

GO2cam



GO2cam V6.10
Tutorial
T02 – Cover

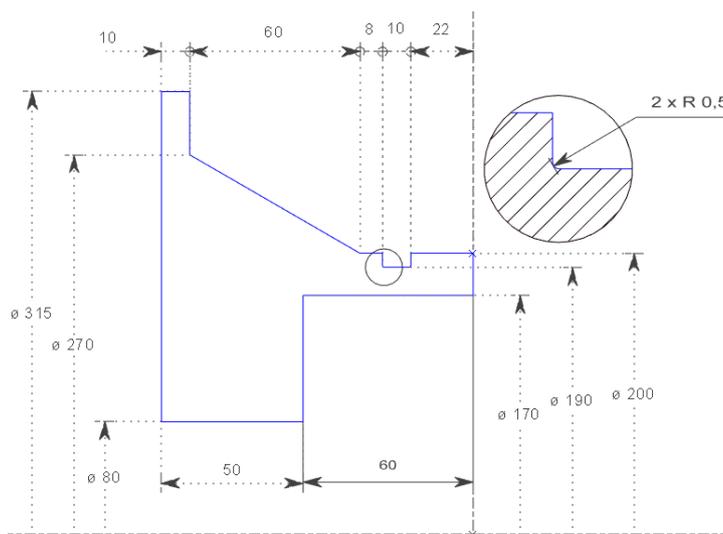
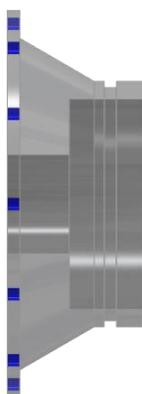
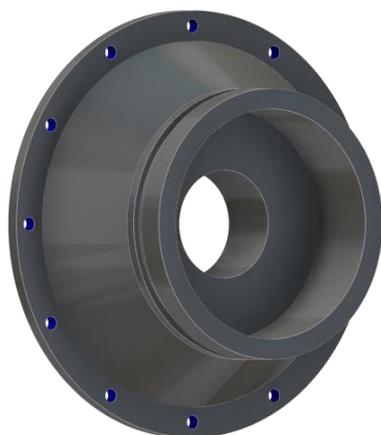
T02 - Cover

Introduction

Welcome to this cover creation tutorial.

This tutorial has the objective to help the user:

- Learn how to use the turning environment in GO2cam
- Understand the uses and capabilities of GO2cam in turning.
- Get used to the user interface and the mouse actions by practicing on a simple example.



Extra files

In the Basic Training Pack, you can find:

- the pdf file of this tutorial
- the layout design

The tutorials are additional resources to GO2cam's online help. You can access the online help by selecting the Help menu in GO2cam or by pressing F1.

I. Process for the Design:

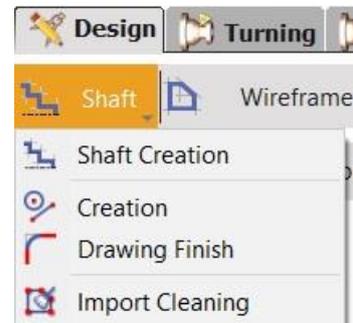
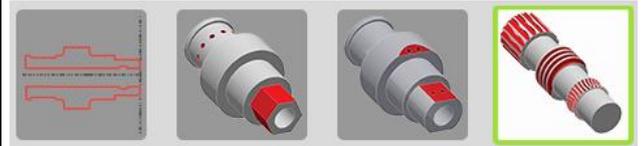
1. Choice of the Turning environment in the homepage:

- 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.

- Left-click on Shaft
- Left-click on Shaft Creation



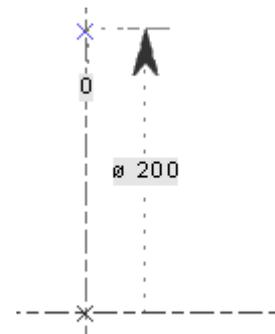
2. Sketch of the part:

- Left-click on the Shaft Creation 

Choice of the starting point: Z=0, D=200

- Type in 0 for Z and 200 for the Diameter
- Left-click on  to confirm the starting point

Note: GO2cam proposes you alternate length and diameter. This choice is shown by an orange line linked to the motion of the cursor.

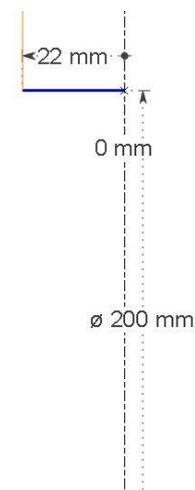


3. Length L=22:

- Left-click on , then choose horizontal segment 

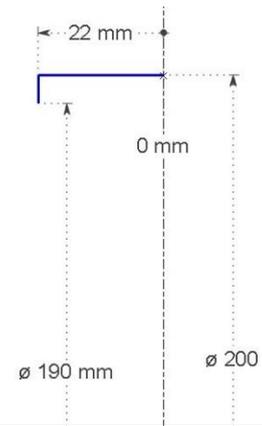
You can enter positive or negative values of length depending on the direction of the + sign defined by Z^- or Z^+ .

- The button should be in that position , if it is not, click on the button to change the direction.
- Type in 22 and press Enter.



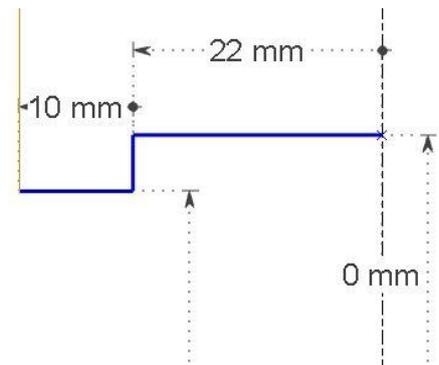
4. Diameter $\varnothing=190$:

- Type in 190 and press Enter.



5. Length L=10

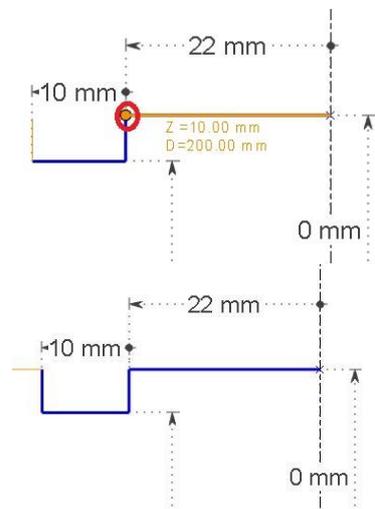
- Type in 10 and press Enter.



6. Groove end:

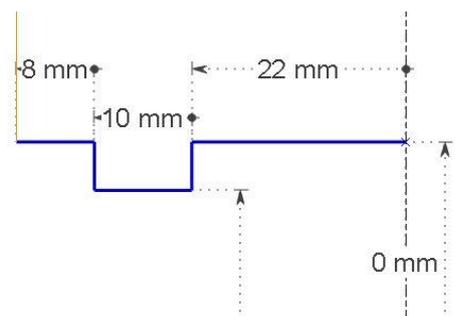
2 solutions :

- Enter the value with the keyboard : type in 200 and press Enter
Or
- Left-click on the first horizontal segment which was first created and which has the same value.



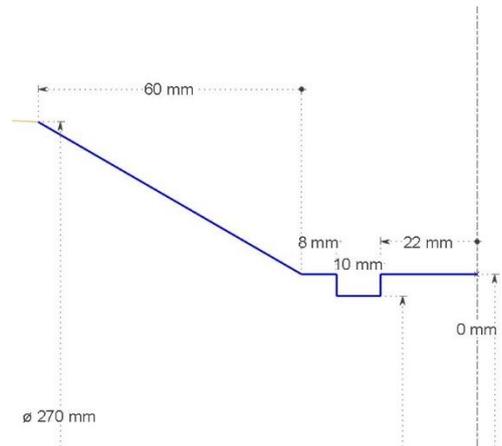
7. Length L=8:

- Type in 8 and press Enter.



8. Creation of a cone L=60, Ø=270:

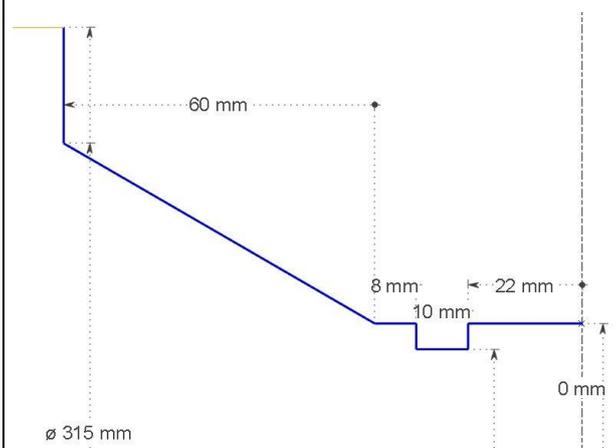
- Left-click on 
- Choose the mode of creation of the cone clicking on  until it changes so that you can define the cone by its length and diameter
- Type in 60 for the length, 270 for the diameter and press Enter
- Left-click on 



9. Diameter Ø=315:

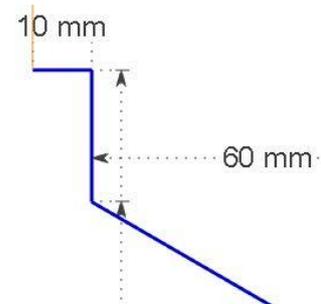
- Left-click on 
- Type in 315 and press Enter.

You are no longer in the cone creation mode. GO2cam proposes you once again to define the length and diameter alternatively.



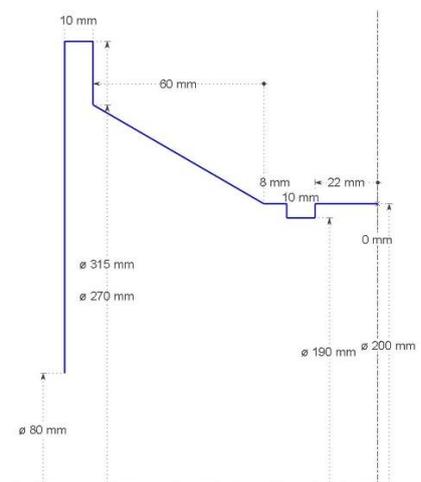
10. Length L=10:

- Type in 10 and press Enter.



11. Diameter Ø=80:

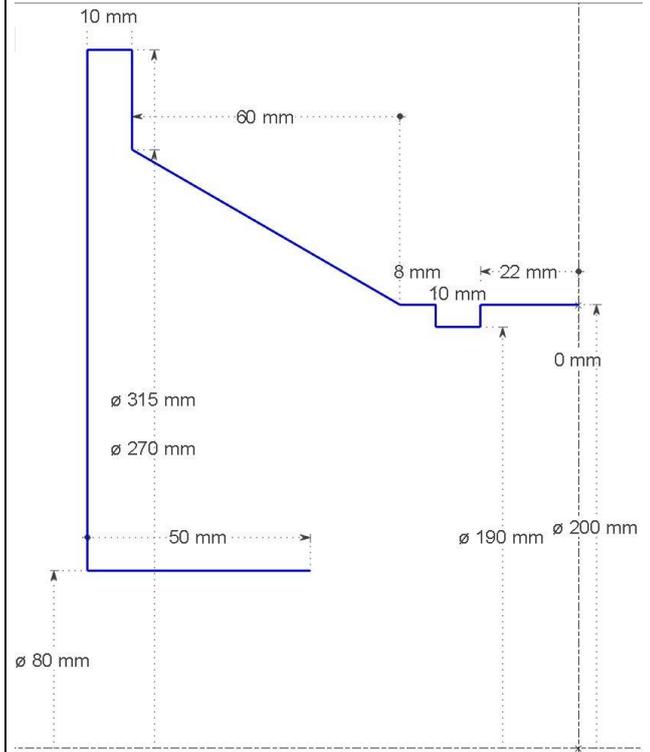
- Type in 80 and press Enter.



12. Length L=50:

The direction of the + sign is still z^- , there are 2 possible solutions:

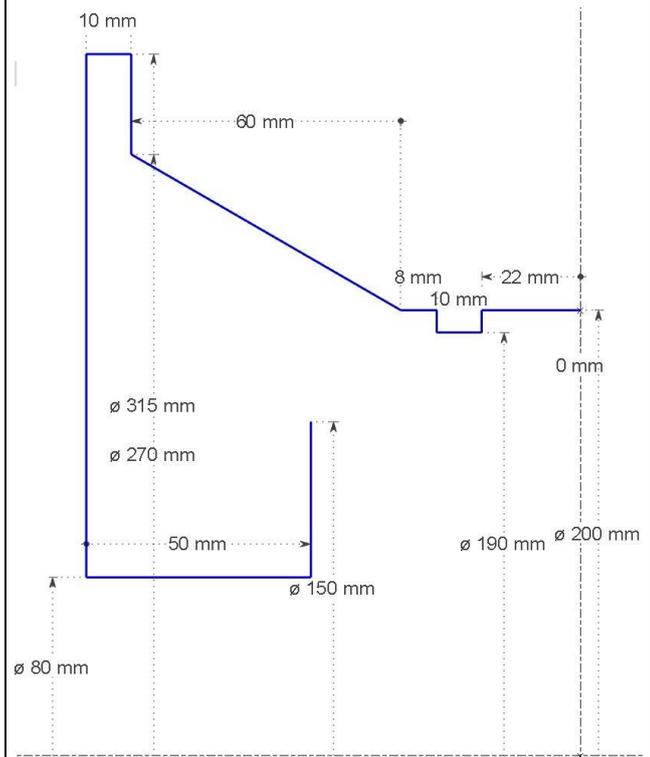
- Type in - 50, Enter key
- Or
- Change the direction with a click on and Left-click on sur z^- , which becomes z^+
 - Type in 50, Enter key



13. Diameter $\varnothing=170$:

We are going to make a mistake on purpose:

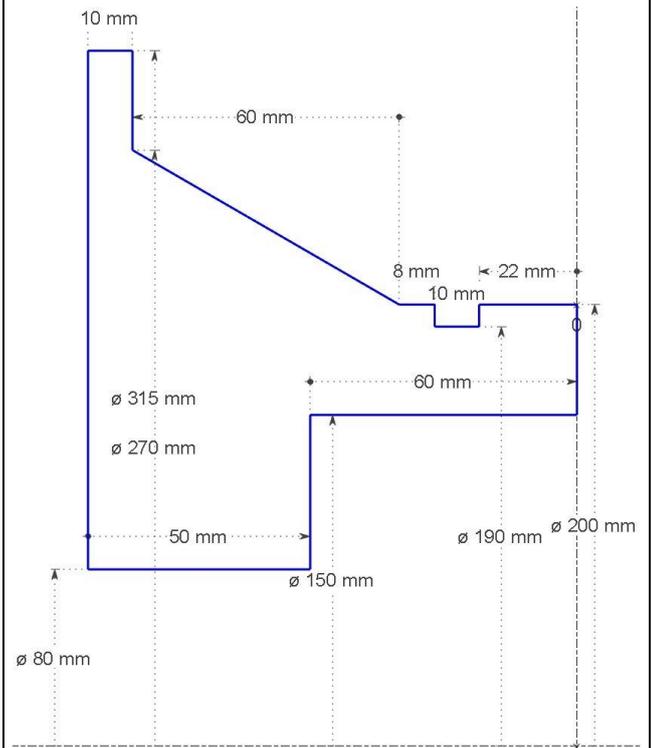
- Type in 150 and press Enter.



14. Finishing of the sketch:

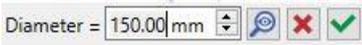
To finish the part, we can click on existing elements:

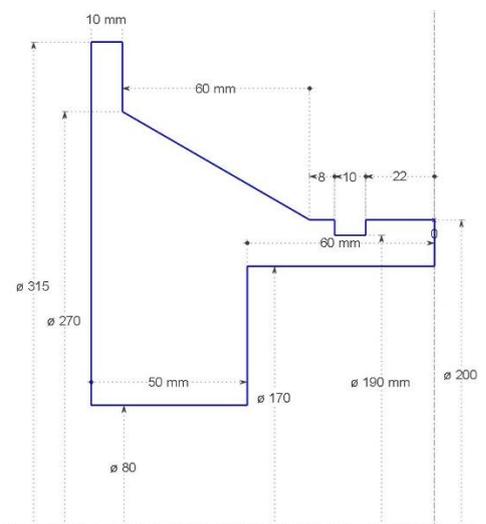
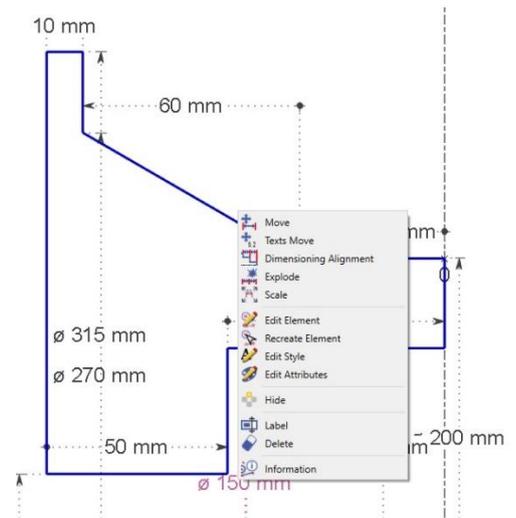
- Left-click on the vertical axis
- Left-click on the first horizontal segment which was first created
- Left-click on 
- Esc key to end the Shaft function



15. Modification :

We are going to correct the mistake done on step 13; the actual diameter is 170, we typed a value of 150!.

- Left-click twice directly on 150
or Right-click on the value then choose 'Edit element'
- 
- Replace the value of 150 with 170 in the field, and enter
 - Left-click on the **Green Tick** to validate



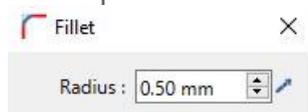
- **Finishing, definition of fillets for the groove:**

- Left-click on Zoom , then click on the screen twice to select the zone you want to zoom like shown on the picture
- Click on **Esc** key to end the zoom function

- Left-click on the **Fillets** function 

To create a fillet you can either click directly on the corner where the fillet is or click on the 2 elements whose intersection is the fillet.

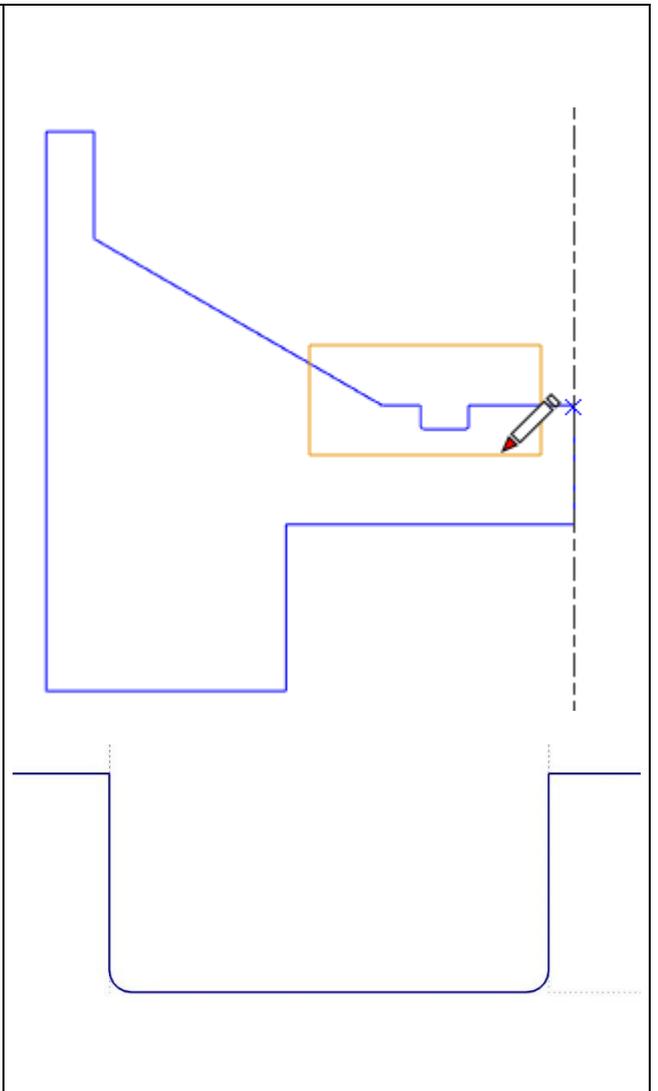
- Type in **0.5** and press Enter.



Note : the decimal symbol is the **point**, not the comma.

- Left-click on the **2 corners** of the groove to apply the fillet.
- **Press ESC** key to end the Fillets function
- Left-click on 

The wireframe design is finished.



II. Operating Procedure for the Machining:

Targets:

- Application of cycles for turning and grooving :
 - Selection of geometry for machining
 - Simple cycle parameters definition
 - Selection of tool
- Use machining tree: modifying a cycle
- Tool path simulation

1. Turning:

- Left-click on Turning
- Left-click on 

The stock is automatically created in relation to the profile.

It is a cylindrical stock defined with a constant 5 mm overflow around the profile.

We are going to modify the stock manually.

- In the Zmaxi field, type 2 to make the stock more than 2mm behind the origin

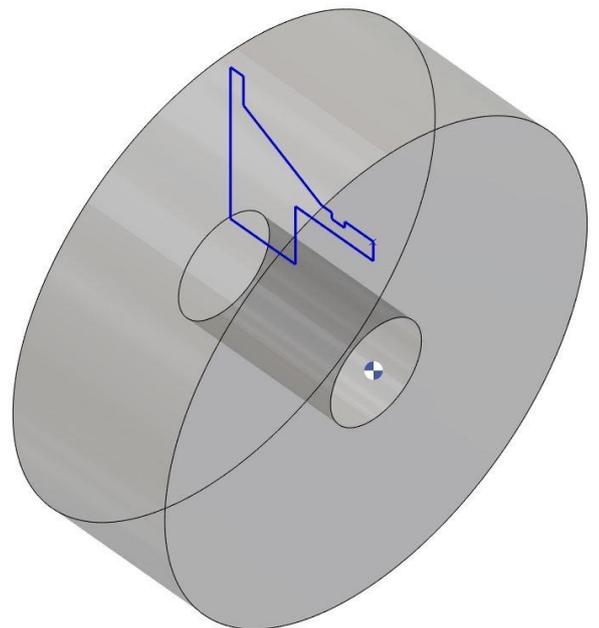
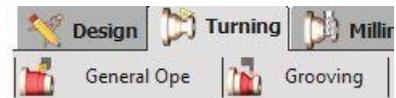


The stock is cored out:

- Change the value in D inner field to 70 and press ENTER



- Left-click on  to validate



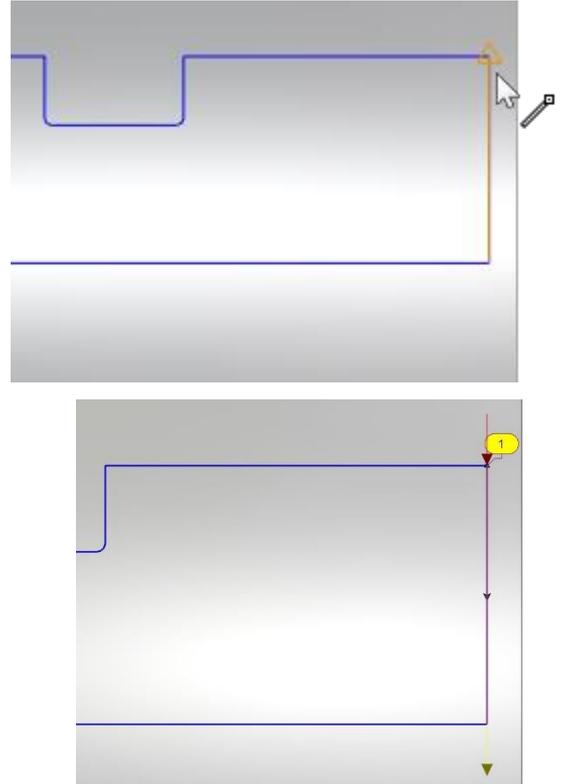
Ope 10 Facing

2. Selection of geometry:

- Left-click on  General Ope

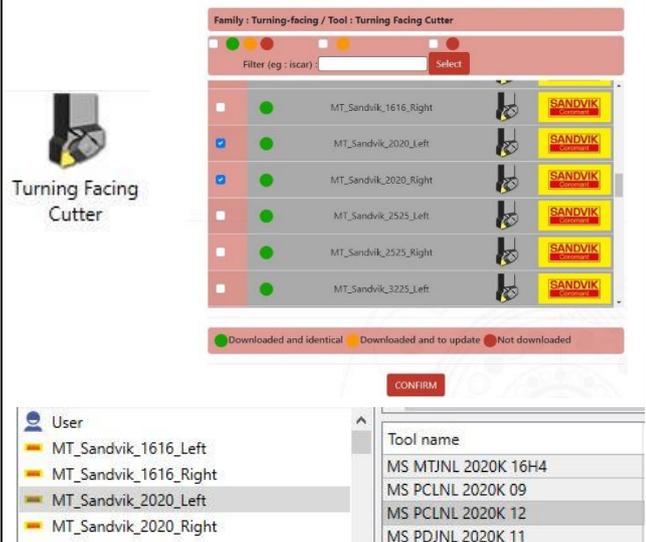
Note: It is recommended to always zoom during selection to facilitate selection of profiles.

- Left-click on Selection 
- Left-click on the first element of the path as in the drawing and ensure that the arrow is pointing downwards.
- Left-click on Profile OK 
- No need to modify the approach and return vectors which are tangent to the element shown.



Tool selection:

- Left-click on , then on Turning Facing Cutter
- Left-click on 
- Select **MT_Sandvik_2020_Right** and **MT_Sandvik_2020_Left**.
- Left-click on **Confirm** to download the tools library.
- After downloading and installing, select **MT_Sandvik_2020_Left** in the library
- In the list of tool, Left-click on "MS PCLNL 2020K 12"



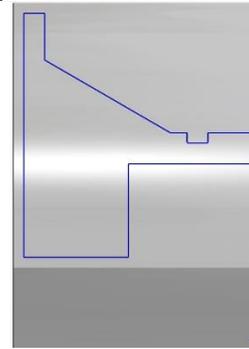
3. Selection of the machining cycle:

- Left-click on  then on Roughing
- In the list of techno, Left-click on "Main Spindle - Face"



4. Toolpath calculation :

- Left-click on **Cycle Calculation** 
- Right-click on the cycle and rename it to 'Facing'



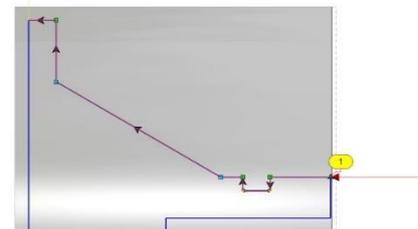
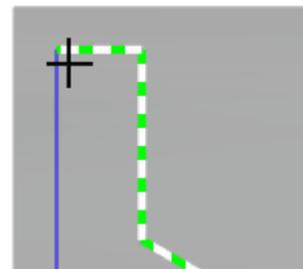
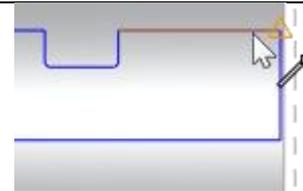
Ope 20 Straight cutting (roughing)

5. Selection of geometry :

- Left-click on  **General Ope**
- Left-click on Selection 
- Left-click on the first path element, on the right (image A), click on OK
- Left-click on the last path element, on the left (image B), click on OK
- Left-click on Profile OK 

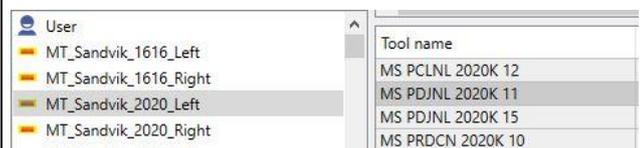
Note: No need to modify the approach vector which are tangent to the element shown.

- Select the Ret Vect check box, then manually enter the value 90 and press Enter



6. Selection of Tool :

- Left-click on , then on Turning Facing Cutter
- Select **MT_Sandvik_2020_Left** under user
- In the list of tool, Left-click on "MS PDJNL 2020K 11"



7. Selection of machining cycle :

- Left-click on  then on Roughing
- In the list of techno, Left-click on "Main Spindle - Outside"



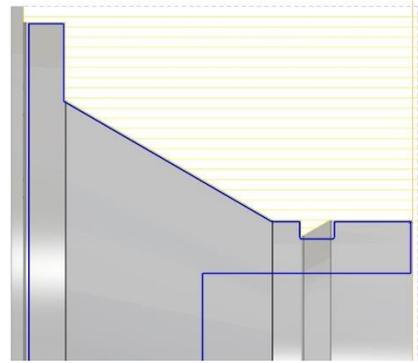
TECHNO NAME	Cycle type	Toolpath
Main Spindle - Outside	Outside	Ima...

8. Toolpath calculation :

- Left-click on Cycle Calculation 

The toolpath is drawn, the evolutive stock is machined.

- Right-click on the cycle and rename it to 'Outside Roughing'



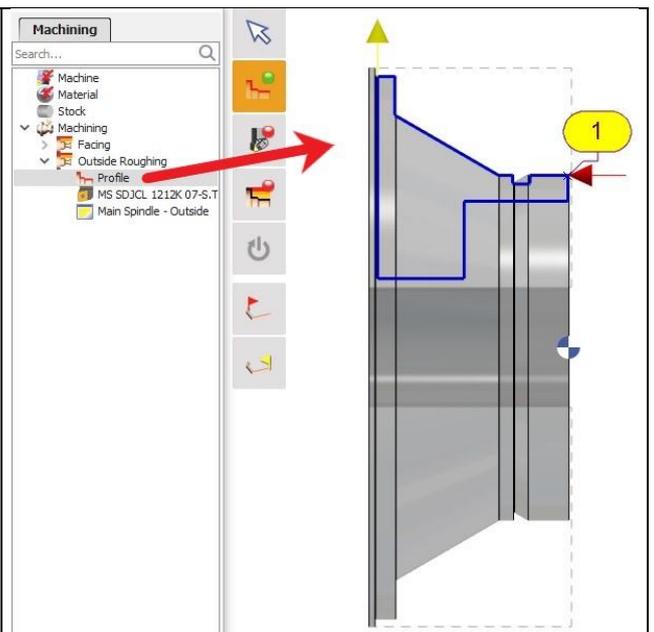
Ope 30 Straight cutting (finish)

9. Selection of geometry :

The menu  General Ope is still active.

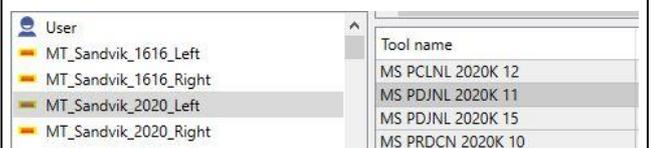
- Left-click on 
- In the machining tree left-click on  to expand the details for the outside roughing cycle
- Left-click and drag and drop the profile on the screen.

Note: This method will copy the profile selection of the roughing cycle to the one currently being defined thus avoiding making the same selection multiple times.



10. Selection of Tool :

- Left-click on 
- Select **MT_Sandvik_2020_Left** under user
- In the list of tool, left click on " **MS PDJNL 2020K 11**"



11. Selection of machining cycle :

- Left-click on Cycles  then on Finish
- In the list of techno, left-click on "Main Spindle - Outside"



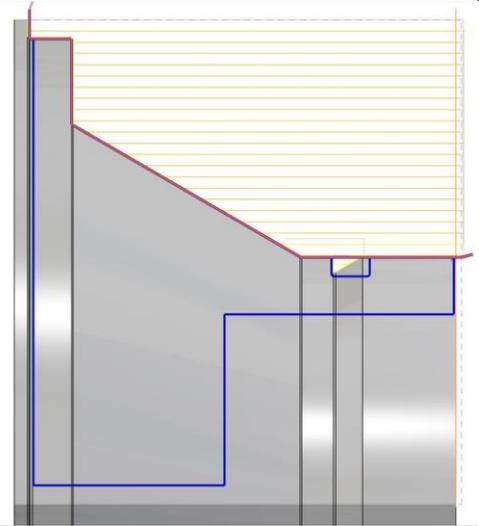
TECHNO NAME	Cycle type	Toolpath
Main Spindle - Outside	Outside	Ima.✓

- Toolpath calculation :

- Left-click on Cycle Calculation



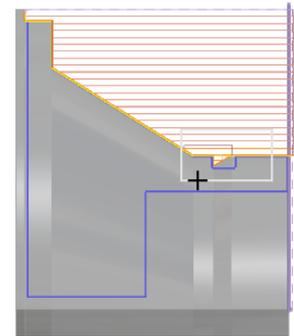
- Right-click on the cycle and rename it to 'Outside Finish'



Opé 40 Groove cutting

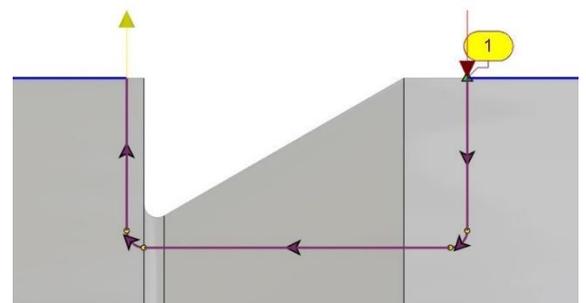
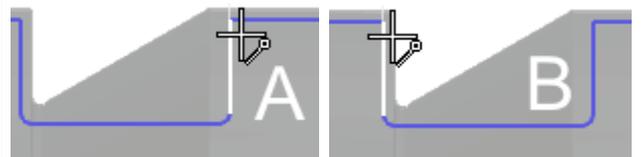
12.Start:

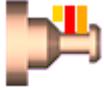
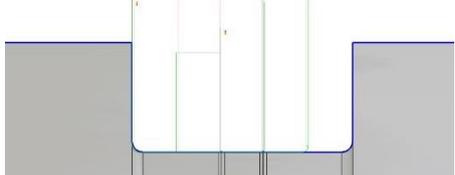
- Left-click on  Grooving
- Left-click on Zoom  then click twice on the screen to select the area you want to zoom like shown on the picture.



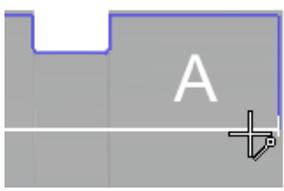
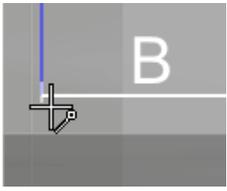
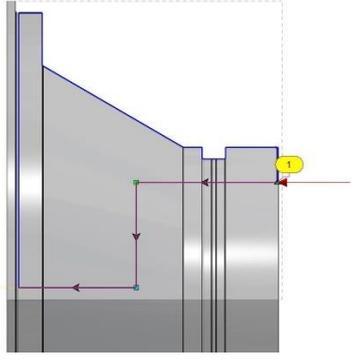
13.Selection of geometry :

- Left-click on 
- Left-click on the first side of the groove (image A)
- Left-click on the last side of the groove (image B)
- Left-click on Profil OK 
- No need to modify the approach and return vectors which are tangent to the elements shown.



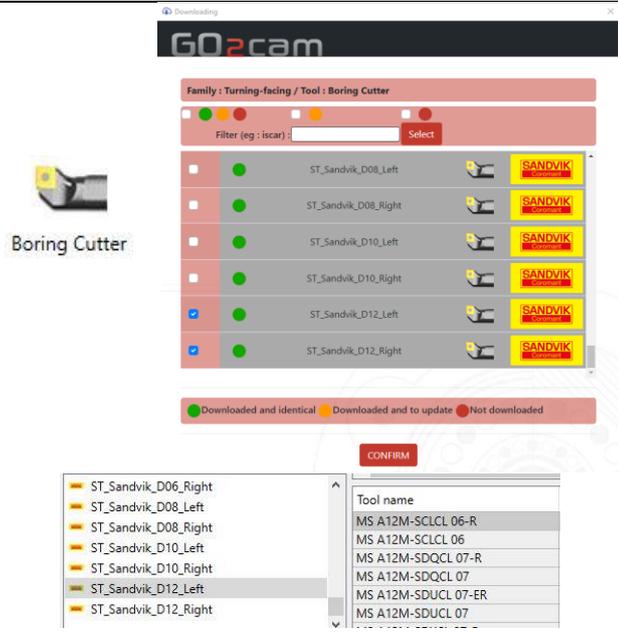
<ul style="list-style-type: none"> • Selection of Tool : • Left-click on Groove cutter  • Select MT_Sandvik_2020_Left under user • In the list of tool, left mouse click on " MS LF123J13-2020BM" 	 <p>Groove Cutter</p> <table border="1"> <thead> <tr> <th>User</th> <th>Tool name</th> </tr> </thead> <tbody> <tr> <td>MT_Sandvik_1616_Left</td> <td>MS LF123H25-2020BM</td> </tr> <tr> <td>MT_Sandvik_1616_Right</td> <td>MS LF123J13-2020BM</td> </tr> <tr> <td>MT_Sandvik_2020_Left</td> <td>MS LF123K08-2020C</td> </tr> <tr> <td>MT_Sandvik_2020_Right</td> <td>MS LF123G07-2020C</td> </tr> </tbody> </table>	User	Tool name	MT_Sandvik_1616_Left	MS LF123H25-2020BM	MT_Sandvik_1616_Right	MS LF123J13-2020BM	MT_Sandvik_2020_Left	MS LF123K08-2020C	MT_Sandvik_2020_Right	MS LF123G07-2020C
User	Tool name										
MT_Sandvik_1616_Left	MS LF123H25-2020BM										
MT_Sandvik_1616_Right	MS LF123J13-2020BM										
MT_Sandvik_2020_Left	MS LF123K08-2020C										
MT_Sandvik_2020_Right	MS LF123G07-2020C										
<p>14. Selection of machining cycle :</p> <ul style="list-style-type: none"> • Left-click on , then on Direct 	 <p>Direct</p>										
<p>15. Toolpath calculation :</p> <ul style="list-style-type: none"> • Left-click on Cycle Calculation  											

Ope 50 Boring

<p>16. Selection of geometry :</p> <ul style="list-style-type: none"> • Left-click on  General Ope • Left-click on geometry selection  • Left-click on the first element of the path, right (Figure A) • Left-click on the last element of the path, left (Figure B) • Left-click on Profil OK  	  
--	--

17. Selection of Tool :

- Left-click on tools , then on Boring cutter
- Left-click on 
- Select **ST_Sandvik_D12_Right** and **ST_Sandvik_D12_Left**.
- Left-click on **Confirm** to download the tools library.
- After downloading and installing, select **ST_Sandvik_1212_Left**
- In the list of tool, select **MS A12M-SCLCL 06-R**



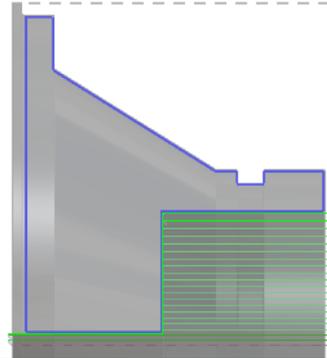
18. Selection of machining cycle :

- Left-click on Cycles , then on Roughing
- In the list of techno, left mouse click on "Main Spindle - Outside"



• Toolpath calculation :

- Left-click on Cycle Calculation 
- Right-click on the cycle and rename it to 'Inside Roughing'

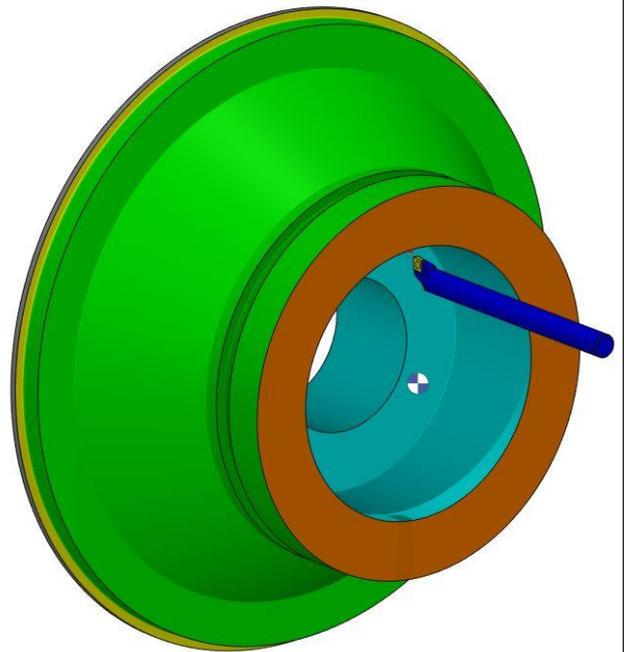


Simulation and NC blocks output

19. Simulation :

- Left-click on  NC File
- Left-click on **Simulation** 
- Left-click on  to activate the step by step mode. Then press the space key to go forward in the simulation step by step.
- If you prefer continuous simulation, click on 
- Left-click on  or *Esc* to stop the simulation

You can rotate the part by holding the left-click and moving the mouse.



20. NC Code:

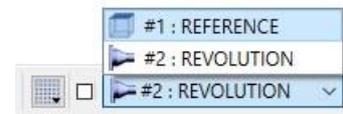
- Left-click on 
- Among the proposed list, left-click on the post-processor "T67_Fanuc"
- Left-click on **Open**
- Left-click on **Confirm**

The NC Code is generated.

C axis operations

1. Switching to Reference Plane for Milling:

- Go back to **Design** menu
- Left-click on current plane tab which is in the **Revolution** plan and change it to **Reference** plane



2. Design of a hole:

- Left-click on **Wireframe**, then select **Holes**
- Left-click on the function **Standard Holes**



- Select type of holes and choose **Through smoothed hole**
- Left-click twice in the **Diameter** box and type 10

Diam. : 10.00 mm

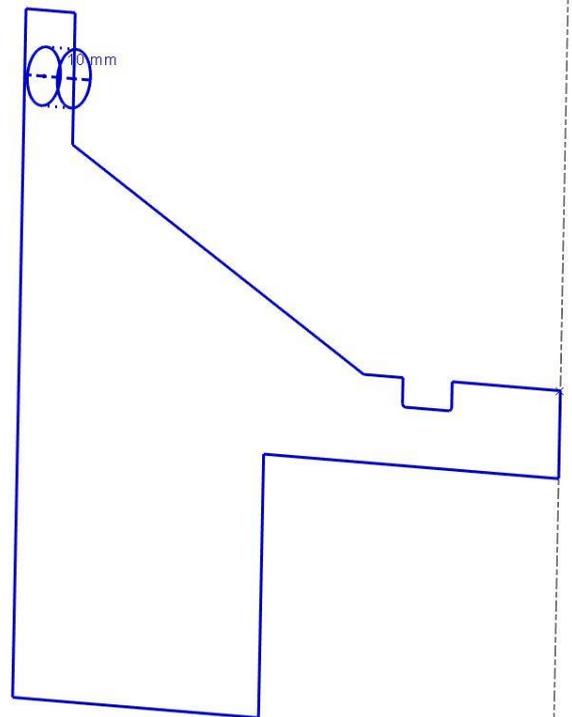
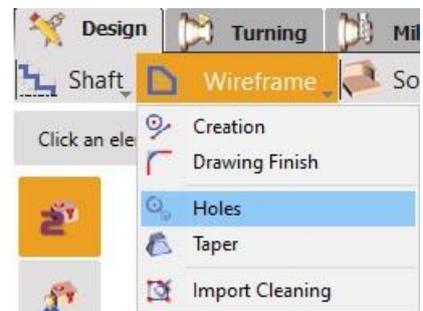
- Left-click twice in **Altitude** box and type -100

Altitude -100.00 mm

- Type the value for X as 292.5/2 and press **Enter**
- Type the value for Y as 0 and press **Enter**

X = 146.25 mm Y = 0.00 mm

- Left-click on



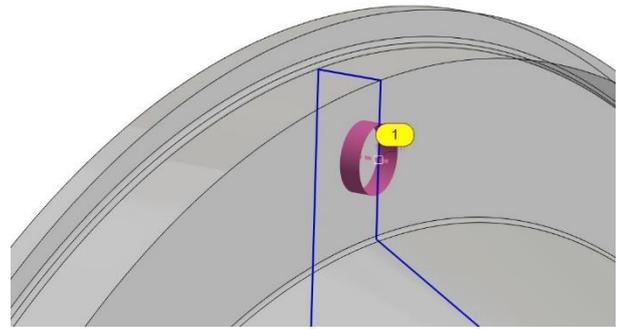
3. Start the milling:

- Left-click on **Milling**
- Left-click on **Hole/Feature**, then select **Manual Holes**



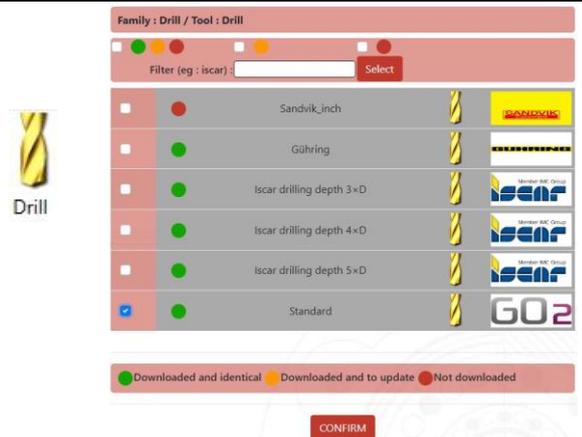
4. Selection of geometry :

- Left-click on 
- Left-click on the hole



• Selection of Tool :

- Left-click on tools , then on drill
- Left-click on 
- Select **Standard**
- Left-click on **Confirm** to download
- After the download and installation, select **Standard** under **User**
- Left-click on "Drill D10"



Iscar drilling depth 5×D	Drill - D10.5
Standard	Drill - D10

5. Selection of machining cycle :

- Left-click on Cycles  then on **Drilling**
- In the list of techno, Left-click on **Simple mode Auto**



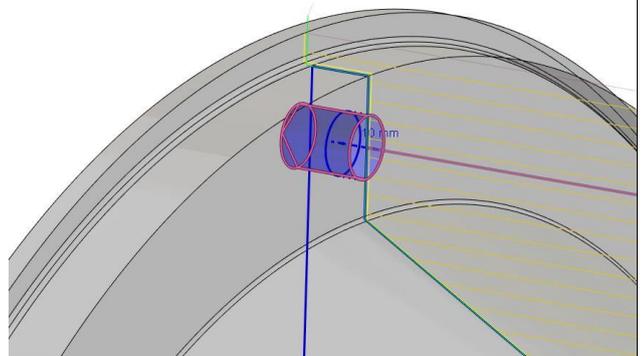
Drilling

Techno. name	Depth	Type
Simple mode Auto	0.00 mm	Simple

• Toolpath calculation :

- Left-click on **Cycle Calculation** 
- Right-click on the cycle and rename it to 'Drilling D10'

The path is displayed on the screen.



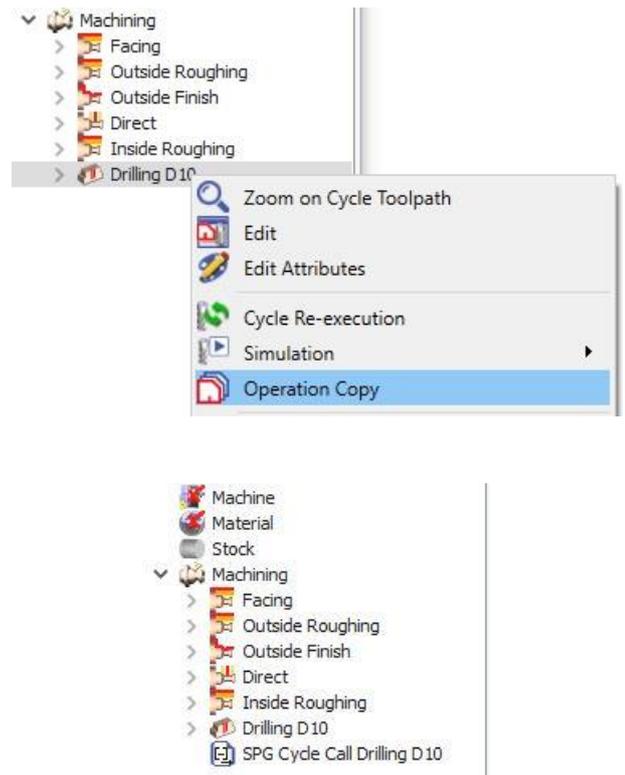
6. Repeat the cycle in the machining tree:

- Left-click on **Drilling D10** cycle in machining tree
- Select **Operation Copy**
- Choose **Rotation**
- Left-click twice in **Number** field, type in **11** and press **Enter**
- Check that the coordinates of the center are **0, 0, 0**
- Left-click twice in **Angle** field, type in **30**, **Enter** key

Rotation Number 11 Center 0.00 mm 0.00 mm 0.00 mm Angle 30.00 deg

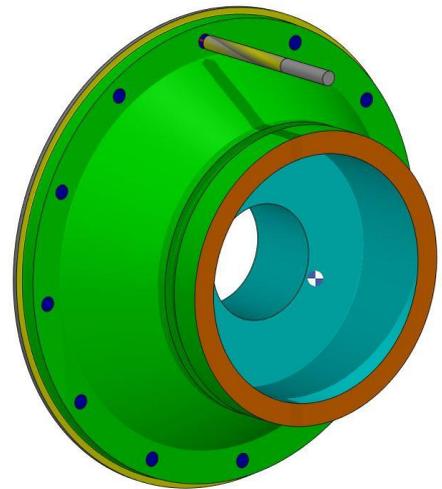
- Left-click on **Drilling** in the machining tree to position the subprogram call after the drilling

The 12 holes are now machined.



7. Simulation :

- Left-click on  **NC File**
- Left-click on **Simulation** 
- Left-click on 



8. NC Code :

- Left-click on 
- Among the proposed list, click on the post-processor **T67_Fanuc**
- Left-click on **Open**
- Left-click on **Confirm**

The NC Code is generated.