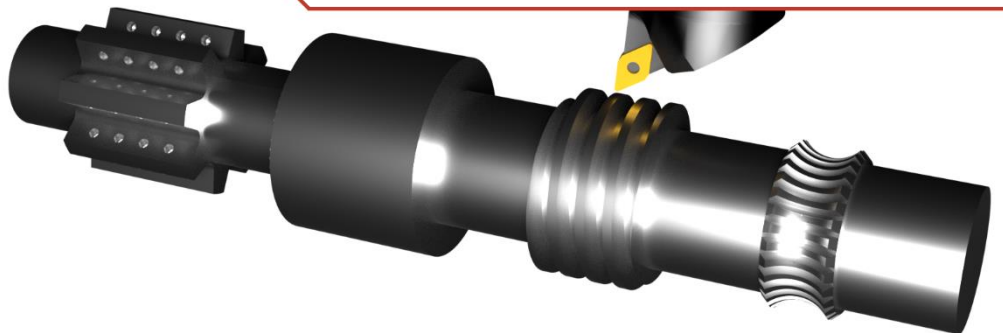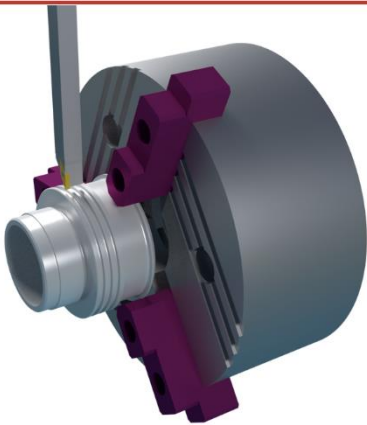GO2cam

GO2cam V6.10
Tutorial
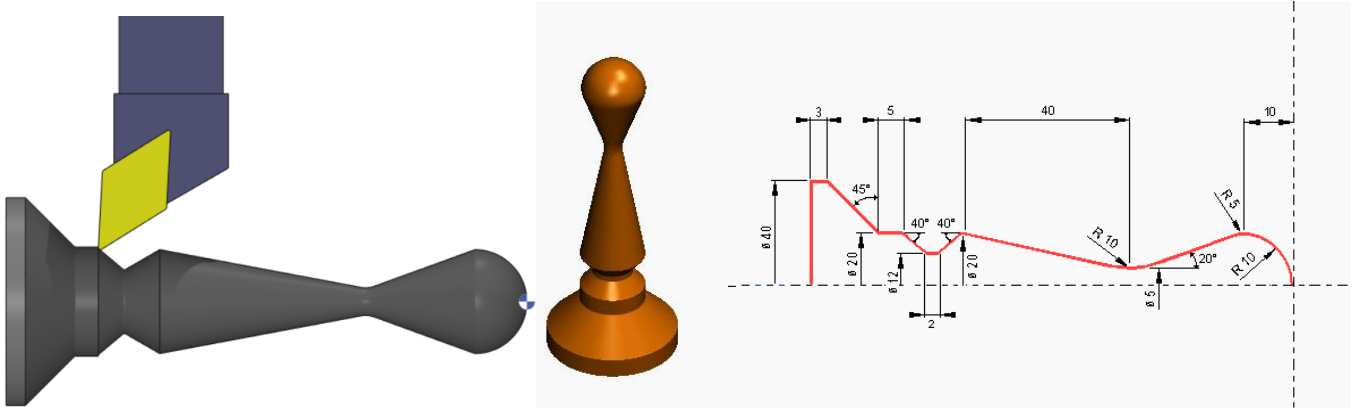# T01 – Pawn

GO2cam
FRANCE

T01 – Pawn

Introduction

Welcome to this chess pawn 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 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.
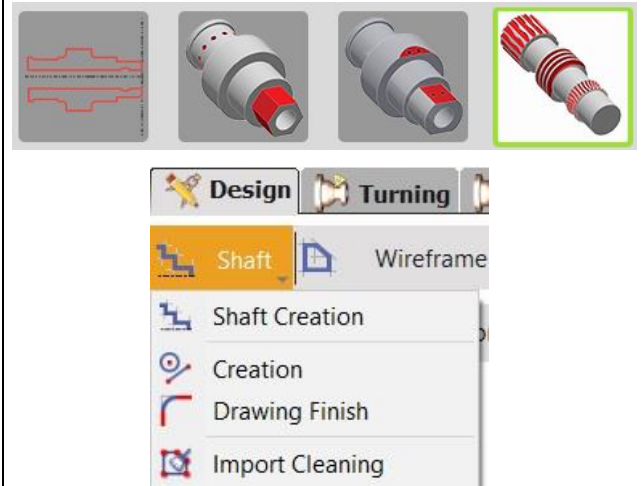
I. Process for the Design :

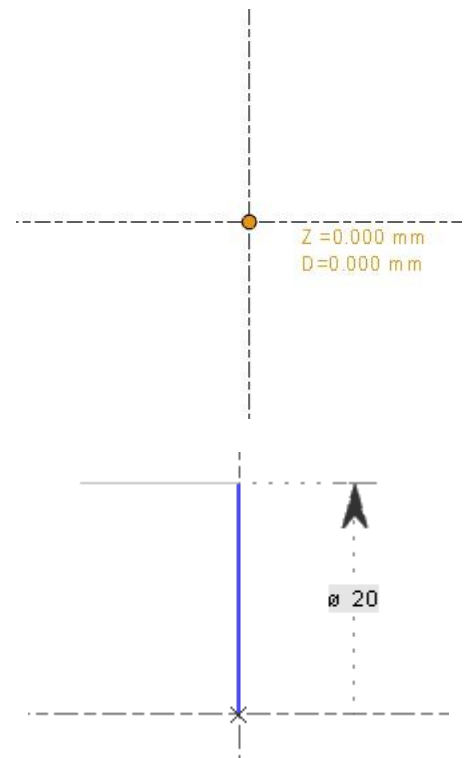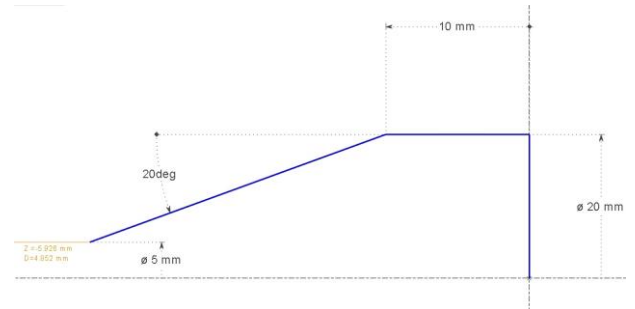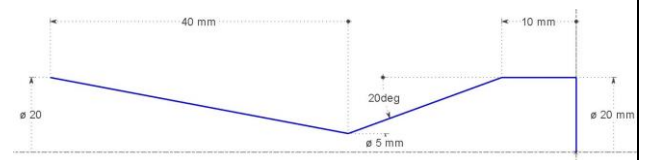|	
1. Choice of the Turning environment in the homepage : • Left-click on the Turning icon the most on the right Note: the icons represent the type of product. If your licence do not include it, the product icon is greyed. The software opens in design mode by default. • Left-click on Shaft • Left-click on Shaft Creation	
2. Sketch of the part : • Left-click on the Shaft Creation • Left-click to check that the is activated in order to choose the direction of the first element • Choice of the starting point: Left-click directly on the origin of the reference plane like shown in the image. GO2cam proposes alternate lengths and diameters. This choice is shown by an orange line linked to the motion of the cursor. • Type in 20and press the Enter key.	
3. Length: 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 the position of z^-, if it is not, Left-click on the button to change the direction.** • Type in 10, Enter key.	

GO2cam
FRANCE
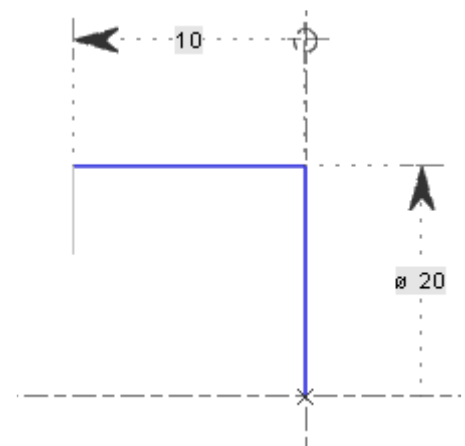
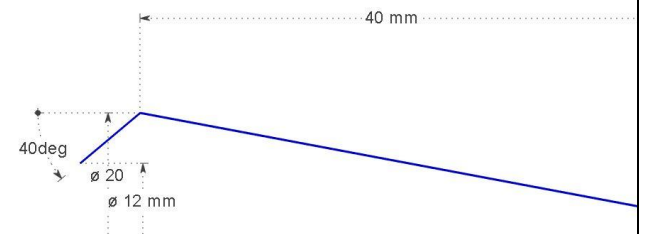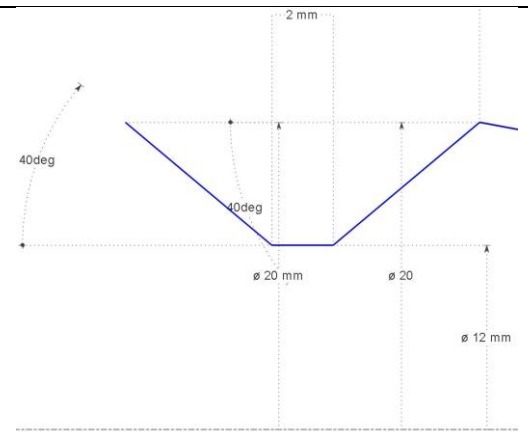|	
4. Creation of a cone A=20˚, Ø=5 : • Left-click on • Choose the mode of creation of the cone clicking on until it becomes so that you define the cone by its angle and diameter. • Type in 20 for the angle, Press the Enter key, Type in 5 for the Diameter, Press the Enter key • Left-click on	
5. Creation of a cone L=40, Ø=20: • Change the mode of creation of the cone clicking on until it turns to , so that you can define the cone by its length and diameter. • Type in 40 for the length and 20 for the Diameter • Left-click on	
6. Creation of a cone A=40˚, Ø=12: • Change the mode of creation of the cone clicking on until it turns to , so that you define the cone by its angle and diameter • Type 40 for the angle and 12 for the Diameter • Left-click on • Left-click on to frame all geometric elements on the screen.	
7. Length : • Left-click on • Type 2 and press Enter	

GO2cam
FRANCE
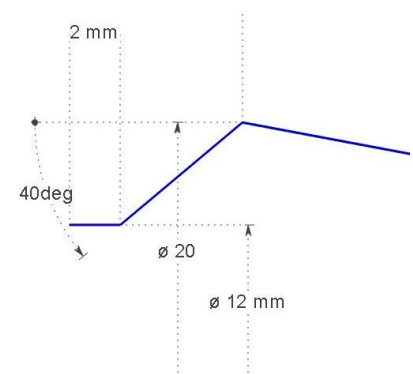
8. Creation of a cone A=40° , Ø=20: • Left-click on • Type 40 for the angle and 20 for the Diameter • Left-click on	
9. Length: • Left-click on • Type 5 and press **Enter**	
10. Creation of a cone A=45° , Ø=40 : • Left-click on • Type 45 for the angle and 40 for the Diameter • Left-click on	
11. Length Z=3 : • Left-click on • Type 3, Enter • Left-click on	
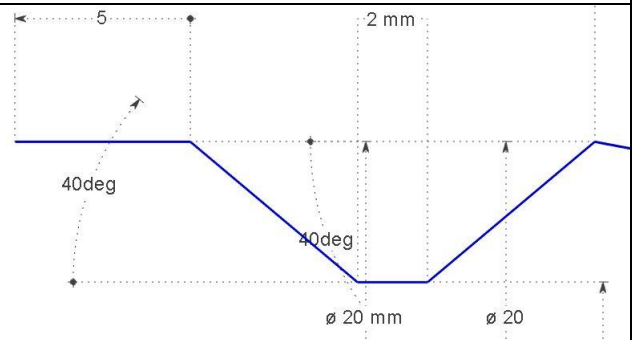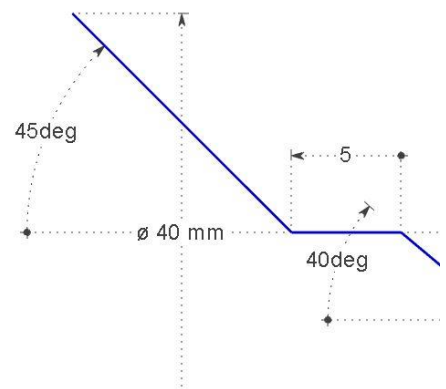
GO2cam
FRANCE
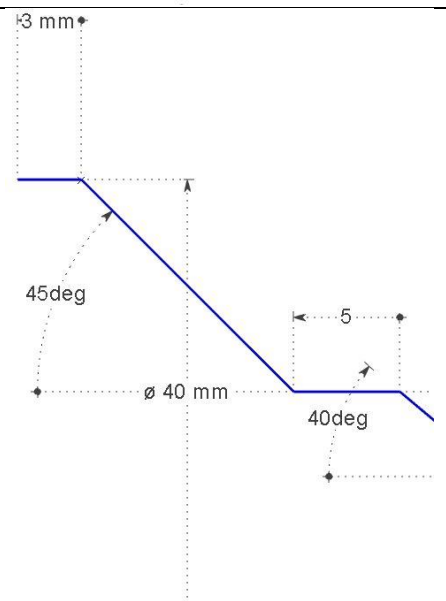
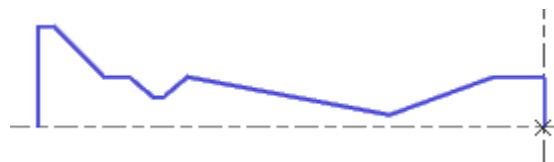|	
12. Diameter Ø=0 : To finish the part, we can click on existing elements as guide: • Left-click on the horizontal axis • Click to verify ✔	
13. Finishing Design : • Left-click on Zoom 🔍, then left-click on the screen twice to select the zone you want to magnify like shown on the picture • Press the 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 10 for the Radius and press Enter • Left-click on the 2 corners where are located the fillets For the 5 mm fillet, it is advised to click on the 2 elements one after another. The order of selection is not important. • Type in 5 and [Enter key • Left-click on an element (image A) • Left-click on the other element (image B) • Esc key to exit the Fillet function • Left-click on 🔍 The part is finished	

GO2cam
FRANCE
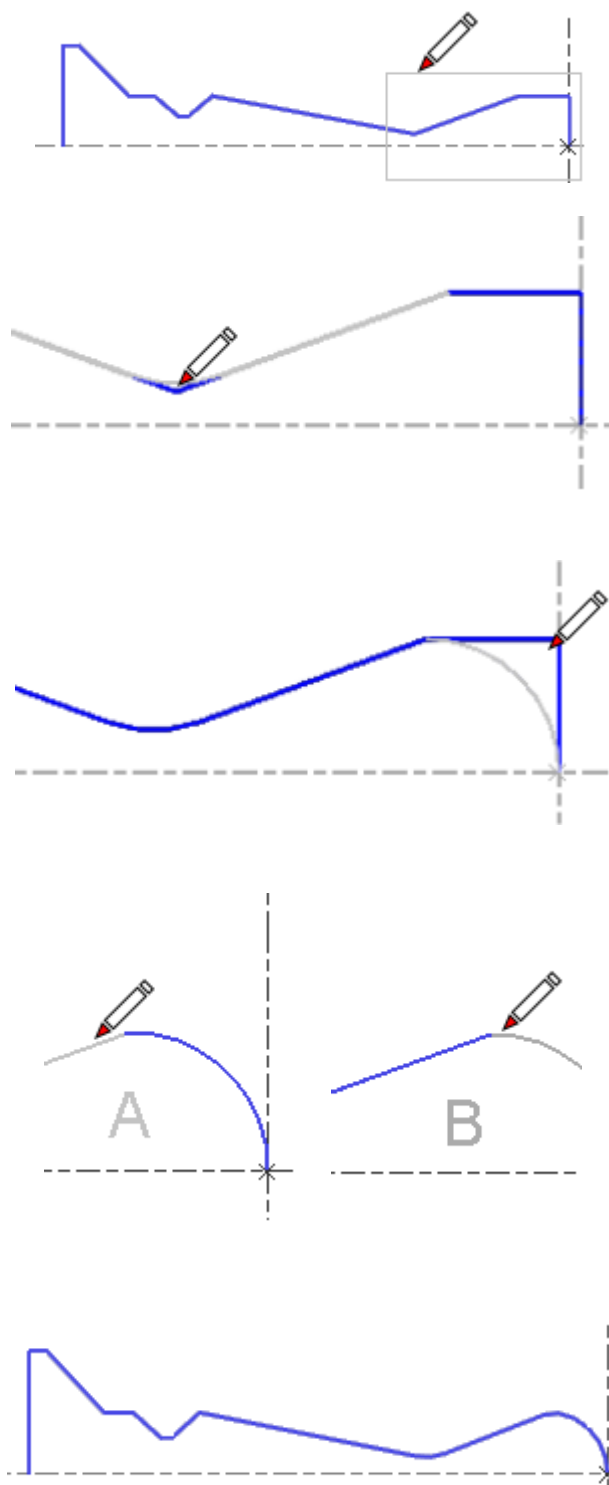
II. Process for the Machining:

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

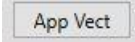|	
1. Turning : • Left-click on the **Turning** menu The stock is automatically created in relation to the profile. **Note :** It is a cylindrical stock defined with a constant 5 mm overflow around the profile. We are going to modify the stock manually. • In the Zmaxi field, type 2 to increase the stock length by 2mm from the origin • Then enter the value of **-110** for the **Zmini** • Left-click on validate ✔	 Overflow = 5.000 mm Cylindric Zmini = -110.000 mm Zmaxi = 2.000 mm

GO2cam
FRANCE
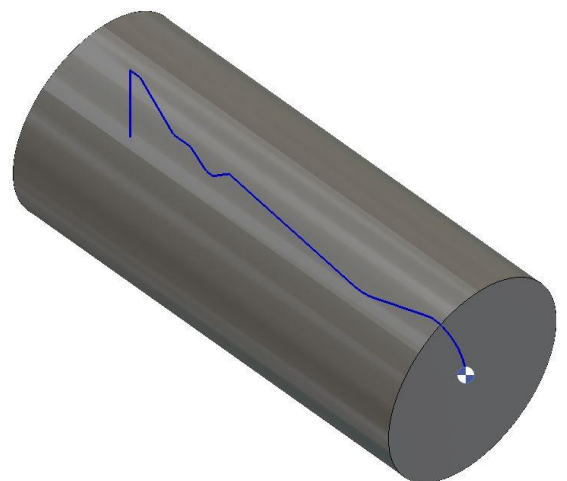
Ope 10 Roughing

2. Geometry selection :

- **Left-click on** | General Ope |

- Left-click on Geometry Selection

- Left-click on the first arc, on the bottom (image A)

- Left-click on the horizontal segment, on the left (image B)

- Left-click on **Profil OK**

Note: the approach and return vectors are tangent to the elements shown.

- Left-click on | App Vect | , to change the value dynamically on the screen. Click on the screen when 180° is displayed.

- Repeat the same action for the Return Vector, with the value of 90°.

| Ret Vect | ☑ | 90.00 deg | ⬍ |
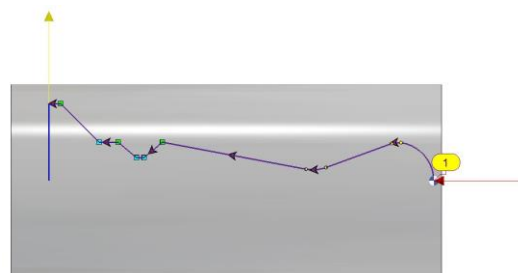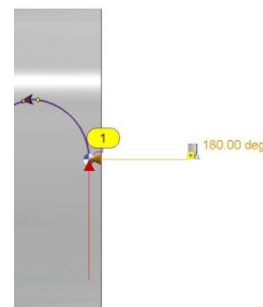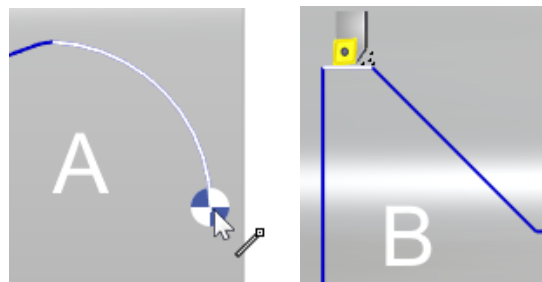
3. Tool selection :

- Left-click on , then on Turning Facing Cutter

- Left-click on | ⬇ |

- Select **ST_Sandvik_1212_Right** and **ST_Sandvik_1212_Left.**

- Left-click on **Confirm**

- After downloading and installing, select ST-SANDVIK-1212-LEFT under "User"

- In the list of tool, select **MSSDJCL1212K07-S**

- Tool light turns green

ST_Sandvik_1212_Left	
ST_Sandvik_1212_Right	
ST_Sandvik_1616_Left	
ST_Sandvik_1616_Right	

MS SDACL 1212K 11-S	D
MS SDJCL 1212K 07-S	D
MS SDJCL 1212K 11-S	D
MS STJCL 1212K 11-S	T
MS SVABL 1212K 11-S-B1	V
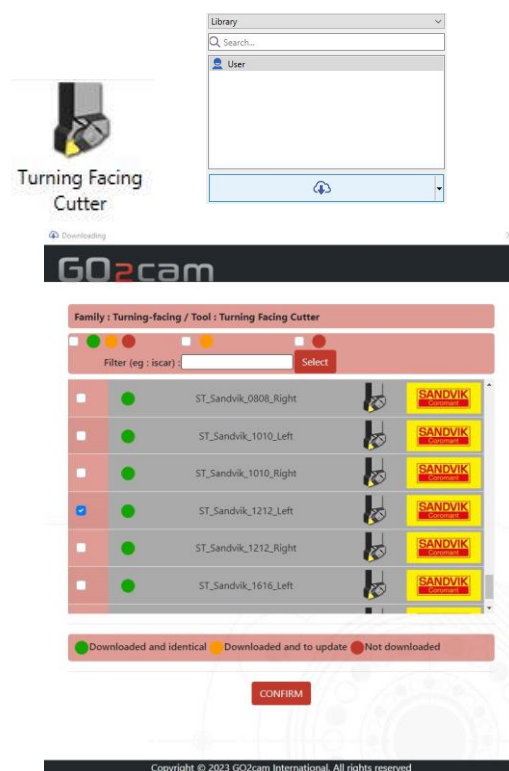
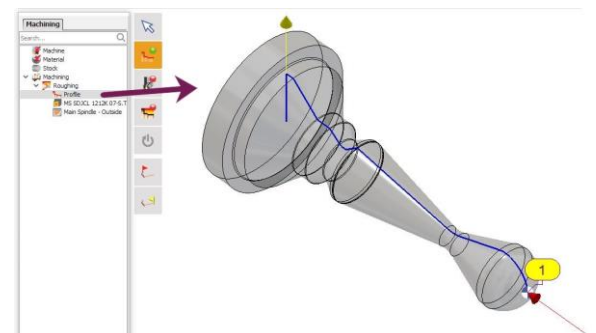|	
4. Machining cycle selection : • Left-click on then on Roughing • In the list of techno, Left-click on **"Main Spindle - Outside"**	Roughing | TECHNO NAME	Cycle type	Toolpath	Offset type
---	---	---	---
Axial Hole Roughing with Finish	Inside	Ima..	Center
ETST	Outside	Ima..	Center
Main Spindle - Face	Face	Ima..	Center
Main Spindle - Outside	Outside	Ima..	Center
Nashil	Inside	Part ..	Center
Sub Spindle - Face	Back	Ima..	Center
Sub Spindle - Outside	Outside	Ima..	Center	
5. Toolpath calculation : • Left-click on **Cycle Calculation** The toolpath is calculated and the evolutive stock is machined.	
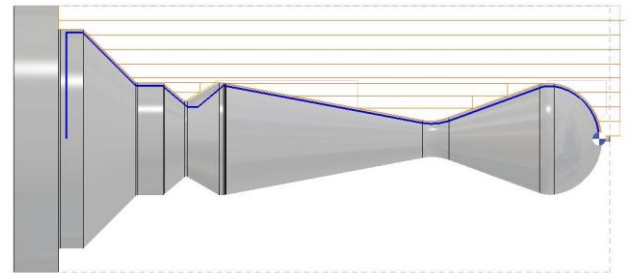
Ope 20 Finishing

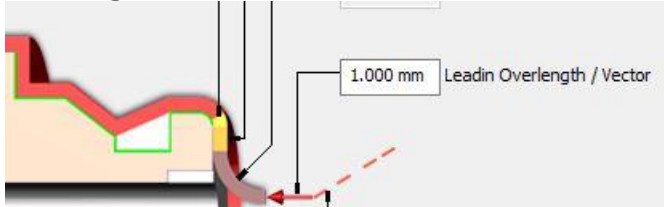|	
6. Geometry selection : The menu General Ope is still active. • Left-click on • In the machining tree click on > to expand the roughing cycle details • Left-click on the 'Profile' drag and drop it on the screen **Note: This operation allows the copy of the existing machining profile, thus avoiding the repetition of the same selection.**	
7. Tool selection : • Left-click on tools then on Turning Facing Cutter • Select **ST_Sandvik_1212_Left** below user • In the list of tool, select **MS SDJCL1212K07-S**	Turning Facing Cutter ST_Sandvik_1212_Left ST_Sandvik_1212_Right ST_Sandvik_1616_Left ST_Sandvik_1616_Right MS SDACL 1212K 11-S MS SDJCL 1212K 07-S MS SDJCL 1212K 11-S MS STJCL 1212K 11-S MS SVABL 1212K 11-S-B1

GO2cam
FRANCE

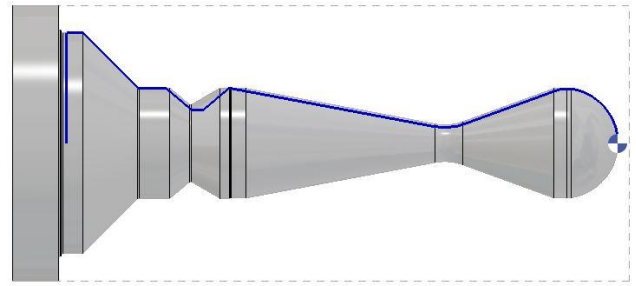|	
8. Machining cycle selection : • Left-click on Cycles then on **Finish**. • In the list of techno, Left-click on Main Spindle-outside. • In the Movement tab, set the Leadin Overlength/Vector to 1. 	Finish
9. Toolpath calculation : • Left-click on Cycle Calculation	

GO2cam
FRANCE

Modification of a cycle

10. Use of the machining tree:

The machining tree is a working tool that allows :
- Obtaining the step-by-step list of operations with all the useful information,
- Modifying the operations (selected geometry, technology, tool)
- Manipulating the cycles : copy, move in the list etc.

**Edit a cycle for modification**

- Note: The Roughing and Finishing cycles have machined the groove, as shown in the image.
- We will modify a parameter of these cycles to automatically avoid this groove.

To obtain this result, we are going to modify one option in the cycles:
- Left-click on the button ⟩ located in front of the name of the Roughing operation to expand the cycle information.
- Left-click twice on the line 'Main Spindle-Outside'.

This strategy dialogue box gathers all the parameters useful to calculate the toolpath.

- The option **Undercut** is set to '**No**', Left-click on '**Yes**',
- Left-click twice in « **Undercut min width** » field, type in 15, and press the Enter key
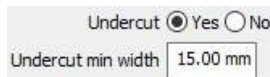
Left-click on Execute, the machining will be updated.
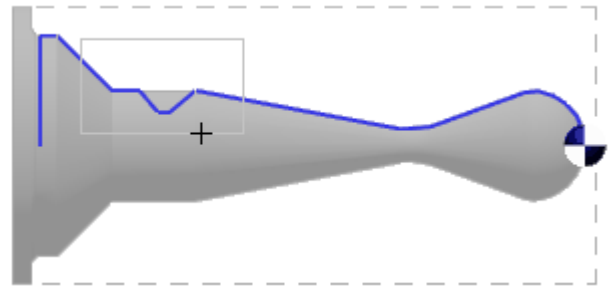
Repeat the same for the Finish cycle.

This setting will only machine the first sub-pocket(>15 mm width) and the second sub-pocket will not be machined (the groove with <15 mm width)
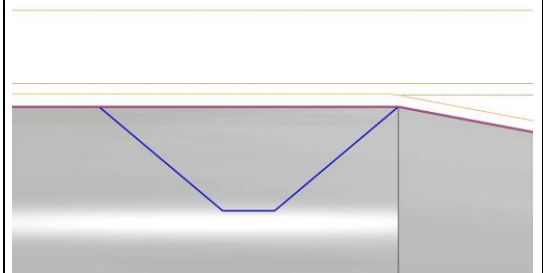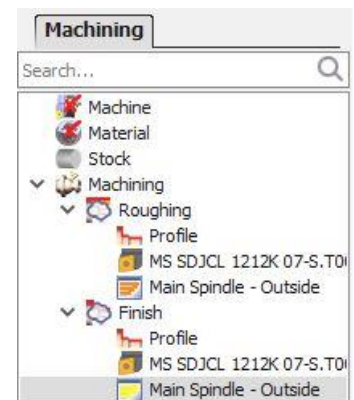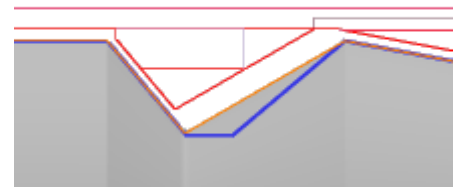
Ope 30 Grooving

<table>
<tr>
<td>

11. Start :
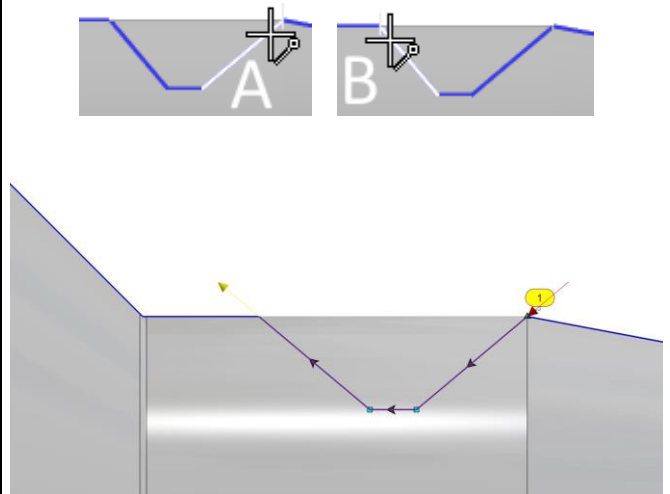
- Left-click on Grooving

- Left-click on Zoom 🔍, then Left-click on the screen twice to select the area you want to see in details.

</td>
<td>

</td>
</tr>
<tr>
<td>

12. Geometry selection :

- Left-click on

- Left-click on the first flank of groove, on the top (image A),

- Left-click on the second flank of groove, on the top (image B),

- Left-click on **Profil OK**

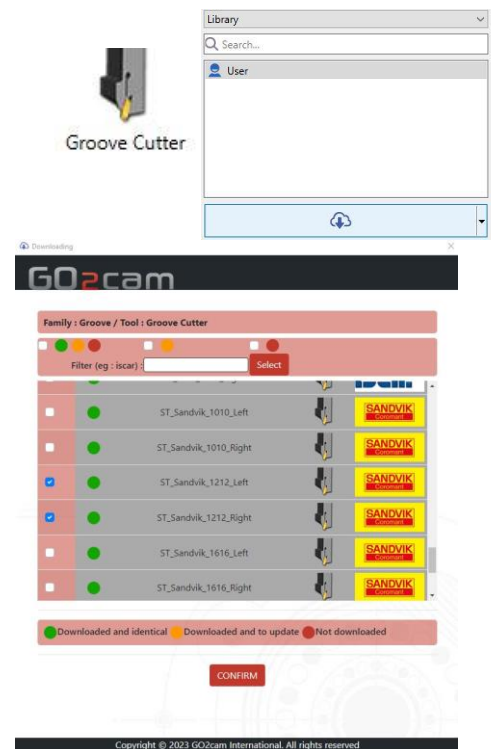No need to modify the approach and return vectors which are tangent to the elements shown.

</td>
<td>

</td>
</tr>
<tr>
<td>

- Tool selection :

- Left-click on , Then select the Groove Cutter

- Left-click on [⬇]

- Select **ST_Sandvik_1212_Right** and **ST_Sandvik_1212_Left.**

- Left-click on **Confirm** to download the tool data

- After downloading and installing, select ST-SANDVIK-1212-LEFT in the tool's library.

- In the list of tool, select **MS LF123E11 1212B-S**

</td>
<td>

</td>
</tr>
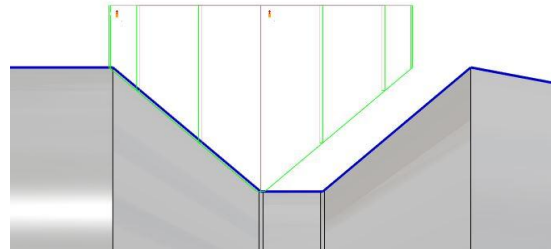</table>

13. Machining cycle selection : • Left-click on [icon], then on **Direct**	 Direct
14. Toolpath calculation : • Left-click on **Cycle Calculation** [icon]	

Simulation and NC blocks

15. Simulation : • Left-click on [icon] NC File • Left-click on **Simulation** [icon] • Left-click on [icon] to reach the step by step mode, Then click on the [icon] or press the space key to go forward. • To simulate in continuous mode, Left-click on [icon]. • Left-click on [icon] or Esc to stop the simulation **Note :** You can rotate the part by pressing and holding the left-click button and moving the mouse.	

GO2cam
FRANCE
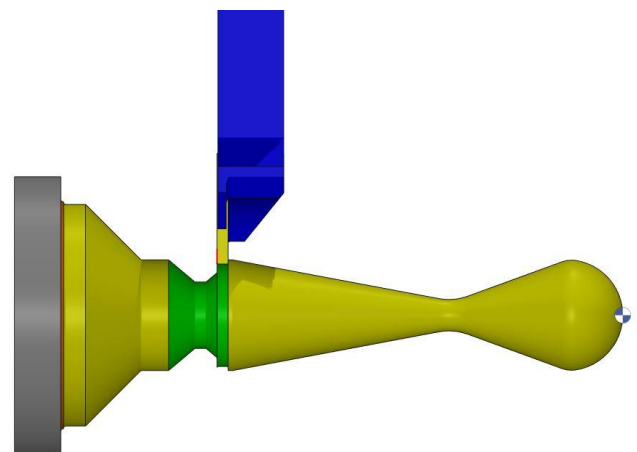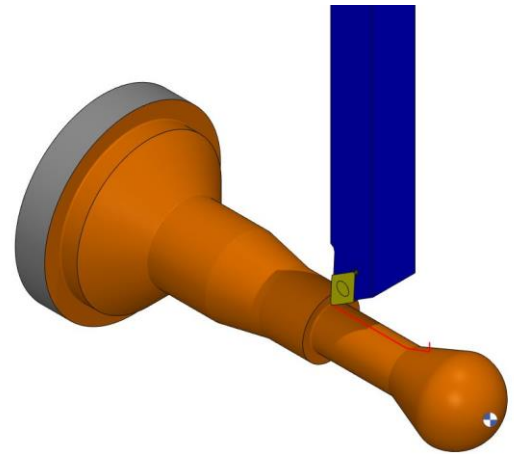
16. NC Code:

- Left-click on

- Among the proposed list, Left-click on post-processor « **T67_Fanuc** »

- Left-click on **Open**

- Left-click on **Confirm**

  The NC codes for the machining of this workpiece are generated.