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

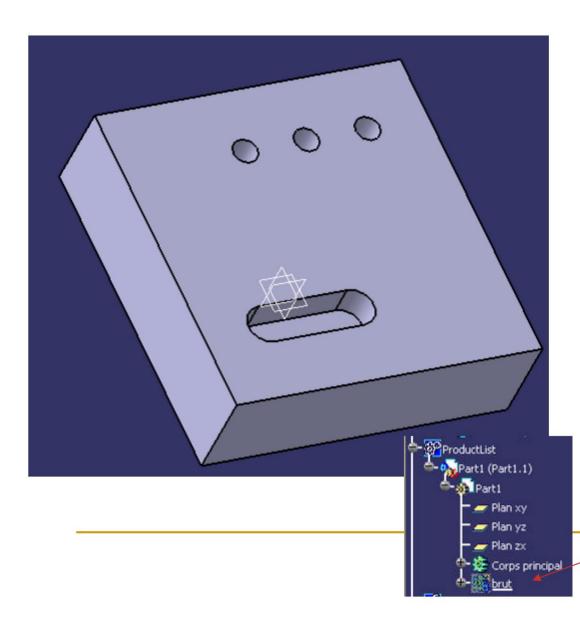
construction philosophy of each machining operation It is a question of defining what is

being machined (entity), how (strategy, approach/retreat), with which tool (and which cutting conditions)

Discover the simulation of material

removal (tool paths, interferences, etc.)

Part drawing



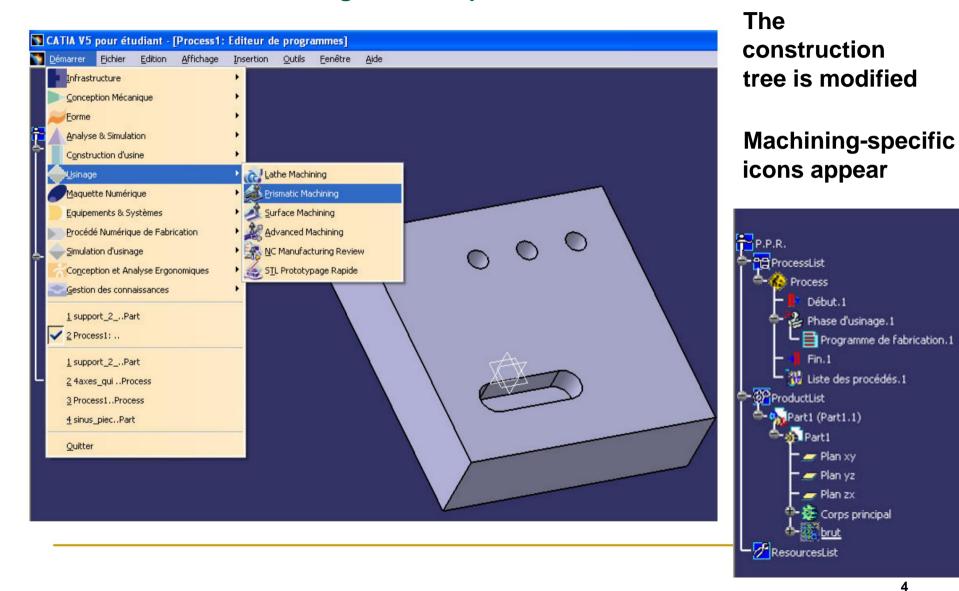
Parallelepiped 100 x 100 x 30 (Extrusion of a 100 x 100 square)

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)

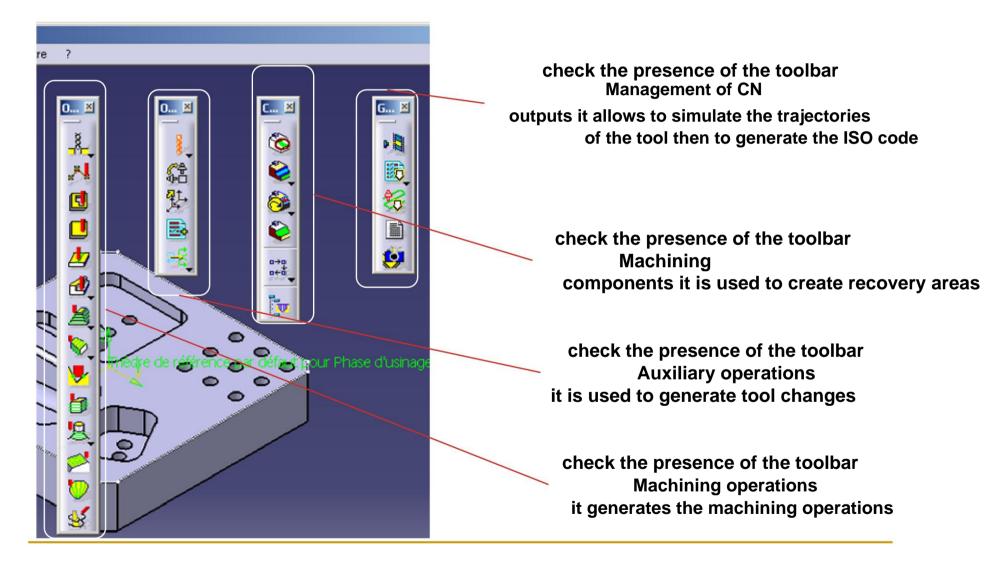
CAM requires the CAD definition of a raw part and a finished part. Copypaste the part body in the same "Part", rename it "stock" (right click, properties), modify the extrusion to 32 mm and delete the holes and the pocket

The "Prismatic Machining" workshop



Save your work regularly

Useful toolbars



Default Options

Dém M M M M	A VS pour étud harrer Eichier	Edition	Affichage Options Général Général Général Ginfrastru Concep Forme Analyse Constru Eabricat Maquett Equipen	Insertion C lage patibilité nètres et me: nériques et R	Qutils Feg Général Post proce Edition du Lors du ca Point de si So Localisatio	esseur Aucun PPs: du trajet ou Stocke I Stocke I Stocke I trajet outil Disponit lcul du trajet Stocker Stocker ortie outil - Bout dk O Centre n trajets out Trajets out Doc, atlier: Code CN:	le trajet outil dans le document i le trajet outil dans un fichier ext bilite de l'editeur du trajet outil et outil les points contact dans le trajel putil outil utils, code CN et documentation il: Sauvegarder au même en C:\TEMP\lescalierc\ C:\TEMP\lescalierc\ C:\TEMP\lescalierc\	M(r) I3\intel_a\startup\Ma courant terne t outil		
1		F					C:\TEMP\lescalierc\			
			- 1			Extension:				
								1	● ок 🛛 ● А	Annuler

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

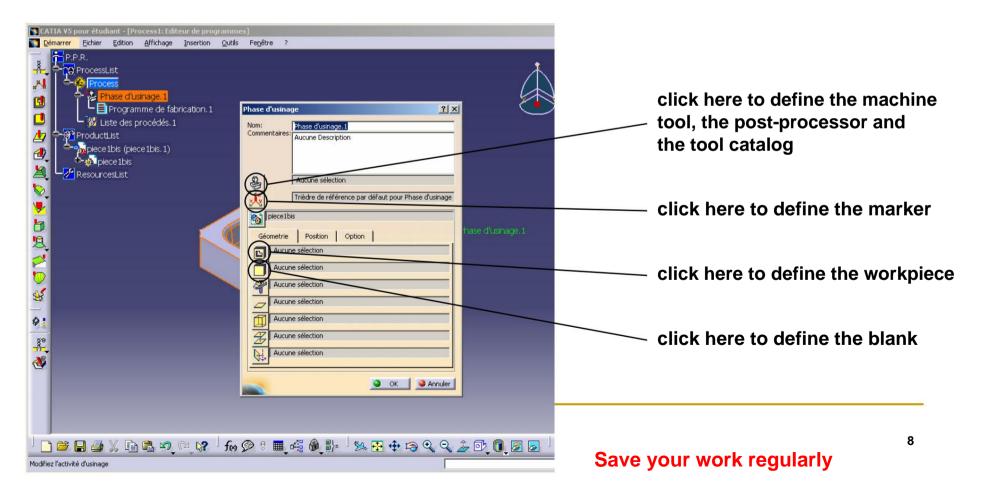
Default Options

Options	<u> </u>
Options Général Général Gompatibilité Minor Paramètres et mesure Périphériques et Réalité Virt Infrastructure Conception Mécanique Forme Analyse & Simulation Constructure d'usine Usinage maquette Numérique	Général essources Opération Programme Sortie Photo/Vidéo Vérification trajet outil Afficher l'outil sur le trajet outil suivant la position de la souris. Afficher le centre de l'outil au lieu de l'extrémité. Afficher les cercles. Couleurs associées aux avances : Coupe (couleur par défaut) Approche ou engagement Retrait ou dégagement Retrait ou dégagement Rapide Finition Chanfreinage Plongée Hors matière
Equipements & Systèmes Procédé Numérique de Fabrica Simulation d'usinage Conception et Analyse Ergono Gestion des connaissances	Gérmétrie complémentaire pour la CN
	OK Annuler

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

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,



icor	ns defining the	architecture		nachine tool used
Définition de la machine			<u>? ×</u>	
Nom 3-axes_Machine	Dar_defaut			
Commentaire				
Commande numérique Ch	nangement d'outil Broche C	Compensation		• the machine
Post Processing	numENSAM.lib	•		
Table de mots Post Processeur	IMSPPCC_MILL.pptable		•	machine (on
Type de données CN	ISO			architectures
Type de format CN	Point (X,Y,Z)			
Stratégie au point de départ	From		1. Silversite	post-process
Rayon d'interpolation mini	0,01mm			processor w
Rayon d'interpolation maxi	5000mm			•
Pas de discretisation mini	1e-004mm			type • the lan
Angle de discrétisation mini	0,1deg			type
Interpolation linéaire 3D	a			
Interpolation circulaire 2D	a		ne na e	
Interpolation circulaire 3D				
Interpolation Nurbs			inter inter	
Vitesse max en usinage	100000mm_mn			
Vitesse rapide	60000mm_mn			
Mouvement Axial et Radial				
		OK	Annuler	Sa

Choice of machine

 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 postprocessor word table is of IMSPPCC_MILL type • the language to be exported is ISO type

Choice of tool catalog

Définition de la machine		? ×
Nom 3-axes_Mad	thine_par_defaut	
Commande numérique	Changement d'outil Broche Compensation	
Catalogue d'outils	Non spécifié	
F 5 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	H12 H21 FeedsAndSpeeds InsertsSampleLathe .atheInsertsAndHoldersSample MillAndDrillStarterSet FoolReplacement FoolSamples_MMG FoolSamples_SMG FoolSSample01 FoolSSample02 FoolSSample03	<u> </u>
	ОК	Annuler

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).

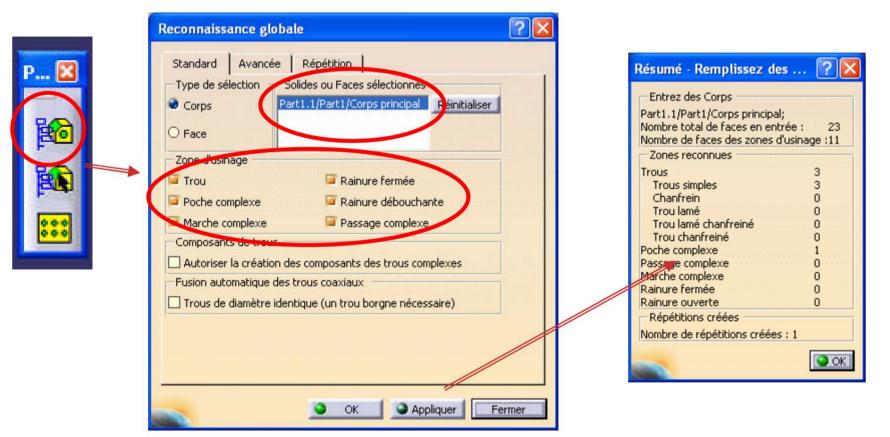
Rough/finished part definition

CATIA V5 pour étudiant - [Process2: Edil		
Démarrer Elchier Edition Affichage Image: State of the state of	piece1bis/piece1bis/Brut.1 Aucune sélection Aucune sélection Aucune sélection Aucune sélection Aucune sélection Aucune sélection Cocc	Annuler
Modifiez l'activité d'usinage	[이 ૢ ૣ ?] fo 💬 8 🔳 여름 🖗 라는	<u>'‰⊕</u> ⊕©©(< <i>≦</i> ⊡

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.

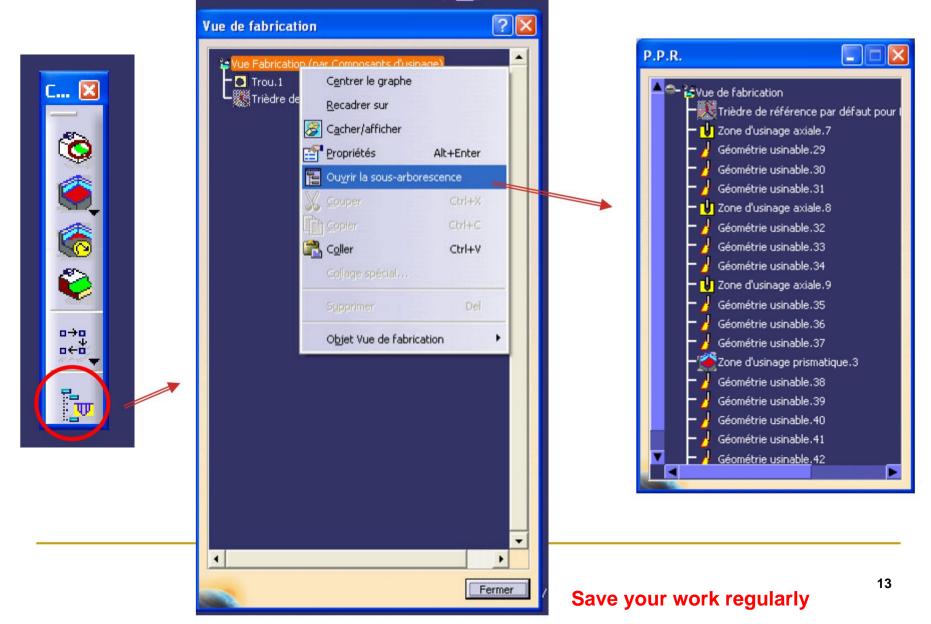
Double-click in the CATIA window to see the Machining phase window reappear. Click OK to exit.

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)

Visualization of machining entities

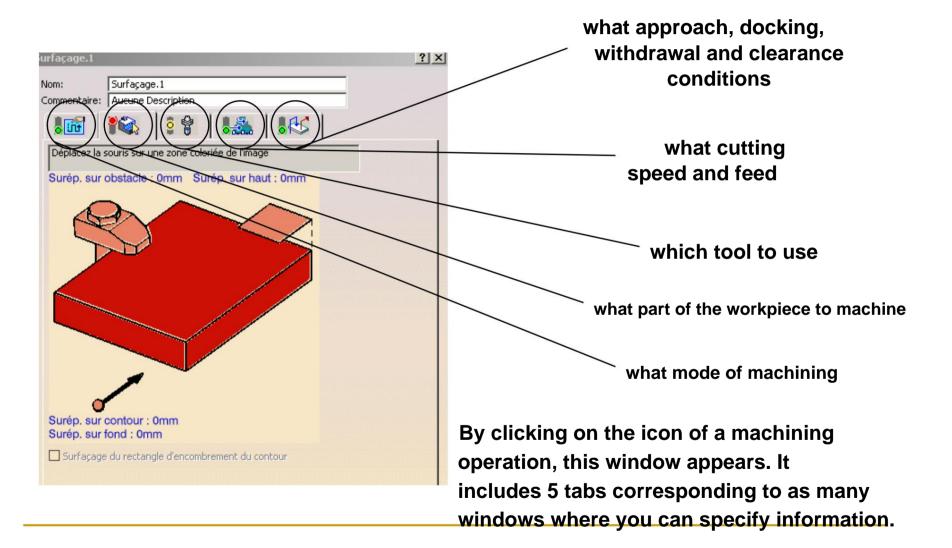


The generation of trajectories starts here. The program is created by a sequence of elementary operations.

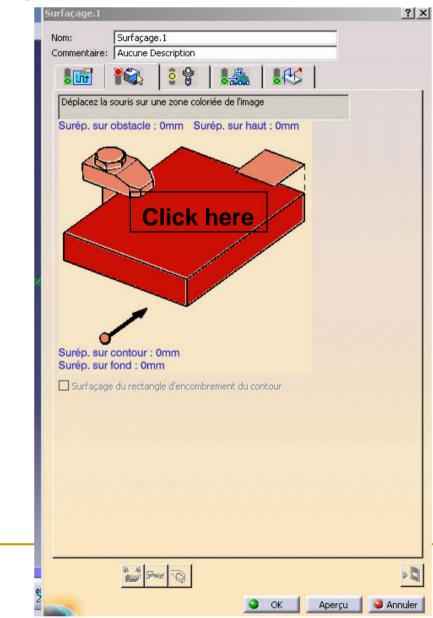
For each of these operations, the programming mode is identical. It consists of explaining to CATIA: • which tool to place in the spindle nose, • which part of the part to machine, • how to machine the surface (defining an operating strategy, ie a type of trajectory, or even engagement conditions, ie 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

Typical presentation of an 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

Surfacing operation
Surfaçage.2
Nom: Surfaçage.2 Commentaire: Aucune Discription
Nom T7-Fraise à Surfacer D 50
Commentaire : Nom d'outil
Numéro d'outil : 2
Da=\$0mm
Plus>>
OK Aperçu Annuler

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

Surfaçage.1		? ×
Nom: Surfaçage.1 Commentaire: Aucune Description		
Déplacez la souris sur une zon	e coloriée de l'image	
	31.5mm	
Mode d'usinage:	Aller-retour	-
Stratégie Radiale Axiale	Spirale vers l'intérieur Aller-retour	
Sens d'usinage:	Aller simple En avalant	,
Tolérance d'usinage:	0,1mm	?
Précision bridage:	0,1mm 🛃	
Contournement arêtes vives:	:ulaire	?
Compensation :	1	•
8 49 Smart 70		•
	OK Aperçu	Annuler

This tab defines the machining strategy.

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

	peration		<u>? ×</u>	
Surfaçage.1				
e: Aucune Descrip	ition			
🎼 🜡	9			
d'avance				
automatique à partir	r des valeurs de l'outill	age		
300mm_mn	-			
2728,37mm_mn	E			
, 2000mm_mn				
2728,37mm_mn				
Linéaire	•			
3031,523turn_mn Angulaire inition				-
			dia dia mandri di secondo di	
See 1	3			
	e: Aucune Descrip automatique à parti 300mm_mn 2728,37mm_mn 2000mm_mn 2728,37mm_mn Linéaire broche automatique à parti vitesse 3031,523turn_mn Angulaire	e: Aucune Description	e: Aucune Description automatique à partir des valeurs de l'outillage 300mm_mn 2728,37mm_mn 2000mm_mn 2728,37mm_mn 2000mm_mn 2728,37mm_mn 2000mm_m	e: Aucune Description

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.

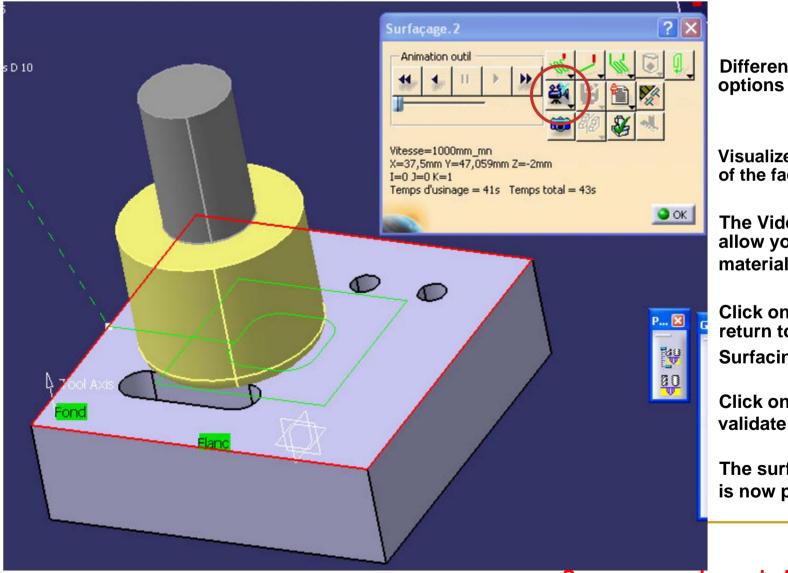
The last tab allows you to modify the tool path in approach (before material removal) and in withdrawal (after).

om:	Surfaçage.1	
ommentaire:	Aucune Description	
-Vitesse d'av	vance	
😇 Calcul aut	tomatique à partir des valeurs de l'outillage	
Approche:	300mm_mn	
Usinage:	2728,37mm_mn	
Retrait:	2000mm_mn	
Finition:	2728,37mm_mn	
Unité:	Linéaire	
Vitesse de l	broche	
🖬 Calcul aut	tomatique à partir des valeurs de l'outillage	
🧧 Sortie vite	esse	
Usinage: 🛐	031,523turn_mn 🗧	
Ar	ngulaire 🔹	
Qualité: Finit		
Qualité: Finit		

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.



Different visualization options are possible.

Visualize the trajectory of the face milling cutter.

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

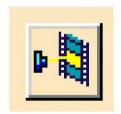
Click on OK to return to the Surfacing window.1

Click on OK to validate the parameters.

The surfacing is now programmed.

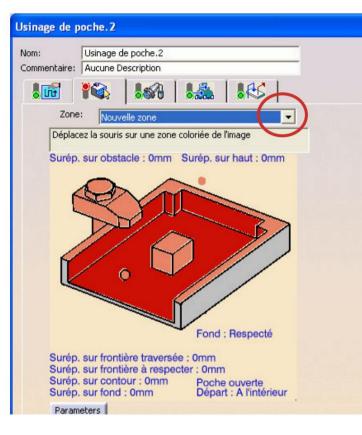
Save your work regularly

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



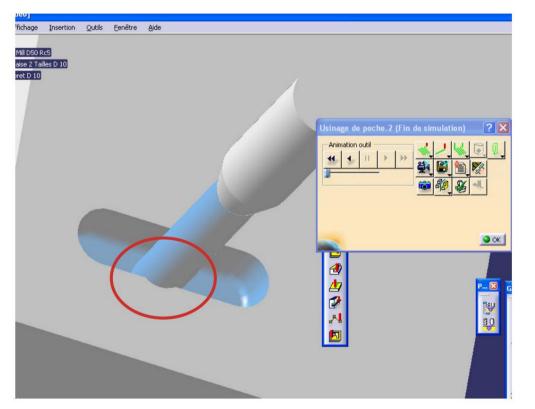


Select the proposed prismatic machining area. It corresponds to the pocket recognized automatically Keep the default machining strategy.

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.



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

Machining of the oval pocket

Usinage de poche.2	? 🔀
Nom: Usinage de poche.2	
Commentaire: Aucune Description	
Im Im Im Im (Im)	
Gestion des Macros	
Metro Nom Mode 🔨	
Approche Approche.1 Construit par I	
Retrait Retrait.1 Construit par I	
O sarde benzontale Garde.1 Jusqu'à un pla	
O Liaison Retrait Liaison.1 Construit par I O Liaison Approche Liaison.1 Construit par I	
O Liaison Approche Liaison.1 Construit par I	
O Liaison sur un niveau App Liaison sur un nive Construit par l	
O Liaison sur passe de finiti Liaison sur passe Construit par I 👽	
Macro en cours	
Définition Options	
Mode: Construit par l'utilisateur	
Fox O	
OK Aperçu	Annuler

Click on the Approach/Retreat tab

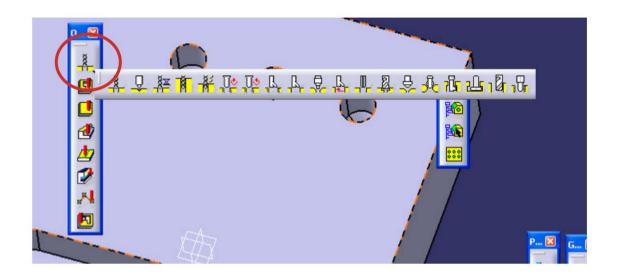
Highlight "Approach"

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.

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

```
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.

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

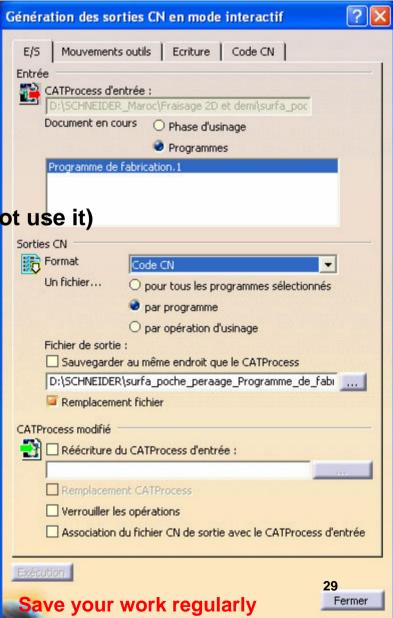
The goal is now to collect these parts of code and export it in text format (.p for example for NUM command director)

NC code generation

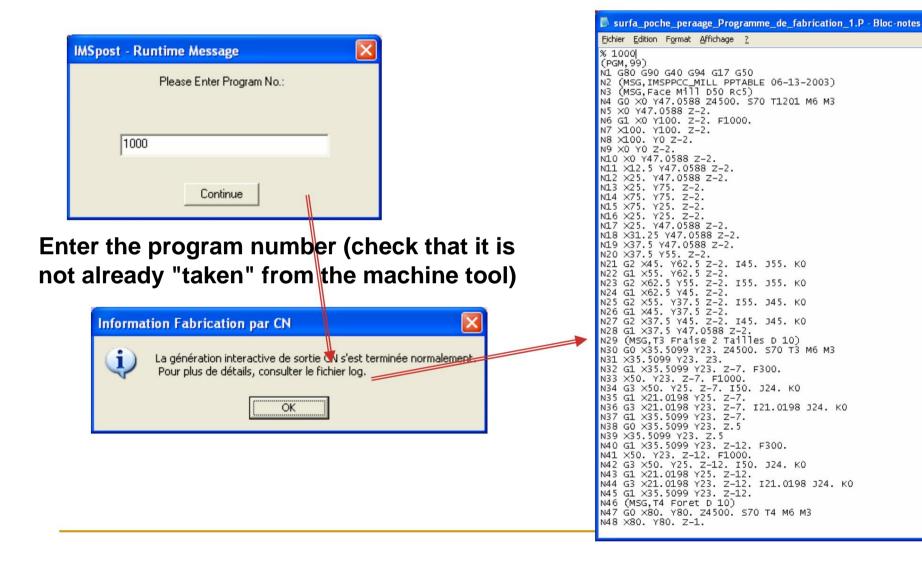


Click on "Generate NC code in Interactive mode" Another generation mode is possible (batch mode), which makes it possible to store and concatenate different programs (we will not use it) Sorties CN Format Un fichier.

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.



NC code generation



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

```
Exercise 2: 3D milling
```

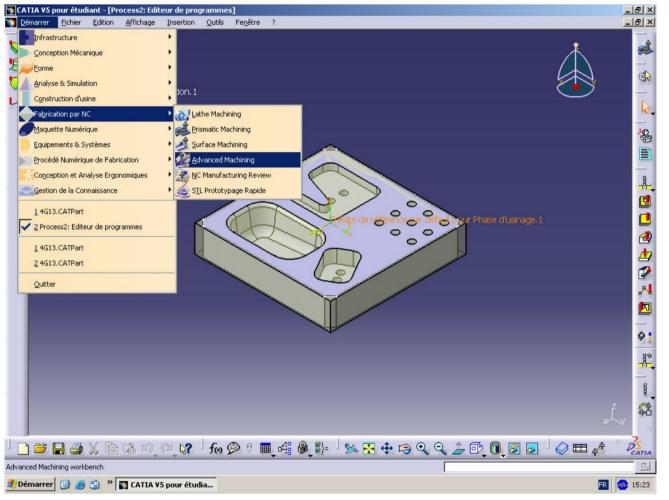
The objectives of this lab are:

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

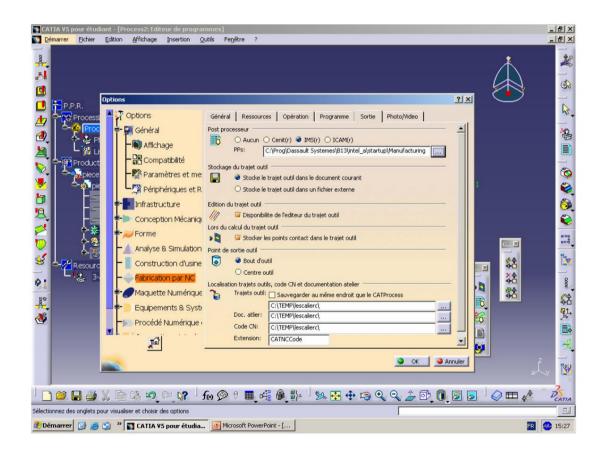
Open the room



open the CATpart file of the lab

start the activity Manufacturing by NC / Advanced Machining

Initial checks



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

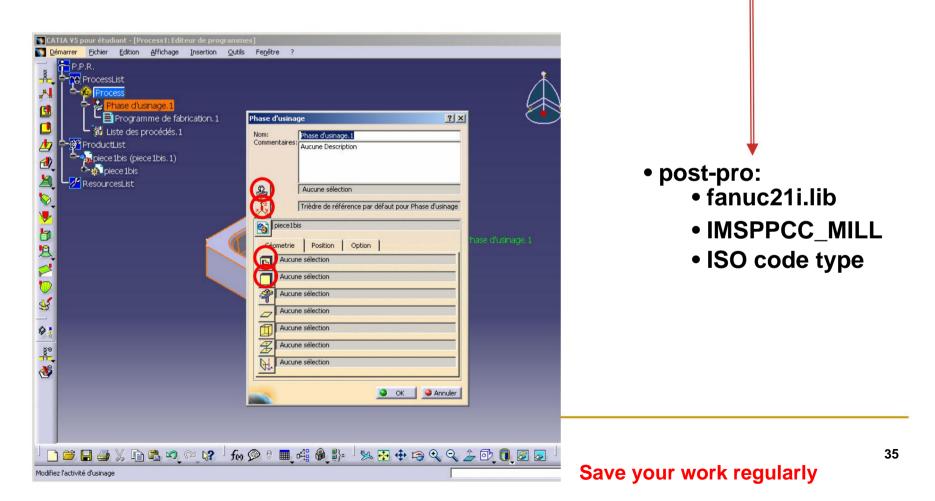
the type of post processor (ie IMS),

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

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

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,

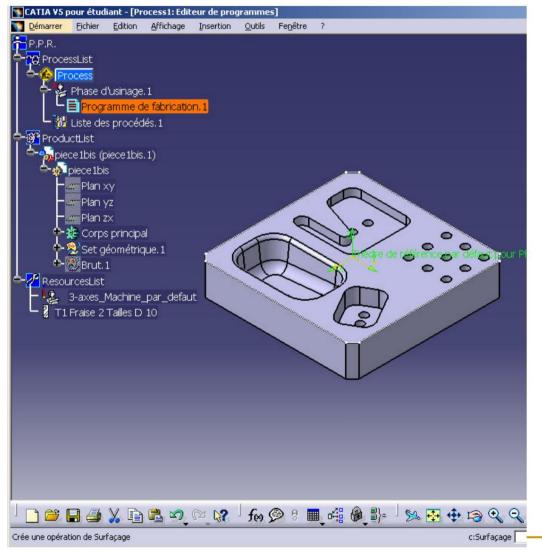


The generation of trajectories starts here.

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

Surfacing



Highlight Manufacturing Schedule

click on the surfacing icon

Select the top surface of the part

Surfacing

CATIA V5 pour étudiant - [Process2: Editeur de programmes]	Surfaçage.1	? × ×	
Démarrer Eichier Edition Affichage Insertion Qutils Fenêtre ?	Nom: Surfaçage.1 Commentaire: Aucune Description Image: I	×	
			Select the Ø63
		n	nm face milling
dg Bare			utter.
Image: Second secon	Commentaire :		
	Numéro d'outil : 1		
		?×	Check
Recherche dans: 4d12		t t	he compensation, he tool number
Recherche dans: 4d12 Recherche dans: 4d12	Angle coupant Diamètre nor		and the presence of cutting
Ø:			conditions implied not
	Bc=1.5mm D=63mm Da=75,5mm		zero)
		.	
	• ок •		
ToolSelection			

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.



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.

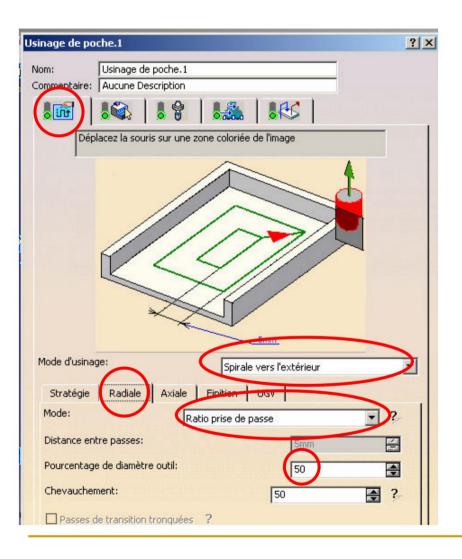
Click on the Pocket Machining icon. Select one of the straight pockets The tool to be used is the 10 mm diameter 2size milling cutter

Usinage de poche.1	? ×
Nom: Usinage de poche.1	
Commentaire: Aucune Description	
Zone: Nouvelle zone	
Cliquez pour sélectionner le fond Puis sélectionnez une face dans la vue 3D	
Surép. sur obstacle : 0mm Surép. sur haut : 0mm	
Effacer	
Analyser	
Surépaisseur	-
Surép, sur frontière tra Surép, sur frontière à Detection des îlots	
Surép. sur fond : Omn	
Surep. sur fond : 0mm	
See See D	•
	nnuler
	muler

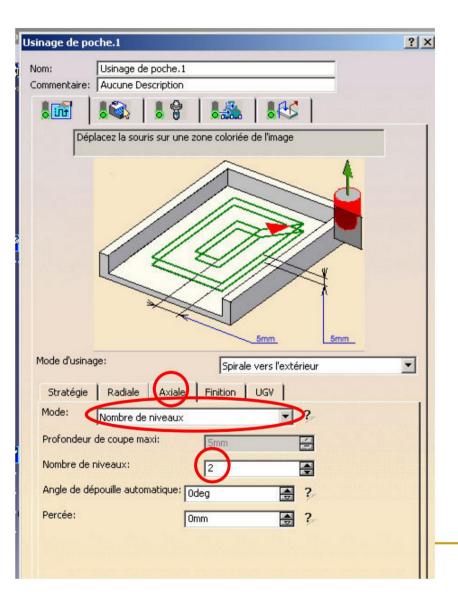
Position the pointer over the image.

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.



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

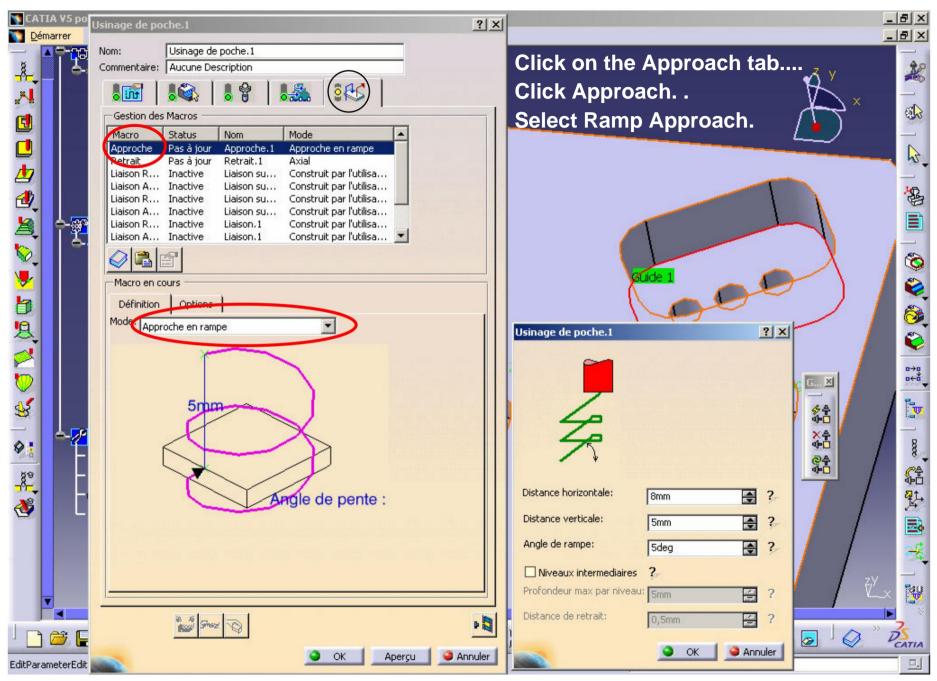


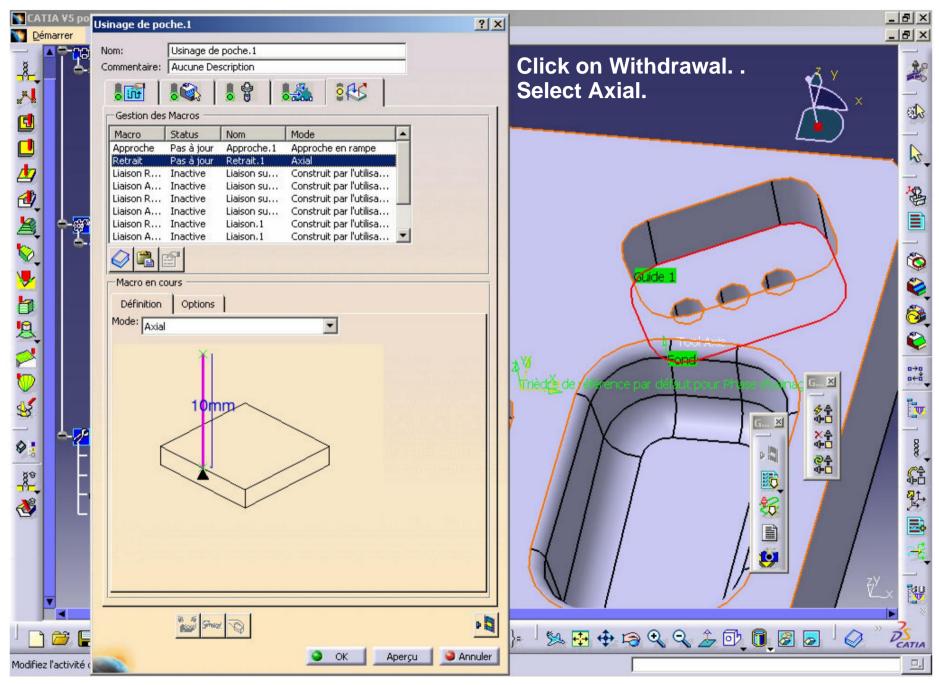
Click on the Axial tab. Select Number of Levels. Set the number of levels to 2. Pocket depth is 10 mm.

The axial depth of cut is therefore 5 mm.

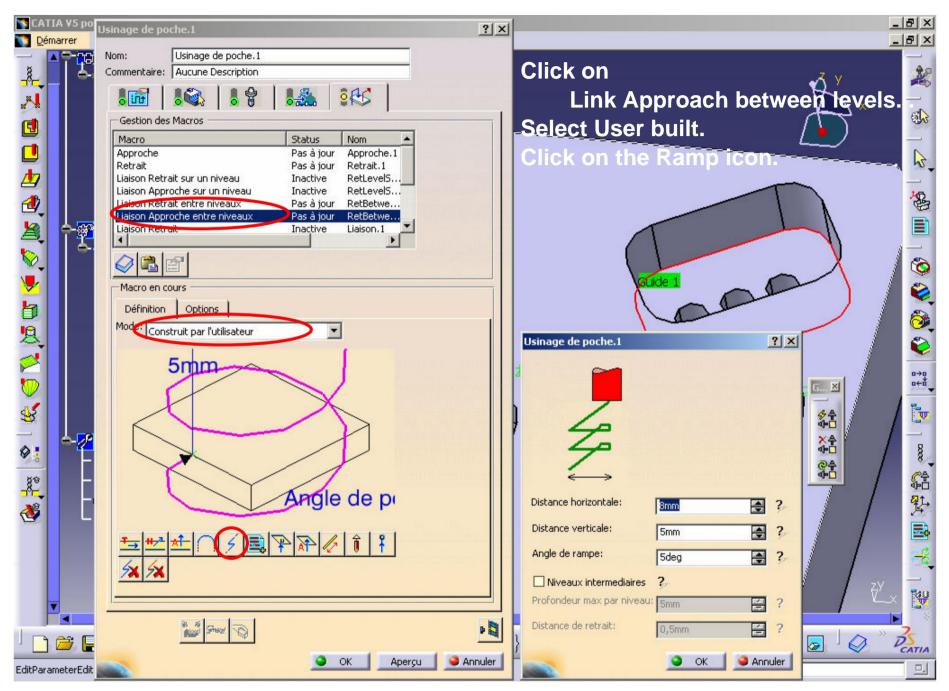
om:	Usinage de poche.1	
ommentaire:	Aucune Description	
	I 🕄 I 🖗 I 🖧 I 🛠	
- Vitesse d'av	ance	
	omatique à partir des valeurs de l'outillage	
Approche:	300mm_mn 📑	
Usinage:	1527,887mm_mn 🛃	
Retrait:	2000,mm_mn	
Finition:	713,014mm_mn	
Taux de raler	ntissement: 1	
Unité:	Linéaire	
-Réduction d	de vitesse dans les coins	
Réduction	n de vitesse dans les coins	
Taux de rédu	uction : 80	
Angle minima	1: 45deg 🚍	
Rayon maxim	ial : 🔤	
Distance ava	nt le coin : 👔 imm	
Distance aprè	ès le coin : 🛛 🔤	
Vitesse de l	proche	
🖬 Calcul aut	omatique à partir des valeurs de l'outillage	
🧧 Sortie vite	sse	
Usinage: 25	646,479turn_mn 🛃	
Unité: An	ngulaire	
Qualite: Eba	uche Calculer	
	B B STORY	• 2

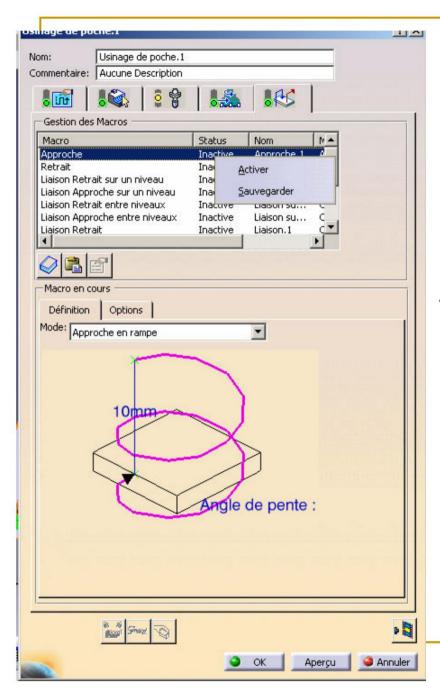
Click on the Speeds tab. Check the withdrawal speed (about 2000 mm/min). Set Quality to Finish.





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.

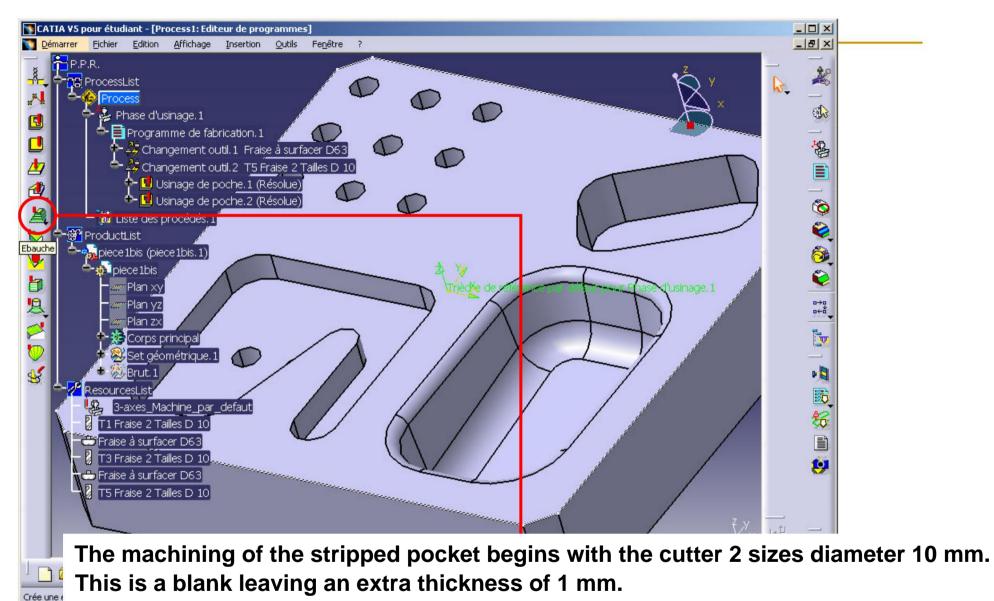
The shallowest one is not fully machined because the diameter of the cutter used is too

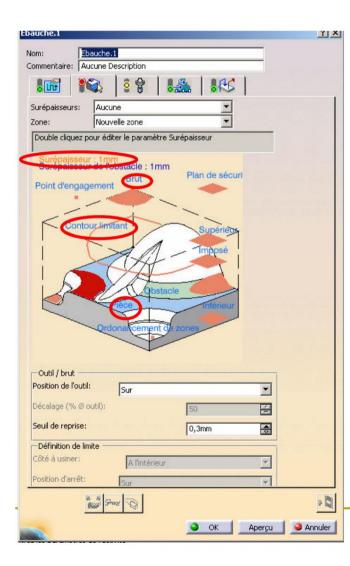
large. To remedy this, we will create a recovery zone.

The choice of a range by decreasing tool diameter having been made, this step will be done after the machining of the stripped pocket.

Sketch Semi-finishing Finishing

Click on the Draft icon.



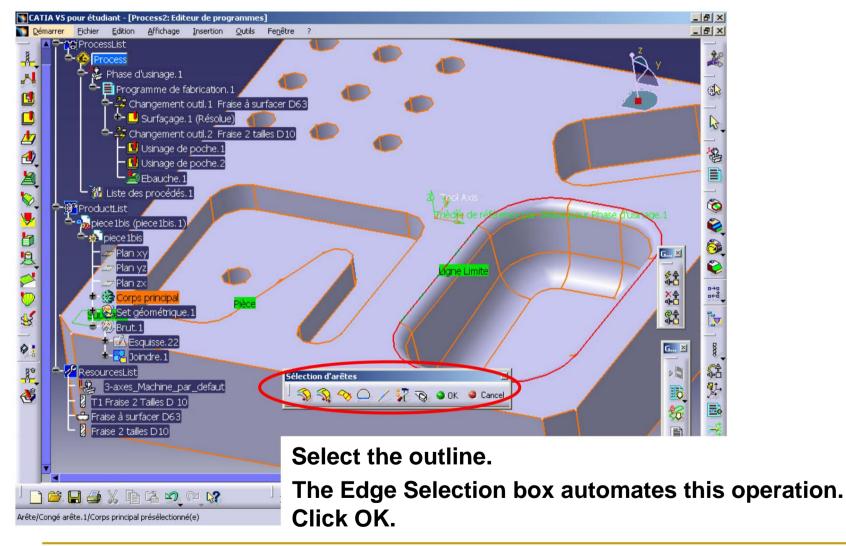


Double-click Stock. Set the Stock allowance to 1 mm.

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

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

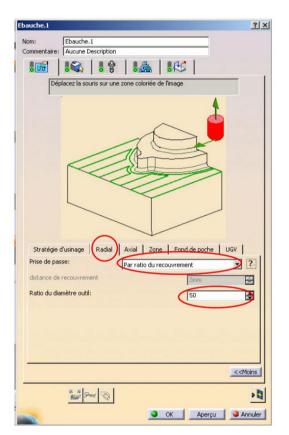
Click on Limiting contour. The limiting contour is the intersection of the surfaces of the drafted pocket with the top face of the part.



Ebauche.1	×
Nom: Ebauche.1	
Commentaire: Aucune Description	
Déplacez la souris sur une zone coloriée de l'image	
Stratégie d'usinage Radial Axial Zanc Fred de poche UGV	
Mode de helevinger.	
Tolérance d'usinage:	
Sens d'usinage: En avalant 💽 🥐	
Mode d'usinage: Par Zone Extérieur et poches 2	
Progression hélicoïdale: Mixte ?	
Rester sur le fond ?	
Contournage ?	
< <moins< td=""><td></td></moins<>	
Sec 3	
OK Aperçu S Annuler	1

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

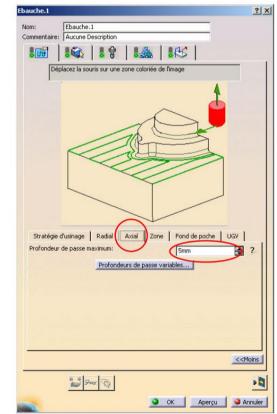
Click on the question marks to understand each of the functions offered



Click on the Axial tab. Set Maximum Depth of Cut to 2 mm.

Click on the Radial tab. Select By Recovery Ratio. Set the engagement width to 50%

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



Eba	uche.1			
nentaire: Auci				
	an a Margara		• 1	
1			>	
estion des Macr		1		
acro auche automati	Statut ique A jour	Nom Ebauche automati	Mc Hế	
ajets de départ	A jour	Trajets de départ.		
ajets de fin	A jour	Trajets de fin.1	CO	
		1	F	
acro en cours -				
-	ptions			1
ode: Hélice				
Optimisation (du retrait			
istance de secu	rite axiale:	5mm	-	
ngle de rampe:		5deg		
iamètre d'hélice	(Pourcentage du d	diamètre): 70		
iamètre de l'out	il de perçage:	25mm		
ngle de l'outil de				
		30deg		
ongueur de l'ou	til de perçage:	100mm		
istance d'appro	che:	10mm		
	rité radiale:	3mm	2	
istance de sécu		Louin		
istance de sécu				
istance de sécu				
	\$ Swx 0			

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

Activate trajectories if necessary.

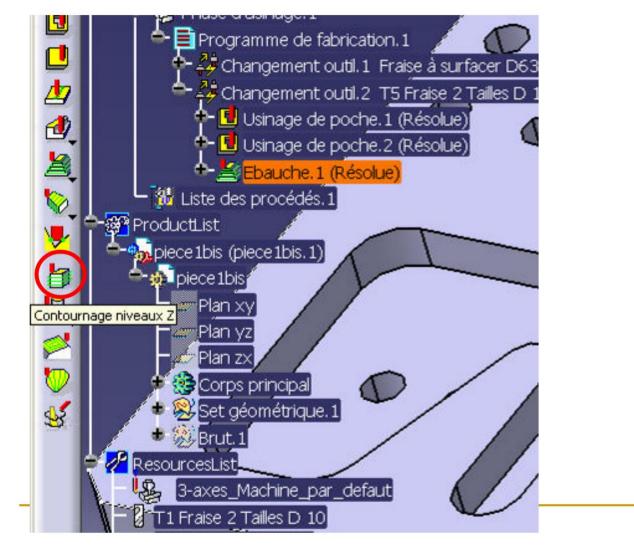
Click on the tool path animation icon to view the programmed paths. Click OK.

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.



Commentaire: Aucune	urnage niveaux Z.1 a Description	-
	icune 🔽	
	r une zone coloriée de l'image	
Surépaisseu Surépaisseu	r : 0mm de l'obstacle : 0mm Plan de sécurité	
Point d'en	gagementContour limitant	
	Supérieur	
6	VA_	
R	Obstacle	
S 1 79		
	Pièce	
	Pièce	
	Plèce	
	Plèce	
Limite automatique Définition des limite		
10 10 10 10 10 10 10 10 10 10 10 10 10 1		
Définition des limite		
Définition des limite: Coté à usiner:	A l'intérieur	
Définition des limites Coté à usiner: Position d'arrêt:	A l'Intérieur Dessus Bout d'outil	_
Définition des limites Coté à usiner: Position d'arrêt: Mode d'arrêt:	A l'intérieur Dessus Bout d'outil	
Définition des limites Coté à usiner: Position d'arrêt: Mode d'arrêt:	A l'Intérieur Dessus Bout d'outil	_
Définition des limites Coté à usiner: Position d'arrêt: Mode d'arrêt:	A l'Intérieur Dessus Bout d'outil	_
Définition des limites Coté à usiner: Position d'arrêt: Mode d'arrêt:	A l'Intérieur Dessus Bout d'outil	_

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

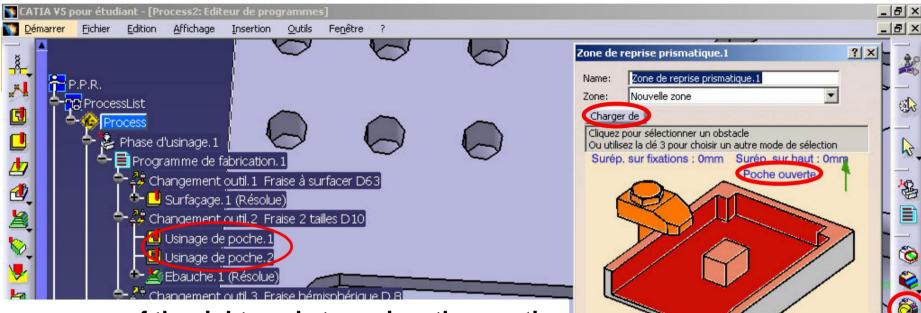
Contournage	e niveaux Z.1	<u>? ×</u>
Nom:	Contournage niveaux Z.1	
Commentaire	Aucune Description	
Dé	éplacez la souris sur une zone coloriée de l'image	
Stratégie	e dusinage	
Prise de pa	asse: Hauteur de crête	•
Passe max	kimum:	2
Passe mini		2
Hauteur d		2
		<moins< td=""></moins<>
	See O	•
		Annuler

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.



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.

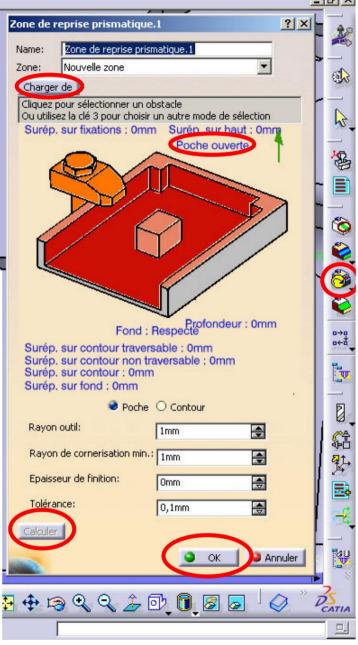
Click the Load From button.

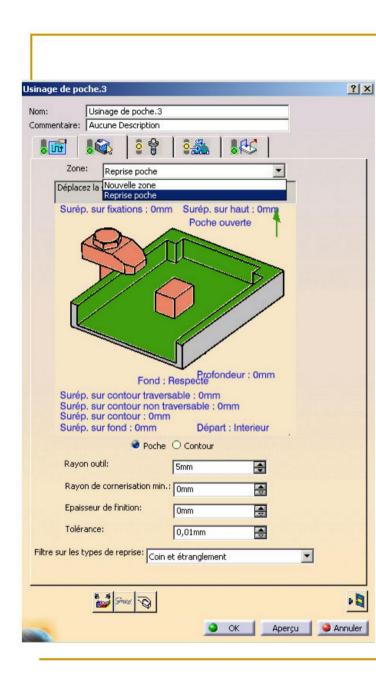
Click on the Pocket machining item in the Process tree.

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.





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.

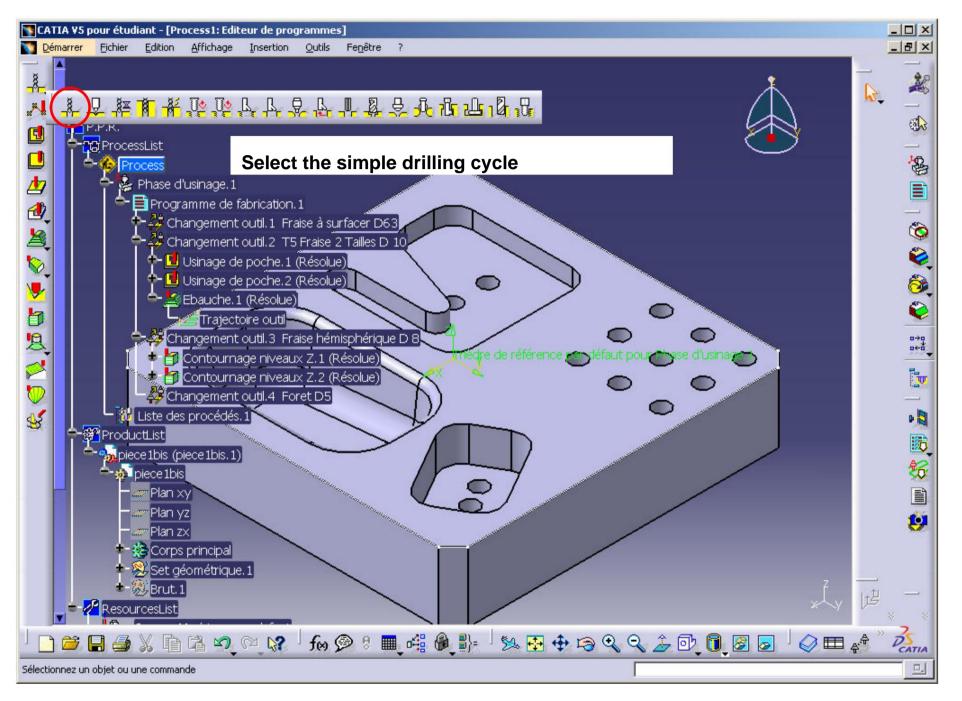
Set the engagement width to 50% of the tool diameter.

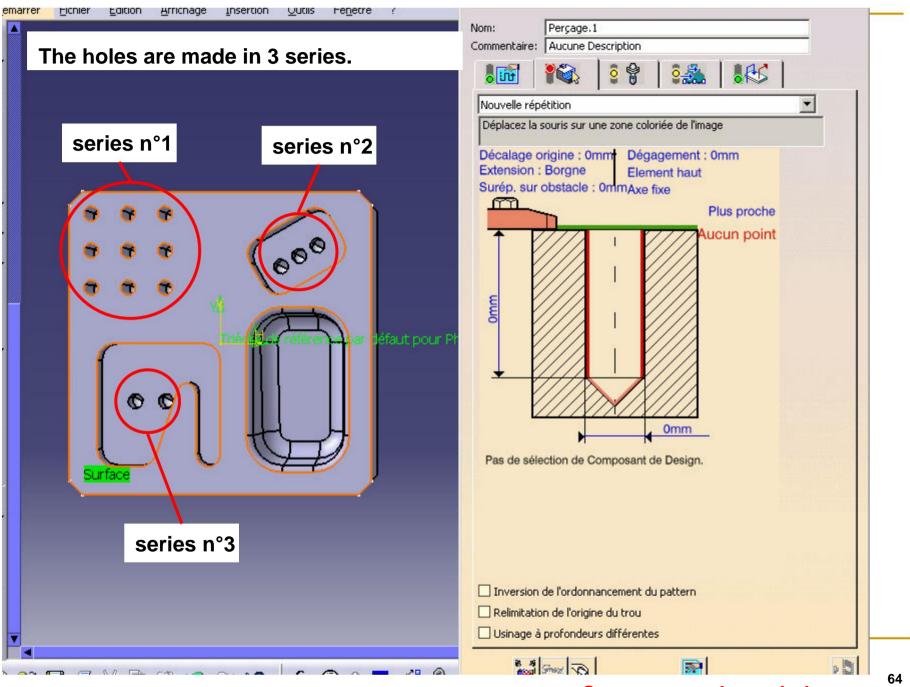
Click on the Axial tab.

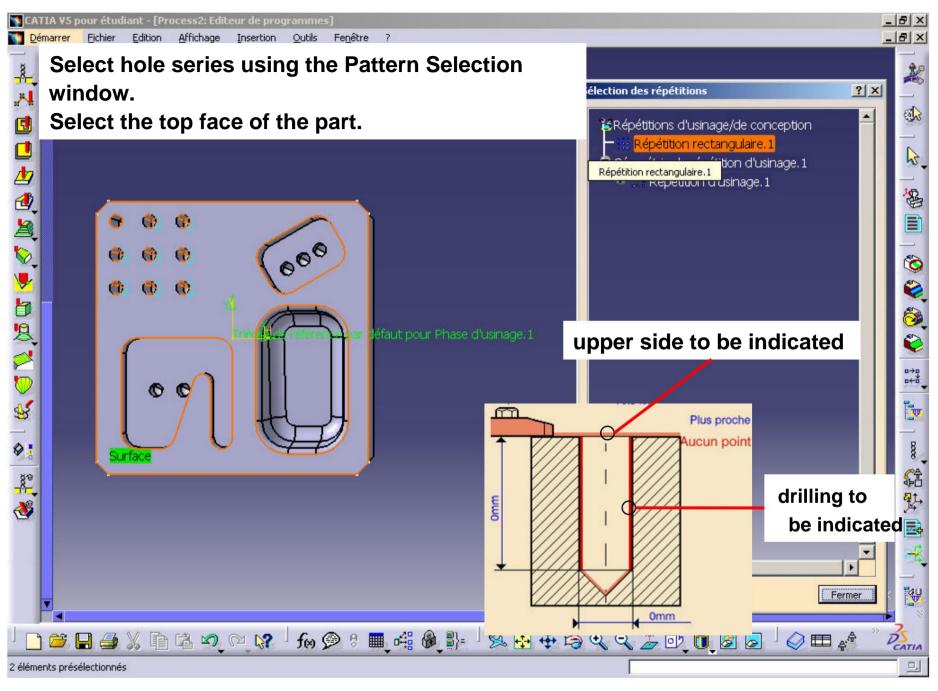
Set the number of levels to 2.

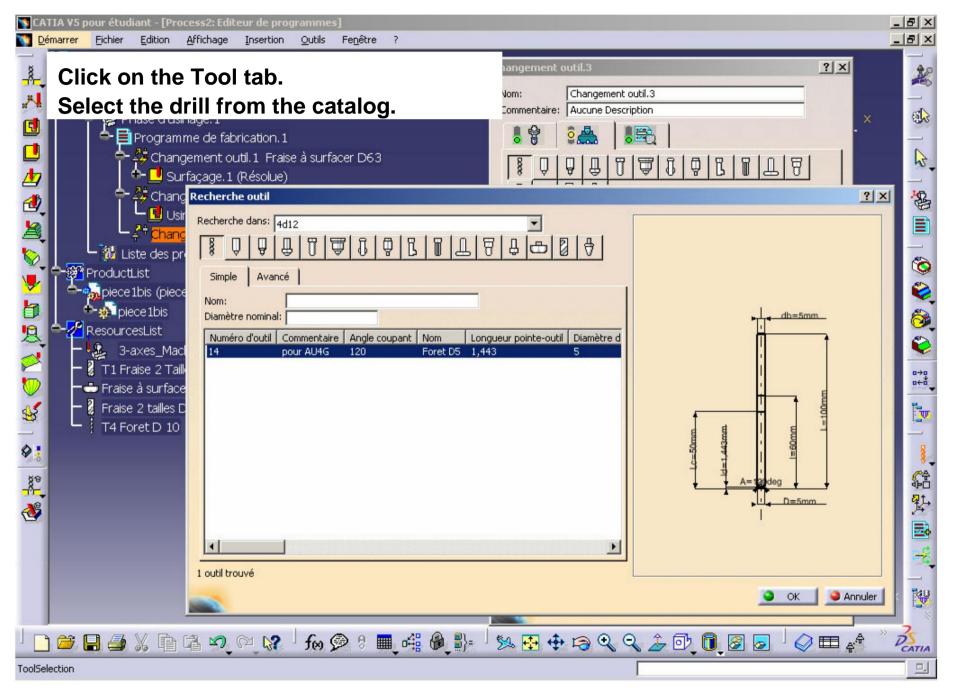
Check speeds and docking and retrieval conditions.

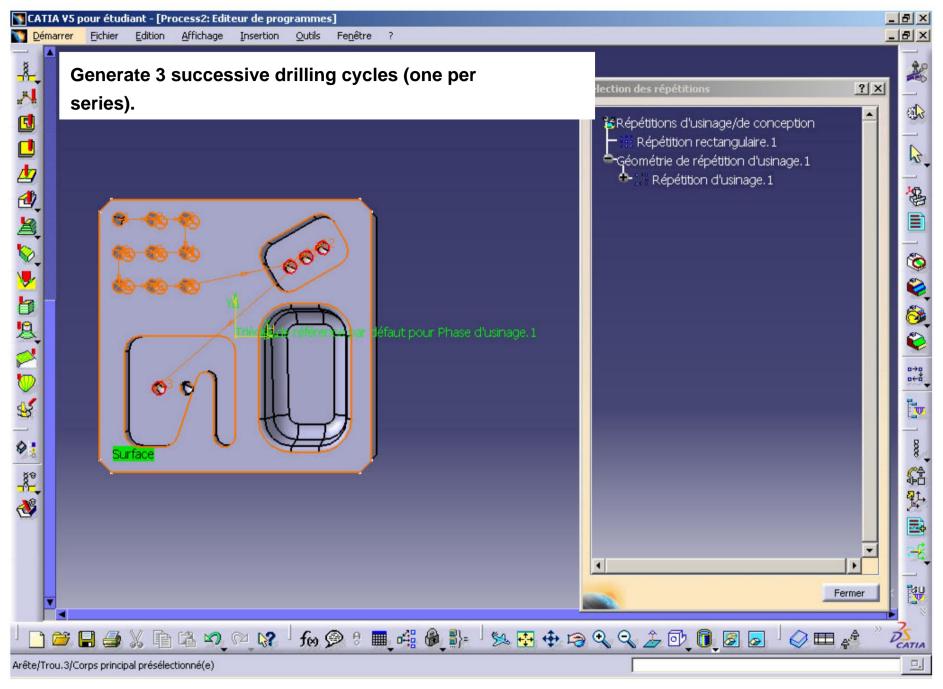
Click on the tool path animation icon to view the programmed paths.











```
Drilling cycles
```

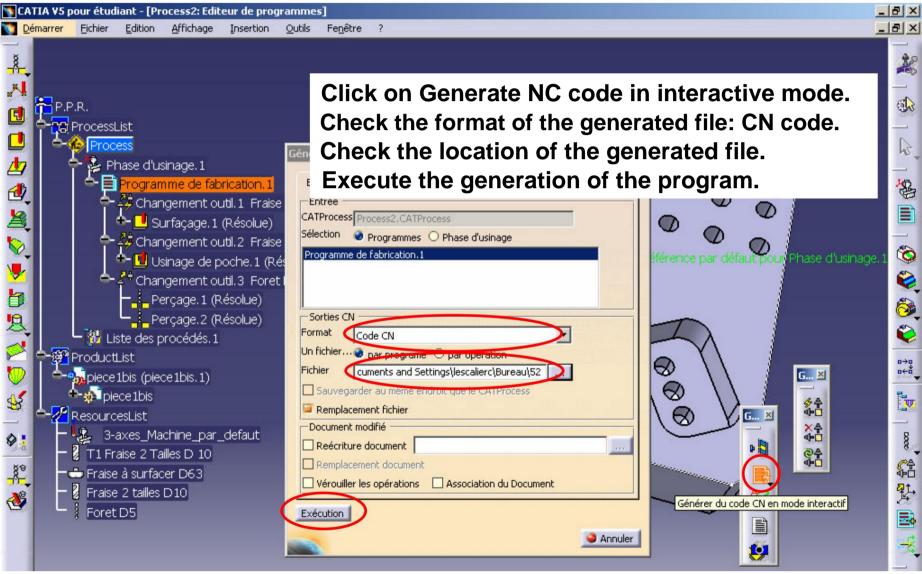
Click on the tool path animation icon to view the programmed paths.

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

NC program generation



Save your work regularly

69

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). **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

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 postpro, the ISO code, etc.)

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

Preliminary step

OP		? ×
Nom du repère : OP		
	Cliquez pour sélectionner la position Puis sélectionnez un point ou un cercle	
Paramètres de sortie CM	Groupe : 1	
	<u> </u>	Annuler

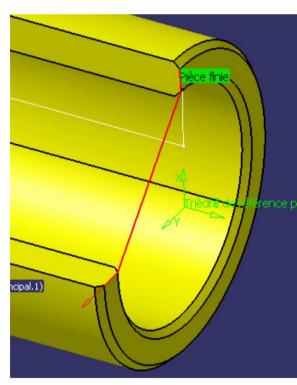
Define the reference triad Click on the origin then on the circular edge of the finished part (so that the origin of the coordinate system is at the center of the right face of the finished part)

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 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

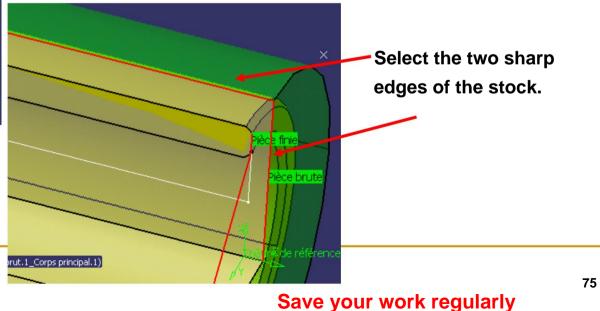
In turning, the definition of the areas to be machined is done by selecting edges and not cylinders, so it is necessary (this is already done here) to cut the part. This can be done either using a pocket, a cut or more simply by generating the part in CAD by revolution of 270° instead of 360°. If the 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°...

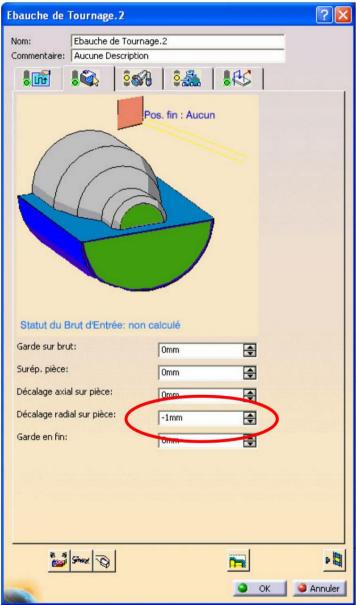


Launch a draft shoot



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.

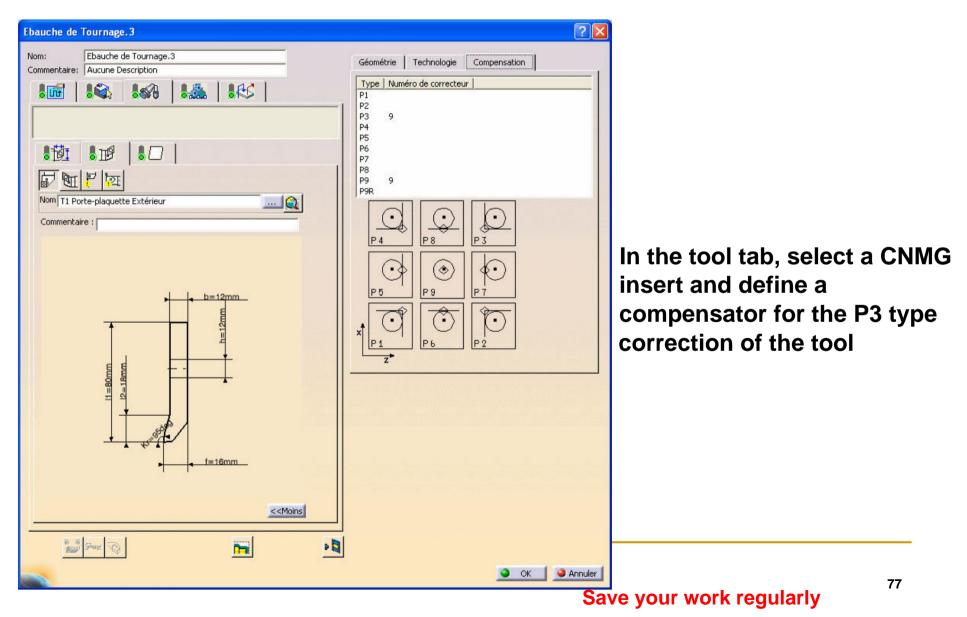


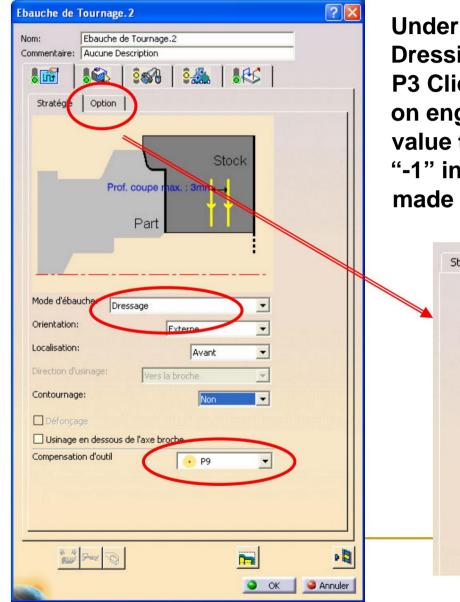


Select a radial offset on the part of "-1 mm".

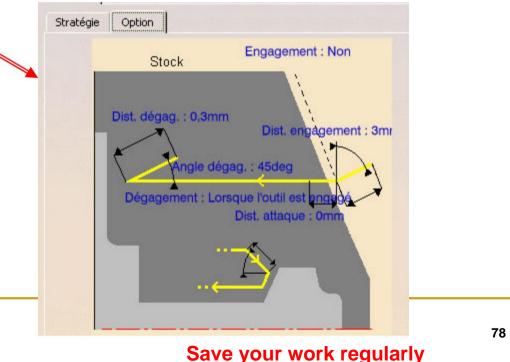
This will completely remove the material on the part (do the simulation at 0 and at -1 to see the difference). Depending on the tool used, there is material left in the center of the part.

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.





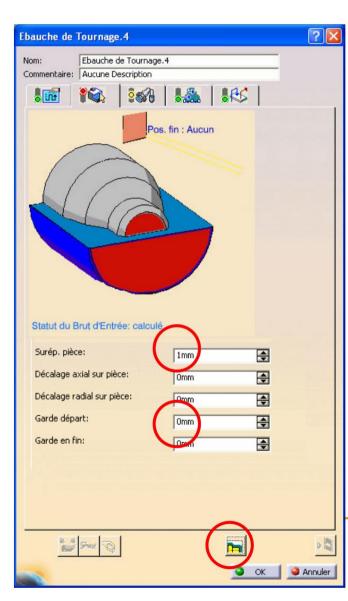
Under the machining strategy tab, select Dressing The tool compensator to use is P3 Click on the option sub-tab Double click on engagement distance and increase the value to 3 mm. This compensates for the "-1" in radial. The paths in this diagram are made at working speed.



bauche de Touri	age. 3	?
Jom: Ebau	the de Tournage.3	
Commentaire: Aucu	ne Description	
Avance		
	ue à partir des valeurs de l'outillage	
Engagement :	0,3mm_turn	
Plongée :	0,3mm_turn	
Usinage :	0,1mm_turn	
Dégagement :	0,8mm_turn	
Finition :	0,1mm_turn	
Remplacer RAPI	DE par avance hors matière	
Avance hors matièr	: 1000mm_mn	
Vitesse de broche		
🧧 Calcul automatic	ue à partir des valeurs de l'outillage	
📁 Sortie broche		
Usinage: 70turn_r	n 🗄 📸	
Unité: Angulaire		
Temporisation —		
Mode de temporisal	ion : Aucun	
<		
Qualité: Finition		
e a Seat		Þ
	4	OK SANNULER
		- Hindici

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

Paraxial draft



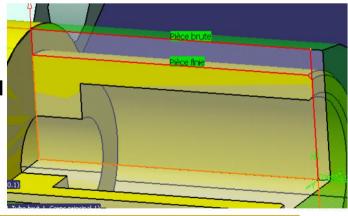
The paraxial roughing will make it possible to obtain the external diameter

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.



Paraxial draft

Ebauche de Tournage.4	? 🛛
Nom: Ebauche de Tournage.4	
Commentaire: Aucune Description	
Stratégie Option	
Stock	
Prof. coupe max. : 1mm	
Part	
Mode d'ébauche: Chariotage	
Orientation:	
Localisation:	
Direction d'usinage: Vers les mors	
Contournage: Sur chaque pa	
Défonçage	
Usinage en dessous de l'axe broche	
Compensation d'outil P3	
🔊 500 🕥	P 🔛
	Annuler

This operation is done with the same tool as dressing

Under the Machining Strategy tab, select "Tilling"

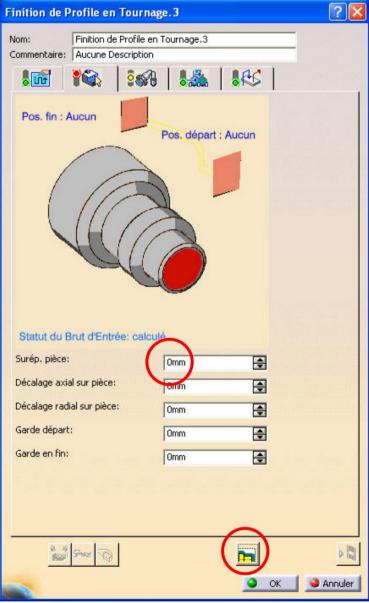
Maximum cutting depth is 1mm The tool compensator to use is P3

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

Turning profile finish



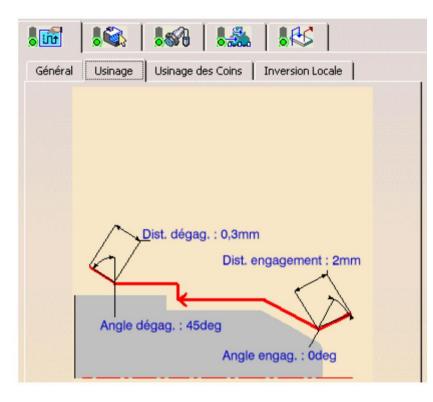
Start a profile finishing operation, always with the same tool

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

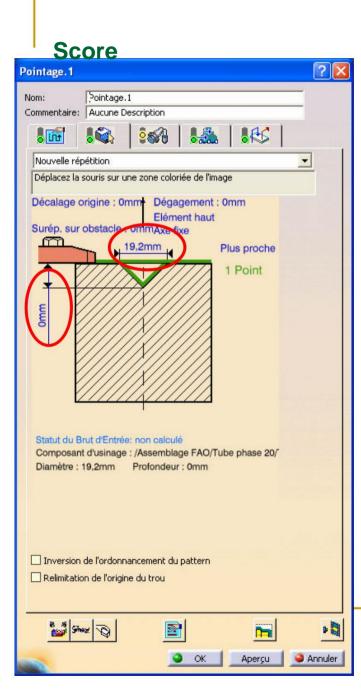
Turning profile finish



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



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

Drilling

-

Perçage pro	fond.1	?
Nom: Commentaire:	Perçage profond.1 Aucune Description	
	14	
Nouv	velle répétition	-
Déca Exter Suré We Statu	alage origine : Omm Dégagement : Omm nsion : Borgne p. sur obstacle : OmmAxe fixe Plus proche Aucun point ut du Brut d'Entvée: non calculé de sélection de Composant de Design.	
1 43 1	de selection de Composain de Design.	
	i de l'ordonnancement du pattern 🗌 Usinage à profonde on de l'origine du trou 👘 Usinage borgnes et	
8 8 9		

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

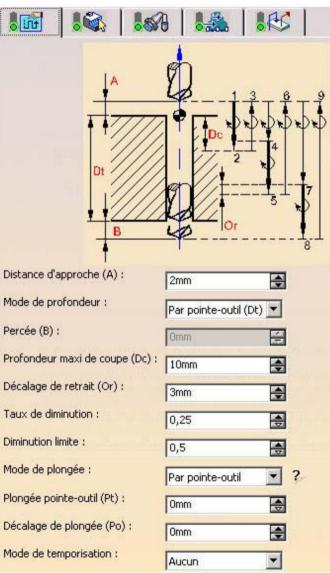
Selections are similar to the pointing operation.

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

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

bore

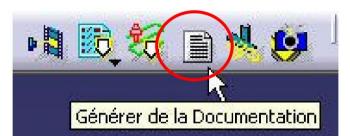
It remains to make the internal

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)

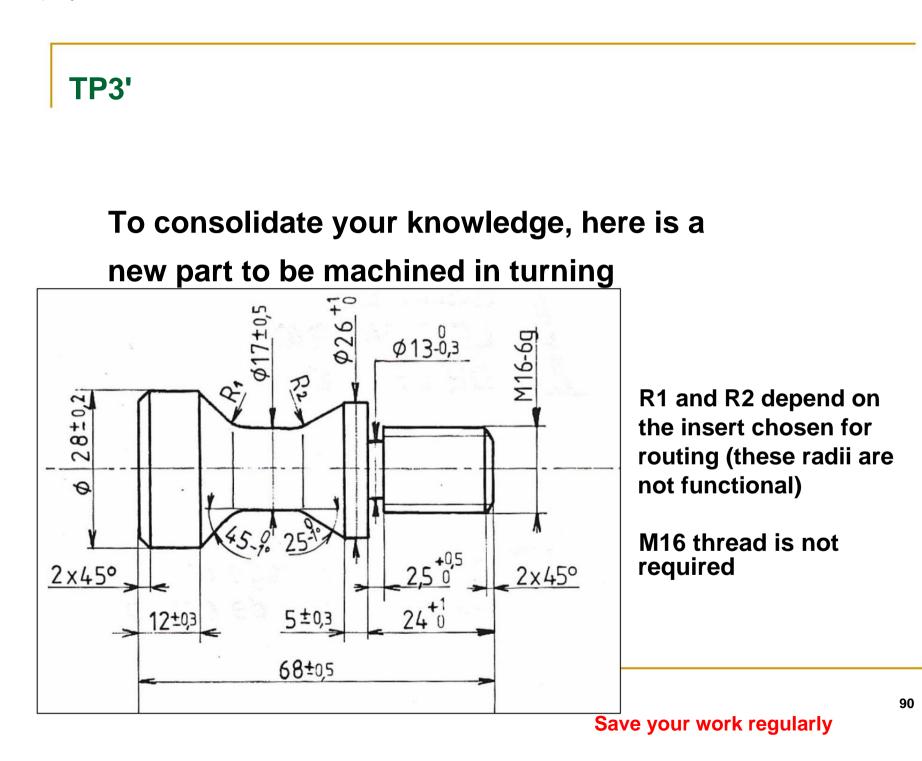
Each type of operation can be configured in a very complex way (angle of engagement, release, etc.)

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'

