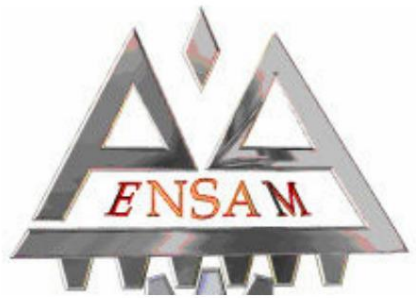
CAM lab on CATIA V5

ENSAM Meknes

2006 2007



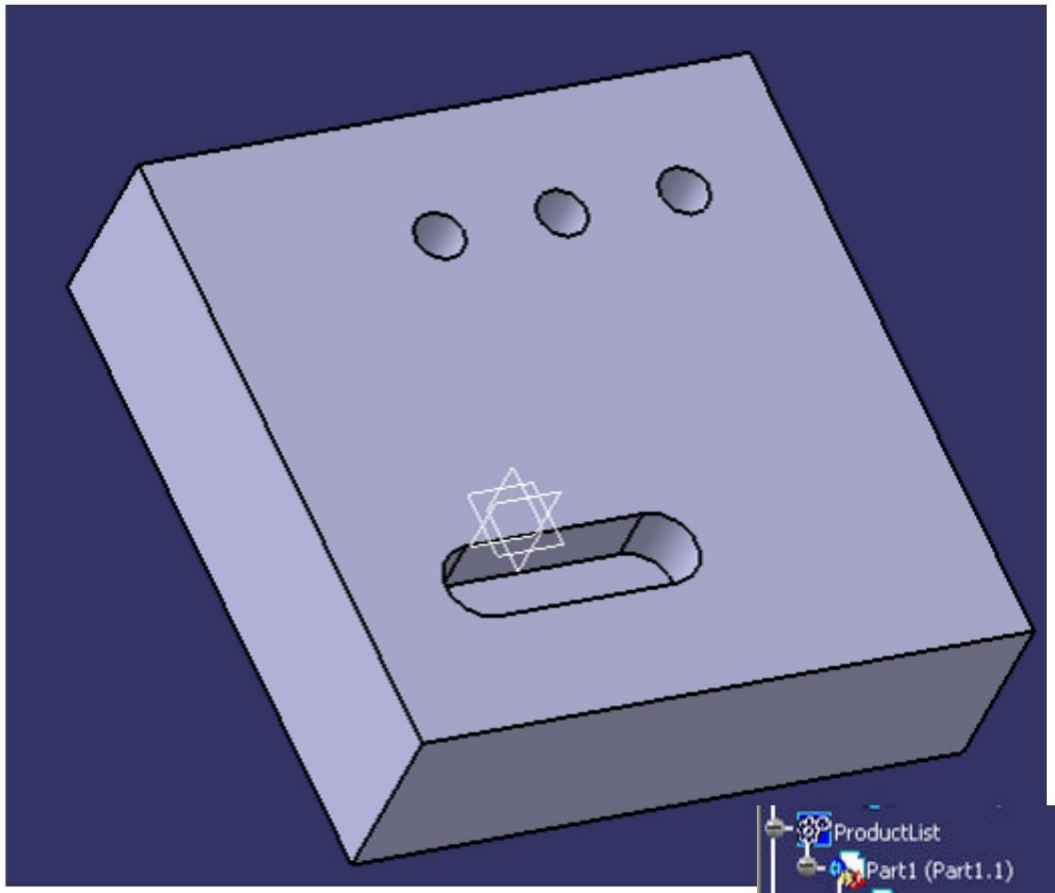
TP 1 2D MILLING 1/2

The objectives of this first TP are:

- **A first approach to CAM on CATIA**
- **Understand the main stages of creating a program on CATIA Immerse yourself in the construction philosophy of each machining operation It is a question of defining what is being machined (entity), how (strategy, approach/retreat), with which tool (and which cutting conditions)**
- **Discover the simulation of material removal (tool paths, interferences, etc.)**

Save your work regularly

Part drawing



Parallelepiped 100 x 100 x 30 (Extrusion of a 100 x 100 square)

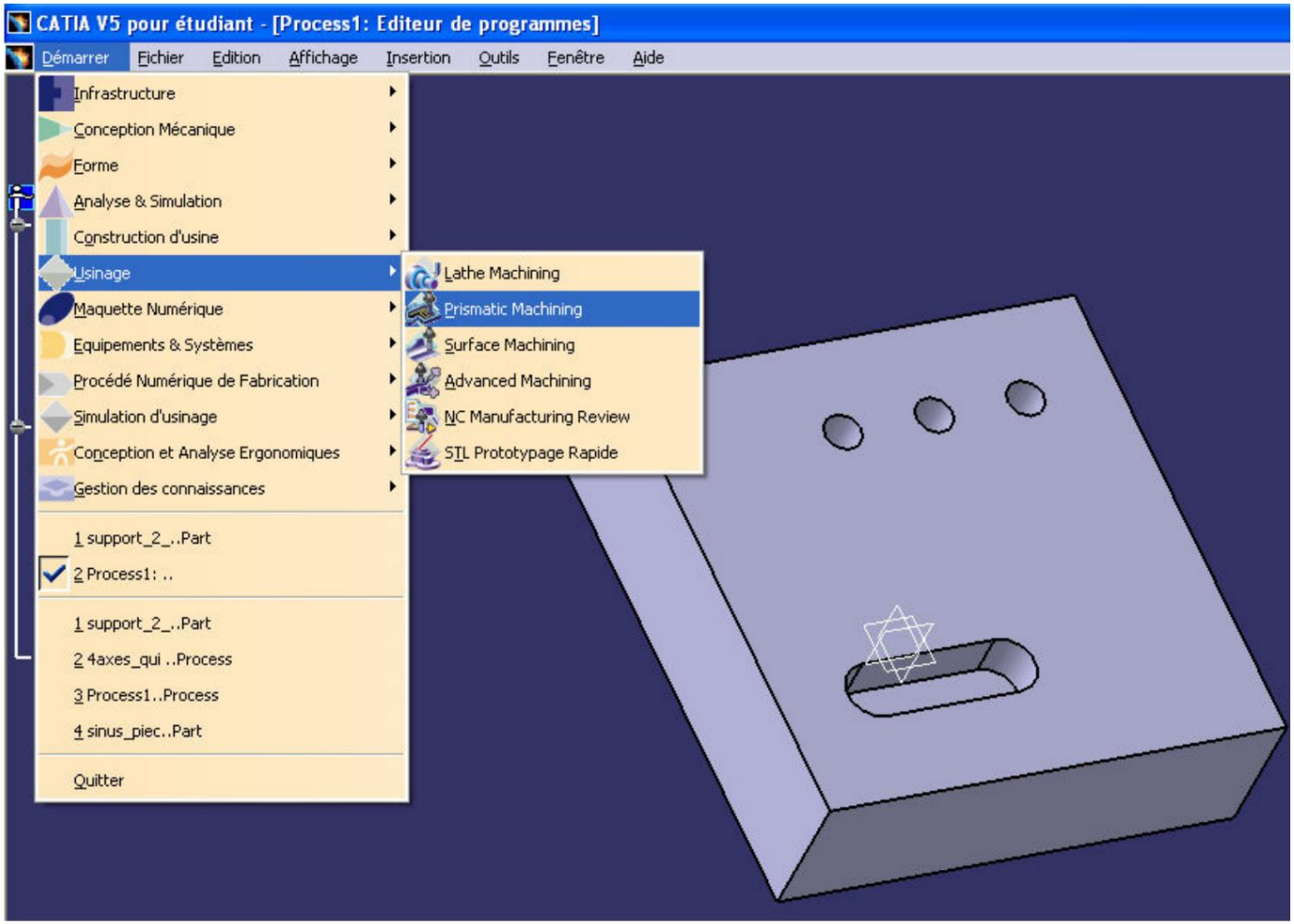
1 rectangular repetition of a hole diameter 6 depth 10 mm, 20 mm from the sides (spacing 20 between instances

A pocket 10 mm deep, diameter of the semi-circles = 12 mm (spacing of the center of the first arc of circles in relation to the two sides: 20)(distance 20 between the two centers of the semi-circles)

CAM requires the CAD definition of a raw part and a finished part. Copy-paste the part body in the same "Part", rename it "stock" (right click, properties), modify the extrusion to 32 mm and delete the holes and the pocket

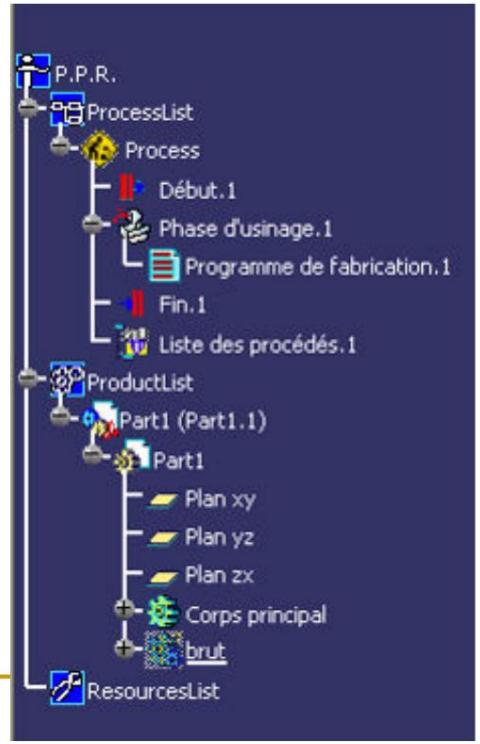
Save your work regularly

The “Prismatic Machining” workshop



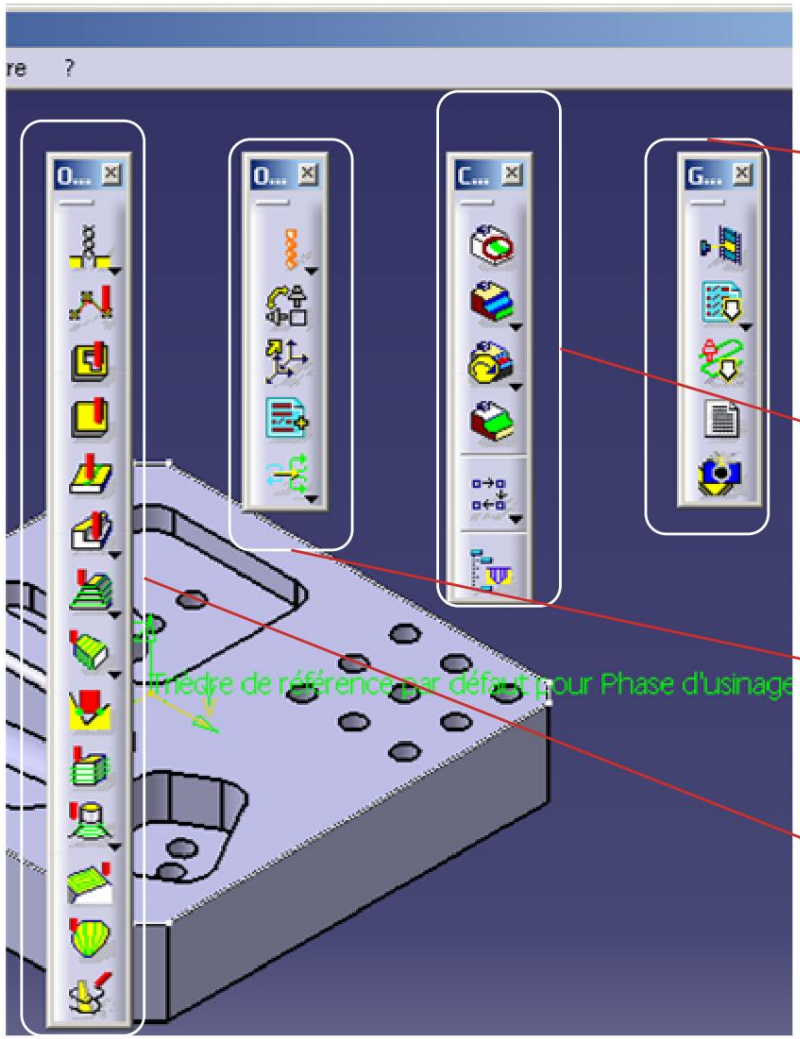
The construction tree is modified

Machining-specific icons appear



Save your work regularly

Useful toolbars



check the presence of the toolbar
Management of CN

outputs it allows to simulate the trajectories of the tool then to generate the ISO code

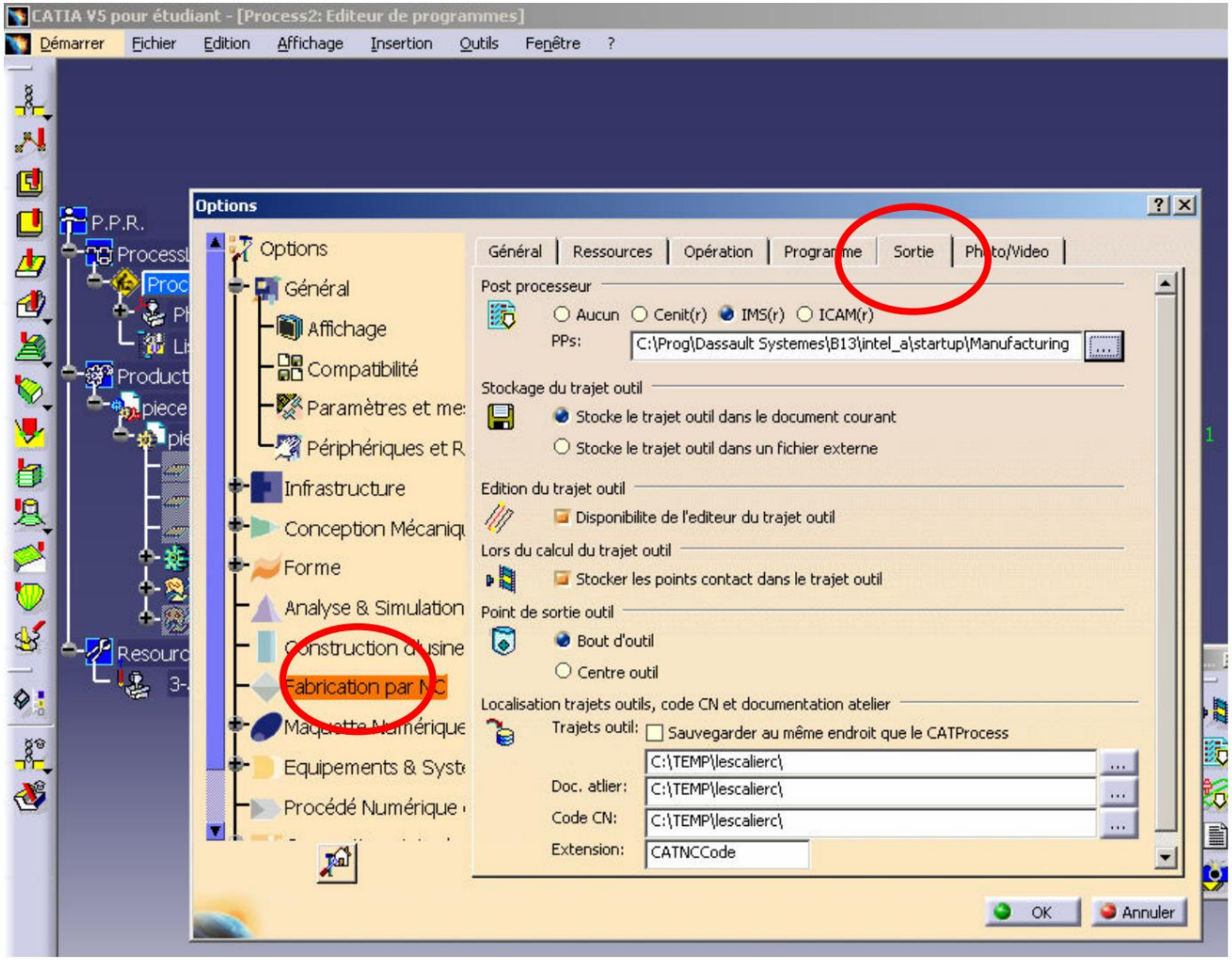
check the presence of the toolbar
Machining
components it is used to create recovery areas

check the presence of the toolbar
Auxiliary operations
it is used to generate tool changes

check the presence of the toolbar
Machining operations
it generates the machining operations

Save your work regularly

Default Options



in the Tools menu / Heading Options Manufacturing by NC (Machining) check:

the type of post processor (ie IMS),

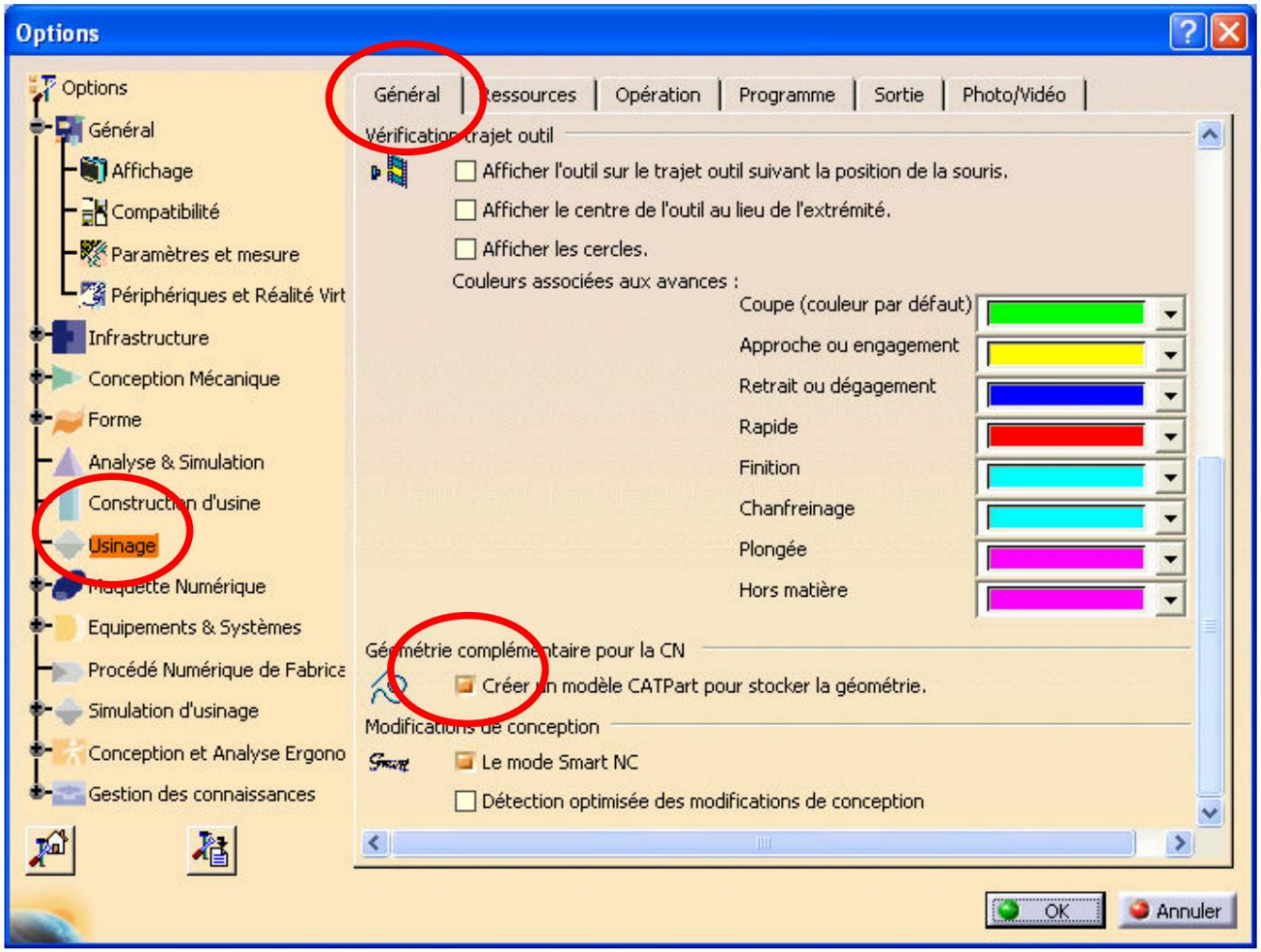
the point of the driven tool (ie end of tool),

the storage directories (ie C:\TEMP\...)

To improve performance, remove the background gradient: General / Display / Visualization

Save your work regularly

Default Options



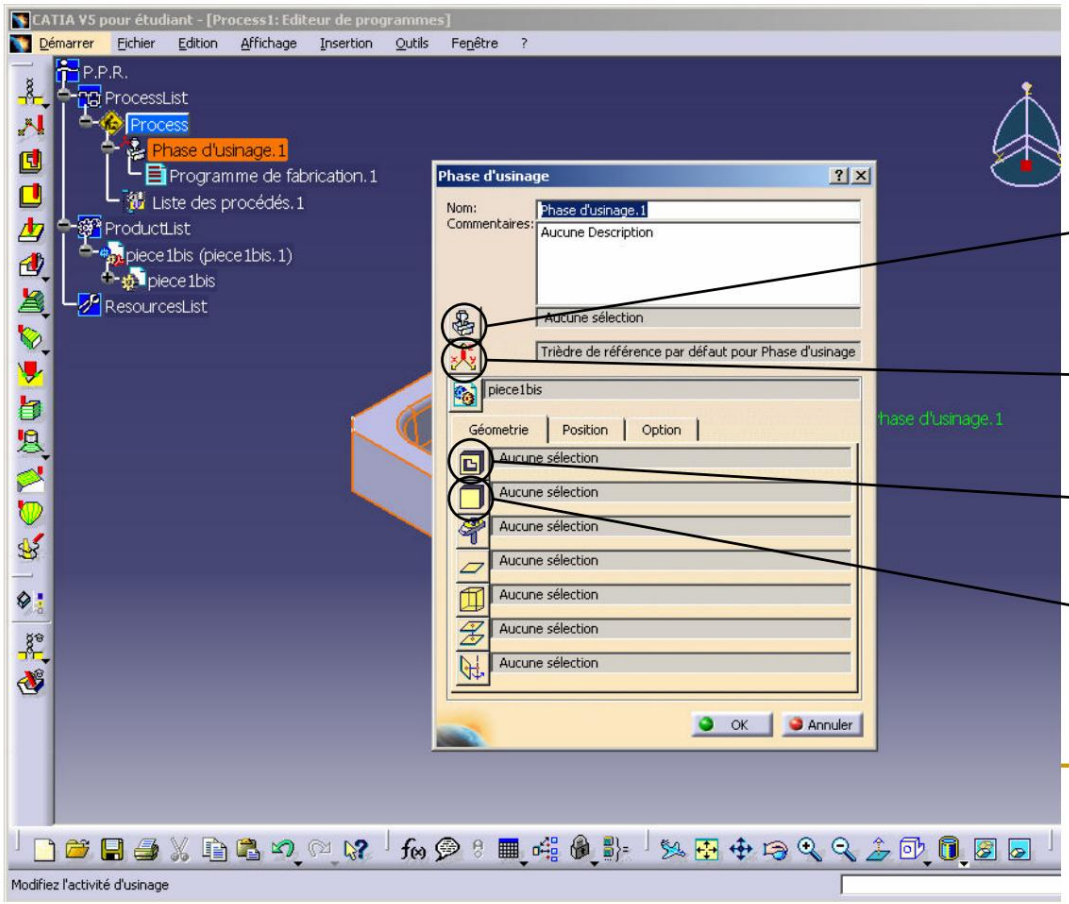
Check the box opposite

It allows to create a CATPart file which stores the intermediate geometries between each operation

Save your work regularly

Configuration of the machining phase

in the PPR tree, double click on Machining phase.1, a window appears, it allows you to define:
• the machine tool used (ie a 3-axis milling machine) and the post-processor to use, • any catalog of tools (ie a set of tools capable of fully producing the part and already present in the tool magazine), • the frame in which the tool moves (ie the position of the program origin and the orientation of the part in relation to the axes of the machine tool), • the part to be machined and the raw part,



click here to define the machine tool, the post-processor and the tool catalog

click here to define the marker

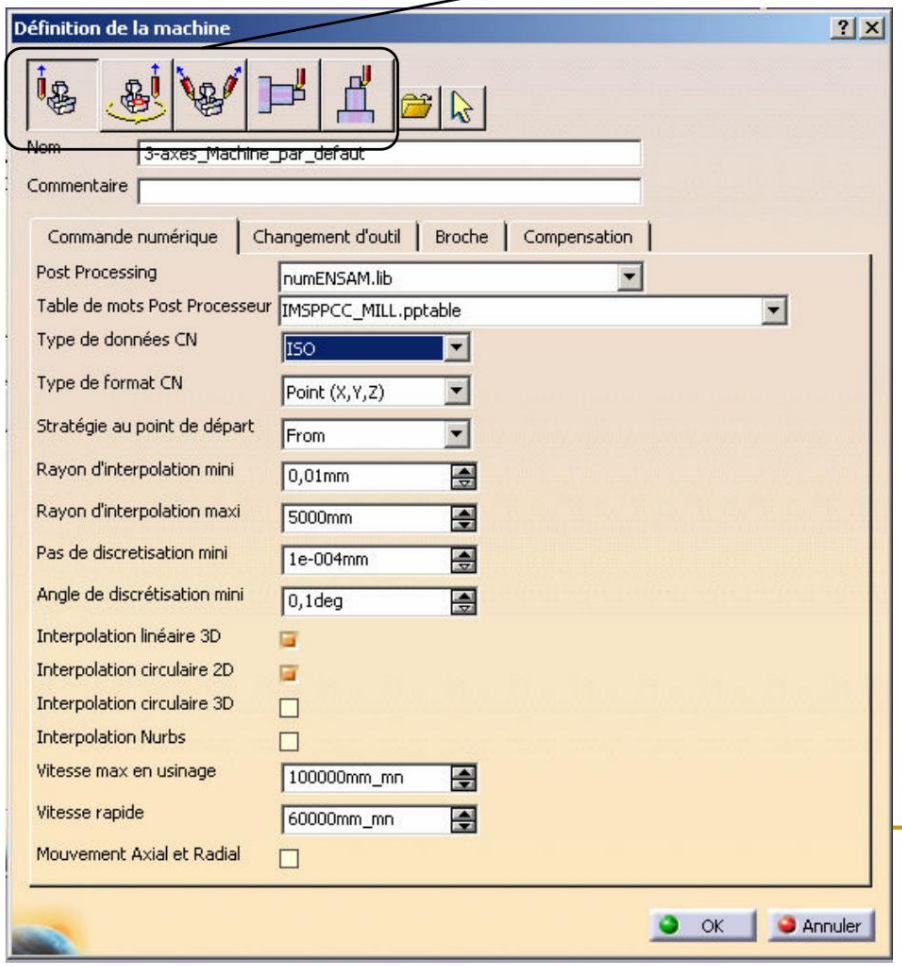
click here to define the workpiece

click here to define the blank

Save your work regularly

Choice of machine

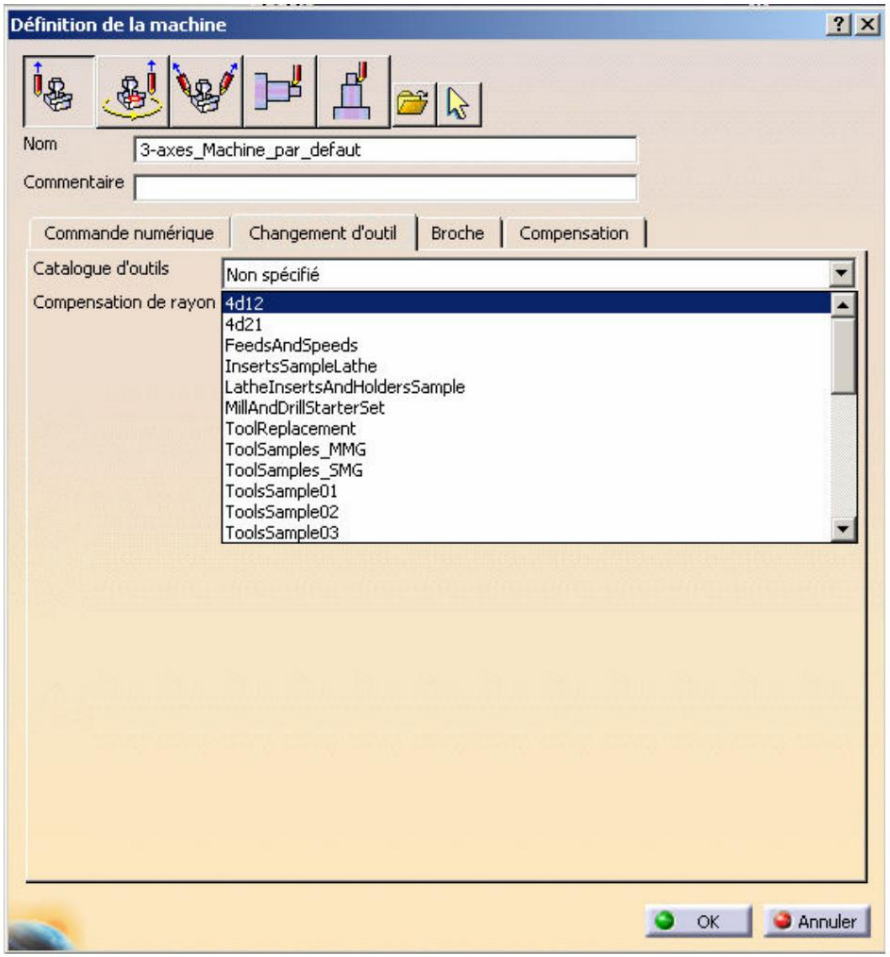
icons defining the architecture of the machine tool used



• the machine tool is a 3-axis milling machine (one of the machine tool architectures offered by default), • the post-processor is fanuc21i.lib • the post-processor word table is of IMSPPCC_MILL type • the language to be exported is ISO type

Save your work regularly

Choice of tool catalog



the catalog of tools to be used is MillandDrillStarterSet

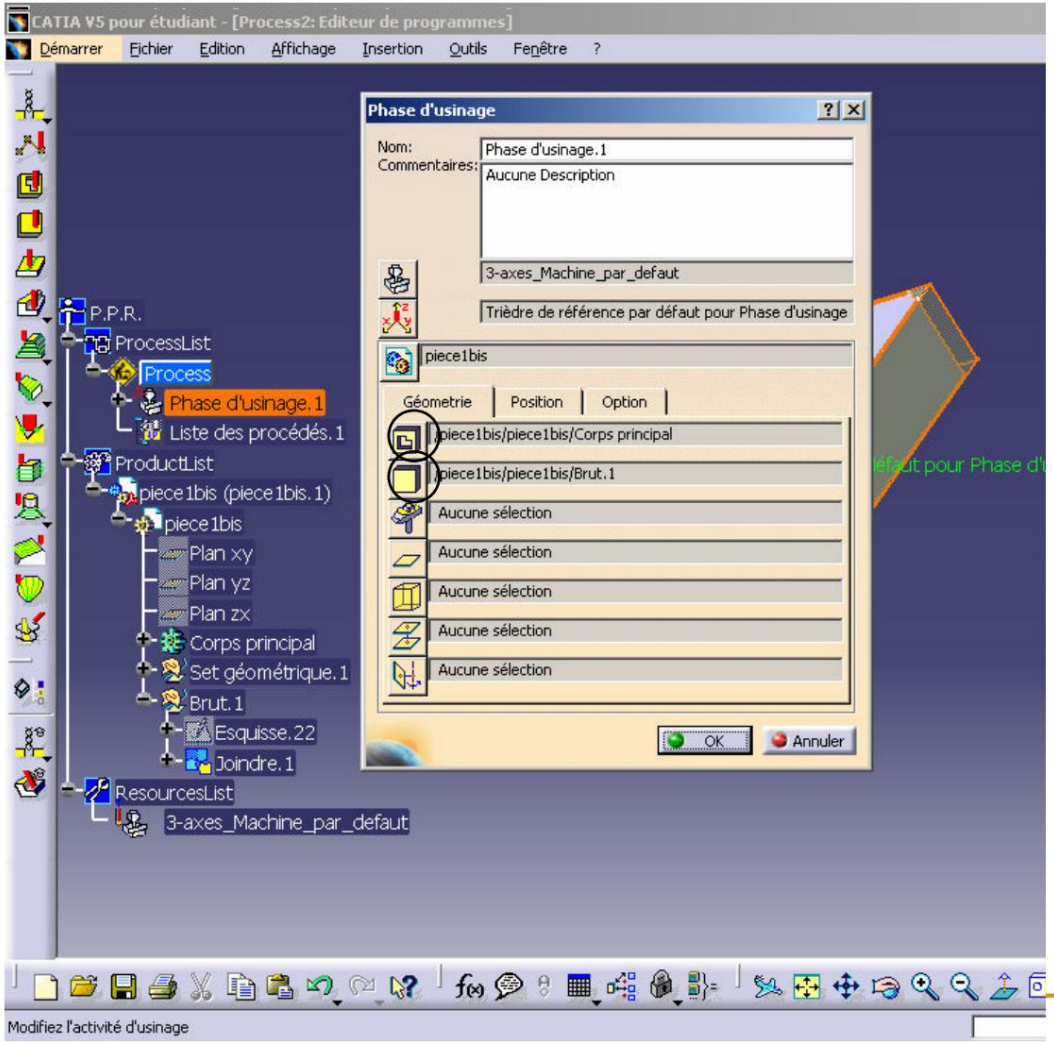
it includes basic milling tools (modifiable) as well as the cutting conditions for these different tools in roughing and finishing

Selecting a tool catalog is optional, but can save you time if you always use the same tools (machining center with tool magazines for example)

The tool catalog can be modified, exported, saved, etc.
It can be generated by an Excel file (requires knowledge of the correct naming labels for tool characteristics) or by modifying existing tools (easier and faster).

Save your work regularly

Rough/finished part definition

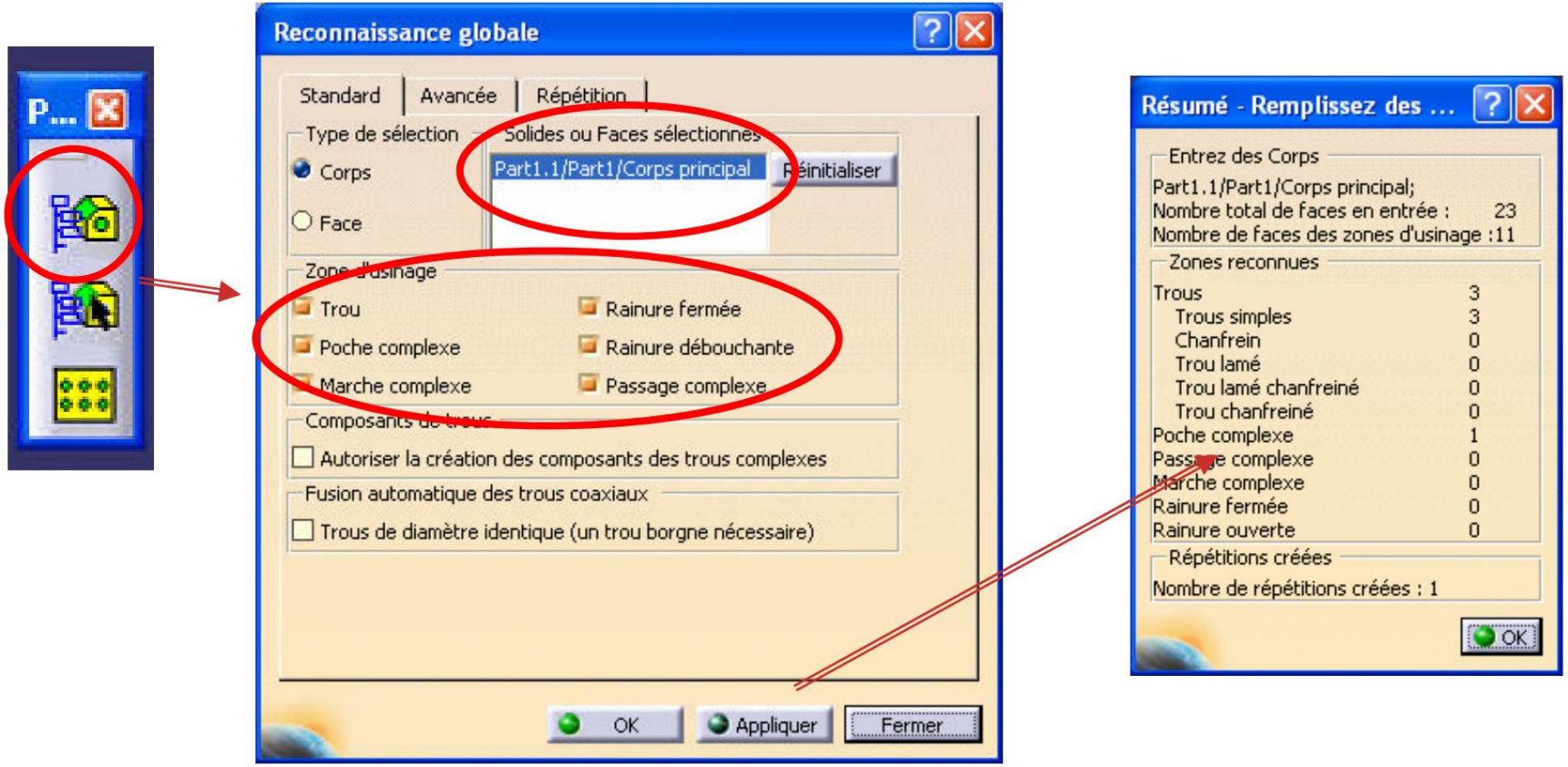


• define the part to be machined (Main body of the construction tree) • define the raw part (“raw” body) to do this, click on one of the icons in the Machining phase window then on the corresponding item in the building tree.

Double-click in the CATIA window to see the Machining phase window reappear. Click OK to exit.

Save your work regularly

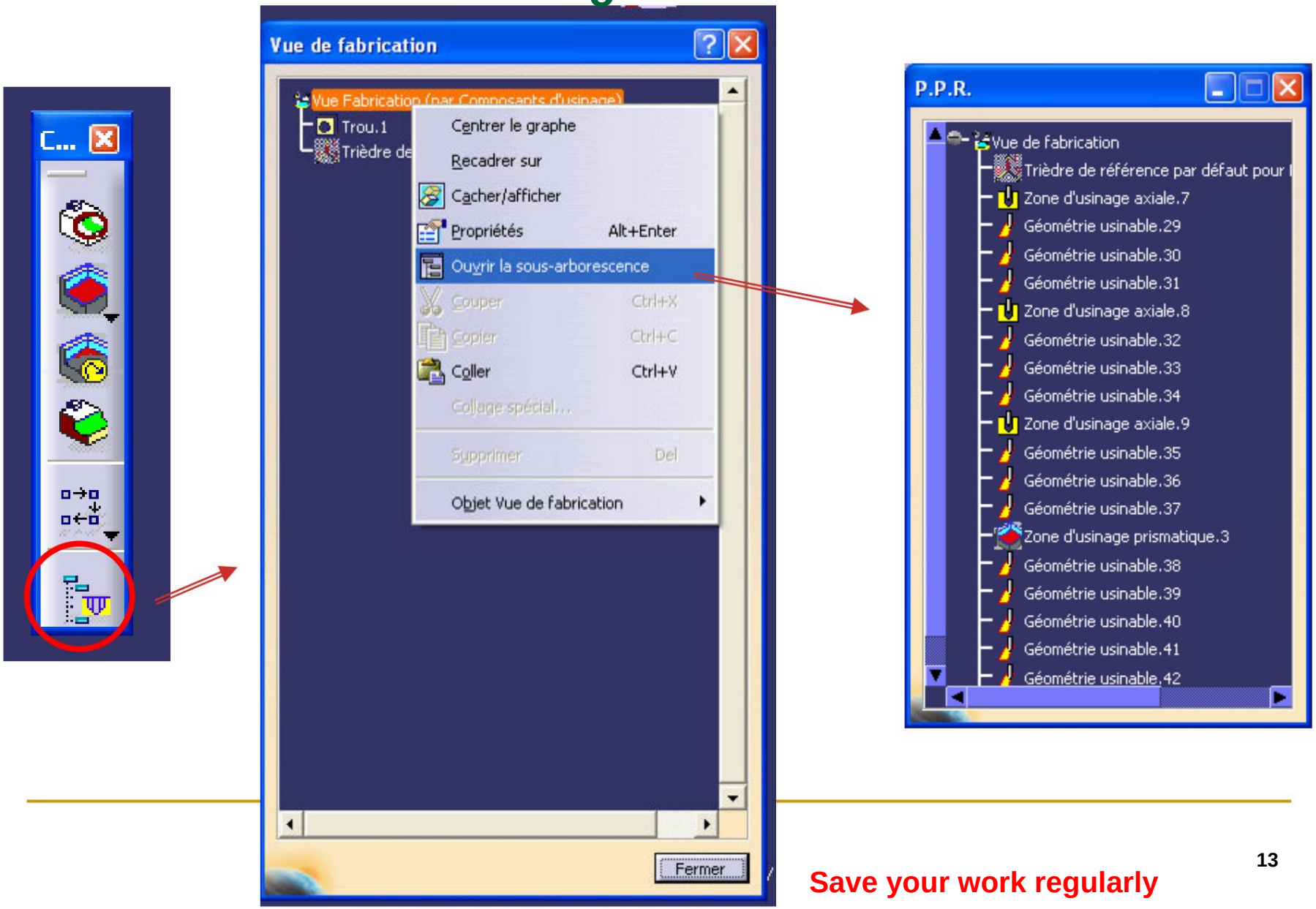
Automatic recognition of machining entities



(This operation is optional and its proper functioning strongly depends on the design of the part: example if a hole has been made with the Pocket icon, it will be recognized as a pocket)

Save your work regularly

Visualization of machining entities



Save your work regularly

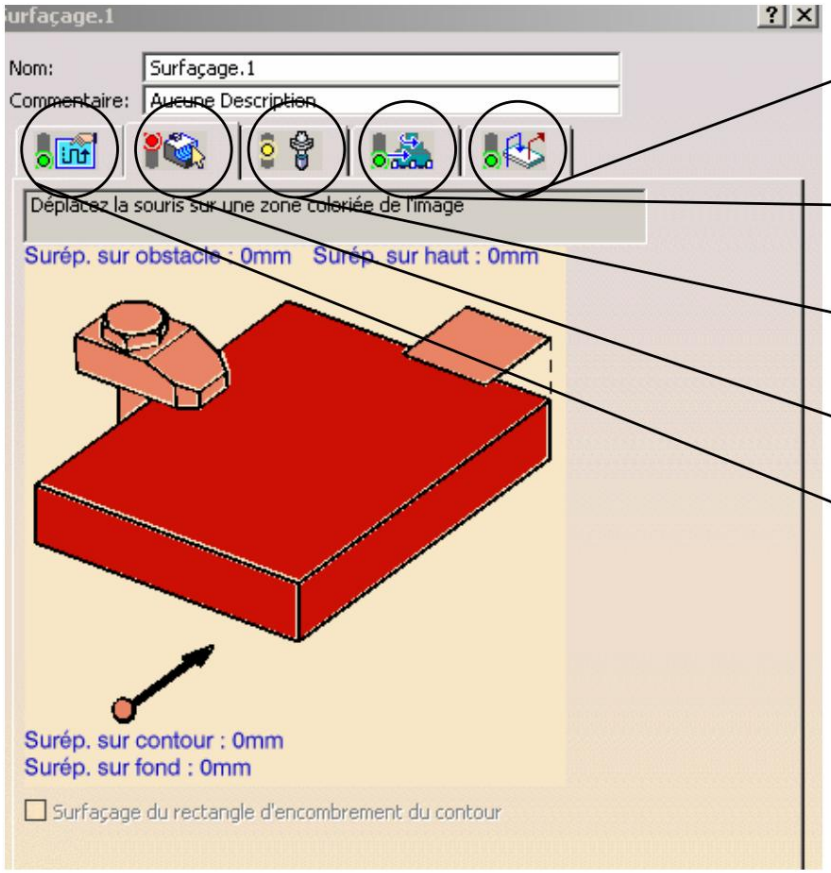
The generation of trajectories starts here. The program is created by a sequence of elementary operations.

For each of these operations, the programming mode is identical. It consists of explaining to CATIA: • which tool to place in the spindle nose, • which part of the part to machine, • how to machine the surface (defining an operating strategy, ie a type of trajectory, or even engagement conditions , ie axial and radial depth of cut), • what cutting and feed rates to use, • how to get to and from the surface to be machined.

• Attention, the operations are created following the entity highlighted in the tree PPR, the easiest way being therefore not to select anything else in the PPR and to generate the operations in the order of their realization

Save your work regularly

Typical presentation of an operation



what approach, docking, withdrawal and clearance conditions

what cutting speed and feed

which tool to use

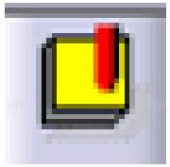
what part of the workpiece to machine

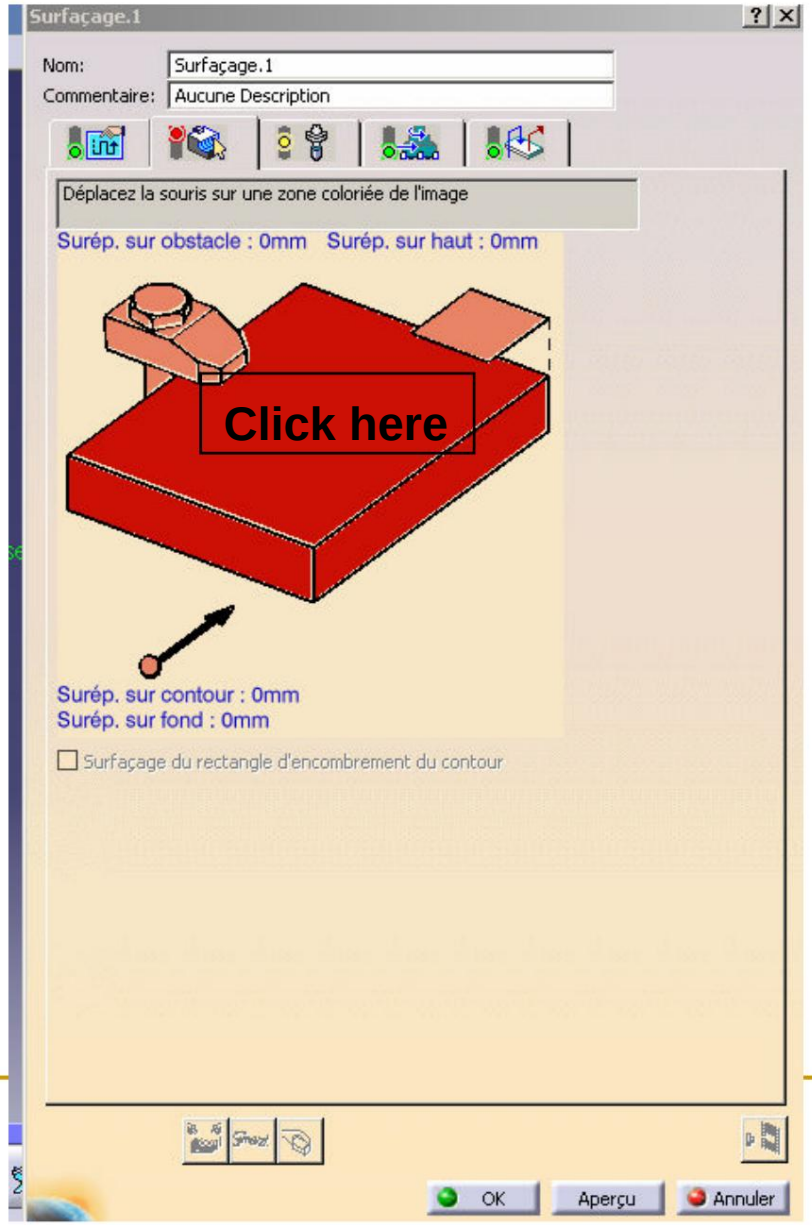
what mode of machining

By clicking on the icon of a machining operation, this window appears. It includes 5 tabs corresponding to as many windows where you can specify information.

Save your work regularly

Surfacing operation





Then click on the top surface of the part.

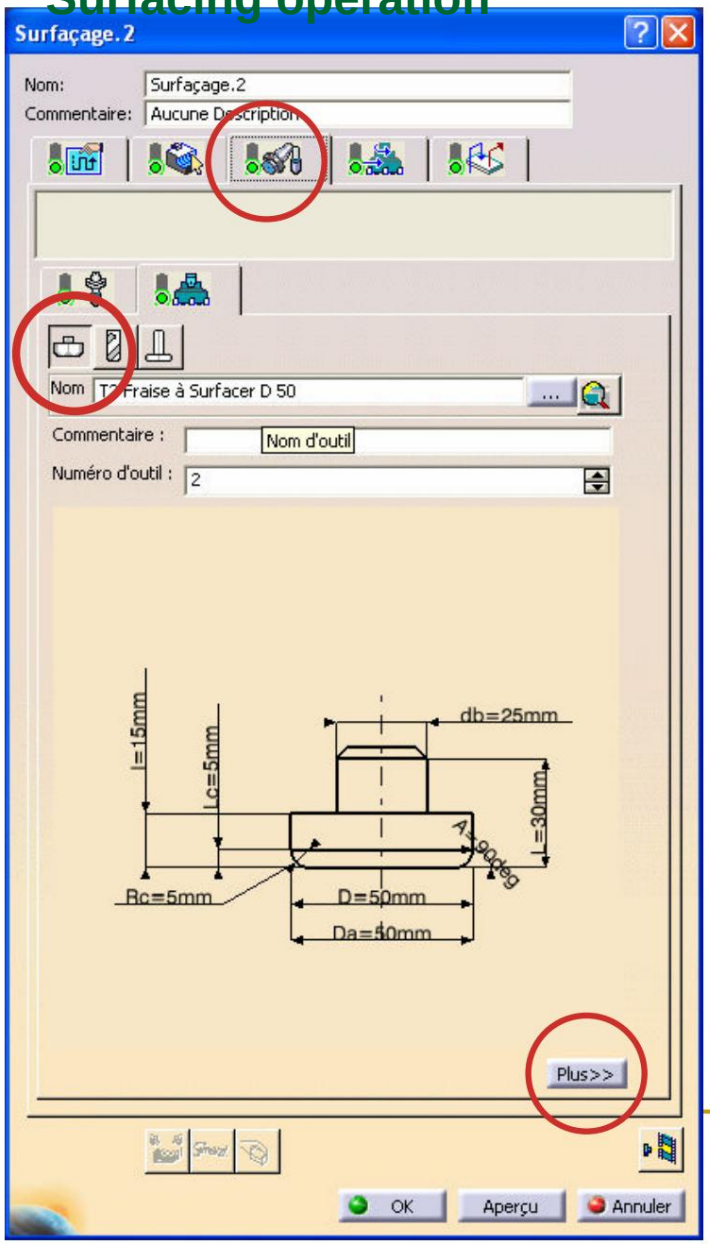
Double-click in the CATIA window to review the Surfacing window.1

The icon indicator (green, orange or red) indicates whether CATIA has sufficient information.

Then click on the side faces of the drawing opposite and select the contour of the surface in the same way

Save your work regularly

Surfacing operation



Click on the Tool tab

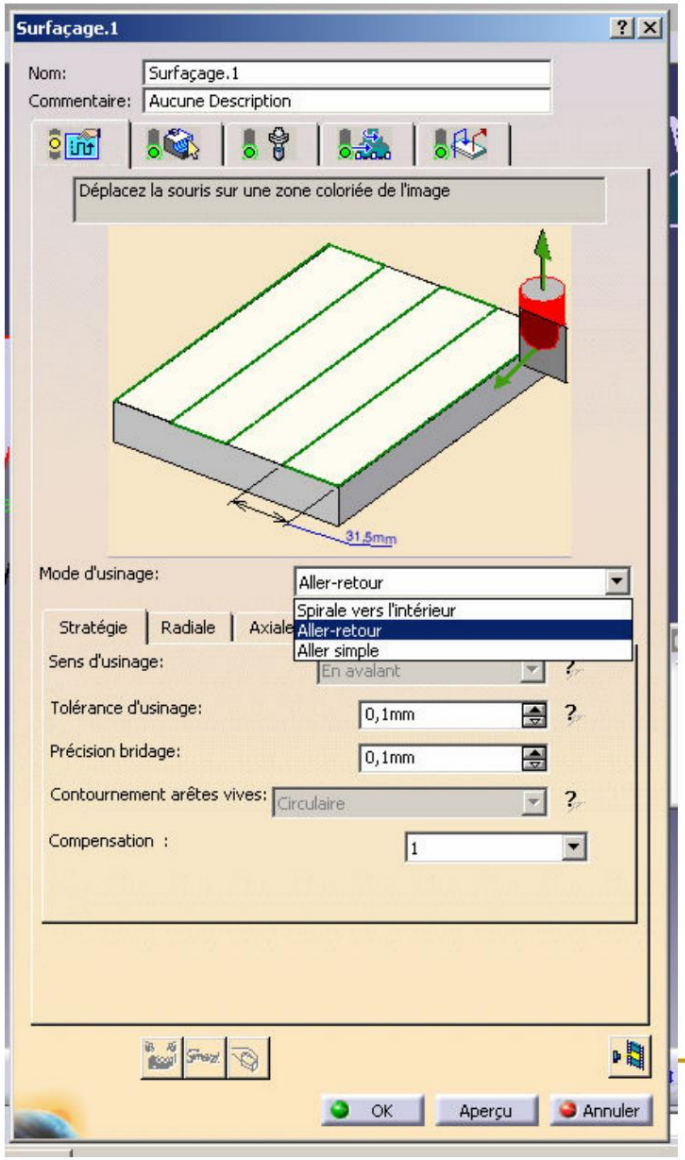
Click on the face milling cutter icon.
The tool number will correspond to the “T” instruction which will appear in the CN code.

You can also find the right face milling cutter by clicking on "Select by request" (icon with the magnifying glass) then browsing through the tool catalogs offered.

Click on “More”
You can modify the geometry of the tool, the offset (compensation tab) and indicate the cutting conditions

Save your work regularly

Surfacing operation

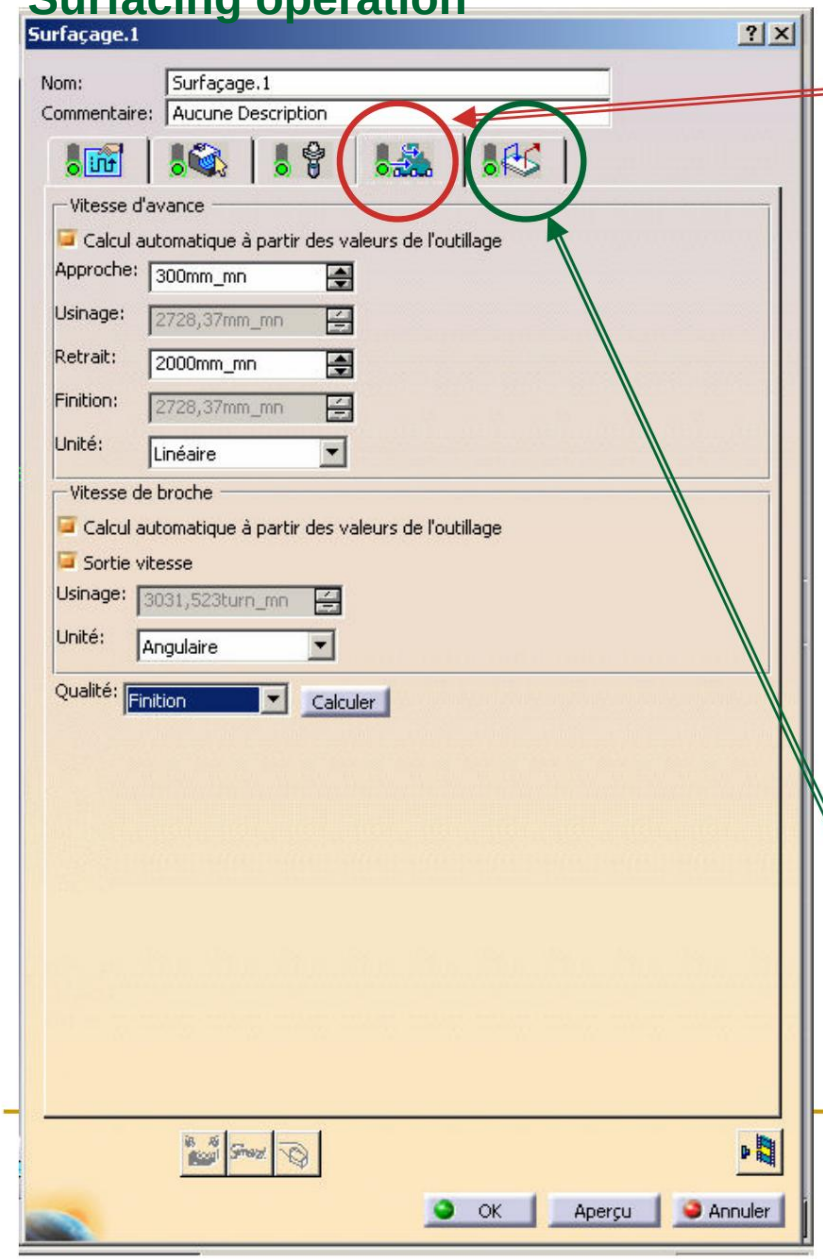


This tab defines the machining strategy.

You can modify the overlap ratio between two passes, modify the peak height, etc.

Save your work regularly

Surfacing operation



Click on the Speeds tab.

Set Quality to Finish.

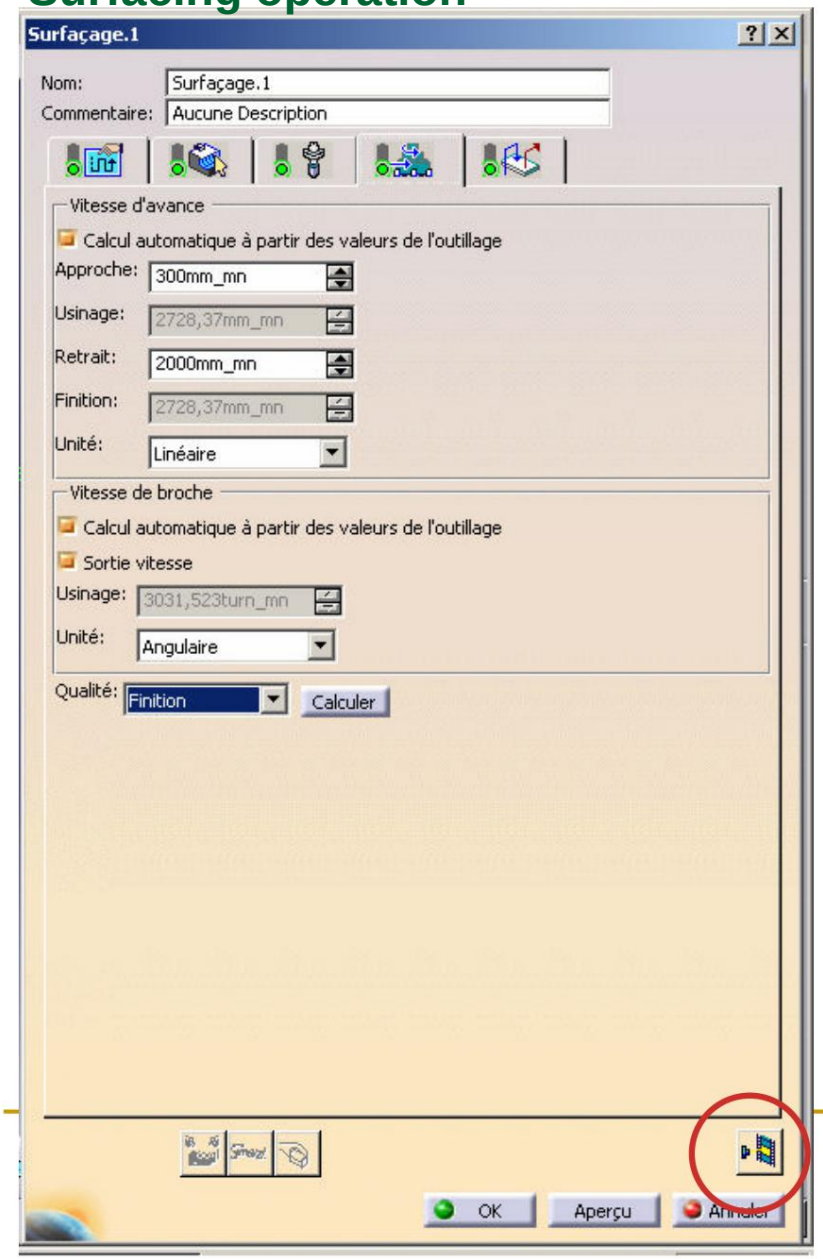
The operating conditions used are those defined in the tool catalogue.

Switching the Quality item from Roughing to Finishing and vice versa changes the rotation frequency and the feed rate.

The last tab allows you to modify the tool path in approach (before material removal) and in withdrawal (after).

Save your work regularly

Surfacing operation



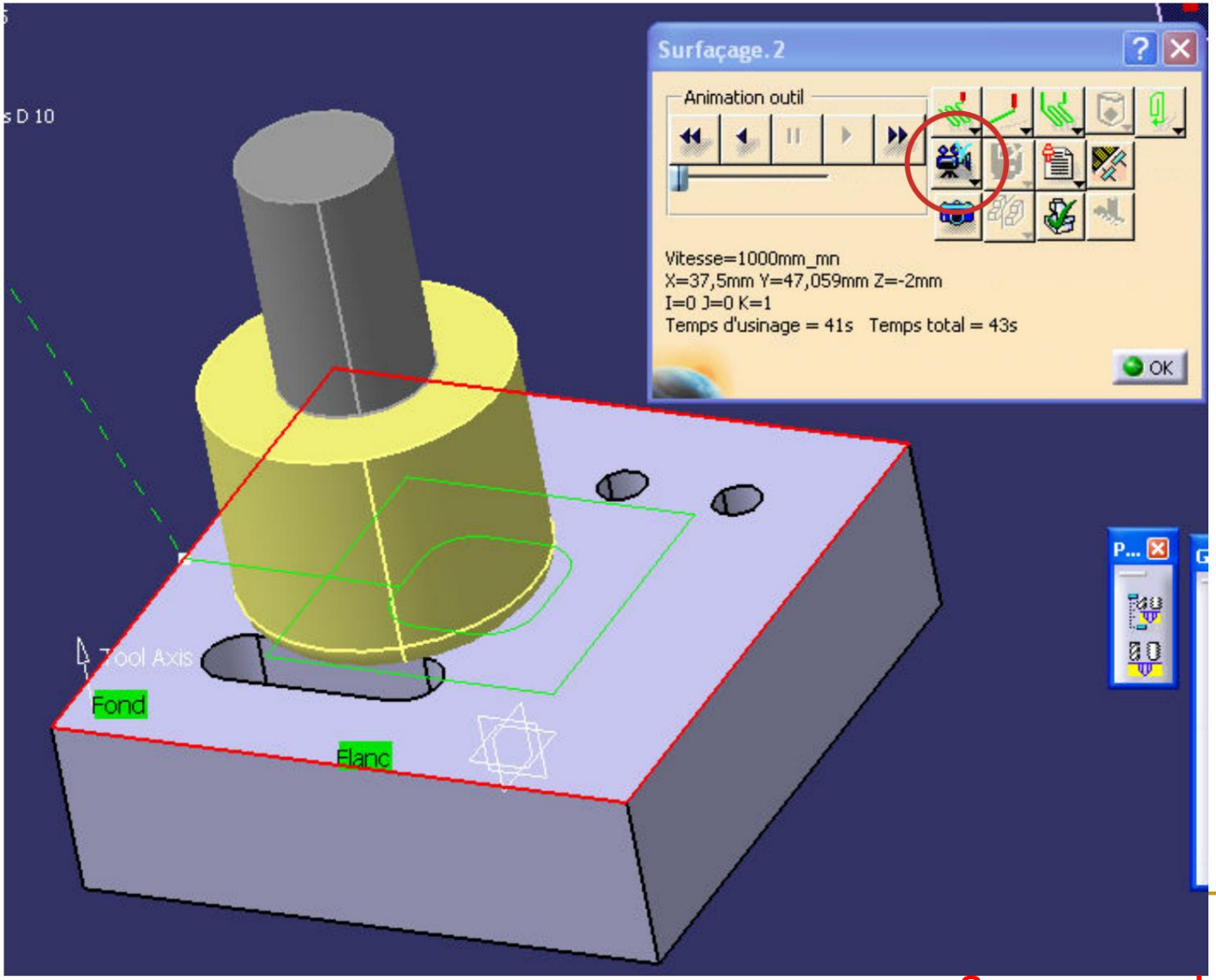
Click on the tool path animation icon.

This icon validates the configuration of the machining operation.

To modify an operation, double-click in the construction tree, make the modifications and validate by clicking again on this icon. Simply clicking OK is not enough.

Save your work regularly

Surfacing operation



Different visualization options are possible.

Visualize the trajectory of the face milling cutter.

The Video icon will allow you to view the material removals.

Click on OK to return to the Surfacing window.1

Click on OK to validate the parameters.

The surfacing is now programmed.

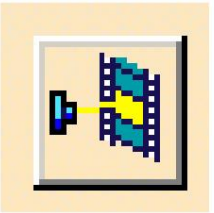
Save your work regularly

Surfacing operation

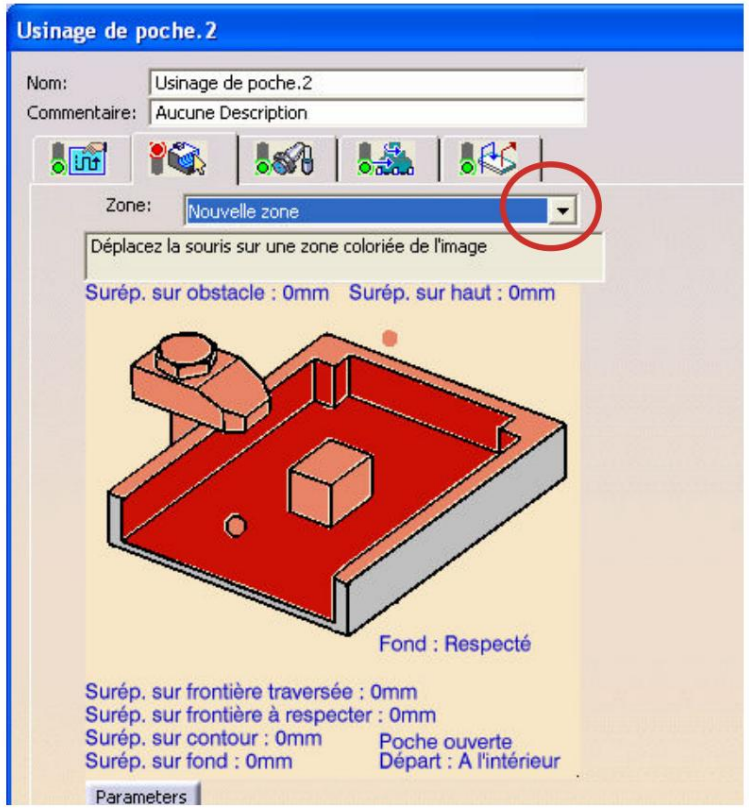
The surfacing operation appears in the tree.

It is said to be Solved therefore correctly programmed (this does not necessarily mean that the conditions of realization are compatible with the tool or the machine tool).

The mention Solved is obtained after animation of the tool path.



Machining of the oval pocket



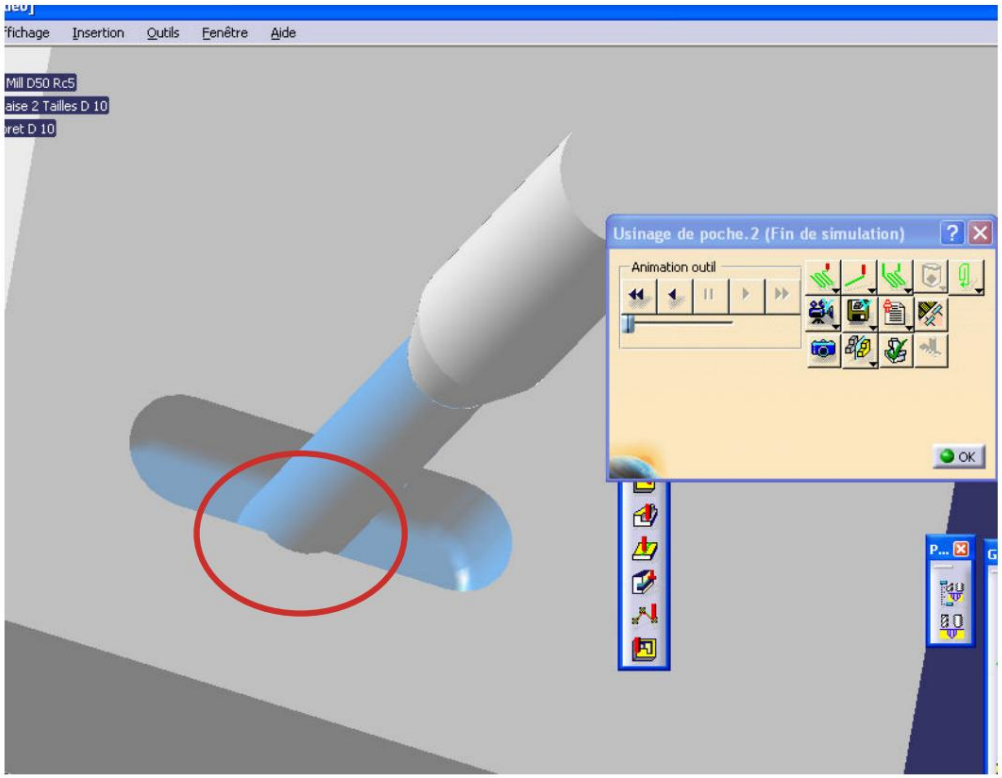
Select the proposed prismatic machining area. It corresponds to the pocket recognized automatically

Keep the default machining strategy.

Select the cutter 3 sizes diameter 10, paying attention to the corrector and the cutting conditions. A message of this type may appear when changing the corrector, this is normal.

Save your work regularly

Machining of the oval pocket

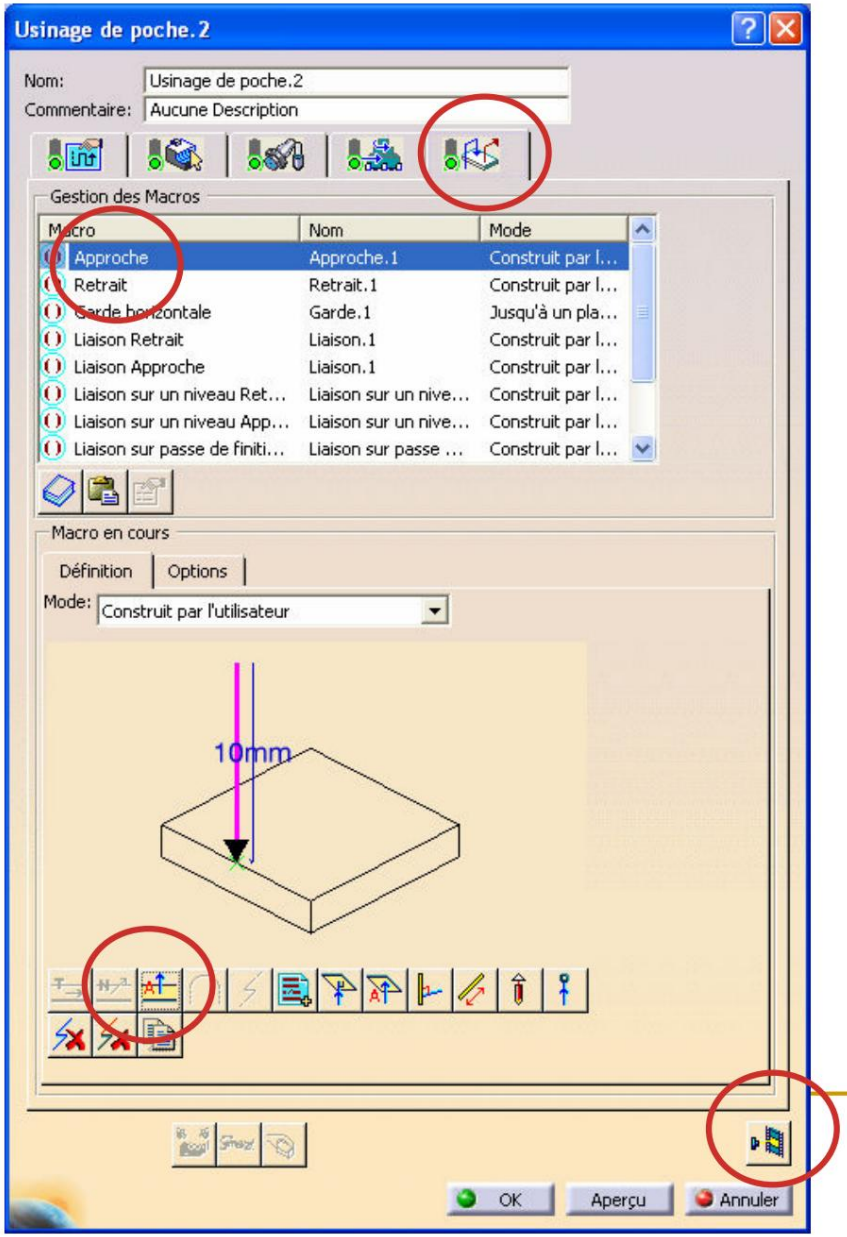


Depending on the strategy chosen (here by swallowing), the simulation reveals a defect during machining. It is actually a collision between the tool and the part when moving the tool from the tool change point to a programmed position

To correct this we will generate an axial approach to the tool

Save your work regularly

Machining of the oval pocket



Click on the Approach/Retreat tab

Highlight "Approach"

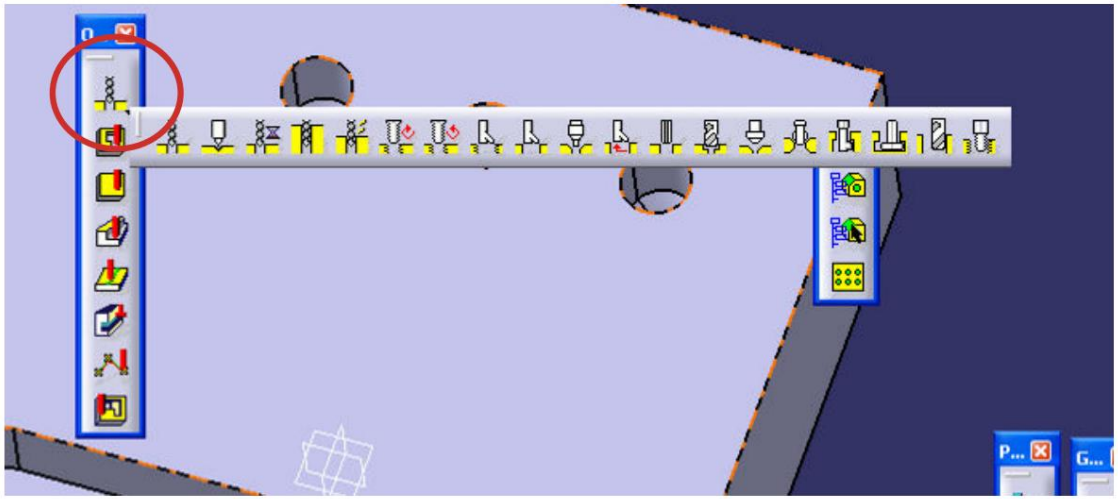
Add an axial path (its value can be modified by double-clicking on "10 mm", a right click allows you to define the approach speed)

Right-click
“Approach” and “Activate”

Start tool path simulation. The defect generated by the tool during its descent into the middle of the material has disappeared.

Save your work regularly

Drilling cycle



There are many pre-programmed solutions for drilling / tapping
Select the simple drilling cycle
Select the hole pattern (the pattern created in CAD allows to have only one drilling operation instead of 3)
Choose the drill, with its compensator (gauge) and its cutting conditions

Save your work regularly

Drilling cycle

Click on the tool path animation icon to view the programmed paths.

If the drill collides with the part in its path between two consecutive holes, add an axial retraction and/or an axial approach as before.

Click on OK when the trajectory is satisfactory.

Save your work regularly

NC code generation

All material removal operations being defined, the CN code will be generated.

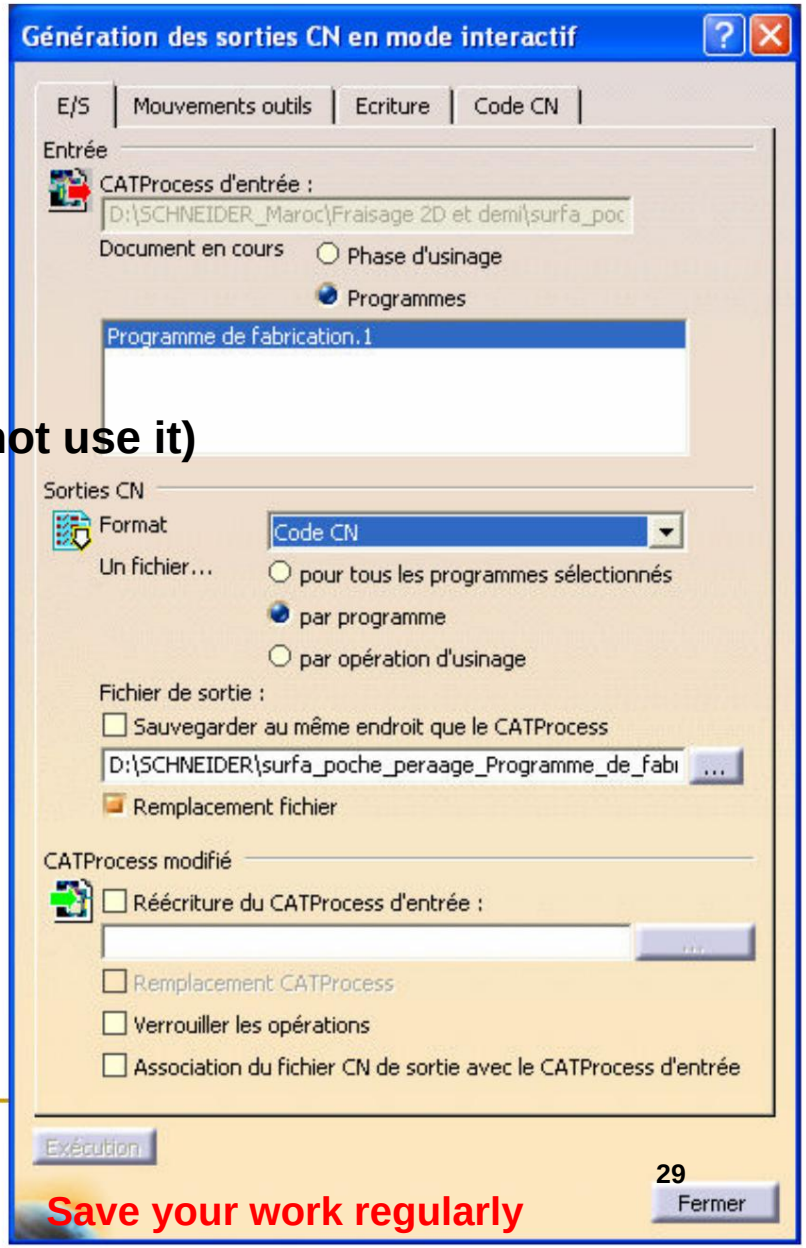
Each operation will generate (from a table of words and macros) part of the code

The goal is now to collect these parts of code and export it in text format (.p for example for NUM command director)

NC code generation

Click on “Generate NC code in Interactive mode”

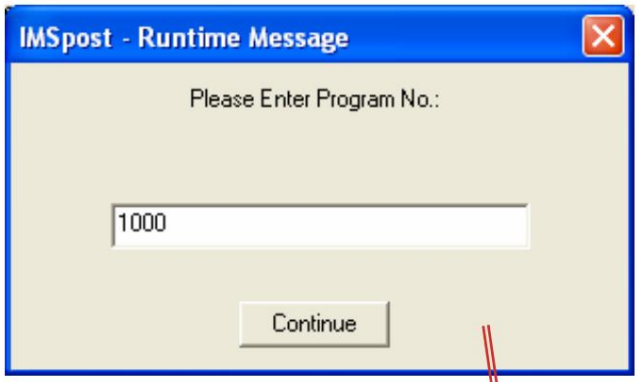
Another generation mode is possible (batch mode), which makes it possible to store and concatenate different programs (we will not use it)



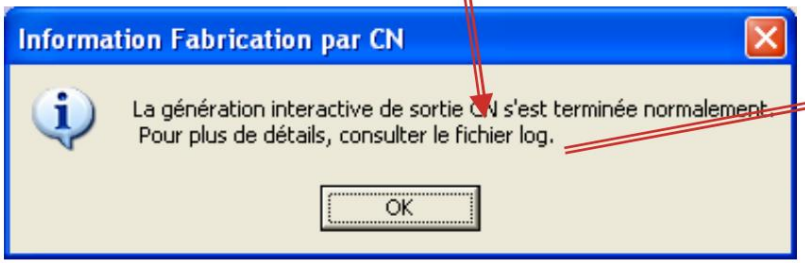
Check the format of the generated file: CN code.
Check the location of the generated file.
Check the post-pro used under the "CN Code" tab
Execute the generation of the program.

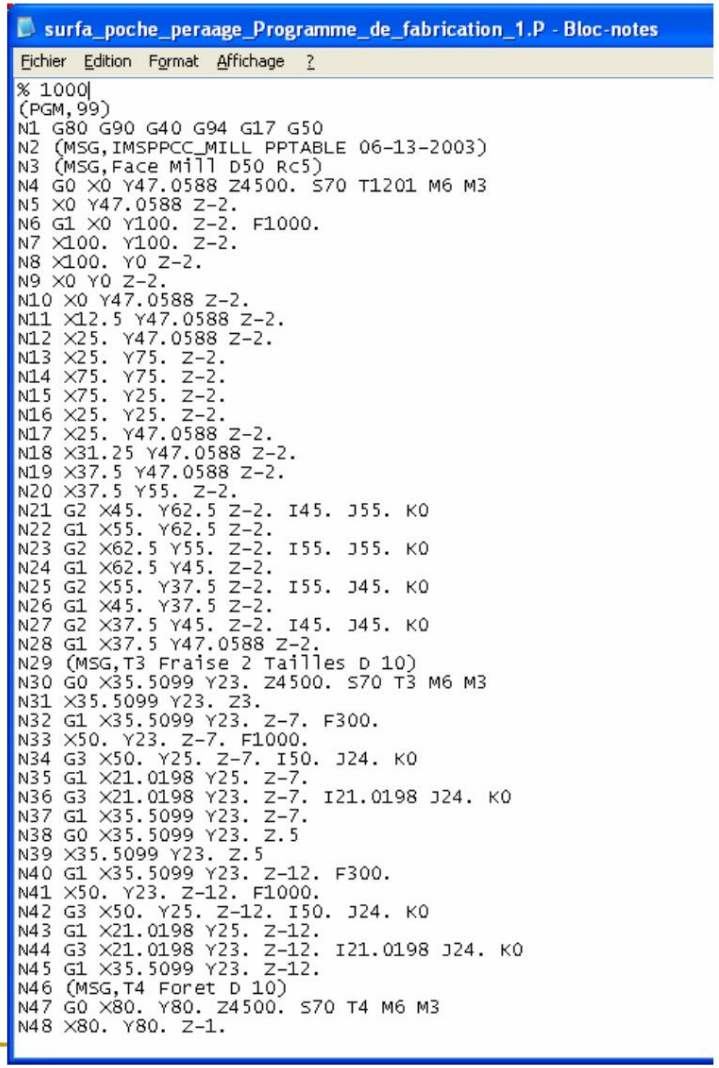
Save your work regularly

NC code generation



Enter the program number (check that it is not already "taken" from the machine tool)





Save your work regularly

End of TP1

You have just completed your first CAM program, congratulations!

To remember :

CAD influences CAM (automatic feature recognition)

Each operation works on the same principle It is a question of defining what is machined (entity), how (strategy, approach/retraction), with which tool (and which cutting conditions)

The simulation makes it possible to check the consistency of the programmed operations and to correct the tool trajectories The generated program should not be used with blind confidence, a rereading of the key points (start of the program, change of tools, loading of the gauges, etc.) and a simulation on the machine tool will be able to deal with the biggest errors. A machining time that is much too long or, on the contrary, much too short (available on the command director) can also be an indicator of a bad feed or programmed cutting speed e

Save your work regularly

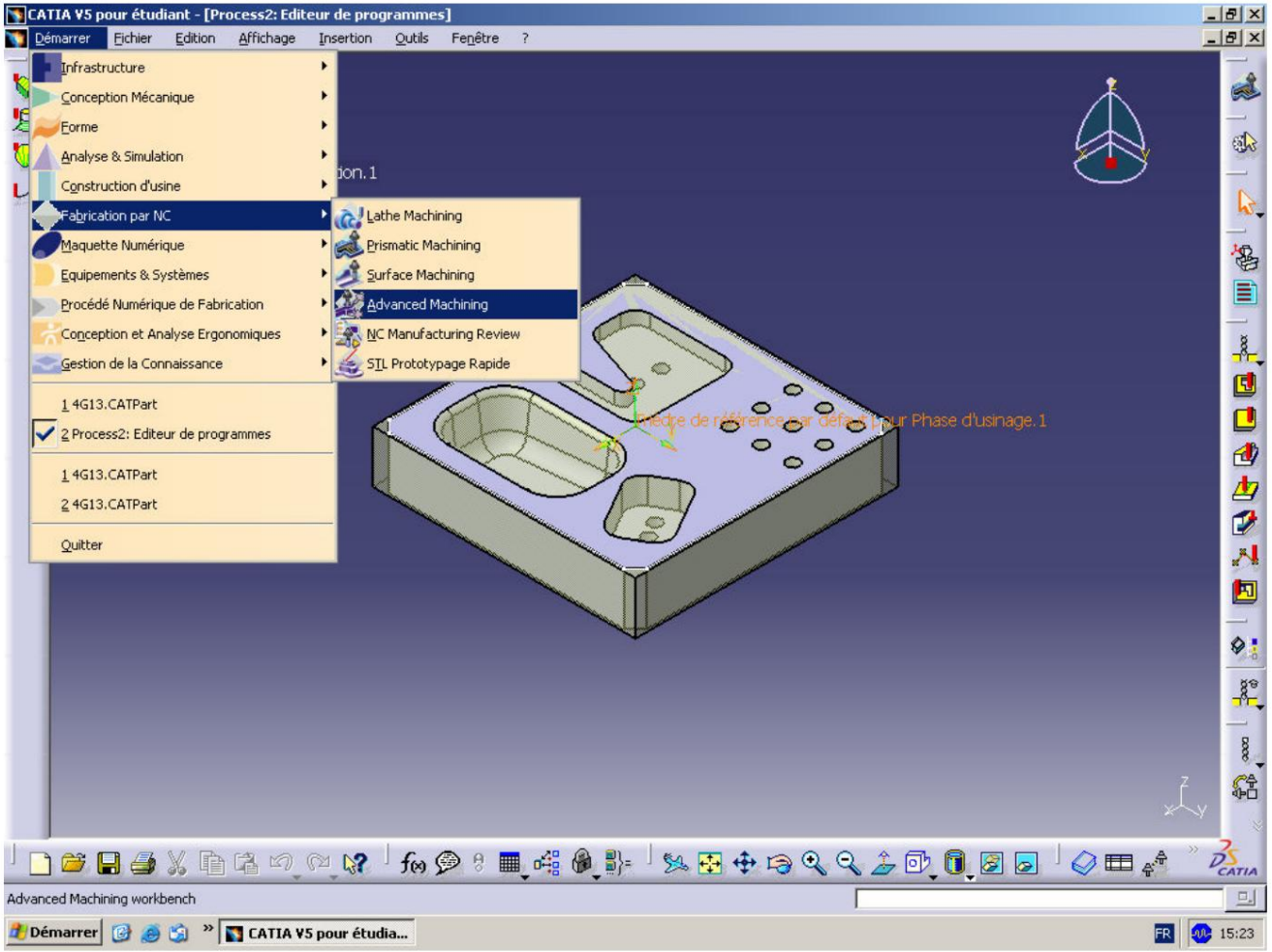
Exercise 2: 3D milling

The objectives of this lab are:

- **Consolidate the skills acquired during the first practical work: the different stages will be less detailed than in the previous practical work, the spirit of initiative is favored over the "press button" approach**
- **Use the functions: Machining recovery zone, Ramp approach**
- **Perform a roughing then finishing operation on a “3D” pocket**

Save your work regularly

Open the room



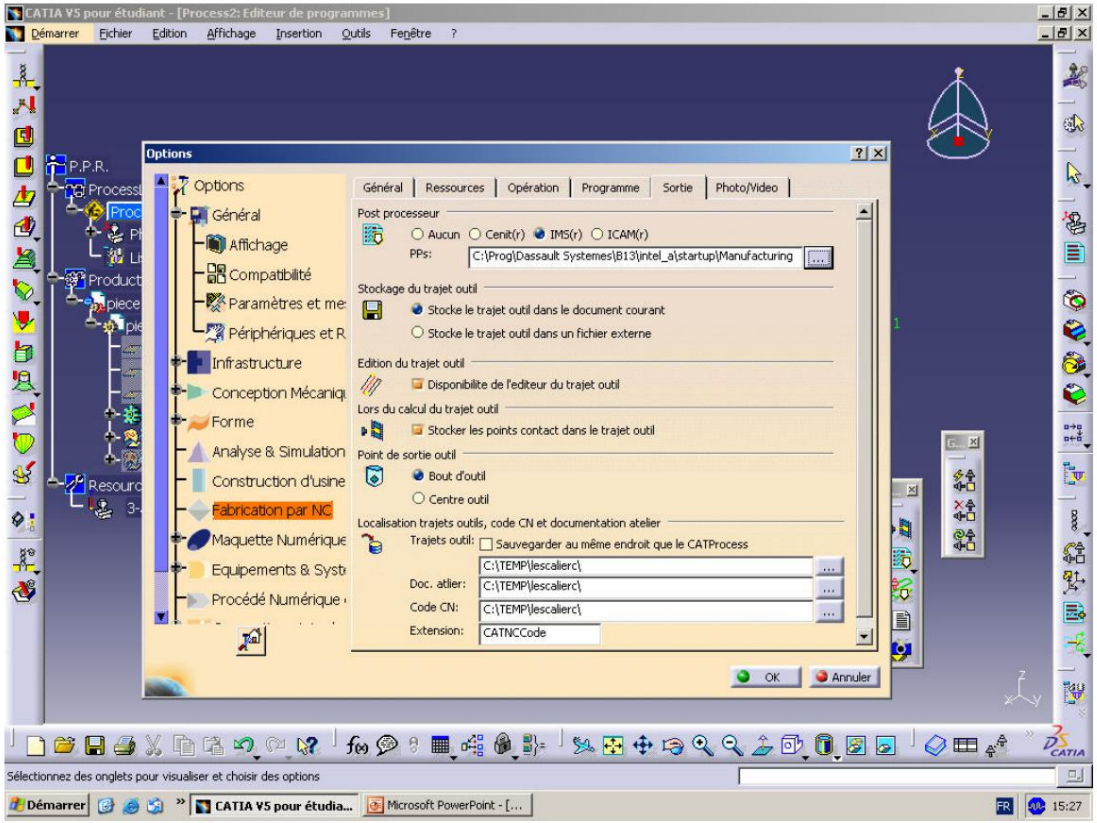
open the CATpart file of the lab

start the activity

Manufacturing by NC / Advanced Machining

Save your work regularly

Initial checks



in the Tools menu / Heading Options Manufacturing by NC check:

the type of post processor (ie IMS),

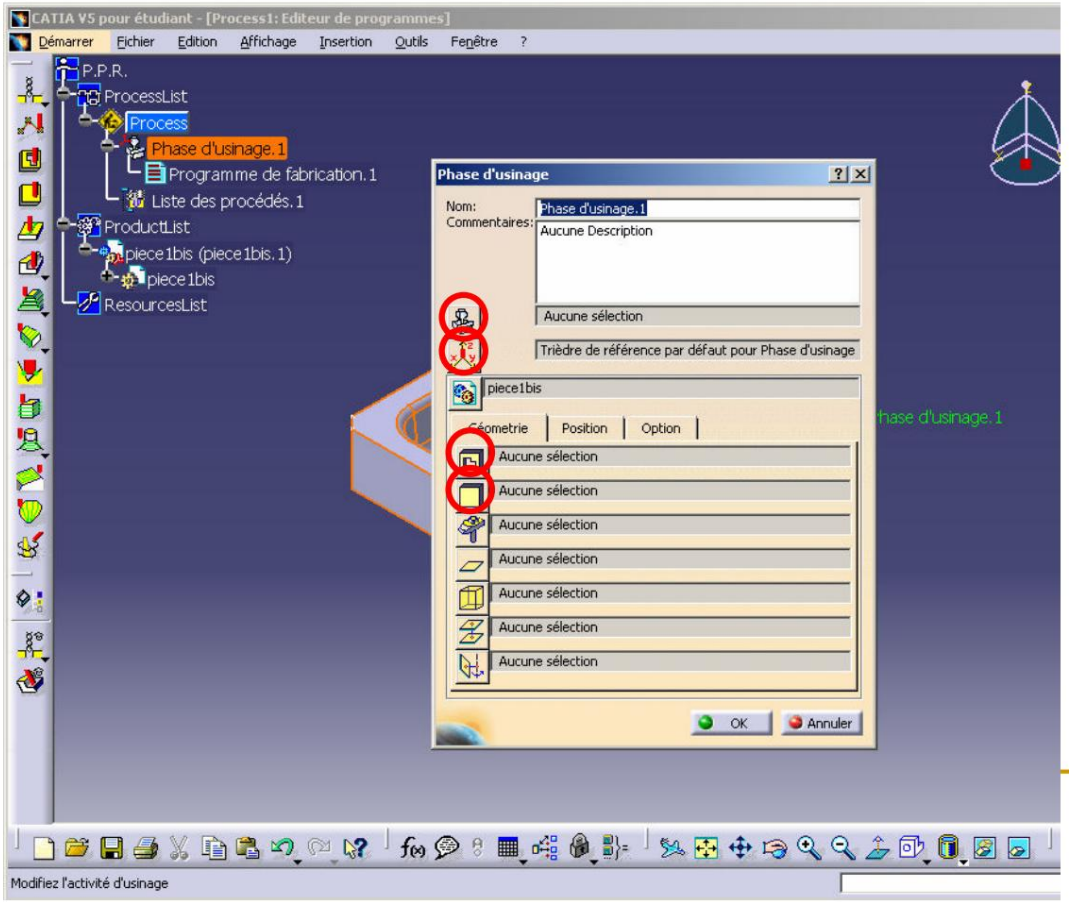
the point of the driven tool (ie end of tool),

the storage directories (ie C:\TEMP\...)

Save your work regularly

Configuration of the machining phase

in the PPR tree, double click on Machining phase.1, define: • the machine tool used (ie a 3-axis milling machine) and the post-processor to use, • The "TP2_Meknes" tool catalog, • the reference trihedron (center of the part, finished upper surface, • the part to be machined and the raw part,



- **post-pro:**
  - **fanuc21i.lib**
  - **IMSPPCC_MILL**
  - **ISO code type**

Save your work regularly

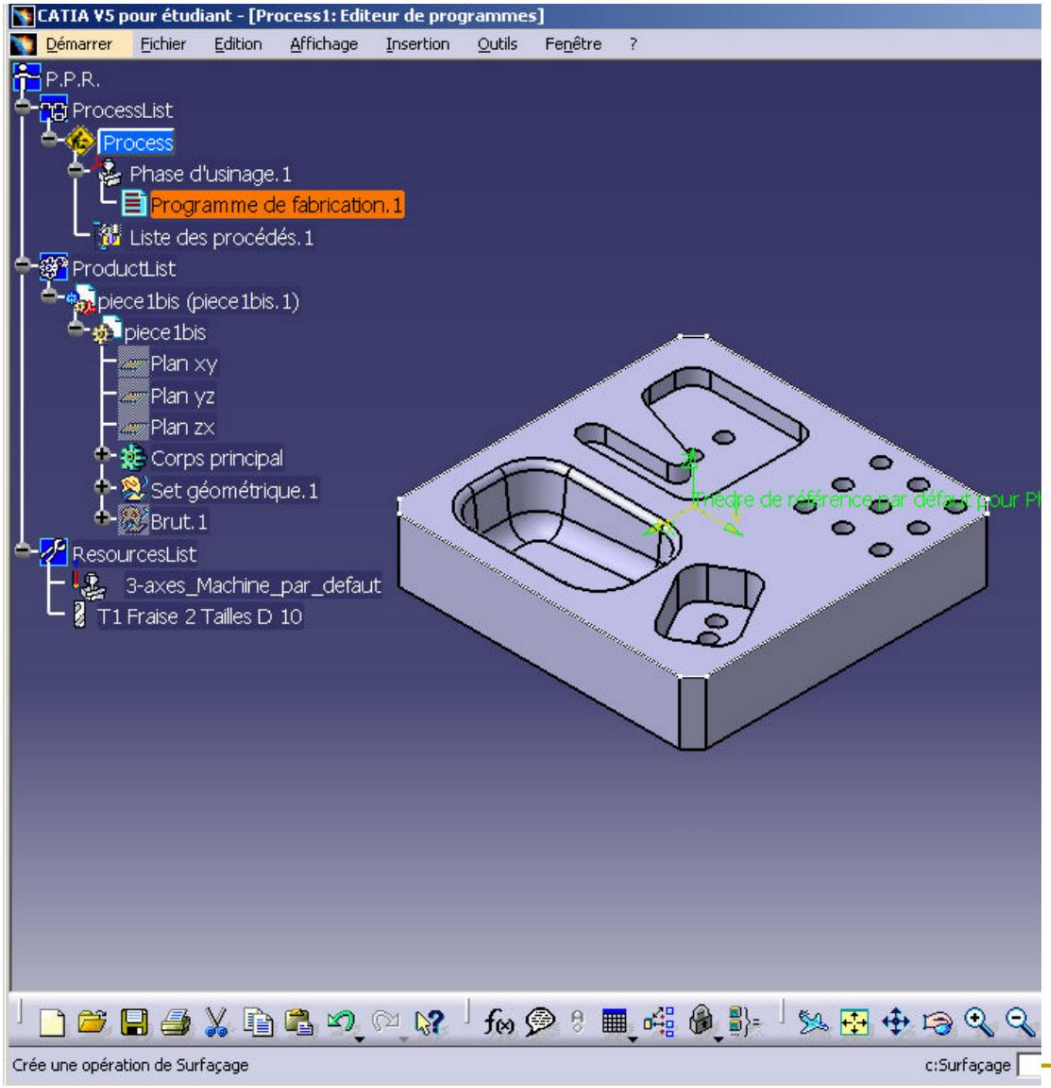
The generation of trajectories starts here.

Reminder : For each of these operations, the programming mode is identical. It consists of explaining to CATIA: • which tool to place in the spindle nose, • which part of the part to machine, • how to machine the surface (defining an operating strategy, ie a type of trajectory, or even engagement conditions , ie axial and radial depth of cut), • what cutting and feed rates to use, • how to get to and from the surface to be machined.

• Reminder: Operations are created after the entity highlighted in the tree PPR, the easiest way being therefore not to select anything else in the PPR and to generate the operations in the order of their realization

Save your work regularly

Surfacing



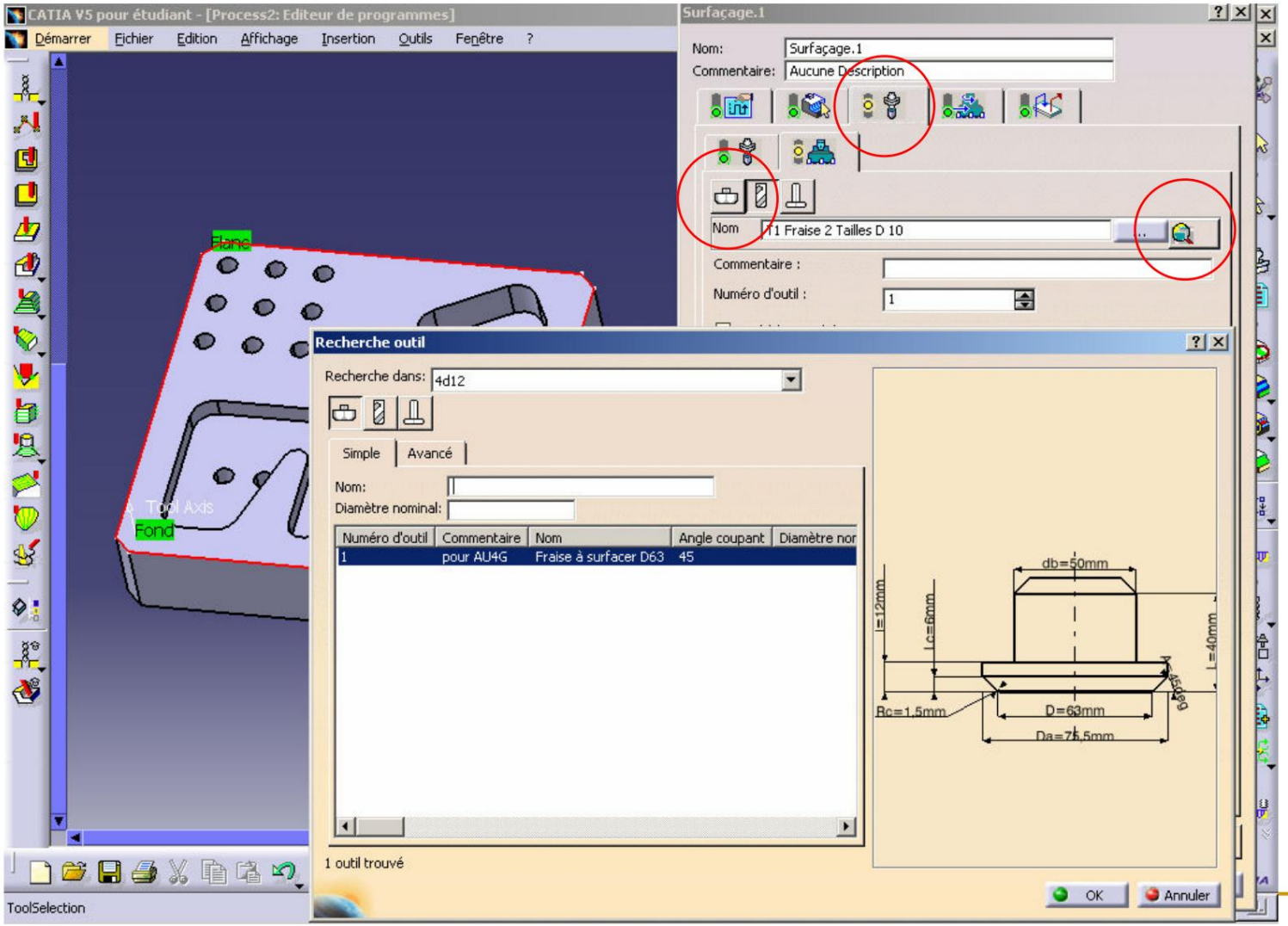
Highlight Manufacturing Schedule

click on the surfacing icon

Select the top surface of the part

Save your work regularly

Surfacing



Select the Ø63 mm face milling cutter.

Check the compensation, the tool number and the presence of cutting conditions (implied not zero)

Save your work regularly

Surfacing

Choose a round-trip machining mode (tab on the far left)

Choose Passing ratio.

Choose a radial engagement equal to 80% of the tool diameter.

Choose an End pass outside the room.

Click on the question marks to know the meaning of these parameters.

Save your work regularly

Surfacing

Set Quality to Finish (fourth tab)

Click on the tool path animation icon.

Visualize the trajectory of the face milling cutter.

Click OK twice and check for Solved in the PPR tree.

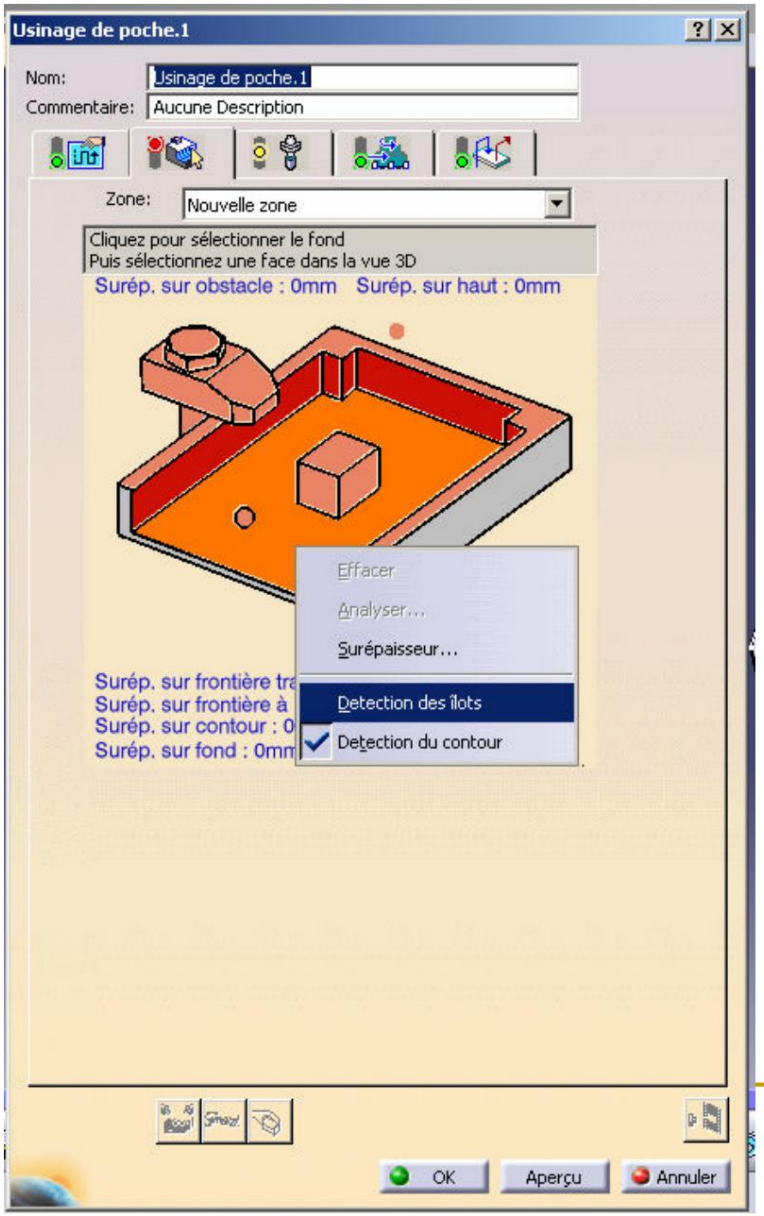
Pocket machining

Click on the Pocket Machining icon.

Select one of the straight pockets

The tool to be used is the 10 mm diameter 2-size milling cutter

Pocket machining



Position the pointer over the image.

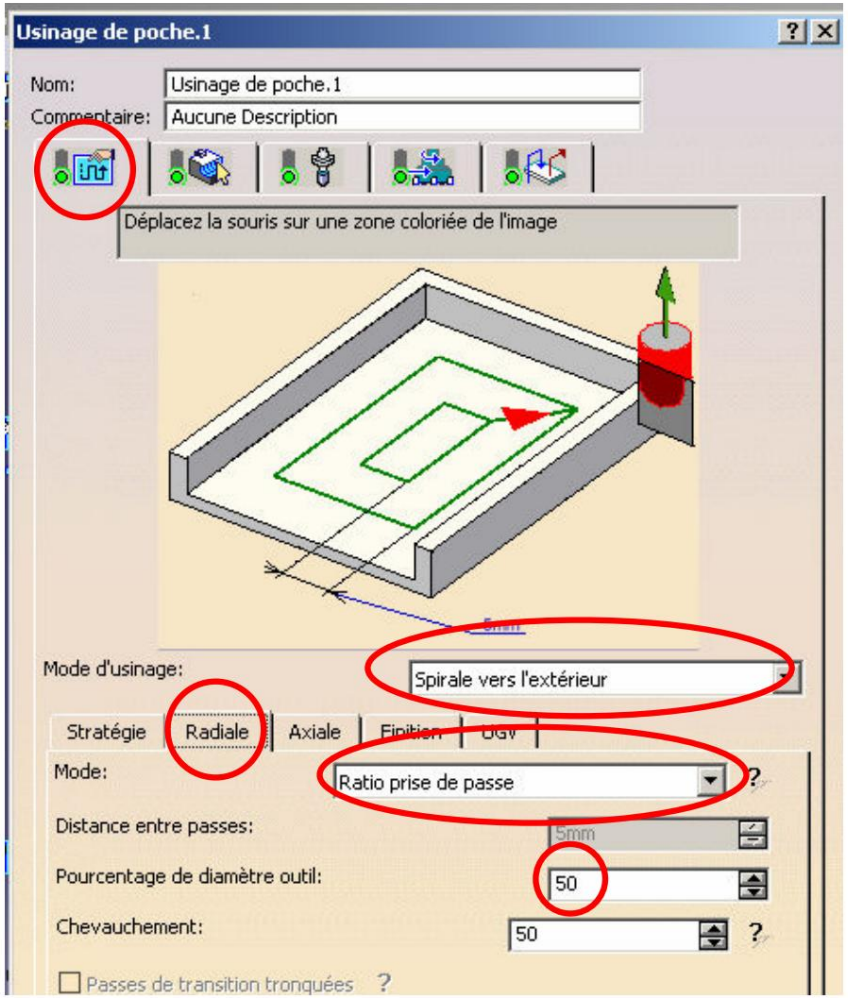
Click with the right mouse button.
Disable Island Detection.

Enable Edge Detection.

The non-detection of the islands makes it possible to machine the pockets above the holes.
Click with the left button.

Save your work regularly

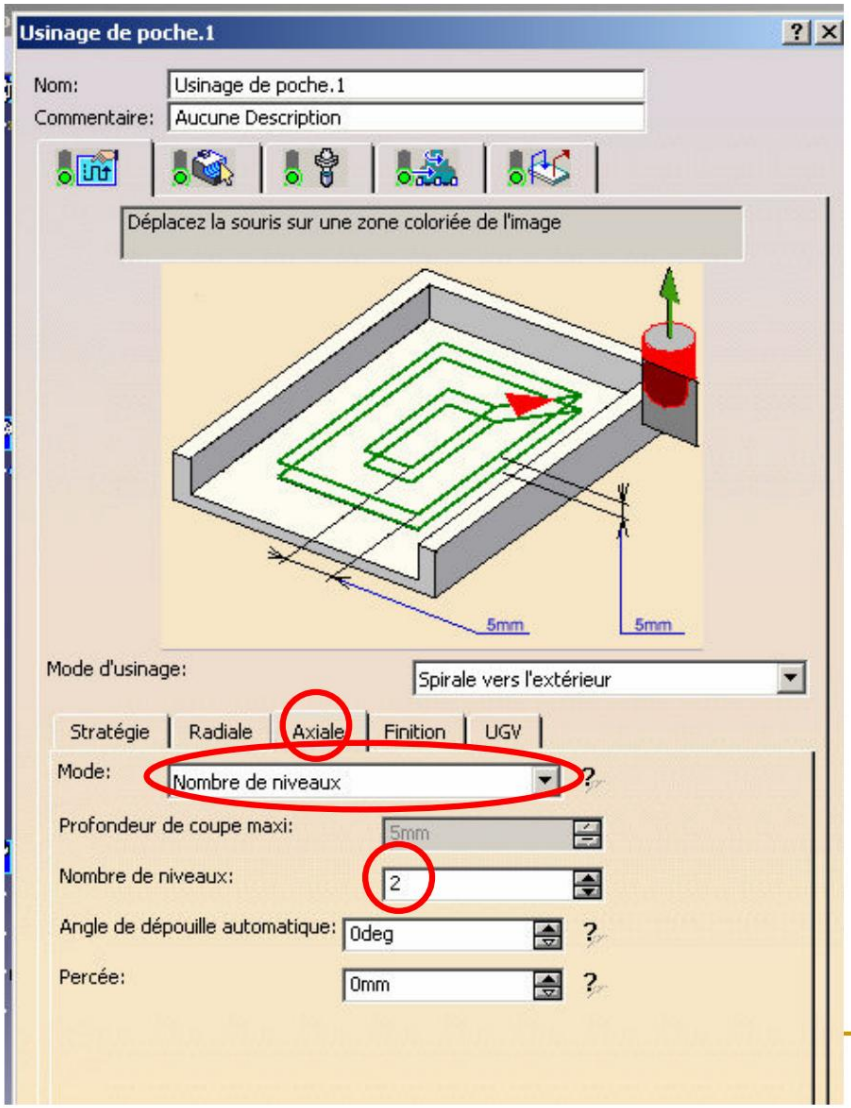
Pocket machining



Click on the Machining mode tab.
Select Outward Spiral.
Click on the Radial tab.
Select Passing Ratio.
Set tool diameter percentage to 50%

Save your work regularly

Pocket machining



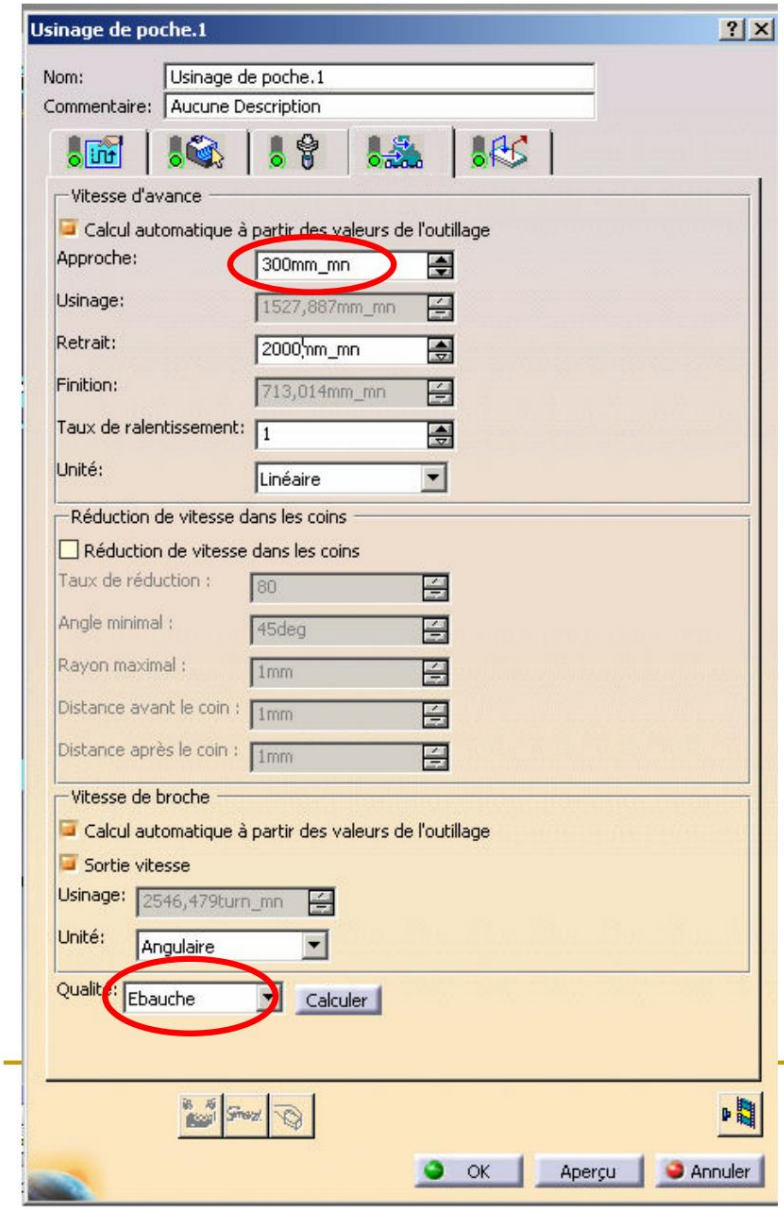
Click on the Axial tab.
Select Number of Levels.

Set the number of levels to 2.

Pocket depth is 10 mm.

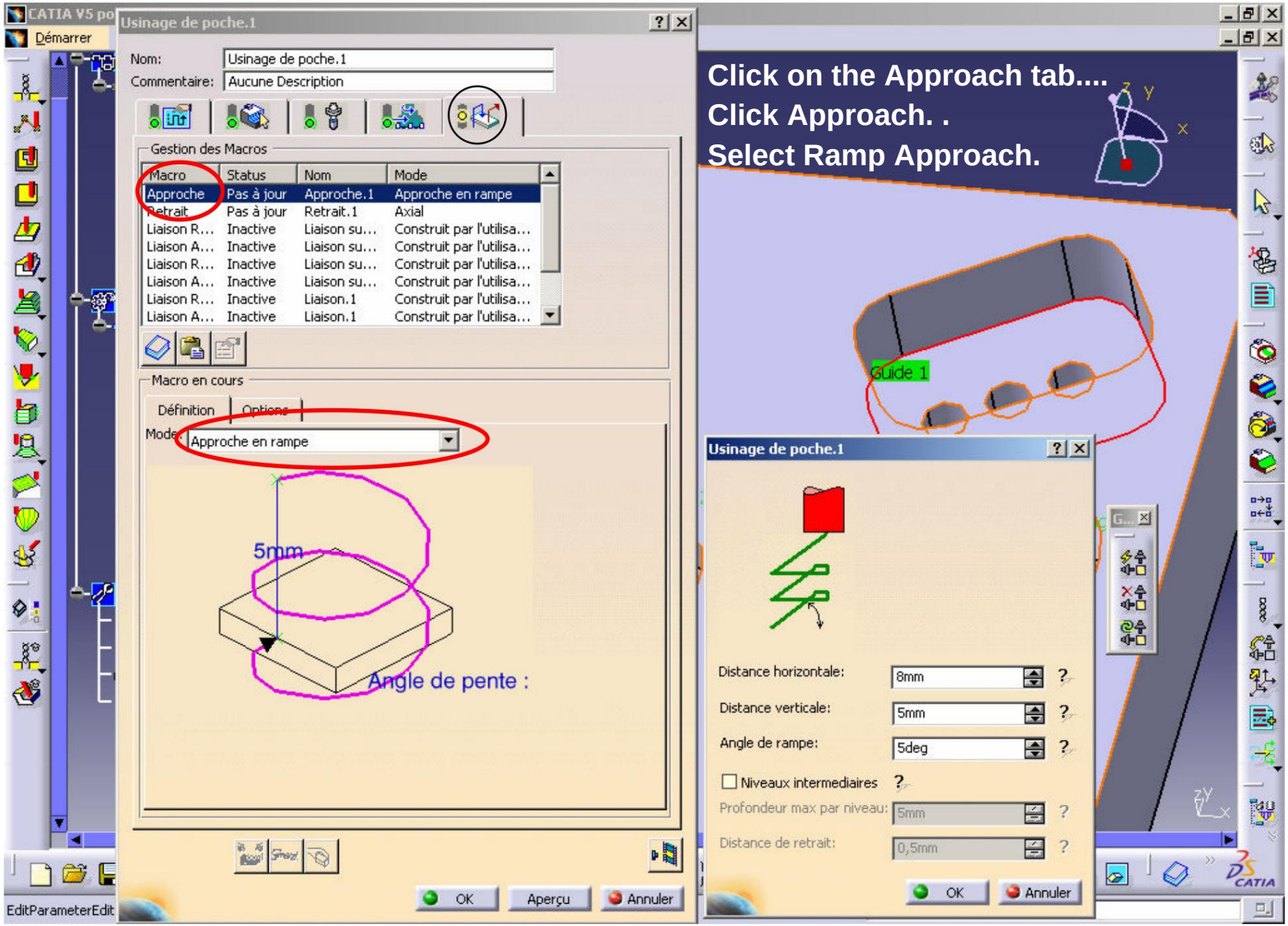
The axial depth of cut is therefore 5 mm.

Save your work regularly

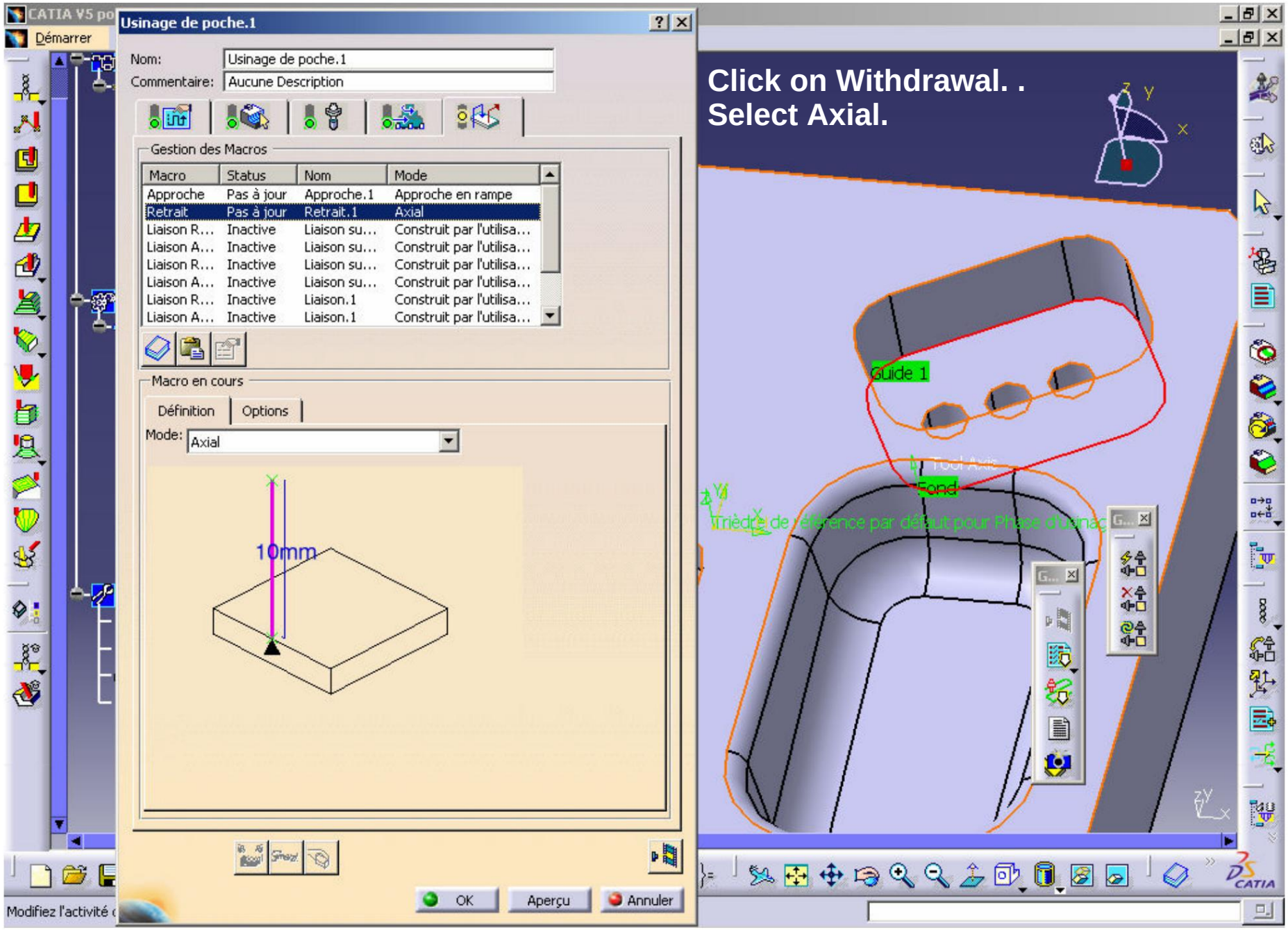


Click on the Speeds tab.
Check the withdrawal speed (about 2000 mm/min).
Set Quality to Finish.

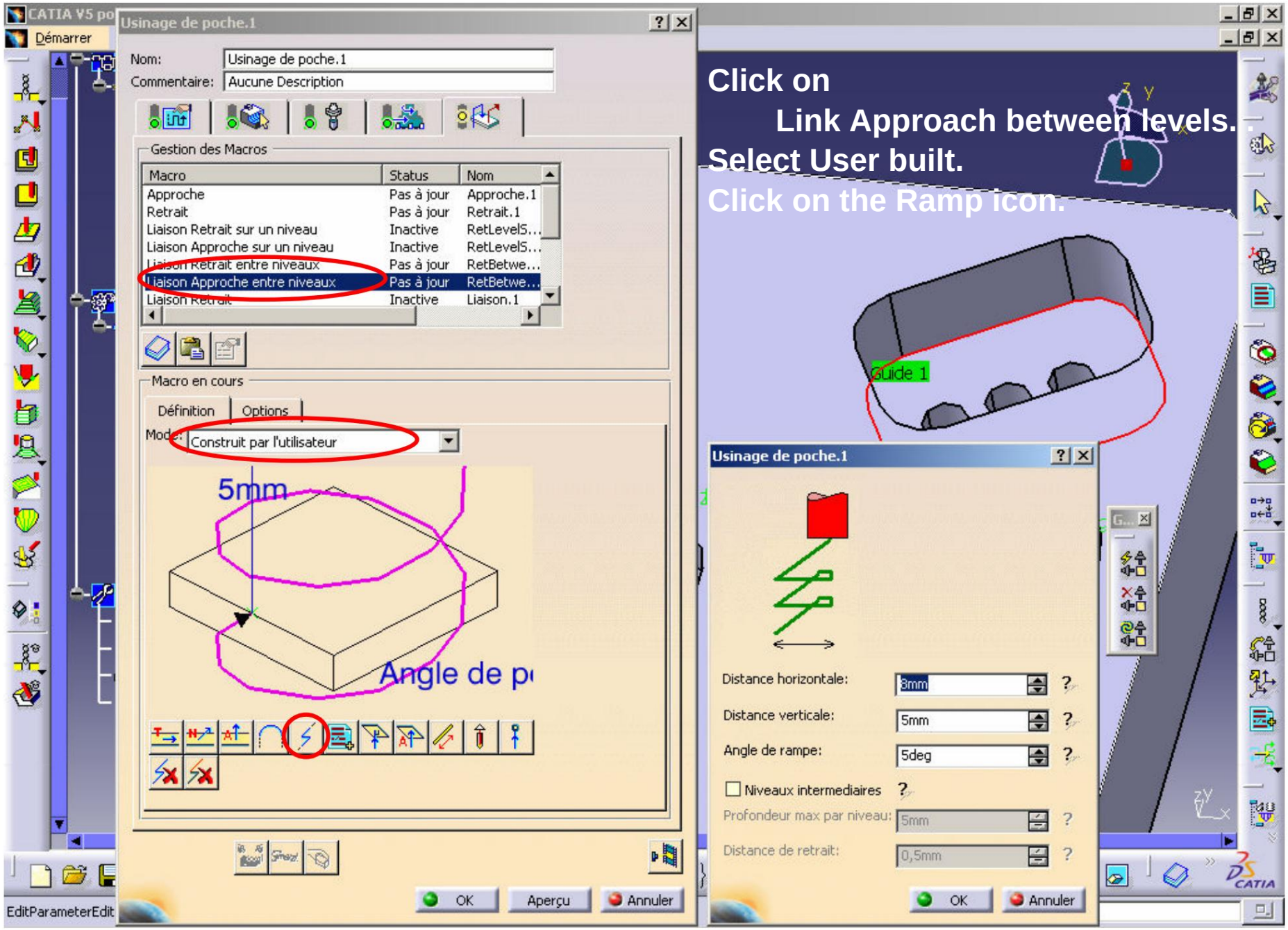
Save your work regularly



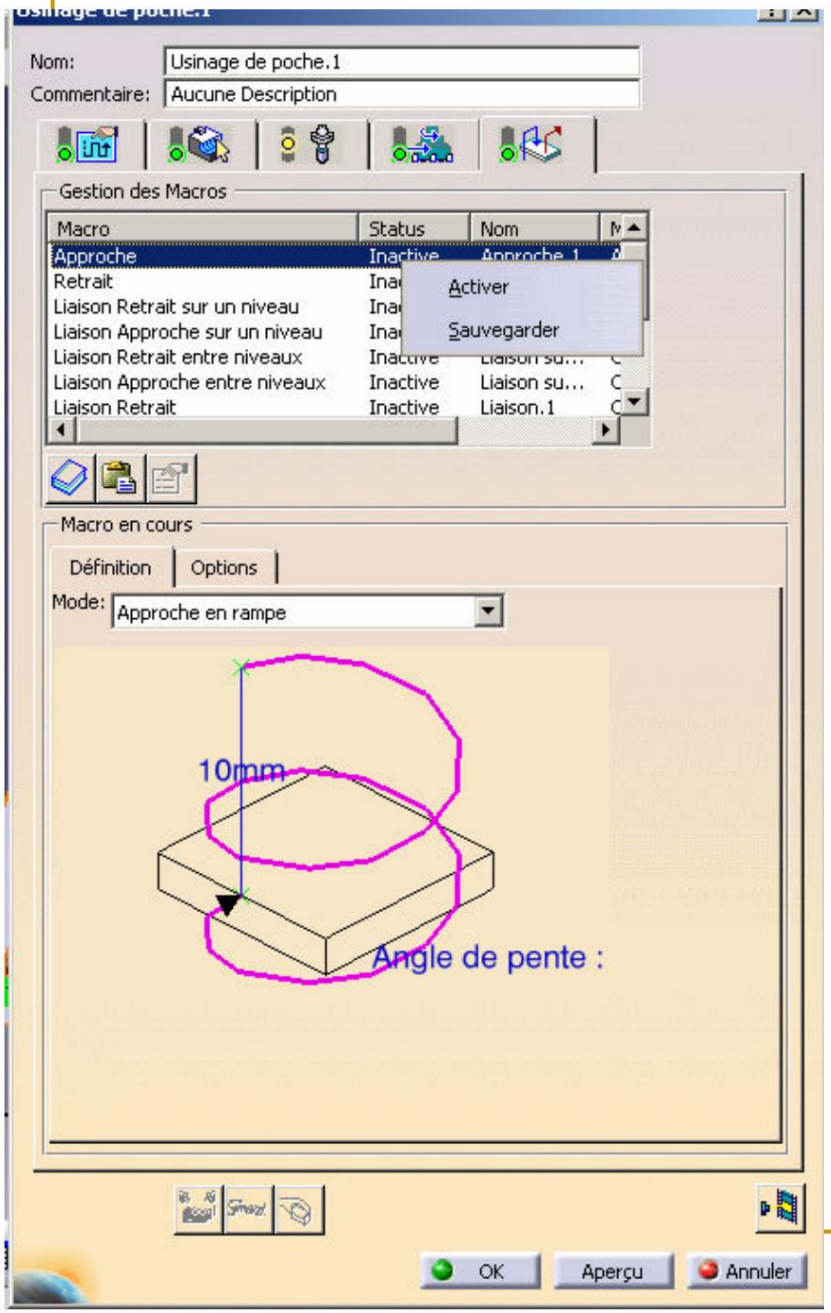
Save your work regularly



Save your work regularly



Save your work regularly



The taking into account of these withdrawal and approach trajectories requires their activation.
Click on the Status of these movements.
Press the right mouse button.
Click Activate.

Click on the tool path animation icon to view the programmed paths.
Click OK if satisfactory.

Do the same for the second right pocket.

The shallowest one is not fully machined because the diameter of the cutter used is too large. To remedy this, we will create a recovery zone.
The choice of a range by decreasing tool diameter having been made, this step will be done after the machining of the stripped pocket.

Save your work regularly

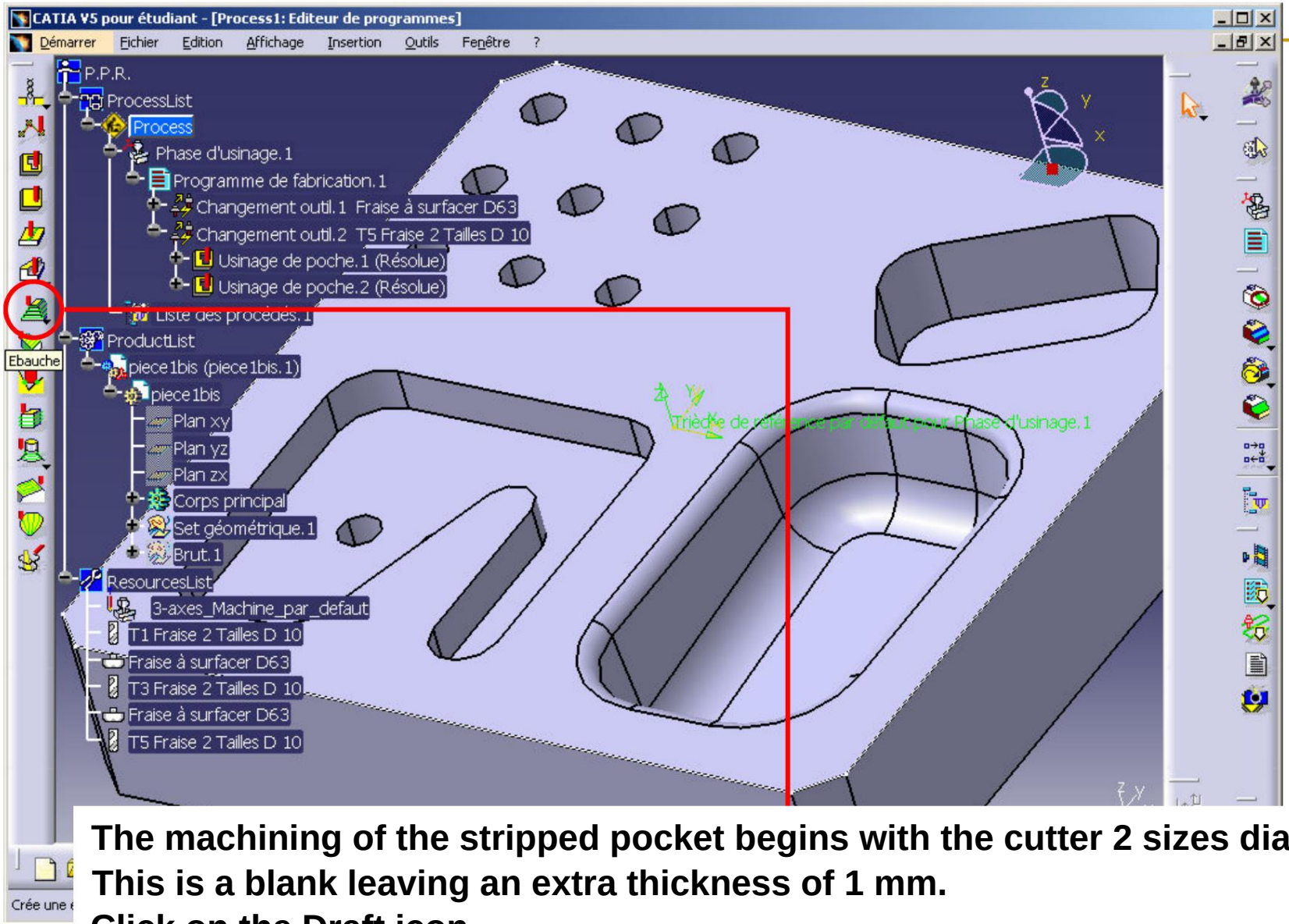
Stripped pocket machining

Sketch

Semi-finishing

Finishing

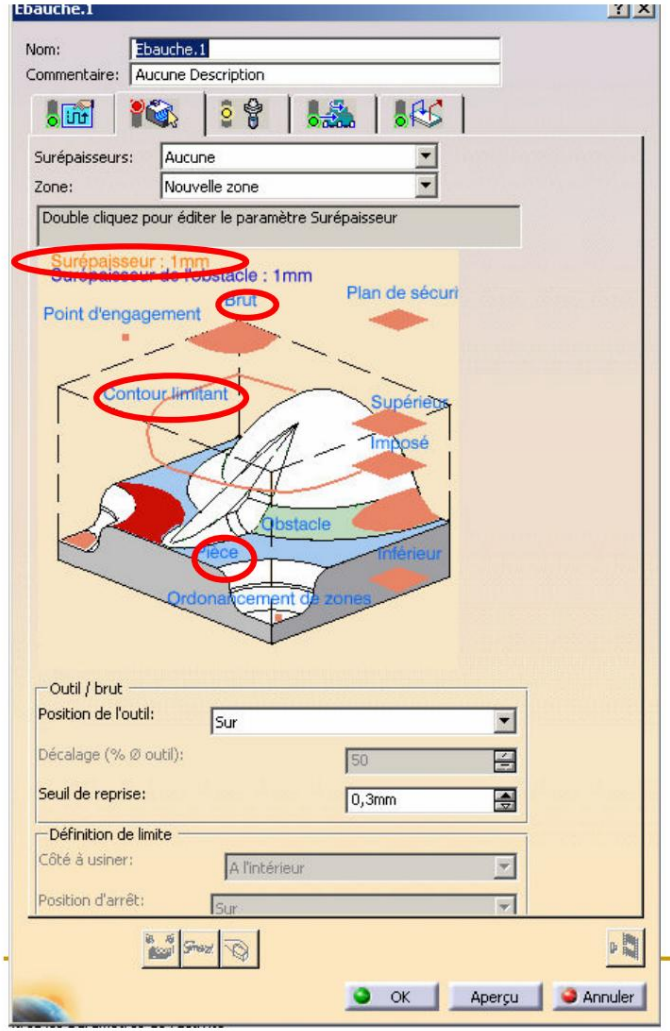
Save your work regularly



The machining of the stripped pocket begins with the cutter 2 sizes diameter 10 mm.
This is a blank leaving an extra thickness of 1 mm.
Click on the Draft icon.

Save your work regularly

Stripped pocket machining



Double-click Stock.
Set the Stock allowance to 1 mm.

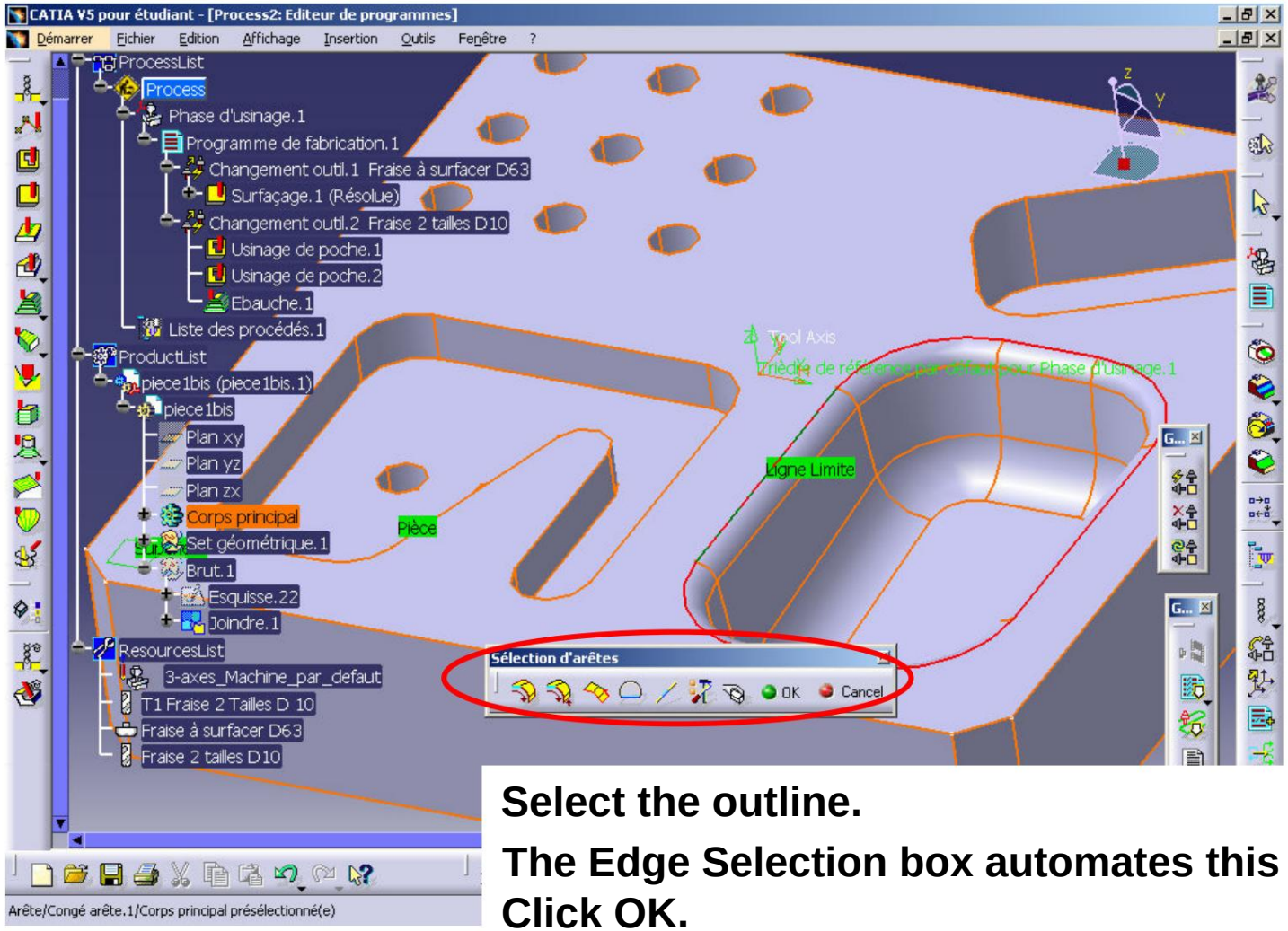
Click on Gross.
Indicate the Brut.1 item in the construction tree.

Click on Room.
Indicate the Main Body item in the construction tree.

Click on Limiting contour.
The limiting contour is the intersection of the surfaces of the drafted pocket with the top face of the part.

Save your work regularly

Stripped pocket machining

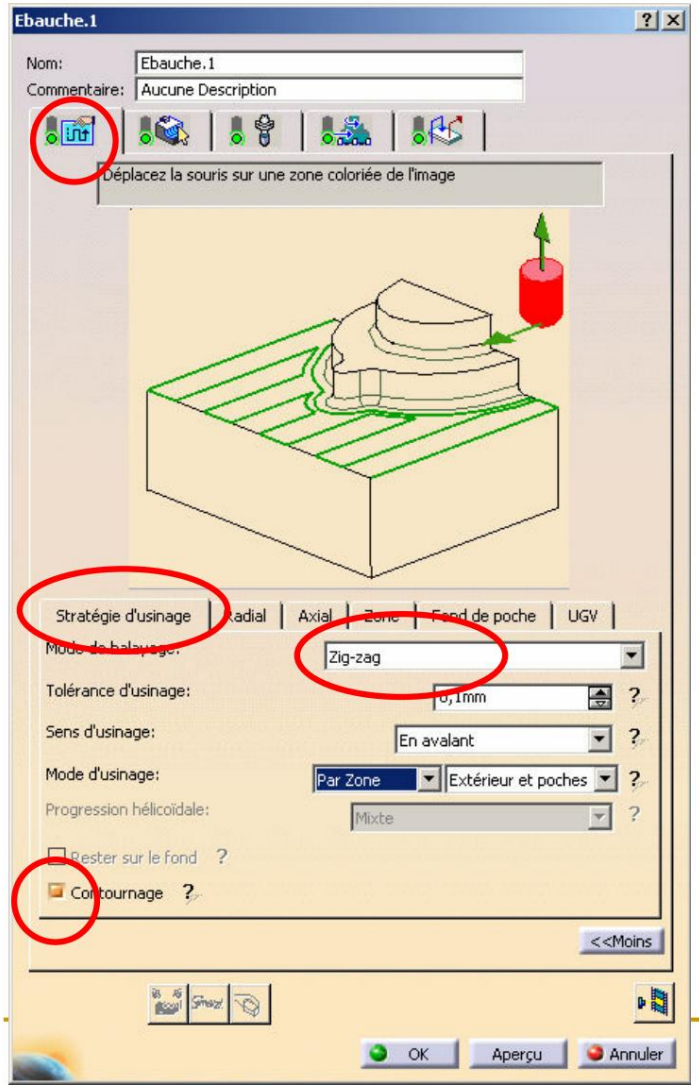


Select the outline.

The Edge Selection box automates this operation.
Click OK.

Save your work regularly

Stripped pocket machining

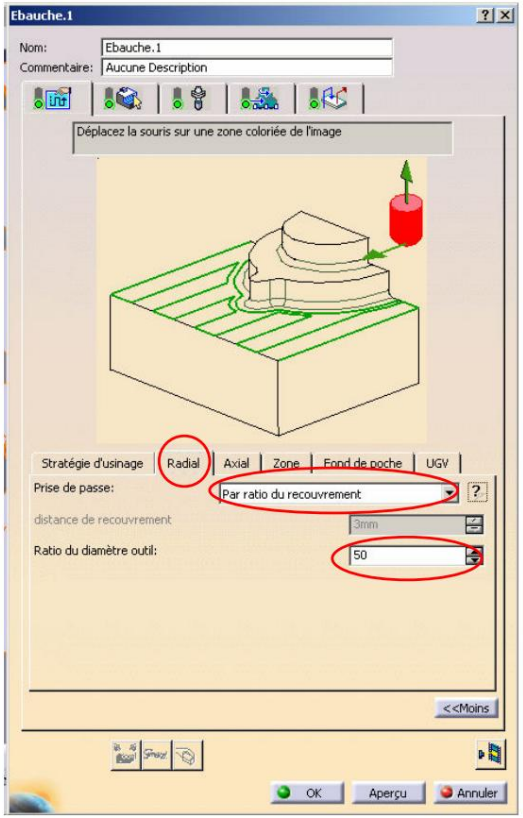


Click on the Machining mode tab.
Click on the Machining strategy tab.
Choose a Zig-Zag Scan Mode.
Check the Contour box.

Click on the question marks to understand each of the functions offered

Save your work regularly

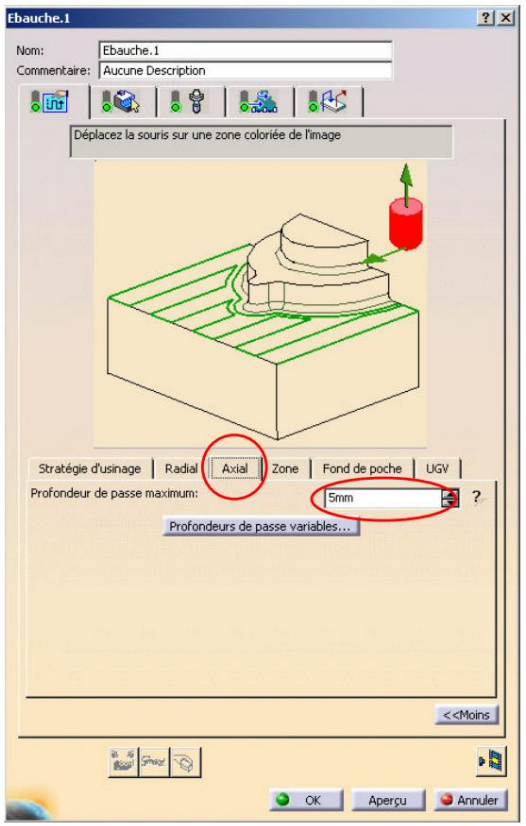
Stripped pocket machining



Click on the Radial tab.

Select By Recovery Ratio.

Set the engagement width to 50% of the tool diameter.

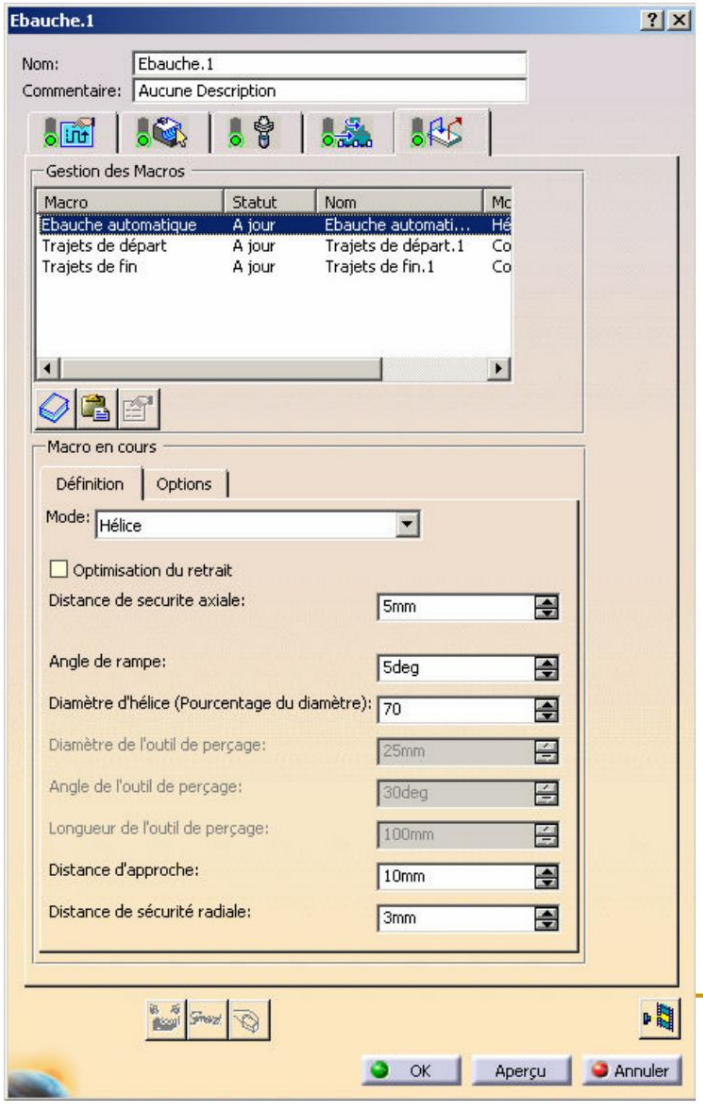


Click on the Axial tab.

Set Maximum Depth of Cut to 2 mm.

Save your work regularly

Stripped pocket machining



Click on the Approach tab....
Click Approach.
Select Propeller.

Activate trajectories if necessary.

Click on the tool path animation icon to view the programmed paths.
Click OK.

Save your work regularly

Stripped pocket machining

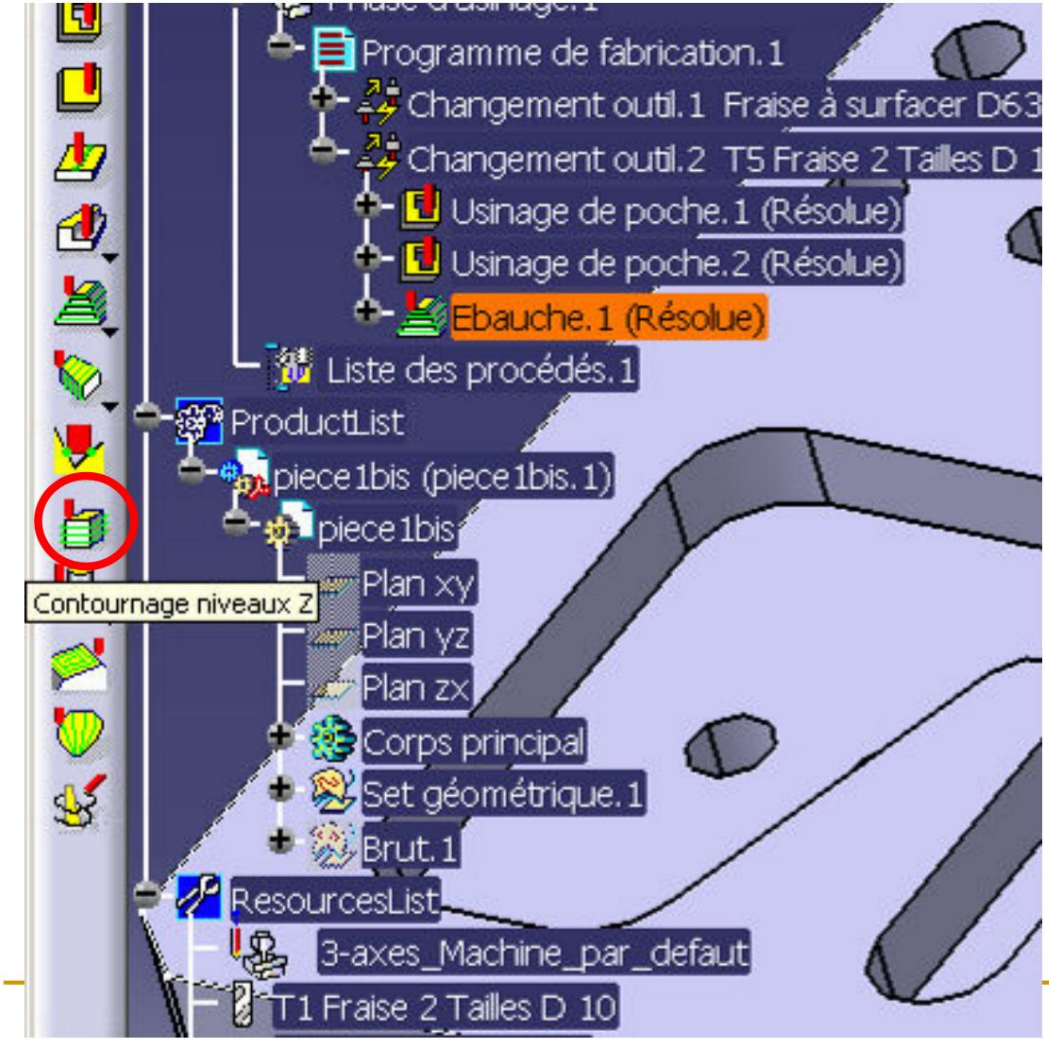
The draft is made,

It is now a question of making the semi-finishing and the finishing

Z-level contouring is used. The cutter

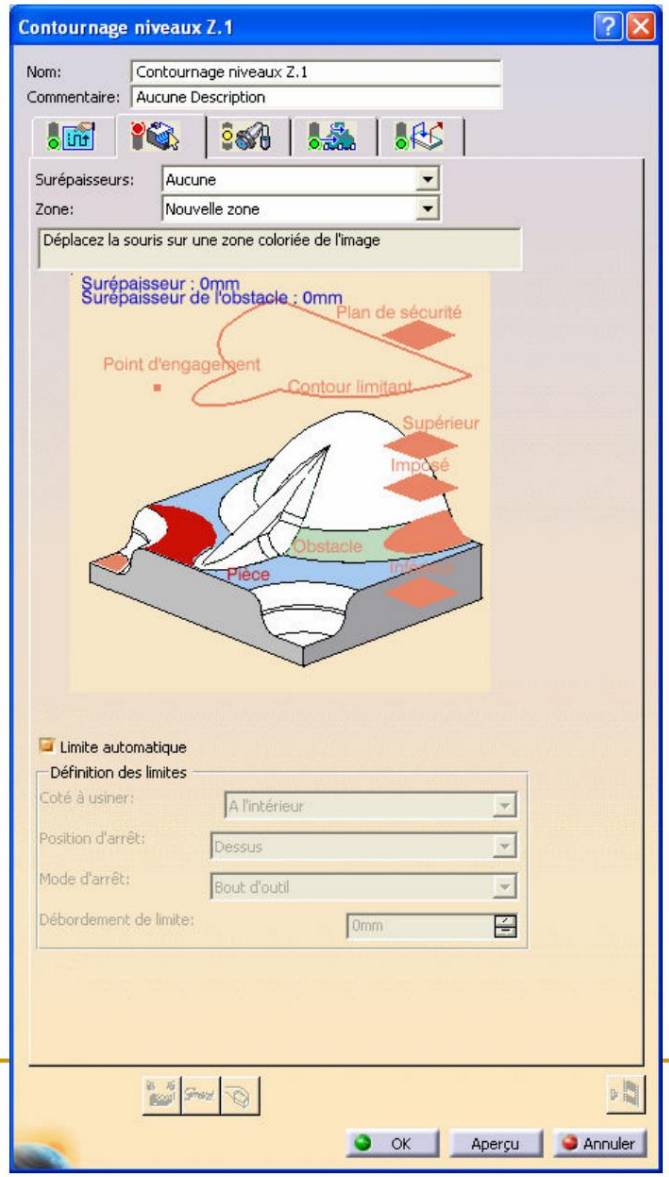
used is the straight cutter (as opposed to conical) with a hemispherical end Ø8.

Stripped pocket machining



Save your work regularly

Stripped pocket machining

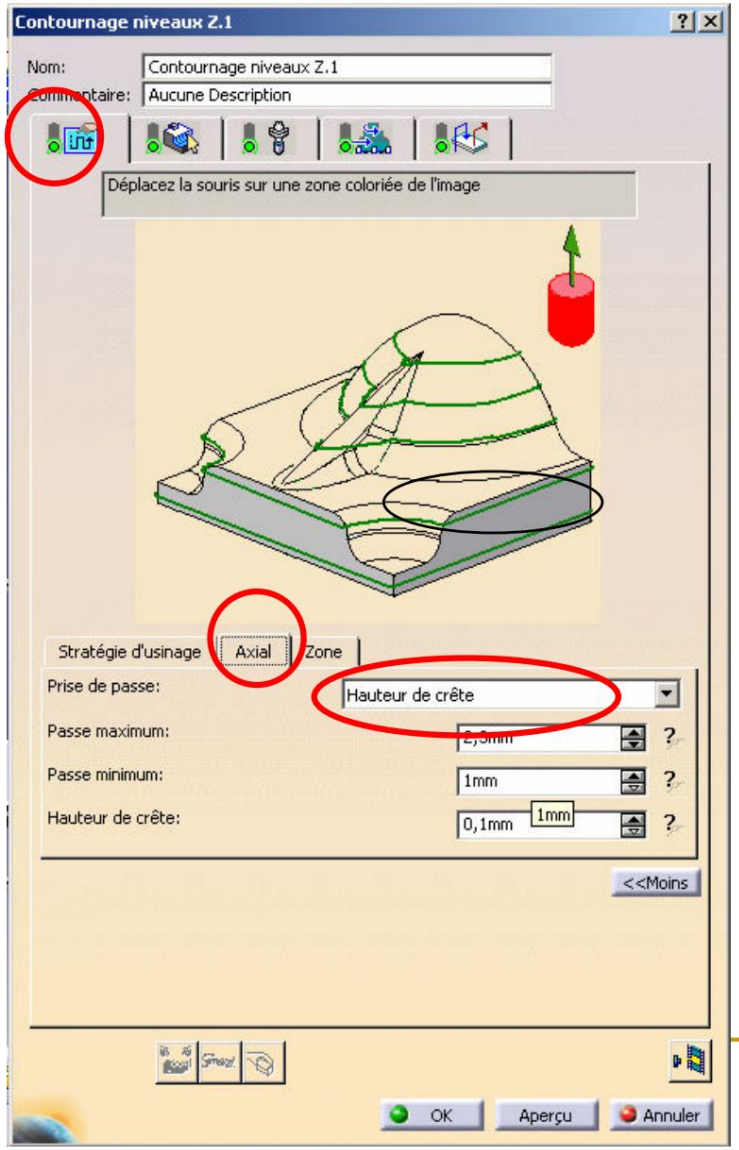


Set Stock to 0.5mm Set Part from Construction Tree.
Define the Limiting contour as for Roughing.

Select the ball end mill, check offsets and cutting conditions

Save your work regularly

Stripped pocket machining

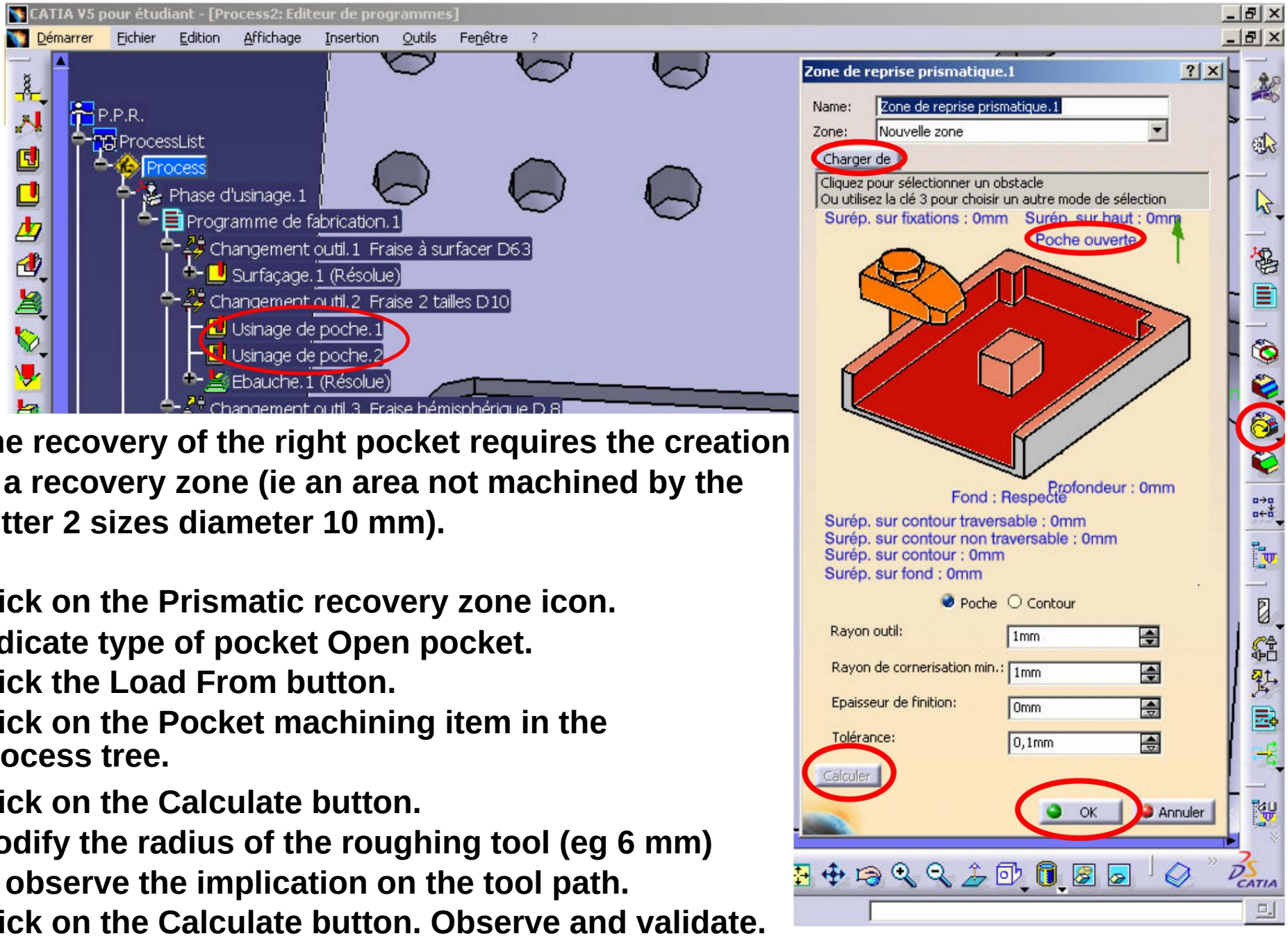


Click on the Machining mode tab.
Click on the Axial tab.
Choose to calculate the Pick-up from a Peak Height value.
Indicate 0.1 mm as the value to be respected.

Click on the tool path animation icon to view the programmed paths.
Click OK if satisfactory.

Proceed in the same way for the finish.
The Machining tolerance changes to 0.01 mm.
The Oversize is 0 mm.
The Peak Height changes to 0.01 mm.

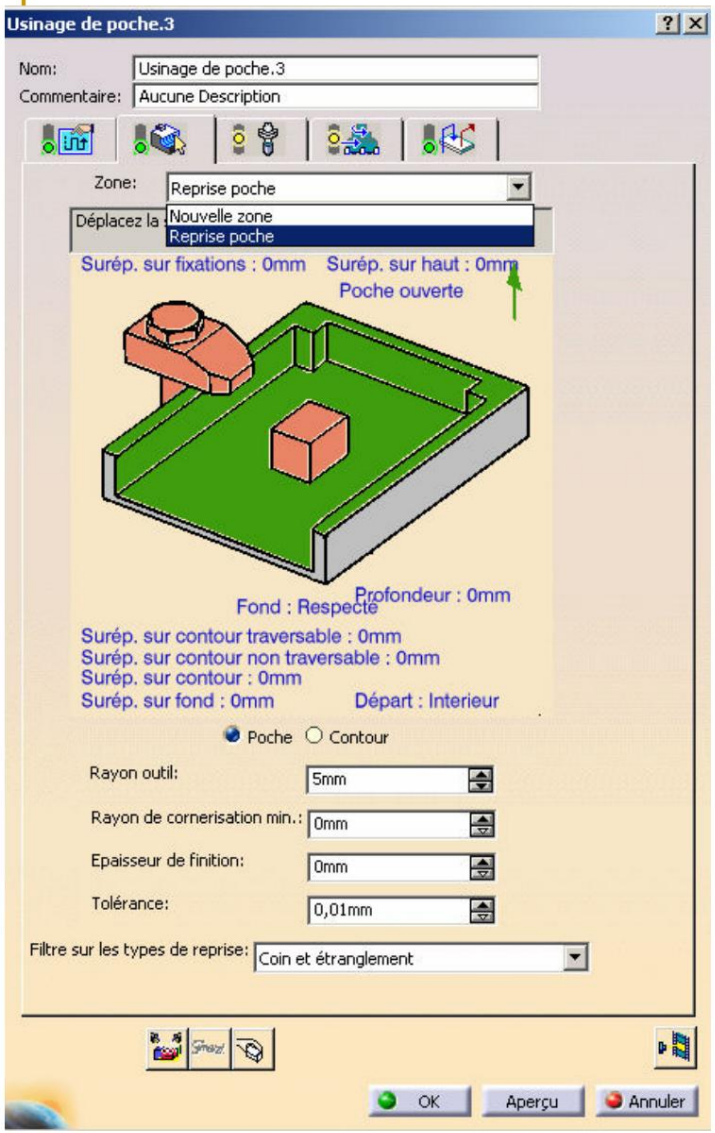
Save your work regularly



The recovery of the right pocket requires the creation of a recovery zone (ie an area not machined by the cutter 2 sizes diameter 10 mm).

Click on the Prismatic recovery zone icon.
Indicate type of pocket Open pocket.
Click the Load From button.
Click on the Pocket machining item in the Process tree.
Click on the Calculate button.
Modify the radius of the roughing tool (eg 6 mm) to observe the implication on the tool path.
Click on the Calculate button. Observe and validate.

Save your work regularly



The recovery zone is created.

Click on the Pocket Machining icon.

Expand the Zone list.

Select the recovery zone created.

Click on the Tool tab.

Select the cutter 2 sizes diameter 4 mm.

Modify the corrector (corrector ID and corrector number change to 13).

Click on the Machining mode tab.

Click on the Radial tab.

Set the engagement width to 50% of the tool diameter.

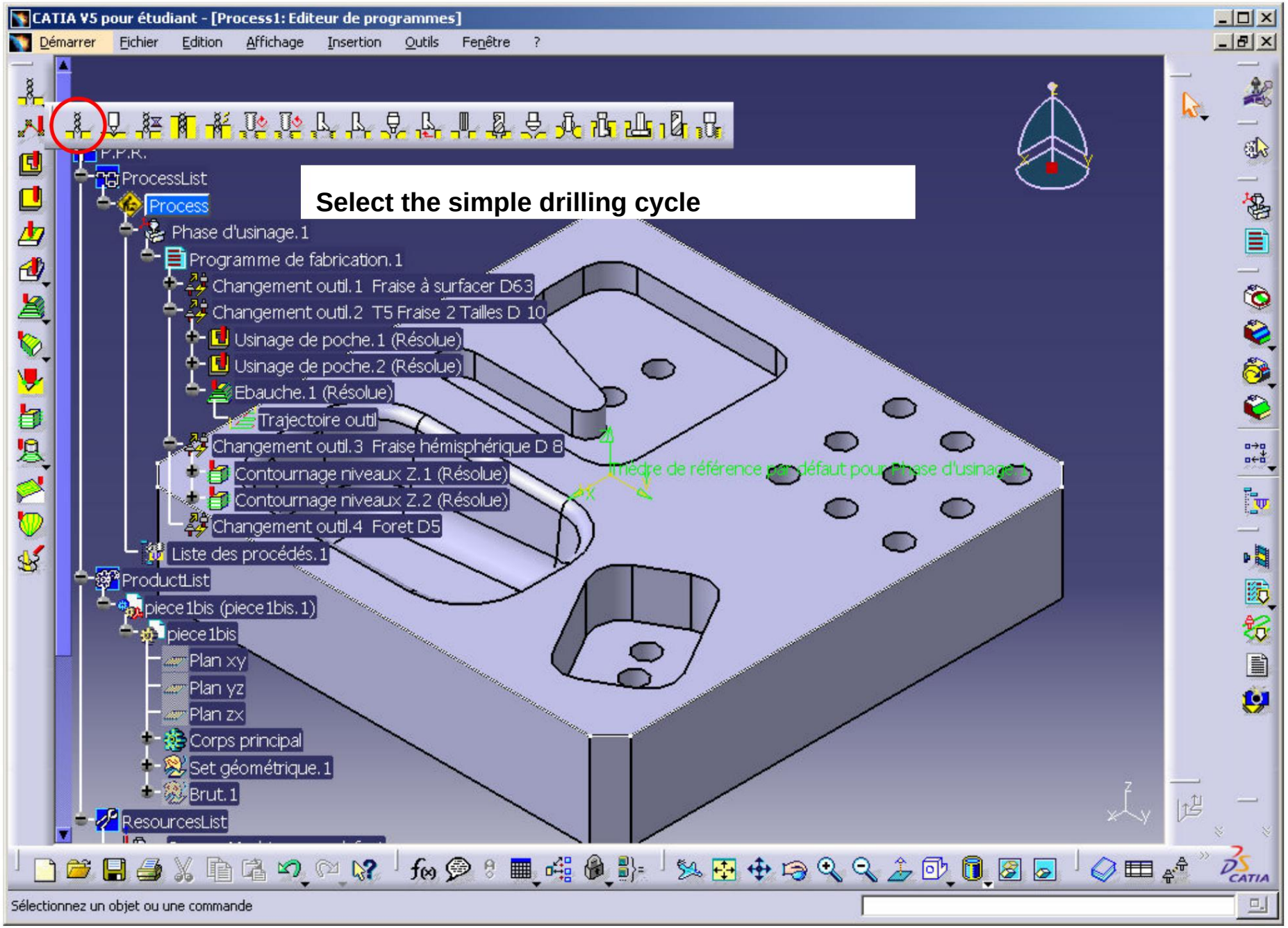
Click on the Axial tab.

Set the number of levels to 2.

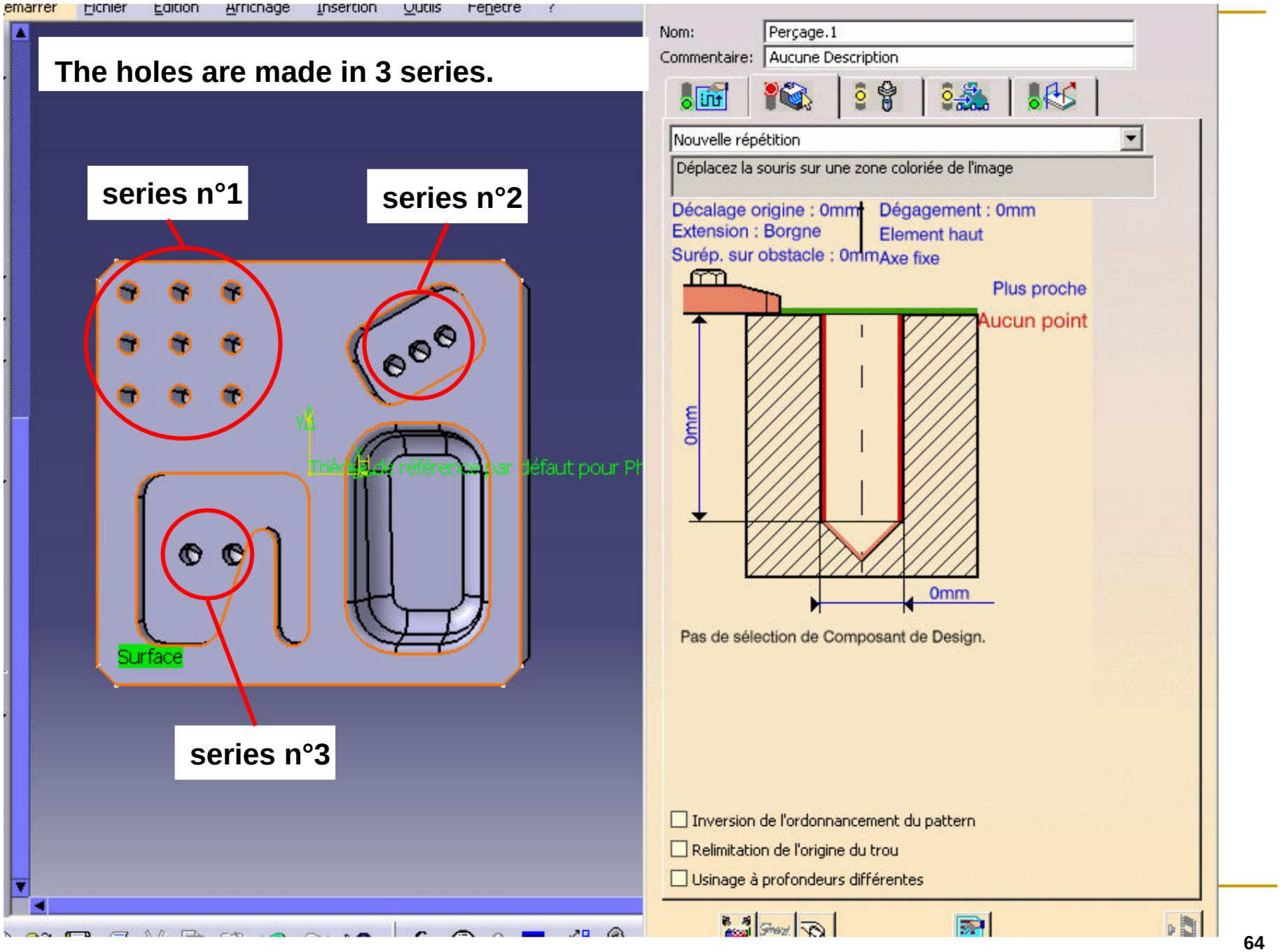
Check speeds and docking and retrieval conditions.

Click on the tool path animation icon to view the programmed paths.

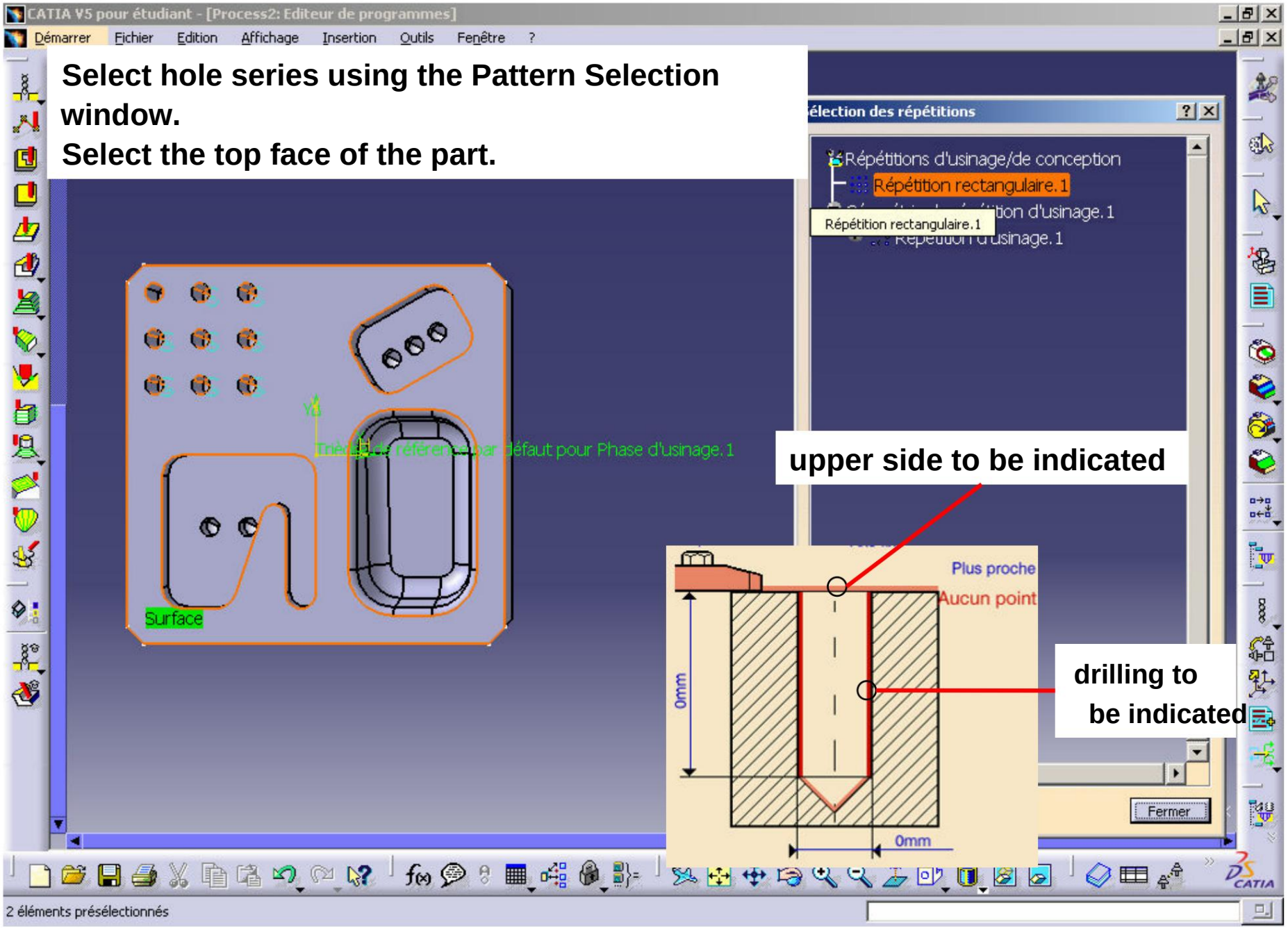
Save your work regularly



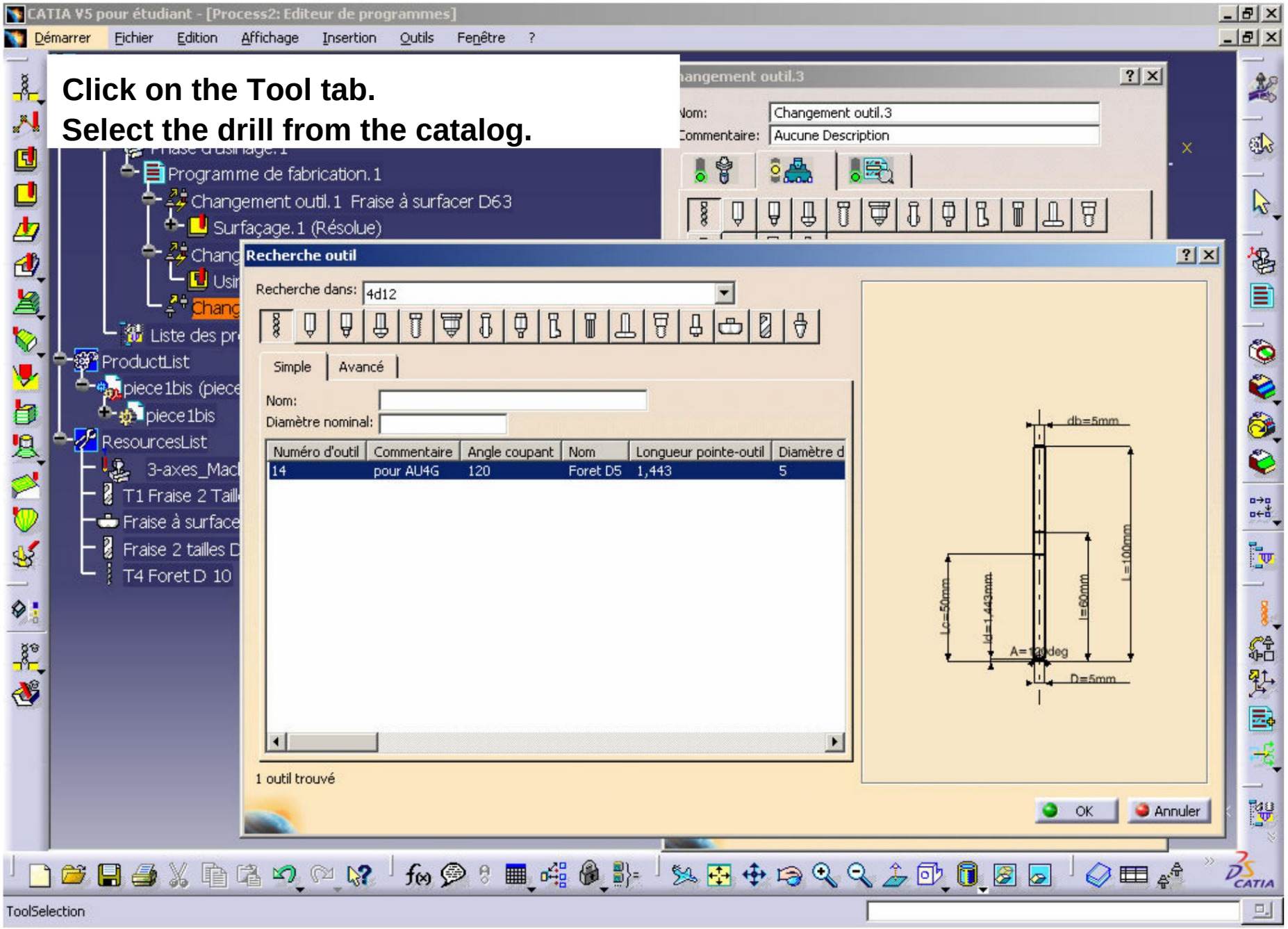
Save your work regularly



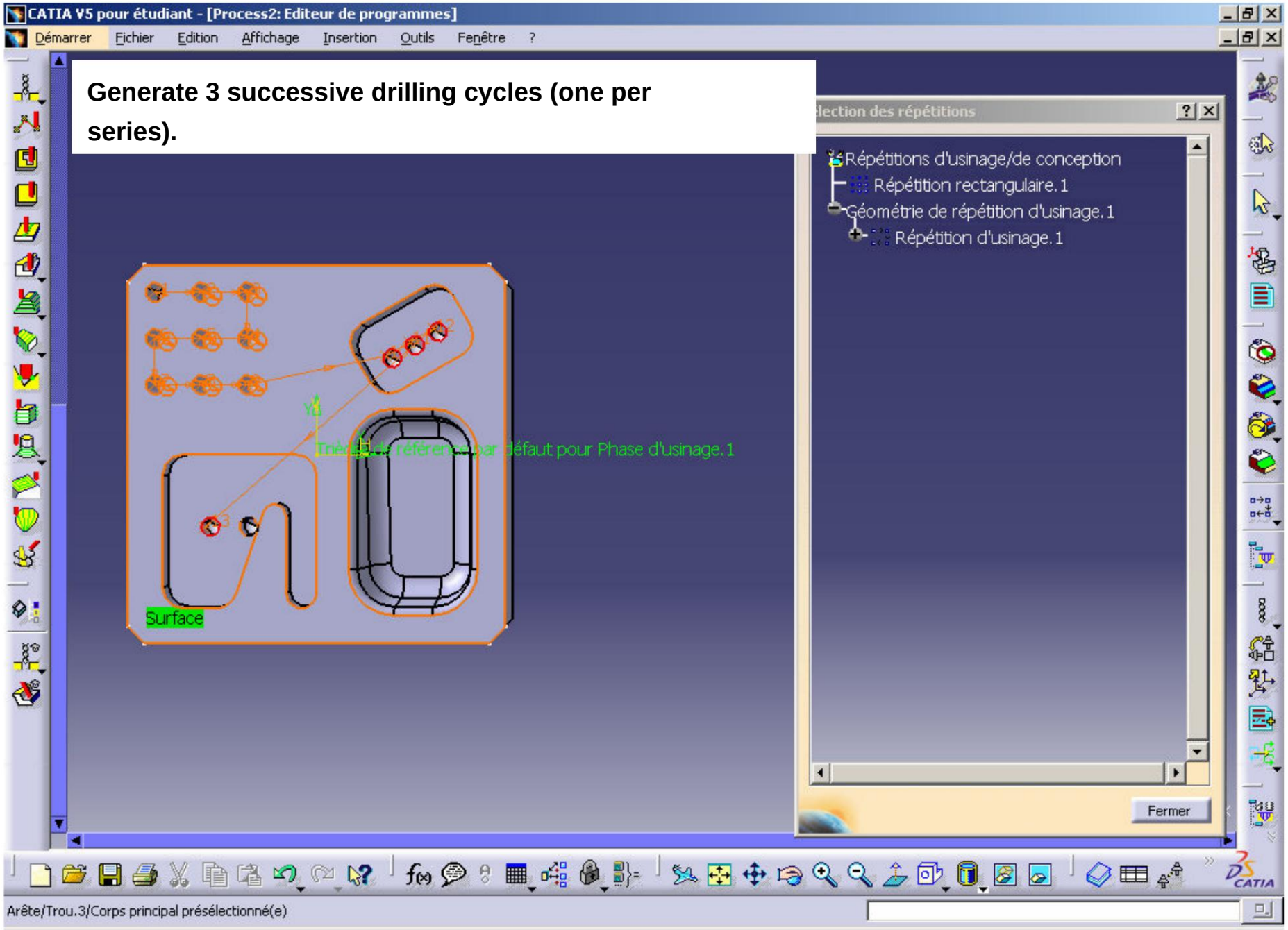
Save your work regularly



Save your work regularly



Save your work regularly



Save your work regularly

Drilling cycles

Click on the tool path animation icon to view the programmed paths.

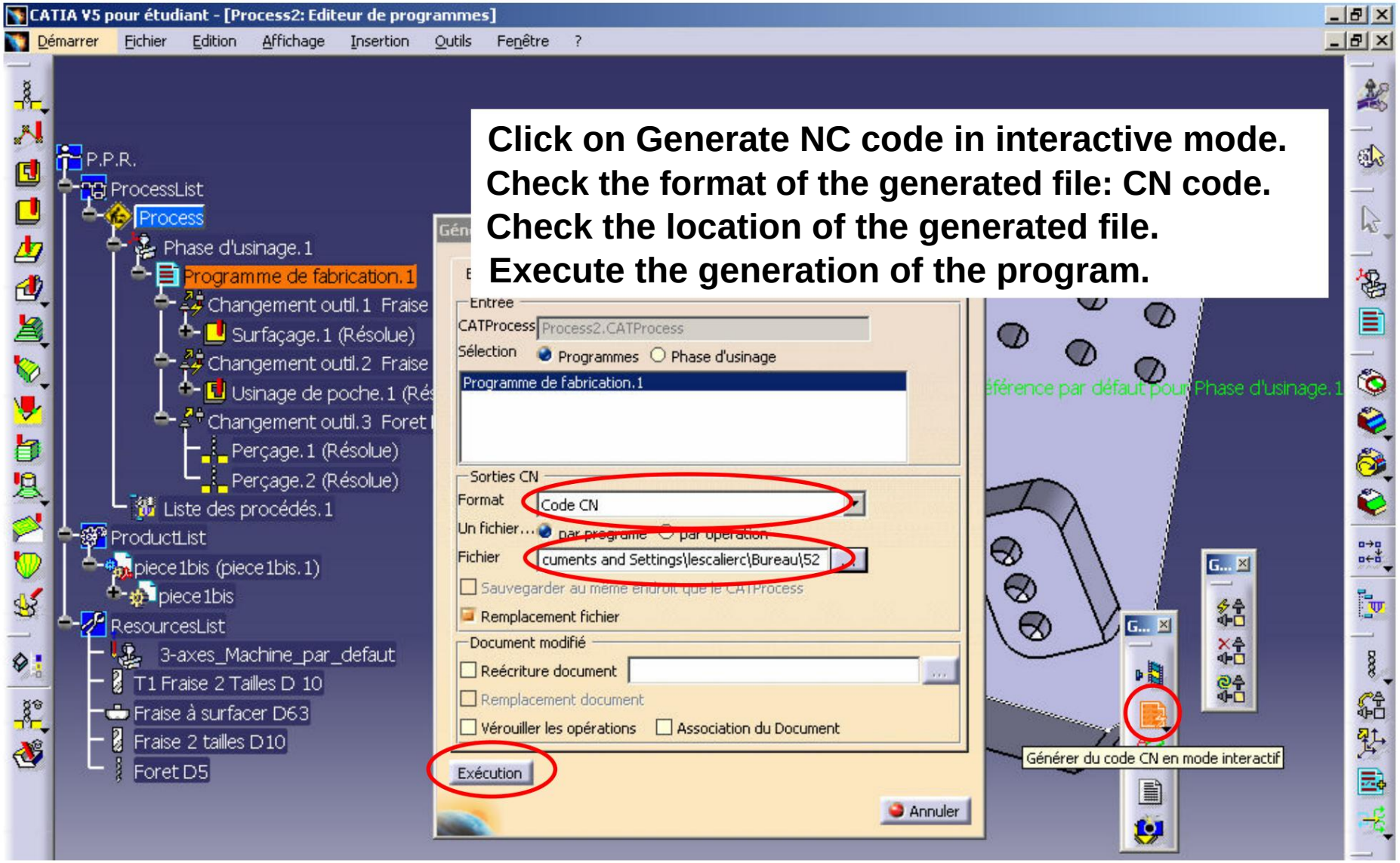
If the drill collides with the part in its path between two consecutive holes, add an axial retraction and/or an axial approach as before.

Click on OK when the trajectory is satisfactory.

The last step is the generation of the NC program

Save your work regularly

NC program generation



Save your work regularly

End of TP2

To remember :

A recovery zone can be created to machine the same pocket with two tools, optimizing the removal of material

We can define the approach and withdrawal of the tools according to the sequence of operations carried out by the same tool

The machining strategy and the number of passes is optimized according to a shape criterion such as ridge height

Needless to insist on the fact that the choice of the values of these criteria must also be chosen according to the type of operation carried out and the precision sought (roughing/ finishing).

Save your work regularly

Practical work 3 and 3': Shooting

Turning is not the primary focus of CAD/CAM given the apparent simplicity of tool paths. We quickly realize the difficulties of the turning workshop in CATIA.

The objectives of this practical

work are: To address the problems inherent in turning in CAM

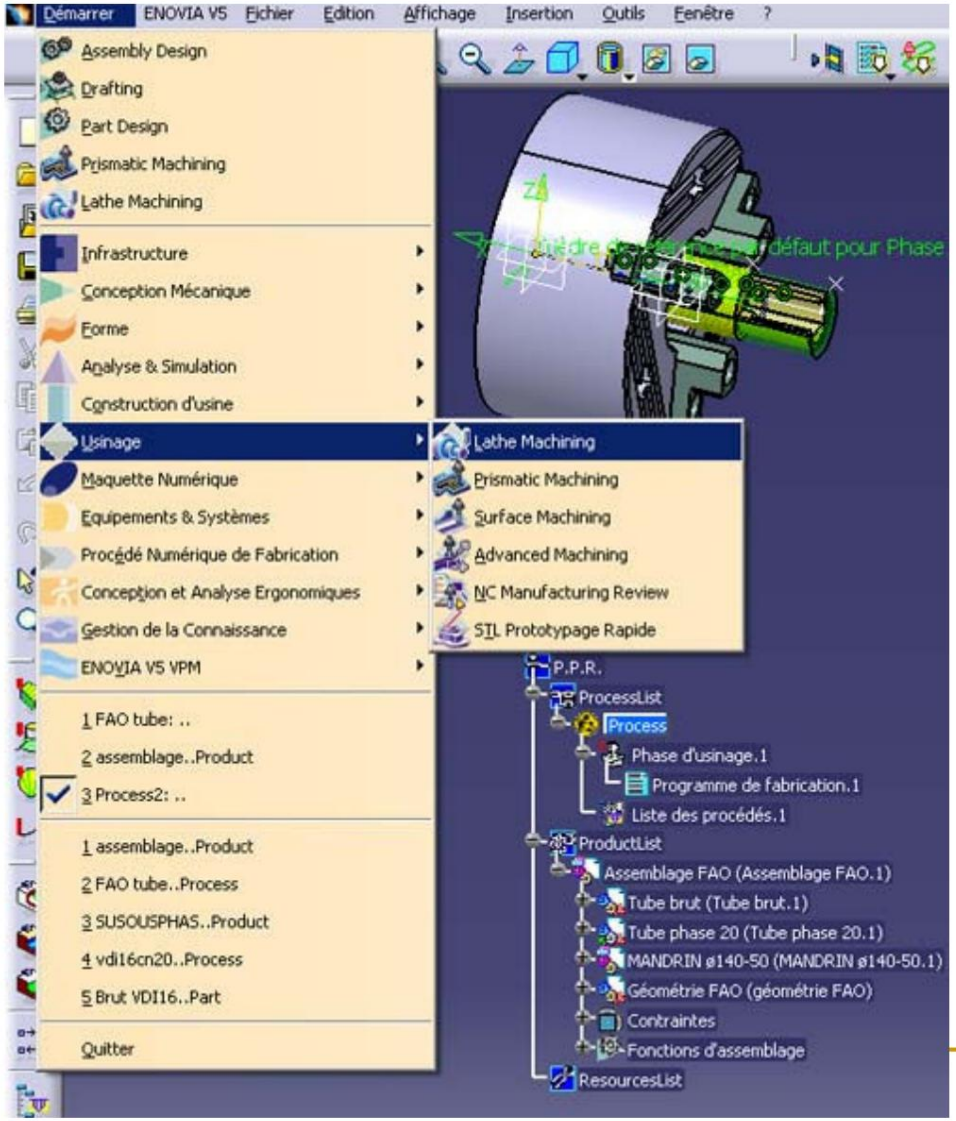
To understand the management of rough turnings

To use the main functions of CATIA corresponding to the usual operations in turning

Generate documentation at the end of the process

Save your work regularly

Preliminary step



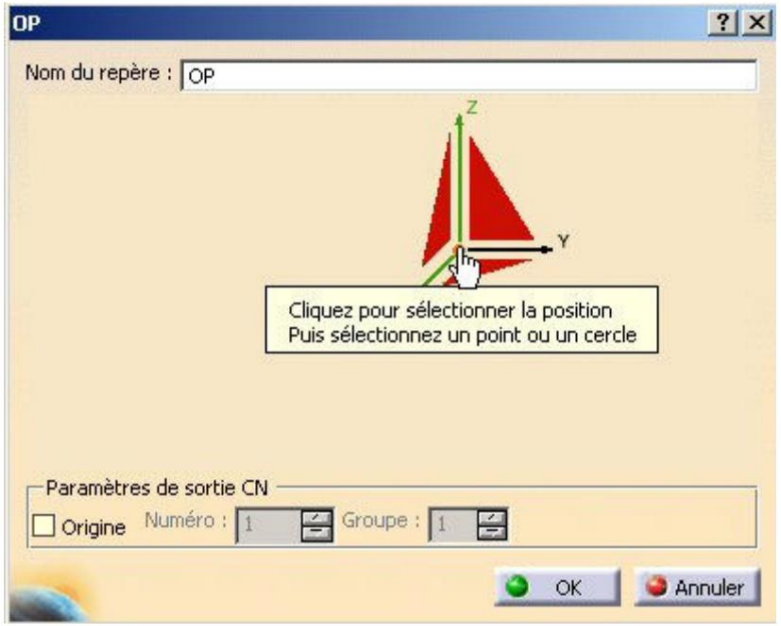
Open the ph20.CATProduct assembly file and launch the “Lathe Machining” workbench

We will use the default horizontal lathe and the “TP3_meknes” tool catalog (remember to check the post-pro, the ISO code, etc.)

Define raw and finished parts (raw tube main bodies and phase 20 tube respectively)

Save your work regularly

Preliminary step



Define the reference triad

Click on the origin then on the circular edge of the finished part (so that the origin of the coordinate system is at the center of the right face of the finished part)

If necessary, click on Z and/or X then on an edge of the part to orient the mark in accordance with the axes of the machine

Save your work regularly

The generation of trajectories starts here.

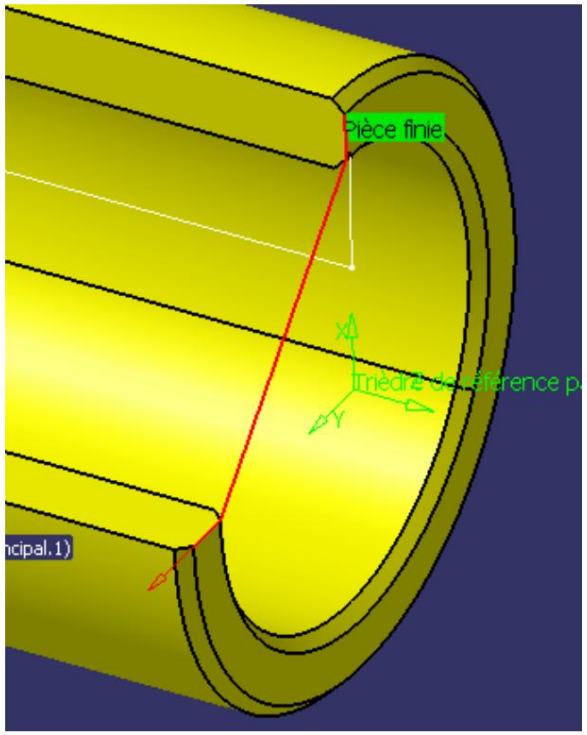
Reminder: For each of these operations, the programming mode is identical. It consists of explaining to CATIA: • which tool to place in the spindle nose, • which part of the part to machine, • how to machine the surface (defining an operating strategy, ie a type of trajectory, or even engagement conditions , ie axial and radial depth of cut), • what cutting and feed rates to use, • how to get to and from the surface to be machined.

• Reminder: Operations are created after the entity highlighted in the tree PPR, the easiest way being therefore not to select anything else in the PPR and to generate the operations in the order of their realization

In turning, the definition of the areas to be machined is done by selecting edges and not cylinders, so it is necessary (this is already done here) to cut the part. This can be done either using a pocket, a cut or more simply by generating the part in CAD by revolution of 270° instead of 360°. If the part wants to be reused for other purposes than in CAM (drawing in particular), we can set the value of the angle of revolution to go more quickly from 270 to 360°…

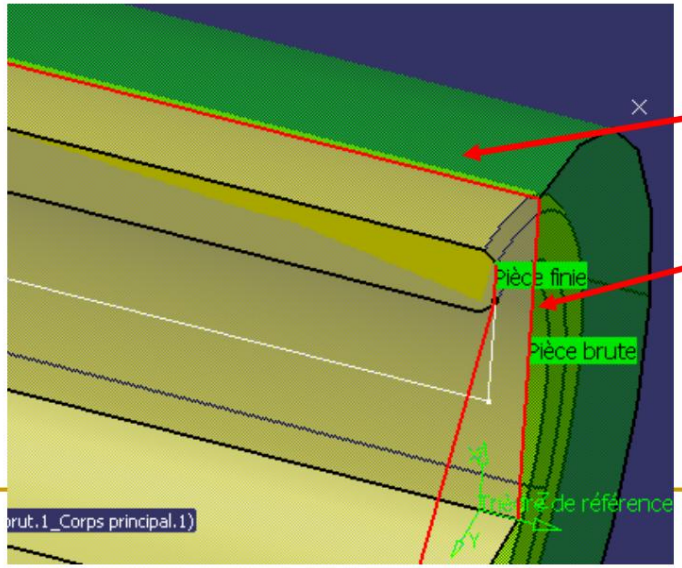
Save your work regularly

Dressage



Launch a draft shoot

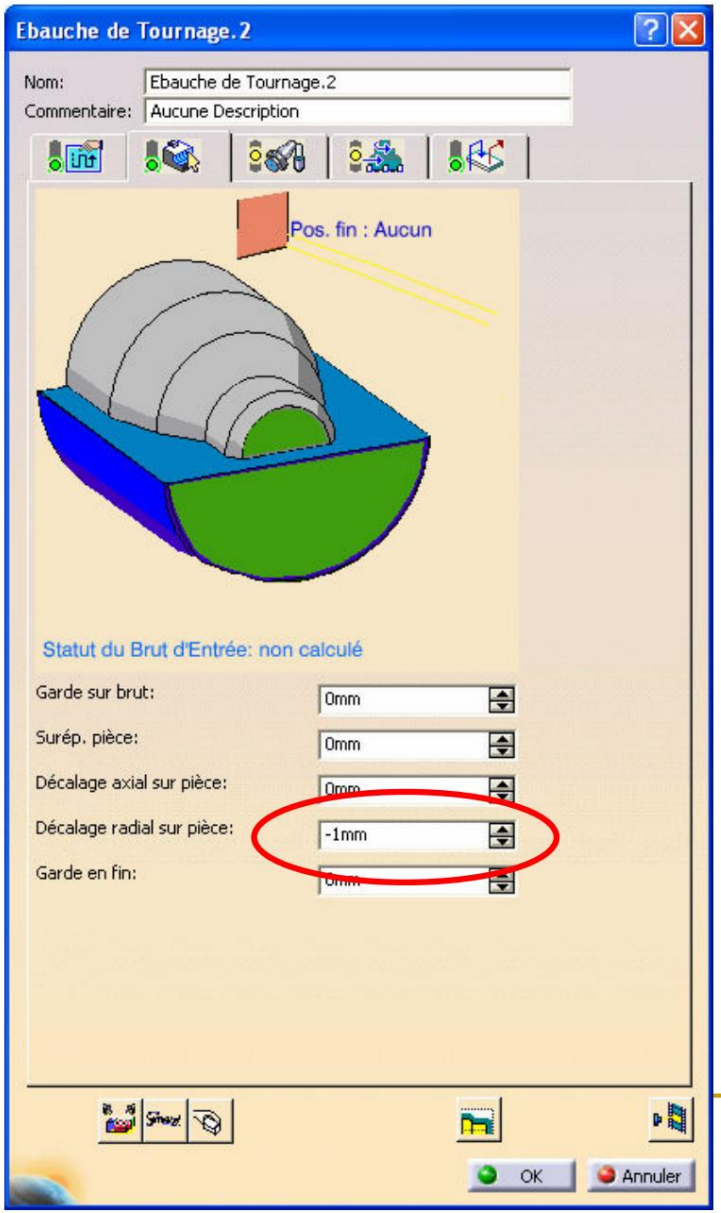
Select the two sharp edges of the part (if only one edge is selected, the tool will not go to the center for facing. This "trick" avoids creating an additional sketch which would only contain a line.



Select the two sharp edges of the stock.

Save your work regularly

Dressage



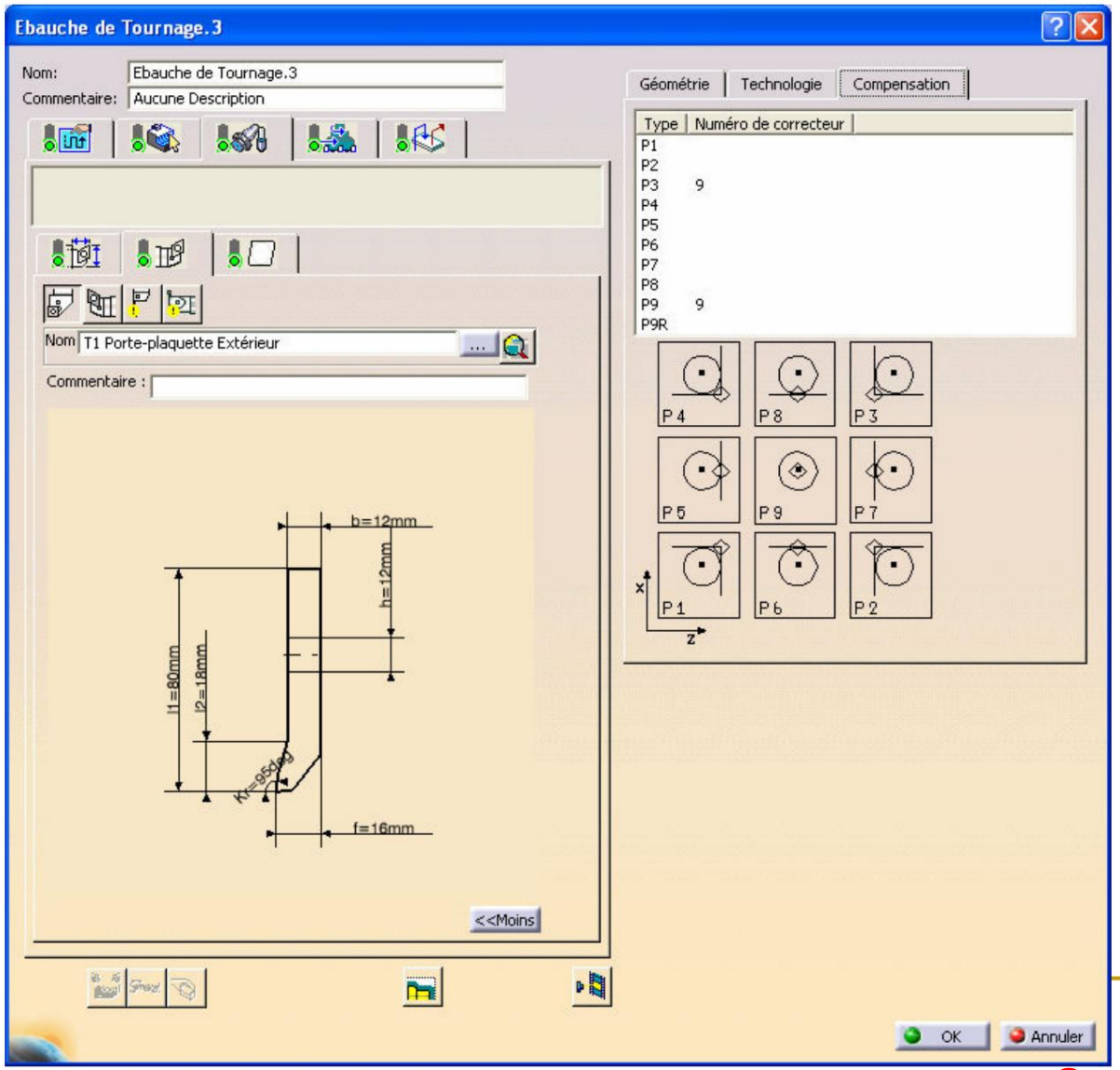
Select a radial offset on the part of "-1 mm".

This will completely remove the material on the part (do the simulation at 0 and at -1 to see the difference). Depending on the tool used, there is material left in the center of the part.

It is in fact a radial offset of the tool path that will have to be compensated for not to enter the material at full speed.

Save your work regularly

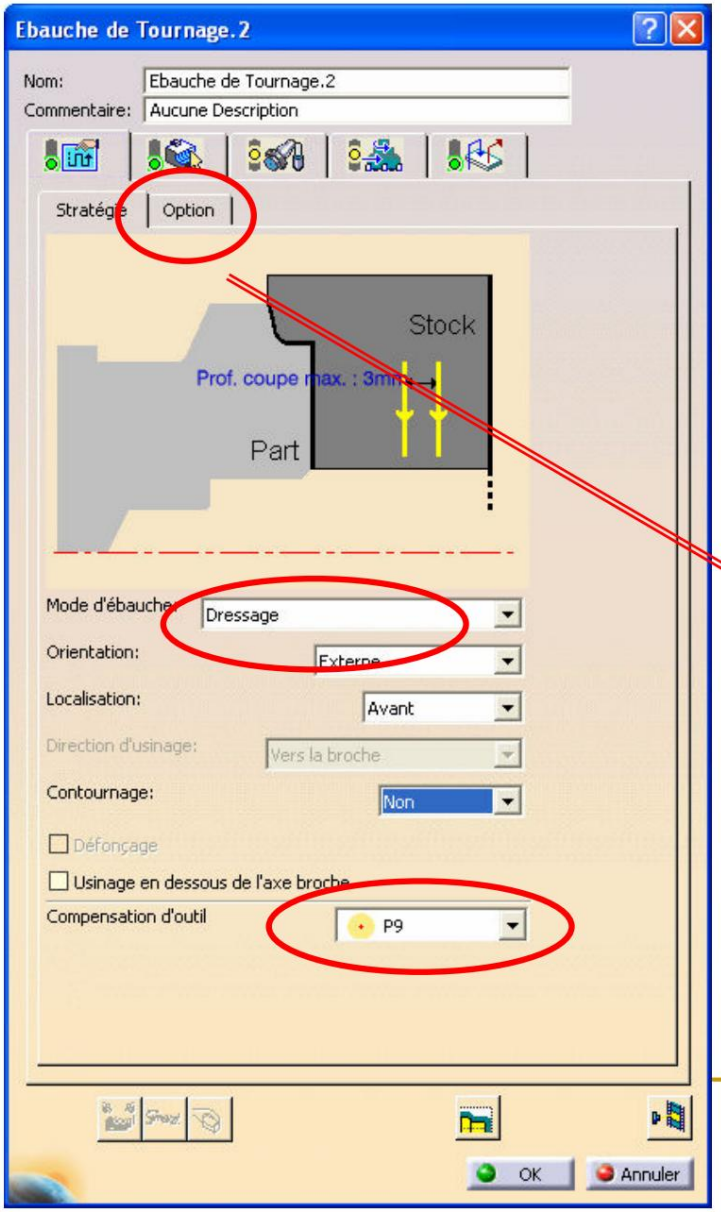
Dressage



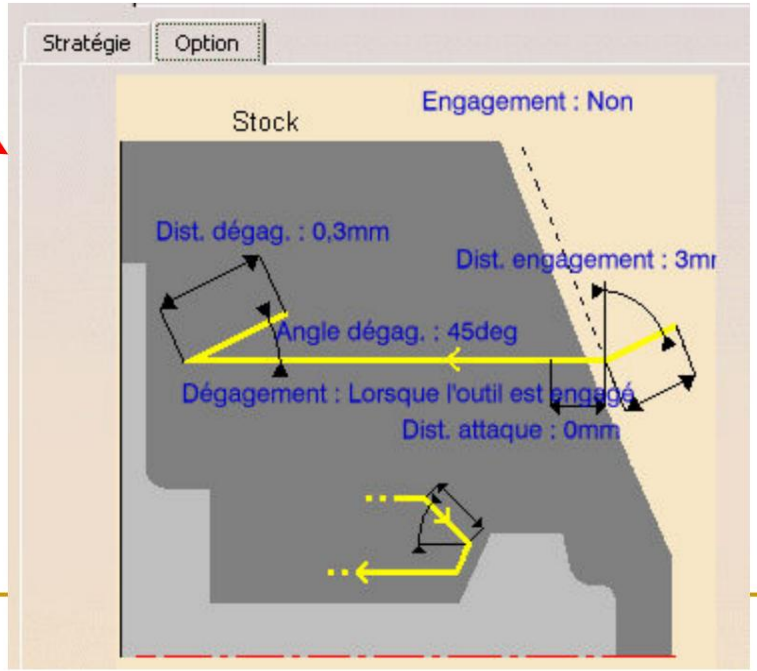
In the tool tab, select a CNMG insert and define a compensator for the P3 type correction of the tool

Save your work regularly

Dressage

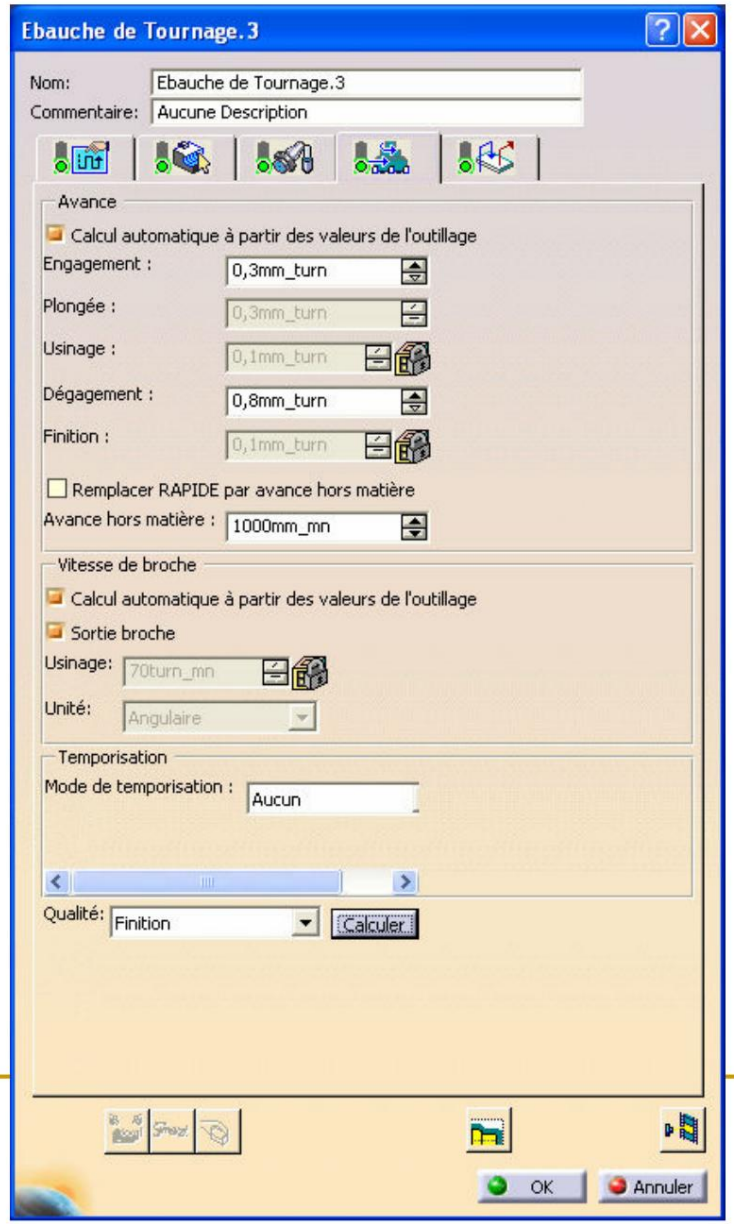


Under the machining strategy tab, select Dressing The tool compensator to use is P3 Click on the option sub-tab Double click on engagement distance and increase the value to 3 mm. This compensates for the "-1" in radial. The paths in this diagram are made at working speed.



Save your work regularly

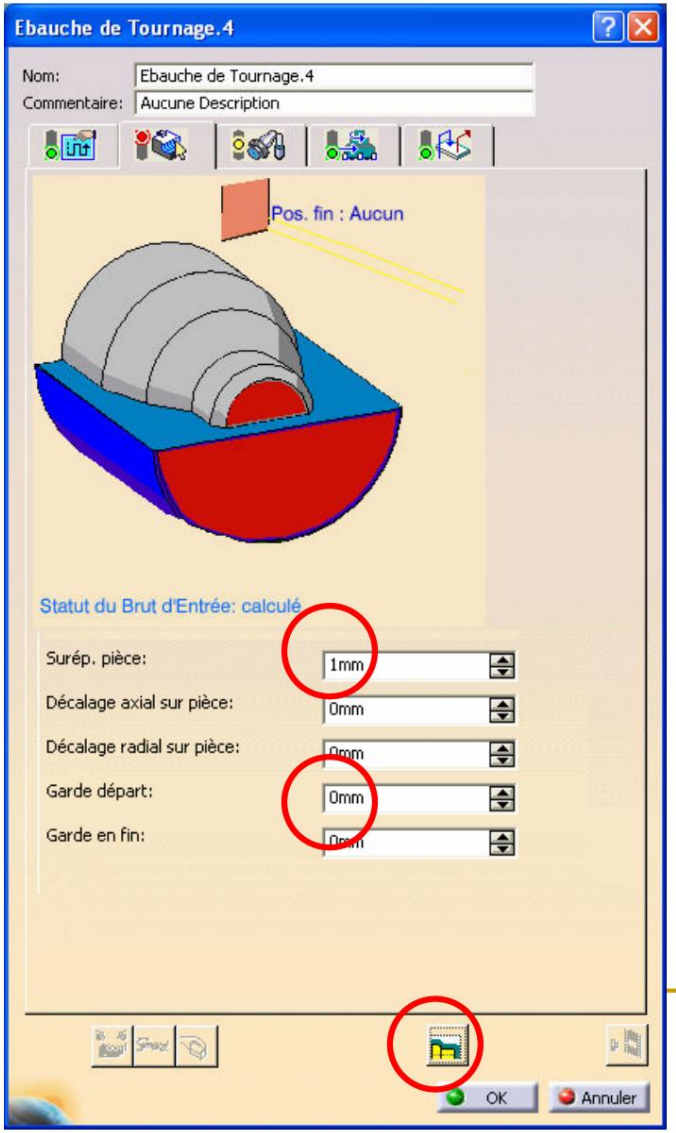
Dressage



Select the finish quality (+ Calculate) and launch the animation of the toolpath to validate the operation (it is now called "Solved").

Save your work regularly

Paraxial draft



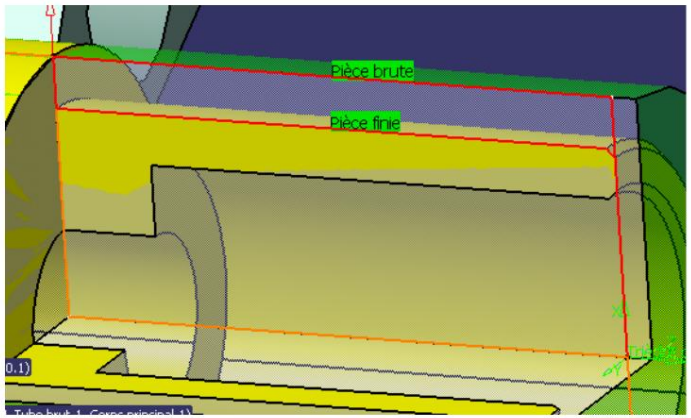
The paraxial roughing will make it possible to obtain the external diameter

Click Turn Rough

Click the Update Input Stock icon

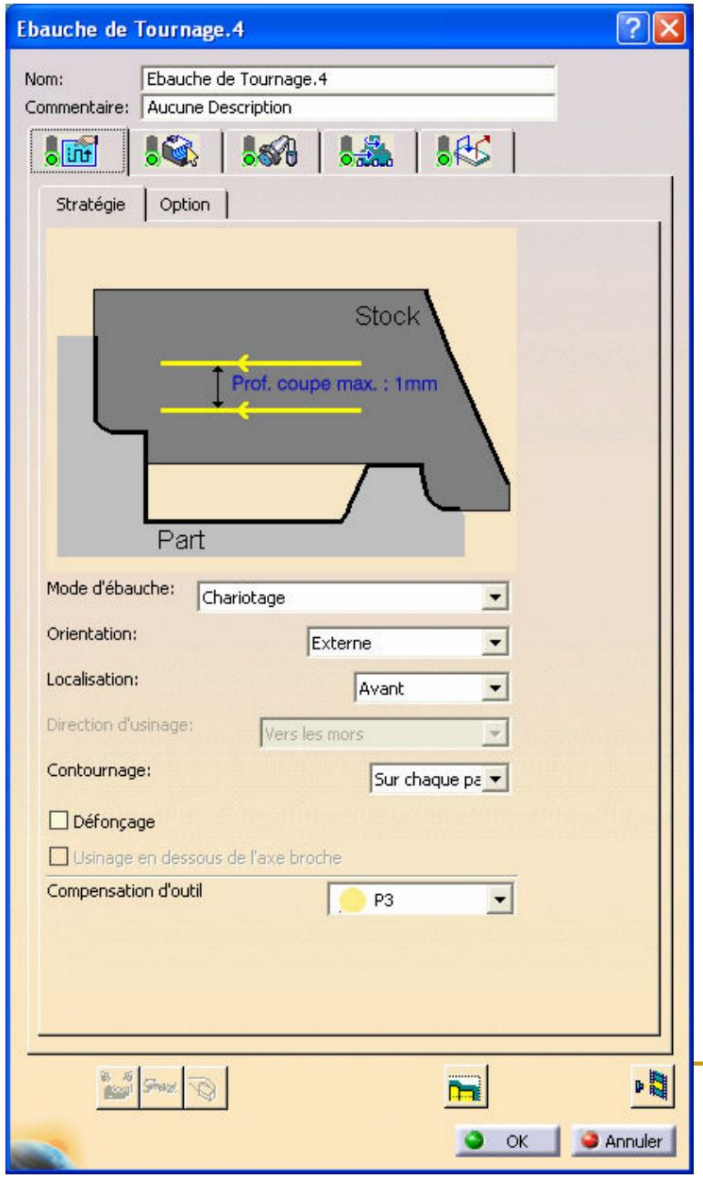
This makes it possible to select as the raw profile of the operation the finished profile of the previous operation (to be repeated therefore at each operation from the second

Select the raw profile (vertical and horizontal edge) and the finished profile (chamfer, horizontal line and vertical line up) Check the zero radial offset and a part allowance of 1 mm.



Save your work regularly

Paraxial draft



This operation is done with the same tool as dressing

Under the Machining Strategy tab, select "Tilling"

Maximum cutting depth is 1mm
The tool compensator to use is P3

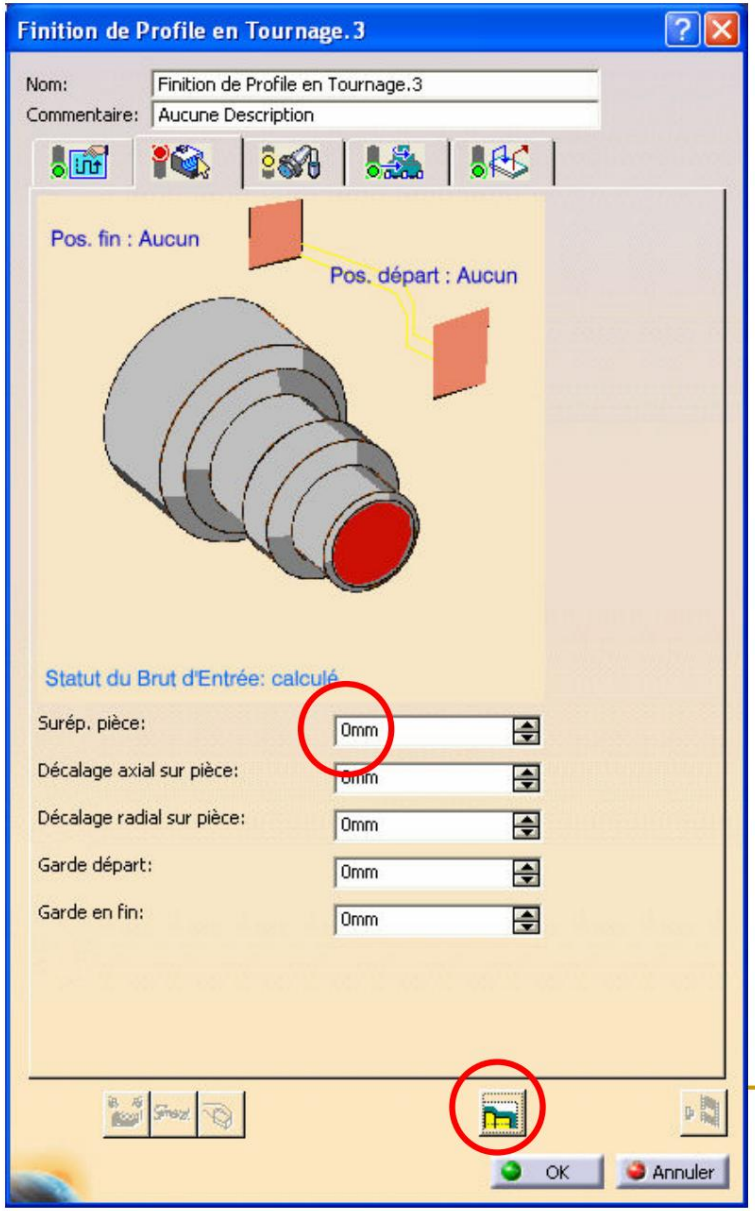
Click on the “options” sub-tab
Reset engagement distance to 2 mm

On the other hand, the clearance distance can be increased to 2 mm.

Run the animation and validate

Save your work regularly

Turning profile finish



Start a profile finishing operation, always with the same tool

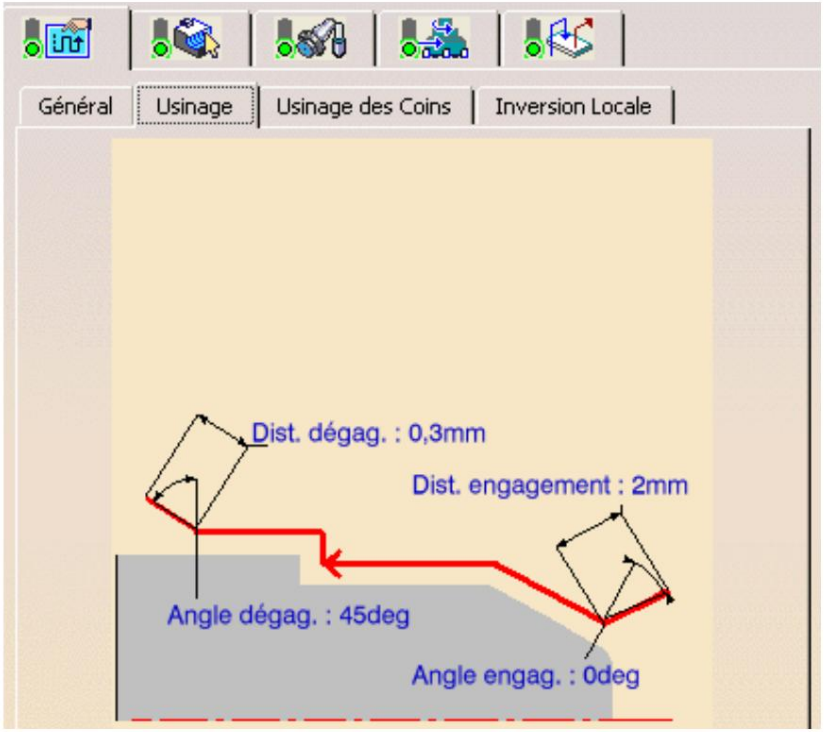
Check that all values are null

Calculate gross input

You notice that the finishing operation does not require a rough definition, so you have to visually check the consistency of what is going to be machined

Save your work regularly

Turning profile finish



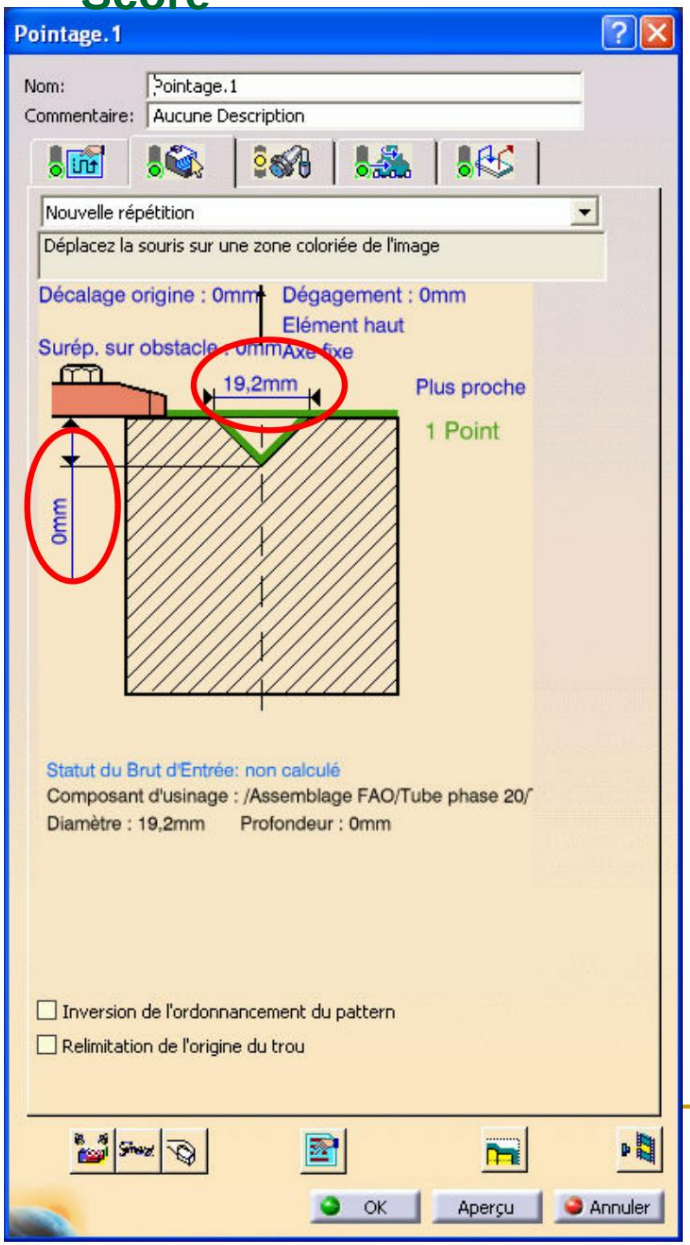
Check the zero engagement angle It appears, after simulation, that the tool collides with the part before the operation (after the paraxial roughing). By setting an engagement distance of 5 mm, the collision no longer occurs.

Under the Speed and Feed tab Select Finishing Quality

Run the animation and confirm

Save your work regularly

Score



Launch a pointing operation (by dragging the drill icon)

Select the front face of the part (hiding the stock may be necessary)
Click on No point then on a circle of the front face (to select its center). Double click to return to the dialog box opposite

Select a diameter of 2.5 mm and a depth of 3 mm

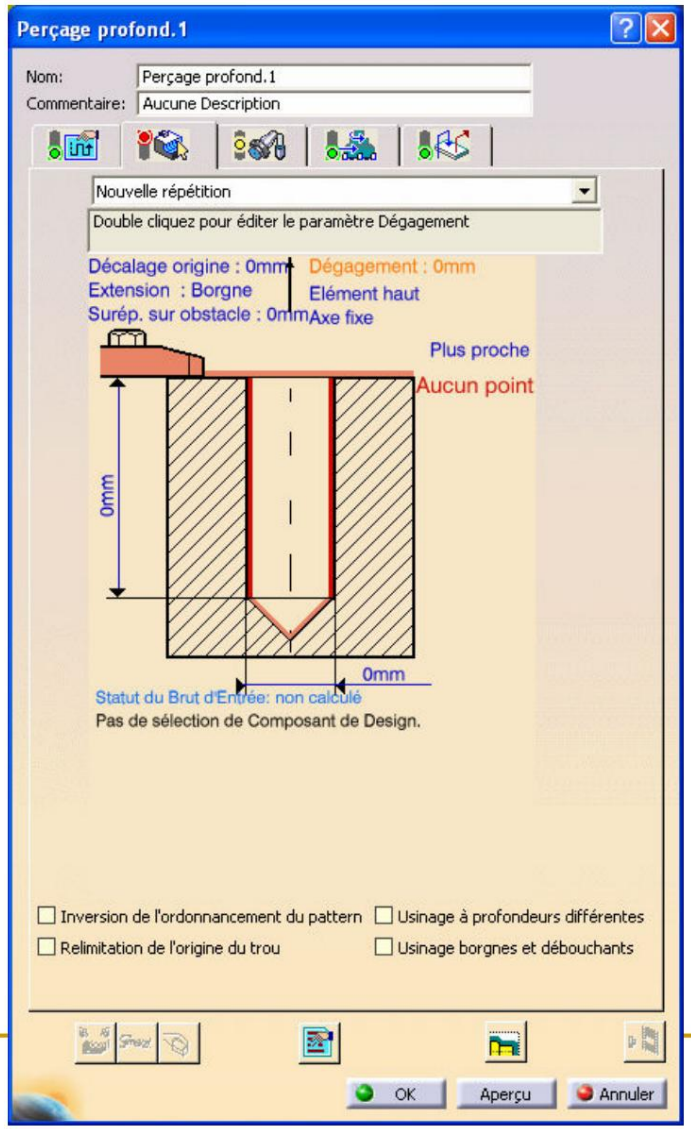
Don't forget to calculate the input gross

Select the 2.5 mm diameter center drill and launch the animation of the tool path.

We can optionally add an axial approach and withdrawal of 5 mm

Save your work regularly

Drilling



Launch a deep drilling operation (by dragging the drilling icon)

Selections are similar to the pointing operation.

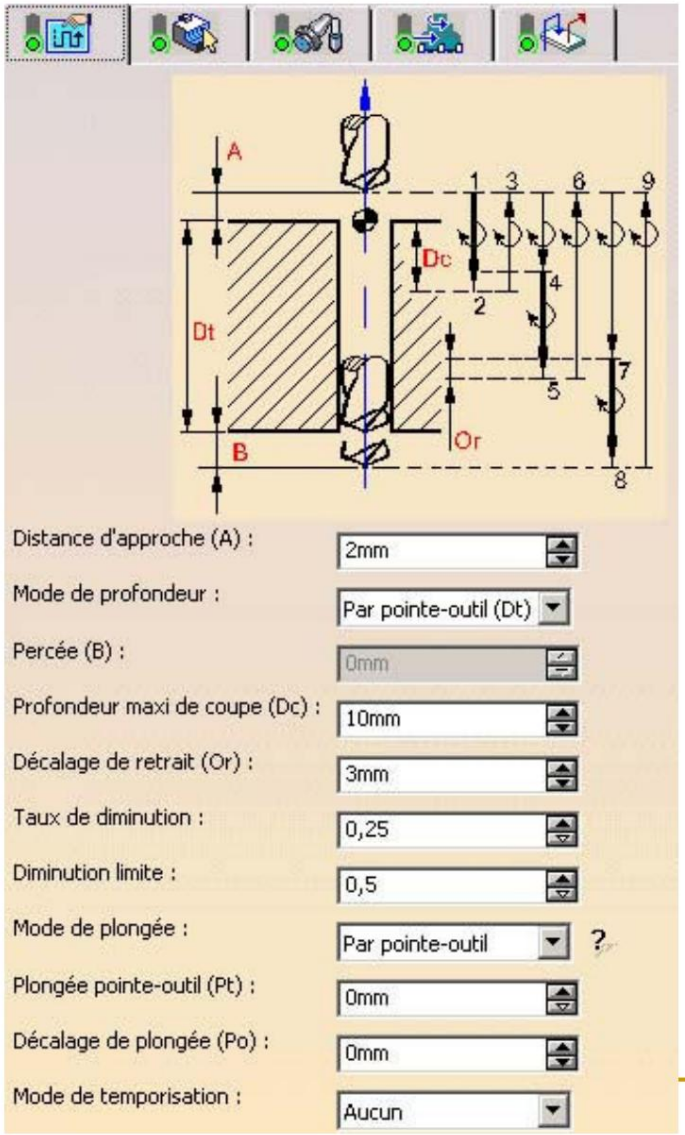
Select a diameter of 10 mm and a depth of 49 mm

Don't forget to calculate the input gross

Select the twist drill diameter 10 mm

Save your work regularly

Drilling



The deep drilling operation is actually a peck drilling operation

Complete the data as opposite

Launch the animation of the tool path and validate the operation.

Create a simple drilling operation diameter 13 (therefore drill diameter 13 from the catalog) and depth 28 at the same point

Save your work regularly

bore

It remains to make the internal

boring Proceed in the same way as for the external diameter: rough turning then profile finishing The turning is internal The tool is

the internal boring tool (S10 L – SCLCL06)

from the catalog with the P2 corrector The production of the part is finished, the CN

Code can be generated (interactive mode, etc.)

Generating a document

Click on the following icon:

The documentation is published in digital form (html) and may contain images and videos.

This documentation is generated from a script (\MachiningDocument.CATScript by default).

According to the caption, to make the document more user-friendly and more in line with the habits of your workshop, the script can be edited, but that is in the domain of VBScript programming and no longer CAD/CAM!

End of TP3

To remember

The raw profile for each operation is calculated automatically from the previous operation (updated input raw icon)

Each type of operation can be configured in a very complex way (angle of engagement, release, etc.)

Be careful not to confuse the engagement and disengagement parameters (in working speed) with the approach and withdrawal (the speed of which can be adjusted by right clicking)

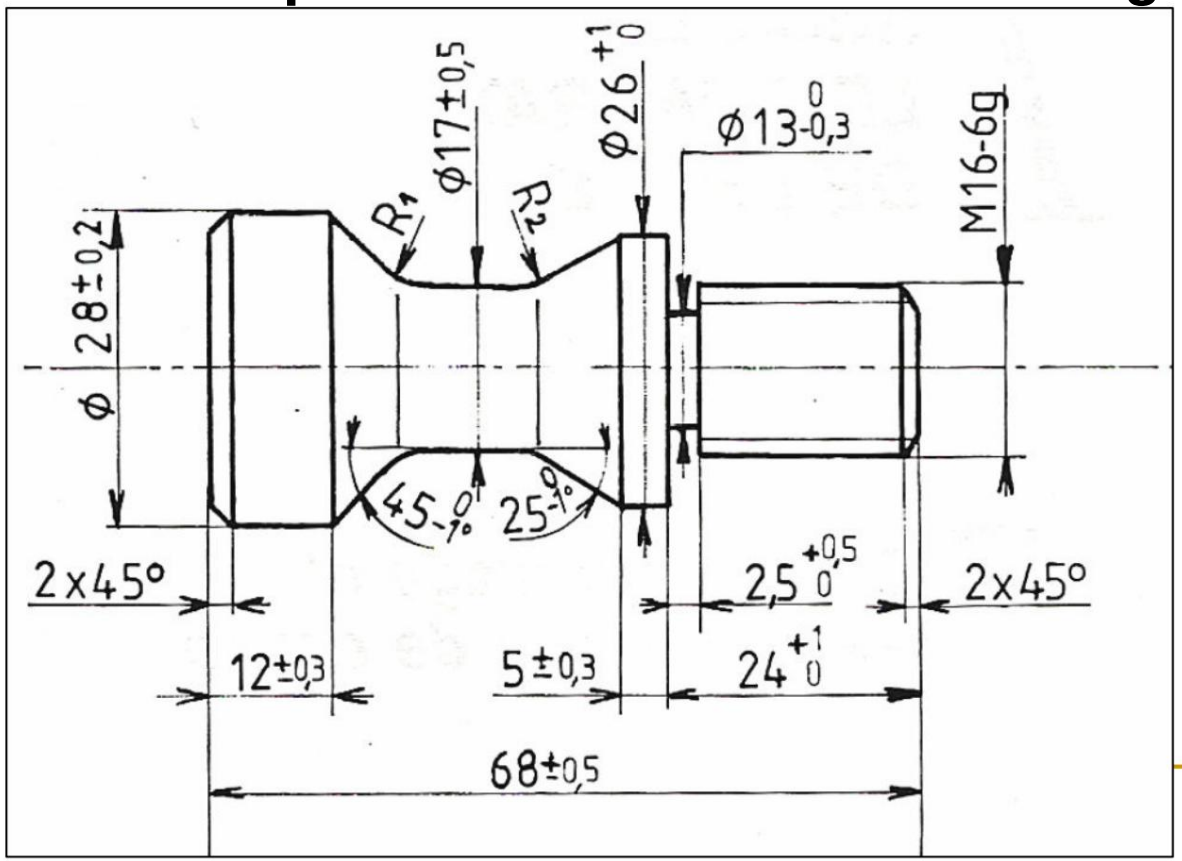
Radius correctors must be configured according to the operation carried out (P3, P2, etc.)

HTML documentation can be edited at the end of programming, it summarizes in detail the operations carried out on the part

Note: The CNC Code generated in turning is by radius (not by diameter)

TP3'

To consolidate your knowledge, here is a new part to be machined in turning



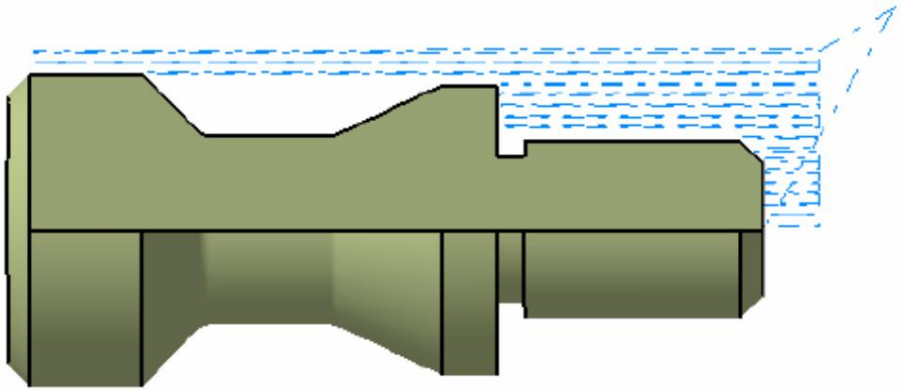
R1 and R2 depend on the insert chosen for routing (these radii are not functional)

M16 thread is not required

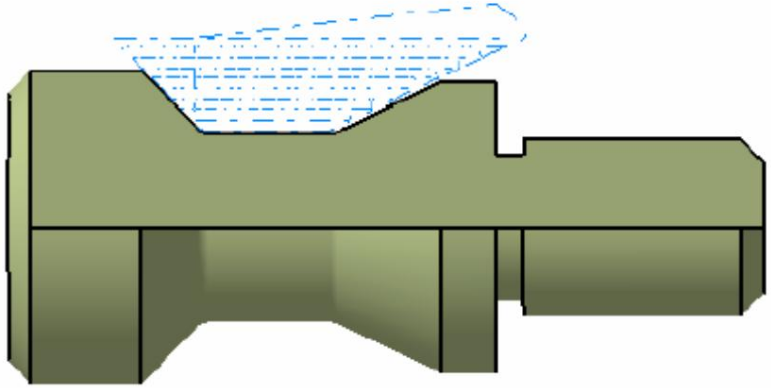
Save your work regularly

TP3'

Ebauche: sélectionner toutes les arêtes du profil fini.



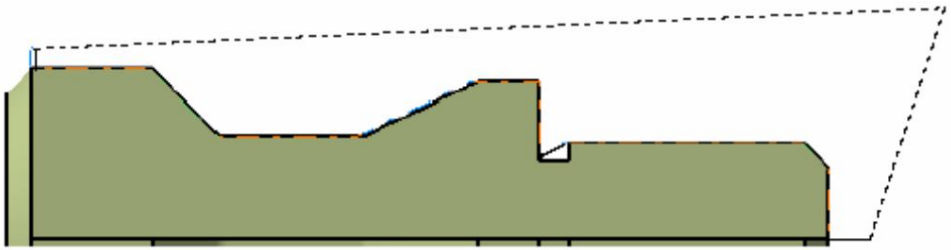
Défonçage: sélectionner seulement le trait horizontal pour définir le brut



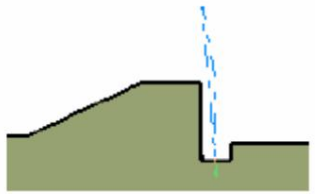
Après la première opération, il est intéressant de redéfinir le brut pour l'opération suivante. Pour cela cliquer sur

pour obtenir le résultat suivant: Défonçage en Tournage.1 (Résolue) (Brut à jour)

Contournage finition



Finition de la gorge



Save your work regularly