GO2cam

GO2cam V6.10 Tutorial T05 – TurnMill
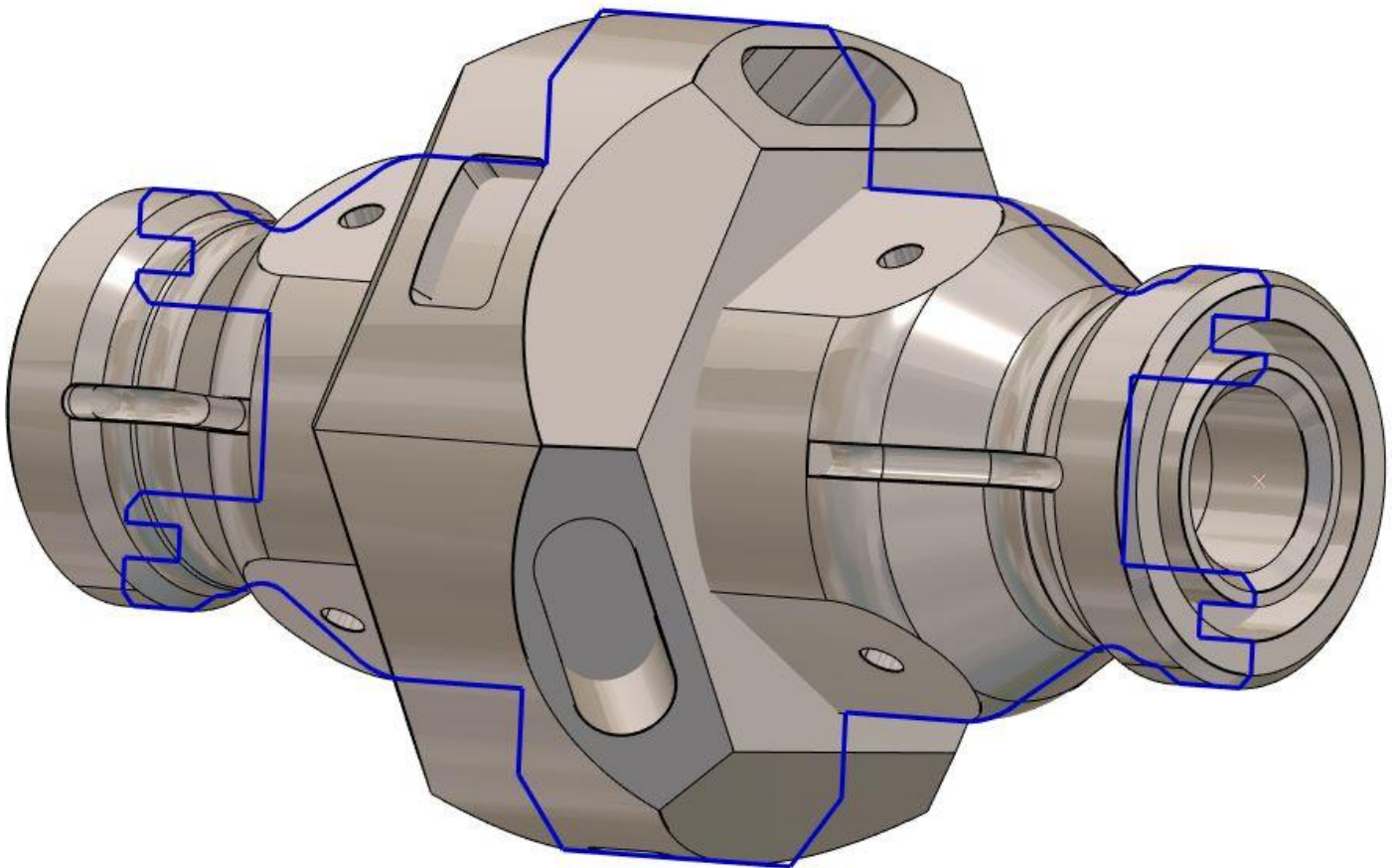
GO2cam
FRANCE

T05 – TurnMill

Introduction

- Import of a revolved 3D solid

- Use of Opelist for machining

- Create axial, radial, and inclined planes

- Creation of developed plane

- Projectionn of profiles in the developed plan

- Application of turnmill operation

Additional Files

You will find in the file :

- PDF file for this tutorial

- Part files for this tutorial: T05_TurnMill.PCE

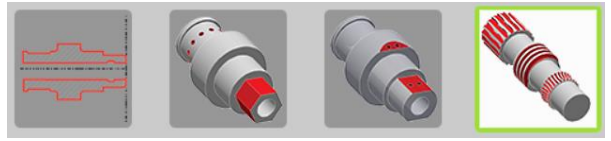- 3D model in Parasolid file format: T05_TurnMill.XT

- Opelist file: T05_Opelist.OPL

I. Import and positioning of parts :

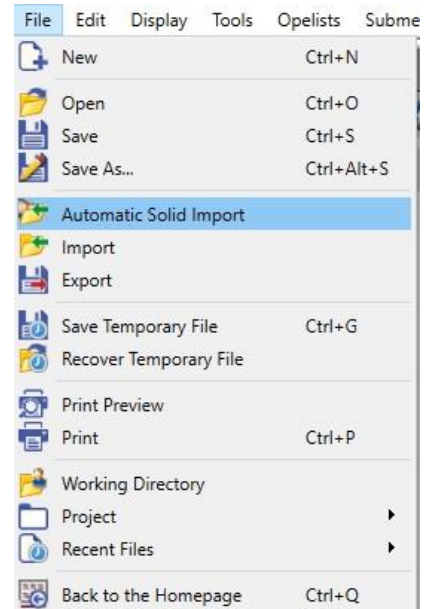|	
1. Start : • In **GO2cam,** left click on the TurnMill module. The module opens with the Design tab active by default.	
2. Import of solid part: • Enter the **File** menu • Left click on Automatic Solid Import • Select the file **T05_TurnMill.XT** from your directory and left click on Import. The solid part is imported.	
3. Positioning : This command is still active. We must determine whether the part is properly oriented. We will rotate the workpiece 180 ° around the Z axis to move the groove face towards us • Left click on **Turn around Z Axis** • In the field, type a value **180** and press Enter • Increase tolerance to 0.005 • Change the silhouette to Symmetrical silhouette. • Left click on ✔ to validate	

GO2cam
FRANCE
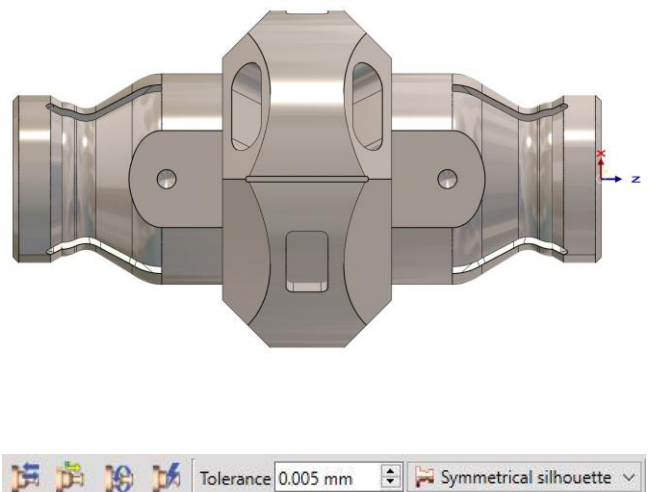
4. Defining the stock:

A cylindrical stock is automatically created for the workpiece with a constant overflow of 5 mm by default.

- Left click on `D outer = 120.000 mm` and change the value to 120 to define the maximum diameter of the stock.
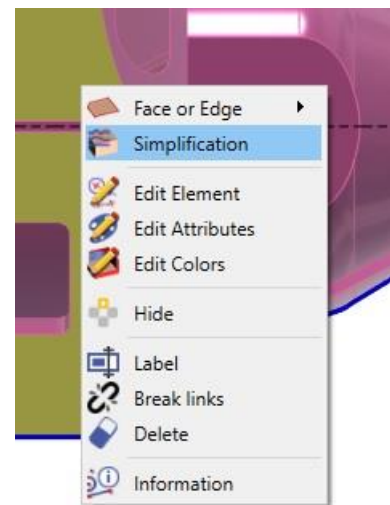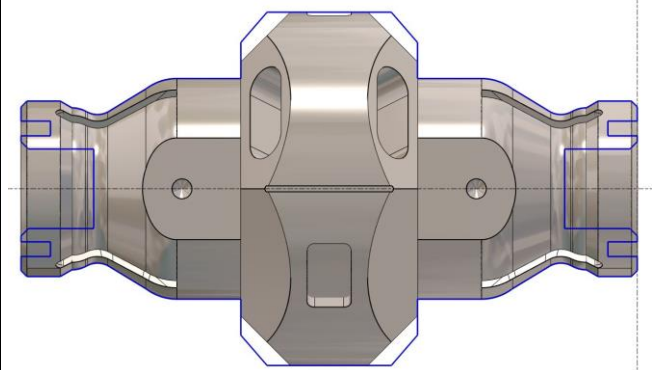
- Left click on ✔ to validate
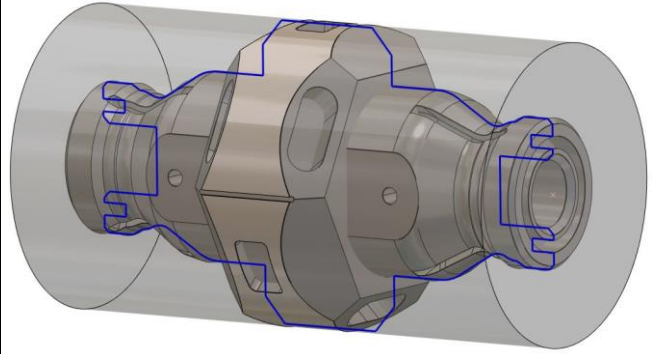
5. Position of origin:

By default, the origin is at the end of the part.

- Since the origin is in the correct position for this workpiece, Left click on ✔ to validate.

Note: Right click on the solid and choose Simplification.

The part is now ready for machining.

GO2cam
FRANCE

II.   Turning operation:

1. Operation Menu:

Note: This menu is dedicated to the opelist created by the user. If there is no opelist, this menu will not be displayed.

You must copy the file **T05_Opelist.OPL**, in the opelist directory of the **GO2cam**

Standard path:

➡ C:\GO2cam_Intl\GO2camV610\opelist

2. Loading the opelist:

- Left click on the Opelists Menu

- Go to **My List**

Note: If there is no other opelist, your opelist will automatically load, else you can click to expand it and find your opelist.

- Select the opelist **T05_Opelist.OPL**

The following dialog box opens: If Opelist is customized, you can change the settings. And provide cutting conditions.

- Left click on **Validate** ✔ to apply the automatic opelist cycles.
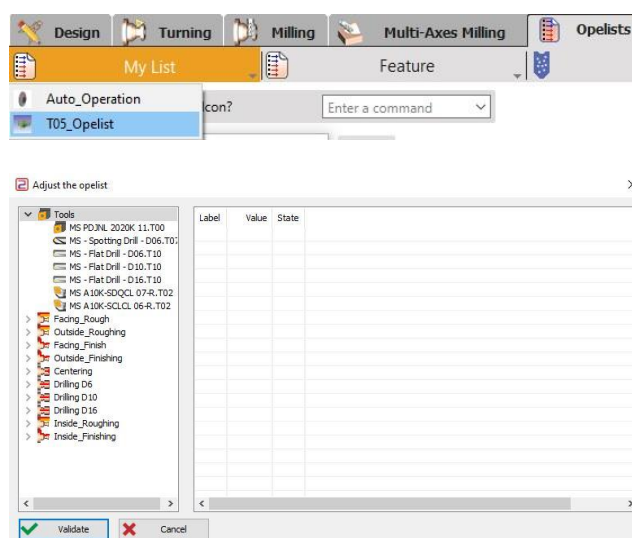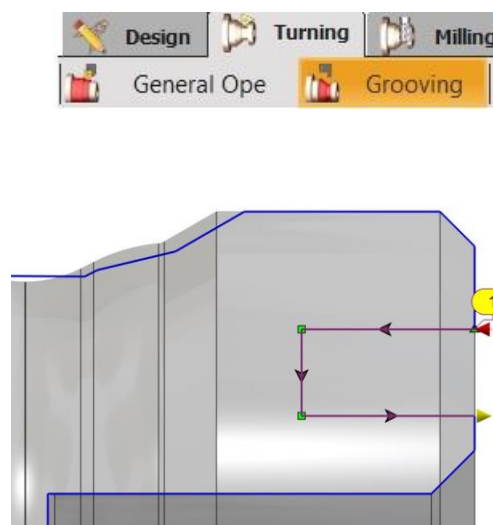
3. Creation of the grooving cycle:

a) Profile definition:

- In the **Turning** menu, Left click on Grooving

- Select the geometry

b) Define the tool:

- Left click on

- Left click twice on Groove Cutter, to access tool parameters settings

- Change the Body Half Taper Angle, Cutter width and thickness values as per the image.

GO2cam
FRANCE

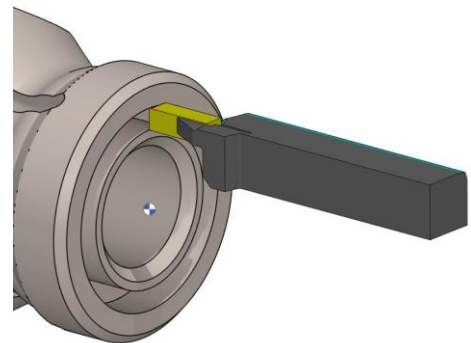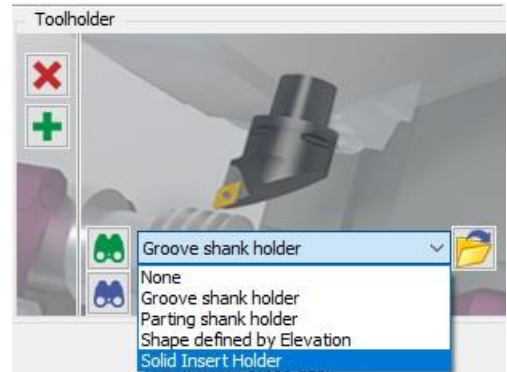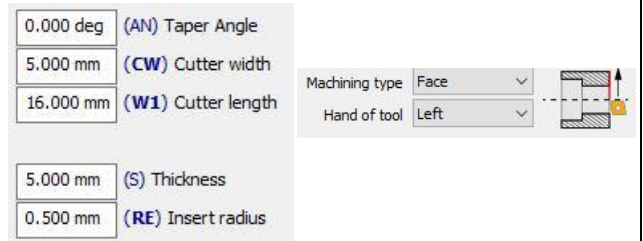|	
• In the machining type drop-down menu, select "Face" We will insert a solid holder and define it to forward direction. • Left click on the icon ➕ To add a holder • Left click on the menu Groove shank holder, Then replace the Groove shank holder with a Solid Insert Holder • Left click on the icon • Select the tool holder provided with the document : **GHFGL25-80-8-AV.SYM** You must have copied this file in the sym directory in: C:\GO2cam_Intl\GO2camV610\sym • Change the orientation value to 0, So that the tool holder is in the forward position c) Choice of cycle: • Left click on • Select the **Direct** cycle • Change the Z step value to 2 • Calculate ⏻	0.000 deg **(AN)** Taper Angle 5.000 mm **(CW)** Cutter width 16.000 mm **(W1)** Cutter length Machining type Face Hand of tool Left 5.000 mm **(S)** Thickness 0.500 mm **(RE)** Insert radius Toolholder Groove shank holder None Groove shank holder Parting shank holder Shape defined by Elevation Solid Insert Holder Drawing
4. Simulation : In order to check our toolpath, we will simulate this: • Left click on Simulation • Check whether the dynamic mode is active, otherwise select this mode. In the display bar for solid on the right side of the screen, you can choose to view the solid with a **120 °** section during simulation : • Left click on the blue arrow to expand **Solid Transparency** • Select the check box with the section icon • Left click on ⏩ To start the simulation in step by step mode • Left click this icon again or press the space bar to advance step by step	

GO2cam
FRANCE
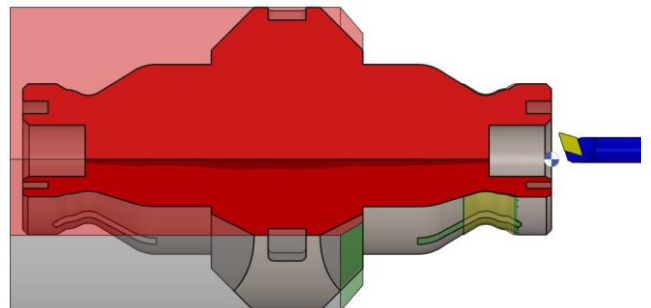
III.   Milling operations:

1. Creation of a custom view:

We will create two custom views, a **Revolution** view and **Reference** view, because the Reference view cannot be accessed in the Turning menu. This is the same for the rotating view which is not available in the Milling menu.

- Set the workpiece in the Revolution plan while in the Turning menu

- Press Ctrl+F2 on the keyboard at the same time

  The Revolution view is now saved as a custom view and assigned to the F2 key.

- Switch to the Milling menu and the solid will automatically be oriented in the Reference plane

- Press Ctrl+F3 on the keyboard at the same time

  The Reference view is now saved as a custom view and assigned to the F3 key.

You can use Ctrl+F4 to create a third custom view.
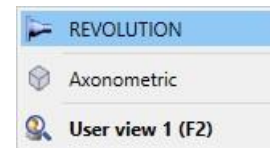
2. Creation of a radial machining plane :

We will create a machining plan using GO2cam's interactive axis system in order to machine the three radial flats faces and their holes.

- In the Milling menu, Press F2 on the keyboard to switch to the Revolution view

- Left click on the origin of the interactive axis system, hold the left click and move the mouse over the radial face. [You can release your hold on the left click at this point]

The radial face is highlighted and the axis is snapped to the center of the face. Left click on the face to have the plane creation bar open.

- In the plane name field, type the Radial plane
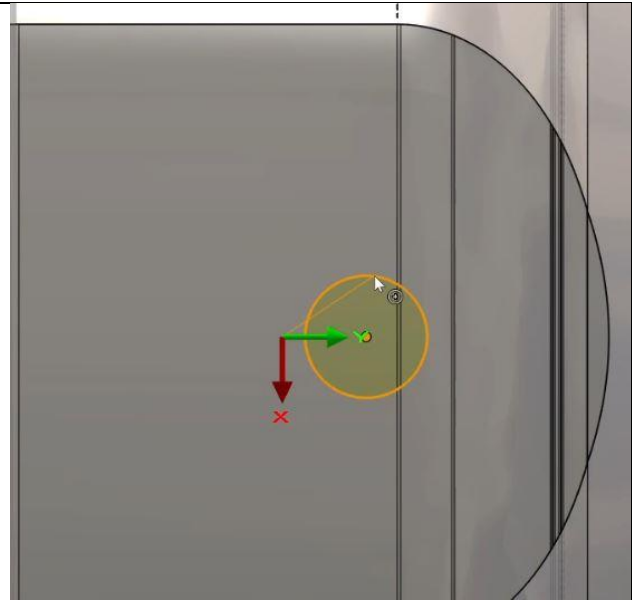
GO2cam
FRANCE
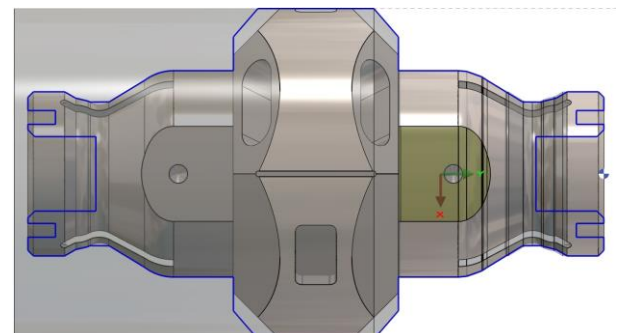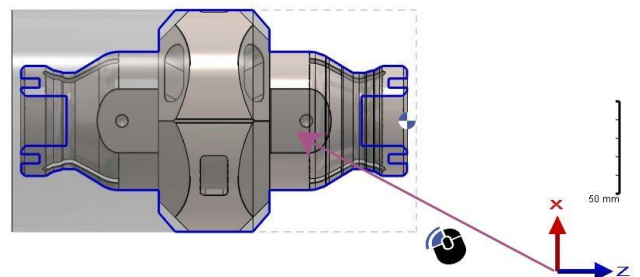
- Left click on to move the plane origin to the center of the hole

- Press and hold Ctrl+Shift key simultaneously and place the cursor on the edge of the hole, and the center point will be highlighted, left click to validate

  The axis is now in the center of the hole surface. To complete the axis must be oriented at 90˚.

- In the orientation field, enter a value of 90

- Left click on to validate

The Radial plane is now created. We will proceed to machine this face.

3. Display of axes:

We have just created a machining plane.

In GO2cam, you can display the axis of the origin of the created plane. To do this, you must activate a filter.
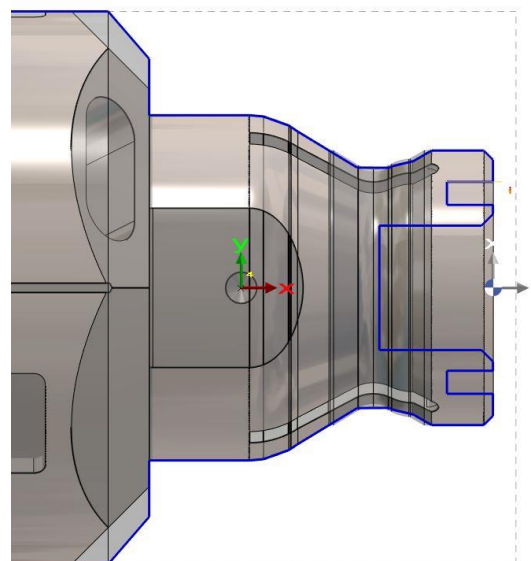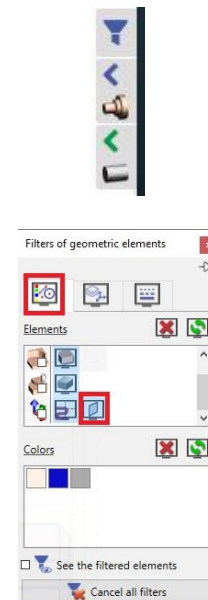
- Go to the filter page

- In the list, Left click on 'Elements and Colors'

- Select the "workplanes systems of axis" check box

Note:

To display the marks, you have two options:

- Show all marks

- Show marks in the current plan

GO2cam
FRANCE

4. Machining on this plane:

a) Machining of the face:

- Enter **Standard** [icon]

- Left click on [icon]

- The selection mode is automatic by default

- Left click on the **bottom face**, Start and end height is acquired from solid and stock

- Left click on [icon]

- Choose **Flat End Mill**, then select the tool **"Flat End Mill D16"** in **Standard**

- Left click on [icon]
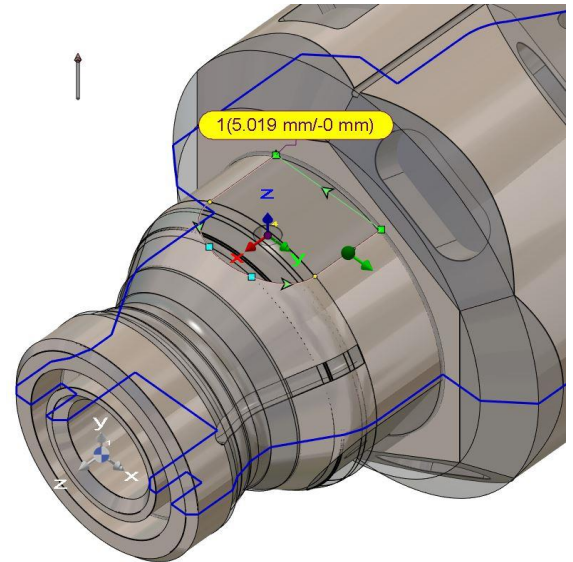
- Choose Pocket+Contour

- Left click on [icon]

b) Drilling Operation - M8 :

- Enter in **Hole/Feature**, select **Manual Holes**

- Left click on [icon], And select the hole

- Left click on [icon], The information page is displayed in the lower right corner, showing the drilling depth and diameter.

Note: On the part, two entities of holes are visible, one corresponding to the drilled hole and the other to the tapped hole, with different depths. The drilled hole goes 5mm deeper than the tapped hole, indicating the depth.
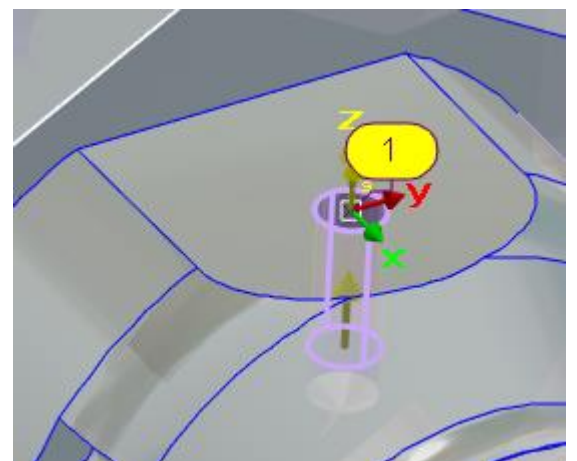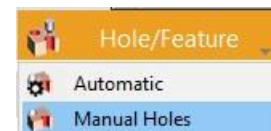
- Check if you are on the X+(radial) and C axis

1(5.019 mm/-0 mm)

Flat End Mill

Tool name	Diameter	Useful length
Flat End Mill - D16.F05	16.000 ...	50.000 mm

Pocket+Contour

Hole/Feature
- Automatic
- Manual Holes

X+(radial) ∨ 0.00 deg ↕ Axis C ∨

GO2cam
FRANCE

- Left click on

- Choose Drill, then select the tool **"Drill D6.75"** in **Standard**

- Left click on

- Choose **Drilling**, then select **Simple mode Auto**

- In the strategy, change the retract value to - 5 and Depth calculation to Diameter

- Left click on

c) Tapping operation of M8 :

- Left click on , then on the command **Wizard**

- In the machining tree, expand the operation Drilling, then Left click on Hole, to copy the selection

- Left click on

- Choose Tap, Download the standard library and select Tools **"TapM08x1.25"**

- Left click on

- Choose tapping, then select **Tapping mode Auto**

- Left click on

Drill

Tool name	Diameter	Point angle	Useful length
Drill - D06.75.F01	6.750 mm	120.00 deg	71.000 mm

Drilling

Techno. name	Depth	Type	Z Offset	Z Step (Ap)
Simple mode Auto	0.000 mm	Simple	0.000 mm	0.000 mm

Depth calculation ○ Tool end
● Diameter

Diameter centering 0.000 mm

Retract -5.000 mm

Tap

Tool name	Diameter	Pitch	Useful length
Tap - M08x1.25.F04	8.000 mm	1.250 mm	38.000 mm

Tapping

Techno. name	Depth	Z Offset
Tapping Mode auto	0.000 mm	0.000 mm

GO2cam
FRANCE
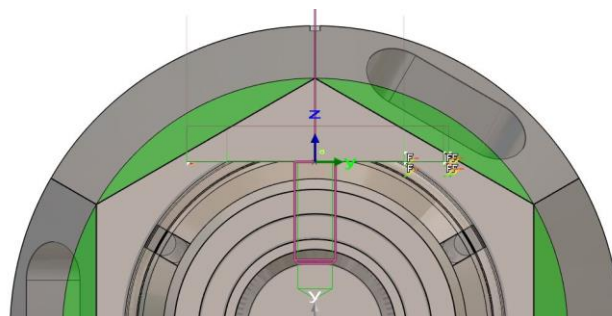
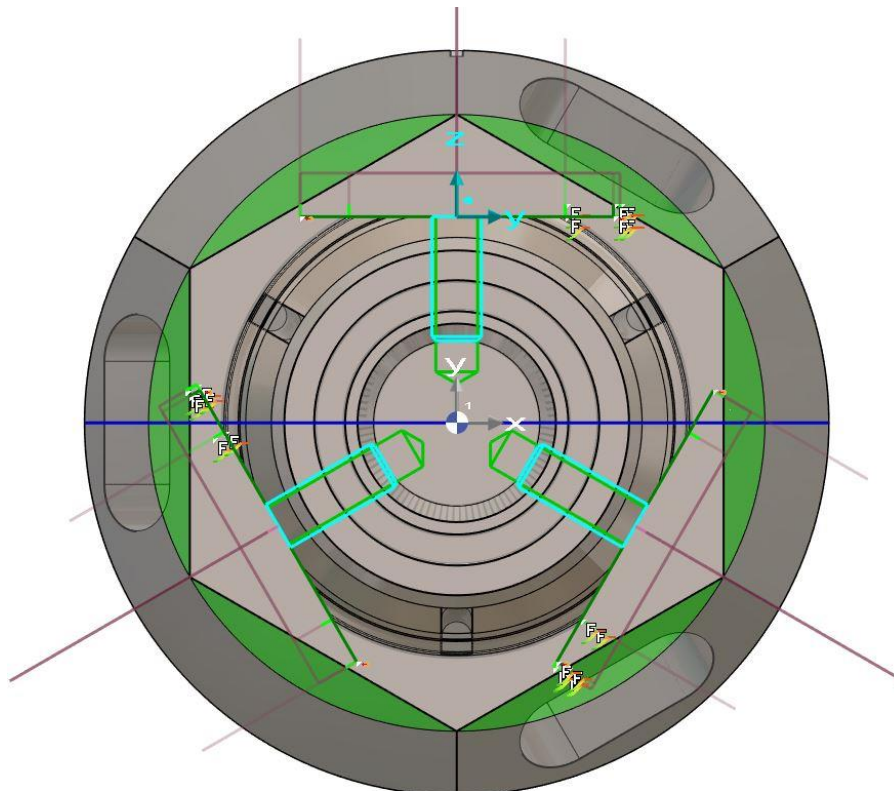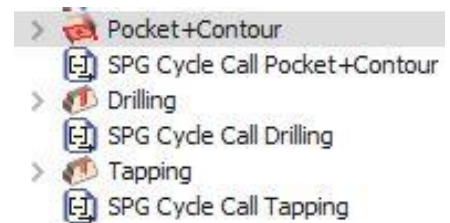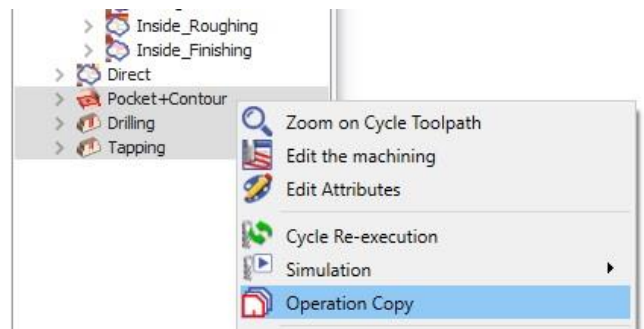## 5. Copy operation by C rotation:

- In the machining tree, use Ctrl to select three cycles applied on the face

- Right click in the machining tree, then Left click on Operation Copy

- In the drop-down menu select C Rot, type 2 in the number field and 120 in the angle field

No 1000	C Rot	Number 2	Angle 120.00 deg

Translation
Symmetry
Rotation
Rescale
A Rot
B Rot
C Rot
A Rot + Dec
B Rot + Dec
C Rot + Dec
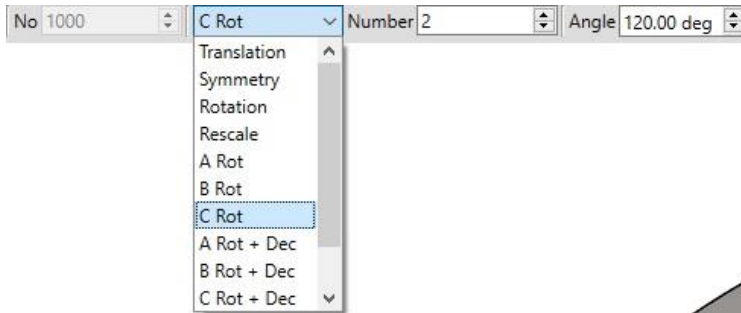
- To complete the operation, Left click on one of the three cycles in the machining axis

The operation copy is located below the original cycle.

You can start the simulation to check your toolpath.

> Inside_Roughing
> Inside_Finishing
> Direct
> Pocket+Contour
> Drilling
> Tapping

Zoom on Cycle Toolpath
Edit the machining
Edit Attributes
Cycle Re-execution
Simulation
Operation Copy

> Pocket+Contour
  SPG Cycle Call Pocket+Contour
> Drilling
  SPG Cycle Call Drilling
> Tapping
  SPG Cycle Call Tapping

GO2cam
FRANCE

6. Use the C-axis to machine on inclined planes :

We will use the C-axis to machine all inclined planes and chamfer them at the same time.

- Change to the reference plane

- Enter manual

- Left click on

- Select the edge of the hexagon, as shown in the figure below, and then select the edge path to be machined

- Check the Bottom Z box, then type - 92 in the field
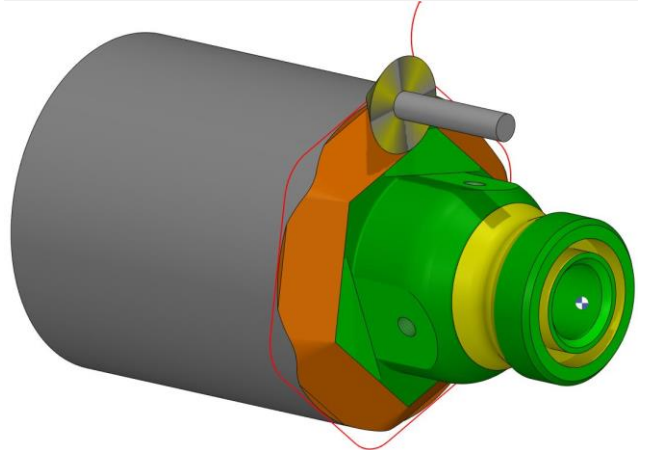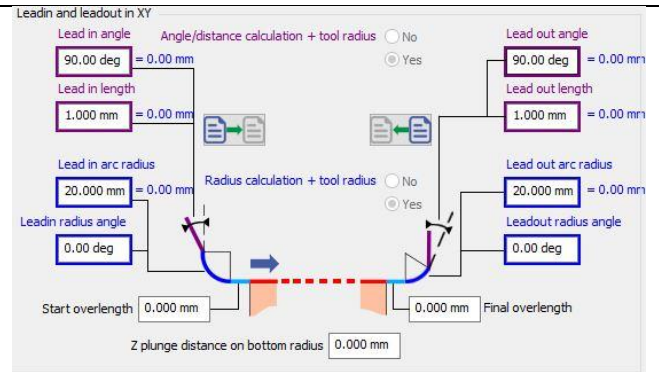
- Set Axis to **C**

- Left click on

- Select Chamfering Mill, download Otelo library. In the selection page, select tool **"16 120 943"**, then change the cut diameter to 35 and the mini diameter to 5.

- Left click on , choose Chamfering
- Left click twice on the Chamfering icon to enter the Strategy.

- In the strategy, the cycle must be on the left, the Z steps is 15, and the toolpath following slope must be "Yes"
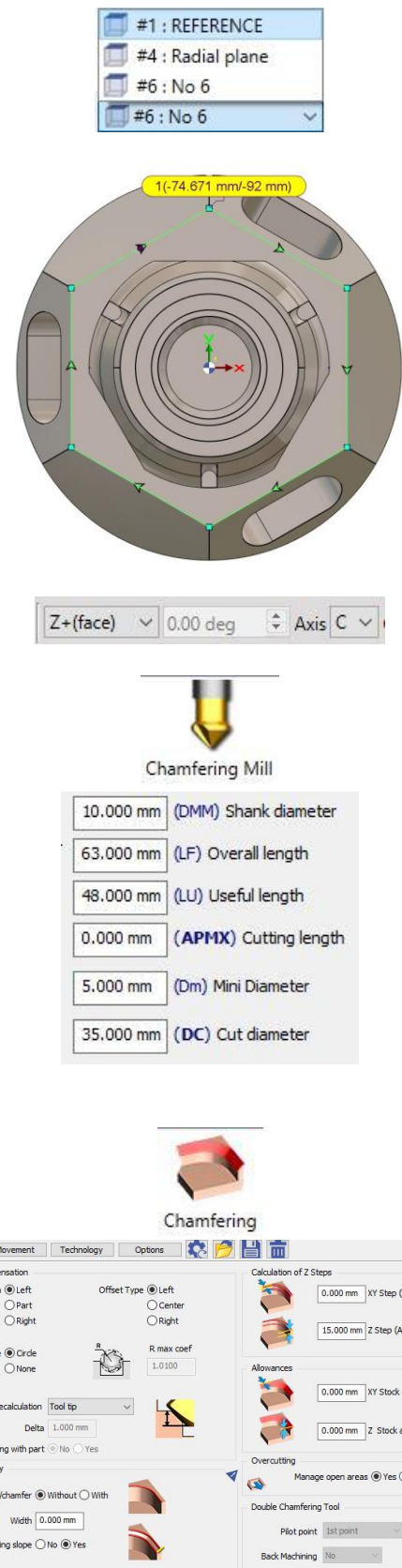
- In "Movement", type 20 in the lead in and lead on arc radius fields; And the angle of lead in/out must be 0

Leadin and leadout in XY

Lead in angle	Angle/distance calculation + tool radius	○ No	Lead out angle
90.00 deg = 0.00 mm		● Yes	90.00 deg = 0.00 mm
Lead in length			Lead out length
1.000 mm = 0.00 mm			1.000 mm = 0.00 mm
Lead in arc radius	Radius calculation + tool radius ○ No		Lead out arc radius
20.000 mm = 0.00 mm	● Yes		20.000 mm = 0.00 mm
Leadin radius angle			Leadout radius angle
0.00 deg			0.00 deg

Start overlength 0.000 mm   0.000 mm Final overlength

Z plunge distance on bottom radius 0.000 mm

- Left click on

7. Pocket on inclined plane :

We will create an inclined plane in the same way as the radial plane with the interactive system of axes.

- Left click on the origin of the system of axes. While holding the left click, move the mouse over to inclined face, then click to select the face.

The inclined plane is highlighted, the axis system is in the center of the face, and the plane creation bar is opened.
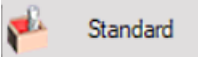
- In the plane name field, type "Inclined plane"

- In the orientation field, enter a value of 90

Plane Name Inclined plane   🔍 ⚙   ↳ Orientation 90.00 deg ⬍

- Left click on ✔ to validate

GO2cam
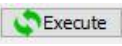FRANCE
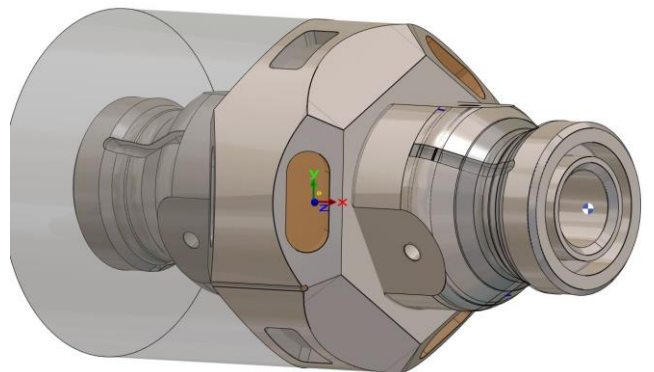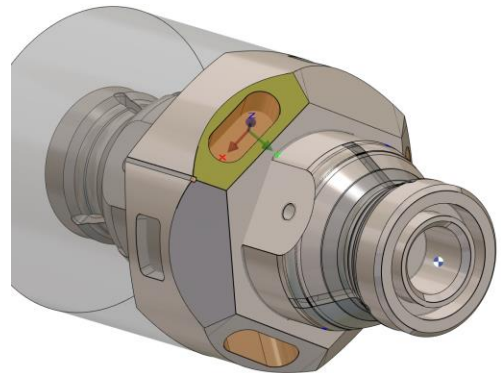
8. Pocket machining :

- Select **Standard**

- Left click on

- The selection mode is automatic by default.

- Left click on the **bottom face** of the pocket. The top and bottom height is acquired from the 3D part and stock.

- Left click on , the information page is displayed in the bottom right corner, which contains the value of the minimum radius to help you select the tool

- Left click on

- Choose **Flat End Mill**, then select tool **"Flat End Mill D10"** from **Standard**

- Left click on

- Choose pocket
- Set the value of Z step to 2.5 and the value of allowance to 0

- Left click on

The cycle is generated, if the Z plunge is not helix. You can change this parameter.

- Left click on ＞ to expand the pocket operation
- Right click on **Strategy**, then left mouse click on **app/ret edition**

- Change Z plunge to Helix if required.

- Left click on to recalculate the toolpath
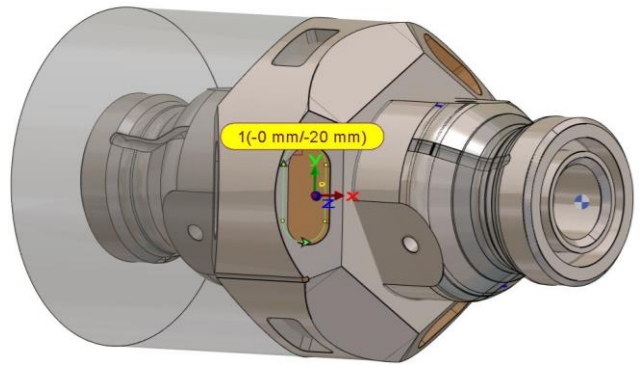
Information

```
<================================>
```

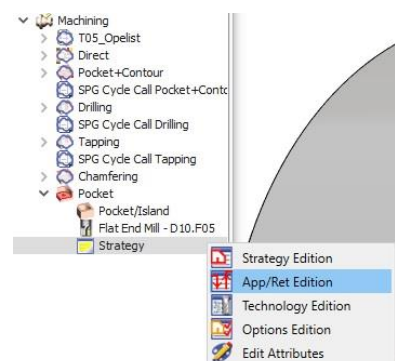Smallest fillet radius = 7.500000 for closed profiles (Diameter = 15.000000)

Flat End Mill

Tool name	Diameter	Useful length
Flat End Mill - D10.F05	10.000 ...	28.000 mm

Pocket

Techno. name	Z Step (Ap)	XY stock allowance	Z stock allowance
	2.500 mm	0.000 mm	0.000 mm

- Machining
  - T05_Opelist
  - Direct
  - Pocket+Contour
  - SPG Cycle Call Pocket+Contc
  - Drilling
  - SPG Cycle Call Drilling
  - Tapping
  - SPG Cycle Call Tapping
  - Chamfering
  - Pocket
    - Pocket/Island
    - Flat End Mill - D10.F05
    - Strategy

| Strategy Edition |
| App/Ret Edition |
| Technology Edition |
| Options Edition |
| Edit Attributes |

Z Plunge

Plunge

Helix

We will copy the operation in rotation C.

- Right click on the **pocket** cycle, then Left click on **operation copy**

- In the drop-down menu of C Rot, type 2 in the number field and 120 in the angle field

| C Rot | ∨ | Number | 2 | ⬍ | Angle | 120.00 deg | ⬍ | REFERENCE | ∨ |
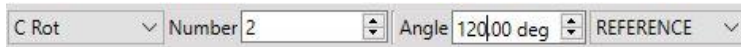
- Left click on the pocket cycle in the machining tree

The operation copy is applied below the original cycle.

You can start the simulation 🖥️▶ to check the toolpath.
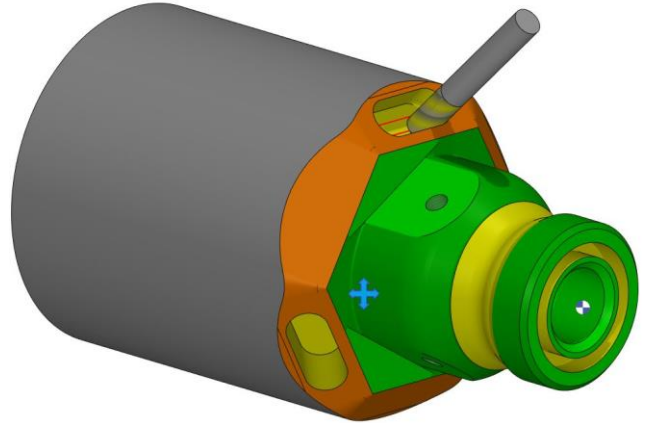
9. Pocket machining on developed plane:

We will machine the pockets located on the largest diameter. It can be observed that the bottoms of these pockets follow the cylinder. It will be necessary to create a Developed plane in order to be able to machine these shapes.
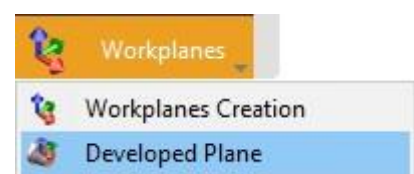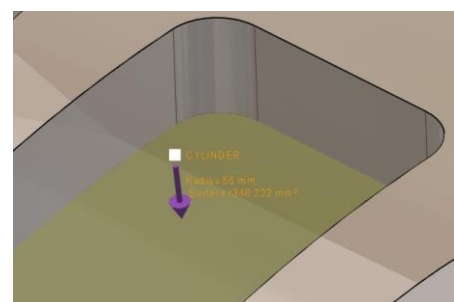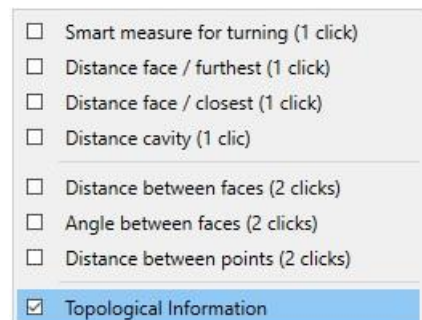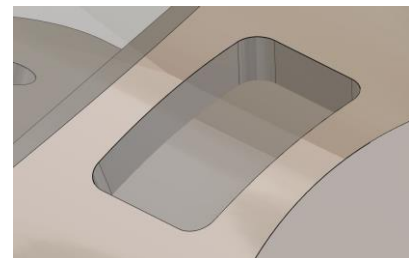
a) Creation of the developed plane :

- Left click on ▸⊢ , then in the drop-down menu of the command bar, select "Measure on Solid" to measure the radius of the bottom of the pocket

- Right click on the screen, select Topological Information

- Left click on bottom of pocket, value reads Radius 55mm

This value will help us to create a plane developed directly on the curved bottom surface.

- Enter Design menu, click on Workplanes

  - Left click on **Developed Plane**

  - Left click on **icon**

☐ Smart measure for turning (1 click)
☐ Distance face / furthest (1 click)
☐ Distance face / closest (1 click)
☐ Distance cavity (1 clic)

☐ Distance between faces (2 clicks)
☐ Angle between faces (2 clicks)
☐ Distance between points (2 clicks)

☑ Topological Information

Workplanes
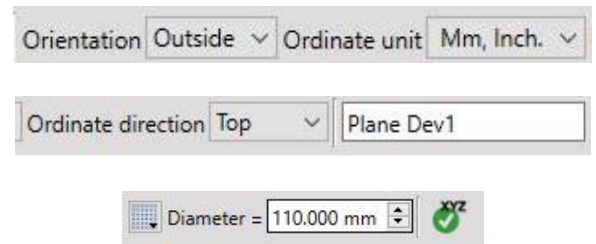🎚️ Workplanes Creation
🎨 Developed Plane

GO2cam
FRANCE

The command bar appears. Before entering the diameter value, you must enter the parameter first.

The parameters for the developed plane are:

- ➢ Orientation : **Outside**
- ➢ Ordinate Unit : **Mm,inch**
- ➢ Ordinate Direction : **Top**
- ➢ Name of plane : **Plane Dev1**

- Enter a value of 110 in the diameter field and click on the green tick to confirm.

The development plane has now been created, and GO2cam will automatically put you in it. We will be able to realize projection in this plane.

b) Projection on developed plane :

- Still in the Workplanes menu, go to the reference plane to view the part
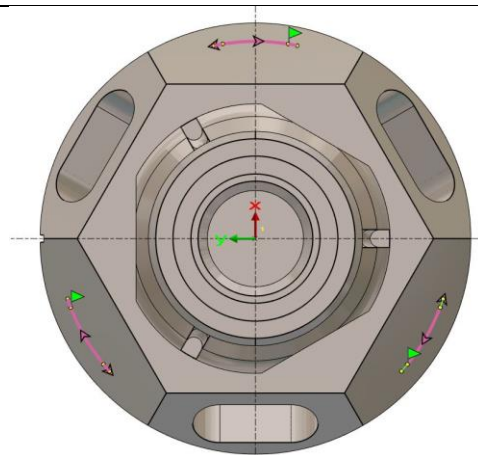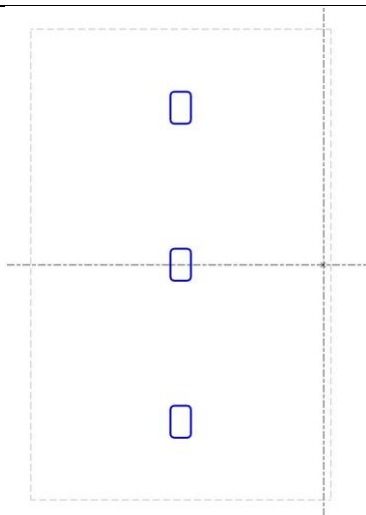
- Left click on

Note: When only one plane is available, the projected flat plane will be automatically selected

- Left click on **Creation of an edge path on the solid**

- Left click on an edge at the bottom of the pocket to get the continuity of the bottom edge, left click on Profile OK
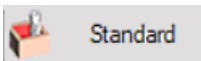
A profile covers the edge path to be projected onto your developed plane. Now, the same step needs to be taken for the other two pockets before validating the function.
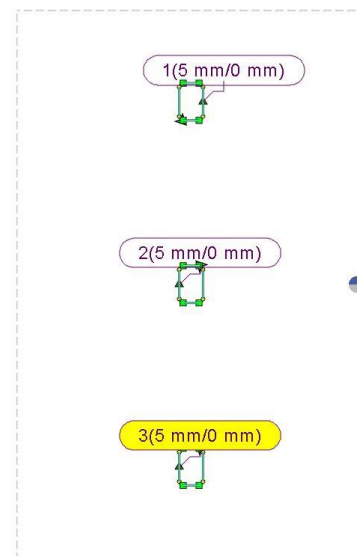
- After selecting three profiles, you can validate the command by clicking ✔

- Return in **Plane Dev1**, To check whether the geometry is correctly projected

GO2cam
FRANCE
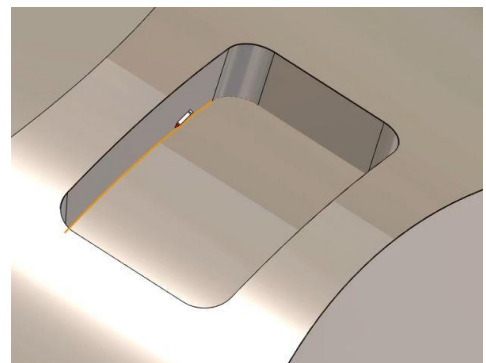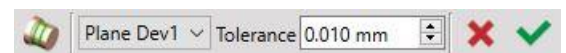
c) Application of pocket on developed plane :

- In **Milling tab**, check whether the dev1 plane is still active

- Select **Standard**

- Left click on

- Select the first profile on the developed plane.

- Enter a value of 5 in the Top Z box and 0 for Bottom Z, and then click the next two profiles. Ensure the values for Top and Bottom Z are similar.

- Left click on

- Choose Flat End Mill, then select tool **"Flat End Mill D04"**

- Left click on

- Choose Pocket, double click on **Pocket** to access the cycle parameters

- In strategy, modify the below parameters:

  ➤ **Z step** to **2,5**

  ➤ **Stepover value** to **0.50**

  ➤ **XY and Z Stock Allowances** to **0**

Flat End Mill

Tool name	Diameter	Useful length
Flat End Mill - D04.F05	4.000 mm	22.000 mm

Pocket

GO2cam
FRANCE

• In page **movement**, modify:
 	
  ➢ **Rapid Plane** to **20**
 	
  ➢ **Approach and Return in Z to** Rapid Plane
 	
• Left click on
 	
You can start simulation to check the toolpath.	
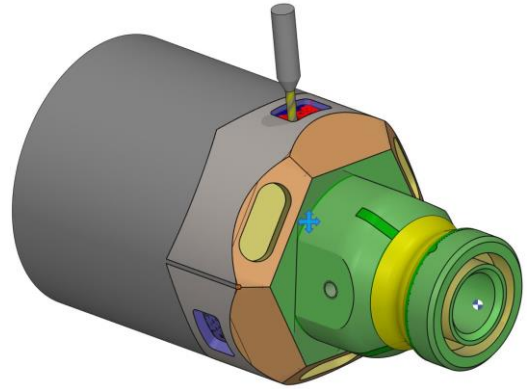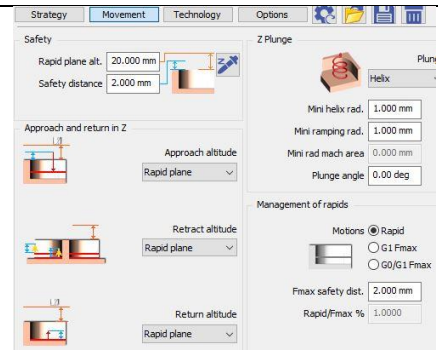
10. Part return : • Enter the **Turning** menu • Left click on the **Part** command and select part Manipulation • Left click on the function **Part return** **Note: In the command bar, the translation value will be automatically filled in the field.** • Left click on • At this point, if you have had a machine selected for the operation, you will be required to set the stock in the chuck and validate. [In this case you will not encounter this] • Now, you can repeat and define all the cycles on this side, similar to those defined from T05_Opelist to Pocket_1	

T05-17

GO2cam
FRANCE