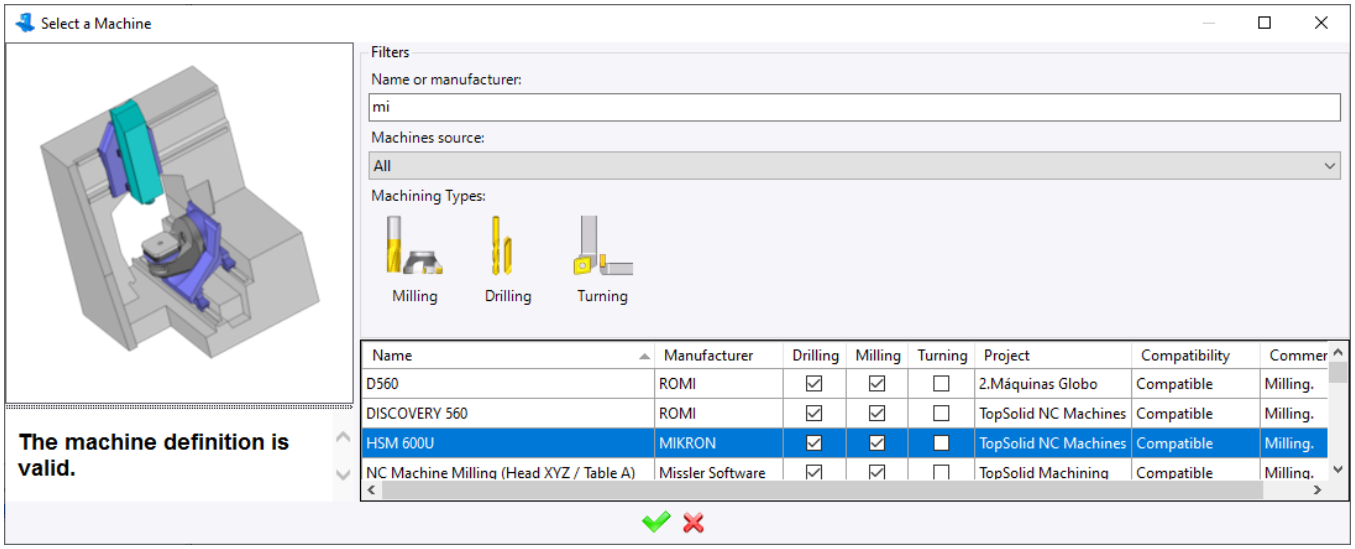


To find the machine more easily, you can filter the list of machines.

- In the **Name or manufacturer** field, enter *mi.*

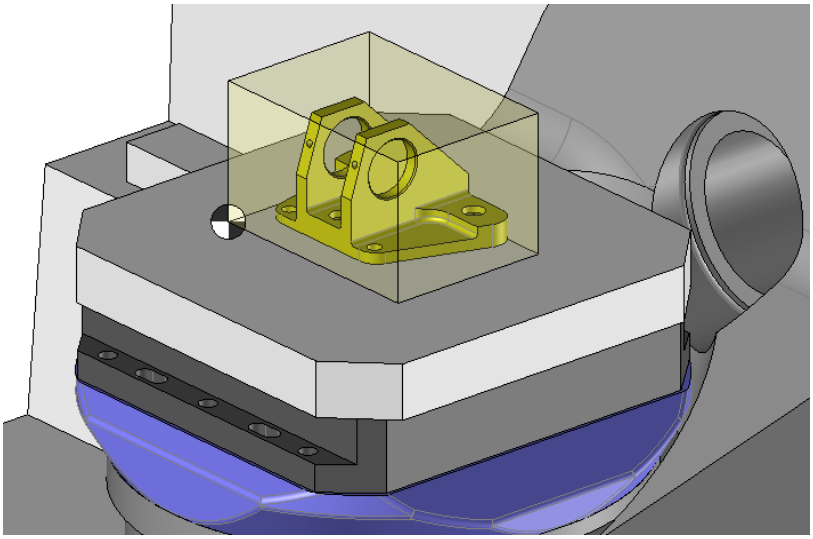
A machine will appear only if the name or the manufacturer contains the text you entered.The search is not case-sensitive and you may type in upper or lower case.

- Select the **MIKRON HSM 600U** 5-axis machine, then click on to **confirm**.



Once the machine is selected, you can insert a part or machining setup. To do this, refer to the *Step 2 – Inserting a part in a blank machining* section.

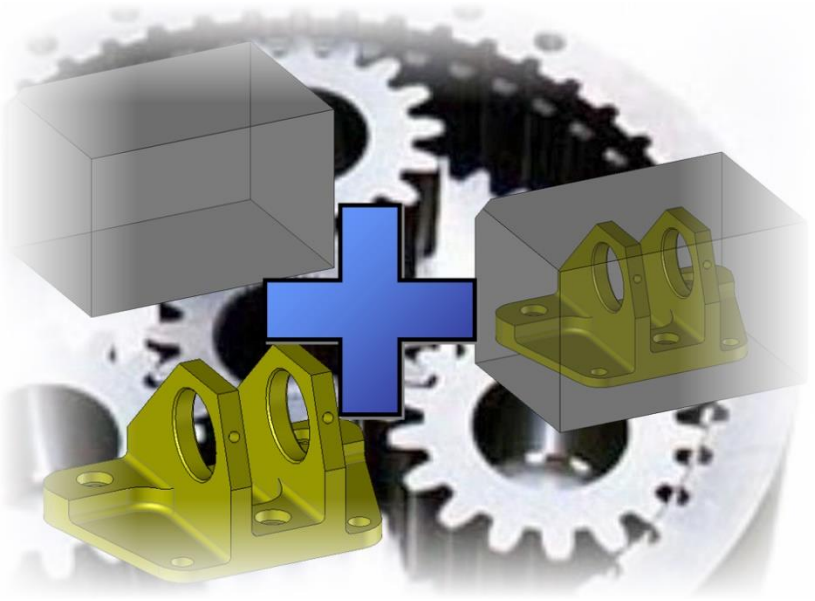
- Position the part so that it is located in the middle of the machine table as shown below.



- **Save** and close all open documents.

You can now start machining the part.

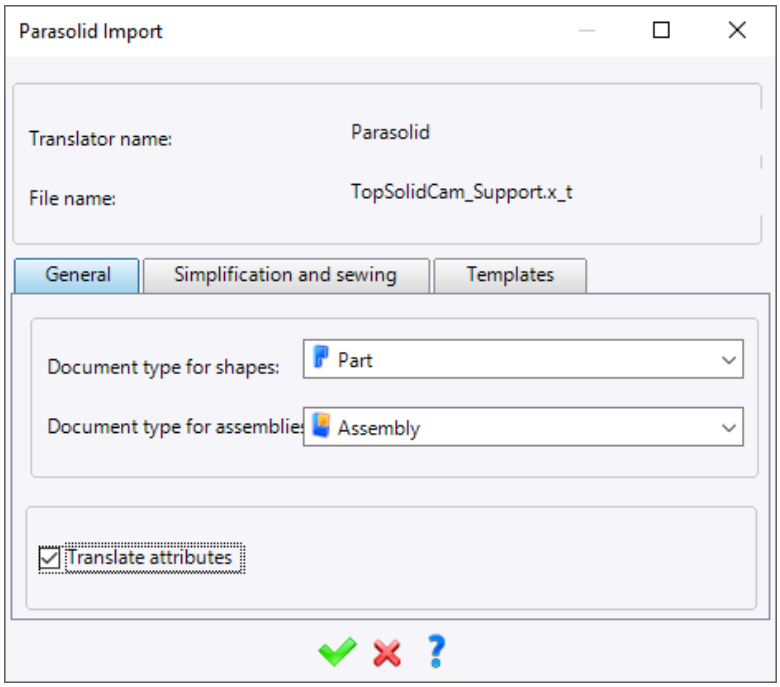
Step 5 – Assembled part and stock, setup, then machining



In this section, we will use a stock and a part that have already been assembled. This method is particularly useful if the stock's geometry is variable or if the stock has already been machined. However, you have to be familiar with the **TopSolid'Design** assembly tool in order to use it.

We will first import the assembly in Parasolid format.

- In the *Support* folder, create a new **folder** and rename it *Step 5*.
- Right-click on this folder and select the **Import/Export** > **Import File with Conversion** command.
- Open the *TopSolidCam_Support.x_t* file (during installation, this file is installed in the "Programs Files\TOPSOLID\TopSolid 7.14\Samples" folder).
- In the import dialog box, select the following parameters.

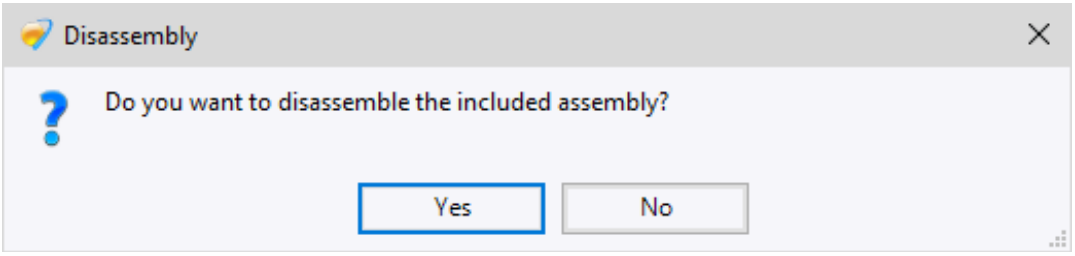


- Click on to **confirm** the import.

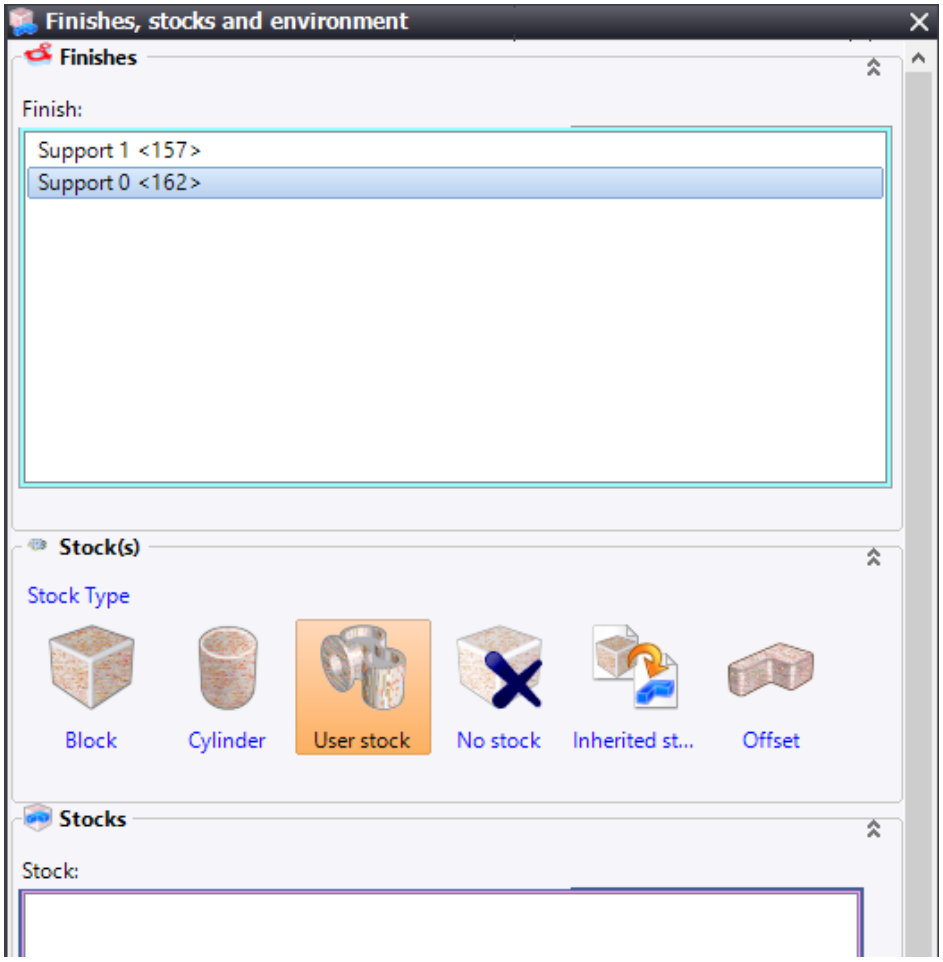
- Rename the assembly document *Support*.

Note: When importing an assembly document, **TopSolid** imports the assembly and each component of the assembly separately. Consequently, you can use only the finish or only the stock for various modifications, for example.

- Right-click on the stock in the graphics area and select the **Attributes** command.
- Adjust the **transparency** to *70%* and click on to **confirm**.
- **Save** the assembly document.
- From the Project tree, right-click on the *Support* assembly document.
- Create a new **Machined Part Setup** document using a **blank template**. The following message appears asking you whether you want to disassemble the assembly.

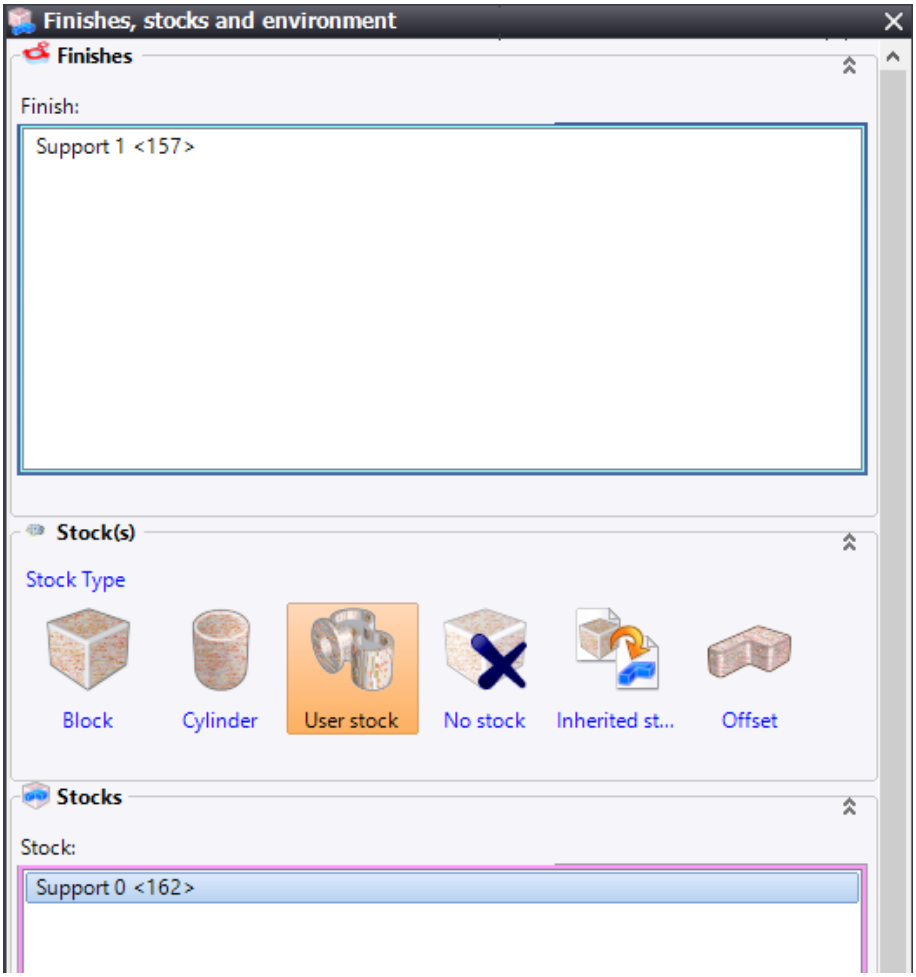


- Click on **Yes**. In this way, each solid of the assembly will be displayed independently in the **Finish** list.
- Click on the **Finishes, stocks and environment** icon.
- In the dialog box that appears, click on **Stock Type** (the blue text) and select **User stock**.



- The **Finish** list contains the two solids, with **Support 1** being the part to be machined and **Support 0** being the stock. You therefore need to move **Support 0** to the **Stock** list. To do this, you simply have to move the mouse over **Support 0**, hold down the left mouse button and drag **Support 0** to the **Stock** list.

Your screen should look like the following image:



- Click on to **confirm**.

Note: You can see that the selection areas are framed in a certain color and when a shape is selected for example, this shape will take on the color of the frame.

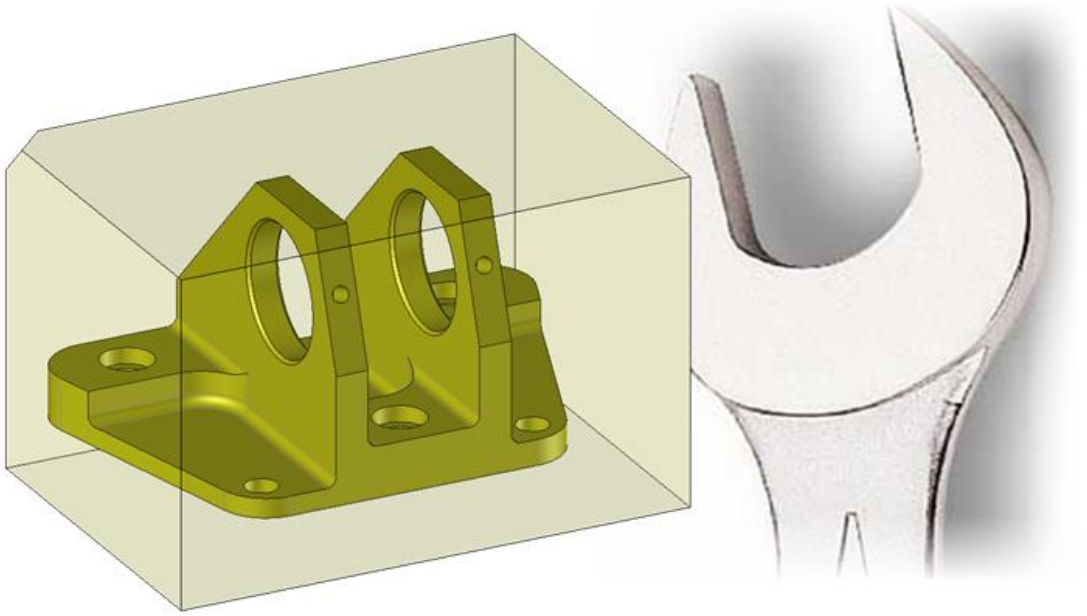
The machining setup document is complete. You can save and close it, and then start machining.

There are two possible ways to create this machining, as seen in Step 1 (creating a machining document from the existing setup) and Step 2 (creating an empty machining document and inserting the existing setup).

Note: We will use this machining setup in the machining tutorial.

You can now start machining the part.

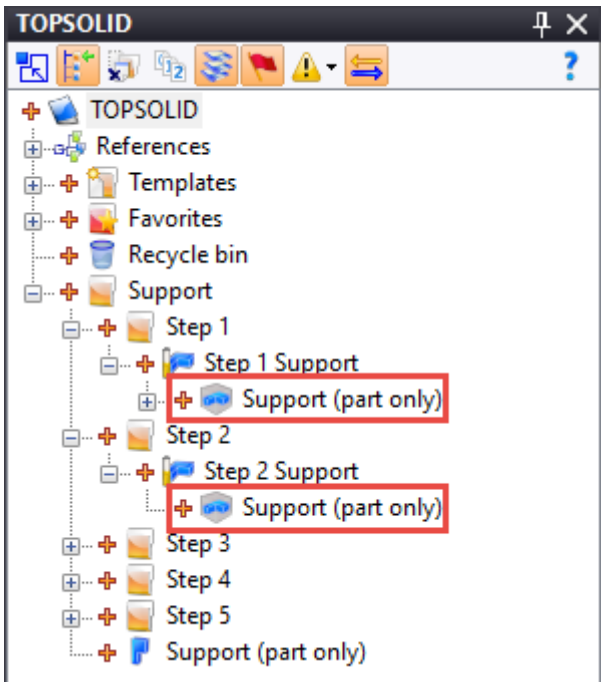
Step 6 – Modifying a stock



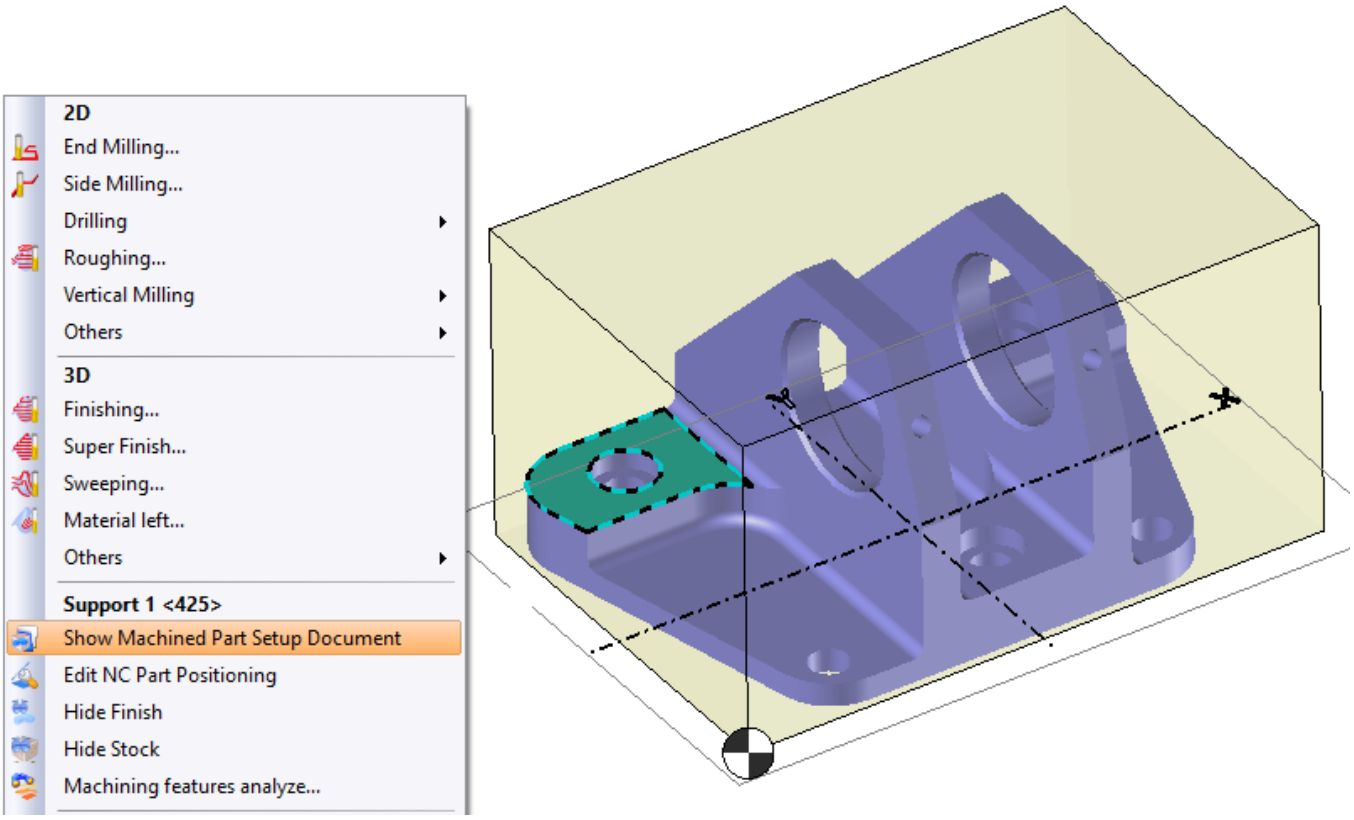
When you scroll down the entire *TOPSOLID* project tree, you can see that machining setup documents have been automatically created for all the machining documents. This is so you can go back and modify the stock or the NC part origin, for example.

There are two ways to access the machining setup:

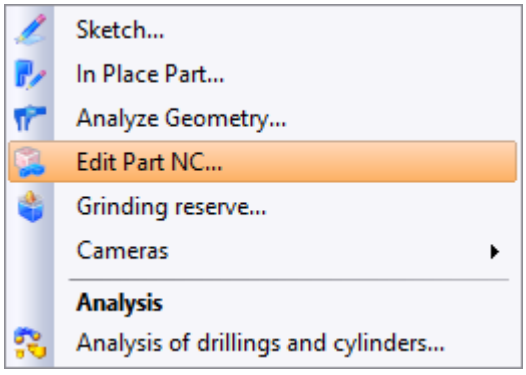
- Scroll down the tree structure and double-click on the automatically created machining setup documents.



- Or select a face of the part, then right-click in the graphics area and select the **Show Machined Part Setup Document** command.



- Once you have accessed the machining setup document, if you want to modify it, right-click in the graphics area and select the **Edit Part NC** command.

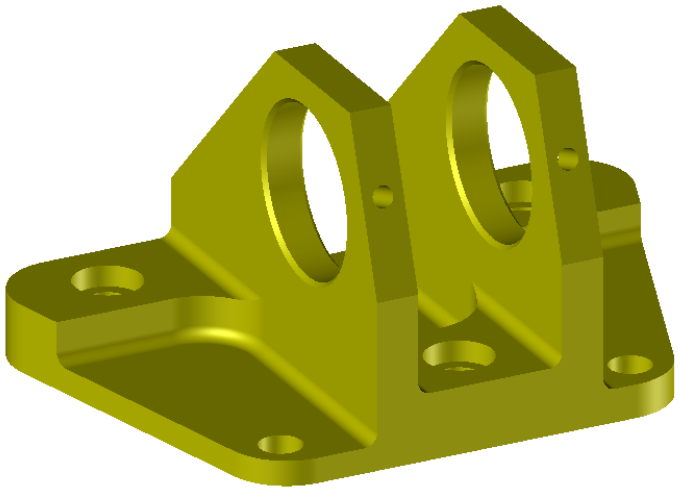


You now have access to the machining setup menus shown in Step 3.

2D Milling

Introduction

The purpose of this chapter is to create a machining program for the following part.



As you can see, to be machined, the part must be positioned on different orientations. To avoid several disassemblies, which can be detrimental to the desired manufacturing quality, we will use a 5-axis machine.

To help you become familiar with **TopSolid 7**, the machining of the part above is divided into several steps. For each step, a machining document is available in which operations have been completed. You can refer to these documents if you run into any difficulties. These machining documents are provided in a TopPkg file named *TopSolidCam_2DPartStepByStep.TopPkg* (this file is installed in the "Programs Files\TOPSOLID\TopSolid 7.14\Samples" folder). In order to follow the tutorial, we recommend that you import this file into a new project.

Example: For Step 6, which consists of performing the external machining of the part, a machining document named *Support_Step_5* is available, in which steps 1 to 5 are already carried out.

The part was designed in **TopSolid**. The designer used the "features" technology, which allows major automation of the machinings.

The overall programming approach is as follows:

1. Defining the stock part
2. Positioning and defining the part (as per **TopSolid'Cam**) and its program origin
3. Defining the operations with integrated stock management
4. Simulation
5. Editing the ISO file

Steps 1 and 2 are explained in the prerequisite tutorial (creating a machining setup document and machining document). We will start at Step 3, which consists of defining the machining operations.

As you have seen in this tutorial, several methods are available for creating a machining document. You can therefore start the machining by selecting the method of your choice. For this chapter, the selected method is method 5, which consists of machining an assembled stock and finish.

Machining

We are now in the machining context. **TopSolid 7** is simple to use and suggests two approaches for creating a machining.

- **Subject – Verb**:
  Select the entities to be machined, and then select a machining command.

With this solution, you can select one or more faces with the mouse and indicate the machining on which to apply the faces.

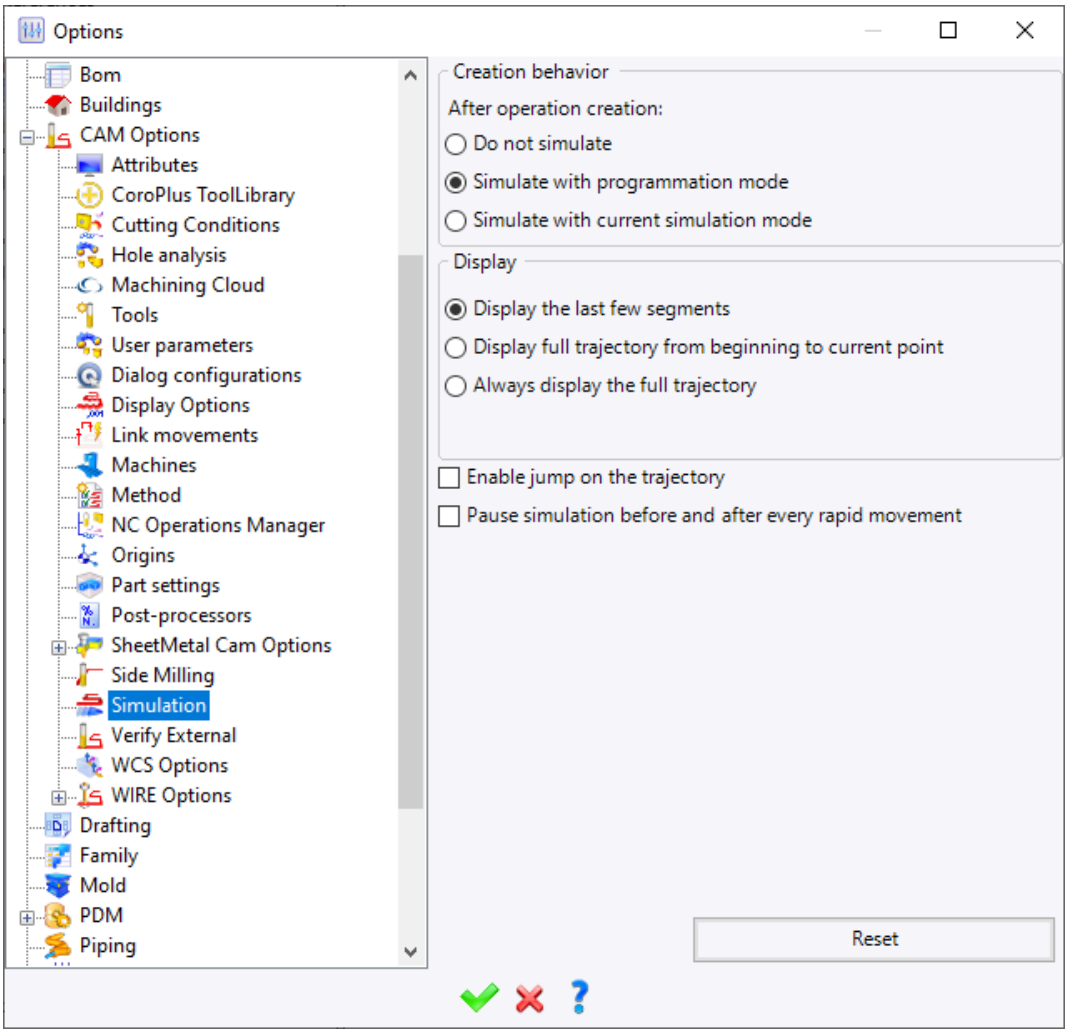
- **Verb – Subject**:
  Select a machining command and indicate the entities on which to apply this command.

These two methods will be used in this chapter.

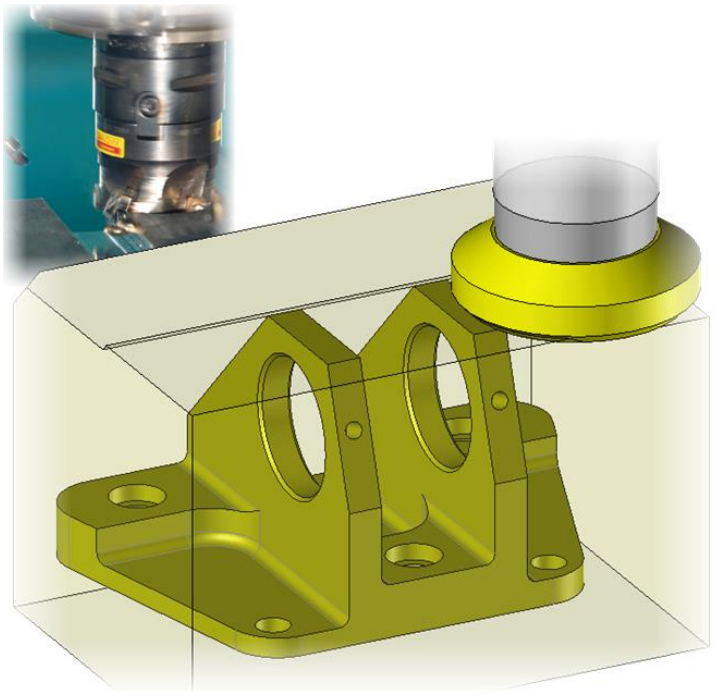
Options

In general, you can adjust the machining options via the **Tools** > **Options** > **Machining** menu.

For example, in the **Tools** > **Options** > **CAM Options** > **Simulation** command, you can choose between different modes of simulation once the machining operation has been confirmed.



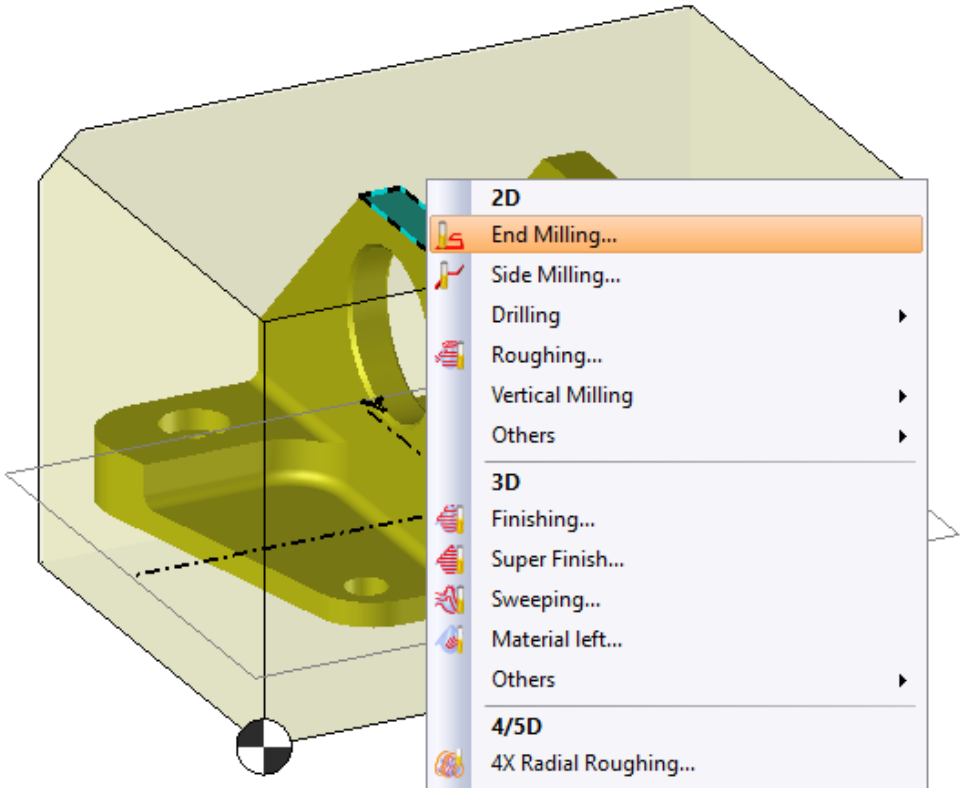
Step 1 – Machining the upper section of the part



- Create a **new project** named *2D Machining* by using a **blank template**.
- **Import** the *TopSolidCam_2DPartStepByStep.TopPkg* package in your project.
- Open the *Support_Step_1* machining document.

We will first machine the upper face.

- Right-click on one of the two upper planar faces and select the **End Milling** command.

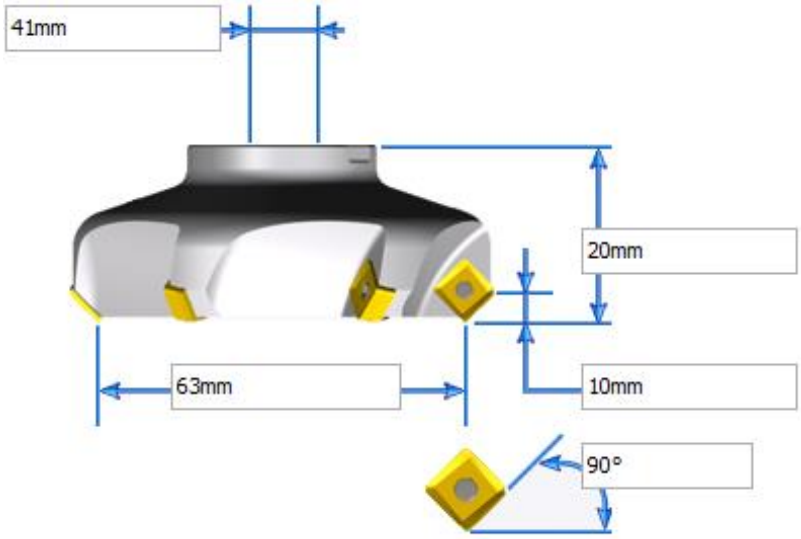


A label such as the one below is displayed. Some values such as the maximal axial depth, which is equal to zero because the tool has not been selected yet, may be different. They can be modified in the **Settings** option.

#1	
Time	00:00:00:000
Altitude...	100mm
Stock to leave on floor	0,2mm
Stock to leave on wall	0,2mm
Stock to leave on wall island	0,3mm
Maximal axial depth	0mm
Final axial depth pass	0mm
Lead in point	
Tool Path Preview	Yes

Note: The icon indicates that a facing operation will be performed. If you want to change the icon, you only have to double-click on it and select the appropriate operation (pocketing, open pocketing or expanded facing).

- Click on the **Tool choice** icon. The blue asterisk indicates a required field.
- Double-click on the Face Mill **Face Mill** thumbnail and create a *63mm* diameter face mill with a *10mm* cutting height.

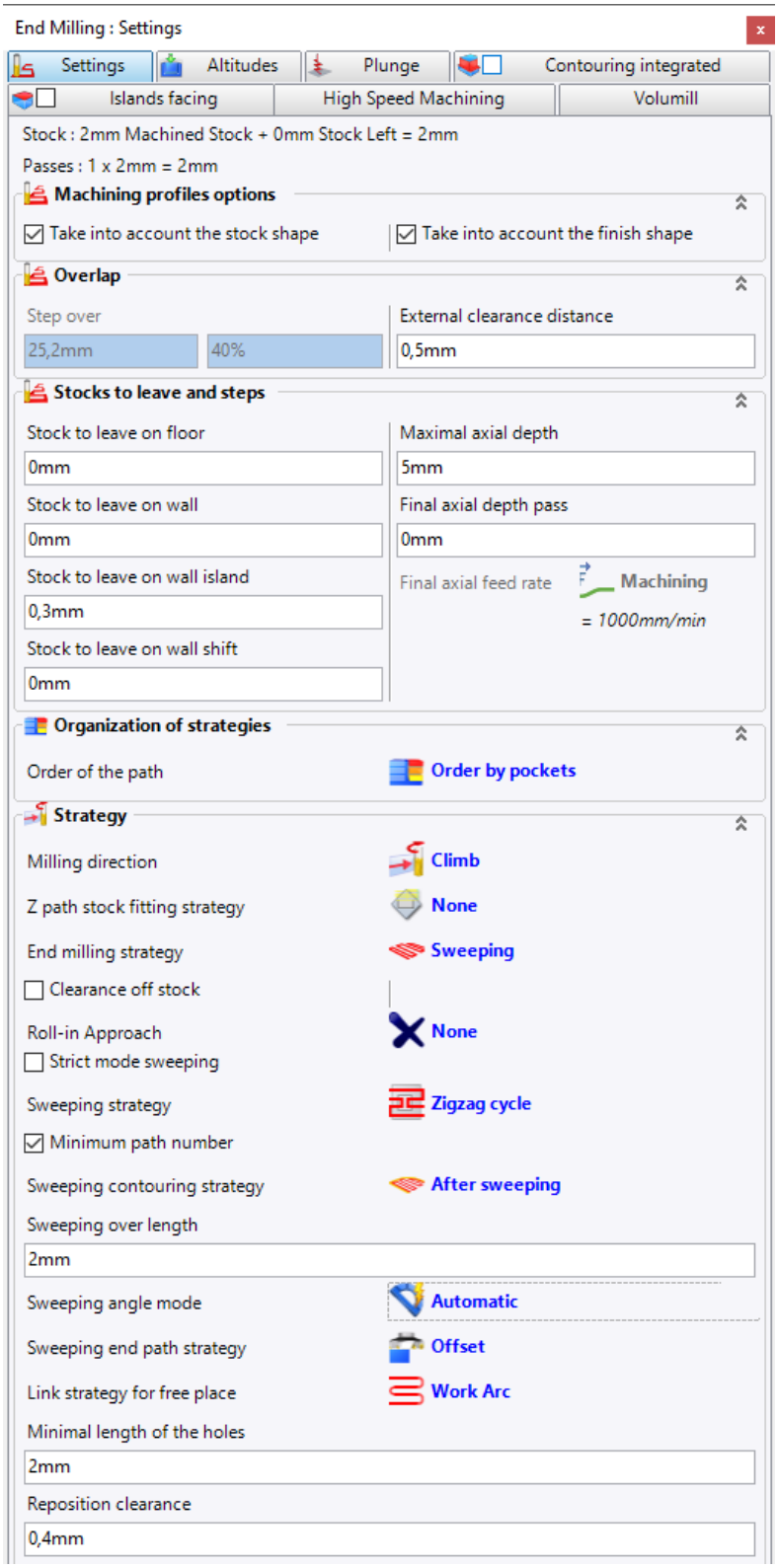


- Click on to **confirm** the tool.

The facing settings remain to be indicated.

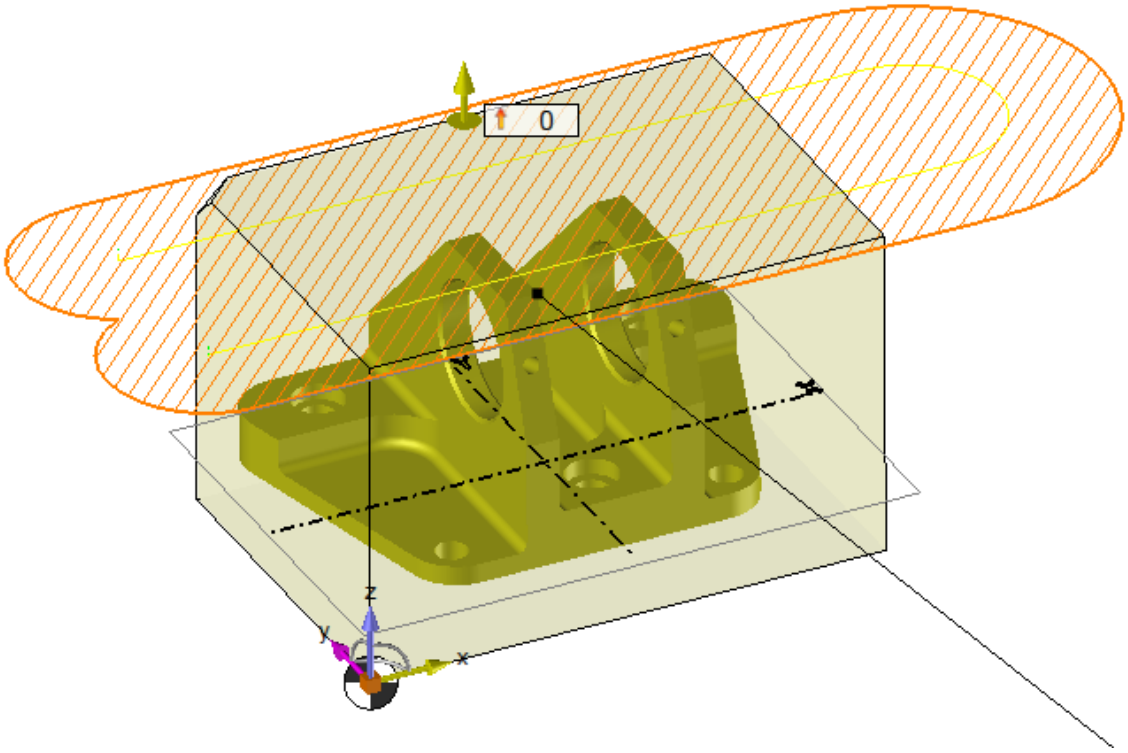
- To do this, click on the **Settings** icon.
- Adjust the following settings:
  - **Stock to leave on floor** and **Stock to leave on wall**: *0mm*
  - **Maximal axial depth**: *5mm*
  - **Final axial depth pass**: *0mm*
  - **End milling strategy**: Select **Sweeping**
  - **Sweeping strategy**: Select **Zigzag cycle**.
  - Check the **Minimum path number** box
  - **Sweeping over length**: *2mm*

You should end up with the following result.



Note: These settings are for guidance only and in no way resulting from a particular methods department. They are simply for verifying that the settings coincide with the obtained tool paths.

With these settings and this tool, the tool path preview should be as follows:



Note: **TopSolid** is configured to avoid unnecessary calculation to improve the calculation time. When modifying a machining setting (for example a stock to leave), the machining area and the tool path preview are not graphically updated (it may take time). An icon at the top right of the screen indicates whether or not the graphics area is updated. This icon has three states:

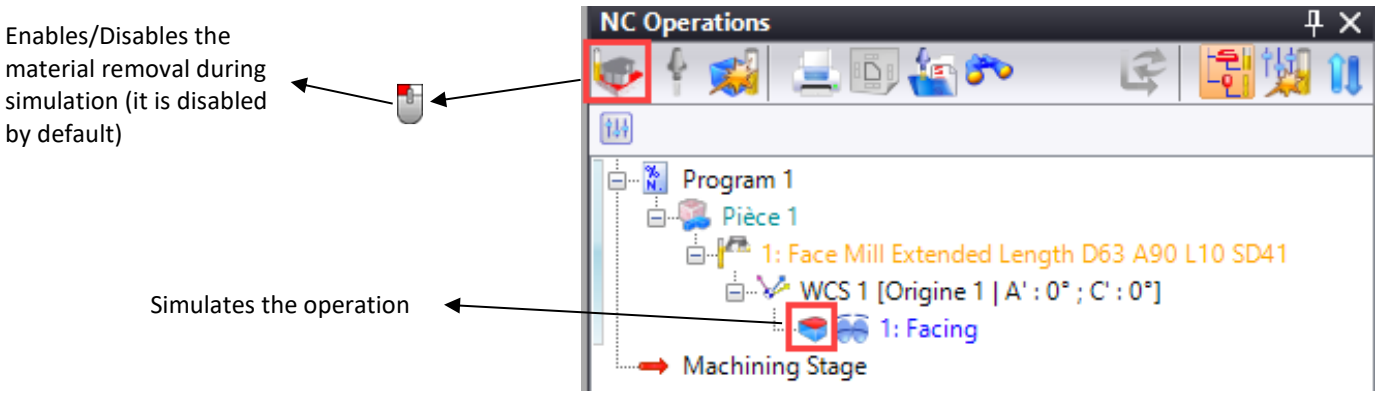
- : The graphical update is manual and the graphics area is currently updated.
- : The graphical update is manual and the graphics area is currently out of date. Clicking on this icon will update the graphics area (it may take time).
- : The graphical update is automatic. Each time a setting is modified, the machined area and the tool path are recalculated (it may take time).

- Click on to **confirm** the machining operation.

Simulating the machining

In this section, we will see how to resimulate the previously performed machining.

First, a very useful option allows you to remove material during machining. Then, to simulate the machining of one operation, you only have to click on the icon that indicates a facing operation, for example.



To simulate one or more operations, you can right-click on the selected operations and select the **Simulate** command.

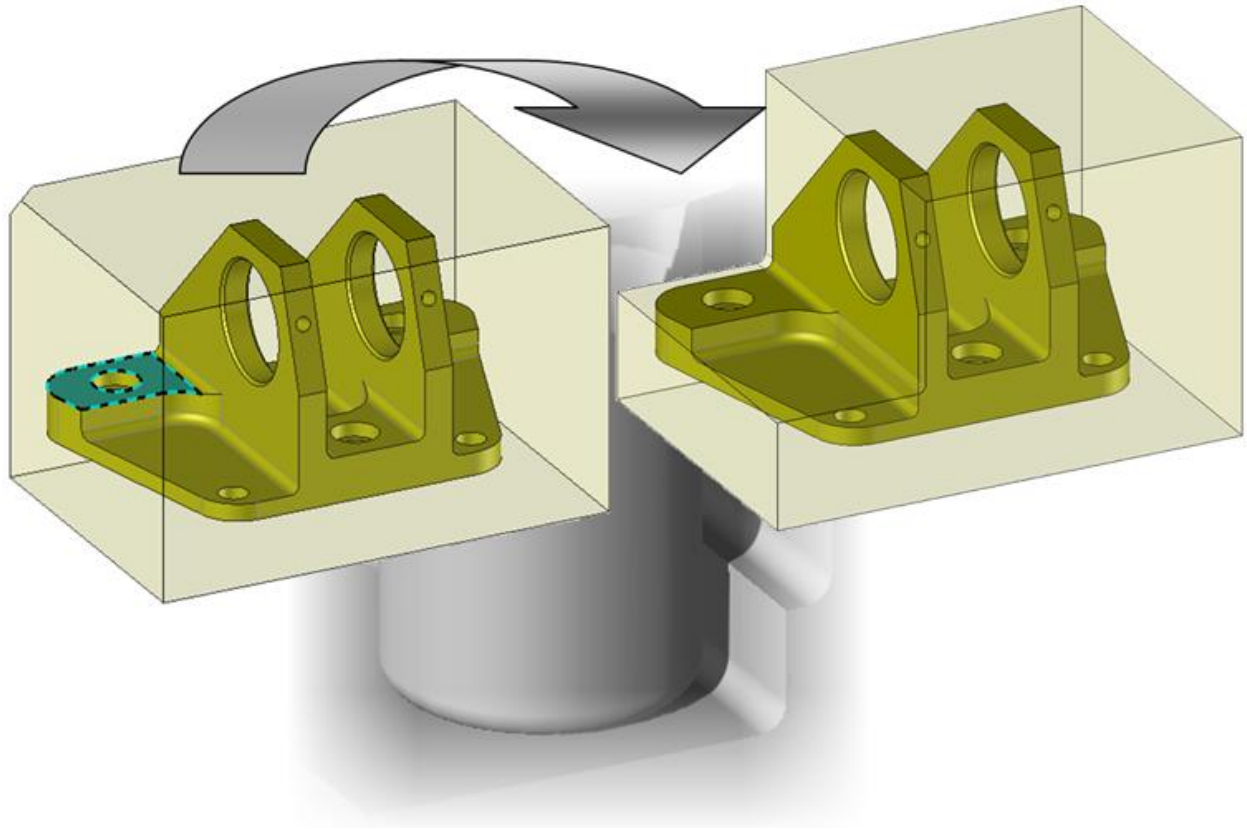
To simulate all operations, right-click on Part 1 and select the **Simulate** command.

- **Save** the document.

Step 2 – Machining the left section of the part

- Open the *Support_Step_2* machining document.

Note: For this step, we will machine the outer section of the part using a radiused mill. The operation will be an open pocketing operation. The technique is the same as in the previous step.



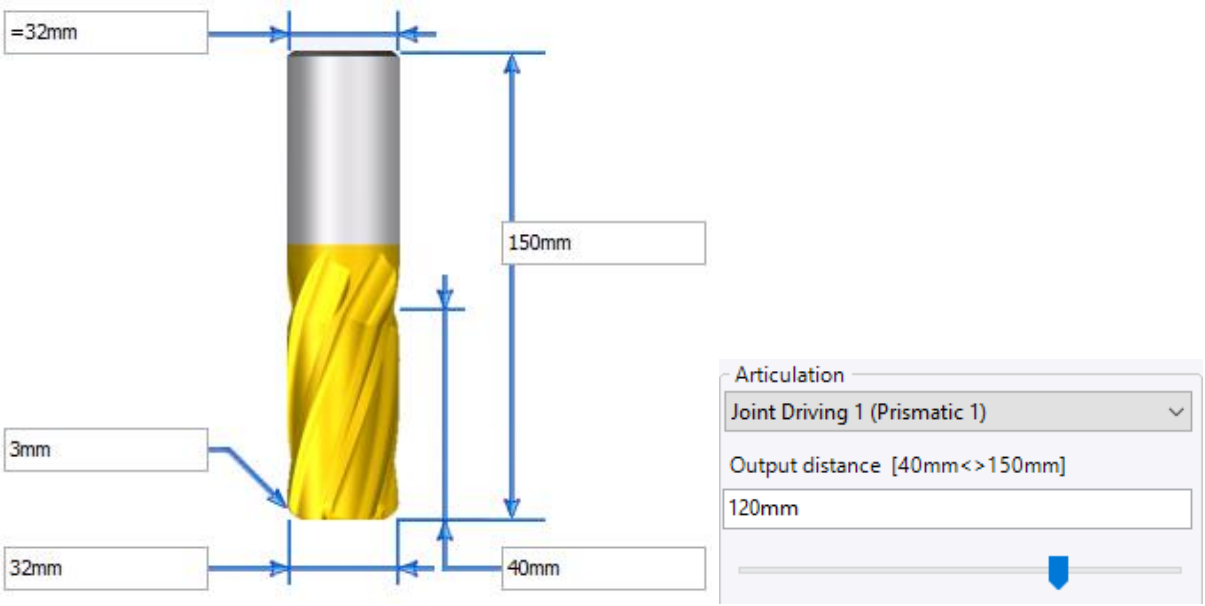
- As before, right-click on the face to be machined shown in blue above.
- Select the **End Milling** 2D machining command.

The following label is displayed.

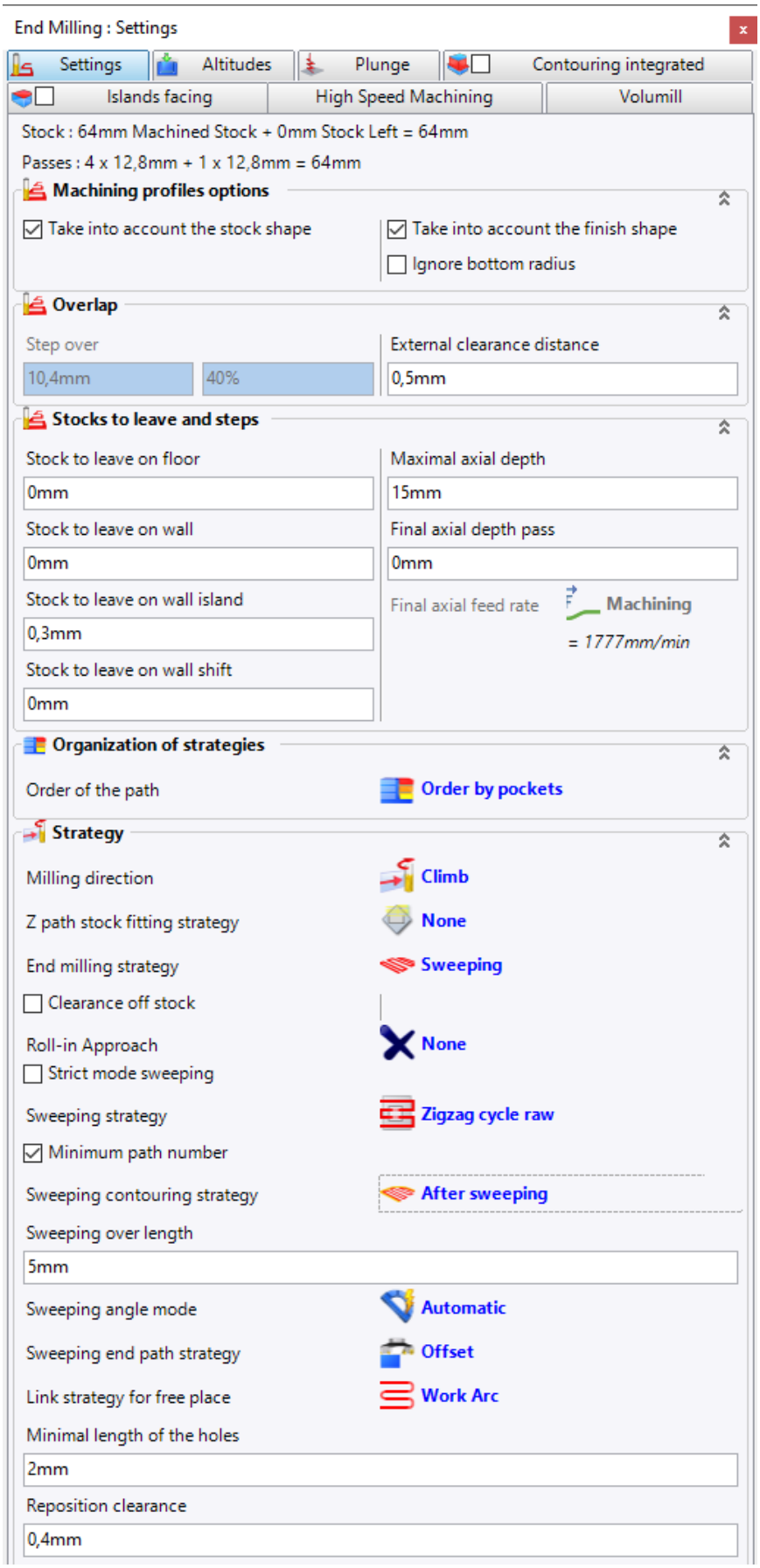
#1	
Time	00:36:36:143
Altitude...	36mm
Stock to leave on floor	0mm
Stock to leave on wall	0,3mm
Stock to leave on wall island	0,3mm
Maximal axial depth	0,9mm
Final axial depth pass	0,3mm
Lead in point	
Tool Path Preview	Yes

The icon indicates that an open pocketing operation will be performed.

- In the **Tool choice** option, double-click on the **Radiused Mill** thumbnail and select a *32mm* diameter radiused mill, with a radius of *3mm* and a total length of *150mm*. From the **Tool Assembly** tab, adjust the **output distance** to *120mm*.

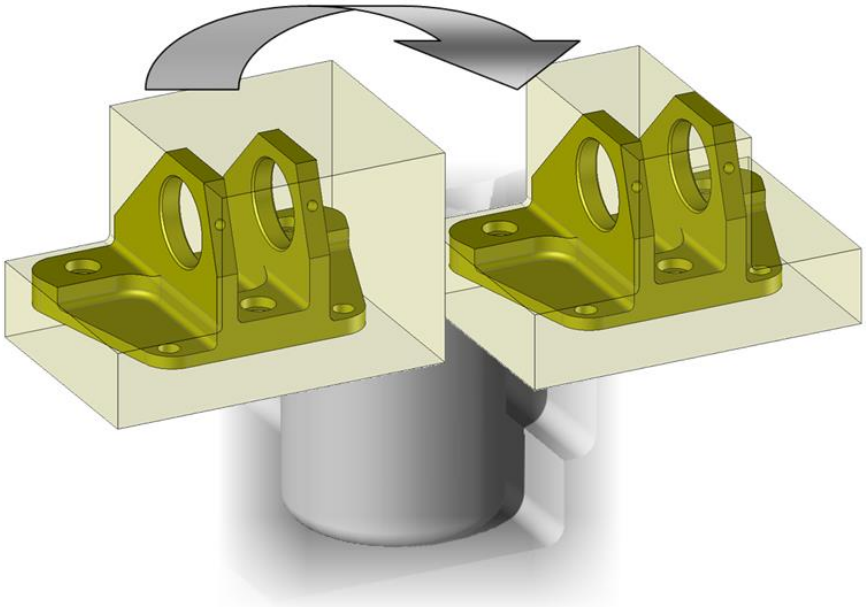


- Click on to **confirm** the tool.
- Click on the **Settings** icon and adjust the following settings:
  - **Stock to leave on floor**: *0mm*
  - **Stock to leave on wall**: *0mm*
  - **Maximal axial depth**: *15mm*
  - **Final axial depth pass**: *0mm*
  - **End milling strategy**: Select **Sweeping**
  - **Sweeping strategy**: Select **Zigzag cycle raw**
  - **Sweeping contouring strategy**: Select **After sweeping**
  - **Sweeping over length**: *5mm*
  - Check the **Minimum path number** box



- Click on ✔ to **confirm** the operation.
- **Save** the document.

Step 3 – Machining the right section of the part



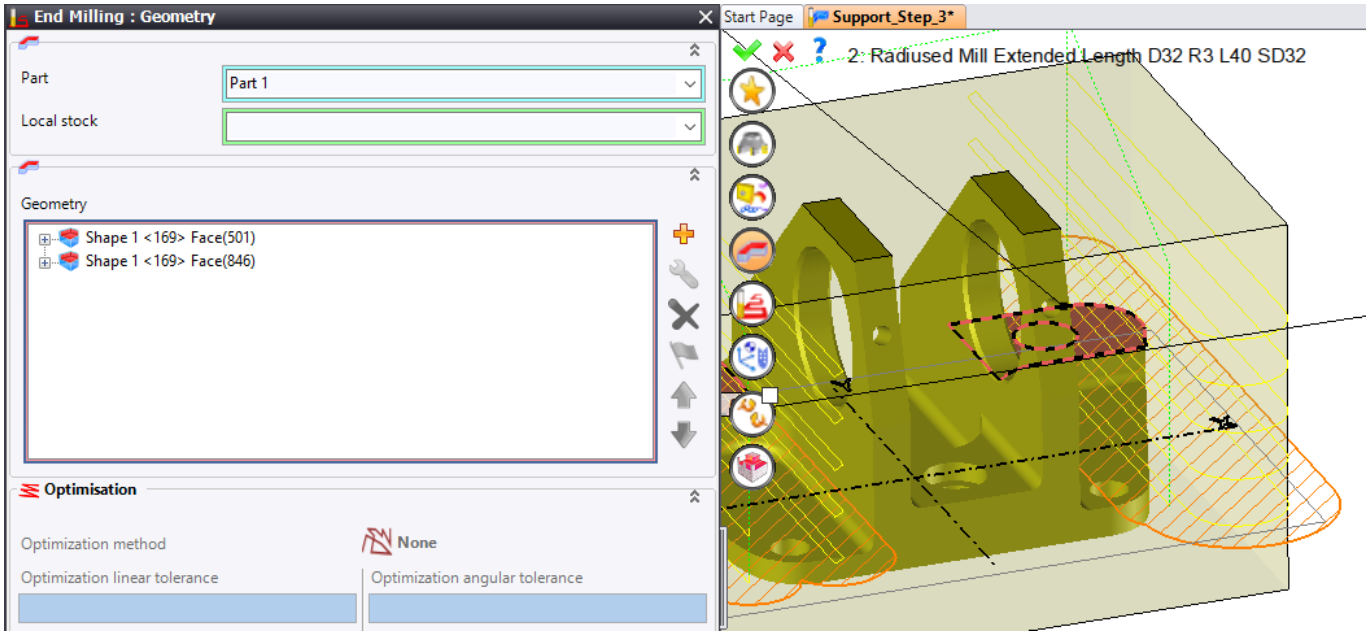
- Open the *Support_Step_3* machining document.

As you can see, the left and right sections clearly match. To repeat this machining, we will add a geometry to the previous operation.

Note: We could have selected the two surfaces to be machined beforehand and perform the open pocketing operation.

- From the NC Operations tree, **edit** the previously performed end milling operation.
- To add a geometry to an operation, click on the **Geometry** icon.
- Add the right-hand surface by clicking on it.

The machining operation is repeated while keeping the same settings. This action also allows you to remove a geometry from an operation.

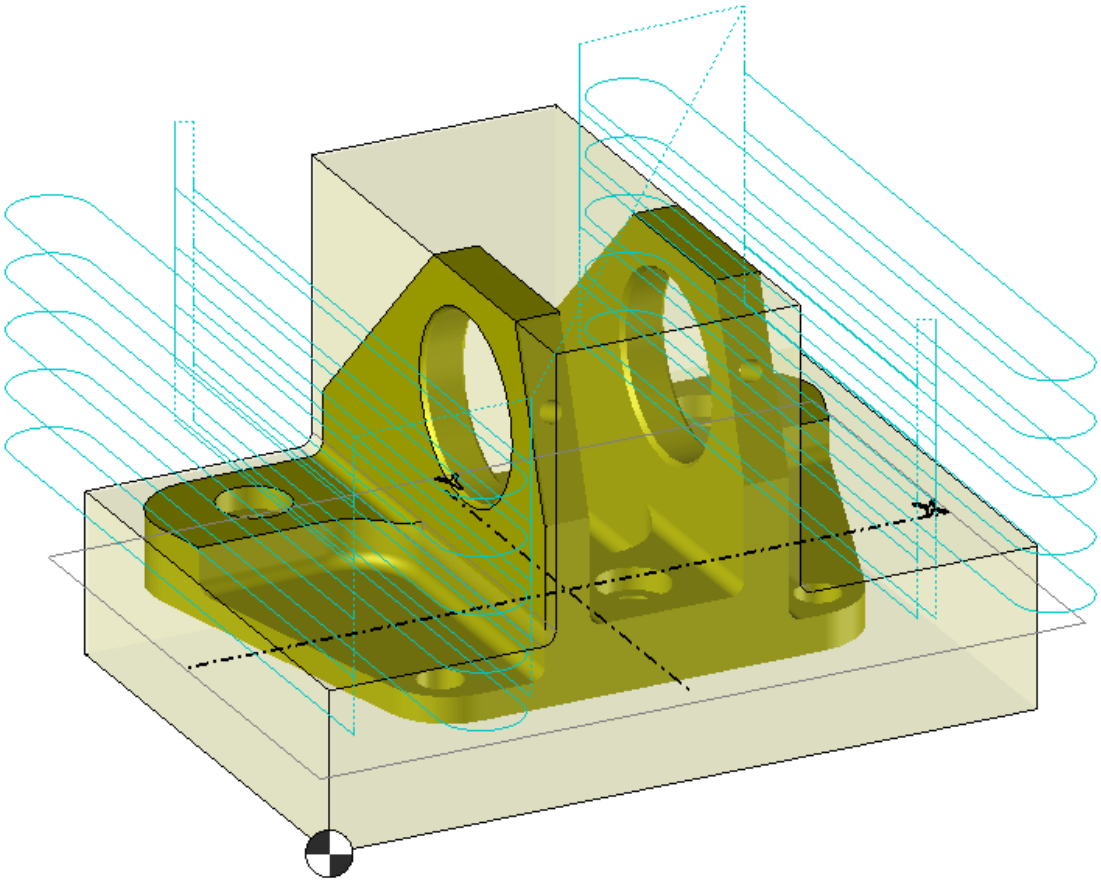


- Click on to **confirm** the operation.

Note: So far, the tool path has been ordered by pockets. We can see here the advantage of such a method for this part with regard to the time saved. The machine will first machine all the left-hand layers and then the right-hand ones.

A strategy by altitudes could have been used if the part had thin walls.

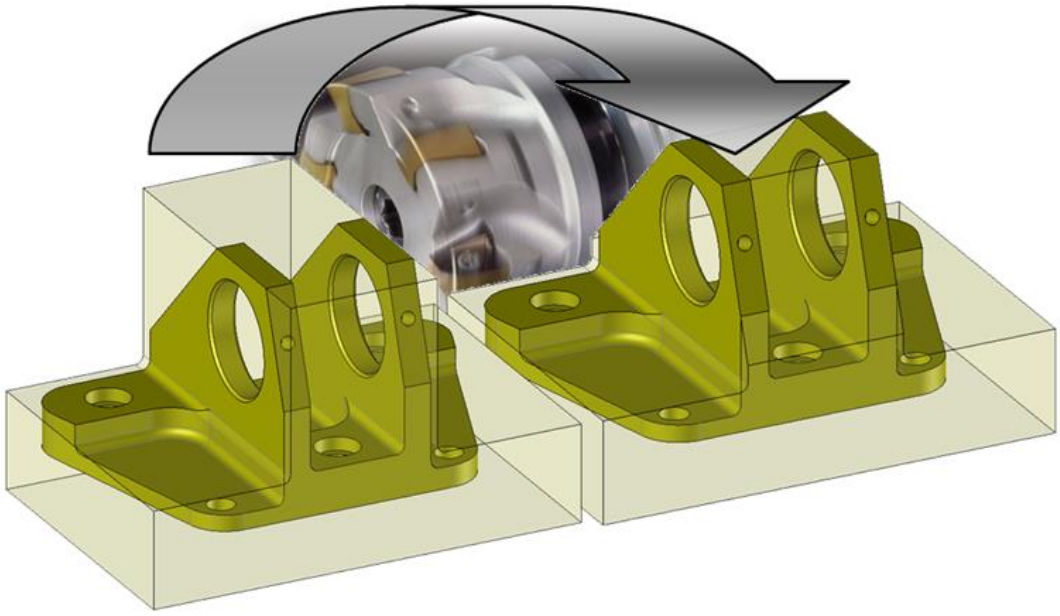
In the end, the machining preview should be as follows:



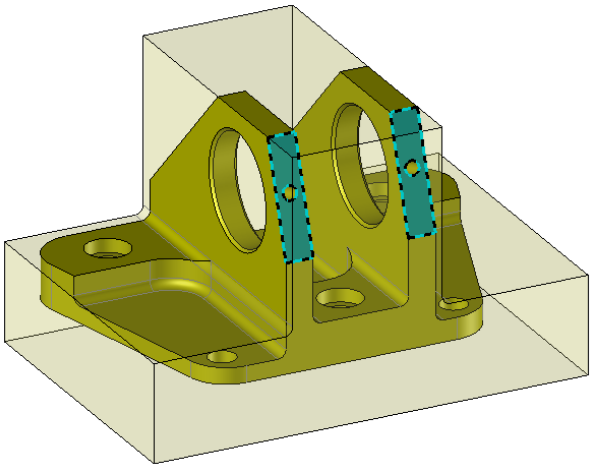
Note: Instead of adding the face in the first operation, you can also request to display the tool paths and then drag and drop this machining onto the new face. You only have to hold down the **Ctrl** key while you perform the drag-and-drop operation.

- **Save** the document.

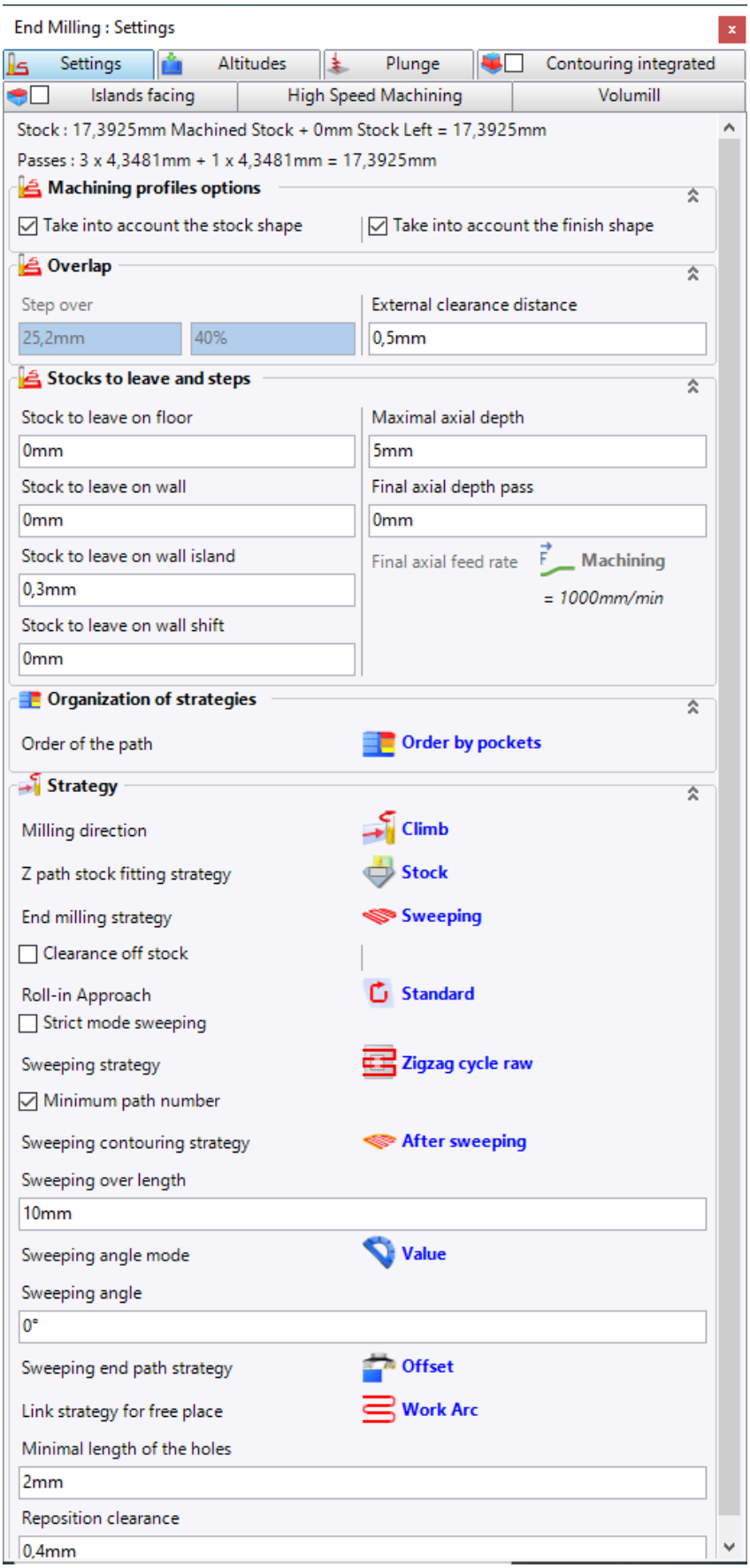
Step 4 – Machining the inclined planes



Machining the small inclined planes



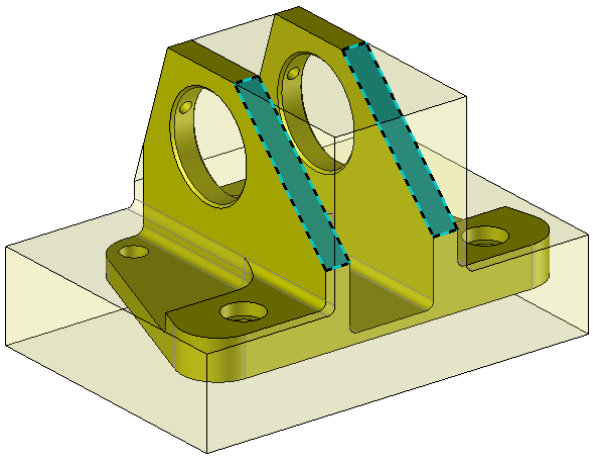
- Open the *Support_Step_4* machining document.
- Right-click on one of the blue faces shown above and select the **End Milling** 2D machining command.
- In the **Tool choice** option, check the **face mill** that was used for the first machining (63mm diameter).
- Click on the **Settings** icon and adjust the following settings:
  - **Stock to leave on floor** and **Stock to leave on wall**: *0mm*
  - **Maximal axial depth**: *5mm*
  - **Final axial depth pass**: *0mm*
  - **Z path stock fitting strategy**: Select **Fit the stock**
  - **End milling strategy**: Select **Sweeping** and the **Zigzag cycle raw** strategy
  - Check the **Minimum path number** box
  - **Sweeping over length**: *10mm*
  - **Sweeping angle mode**: Select **Value** and **Sweeping angle**: *0°*



Note: As you can see, once the tool is selected, it automatically orients itself.

- Click on ✔ to **confirm** the operation.

Machining the large inclined planes



To machine the large inclined planes, you can reproduce the same operations as before or use the **Ctrl** + drag-and-drop technique by dragging the first machining operation onto the new face in the graphics area.

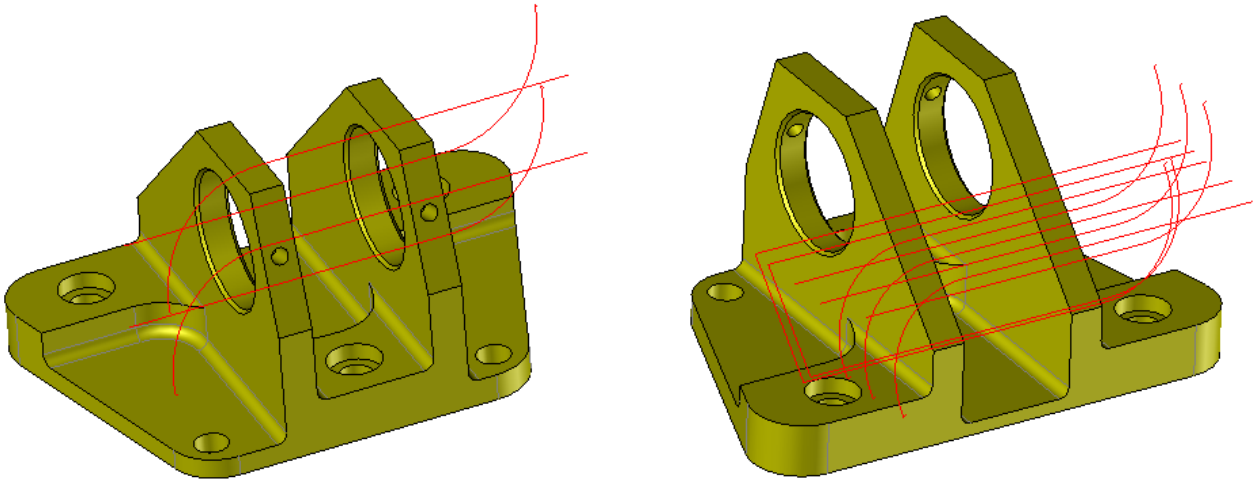
To reproduce the machining on the other inclined plane, proceed as follows:

- From the NC Operations tree, left-click on the 3: Facing operation.
- Place the mouse cursor over the tool path that appears in the graphics area. The color turns red.
- Press the **Ctrl** key.
- While holding down the **Ctrl** key, left-click on the tool path and drag and drop it onto the new face.

The operation is then copied identically. You can edit some settings.

- **Edit** the operation and click on the **Settings** icon.
- In the **Link strategy for free place** field, select the **Overlength link direct at working feed rate** option.

The machining previews of the two inclined planes should be as follows:

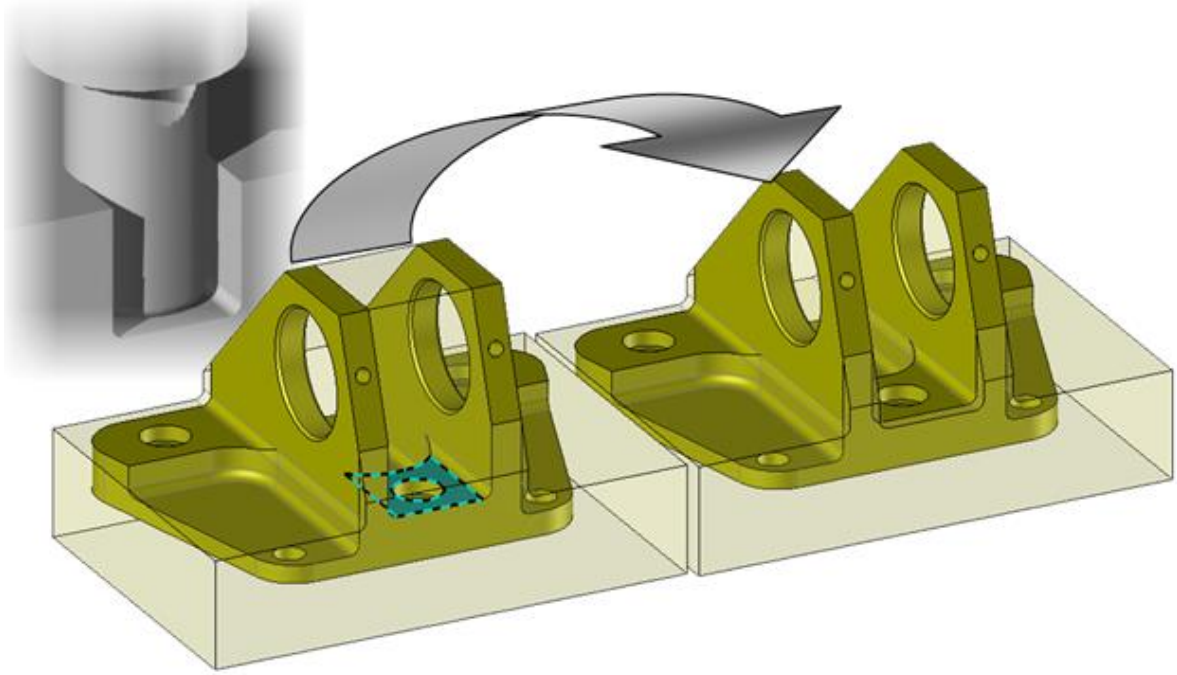


- Click on to **confirm** the operation.
- **Save** the document.

Step 5 – Machining the center groove

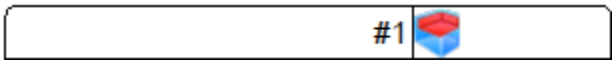
- Open the *Support_Step_5* machining document.

For this step, we will machine a center groove. We will use a method that consists of selecting the geometry as late as possible.



- From the **2D/3D** tab, select the **End Milling** 2D machining command.
- In the **Tool choice** option, select the **radiused mill** with a 32mm diameter and 3mm radius from the tool magazine.
- In the **Geometry** option, select the blue-colored plane as shown above.
- Click on the icon to update the graphics area.

On the label in the graphics area, you can see that **TopSolid** has automatically recognized the open pocketing operation.



- In the **Settings** option, enter zero stock to leave and keep all other default settings.

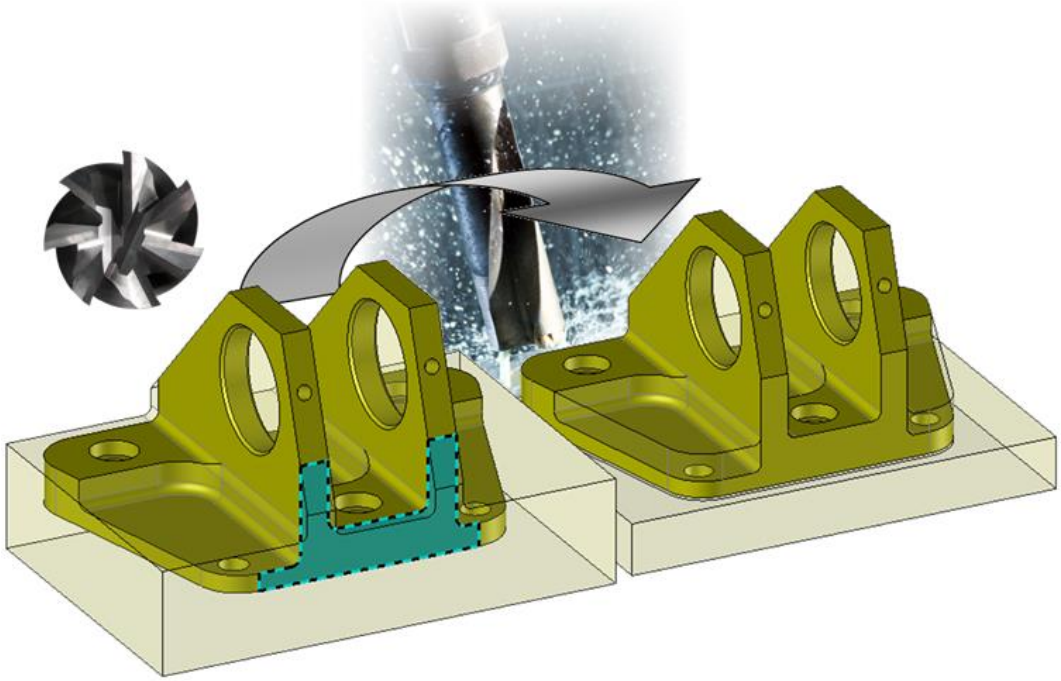
Note: This operation could have been included in Step 3. In fact, we could have selected this geometry during the first open pocketing operation. We could also have used the copy and paste method.

- Click on to **confirm** the operation.

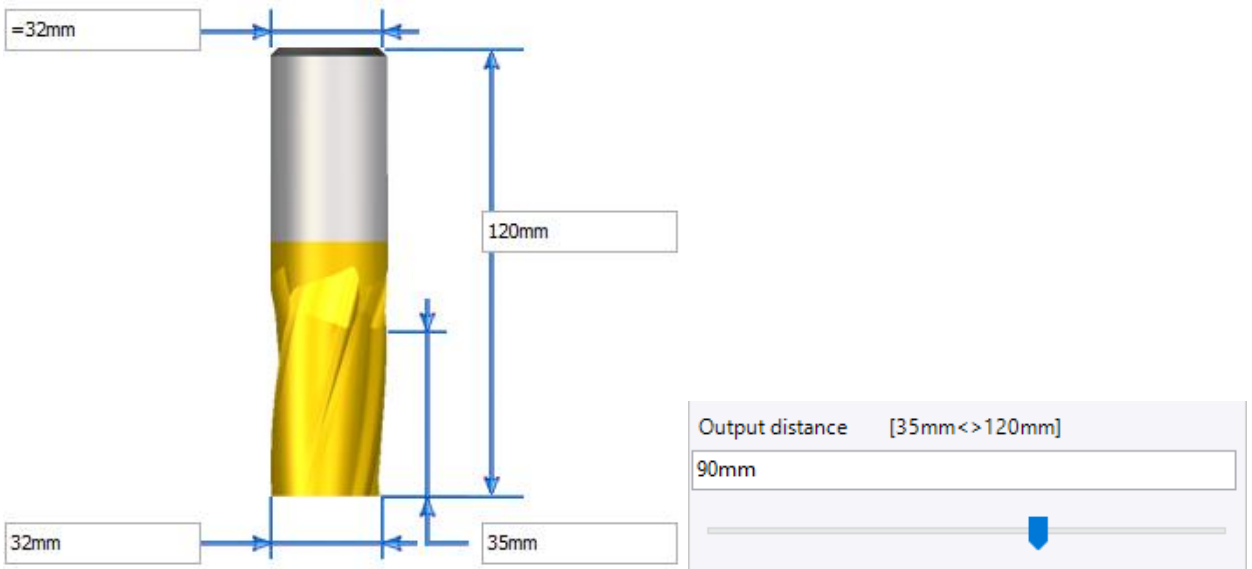
Step 6 – Machining the outer section

- Open the *Support_Step_6* machining document.

We will machine the outer section of the part by using a contouring cycle.

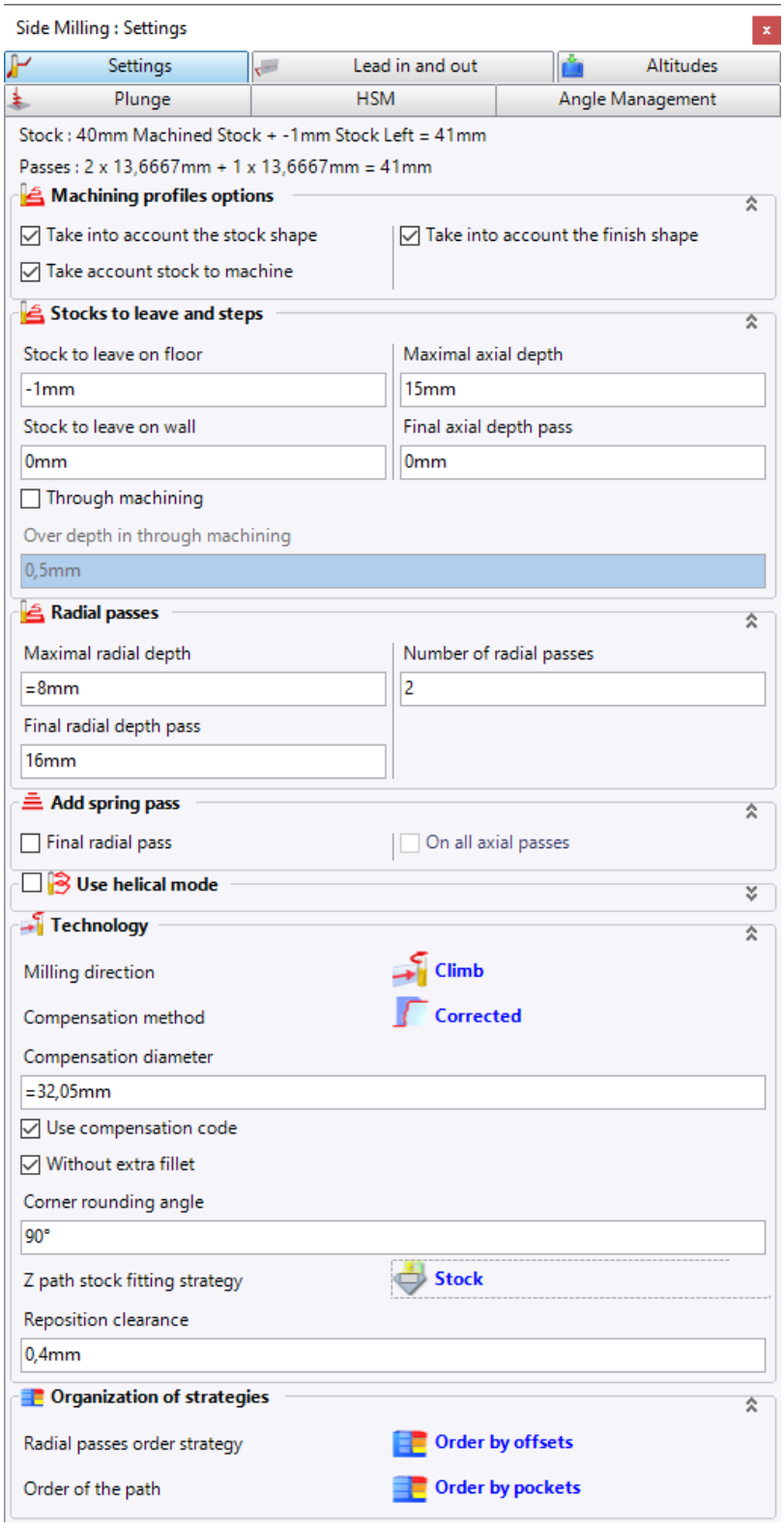


- Right-click on the blue-colored face as shown above.
- Select the **Side Milling** command.
- In the **Tool choice** option, create a *32mm* diameter **side mill** with a total length of *120mm* and an output distance of *90mm*.



- Click on to **confirm** the tool.

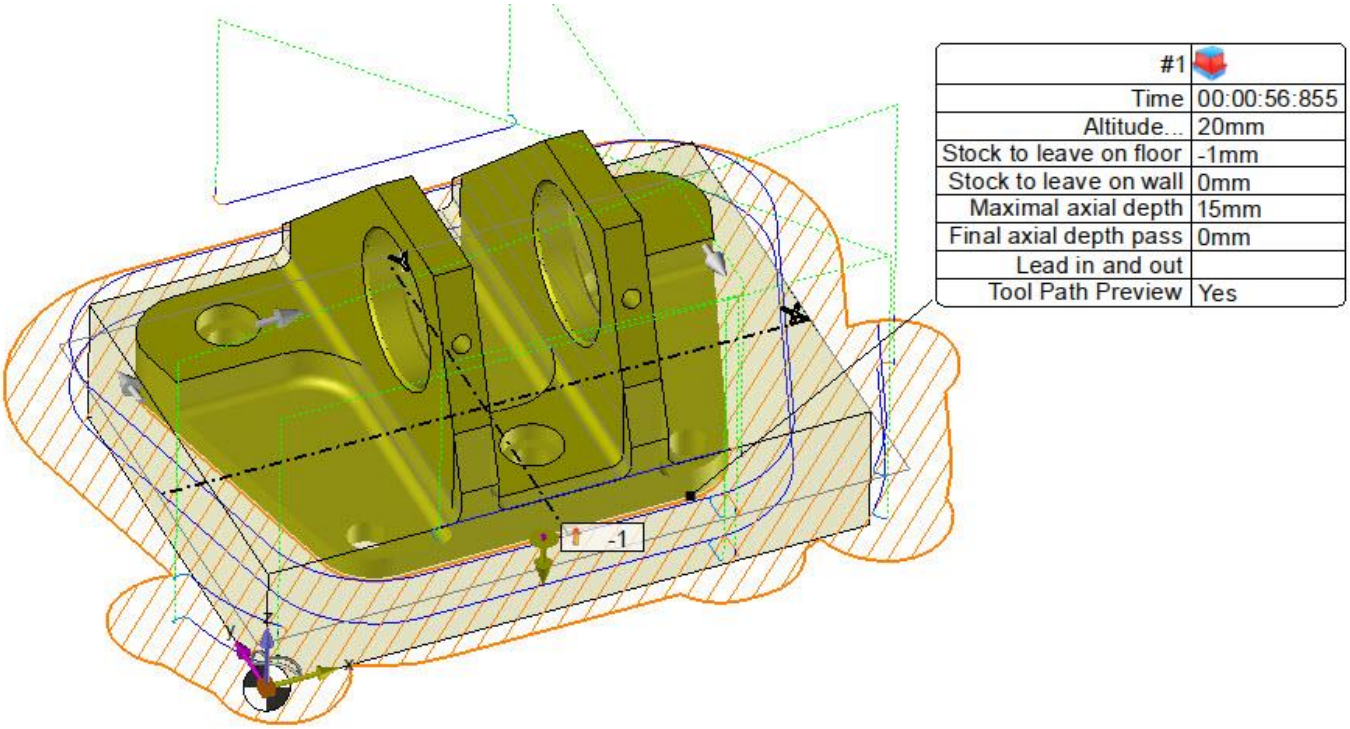
- In the  **Settings** option, adjust the following settings:
  - **Stock to leave on floor**: *-1mm*
  - **Stock to leave on wall**: *0mm*
  - **Maximal axial depth**: *15mm*
  - **Number of radial passes**: *2*
  - **Final radial depth pass**: *16mm*
  - **Z path stock fitting strategy**: Select **Fit the stock**



Note: You can select any face on the part's contour. The choice of face only determines the end and return points of the tool.

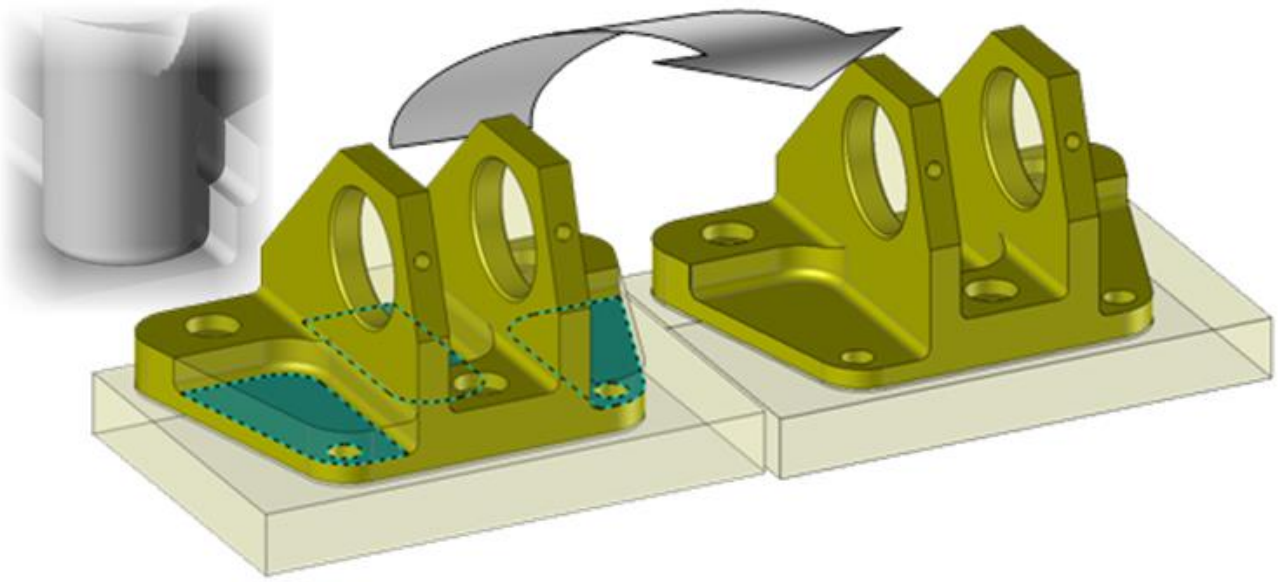
A negative stock to leave on floor is left in order to avoid having to redo the contour during a possible additional phase.

The machining preview should be as follows:



- Click on to **confirm** the operation.
- **Save** the document.

Step 7 – Machining the three steps

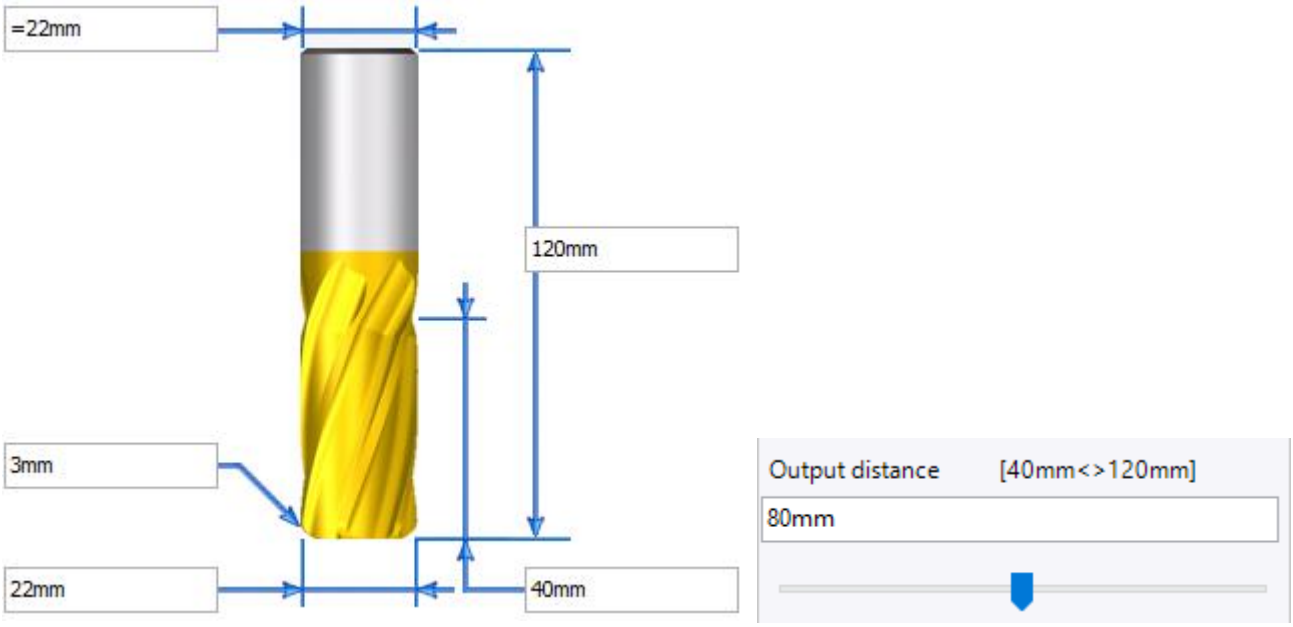


- Open the *Support_Step_7* machining document.

We will machine the three steps by using a multiple pocketing cycle.

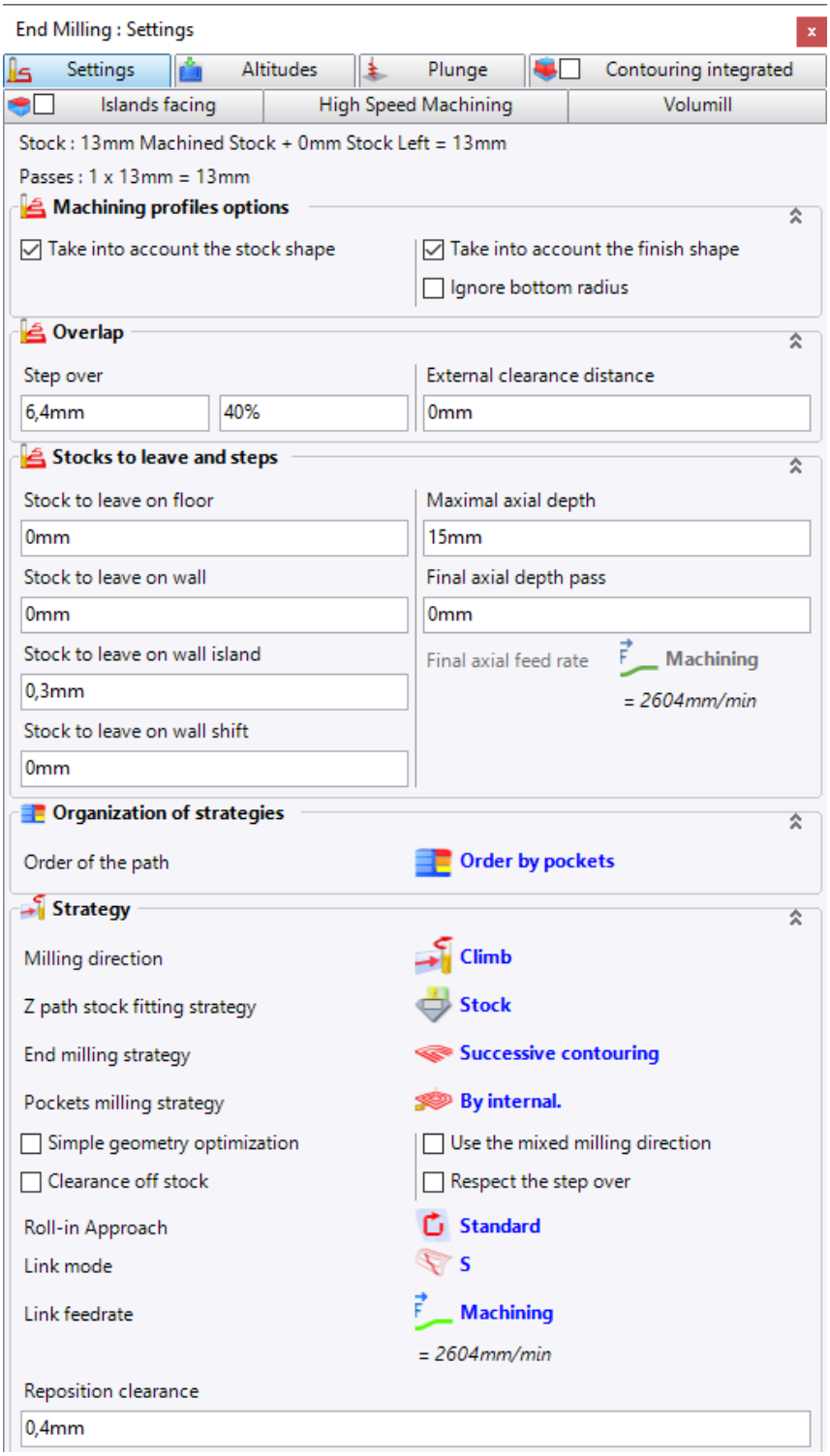
To perform this machining, we will make a multiple selection by holding down the **Ctrl** key while selecting the three blue-colored faces.

- Hold down the **Ctrl** key and select the three blue-colored faces as shown above.
- Right-click in the graphics area and select the **End Milling** command.
- In the **Tool choice** option, create a *22mm* diameter **radiused mill** with a radius of *3mm*, a total length of *120mm* and an output distance of *80mm*.



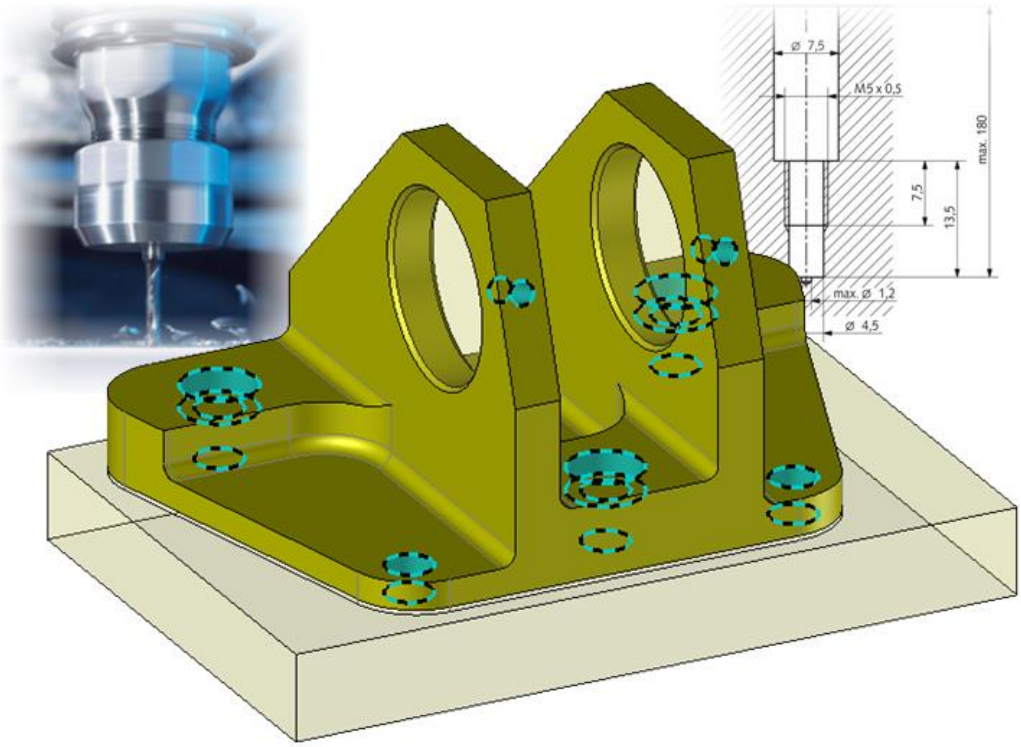
- Click on to **confirm** the tool.

- In the **Settings** option, adjust the following settings:
  - **Stock to leave on floor** and **Stock to leave on wall**: *0mm*
  - **External clearance distance**: *0mm*
  - **Maximal axial depth**: *15mm*
  - **Final axial depth pass**: *0mm*
  - **Z path stock fitting strategy**: Select **Fit the stock**
  - **End milling strategy**: Select **Successive contouring**

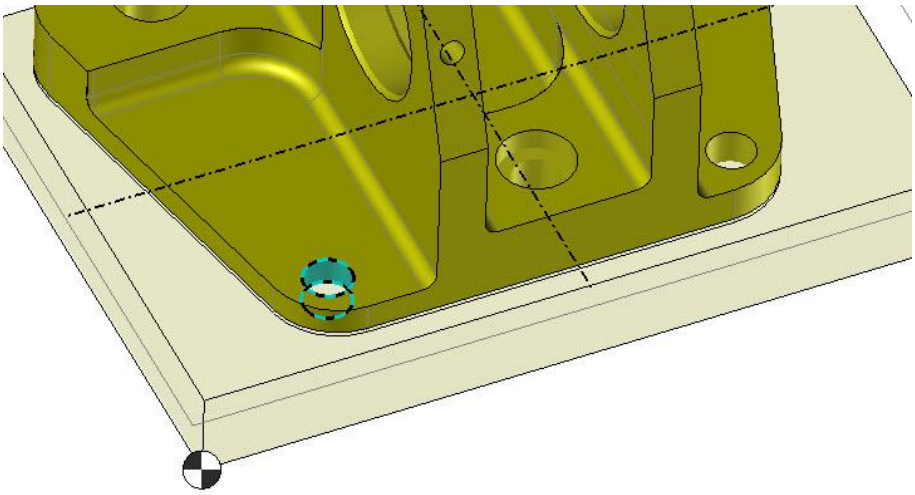


- Click on to **confirm** the operation.
- **Save** the document.

Step 8 – Performing the drillings



- Open the *Support_Step_8* machining document.
- Select the blue-colored cylindrical face as shown below.

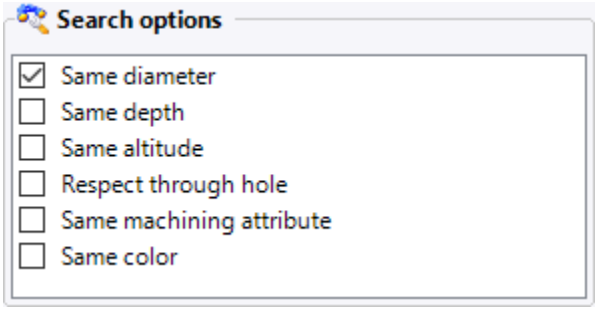


- Right-click in the graphics area and select the **Drilling** > **Hole Machining** command.

The label provides some information, including the hole diameter (diameter of 10mm in our case).

Kind of machining	
Time	00:00:00:000
Diameter	10mm
Machined depth	12,8868mm
Geometry type	
Through hole overdepth	2mm
Tool Path Preview	Yes

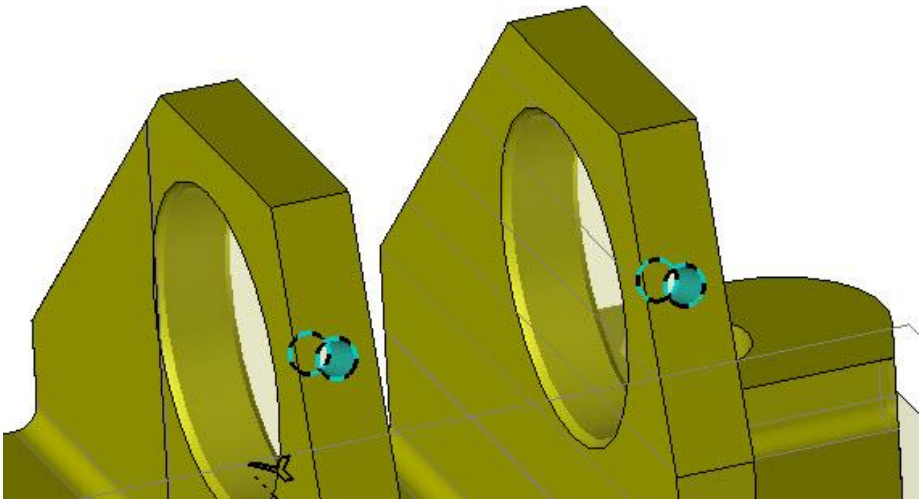
- Click on the **Geometry** icon. In the search options, check the **Same diameter** box and uncheck the **Same depth**, **Same altitude**, **Respect Through Hole**, **Same machining attribute** and **Same color** boxes.



- Click on the icon to search for similar holes. We have five holes to drill.
- Do not include the partial cylinders in the analysis.
- Click on the **Tool choice** icon.
- Double-click on the **Twist Drill** thumbnail. By default, the tool diameter is the hole diameter. Adjust the total length of the drill to *120mm* and the output distance to *90mm*.
- **Confirm** the tool, then **confirm** the operation.

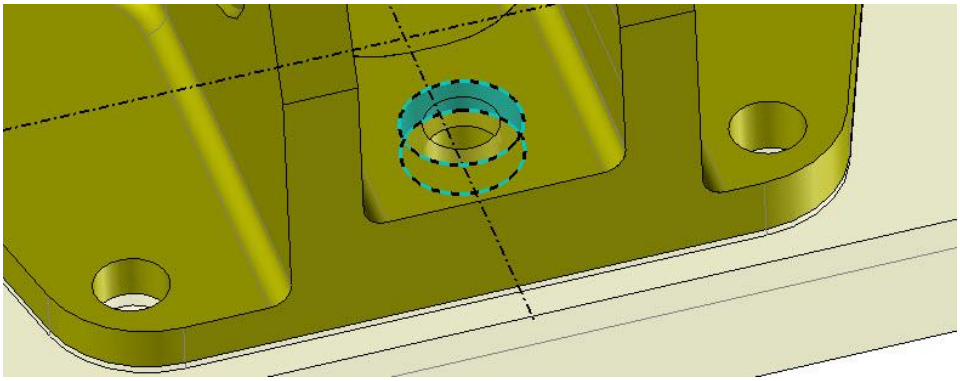
The drilling operation is achieved.

- Repeat the procedure to drill the two holes shown in blue below.

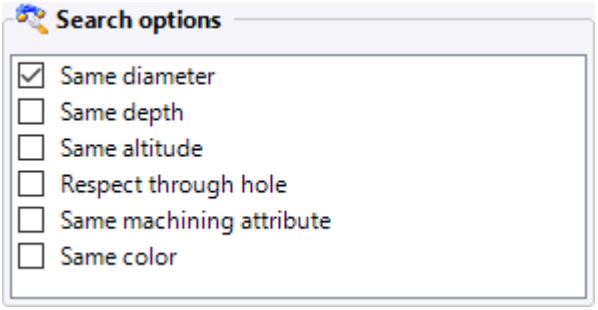


We will now machine the spot faced holes.

- Select the blue-colored cylindrical face as shown below.



- Right-click in the graphics area and select the **Drilling** > **Hole Machining** command.
- In the **Geometry** option, check the **Same diameter** box and uncheck the **Same depth**, **Same altitude**, **Respect Through Hole**, **Same machining attribute** and **Same color** boxes.

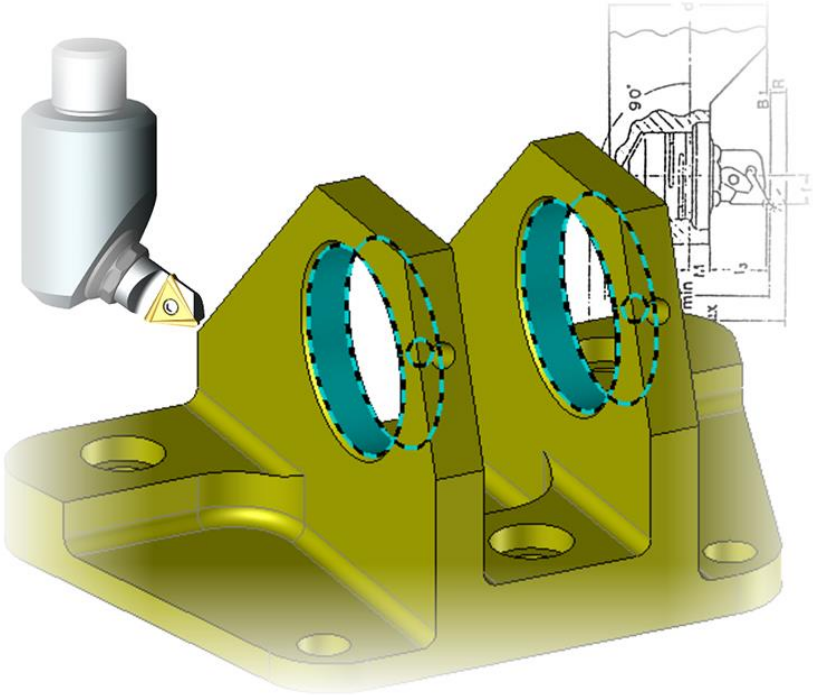


- Click on the icon to search for similar holes. We have three holes to drill.
- On the label, in the **Kind of machining** field, double-click on the icon and select **Spot Facing**.
- In the **Tool choice** option, double-click on the **Spot Face Mill** thumbnail. By default, the tool diameter is the hole diameter. Adjust the total length of the drill to *120mm* and the output distance to *90mm*.
- **Confirm** the tool, then **confirm** the operation.

The spot facing operation is complete.

- **Save** the document.

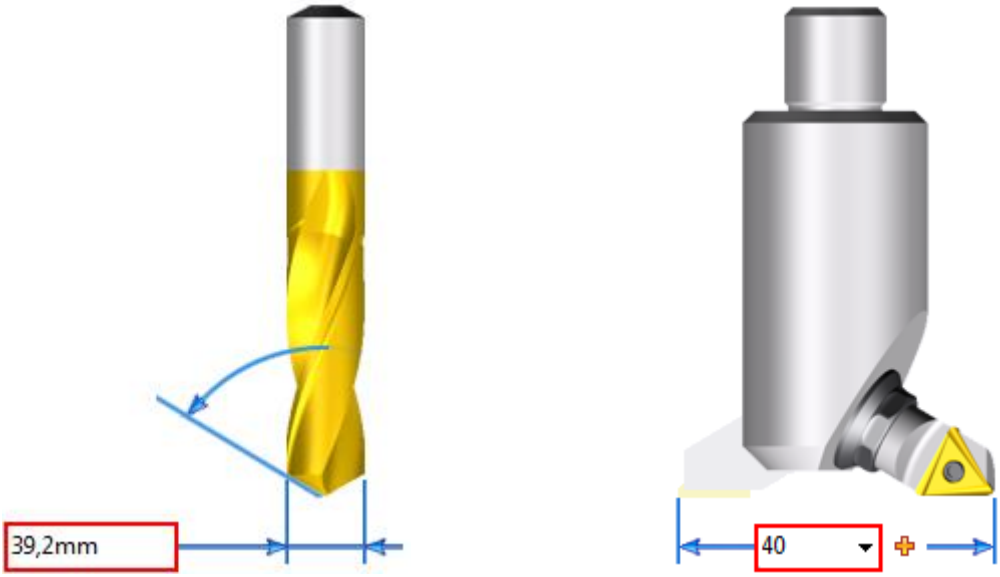
Step 9 – Performing the borings



- Open the *Support_Step_9* machining document.

We will machine the blue-colored bores as shown above. First, we will load the tools in the tool magazine.

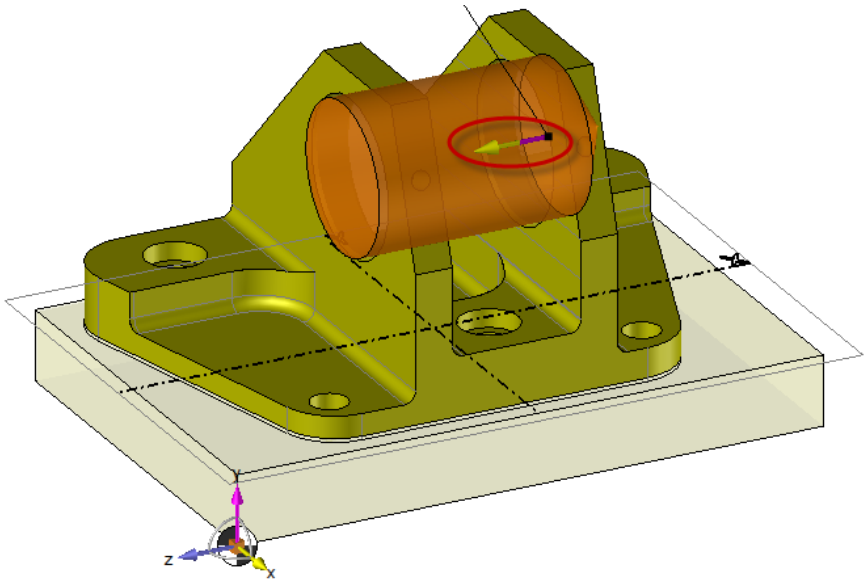
- From the **Equipment** tab, select the **Tool Manager** command.
- Double-click on the **Twist Drill** thumbnail, then create a *39.2mm* diameter drill with a total length of *120mm* and an output distance of *90mm.* Click on to **confirm**.
- Double-click on the **Boring Bar** thumbnail, then create a *40mm* diameter bar with a total length of *120mm* and an output distance of *90mm.* Click on to **confirm**.



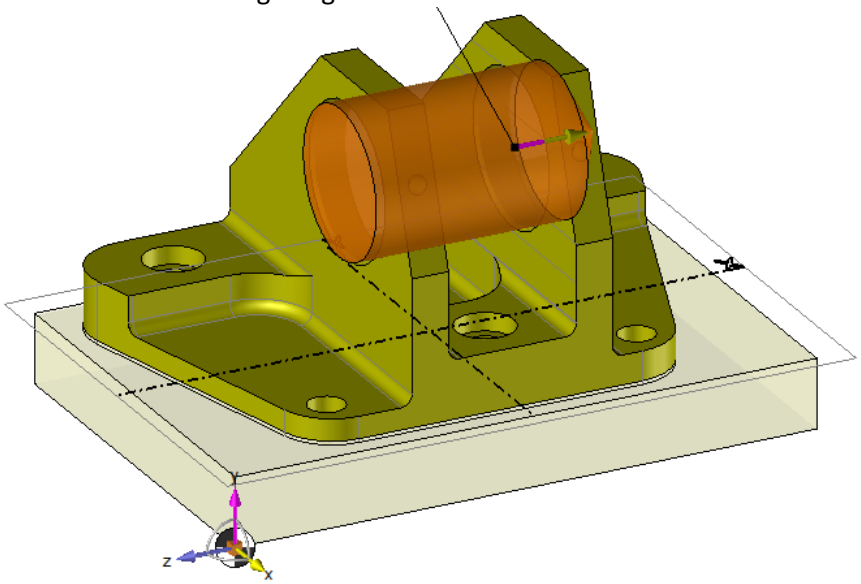
The tools are added to the tool magazine.

- Right-click on one of the blue-colored cylinders as shown on the previous page.
- Select the **Drilling** > **Hole Machining** command.
- Select the 39.2mm diameter drill as the tool.

You can change the orientation of the drilling by double-clicking on the yellow arrow (circled in red below) located in the axis of the drilling in the graphics area.



Your screen should look like the following image:



The WCS solution is then automatically modified.

The two faces can be drilled either in a single machining operation, or in two machining operations.

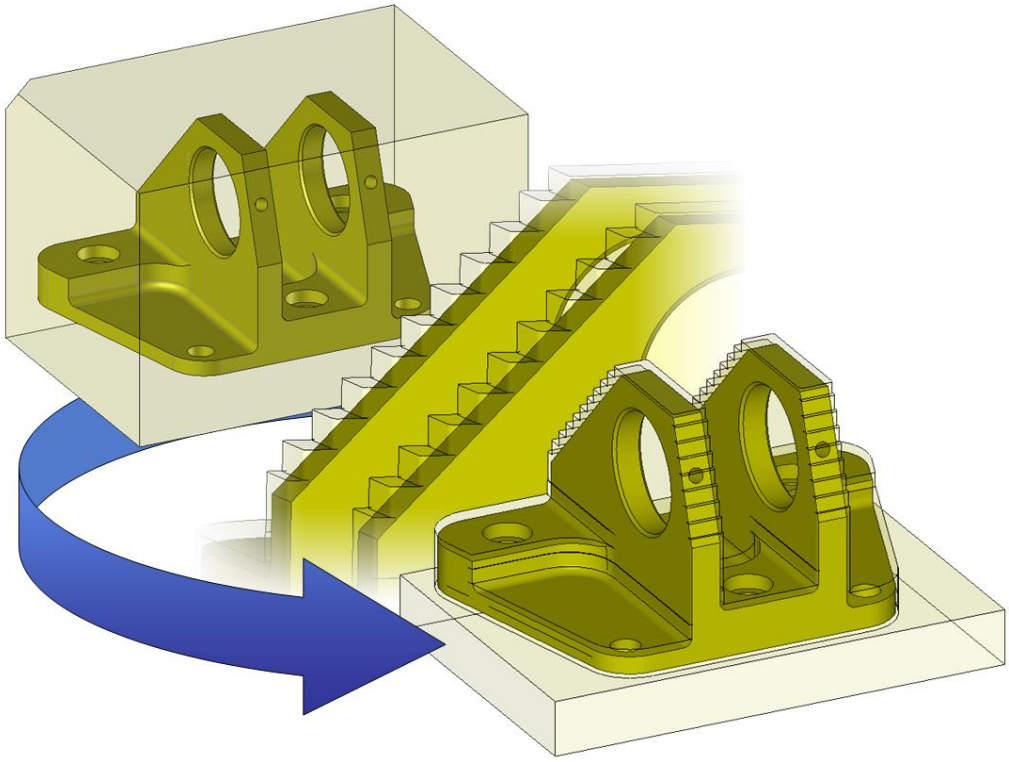
- Repeat the operation by selecting the boring bar as the tool and adjusting the drilling type to **Reaming** in the **Kind of machining** field of the label.

Note: If the tool does not appear in the toolbar when selecting the tool, you only have to right-click and select **Show all pockets**.

As a standard, 98% of the nominal diameter should be bored.

- **Save** the document.

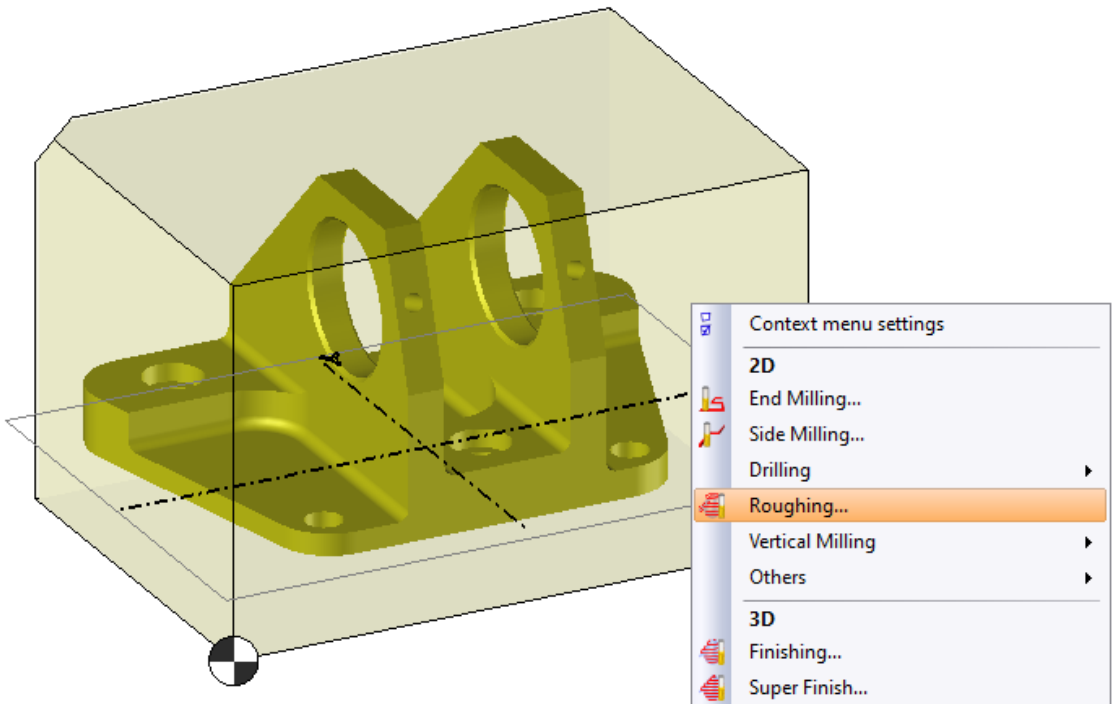
Step 10 – Alternative machinings



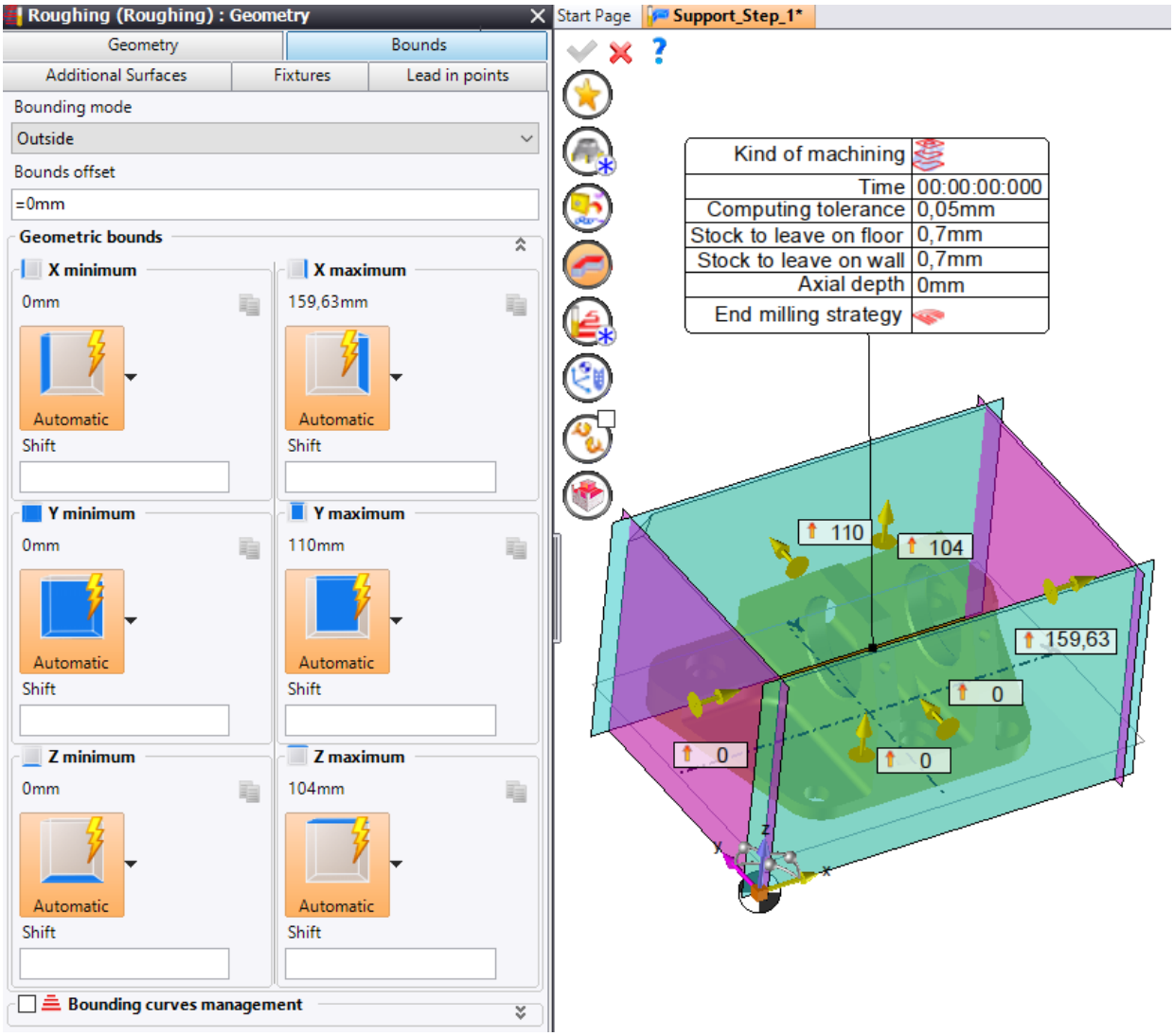
- Open the *Support_Step_1* machining document again.

We have previously seen how to completely machine a 2D part. Here, we will see a process that could reduce the amount of finishing operations. This roughing could potentially save you time (if using the same mill!).

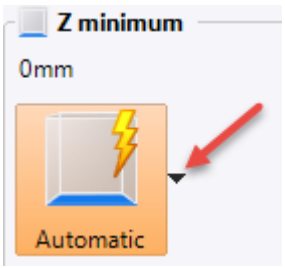
- To do this, open the machining in its initial state (no machining operation).
- Right-click in the graphics area and select the **Roughing** command.

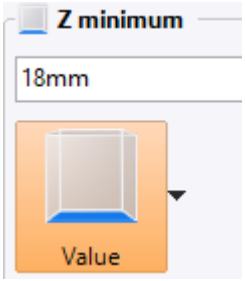


For this operation, we will machine the maximum material. The color planes display the machining bounds. You can manually move them or define them exactly.



- Click on the **Geometry** icon.
- Go to the **Bounds** tab.
- Leave the default bounds, except for the Z minimum, which will be 18mm. To enter the Z minimum, first click on the black triangle icon, then click on the **Value** icon and enter *18mm*.





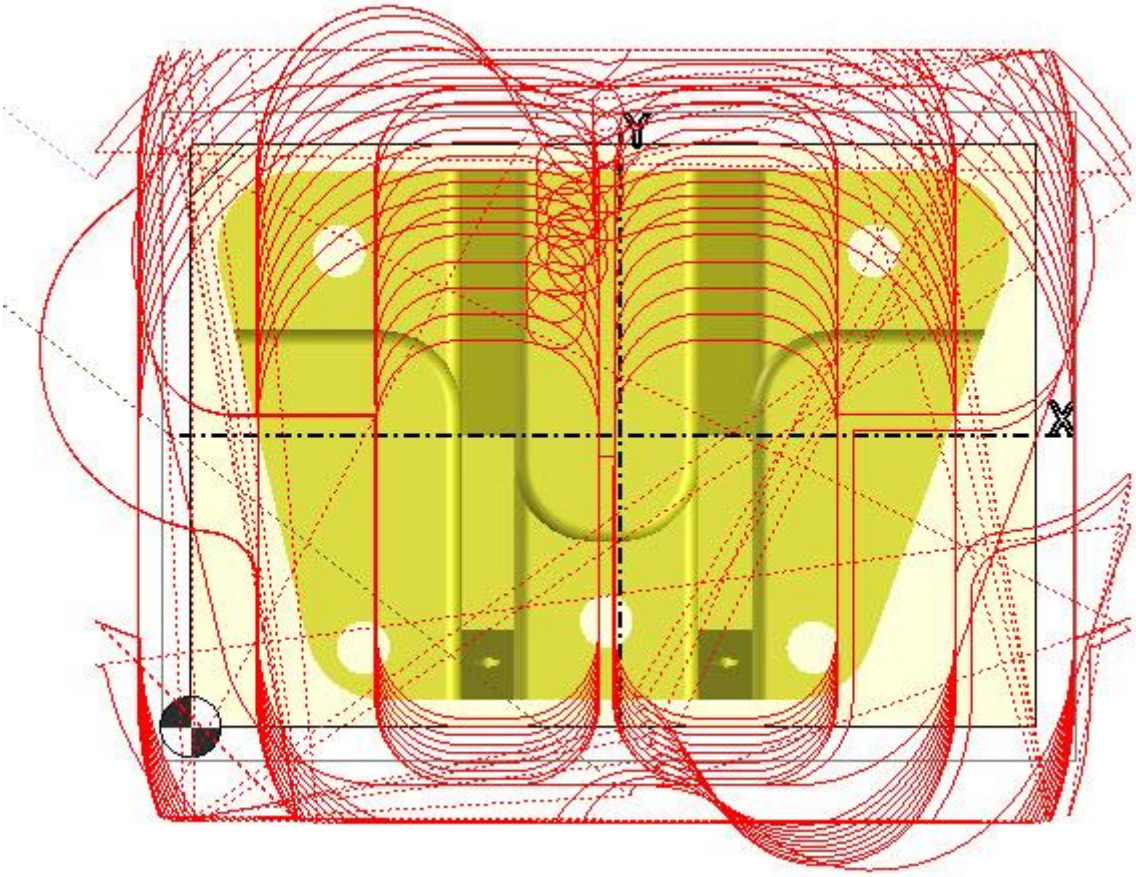
- In the **Tool choice** option, create a *32mm* diameter **side mill** with a total length of *120mm* and an output distance of *90mm*.

- On the label, adjust the **stock to leave on floor** to *1mm*, the **stock to leave on wall** to *0mm* and the **computing tolerance** to *0.01mm*.

Kind of machining	
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave on floor	1mm
Stock to leave on wall	0mm
Axial depth	7mm
End milling strategy	

- Click on ✔ to **confirm** the roughing operation.

In top view, the machining tool paths should be as follows:



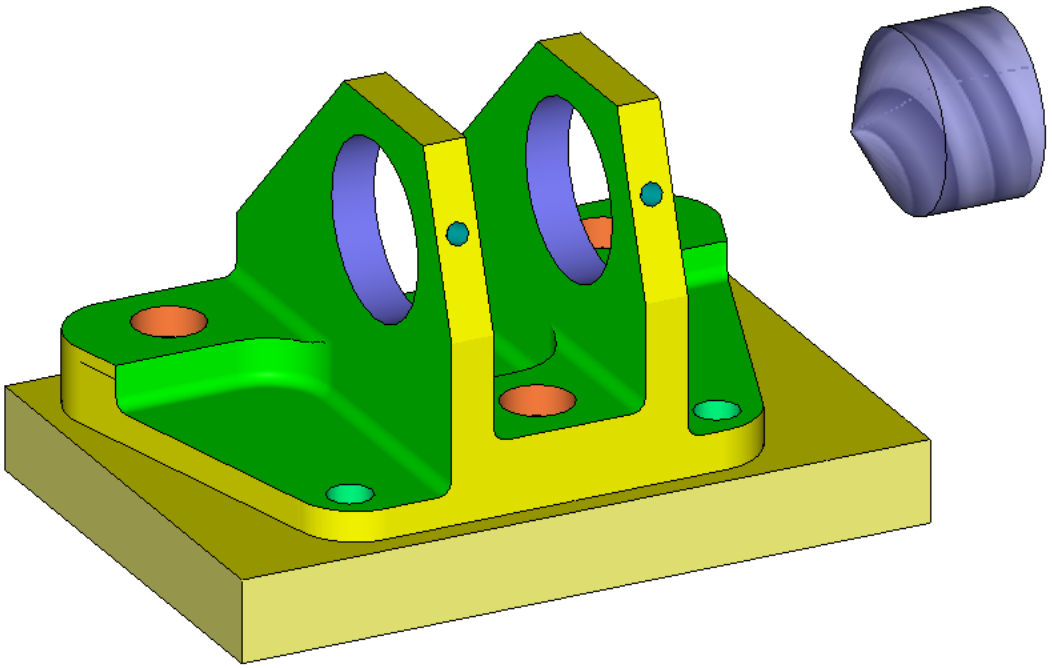
By default, the stock is not updated.

- In order for the operation to update the stock, **edit** the roughing operation, click on the **Machine Dialog Flags** icon and select the **Update stock** option.

Note: If you want to redo the machining by starting with a roughing operation, note that for the facing operations you must select multiple geometries.

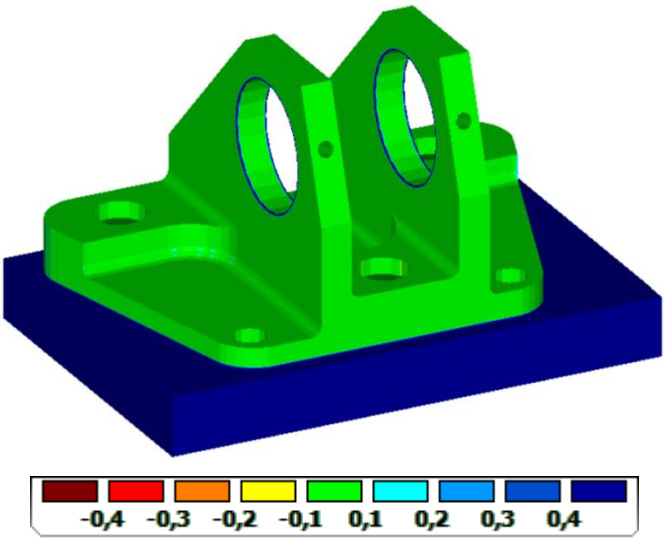
Conclusion

At the end of the cycle, the part is practically machined and the following result is obtained.



The **Verify** tab allows you to check the machining and assess the tool paths of each tool. Each color corresponds to a different tool.

Some stocks to leave may not be machined, which is why you can use the **Comparison** tool to measure the distance between the finish and the fictitious stock. If the part is green, this distance is assumed to be zero. If it is closer to blue, not enough material was machined. Alternatively, if it is closer to red, too much material was machined.



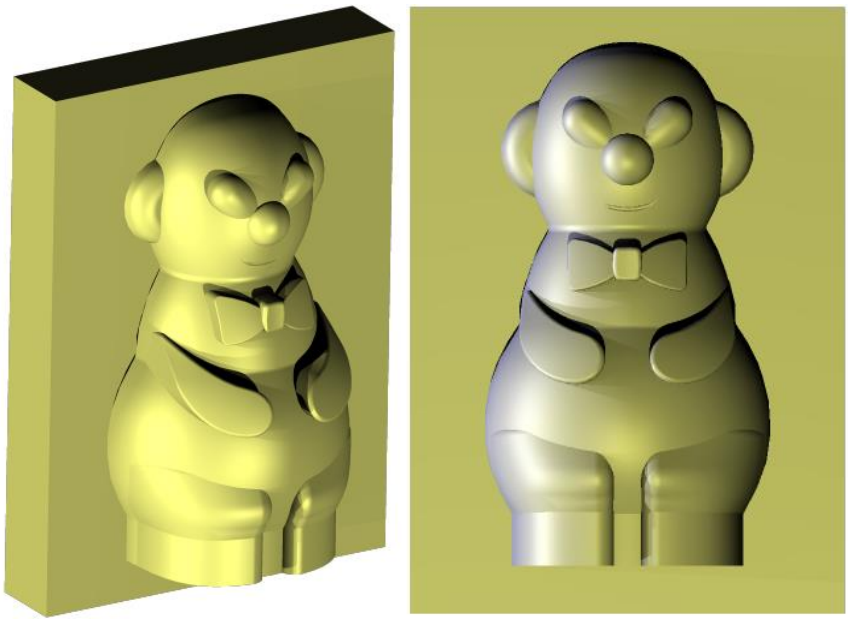
You can see here that only the boring chamfers are not machined.

The purpose of this chapter is to introduce you to **TopSolid 7**'s Cam module and present its range of 2D features. By no means are the order of operations, choice of tools, or operation settings optimized or defined for industrial use.

3D Milling

Introduction

The purpose of this chapter is to create a machining program for the following part.



This is a sinking electrode for a perfume bottle mold.

To help you become familiar with **TopSolid 7**, the machining of the part above is divided into several steps. For each step, a machining document is available in which the operations have been completed. You can refer to these documents if you run into any difficulties. The machining steps are numbered from 1 to 4, so there will be four numbered machining documents. These machining documents are provided in a TopPkg file named *TopSolidCam_3DPartStepByStep.TopPkg* (during installation, this file is installed in the "Program Files\TOPSOLID\TopSolid 7.14\Samples" folder).

This tutorial is based on the assumption that the operator has experience with **TopSolid'Design** features and that they have already become familiar with the 2D features. It is therefore assumed that the user is familiar with the finish part, stock, machine, and tool concepts.

The overall programming approach is as follows:

1. Selecting the machine tool
2. Defining the stock part
3. Positioning and defining the part (as per **TopSolid'Cam**) and its program origin
4. Defining the work coordinate systems automatically (angular positions that must take the two machine tool’s plates)
5. Defining the operations with integrated stock management
6. Simulation
7. Editing the ISO file

Given that this is a 3D tutorial, we will directly begin at Step 5. Therefore, the Cam part has already been created and the tool magazine is loaded with the required tools.

Machining

As you saw in the 2D tutorial, **TopSolid** suggests two approaches to perform a machining.

- **Subject – Verb**:
  Select the entities to be machined, and then select a machining command.

With this solution, you can select one or more faces with the mouse and indicate the machining on which to apply the faces.

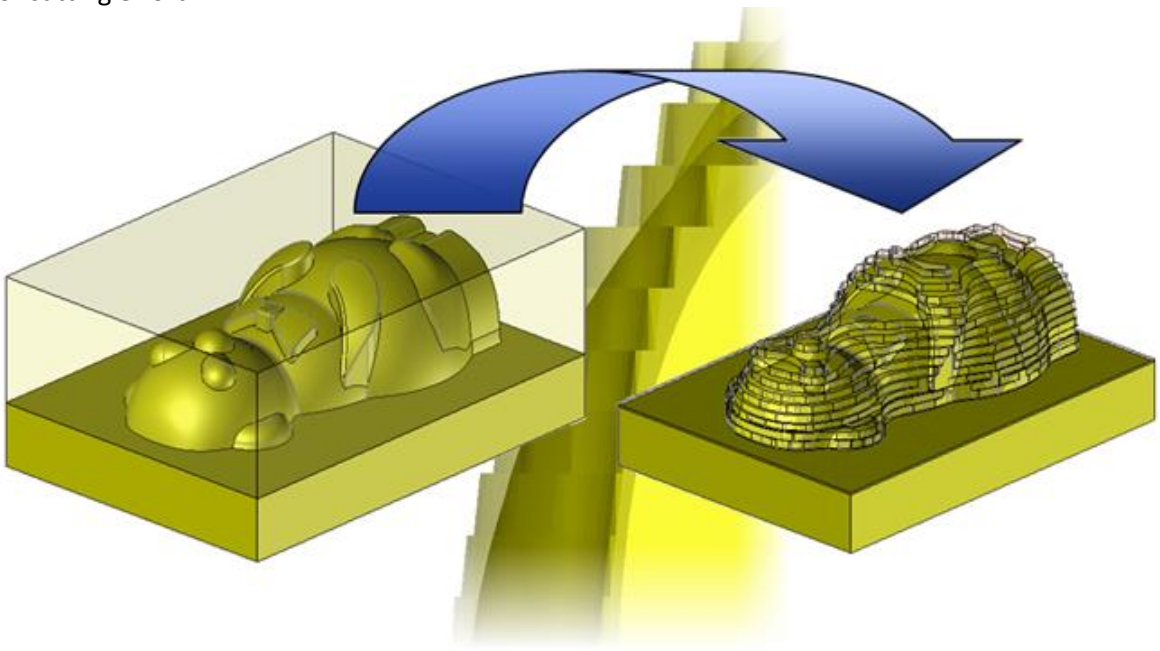
- **Verb – Subject**:
  Select a machining command and indicate the entities on which to apply this command.

For 3D machinings, these two methods can be applied. Given that the entire part is machined, selecting a face opens the 3D machinings even though the entire part will be machined by default. You can also right-click in the graphics window to avoid having to select a face.

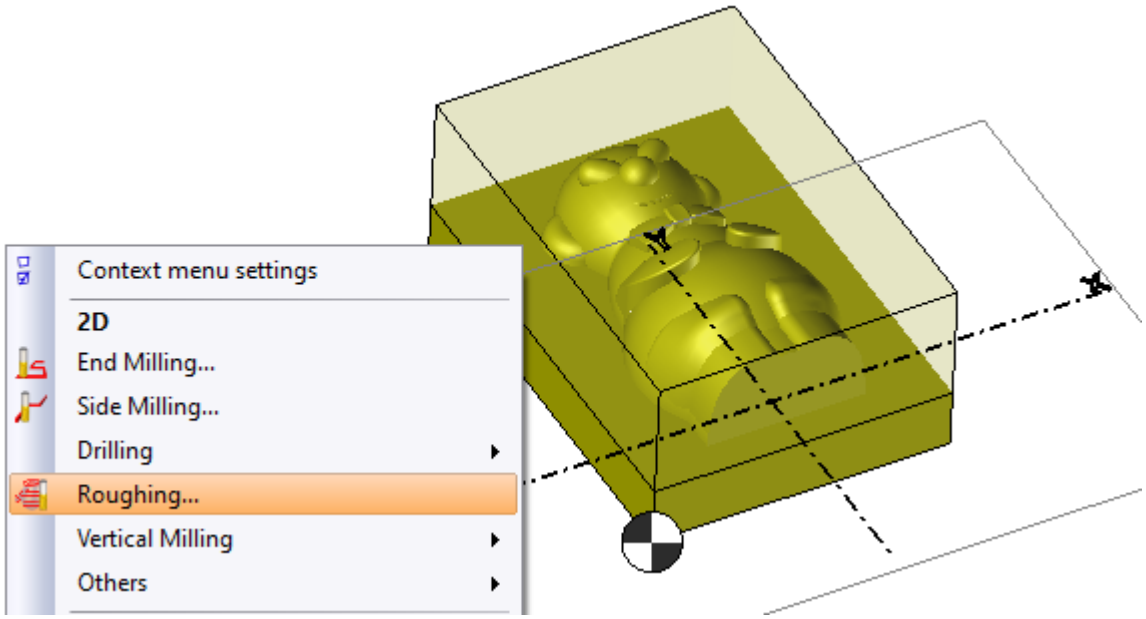
Step 1 – Roughing

- Create a **new project** named *3D Machining* using a **blank template**.
- **Import** the *TopSolidCam_3DPartStepByStep.TopPkg* package in your project.
- Open the *Nounours_Step_1* machining document.

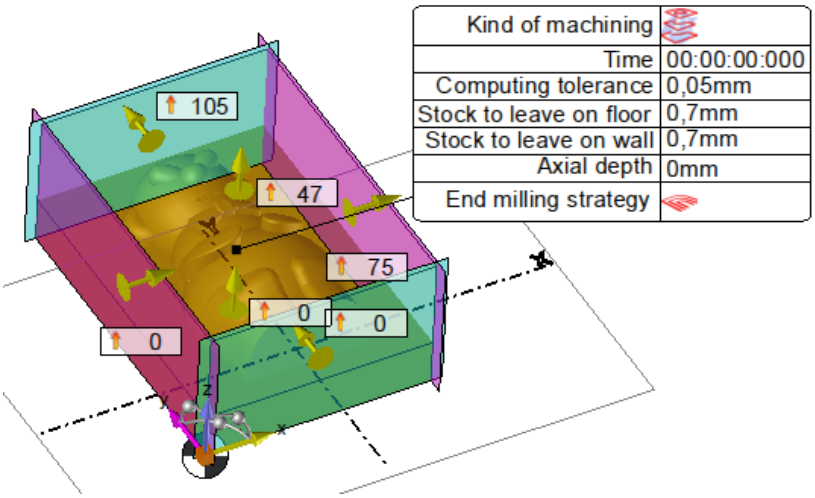
During this step, we will completely rough the model. This operation consists of removing the maximum amount of material with a dimensioned roughing tool in order to remove the greatest amount of material with the least amount of cutting effort.



- Right-click in the graphics area and select the **Roughing** command.

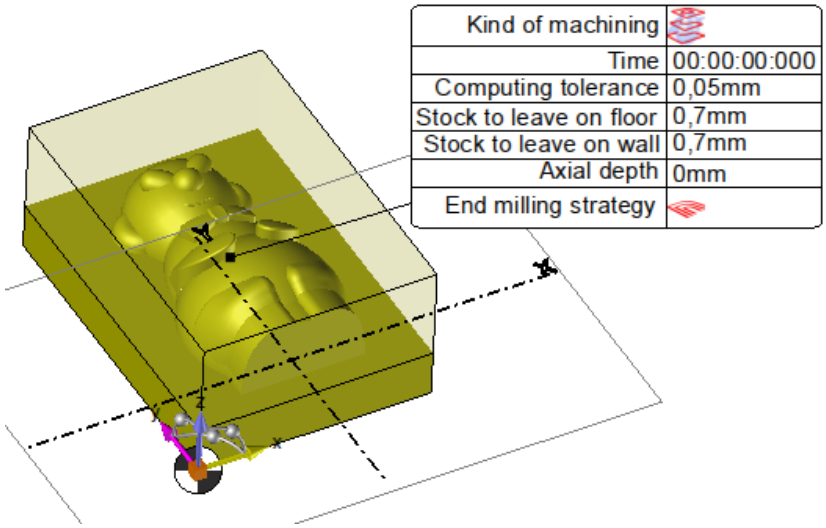


The color planes indicate the machining bounds, which means that the material will be machined within these limits.

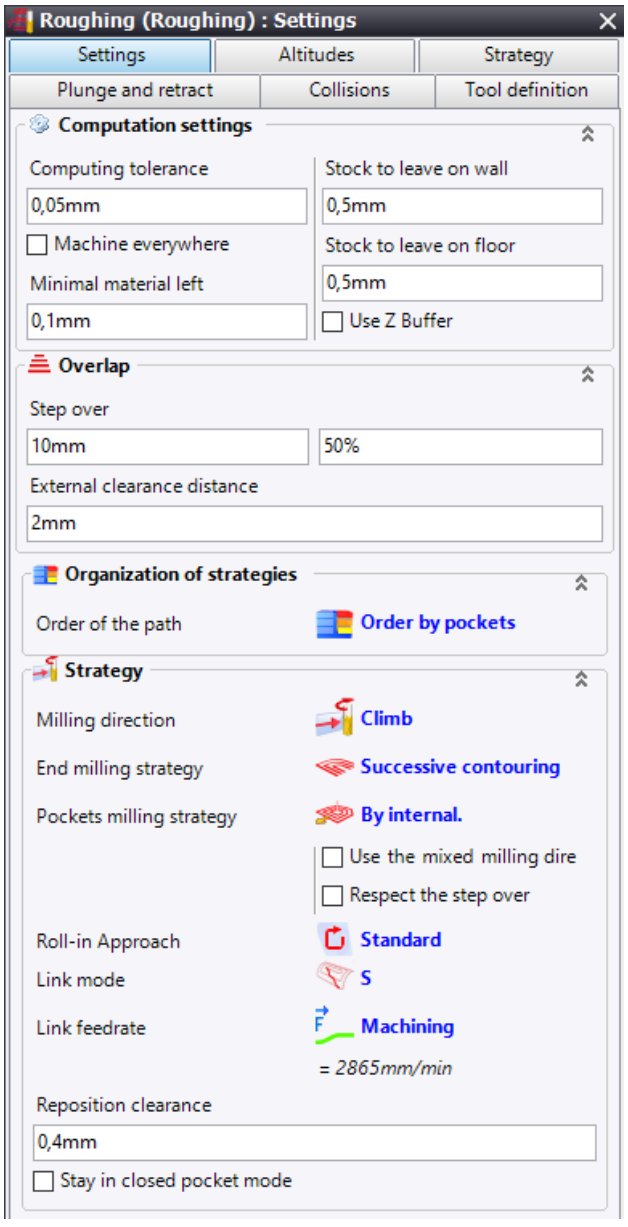


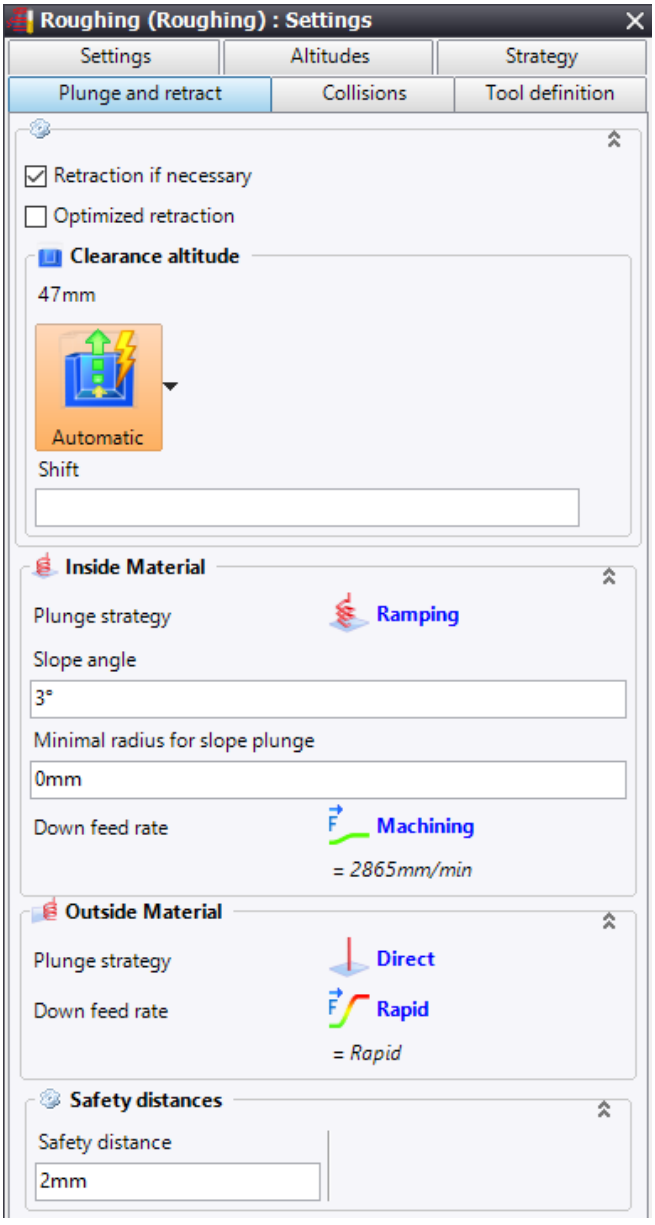
Note: You can hide the color planes by disabling the **Preview** icon at the top right of the **TopSolid** window.

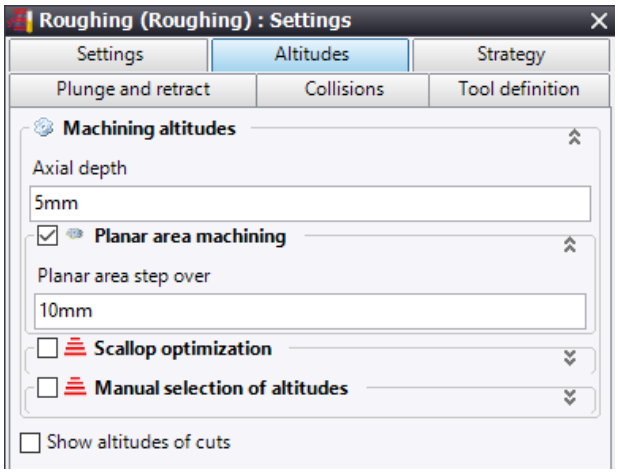
- **Preview** enabled: The color planes are visible as shown in the image above.
- **Preview** disabled: The color planes are invisible as shown in the image below.



- In the **Tool choice** option, create a *20mm* diameter **side mill** with a total length of *72mm* and an output distance of *50mm*.
- Click on the **Settings** icon and adjust the following settings:







- In the **Settings** tab:
  - **Computing tolerance**: *0.05mm*
  - **Minimal material left**: *0.1mm*
  - **Stock to leave on wall** and **Stock to leave on floor**: *0.5mm*
  - **External clearance distance**: *2mm*
  - **Step over**: *10mm*
- In the **Altitudes** tab:
  - Check the **Planar area machining** box.
  - **Planar area step over**: *10mm*
- In the **Plunge and retract** tab:
  - Check the **Retraction if necessary** box
  - Adjust the **inside material plunge strategy** to **Ramping** and the angle to *3°*

Note: It may take a while to update the stock, which is why it is not selected by default.

Roughing operations do not require an updated stock since the current stock is transparently calculated so that the remaining material to machine is known. The roughing operation analyzes all the previous active machining operations to know the actual shape of the stock at this point.

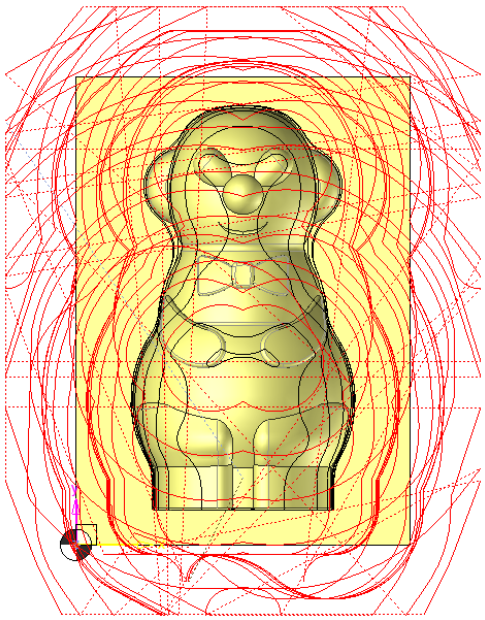
For 2D operations, if the stock is updated, the default start Z values will be correct. If not, they must be updated.

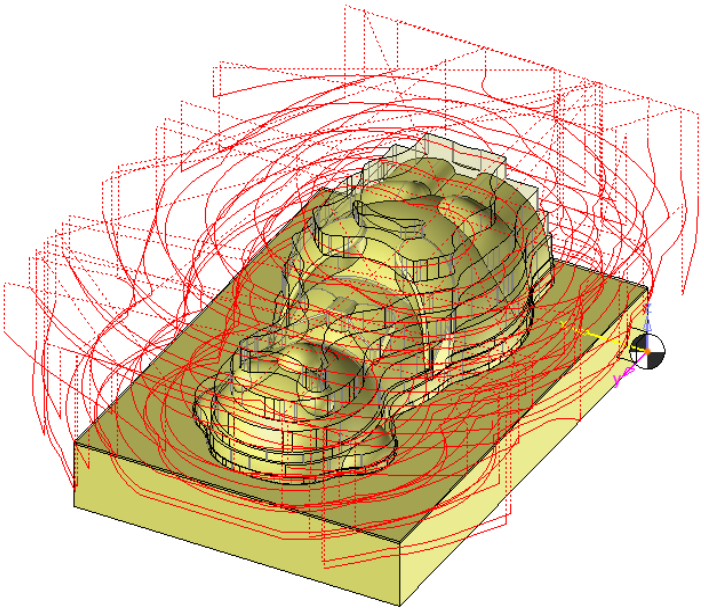
By default, the stock is not updated.

- In order for the operation to update the stock, click on the **Machine Dialog Flags** icon and select the **Update stock** option.

- Once all of these settings are defined, click on to **confirm** the roughing operation.

The machining previews should match the previews shown below.

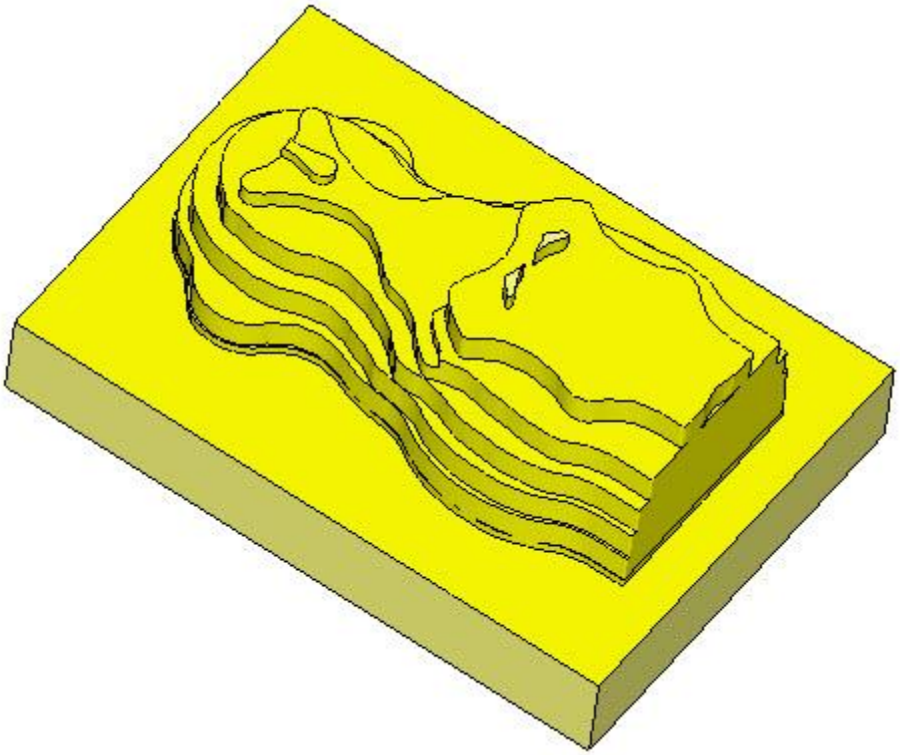




The verification module quickly captures a realistic image of what will actually happen on the machine. It is important to confirm this step when the stock is not updated. You can view the state of the stock after this roughing.

- To use this module, simply select the operations to view from the NC Operations tree, right-click on them and select the **Verify** command.
- Click on the **Start animation mode** icon.

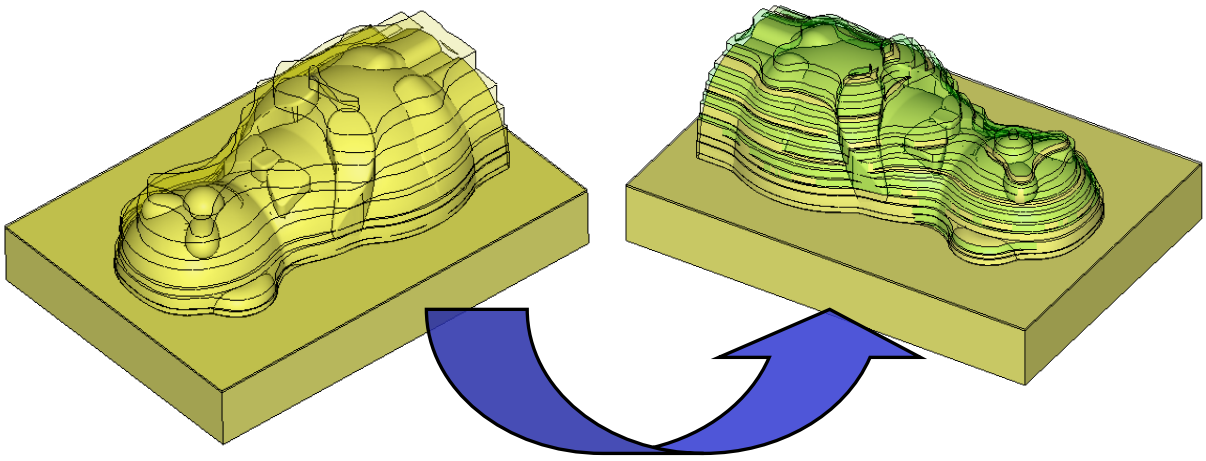
Note that you can modify the transparency by right-clicking on the part and moving the transparency cursor.



Note: The icon allows you to start the verification in **Turbo** mode. With this mode, response times are much faster.

- Click on the icon to exit the verification module.

Step 2 – Re-roughing



- Open the *Nounours_Step_2* machining document.

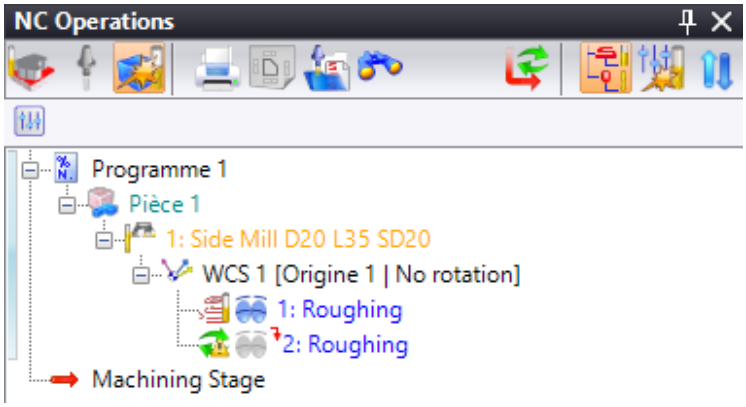
The diameter of the previously used mill was relatively large with respect to the size of the part to be machined. It therefore left areas in which the remaining amount of material was too large to be removed only by the finishing tool. We will perform a re-roughing operation that consists of a new roughing operation performed with a smaller diameter tool.

As you saw at the start of this chapter, a roughing takes into account the current state of the stock (even though it is not up-to-date) in order to determine the areas to be machined. The re-roughing command is therefore the same as the roughing command.

It may be worthwhile to copy the roughing that has already been calculated so that you only have to modify a few settings (such as the tool used). This is easily done from the NC Operations tree.

- Select the first roughing operation.
- Simultaneously hold down the **Ctrl** and **C** keys.
- Simultaneously hold down the **Ctrl** and **V** keys in the location where you want to copy (in this case, at the end).

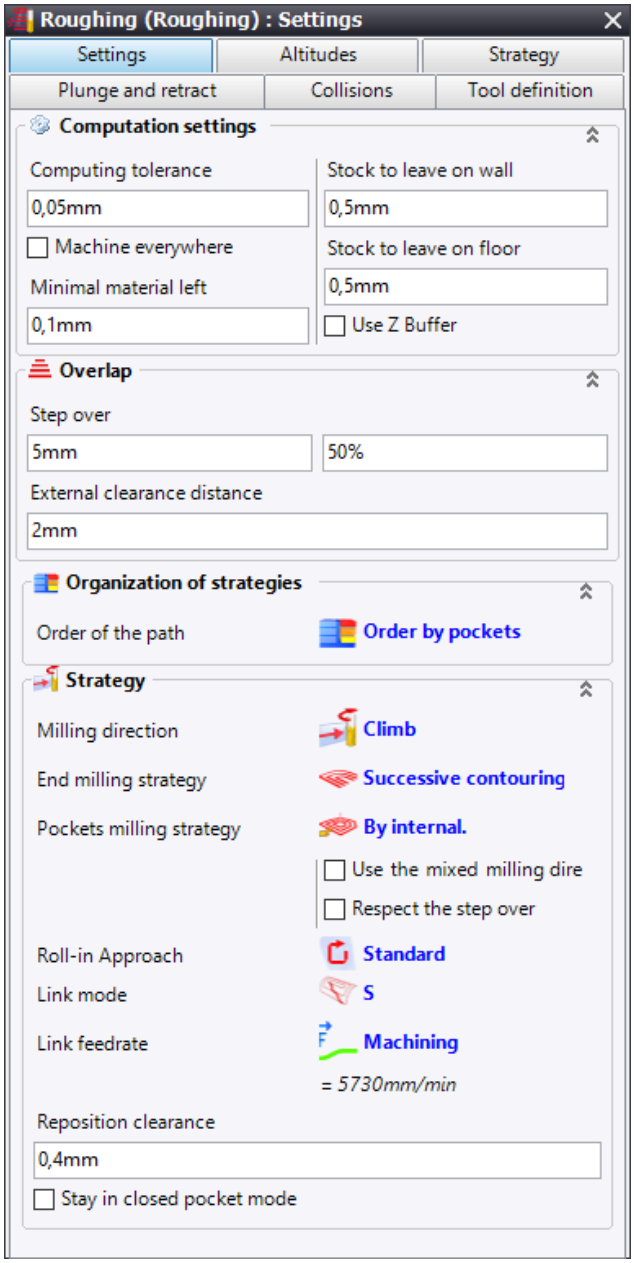
Your screen should look like the following image:

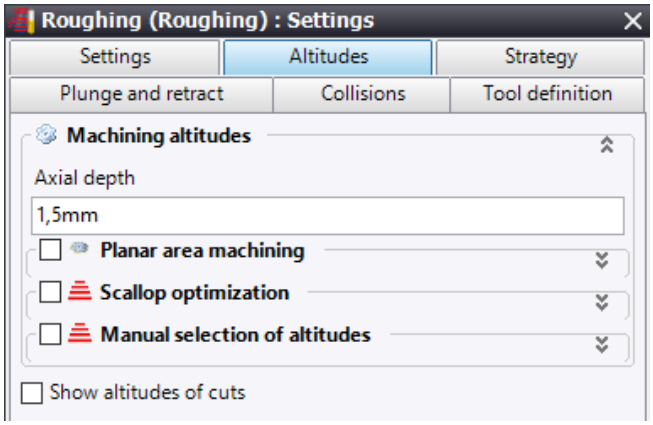


A new operation is then created.

- Double-click on the new operation (operation 2) to edit it.
- Click on the **Tool choice** icon, then create a *10mm* diameter **side mill** with a total length of *72mm* and an output distance of *40mm*.

- In the **Settings** option, adjust the following settings.



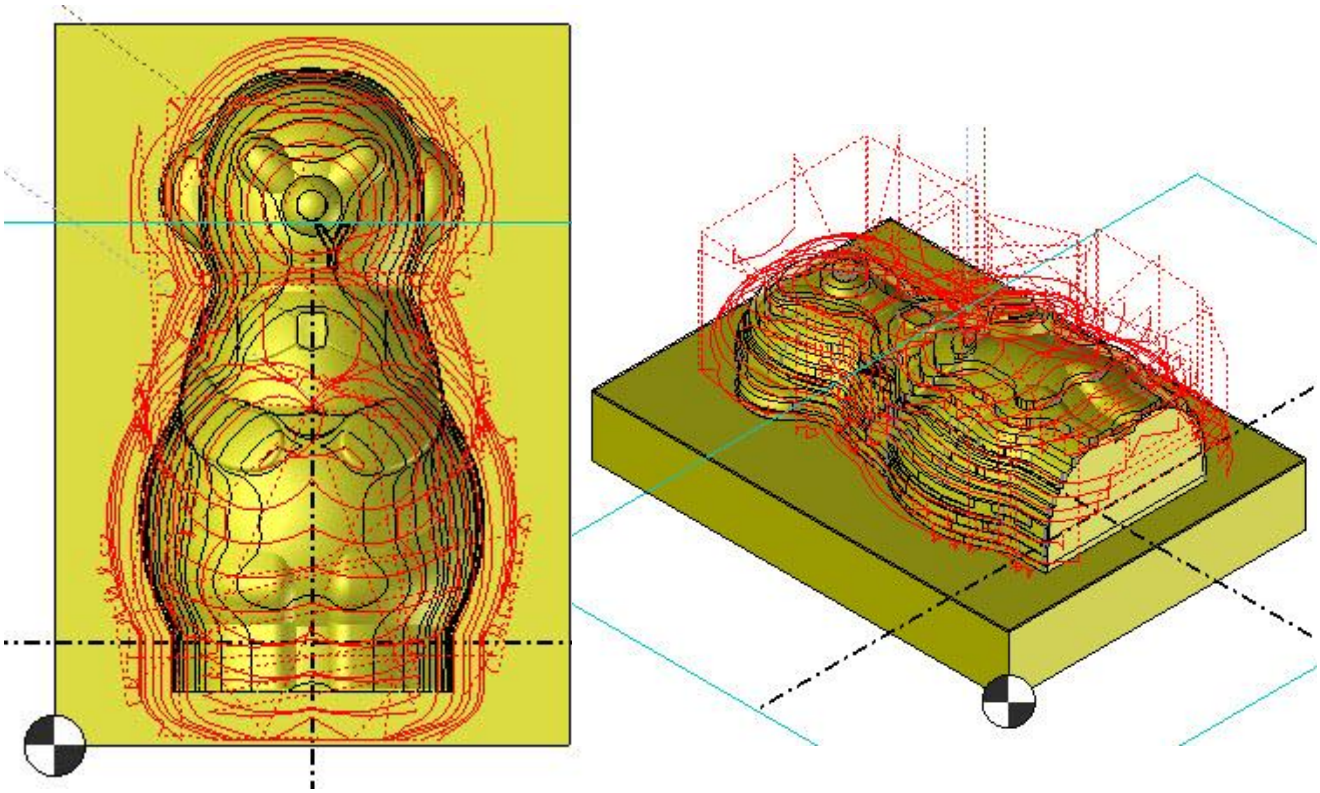


- In the **Settings** tab:
  - **Step over**: *5mm*
- In the **Altitudes** tab:
  - **Axial depth**: *1.5mm*
  - Uncheck the **Planar area machining** box

Note: It is no longer necessary to machine the planar faces since it was already done for the first roughing operation.

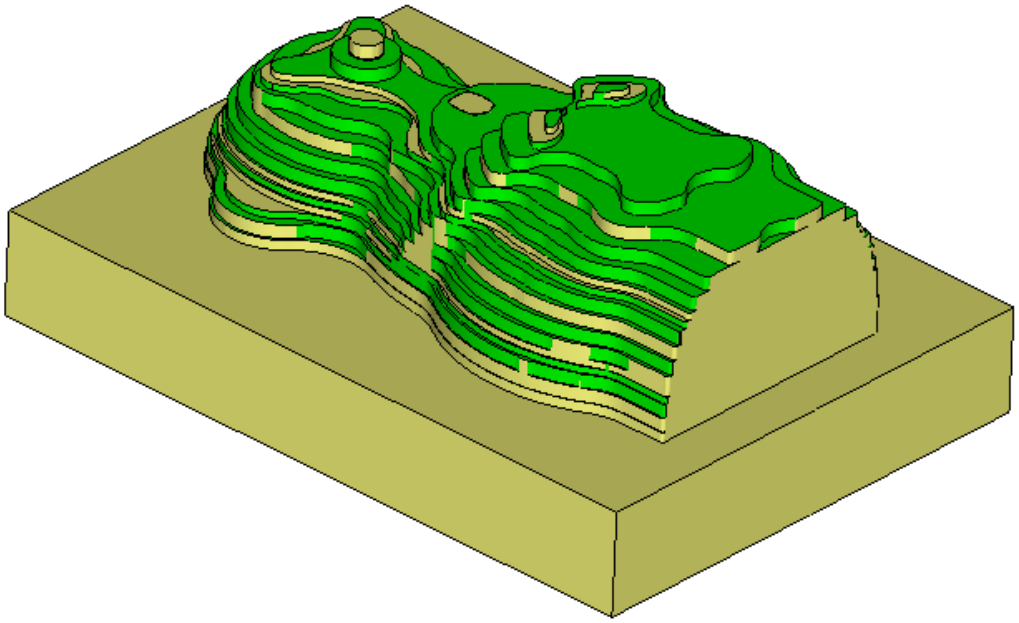
- Click on to **confirm** the operation.

The machining previews should match the previews shown below.



As before, you can check the result by using the verification module.

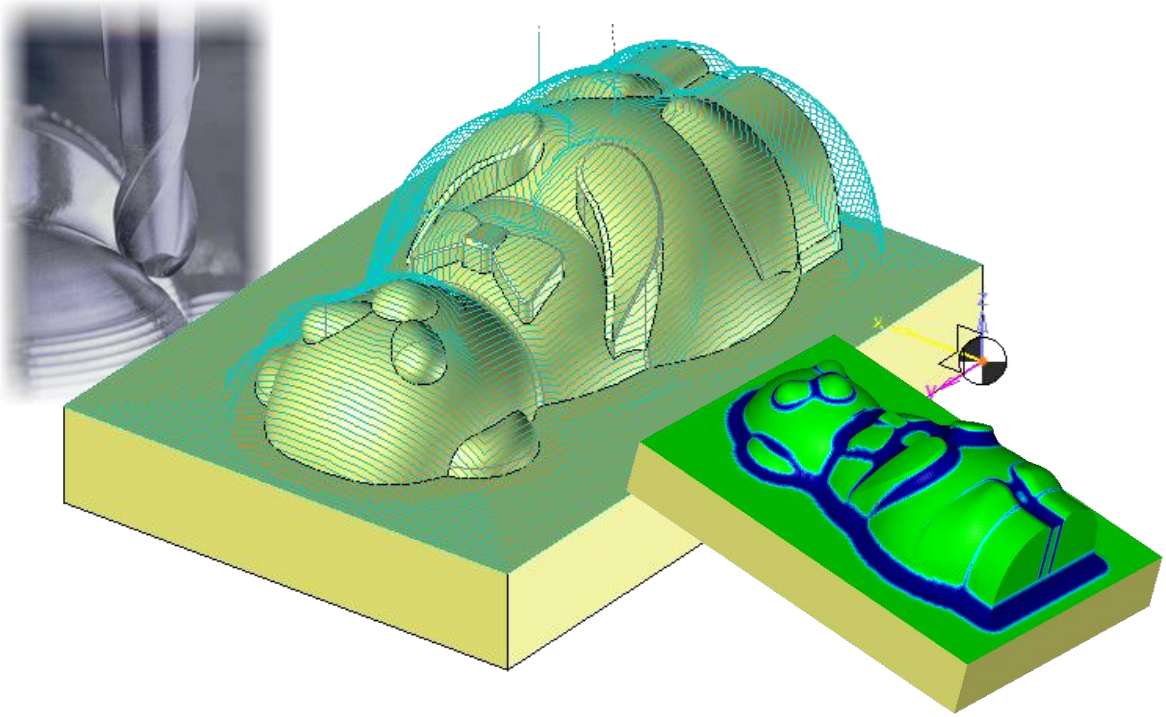
You can see in the image below that the finishing tool (shown in green) machined the affected areas.



Step 3 – Finishing

- Open the *Nounours_Step_3* machining document.

For this step, we will finish the machining by milling the shape using a 12mm diameter ball nose mill.



- Right-click in the graphics area and select the **Finishing** command.

The following label is displayed.

Kind of machining	
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave on floor	0mm
Stock to leave on wall	0mm
Step over	0,5mm
Scallop height	0,0125mm
Sweeping angle	0°

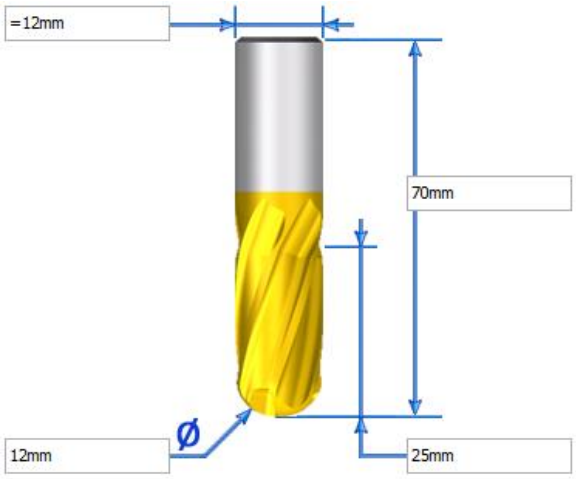
- Double-click on the icon, which indicates the type of 3D finishing, then select a **Constant step-over** finishing.

Kind of mach	
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave on floor	0mm
Stock to leave on wall	0mm
Step over	0mm
Scallop height	0mm

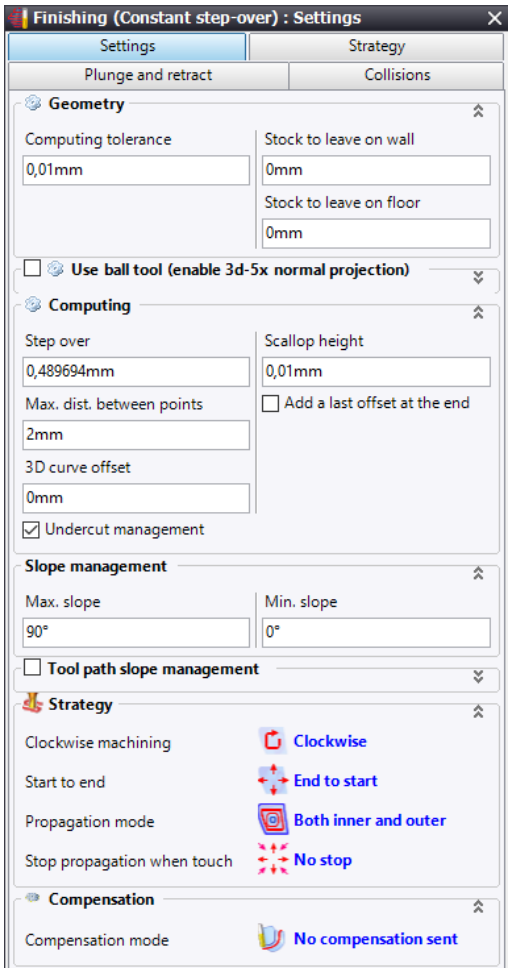
The following label should be displayed:

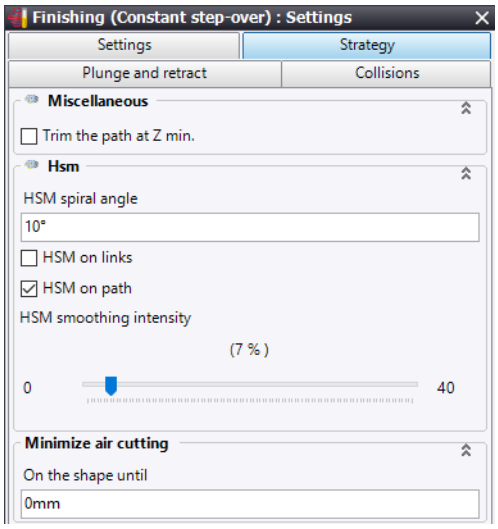
Kind of machining	
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave on floor	0mm
Stock to leave on wall	0mm
Step over	0mm
Scallop height	0mm

- In the **Tool choice** option, create a *12mm* diameter **ball nose mill** with an output distance of *50mm*.



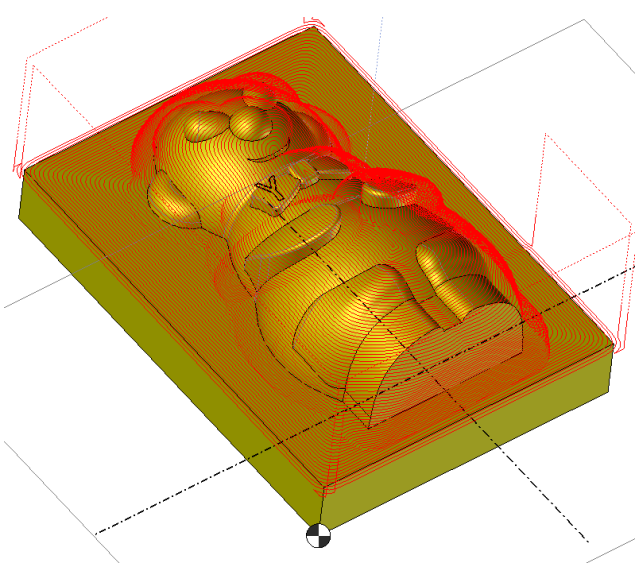
- In the **Settings** option, adjust the following settings.

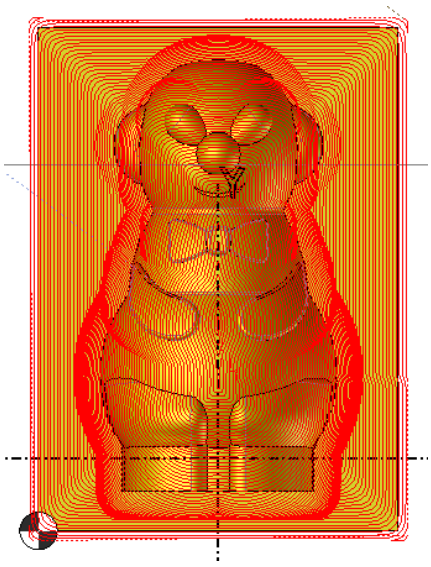




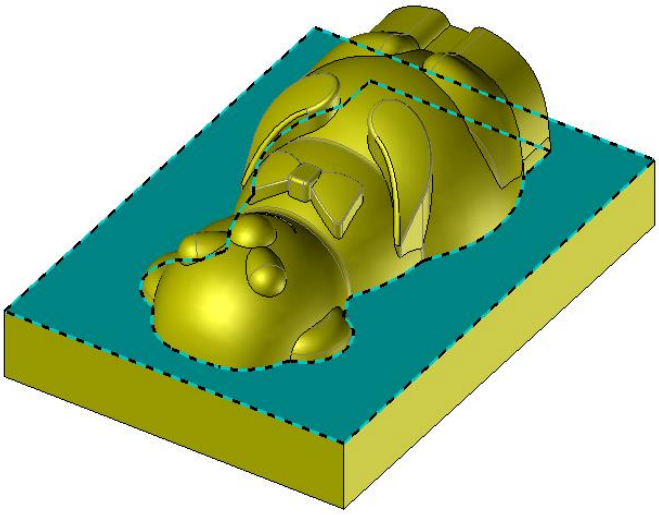
- In the **Settings** tab:
  - **Scallop height** and **Computing tolerance**: *0.01mm*
  - **Start to end**: Select **End to start**
  - **Max. dist. between points**: *2mm*
- In the **Strategy** tab:
  - **HSM spiral angle**: *10°*
  - Uncheck the **HSM on links** box
  - Check the **HSM on path** box
- Click on to **confirm** the finishing operation.

As before, tool paths can be reviewed which, for this operation, are as follows:





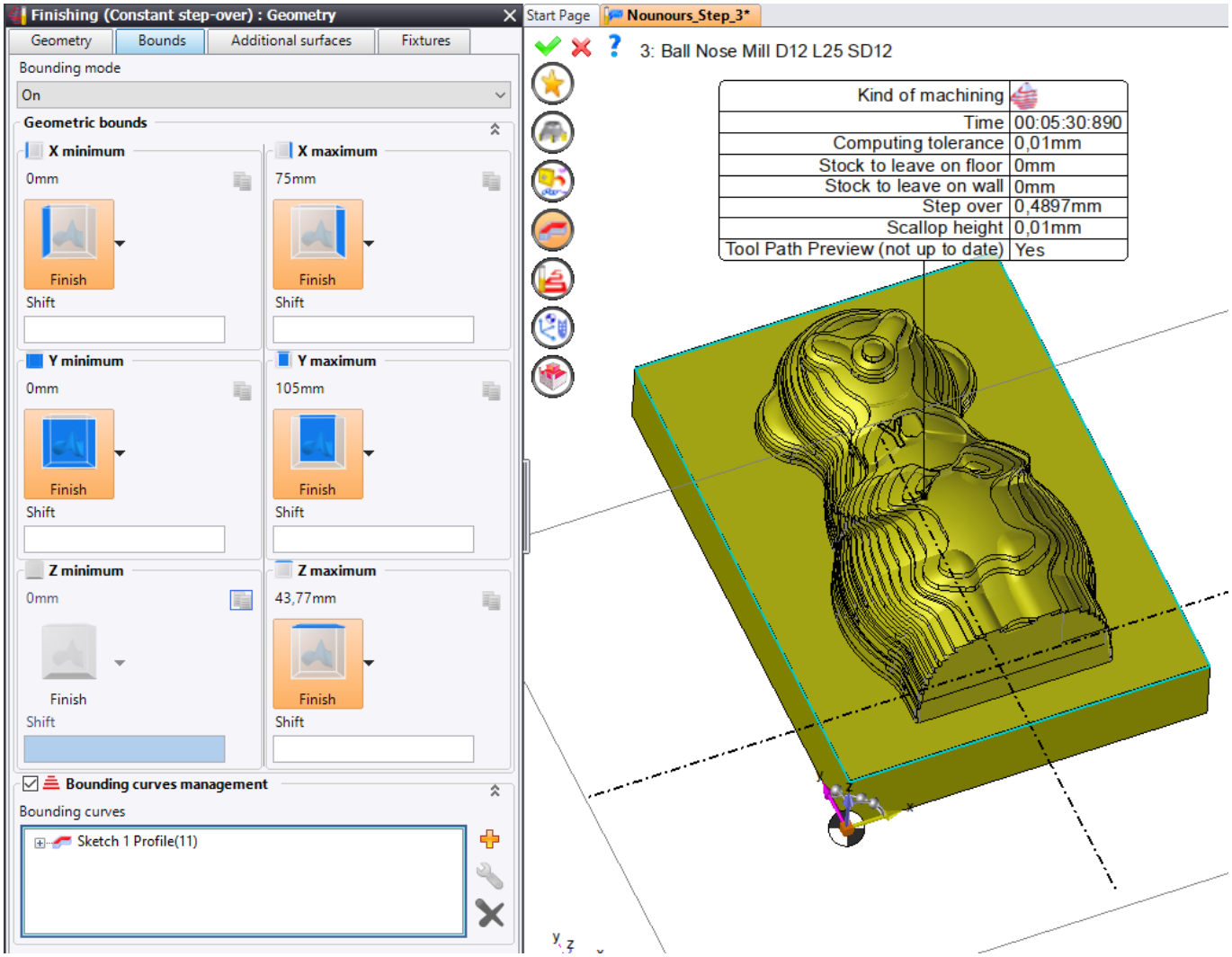
We will now trim the finishing pass above the upper plane. We could use the Z minimum, but for this example, we will see how we can also trim the machining with a sketch.



- Click on the icon at the top left of the screen to switch to design mode.

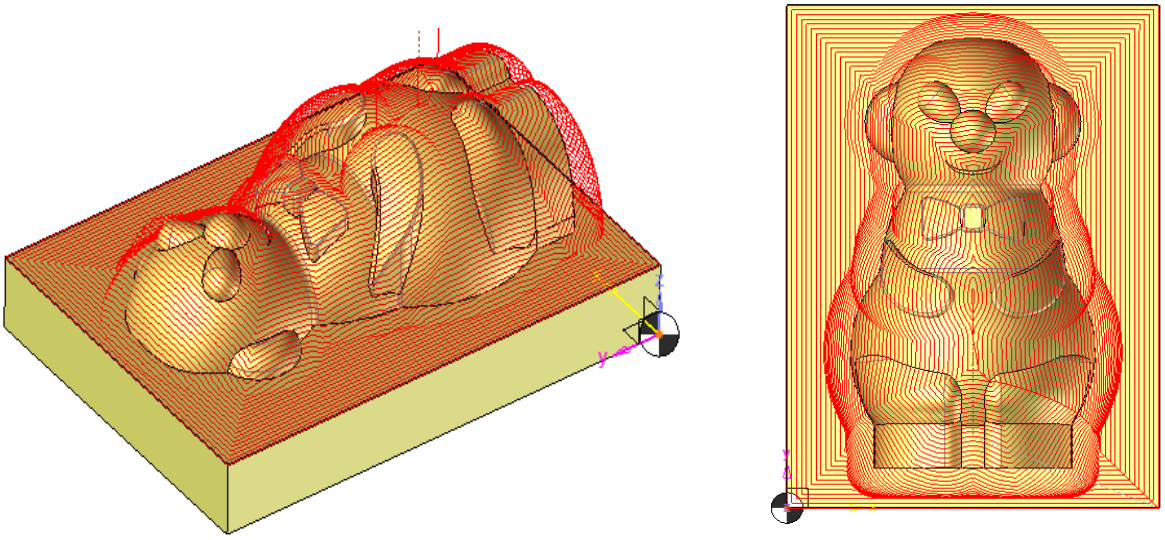
Note that you can also use the **Ctrl** + **W** default shortcut to switch between the machining and the design modes, and conversely.

- Right-click on the blue-colored plane as shown on the previous page and select the **Sketch** command.
- **Project** the outer contour of the blue-colored surface.
- **Confirm** the sketch by clicking on the Sketch 1 button.
- Switch back to machining mode by clicking on the icon in the system bar or using the **Ctrl** + **W** shortcut.
- **Edit** the finishing operation.
- Click on the **Geometry** icon.
- In the **Bounds** tab, adjust the following settings:
  - **Bounding mode**: Select **On**
  - Make sure that the **Bounding curves management** box is checked.
  - Select the sketch as the bounding curve.

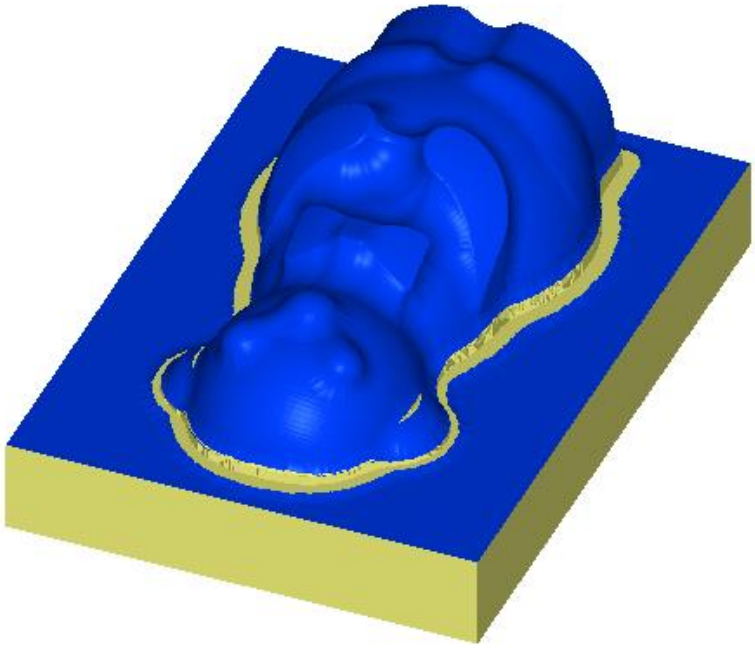


- Click on to **confirm** the finishing operation.

You should end up with the following result:



As a reminder, the verification controls the removed material. Therefore, you can see that the ball nose mill machined a large amount of material and thus finished the part in several areas (shown in blue below).

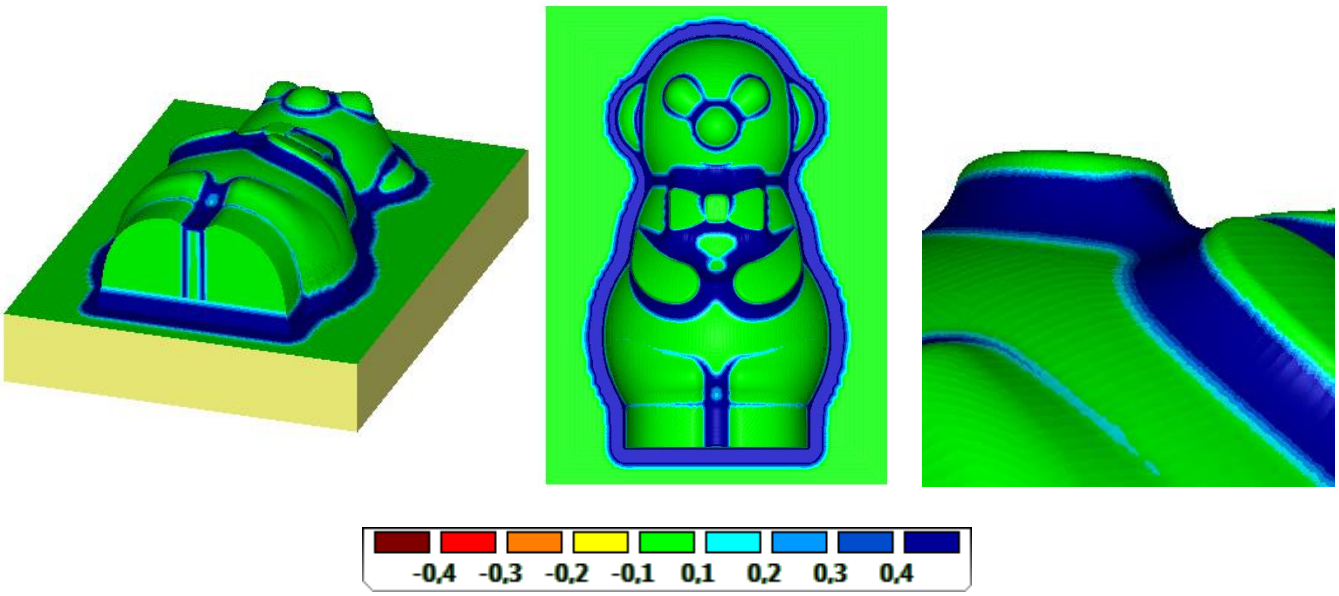


However, some material remains to be machined because the diameter of the finishing tool was not able to pass over the entire part.

You can request that the verification module clearly indicates the unmachined areas by comparing the current stock to the theoretical part to be machined. This also indicates whether a problem occurred.

- Always in the verification context, right-click in the graphics area and select the **Display comparison** command, or click on the icon at the top left of the screen.

You should end up with the following result:

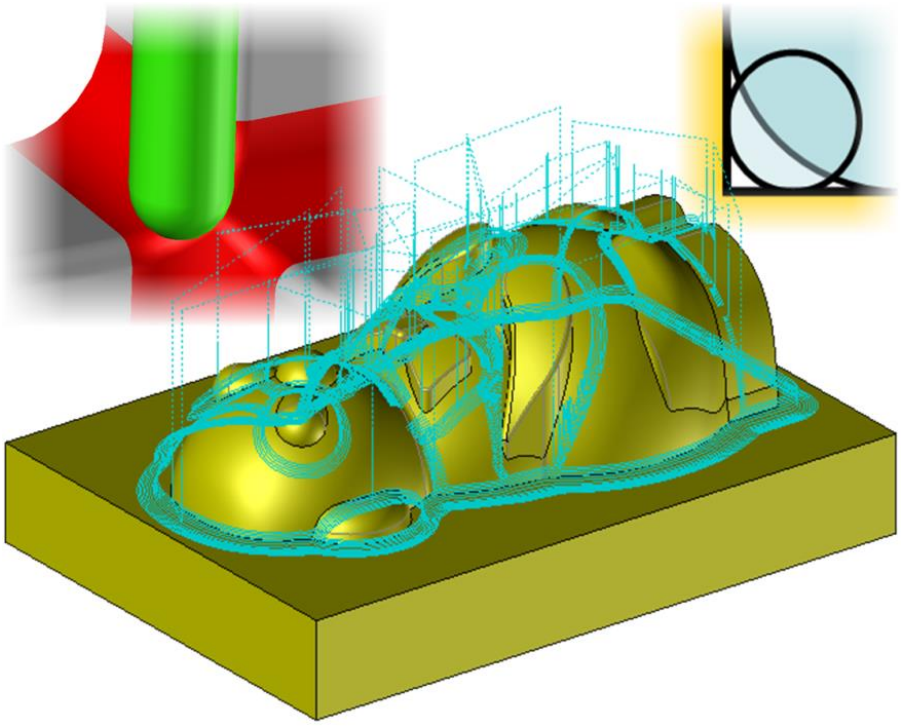


As you can see, the closer the color is to blue, the more the material remains to be machined. In this case, it is clear that there is a lot of material left to be machined. We will therefore perform a left material machining using a smaller radius.

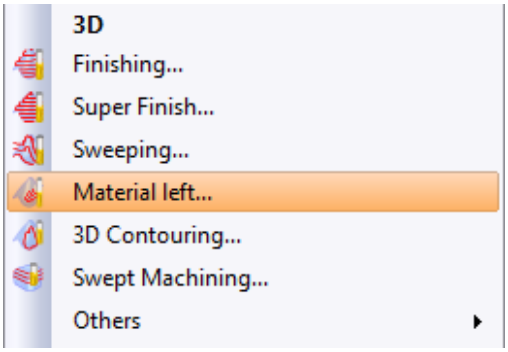
Step 4 – Re-finishing

- Open the *Nounours_Step_4* machining document.

For this step, we will machine the areas that were not machined in the previous operation because the tool diameter was too large. We will therefore redo the finishing using a reference tool. The reference tool will be the one that was used in the previous step.



- Right-click in the graphics area and select the **Material left** command.



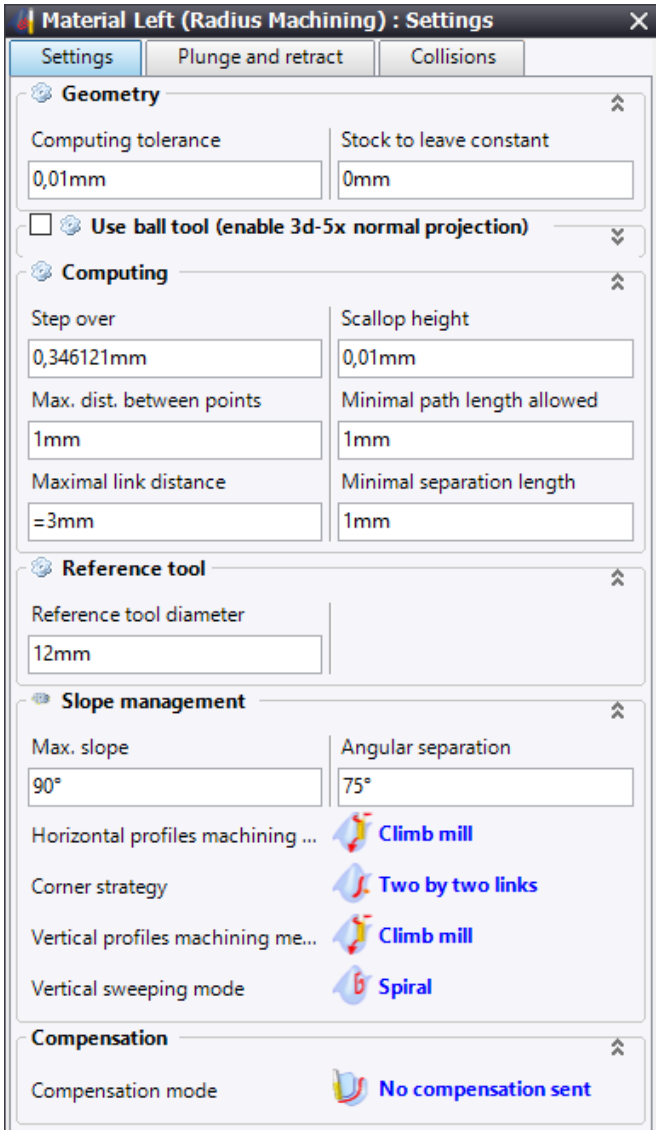
The following label is displayed.

Kind of machining	
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave constant	0mm
Axial depth	0,6922mm
Scallop height	0,02mm
Reference tool diameter	10mm
Reference tool corner radius	5mm

- Double-click on the icon, which indicates a constant Z left material machining, then select **Radius machining**.

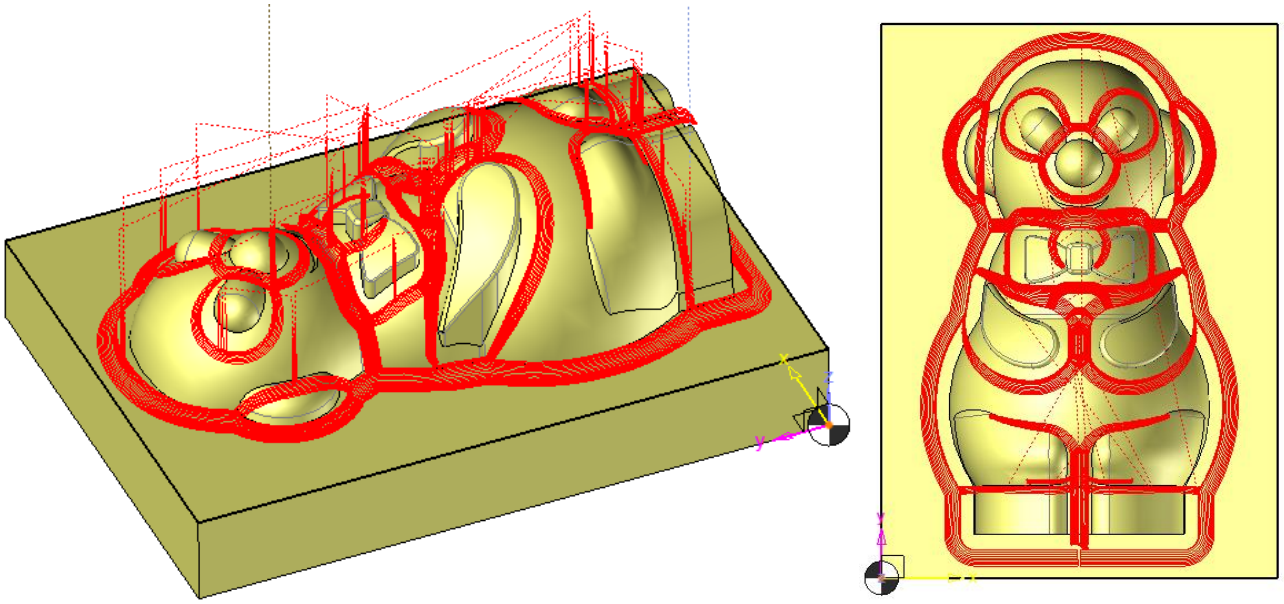
Kind of machinin	
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave constant	0mm
Step over	0,6922mm
Scallop height	0,02mm
Reference tool diameter	10mm

- In the **Tool choice** option, create a *6mm* diameter **ball nose mill**.
- In the **Settings** option, adjust the following settings:
  - Adjust the **computing tolerance** and the **scallop height** to *0.01mm*.
  - Since the reference tool in the previous step was a 12mm diameter ball nose mill, adjust the **reference tool diameter** to *12mm*.
  - Adjust the **angular separation** to *75°*.

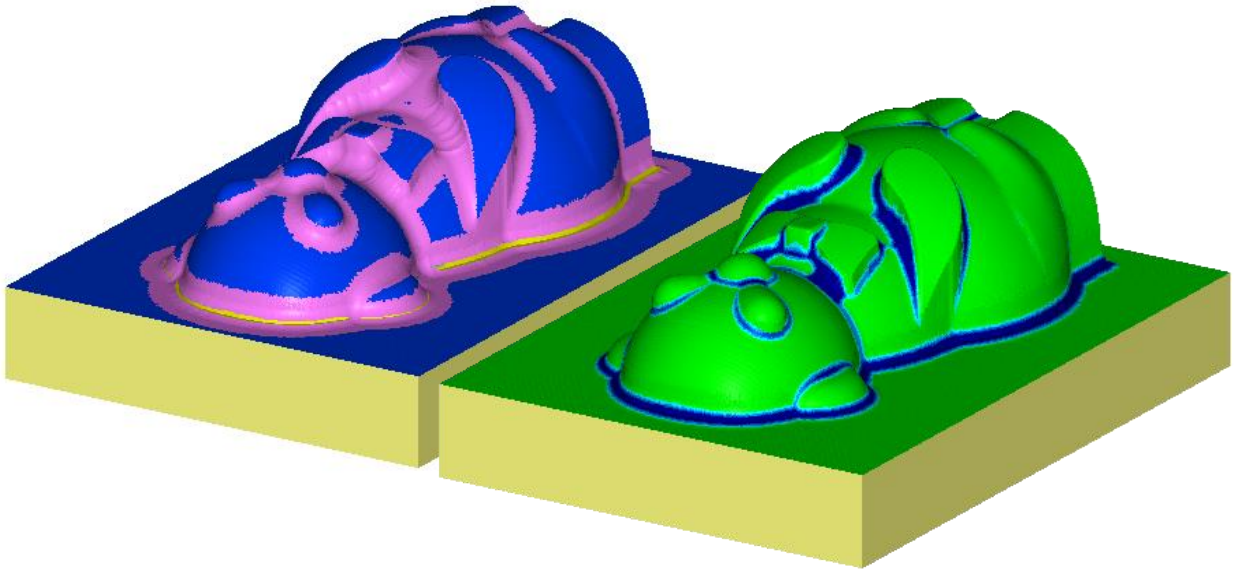


- Click on to **confirm** the material left operation. During calculation, information is displayed at the bottom left of the **TopSolid** window.

As in previous steps, we will verify the machining and tool paths. The stock is updated in machining operations 1 and 2, so we have to select machining operations 3 and 4 before launching the verification.



Verification and comparison:



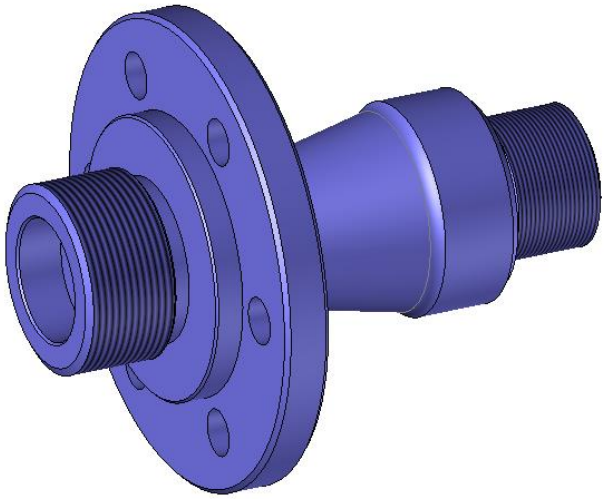
As you can see here, some material remains to be machined. You must therefore repeat the material left operation by reducing the diameter of the mill. For the parting surface, in order to keep the sharp angle, you can use a side mill to contour the shape (see 2D chapter).

The purpose of this chapter is to introduce you to **TopSolid 7**'s Cam module and present its range of 3D features. By no means are the order of operations, choice of tools, or operation settings optimized or defined for industrial use.

Turning

Introduction

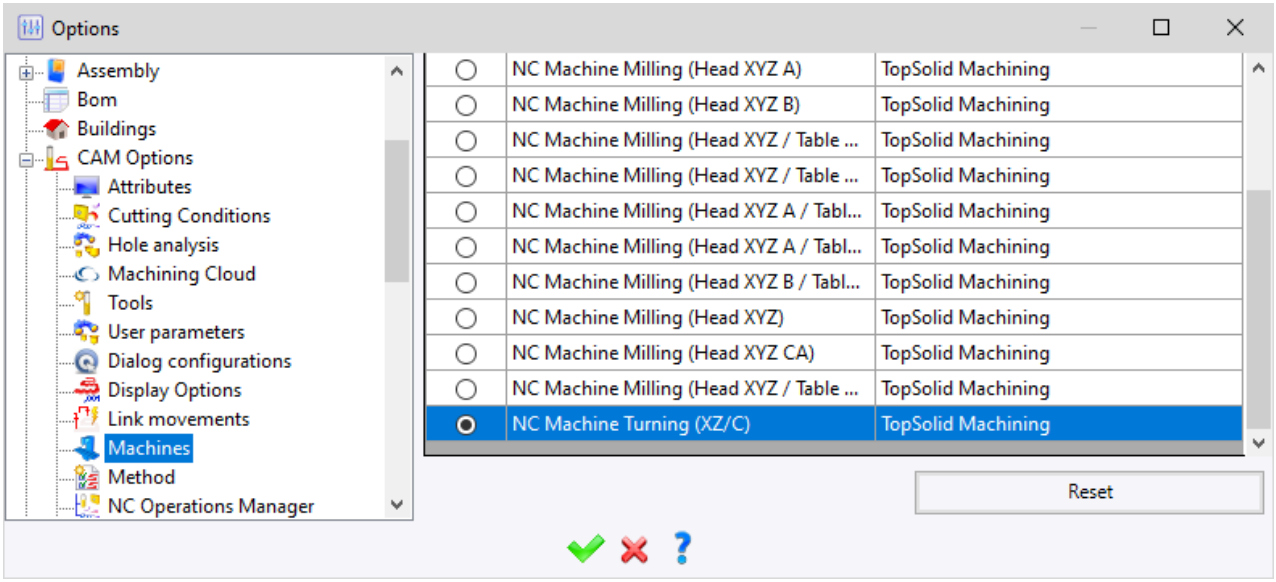
The purpose of this chapter is to create a machining program for the following part.



As you can see, the part must be positioned on a turning machine in order to be machined.

Warning: If the default machine is not a turning machine, right-clicking on the design document and selecting the **Machining** command will not be suitable.

- In this case, select the **Options** command from the **Tools** tab.
- Expand the **CAM Options** menu, go to the **Machines** section and select the **NC Machine Turning (XZ/C)** machine.



To help you become familiar with **TopSolid 7**, the machining of the part above is divided into several steps. For each step, a machining document is available in which the operations have been completed. You can refer to these documents if you run into any difficulties. The machining steps are numbered from 1 to 7, so there will be seven numbered machining documents.
These machining documents are provided in a TopPkg file named *TopSolidCam_ArbreCourt.TopPkg* which is installed in the "Program Files\TOPSOLID\TopSolid 7.14\Samples" folder. In order to follow the tutorial, we recommend that you import this file into a new project.

- Create a  **new project** named *Turning* using a **blank template**.
-  **Import** the *TopSolidCam_ArbreCourt.TopPkg* package in your project.

The part was designed in **TopSolid**. The designer used the "features" technology, which allows major automation of machinings.

The overall programming approach is as follows:

1. Defining the stock part
2. Positioning and defining the part (as per **TopSolid'Cam**) and its program origin
3. Defining the operations with integrated stock management
4. Simulation
5. Editing the ISO file

Steps 1 and 2 are explained in the prerequisite tutorial (creating a machining setup document and machining document). We will start at Step 3, which consists of defining the machining operations.

As you have previously seen, several methods are available for creating a machining document. You can therefore start the machining by selecting the method of your choice. For this chapter, method 5 will be selected, which consists of machining an assembled stock and finish.

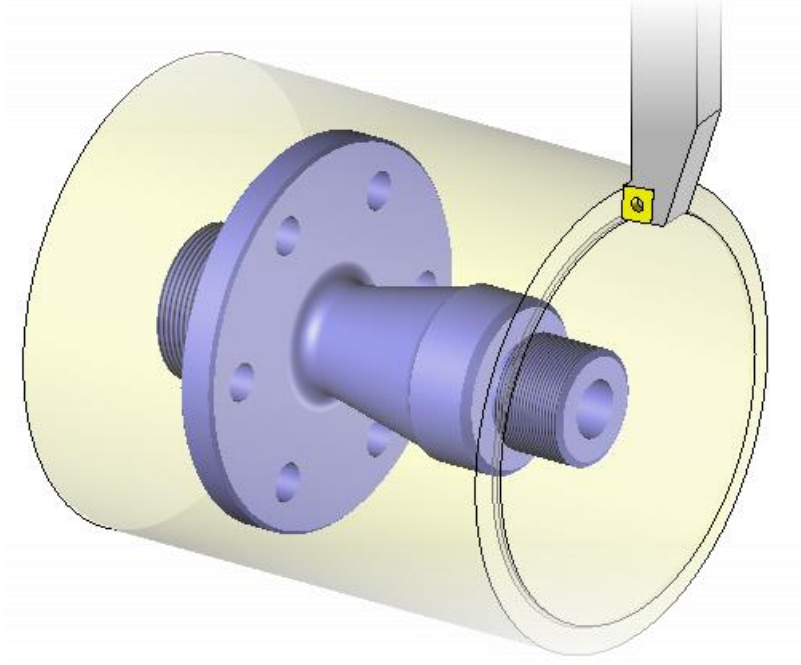
In this chapter, we will perform various basic turning operations, including:

- Facing
- Turning
- Groove roughing
- Groove machining
- Threading
- Hole machining

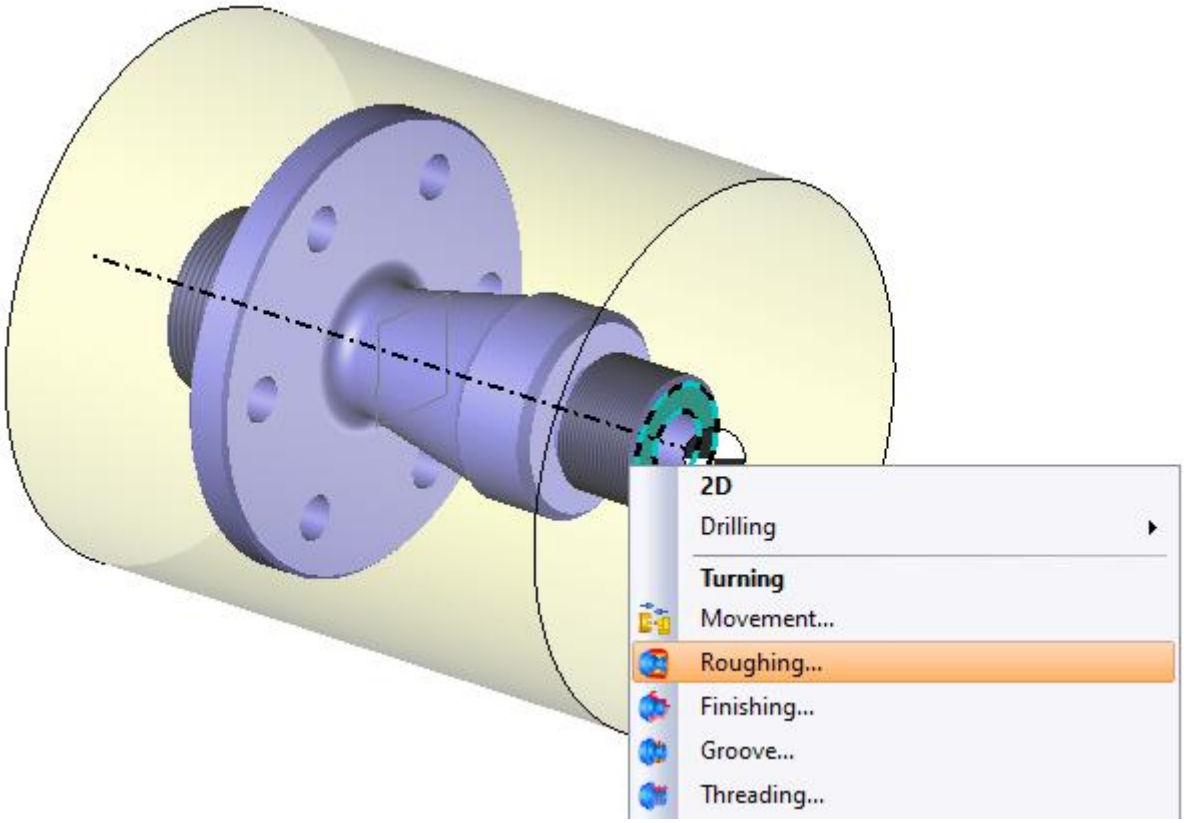
Step 1 – Facing the end of the part

- From the Project tree, open the *Arbre_Court_1* machining document.

During this step, we will machine the right end of the part. We will plunge into the material with a facing tool oriented along the X axis.

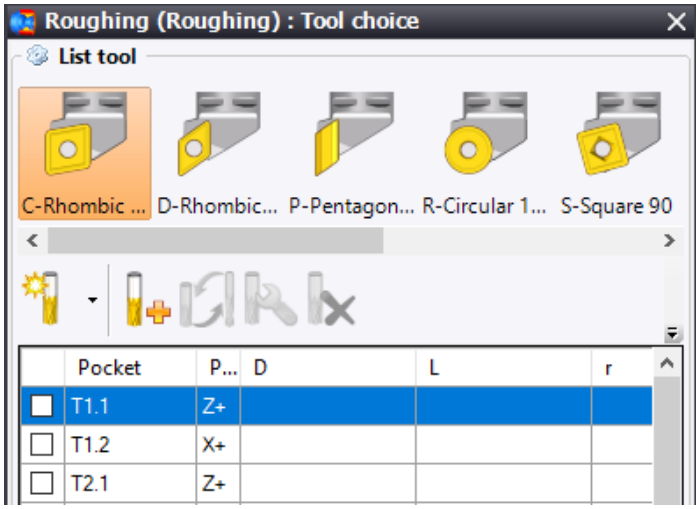


- Select the extreme right face shown in blue below.
- Right-click in the graphics area and select the **Roughing** command.

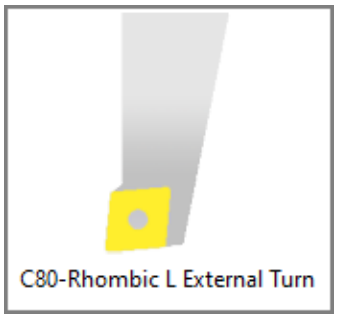


- Click on the **Tool choice** icon.

The following dialog box appears:

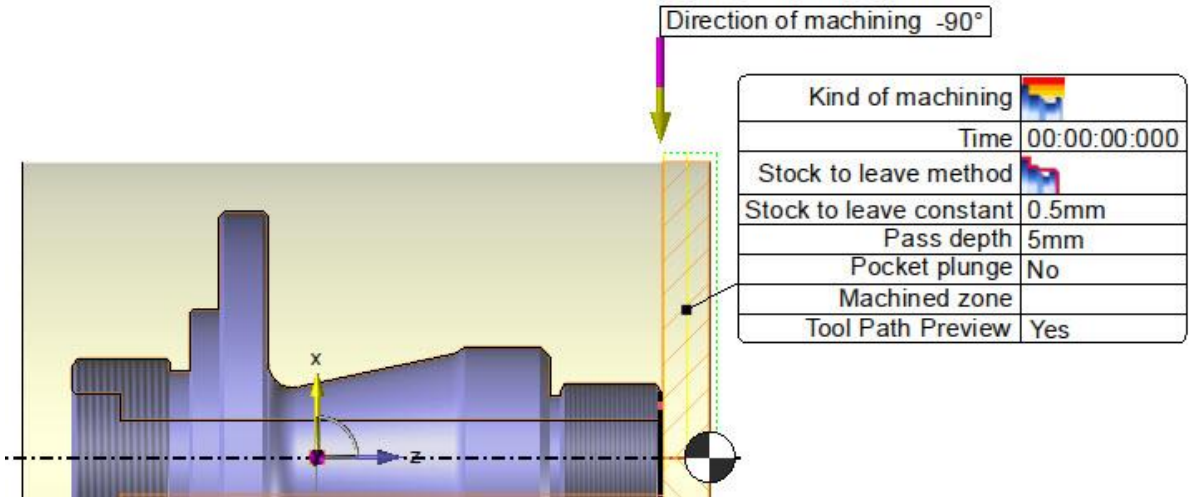


- Select an empty pocket in **X+**. If you select Z+, the tool will not be oriented properly.
- Double-click on the **C-Rhombic 80** thumbnail.
- In the **Template(s)** tab, double-click on **C80-Rhombic L External Turn**.



- Click on to **confirm** the tool.

The default machining direction is based on the first selected face. The default value will be +90° or -90° or 0° or 180°.



Since the pass depth is too large, you must modify its value.

- On the label, change the values as follows:
  - **Stock to leave constant** = *0.3mm*
  - **Pass depth** = *2mm*

Kind of machining	
Time	00:00:00:000
Stock to leave method	
Stock to leave constant	0,3mm
Pass depth	2mm
Pocket plunge	No
Machined zone	
Tool Path Preview	Yes

- Click on to **confirm** the operation.

If you want to simulate the machining operation, you only have to click on the operation icon in the NC Operations tree, or right-click on the desired operation and select the **Simulate** command.

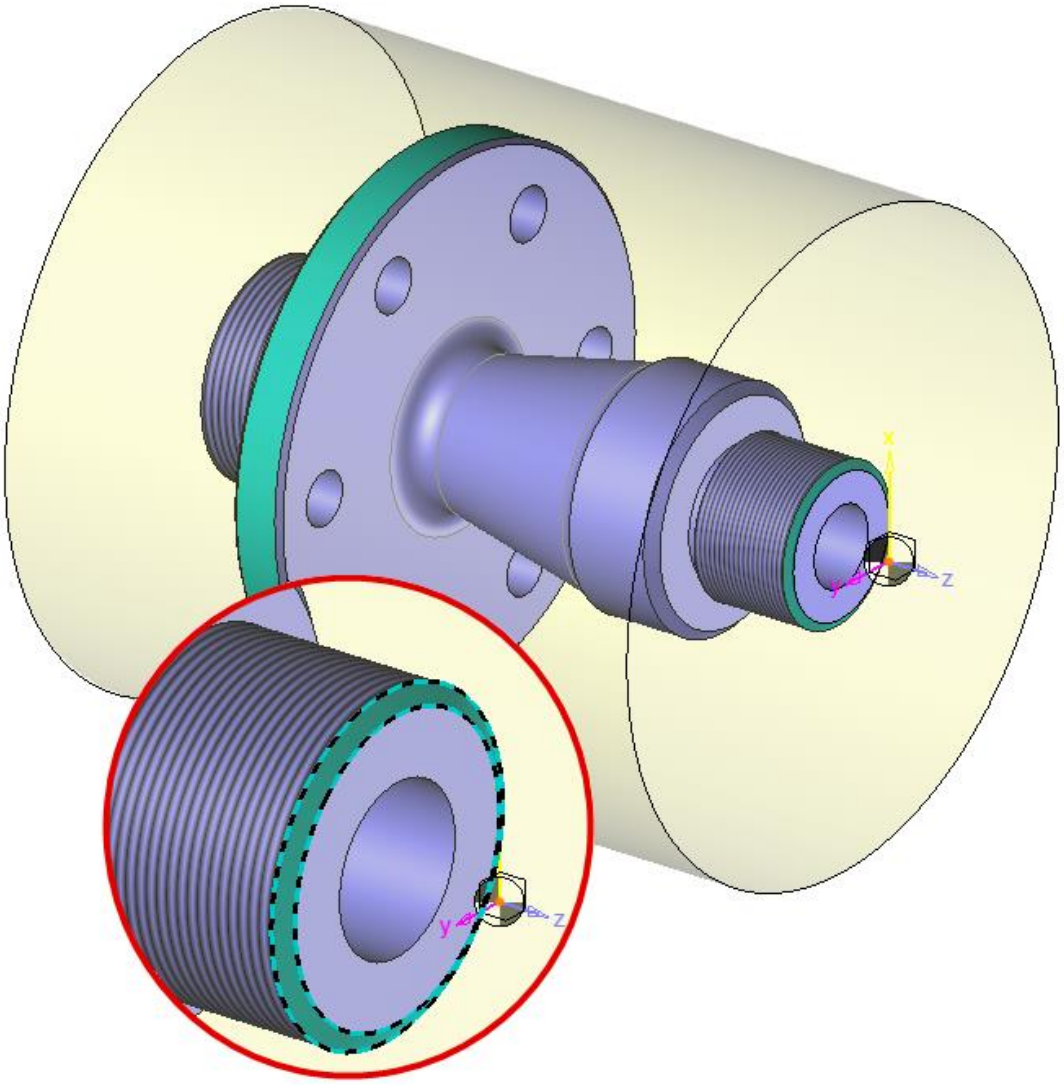
- **Save** the document.

Step 2 – Turning

- From the Project tree, open the *Arbre_Court_2* machining document.

For this step, we will execute the same paraxial roughing operation as before, but this time the tool path will be oriented along the Z axis and not along the Y axis.

- Click on one of the blue-colored faces as shown below, then hold down the **Alt** key and click on the second face.



Note: The **Alt** key allows you to select all the faces between the two selected faces (included).

- Right-click in the graphics area and select the **Roughing** command.

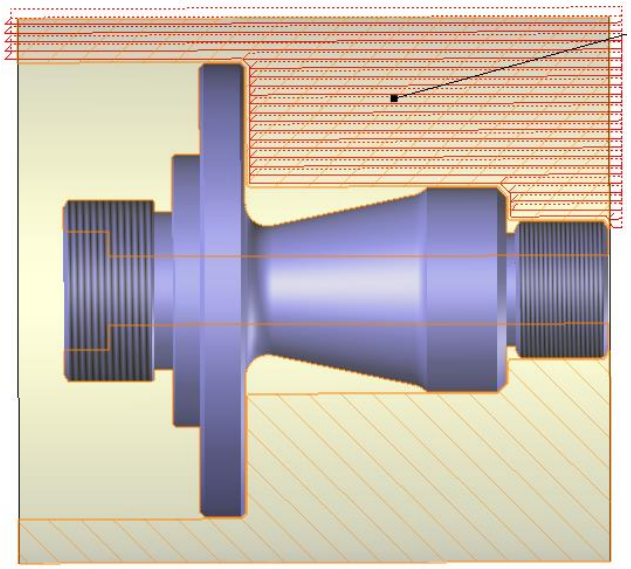
The default machining direction is 180° and the previously chosen tool is still selected.

- On the label, change the **Stock to leave constant** and **Pass depth** values like you did before to obtain the following result:

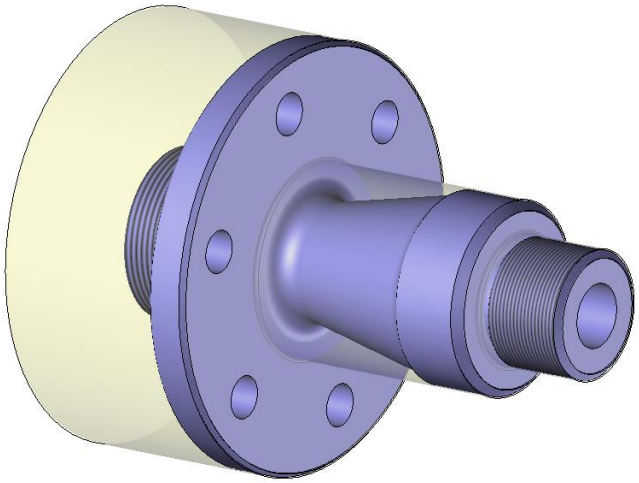
Kind of machining	
Time	00:00:00:000
Stock to leave method	
Stock to leave constant	0,3mm
Pass depth	2mm
Pocket plunge	No
Machined zone	
Tool Path Preview	Yes

- Click on to **confirm** the o>peration.

The part preview should be as follows:



Tool path



Machined part

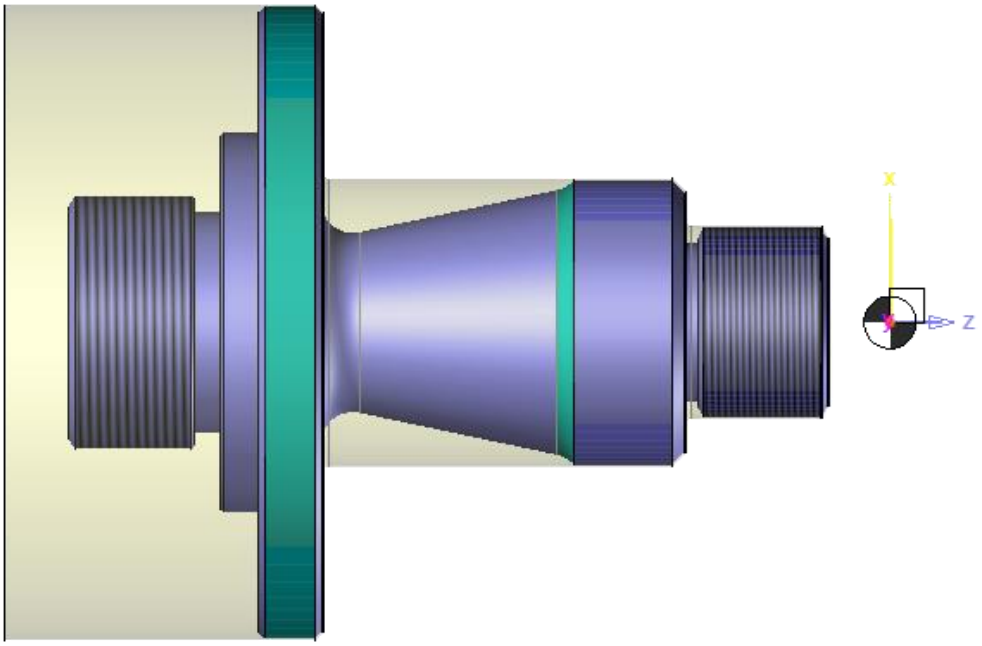
- **Save** the document.

Step 3 – Groove roughing

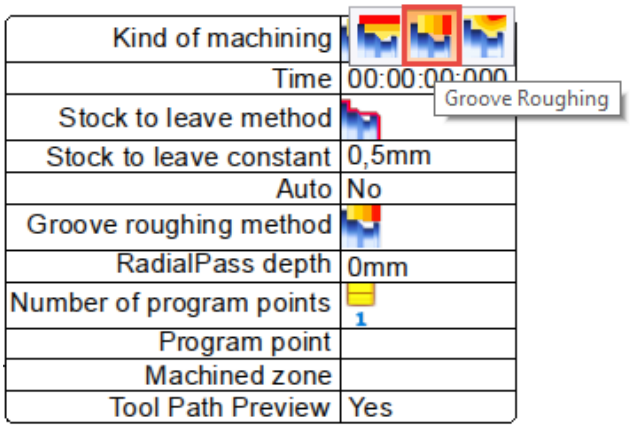
- From the Project tree, open the *Arbre_Court_3* machining document.

For this step, we will machine the cone-shaped section of the shaft using the groove roughing technique, i.e. machine in the direction of the tool axis.

- Hold down the **Alt** key and select the two blue-colored faces as shown below.

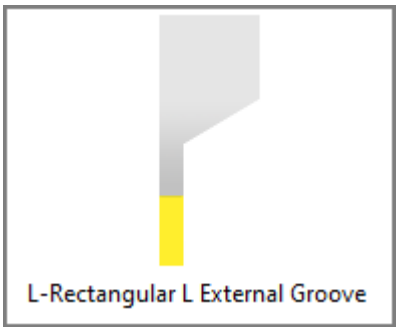


- Right-click in the graphics area and select the **Roughing** command.
- On the label, double-click on the **Kind of machining** icon and select **Groove Roughing**.

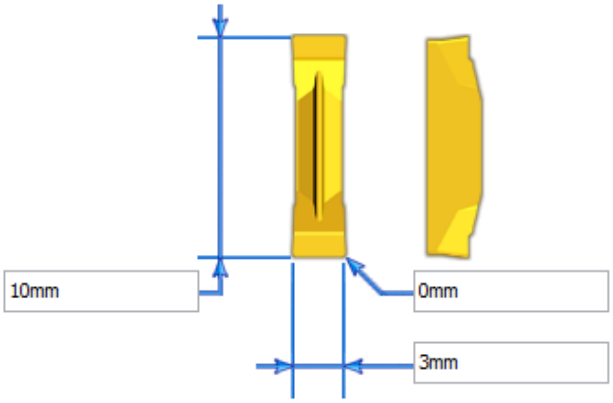


- In the **Tool Choice** option, select an empty pocket in **X+**.
- Double-click on the L-Rectangu... **L-Rectangular** thumbnail.

- In the **Template(s)** tab, double-click on **L-Rectangular L External Groove**.



- In the **Component(s)** tab, adjust the tool width to *3mm* and the radius to *0mm*.



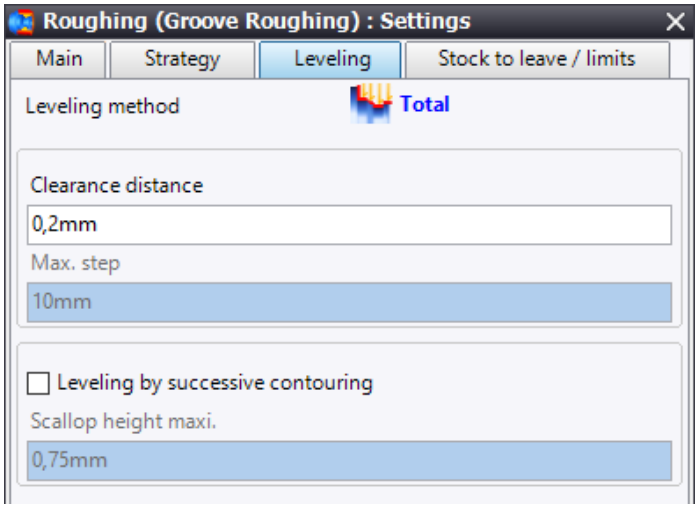
- Click on ✔ to **confirm** the tool.

- On the label, change the values as follows:
  - Adjust the **stock to leave constant** to *0.3mm*
  - Enable the **Auto** option: **Yes**
  - Adjust the **radial pass depth** to *2mm*
  - Select **2 program points**

Note: Double-clicking in the **Program point** or **Tool Offset XXXXX** box automatically opens the dialog box of the cutting conditions in the **Gauges** tab, which allows you to select the driven points (right and/or left).

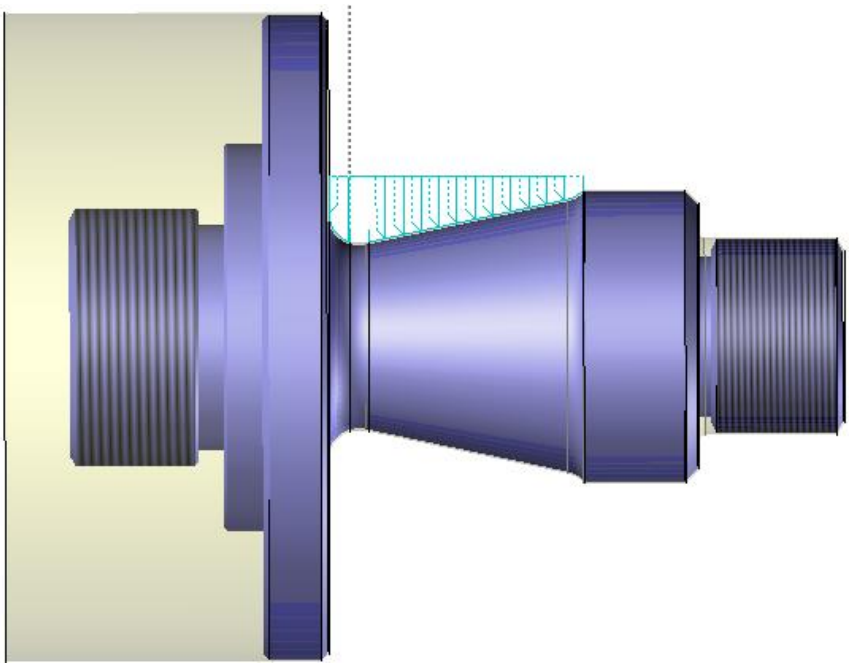
Kind of machining	
Time	00:00:00:000
Stock to leave method	
Stock to leave constant	0,3mm
Auto	Yes
Groove roughing method	
RadialPass depth	2mm
Number of program points	2
Program point	Tool Offset 2 Left Auto Tool Offset 3 Right Auto
Machined zone	
Tool Path Preview	Yes

- In the  **Settings** option, go to the **Leveling** tab and select a **Total** leveling method.



- Click on  to **confirm** the operation.

You should end up with the following result.

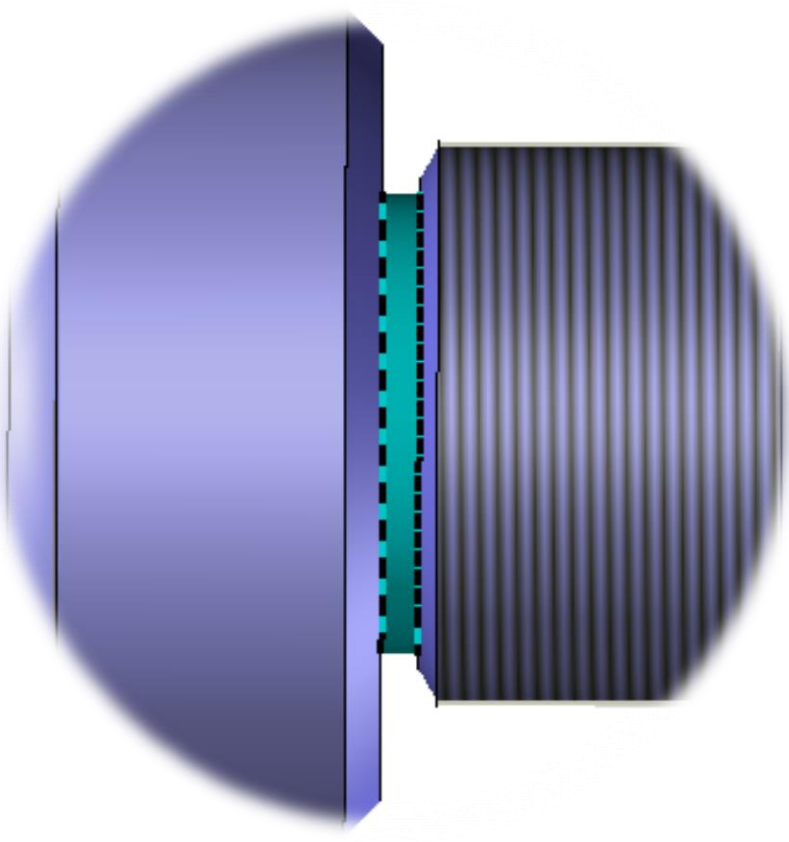


-  **Save** the document.

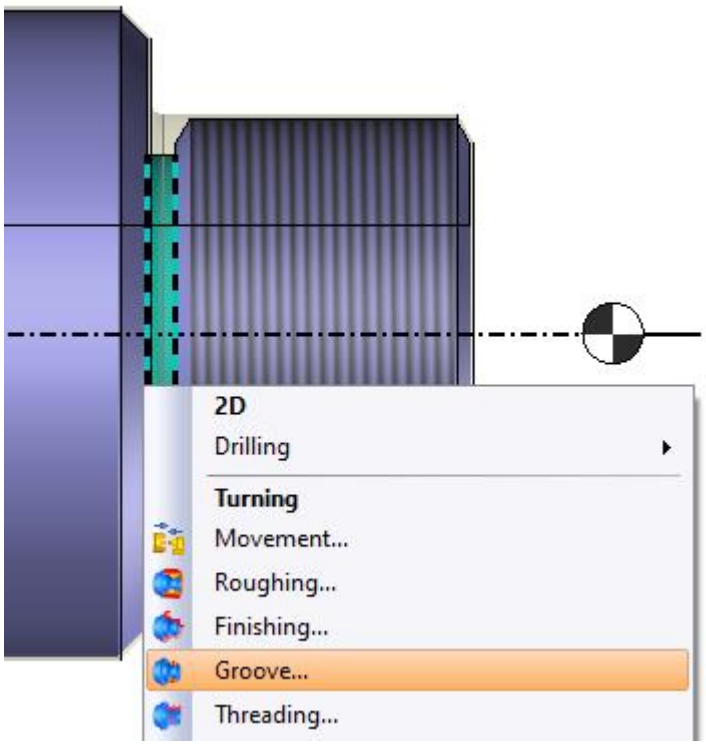
Step 4 – Groove machining

- From the Project tree, open the *Arbre_Court_4* machining document.

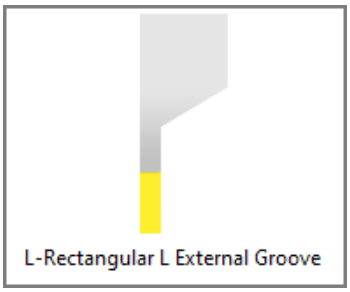
We will now machine the groove.



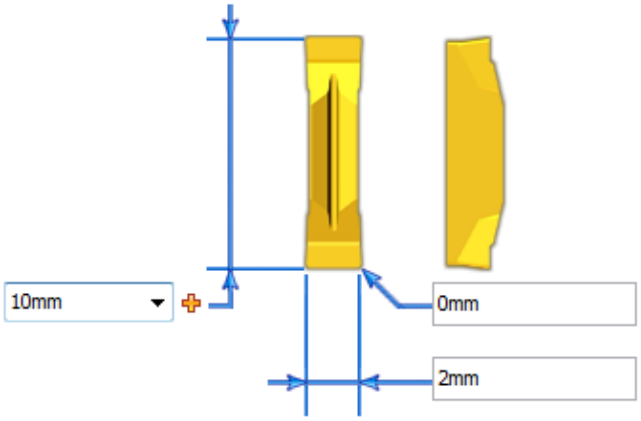
- Select the blue-colored face as shown below, then right-click and select the **Groove** command.



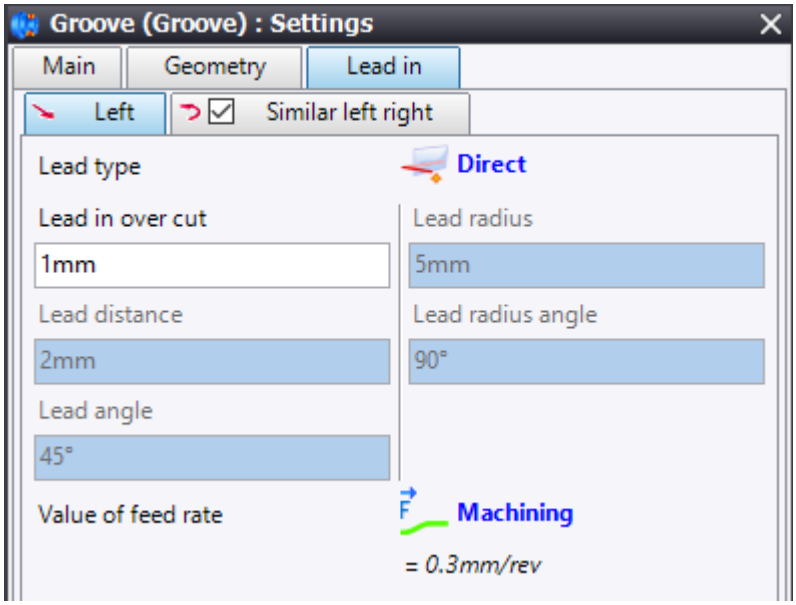
- Go to the tool magazine.
- Select an empty pocket in **X+**.
- Double-click on the L-Rectangu... **L-Rectangular** thumbnail.
- In the **Template(s)** tab, double-click on **L-Rectangular L External Groove**.



- In the **Component(s)** tab, adjust the tool width to *2mm* and the radius to *0mm*.



- Click on to **confirm** the tool.
- In the **Settings** option, go to the **Lead in** tab and adjust the **lead in over cut** to *1mm*.



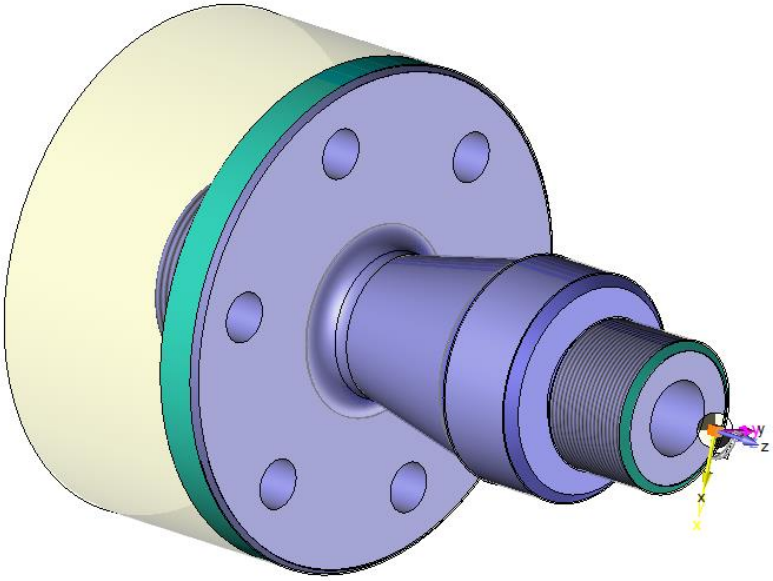
- Click on to **confirm** the operation.
- **Save** the document.

Step 5 – Finishing

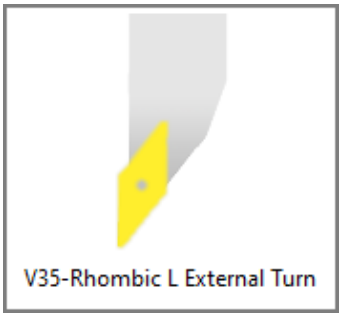
- From the Project tree, open the *Arbre_Court_5* machining document.

For this step, we will remove the remaining stocks to leave resulting from the roughing operations.

- Hold down the **Alt** key and select the two blue-colored faces as shown below.



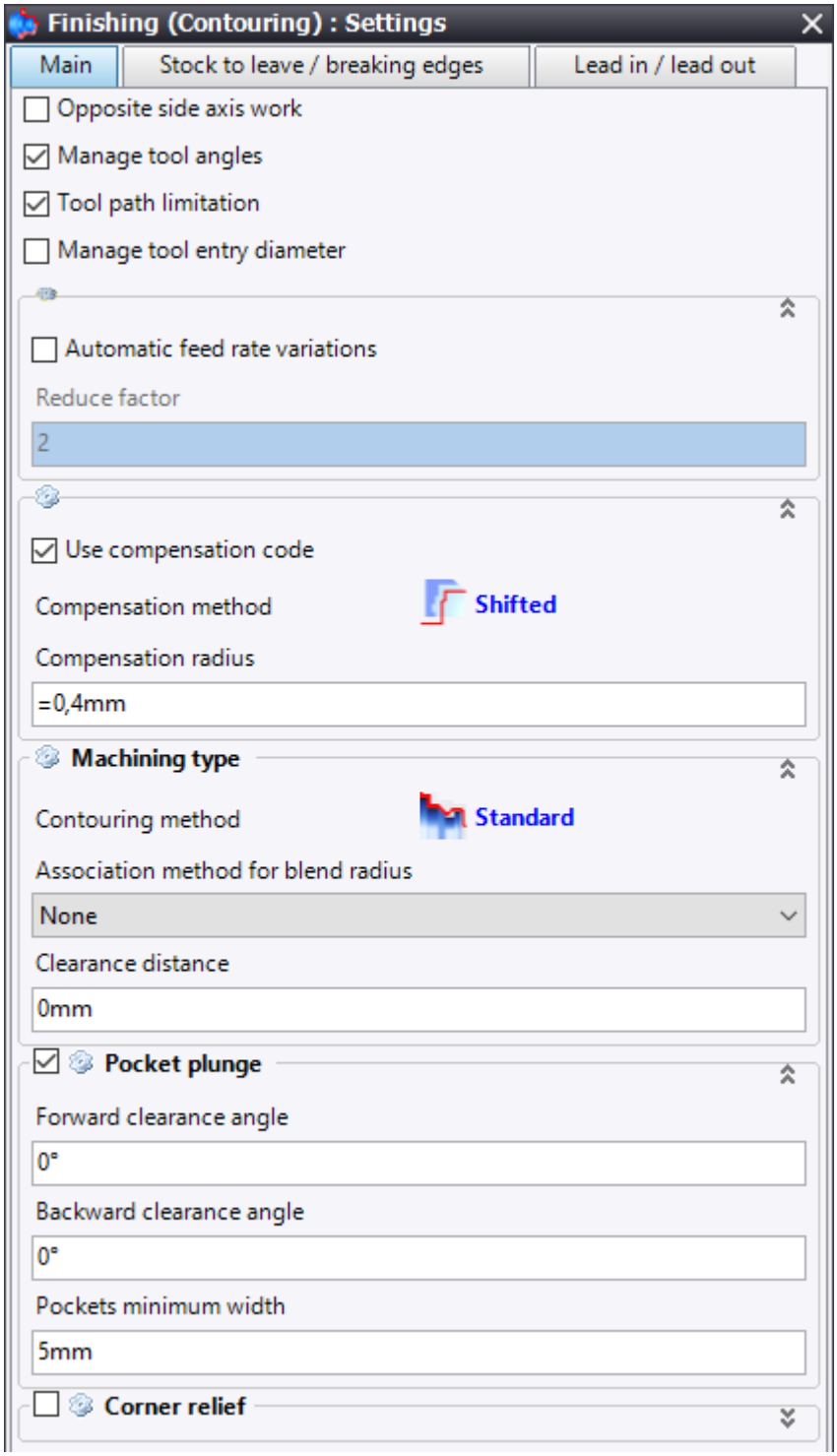
- Right-click in the graphics area and select the **Finishing** command.
- Go to the tool magazine.
- Select an empty pocket in **X+**.
- Double-click on the V-Rhombic ... **V-Rhombic 35** thumbnail.
- In the **Template(s)** tab, double-click on **V35-Rhombic L External Turn**.



- In the **Component(s)** tab, adjust the insert radius to *0.4mm*.
- Click on to **confirm** the tool.
- On the label, enable the **pocket plunge** option.

Kind of machining	
Time	00:00:00:000
Stock to leave method	
Compensation method	
Pocket plunge	Yes
Tool Path Preview	Yes

- In the  **Settings** option, go to the **Main** tab and adjust the **pocket minimum width** to *5mm*.

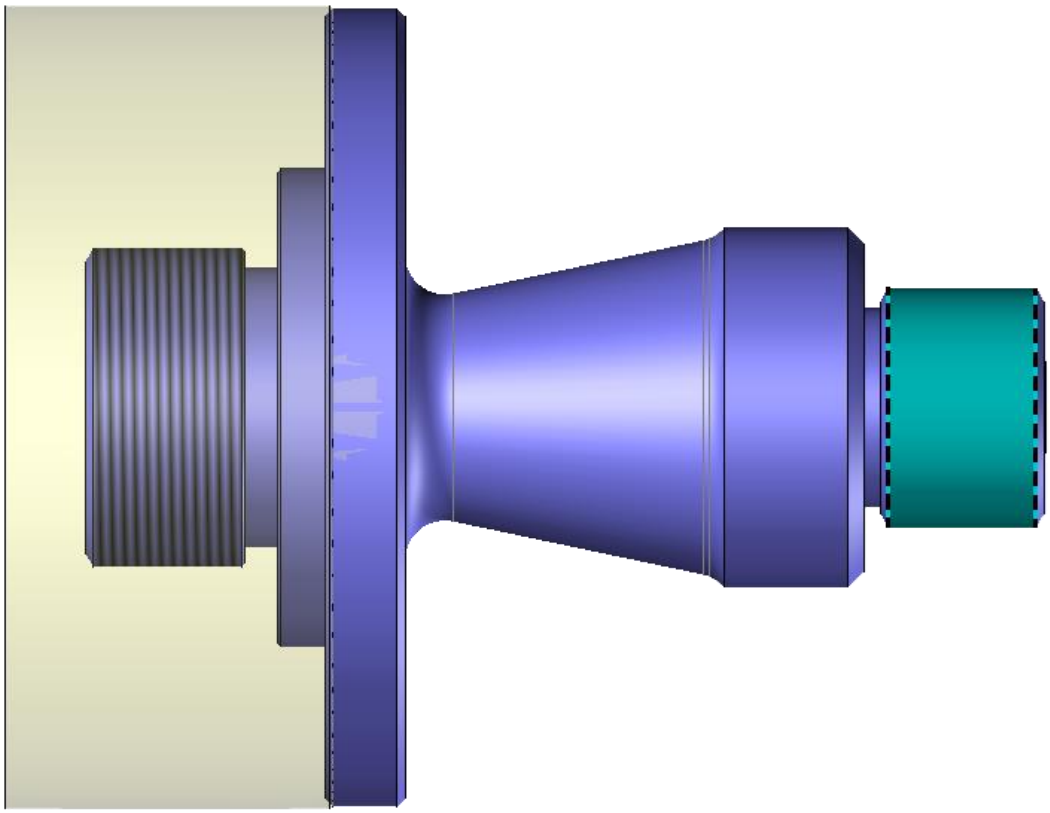


- Click on  to **confirm** the operation.
-  **Save** the document.

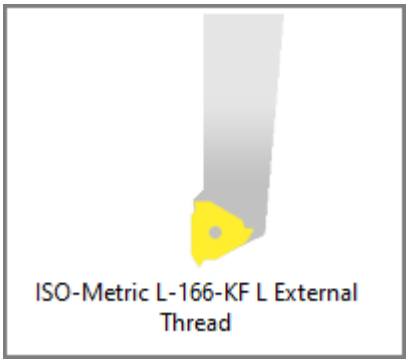
Step 6 – Threading

- From the Project tree, open the *Arbre_Court_6* machining document.

We will now thread the blue-colored face as shown below.



- Right-click on the blue-colored face as shown below and select the **Threading** command.
- Go to the tool magazine.
- Select an empty pocket in **X+**.
- Double-click on the **ISO** thumbnail.
- In the **Template(s)** tab, double-click on **ISO-Metric L-166-KF L External Thread**.



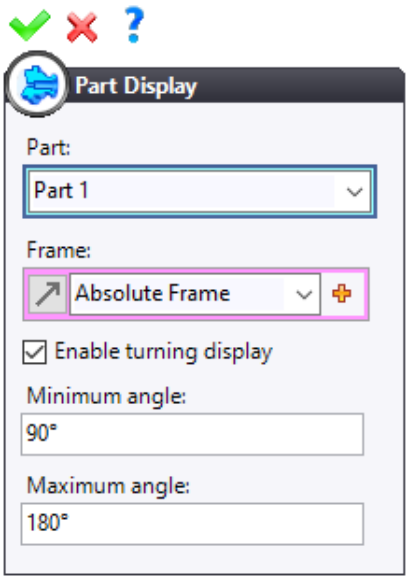
- In the **Component(s)** tab, make sure that the step is 1mm.
- Click on to **confirm** the tool, and then the operation.
- **Save** the document.

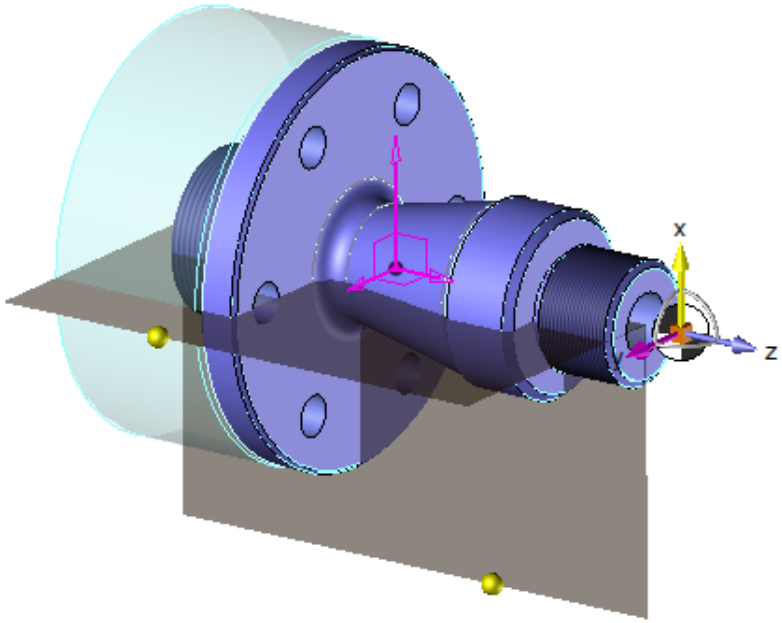
Step 7 – Hole machining

- From the Project tree, open the *Arbre_Court_7* machining document.

For some turning operations, it may be interesting to view the inside of the part.
Here, in particular, it is useful to create a sectional view for our drilling.

- To create the sectional view, select the **Part Display** command from the **Turning** tab.
- Make sure to check the **Enable turning display** box.

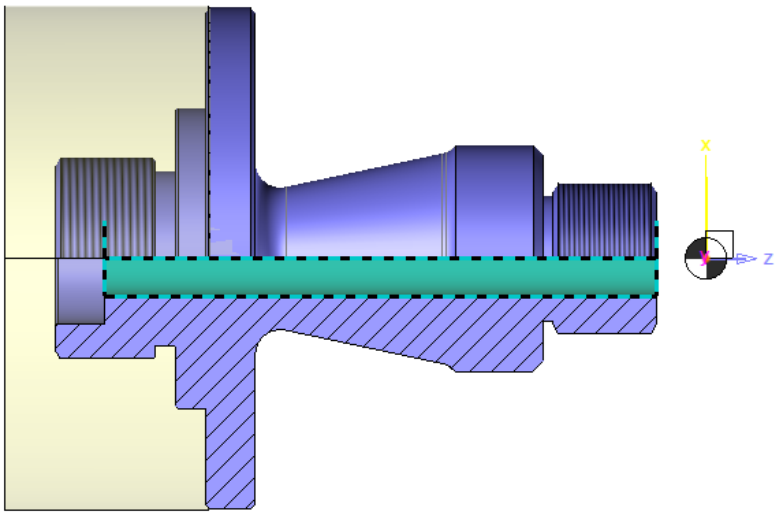




- Click on to **confirm** the display.

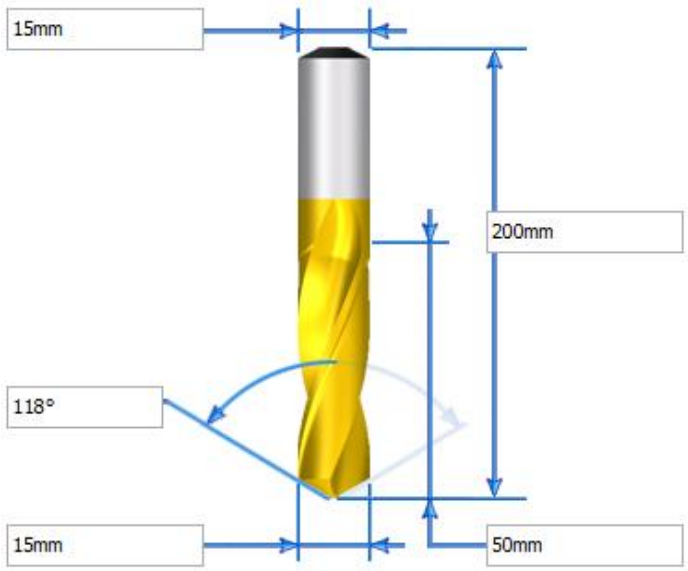
We will now perform the drilling operation.

- Select the blue-colored cylindrical surface as shown below and select the **Hole Machining** command from the **2D/3D** tab.



- Go to the tool magazine.
- Select an empty pocket in **Z+**.

- Create a *Ø15mm* **twist drill**.
- Adjust the usable length to *50mm*, the total length to *200mm* and the output distance to *160mm*.

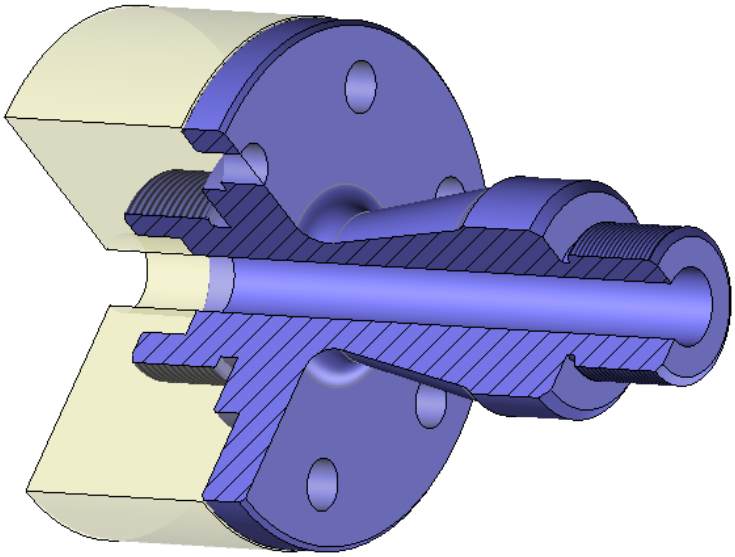


- Click on  to **confirm** the tool.
- On the label, adjust the geometry type to  **Through stock** and the rotating element to  **Part**.

Kind of machining	
Time	00:00:00:000
Diameter	15mm
Machined depth	138,6301mm
Rotating element	
Off center	0mm
Geometry type	
Through hole overdepth	2mm
Tool Path Preview	Yes

- Click on  to **confirm** the operation.

You should end up with the following final result.



-  **Save** the document.