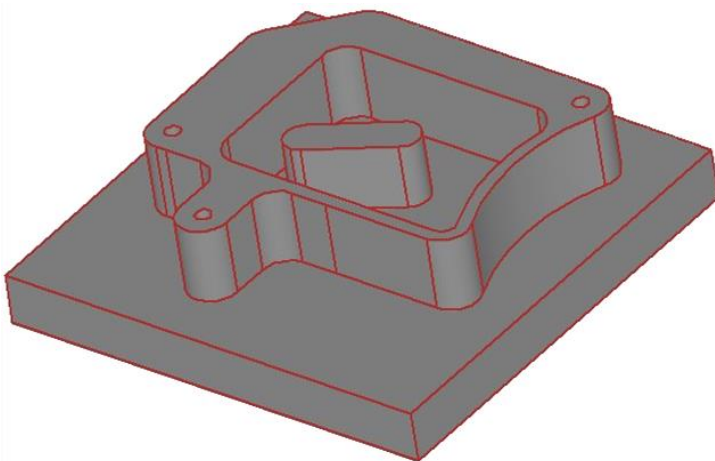
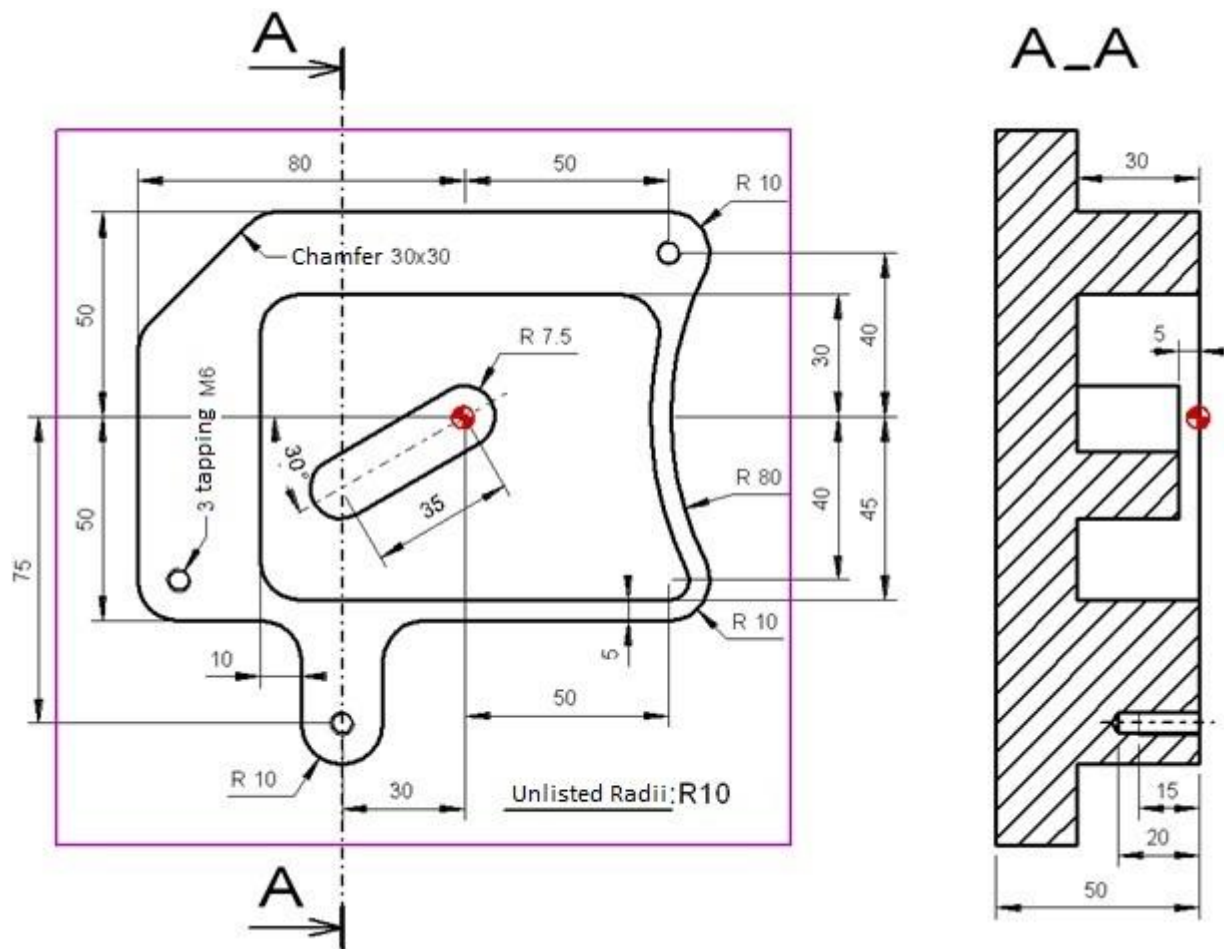


GO2cam V6.10 Tutorial M04 – Fixing plate

I. Part Layout



Quantity : 150
Material : Au4G

II. Process for Machining

Targets :

- Import 3D parts
- Application des cycles de Facing, Pointing, Drill Hole, Rework, Slotting and Contour
- Get the surface to be machined
- Basic parameters of operation
- Tool selection

1. Import 3D parts :

The import function consists of three steps:

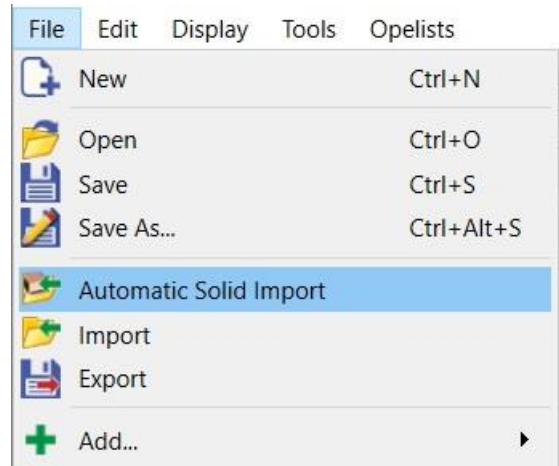
- Positioning, Creation of stock and Creation of origin.



Do not quit this function until all the 3 steps are completed. Otherwise you cannot go back to completed the missing step.

- Left click on File
- Left click on automatic solid import

Select the file 'M04_Plaque de fixation', if it is not visible check "File Type"



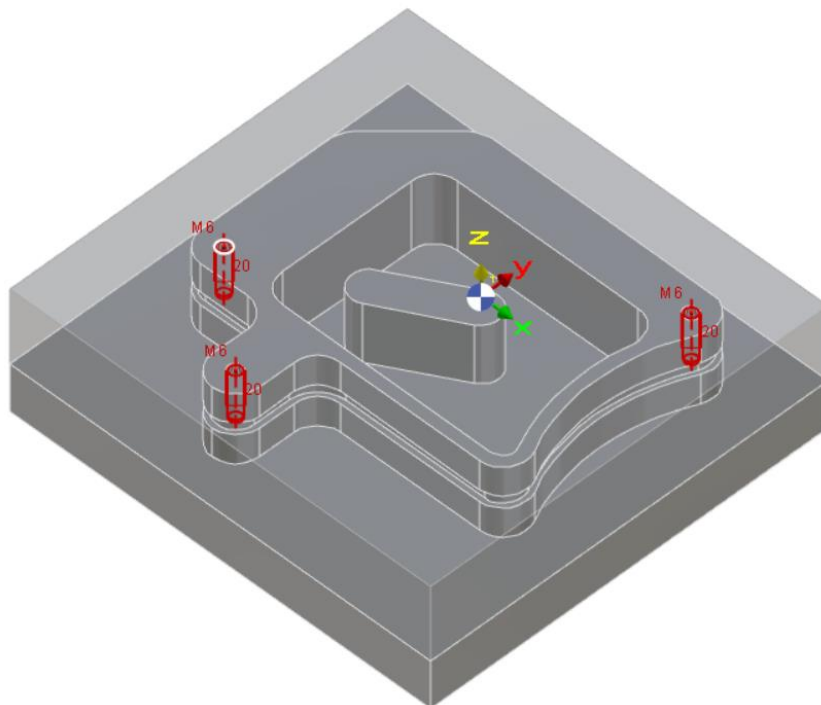
File name:

Import

Files of type:



Parasolid (*.X_T;*.X_B)

Cancel




a. Face support



This step allows the support surface to be positioned on the mounting system to orient the Z axis.

- By default, select one side of the stock. If this is not the desired side, you can click the other side of the stock.
- Icon  allows to get a face on the part
- In this tutorial, we keep the bottom surface as the support surface
- Left click on  to validate the first step

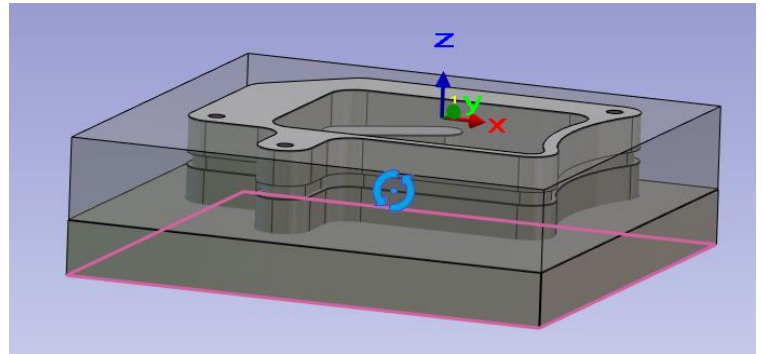
b. Stock size and direction

- Enter the value 57 in the height box
- Switch the position to the top offset and type a value of 2
- The direction allows the workpiece to rotate around the Z axis. In this example, we keep the value 0.
- Also set the overflow to 0.
- Left click on  to validate this second step


c. Creation of origin

- Left click on  to enter manual origin
- Press Ctrl+Shift and select the central island arc
- Enter a value of 0 in the Z value box
- Left click on  to validate this third step

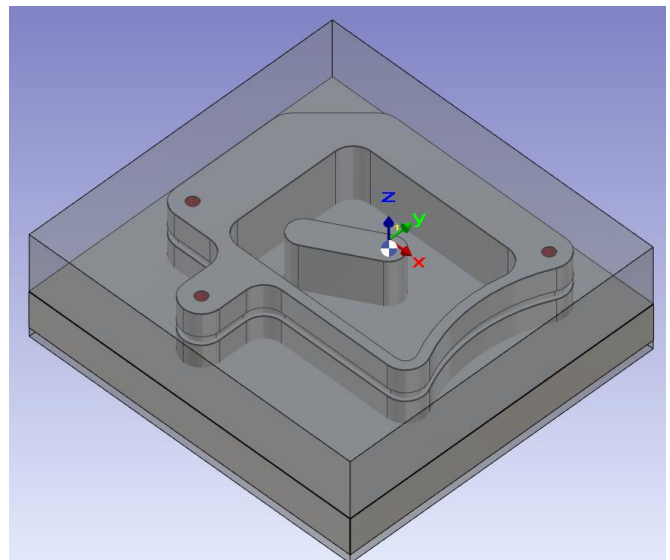
Select Support Face



Height 57.000 mm 

Positioning:  Top offset 2.000 mm

Orientation 0.00 deg 




Ope 10 Facing

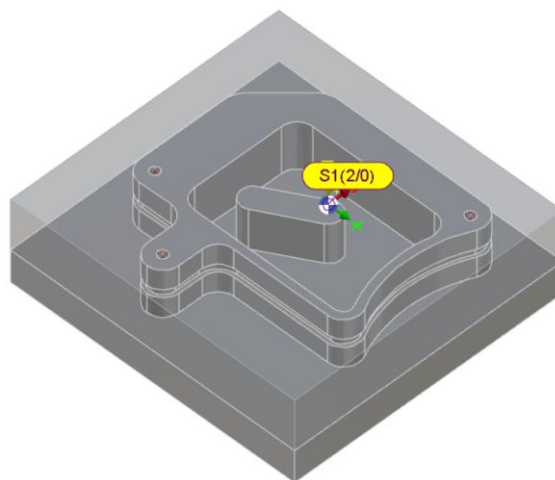
1. Start :

- Left click on the sub-menu Standard




2. Geometry selection :

- Left click on 
- Select the upper surface of the part



3. Tool selection:


- Left click on 
- Left click on Face Mill Cutter
- Automatic creation of Ø 63 tool



Face Mill Cutter

Tool name	Diameter	Useful length
	63.000 mm	30.000 mm

4. Machining cycle selection:

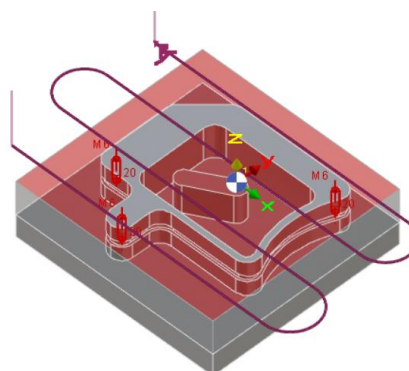
- Left click on 
- Left click on Facing Pocket



Facing Pocket


5. Toolpath calculation:


- Left click on Cycle Calculation



Ope 20 External profile

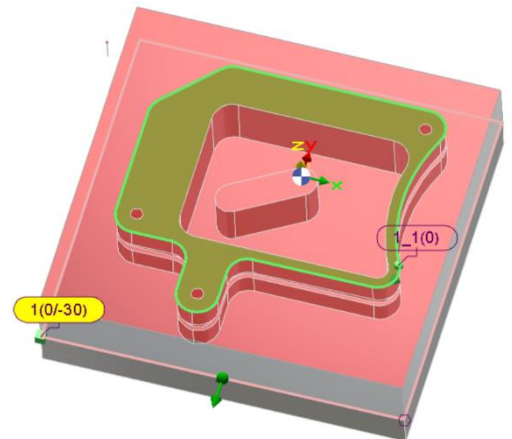
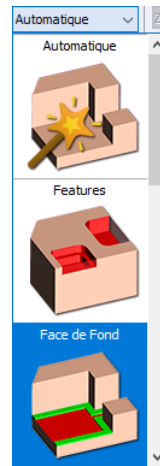
1. Geometry selection :

The mode  Standard is still active.


- Left click on 
- In the drop-down menu of profile creation mode, select Bottom Face
- Left click on the plane outside

It displays the outer profile of the island and create an overflow at all edges of the part

Height is automatically acquired




2. Tool selection :

- Left click on 
- Left click on Flat End Mill
- Double-Click on Diameter and type in 20



Tool name	Diameter	Useful length
	20.00 mm	70.00 mm

3. Machining cycle selection :

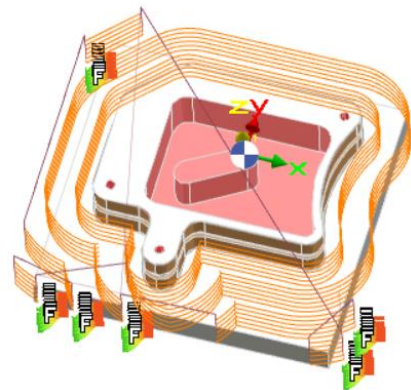
- Left click on 
- Left click on Pocket.
- In the strategy setting, change the allowance value in Z to 0



Techno. name	Z Step (Ap)	Stepover (Ae)
	5.00 mm	15.00 mm


4. Toolpath calculation :


- Left click on Cycle Calculation 



Ope 30 Inside Pocket

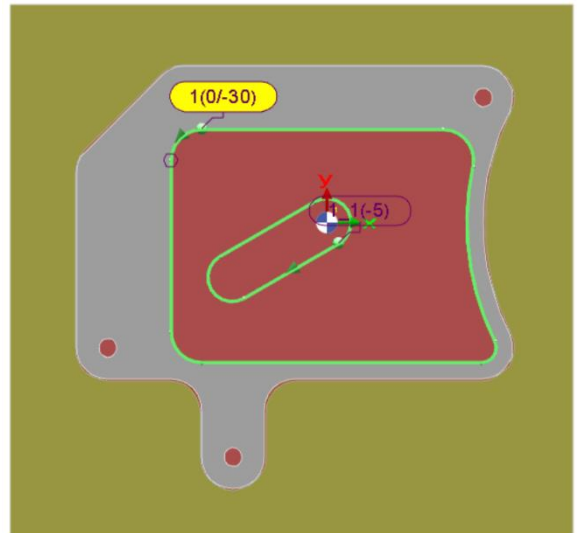
1. Geometry selection :

The mode  Standard is still active.


- Left click on 
- In the drop-down menu of profile creation mode, select bottom face
- Left click on the flat inside pocket

It displays internal profile of pocket and external profile of island

Height is automatically acquired



2. Tool selection :

- Left click on 
-
- Left click on Flat End Mill
- Left click twice in Diameter then type 20



Tool name	Diameter	Useful length
	20.00 mm	70.00 mm

3. Machining cycle selection :

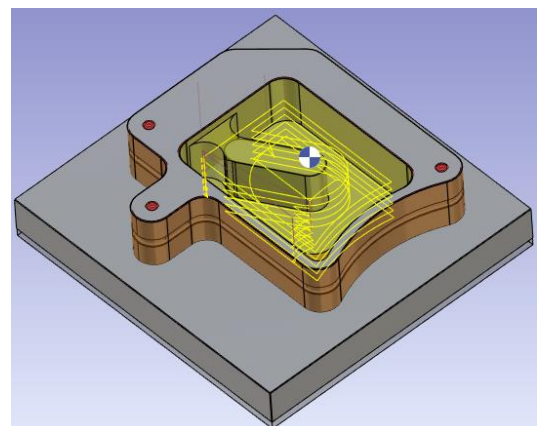
- Left click on 



Techno. name	Z Step (Ap)	Stepover (Ae)
	5.00 mm	15.00 mm


4. Toolpath calculation :




- Left click on Cycle Calculation 



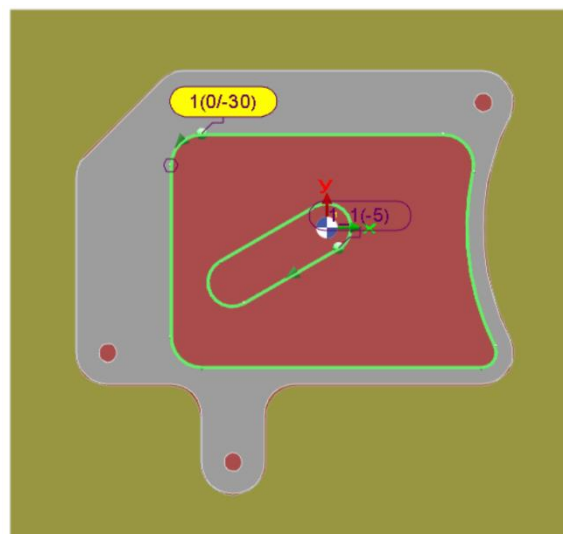
Ope 40 Inside Pocket rework

1. Geometry selection :


The mode  Standard is still active.

- Left click on 
- In the machining tree, left click on the  Pocket_1 arrow before the cycle name
-  Left click on and drag the pocket/island icon of the last pocket operation to the window on right side.

The profile and height of pocket operation will be copied into this new operation.




2. Tool selection :

- Left click on 
- Left click on Flat End Mill



Tool name	Diameter	Useful length
	10.00 mm	70.00 mm

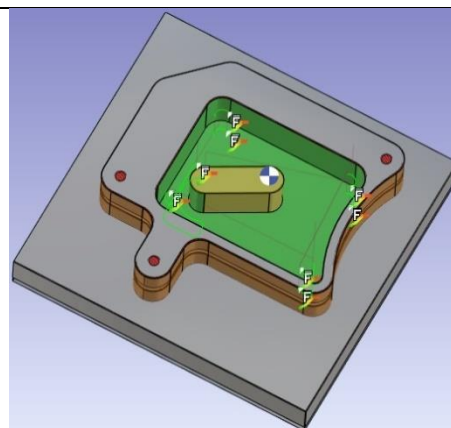
3. Machining cycle selection :

- Left click on 
- Set the Z Step (Ap) to 0.
- In the strategy setting, change the reference diameter to 20.




4. Toolpath calculation :




- Left click on Cycle Calculation 



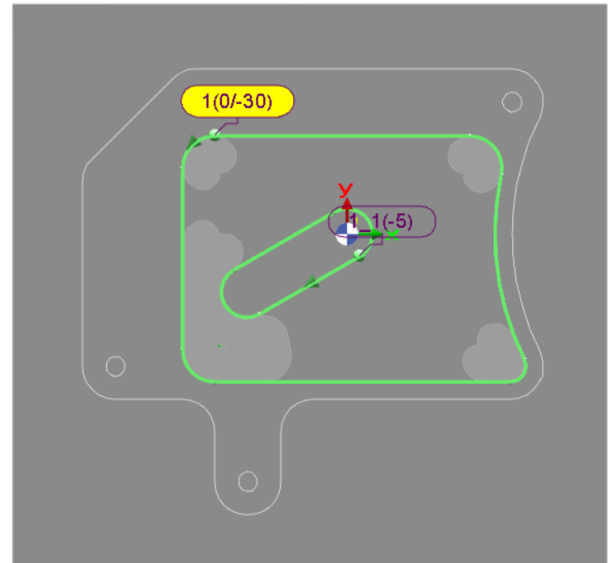
Ope 50 Bottom Finishing of inside pocket

1. Geometry selection :


The mode  Standard is still active.

- Left click on 
- In the machining tree, left click on the  Pocket Rework arrow before the cycle name.
-  Left click on and drag the pocket/island icon of the last pocket rework operation to the window on right side.

The profile and height of the pocket rework operation will be copied into the new operation.



2. Tool selection :

-  Click and drag the Pocket Rework tool to enter the big window on right side.


The tool from the last Pocket Rework operation is now copied in this new operation



Flat End Mill

Tool name	Diameter	Useful length
	10.00 mm	70.00 mm

3. Machining cycle selection :

- Left click on 
- Left click on Pocket.
- Set the Z stock allowance to 0.
- In the strategy setting, change the Stepover (Tool ratio) to 0.6.
- Set the Calculation of Z Steps to «Residual material » and thickness to 0.5 mm.
- Set the Island reworking and Bottom reworking to « Yes ».



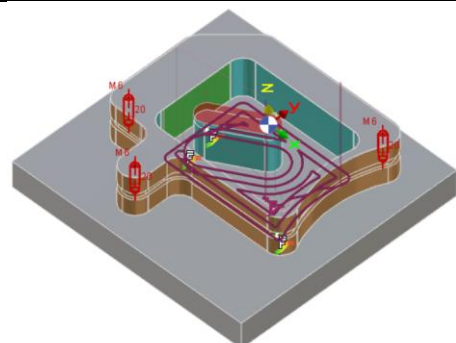
Pocket

Techno. name	Z Step (Ap)	Stepover (Ae)
	5.00 mm	15.00 mm







4. Toolpath calculation :

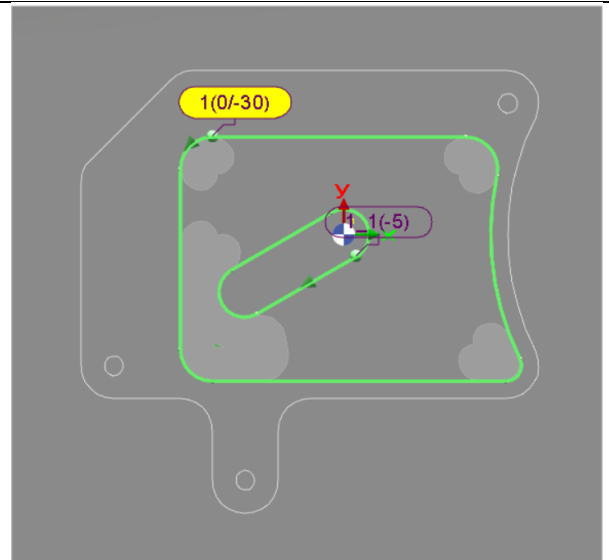
- Left click on Cycle Calculation 




Ope 60 Inside Contouring

1. Geometry selection :

- Left click on sub menu  Manual
- Left click on 
- Left click on the arrow before the cycle
 Pocket Rework name
-  Left click and drag the Pocket/Island icon from the Pocket Rework operation to the big window on the right side.



2. Tool selection :


-  Click and drag the Rework Pocket tool to enter the big window on right side.

The tool from the last Pocket Rework operation is now copied in this new operation.



Tool name	Diameter	Useful length
	10.00 mm	70.00 mm

3. Machining cycle selection :

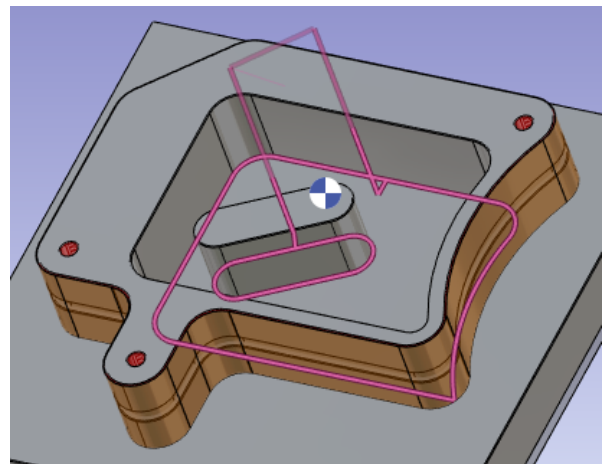
- Left click on 
- Left click on **Contouring**.
- In the strategy, set the Z Step to 0.



Techno. name	Z Step (Ap)
	0.000 mm

4. Toolpath calculation :


- Left click on Cycle Calculation 




Ope 70 Outside Contouring

1. Geometry selection :


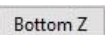
The  Manual is still active.

- Left click on 
- In the drop-down menu of profile creation mode, select Lateral Face
- Left click on an outside vertical surface, if necessary, click the green arrow to reverse the direction
- Repeat this step on the arrival surface (red arrow on the image)

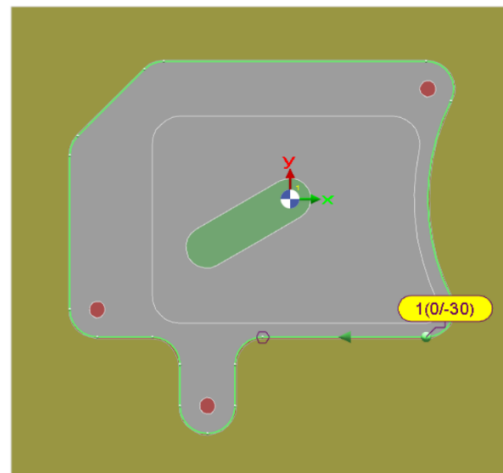
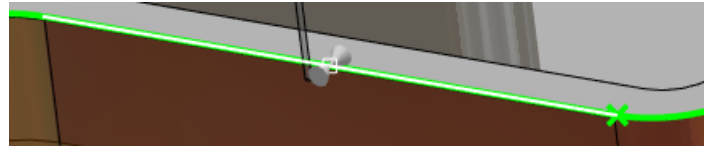


- Click  Start/End Point to change the entry/exit point of the tool on the profile to same as on the image.




- Left click on  Profile OK
- Click  Bottom Z and select a face or point with a height of - 30

Profile is created as the external profile of the island and create an overflow at all edges of the room



2. Tool selection :


- Left mouse click on 
- Left mouse click on Flat End Mill
Select the Standrd tool 6.5 mm.



Flat End Mill

Tool name	Diameter	Useful length
Flat End Mill - D06.5.F05	6.500 mm	18.000 mm

3. Machining cycle selection:

- Left click on 
- Left click on **Contouring**.

Inside the strategy, in the movement tab set the parameters as shown on the image.



Contouring

Leadin and leadout in XY

Lead in angle = 0.00 mm

Lead in length = 4.25 mm

Lead in arc radius = 6.25 mm

Lead in radius angle

Angle/distance calculation + tool radius ☒ No ☐ Yes

Radius calculation + tool radius ☒ No ☐ Yes

Lead out angle = 0.00 mm

Lead out length = 4.25 mm

Lead out arc radius = 6.25 mm

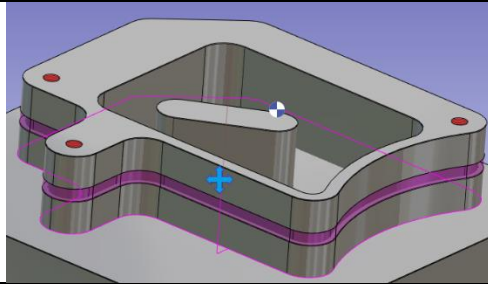
Leadout radius angle

Start overlength Final overlength

Z plunge distance on bottom radius




4. Toolpath calculation :

- Left click on Cycle Calculation

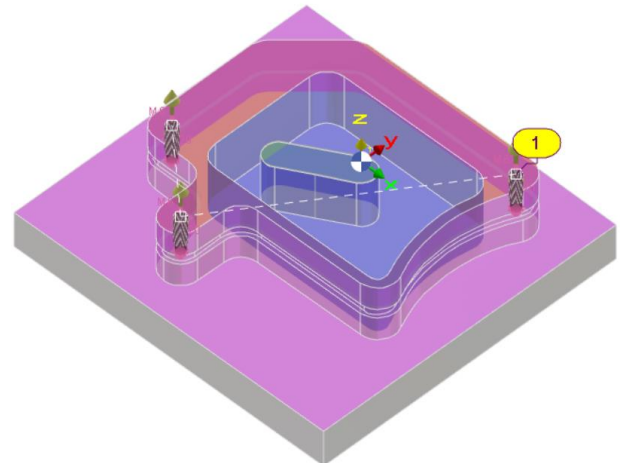


Ope 80 Drilling 2 holes



1. Geometry selection :

- Left click on the mode  and select **Manual Holes**
- Left click on 
- Left click on  to activate **multiple selection**

Left click on any of the 3 holes, select all holes of the same diameter



2. Tool selection :


- Left click on 
- Left click on **Drill**
- Left click on  to cancel the filter and select a drill tool with diameter of 5



Drill

Tool name	Diameter	Point angle	Useful length
	5.000 mm	120.00 deg	55.000 mm

3. Machining cycle selection :

- Left click on 
- Left click on **Drilling**
- On the strategy page, select "Diameter" in depth calculation, and then type - 3 mm in Retract



Drilling

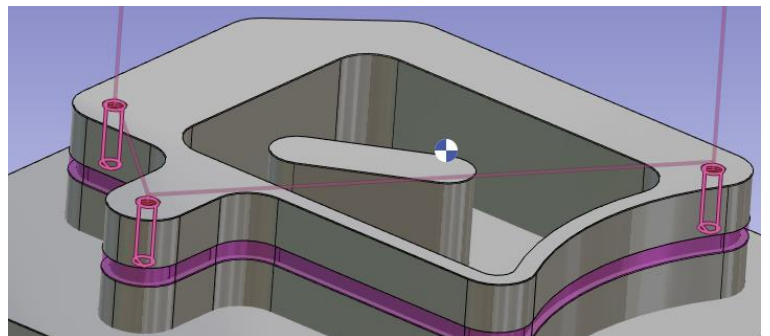
Depth calculation ☐ Tool end ☒ Diameter

Diameter centering

Retract



4. Toolpath calculation :

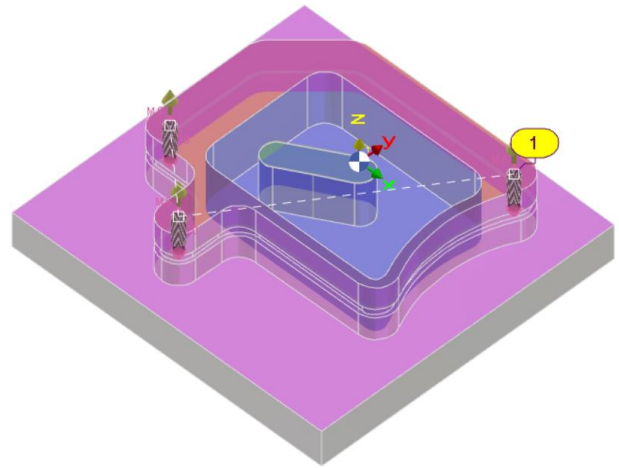
- Left click on Cycle Start 




Ope 90 Tapping

1. Geometry selection :

- **Manual Holes** is still activated.
- In the machining tree, left click the  **Drilling** arrow in front of the cycle name.
-  Left click and drag the Hole selection of the drilling operation until you enter the window on right side.
- The 3 holes are selected.



2. Tool selection :


- Left click on 
- Left click on Tap



Tap

Tool name	Diameter	Pitch	Useful length
	6.000 mm	1.250 mm	70.000 mm

3. Machining cycle selection :

- Left click on 
- Left click on **Tapping**
- Set the depth to 0
- On the strategy page, select "Diameter" in depth calculation, and then type - 3 in retract.

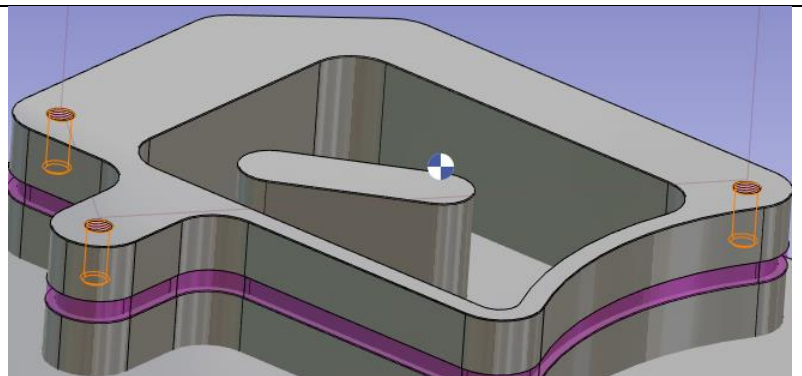


Tapping

Techno. name	Depth	Z Offset	Final dwell	Z Step (Ap)
	0.000 mm	0.000 mm	0.0000	0.000 mm

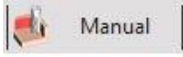



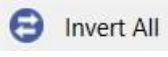
4. Toolpath calculation :

- Left click on Cycle Calculation 

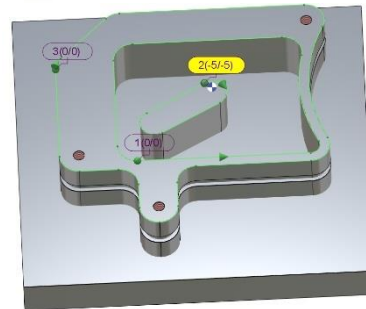
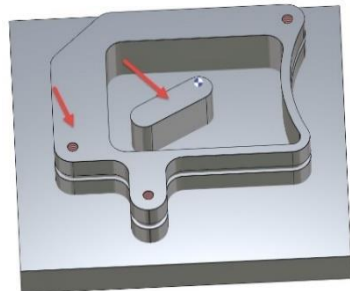


Ope 100 Chamfering


1. Geometry selection :

- Left click on sub menu  Manual
- Left click on 
- In the drop-down menu of profile creation mode, select Sharp edges
- Left click on Select by click 
- Left click on both sides as shown in the figure.
- If the direction of the profile is opposite Left click on  Invert All
- Left click in background and Left click on  Invert All

Mode de sélection de...



2. Tool selection :

- Left click on 
 - Left click on Chamfering Mill
- In the tool settings, switch the Piloted Diameter on Tip to "Yes"




Chamfering Mill

Tool name	Diameter	Point Angle	Useful length
	20.000 mm	90.00 deg	70.000 mm

Piloted Diameter on Tip ☐ No ☒ Yes

3. Machining cycle selection :

- Left click on 
- Left click on **Chamfering**
- Edit the strategy and change the Depth Recalculation to the Z Delta, the delta value is 1,



Chamfering

Depth Recalculation Z Delta

Delta 1.000 mm

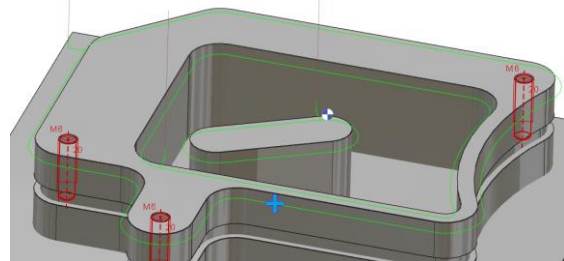
Gouge checking with part ☒ No ☐ Yes



Enter a value of 0 in the width of the chamfer

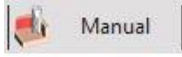


4. Toolpath calculation :

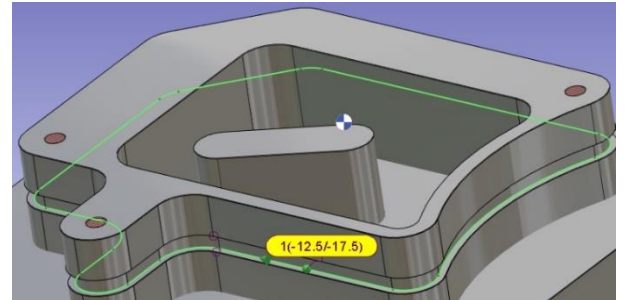
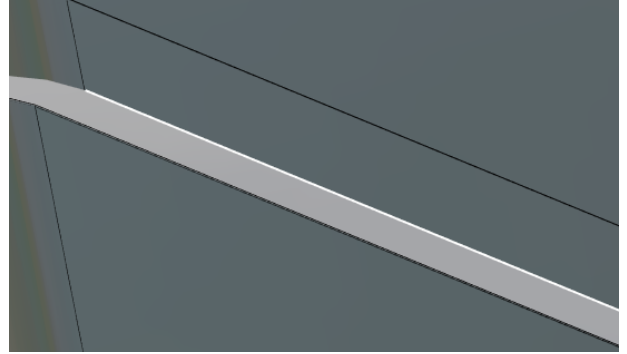
- Left click on Cycle Calculation 




Ope 110 Slotting

1. Geometry selection :

- Left click on the sub menu  Manual
- Left click on 
- In the drop-down menu of profile creation mode, select the edges path
- Left click on the bottom edge of the internal groove.
- Left click on  Profile OK
- Left click on top Z then click the edge at the top of the groove to obtain - 12.5
- Left click on bottom Z then click the edge at the bottom of the groove to obtain - 17.5




2. Tool selection :

- Left click on 
- Left click on **Side and Face Mill Cutter**


Side and Face
Mill Cutter

Tool name	Diameter	Cutting height	Useful width
	100.000 mm	15.000 mm	15.000 mm

3. Modification of tool:

- In Tool selection, left click twice on the tool icon of the **Side and Face Mill Cutter**
- Modify the different parameters of the tool, as shown on the image
- Click on the  icon to insert a toolholder.
- Modify the different characteristics of the tool holder, as shown on the image

10.000 mm (DAH) Hole Diameter

2.000 mm (LU) Useful width



2.000 mm (APMX) Cutting Height



20.000 mm (DC) Cut diameter

0.000 mm (RE1) Tool radius

0.000 mm (RE2) Top radius

12 (Z) Nb of flutes

 Toolholder chuck W/ hole 


 Smooth hole 

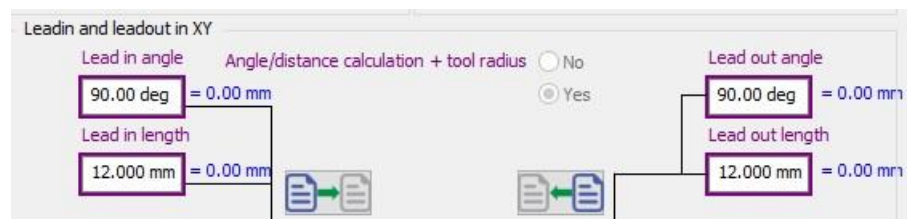
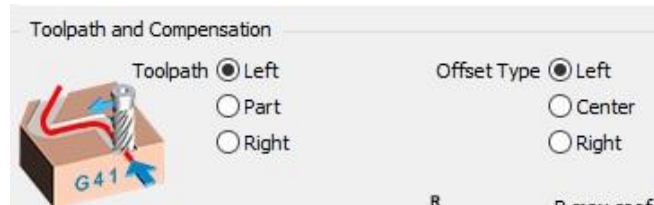
60.000 mm (L) Programming length

10.000 mm (D) Diameter

10.000 mm Tool hole diameter

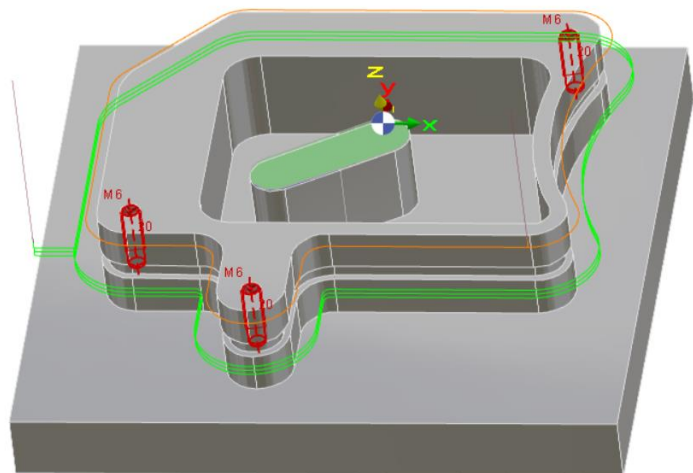
4. Machining cycle selection :

- Left click on 
- Left click on **Slotting**.
- In the strategy, change offset type to the left
- Change the Slot width to 2 and the profile selected to "side"
- Change Z machining to Middle+Depth+Alt
- Change Z Steps to 2
- Change the lead in and lead out length to 12 in movement page



5. Toolpath calculation :

- Left click on Cycle Calculation




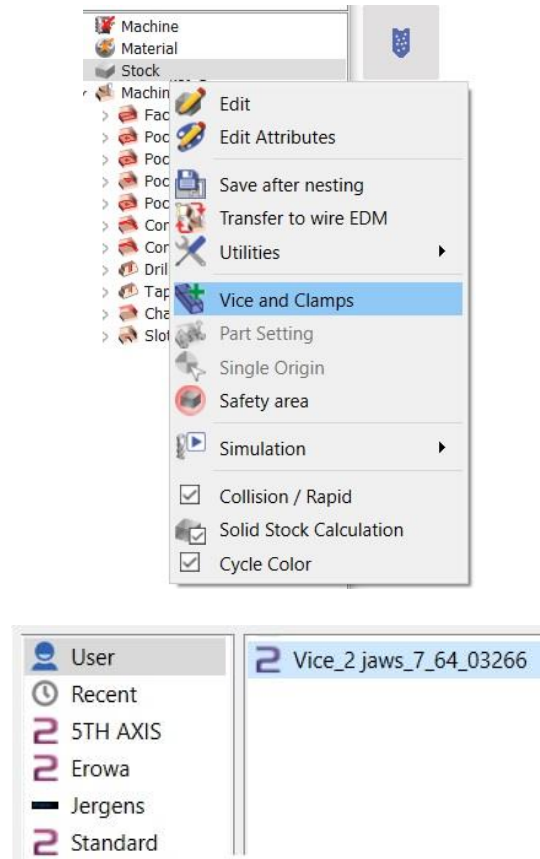
III. Import of fixture

Objective:

- Guide in fixture
- Positioning of fixture
- Collision management with fixture


1.Import of fixture :

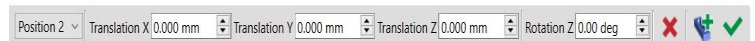
- Right click on stock
- Left click on vice and clamp
- On the user, select
«Vice_2_jaws_7_64_03266 »
- Left click on  Valider





2.Positioning of fixture:

When importing the fixture, a window will appear

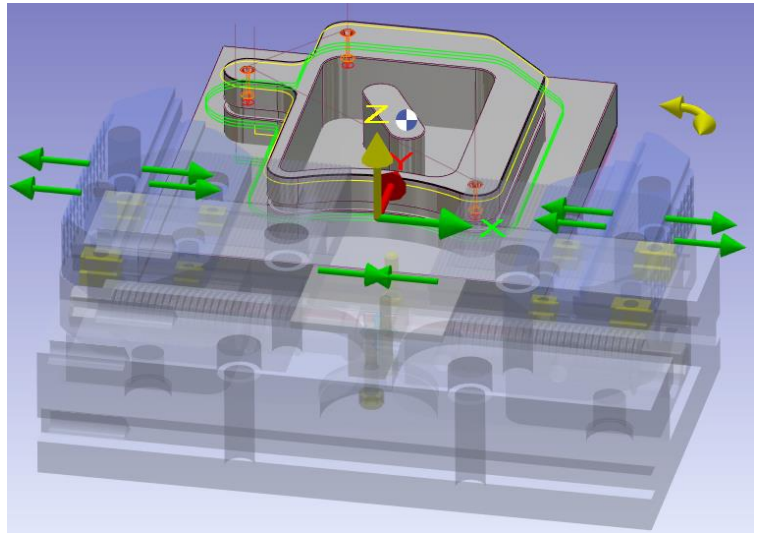
- Left click on position 1 and switch to position 2
- Click on  to validate



3. Define the fixture movement :

- Left click on  to rotate the fixture along Z, or enter a value in the Rotate Z box (if necessary)
- Use the axis system of the vice to perform translation on each axis (x, y, or z), or enter values in the window
- Left click on each green arrow to position each jaw element relative to the workpiece or stock
- Click on  to validate

Note : Your fixture is imported and positioned, and collision of fixture is managed



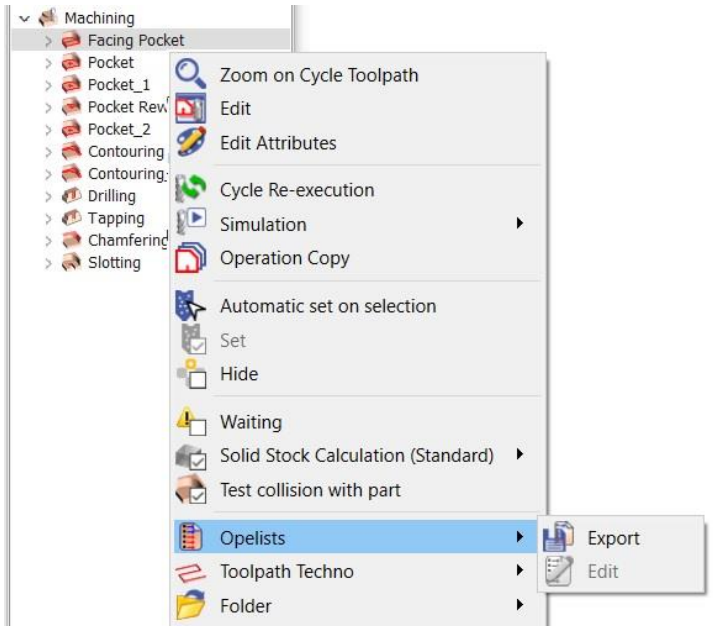
IV. Creation of Opelist

Objective:

- Automate machining cycles

1. Selection d'un cycle :

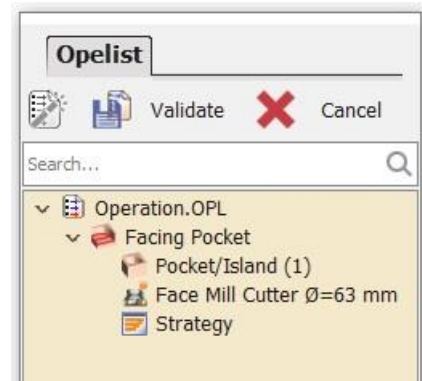
- Right click on cycle Facing Pocket
- Left click on Opelists
- Left click on Export



2. Edit Opelist :

You are now in the Opelist editor. The background is orange, different from the standard machining tree background.

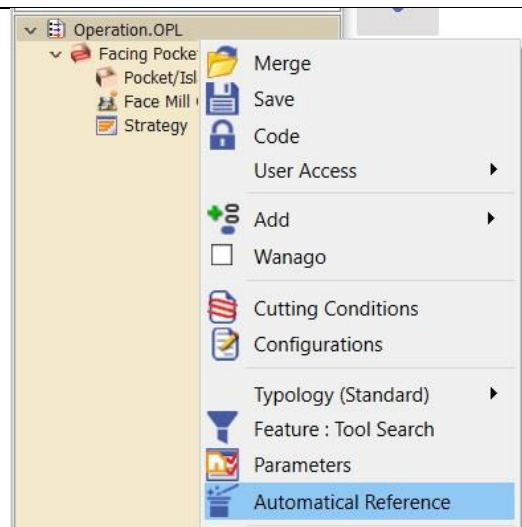
- If you click Validate, the operator will be created
- Don't confirm now, but please read the following steps.



3. Include automatic reference :

We will make some changes and customizations to opelist.

- Right click on Operation.OPL
- Left click on Automatical Reference



- Change the Pocket/Island parameter to Auto Facing

- Then click on



Note : This parameter allows automatic search of solid surfaces for Face Milling.

Label	Action	Value	Function Label
Pocket/Island	Auto Facing		
Open	None		
Plane number	Entities		
Altitude	Automatic	0.000 mm	
Bottom altitude	Automatic	0.000 mm	
Approach point	None		
Return point	None		
Predrilled hole	None		
Opelists			
Facing Pocket	Facing Pocket		

4. Save Opelists:

- Click on



- Create « Aluminum » folder in Opelists folder using icon

- Save your opelist in the name of Facing in « Aluminum » folder

Aluminum

Name	Date modified	Type	Size
Facing.OPL	14/03/2023 14:41	GO2cam model file	5 KB

File name: Facing.OPL

Save as type: Opelists (*.OPL)

Save Cancel





5. Opelists of pocket cycle

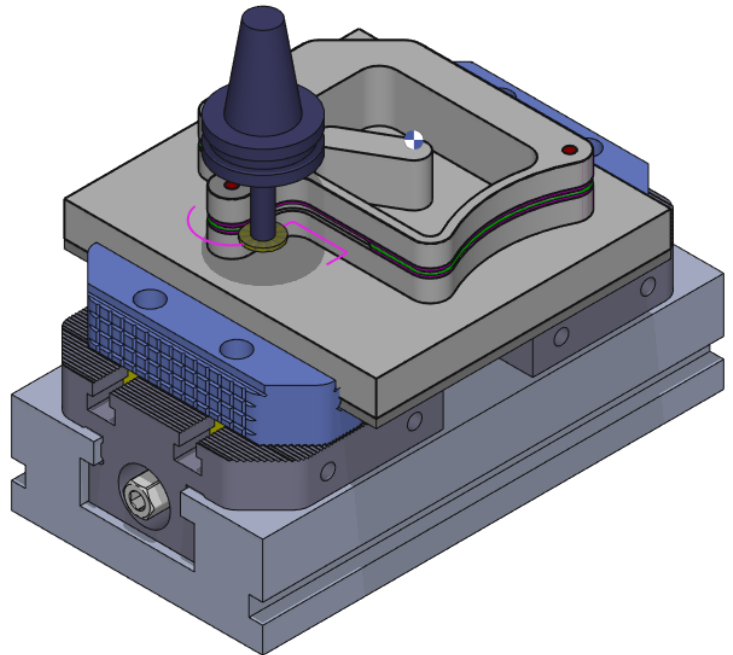
- According to the same principle as the Facing Pocket, create an Opelists for the second **Pocket cycle** (**Pocket_1**) and a second Opelists for the outside **Contouring cycle** (**Contouring_1**).
- For automatic reference, select all the features (available only for Pocket Cycle).
- Name the first Opelists 'Pocket' and the second Opelists 'Contouring G41'.

Label	Action	Value	Function Label
Pocket/Island	All the Features		
Open	None		
Plane number	Entities		
Altitude	Automatic	0.000 mm	
Bottom altitude	Automatic	0.000 mm	
Approach point	None		
Return point	None		
Predrilled hole	None		
Opelists			
Pocket_1	Pocket		

Simulation and NC output

1. Simulation :

- Enter the menu  NC File
- Left click on 
- Left click or  to start simulation on all machining operations
- If you want to switch to step-by-step mode, use the space bar
- Left click on  Or press the Escape key to stop the simulation



2. Generation of NC Code :

- Left click on 
- Select the post processor M67-FANUC from the list
- Left click on open
- Left click on Confirm
- The ISO program is generated

