

GO2cam



GO2cam V6.10
Tutorial
T04 – Shaft

T04 – Shaft

Introduction

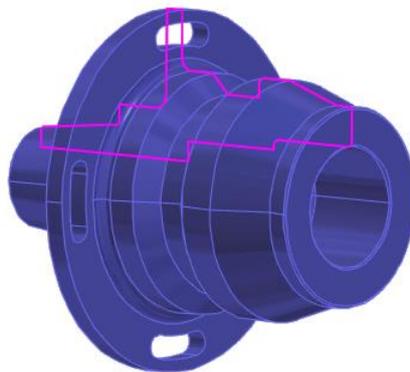
- Import of 3D model from CAD
- Positioning of 3D model
- Creation of 2D silhouette
- Machining with an opelist
- Modification of machining order
- Modification of strategies and tools
- Creation of axial plane
- Milling operations on the C axis

Extra files

In the Training Pack Basic, you can find:

- the pdf file of this tutorial,
- the 3D model in STEP file format T04_Shaft.STP,
- the finished part file of the tutorial T04_Machining.PCE.

These tutorials supplement the go2cam online help. You can read the online help from the help menu of GO2cam or press F1



I. Import and Positioning of the Part:

1. Start:

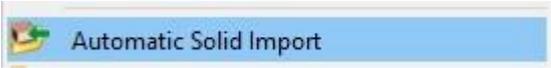
- In **GO2cam**, Left click on the Turning icon the most on the right

Note: the icons represent the type of product. If your licence do not include it, the product icon is greyed.

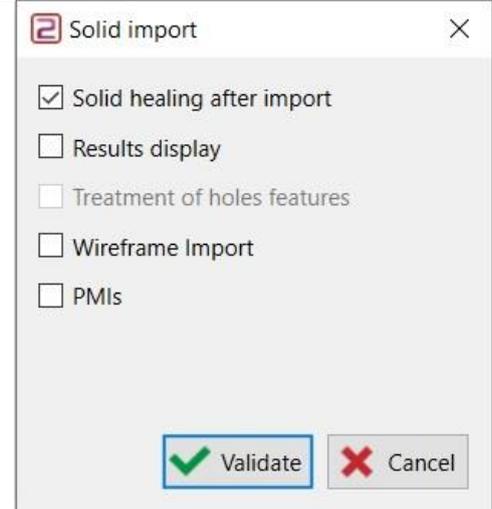
The software opens in design mode by default.



2. Loading the part:

- Select the menu **File**
- Left click on 
- Choose the part called **T04_Shaft.STP**
- In the next dialog, check « Solid healing after import » then left click on **Validate**

Part is loaded.

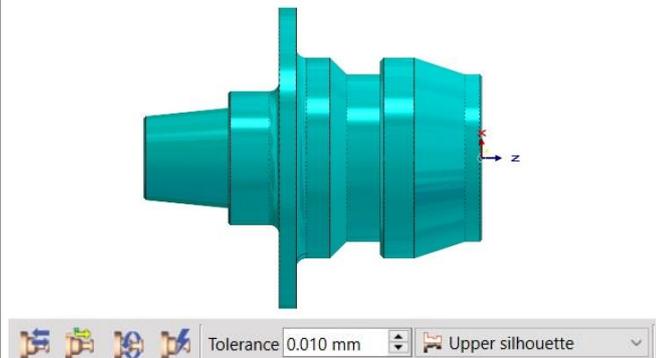


3. Positioning:

This function is always active. Must determine if the part is in good position.

- The part is in the correct position, left click 

Note: If the part location is incorrect, there are three solutions to choose from so that you can correctly locate the part.



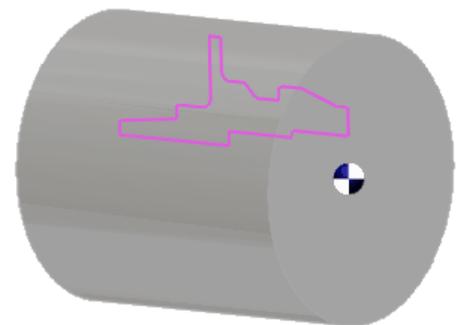
4. Stock definition:

The contour of the part is calculated automatically. Now you are required to define the stock.

Stock is automatically created relative to the part.

Note: This is a defined cylindrical stock with a constant overflow of 5 mm around the geometric shape.

- The stock is defined, Left click on 



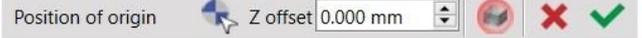
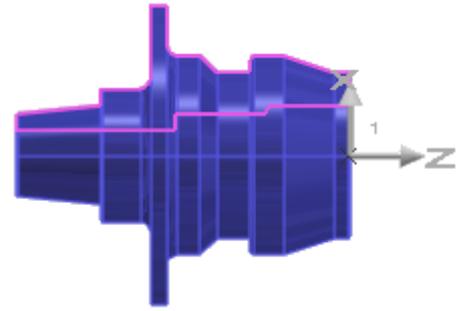
5. Calculation of origin position:

- Rotate the 3D part to check the origin position
- The origin is in the correct position, left click on



Note: If the origin is not good, you can reposition the origin by clicking  and then clicking the new origin

Parts are ready for machining.

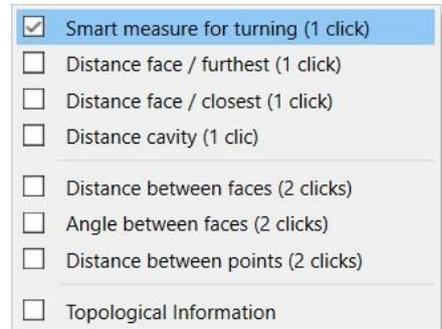


II. Modify tolerance dimension of 3D part:

1. Selection of smart measure:

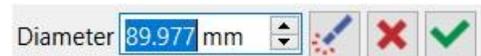
- Left click on 

Left click in the background, and then select Smart Measure for Turning.



2. Select the diameter to modify :

- Left click on the diameter 90 mm
- Left click on the pencil then write 90g6 in the diameter field
- Calculate the tolerance dimension and then validate.
- Both the 3D part and the silhouette is modified.



III. Process for the machining:

1. Operation menu :

- Left click the menu **Opelists**

Note: This menu is dedicated to user-created opelists. In this menu, you will not find the standard cycle, only the opelist. If there is no opelist, this menu does not exist.



2. Load an Opelist :

- Left click the menu **My List**
- Select Opelist **T03_Opelist.OPL**

Note: If there is no other opelist, your opelist will automatically be loaded.

The dialog box opens: here you can find the parameters customized when creating Opelist.

- Change the pass depth of the Outside Roughing, Left click in the Value field, type 2.5
- Left click on Validate, and the Opelist applies to the part.



| Label | Value | State |
|----------------------|----------|-------|
| < Facing > | | |
| Pass Depth | 3.000 mm | |
| < Outside Roughing > | | |
| Pass Depth | 2.500 mm | |
| < Inside Roughing > | | |
| Pass Depth | 3.000 mm | |

3. Simulation :

In order to verify the toolpath, we execute the simulation:

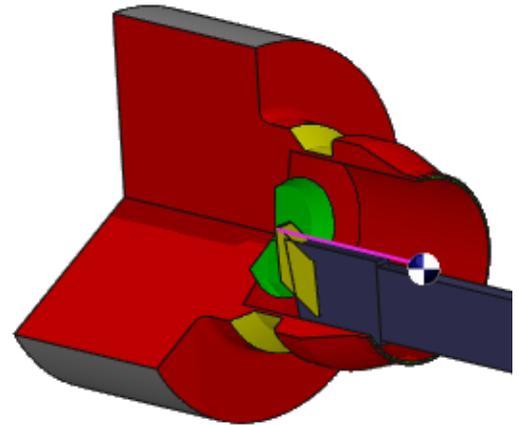


- Left click on **Simulation**
- Check whether the dynamic mode is active, otherwise select this mode.

In the filter for solid on the right side of the screen, a 120 ° cross section view can be obtained during the simulation :



- Left click on the Blue arrow above the solid
- Select the check box with the section icon
- Left click on  To start the simulation in step mode
- Click the icon again with the left mouse or press the space bar to play forward step by step



4. Conclusion :

We can see that collision is detected, icon  turns red in the machining tree, and the cycle name containing collisions also turns red

There are two conclusions after this simulation:

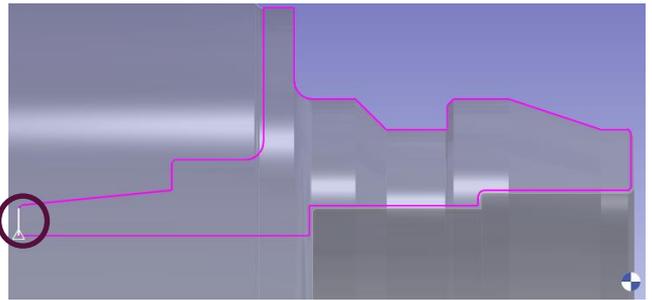
- We need to add a drilling operation before the Inside Roughing.
- We need to change the size of the tool for the Inside Roughing and select a smaller tool holder



IV. Add operations and modify tools:

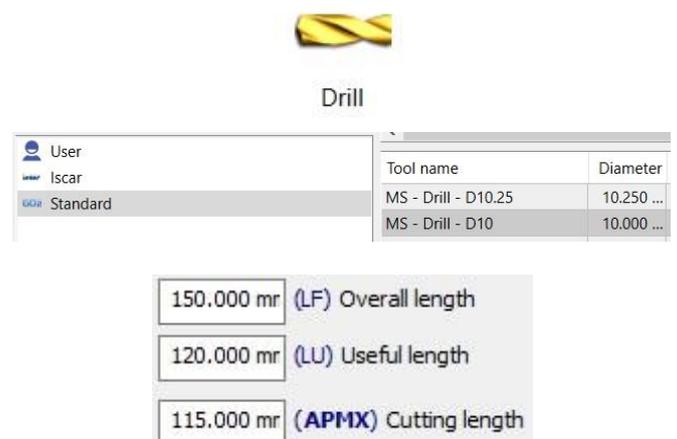
1. Create a drilling cycle :

- Left click on  Axial Hole
- Left click on 
- Left click the point, as shown in the figure 



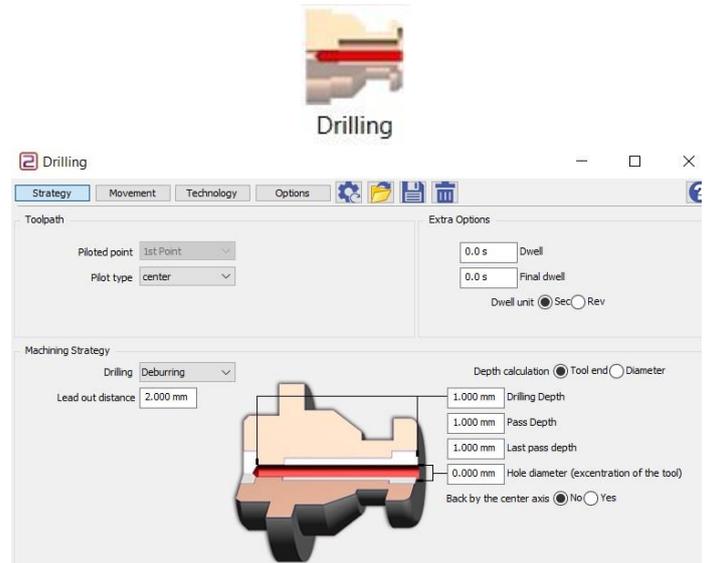
2. Selection of tool:

- Left click on , then on Drill
- Select download and then **Standard**
- Left click on "MS - Drill - D10"
- Modify the **overall length**, **useful length** and **cutting length** in the tool settings



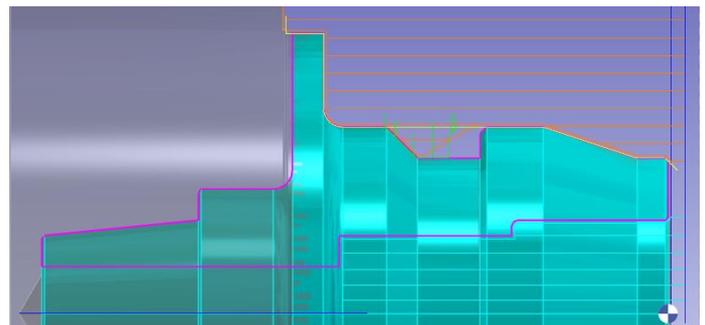
3. Selection of machining cycle :

- Left click on , then on Drilling
- In the strategy, modify the parameter as shown on the image



4. Toolpath calculation :

- Left click on 



5. Reposition the Drilling operation:

Obviously, the drilling must be carried out before the Inside Roughing. So we have to move it.

Note: In the machining tree, operations created by opelist are grouped together. To change the cycle order, you must first break the opelist.

- Right click on T03_Opelist in machining tree
- Left click on **Multi-Cycle Breaking Out**
- Next, select the Drilling cycle and hold down the left mouse button
- Then put the hole in front of the Inside Roughing and release the left mouse button.

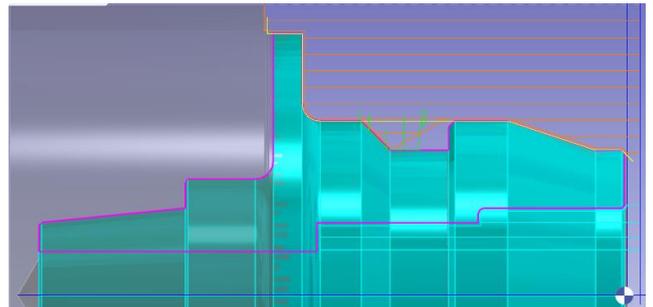
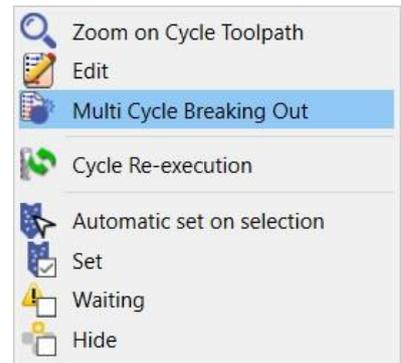
- Left click on **Update**



The orange ball indicates that one or more operations can be updated

- Select Machining

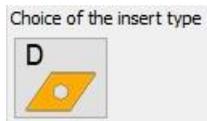
After the Drilling, the Inside Roughing is recalculated based on the remaining stock (removed material).



6. Modification of tool:

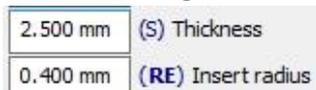
As seen in the simulation, the boring cutter for the Inside Roughing is not suitable for the part size. Therefore, we must change it.

- In the machining tree, Left click on icon  before the Inside Roughing operation
- Right click on tool and select **tool edition**
- The tool definition dialog box opens:

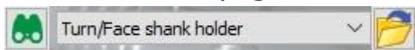


Change the insert to

And then change the following values :



- Left click on the binocular icon for Insert holder and choose the Turn/Face shank holder to access the attachment page

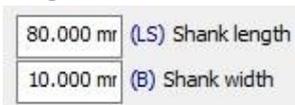


Change the values as shown on the picture. 

- Left click on the  plus icon to add a Round Shank

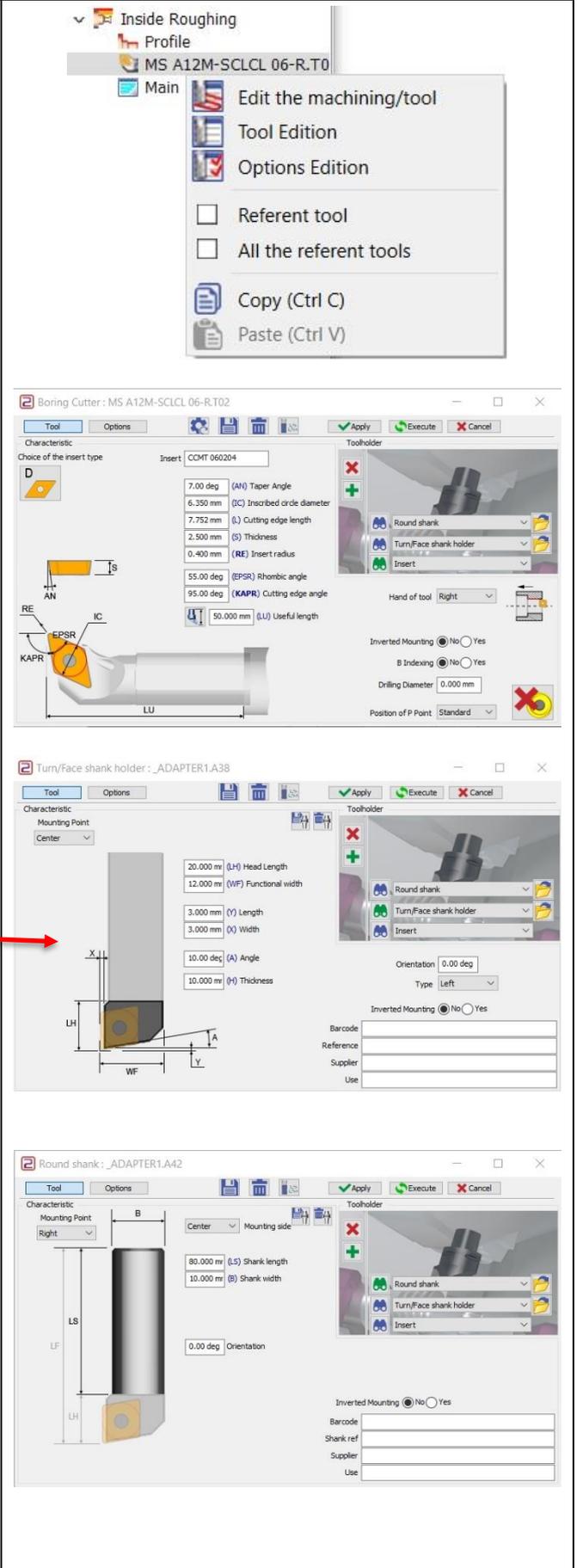


Change the following values:



The tool and its accessories have now changed.

- Left click on  Execute to validate the changes.



V. Part Return and back machining

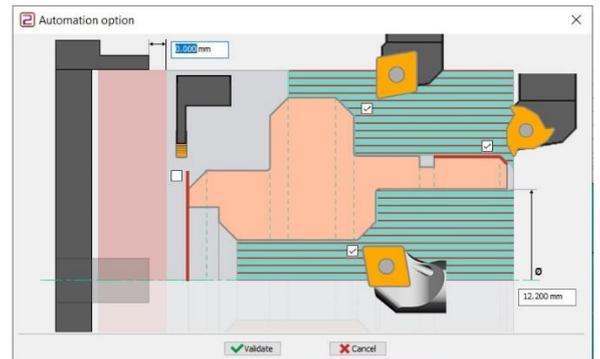
7. Creation of opelist T04 :

Now we will create an Opelist to machine the back of part. (See T03_Opelist_Creation)

- Select the operations, as shown in the picture
- Right click in the machining tree
- Left click on **Opelists/Export**
- If necessary, input parameters and **automatise**  the cycles by indicating a minimum diameter of 12.2 corresponding to the drilling diameter.
- Left click on  **Validate** to validate
- Name it T04_Opelist

Opelist T04 is now created. Therefore, we can apply it to the back of part.

But before that, we must return the part.

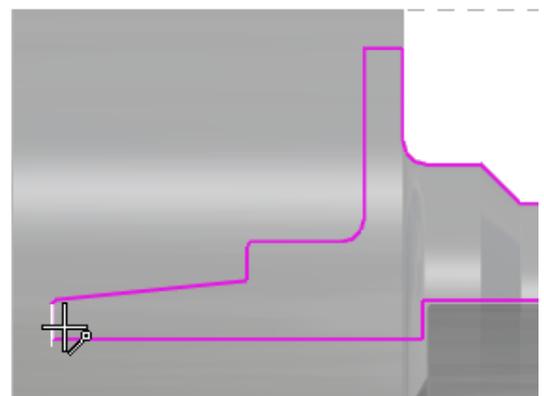
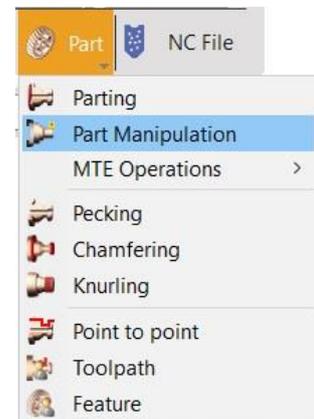


8. Part Return:

- Go to the «Part» menu.
- Left click on **Part manipulation**
- Left click on the function **Part Return** 

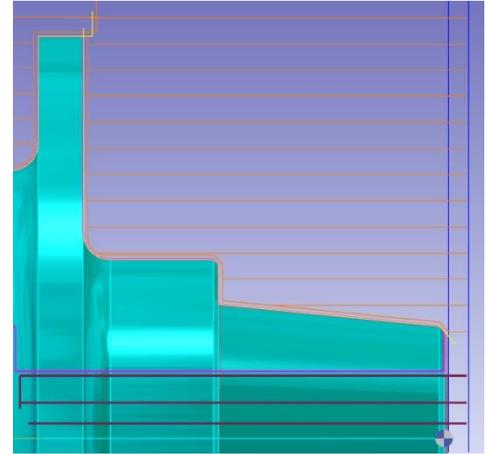
Note: Opens a dialog box where you can define the Z position of the new origin or click the geometry directly. Software will automatically provide the corresponding value.

- Left click on the point of the desired face if the Z value is different. Then left click on 



9. Application of opelist T04 :

- Right click in machining tree
- Go on Opelist and left click on  Import opelist
- Choose T04_Opelist
- Change parameters if necessary
- Left click on icon Confirm



10. Conclusion :

Note: All turning operations on the machining tree have been correctly programmed, which is the objective of this tutorial document.

However, you may notice that there are still some changes to be made, including adding finishing operations for facing and inside roughing. Rather than adding operations, you can also delete the allowance defined in these operations in the strategy page..



VI. Creation of milling operations :

1. Milling operations:

We will create a C-axis milling operation and copy it to machine four pockets.

- Enter menu  **Milling**
- The automatically selected plane is a reference plane. This is the only plane created by milling. This plane is only for axial rotation tools.

- Enter menu  **Standard**

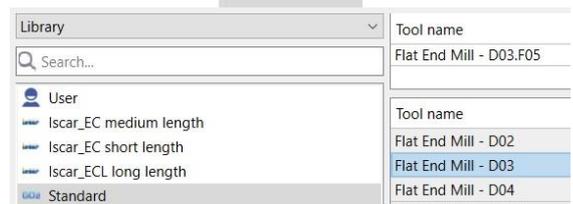
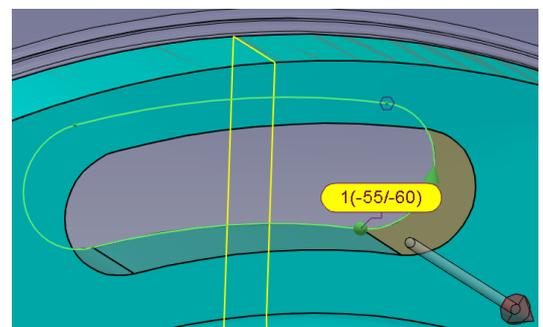
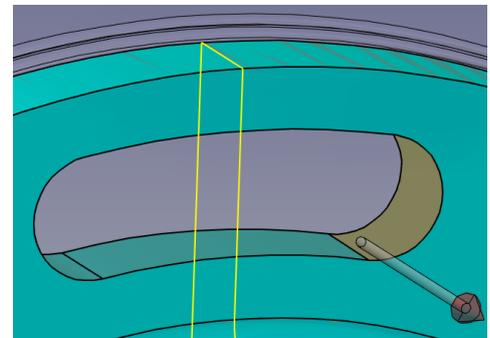
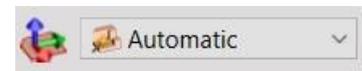
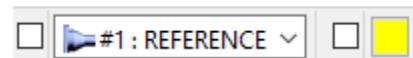
- Left click on 
- The selection mode defaults to automatic
- Left click on one side of the pocket

The pocket contour has just been created at the correct height, which is directly created on the solid.

- Left click on 
- Choose Flat End Mill, Download Standard library
- Select tool "Flat End Mill D03"

- Left click on 
- Choose **Pocket+Contour**, type-in 0.5 mm for the Z step

- Validate 



Pocket+Contour

| Techno. name | Z Step (Ap) |
|--------------|-------------|
| | 0.500 mm |

2. Copy of operation :

We will copy the last operation 3 times, but it is simple and fast.

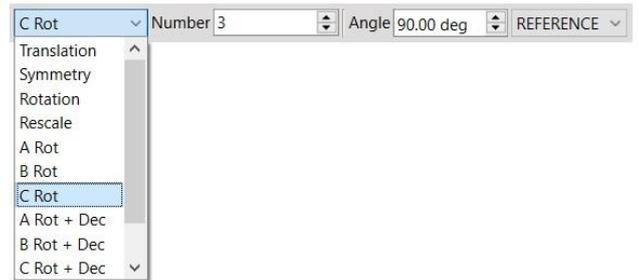
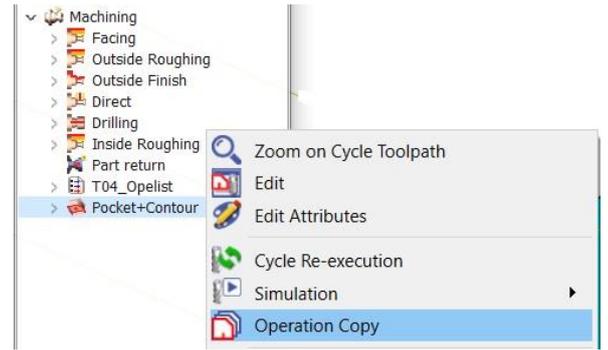
- Stay in the menu 
- Right click on **Pocket+Contour** in machining tree
- Left click on **Operation Copy**

In the dialog box, you must define :

- Copy and move, select Rot C
- Number of copies to replicate, type 3
- The angle of the copy operation relative to the copy operation, enter **90°**
- The plane on which to copy is automatically selected with the current plane.
- Finally, you need to define the operation to copy in the machining tree, left click on **Pocket+Contour**

So the part is finished. All processing has been completed. Now you know :

- Import, repositioning, and create a 3D model profile
- Work with the opelist, cycle break out and add new operation
- Modify strategy and tool
- The axial plane is also created to perform the C-axis milling operation



Show SPG call position

