

TABLE OF CONTENTS

Introduction	1
CATIA Version 5 Manufacturing	1
Types of NC Machines	2
Three Axis Machines	2
Multi Axis Machines	4
Lathes	4
Machining Modes	5
Milling Modes	5
Facing Mode	5
Pocketing	5
Contouring	5
Curve Following	5
Axial Modes	6
Drilling	6
Spot Drilling	6
Drilling Dwell Delay	6
Drilling Deep Hole	6
Drilling Break Chips	6
Tapping	6
Reverse Threading	7
Thread Without Tap Head	7
Boring	7
Boring and Chamfering	7
Boring Spindle Stop	7
Reaming	7
Counter Boring	8
Counter Sinking	8
Chamfering 2 Sides	8
Back Boring	9
T-Slotting	9
Circular Milling	9
Thread Milling	9
NC Tools	10
Facing Tool	10
End Mills	11
Center Drills	12
Spot Drills	13
Drill	14
Countersink	15
Reamer	16
Boring Bar	17
Tap	18
T-Slotter	19
Multi-Diameter Drill	20
Two Sides Chamfering Tool	21
Boring and Chamfering Tool	22

Conical Mill	23
Thread Mill	24
Milling Directions	25
Conventional Milling	25
Climb Milling	26
Prismatic Machining Workbench	27
Specifications Tree	27
Toolbars	29
Preparing to Machine	31
Part Design Review	31
Measurement Review	35
Assembly Review	38
Part Operation Setup	43
Defining the Part Operation	43
Basic Machining	51
Facing	51
Geometry tab	53
Edge Selection	55
By Belt of Faces	56
By Boundary of Faces	56
Sectioning	56
Machining operation parameters tab	59
Tool tab	63
Speeds and Feeds tab	64
Macros tab	64
Replaying	67
Contour Milling	71
Profile Contouring Methods	71
Between Two Planes	72
Between Two Curves	73
Between Curve and Surfaces	73
By Flank Contouring	74
Simulating the Replay	80
Manually Defining Tool Changes	81
Pocketing	87
Hard Bottom, Closed Pocket	87
Soft Bottom, Closed Pocket	88
Hard Bottom, Open Pocket	88
Soft Bottom, Open Pocket	89
Curve Following	92
Point to Point	97
Axial Machining	101
Spot Drilling	102
Drilling	107
Drilling Pre-Defined Patterns	110

Drilling Deep Hole	113
Drilling Break Chips	116
Countersinking	118
Counterboring	119
T-Slotting	120
Multiple Part Operations	129
 Machining Axes	 137
 Machine Rotation	 144
 Advanced Machining Topics	 161
Copy Transformations	161
Circular Milling	167
Manufacturing Knowledgeware	170
Post Processor Instructions	178
NC Documentation	182
NC Code	186
APT Code Generation	186
Generating Post Processed Code	191

Introduction

CATIA Version 5 Manufacturing

Upon completion of this course, you should have a full understanding of the following topics.

- Build stock material for a finished part
- Define Part operations in a machining process
- Define machining operations in a machining process
- Replay the machining operations, visualizing the material removal
- Modify part geometry, fixing machining operations to reflect changes
- Generate Apt code from machining operations

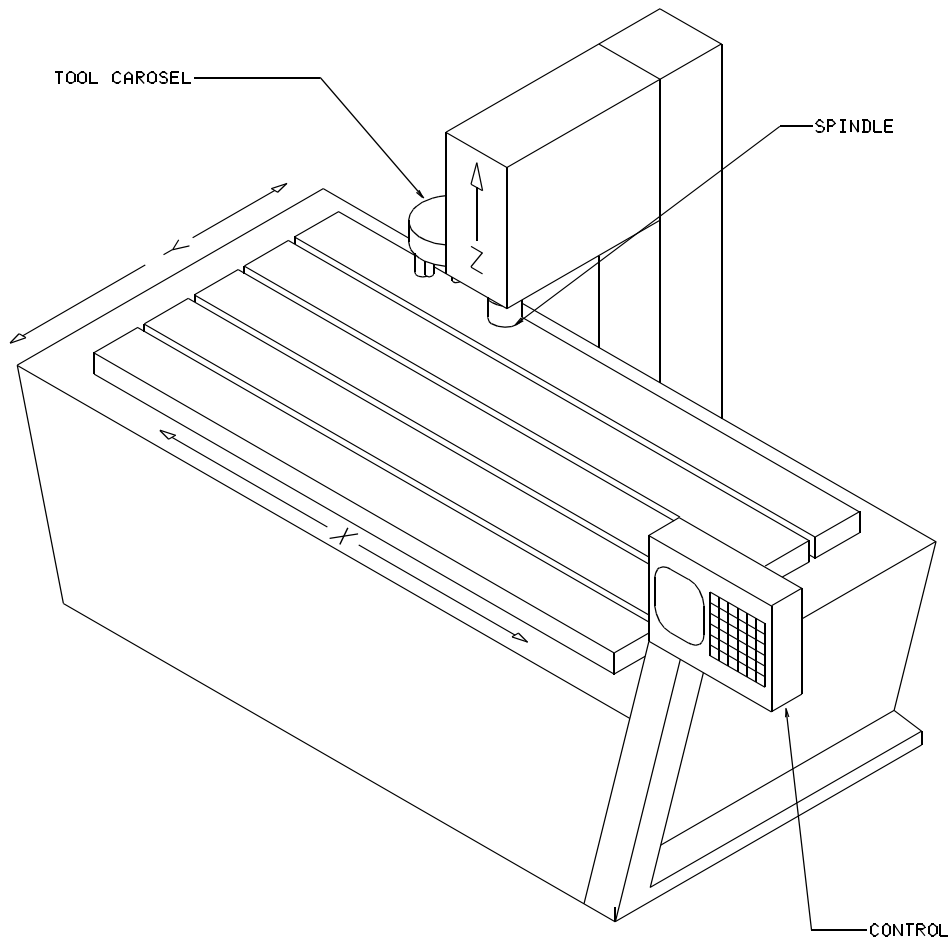
Designing and drawing parts is an important part of any company process. However, just designing the part does not make the airplane, automobile or any other product leave the assembly line. The parts for the assemblies must be manufactured. The manufacture of three axis machine parts will be the emphasis of this course. It will be assumed that you are proficient in the Part Design, Sketcher, and Assembly Design workbenches. If you feel your skills are not what they should be, or if you have trouble in some sections of this manual, you may want to look back at your Part Design and Sketcher or Assembly Design books for review.

Types of NC Machines

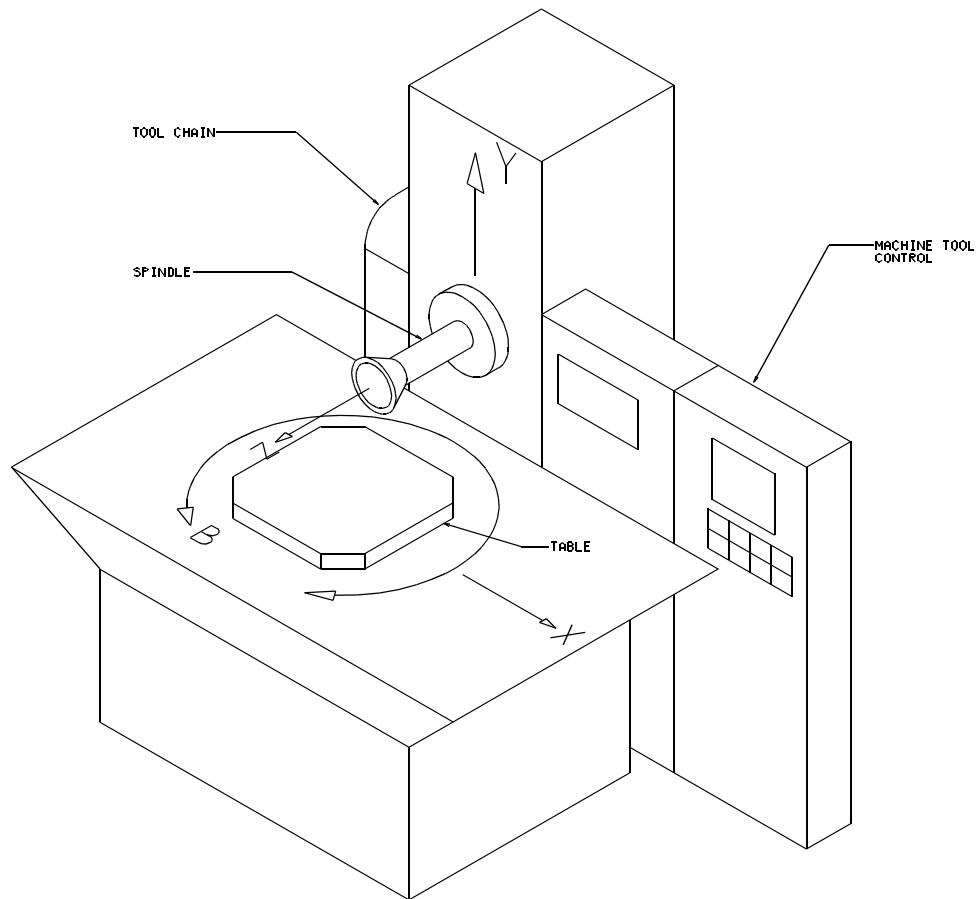
There are many different types of CNC machines used in production lines all over the world. The prismatic machining portion of CATIA V5 concentrates on a few different types. Those types will be highlighted here, although not all machine types will be used in programming.

Three Axis Machines

Three axis machines are most commonly used for simple parts. Three axis machines come in two styles, vertical and horizontal machining centers. Vertical machines have the tool axis locked along the Z axis. The X axis generally points the length of the table, while the Y axis runs forward and aft on the table. Several tools are usually carried in a carousel near the head of the machine.

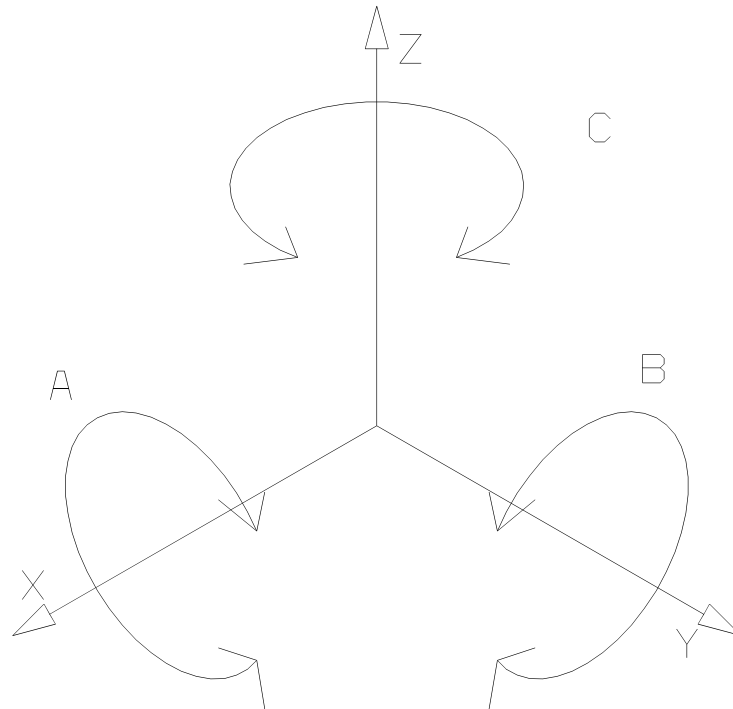


Horizontal machines work in a similar fashion. The Z axis of a horizontal machine still runs along the tool axis, while the Y axis points along the machine arm, and the X axis runs along the table. It is very common to find another axis on a horizontal machine. A rotation axis is commonly found on the table.



Multi Axis Machines

There are three rotation axes associated with the three Cartesian axes (X,Y,Z). The three rotation axes are A, B, and C, all respectively associated with X, Y, and Z. It is not uncommon to find CNC machines with one, two, or even all three rotation axes. Machines with more than one rotation axis are commonly considered multi axis machines. The most common multi axis machine is a five axis machine that has the three X, Y, and Z directions, as well as A and B rotational components. Multi axis machines are generally more expensive to operate, and keep operational, therefore are only used when necessary.



Lathes

Horizontal and vertical lathes are another type of machine that can be programmed for in CATIA V5. Lathes are most generally used for making round, or round shaped, parts. This is due to the nature of the lathe. The stock material is held in a set of grips at each end and then the material is spun as a tool cuts.

Machining Modes

There are two different types of machining modes. There are axial modes, which include drilling, reaming, and tapping, basically making the machine behave as a drill, and then there are milling modes. This mode includes pocketing, facing, and contouring motions. For each type of mode there are specific tools that are used. You will look at each of the modes in a bit more detail.

Milling Modes



Facing Mode

Facing is a machining mode where excess material is removed from the top of the finished part. In most cases, a face mill is used due to its large size and ability to remove a lot of material quickly. Face mills come in many different sizes and shapes. Some look like a large end mill, while others seem to have more of a “shell” shape, giving them the name of shell mills.



Pocketing

Pocketing is where the milling machine will cut out material within an enclosed area. Generally the cutter is ramped into the pocket and then the tool will clear out a level. Depending on the depth of the pocket, the milling machine may make several levels before reaching the bottom of the pocket. Pockets can also be open on the bottom, similar to a large hole in the part.



Contouring

Profile contouring is where the milling machine will cut the profile or around a guide curve on a part. Contour milling will make several radial passes, as well as a number of necessary axial passes as needed. Contour milling will usually be used when the outside of the part is needing to be machined.



Curve Following

Curve following is one of the more simplified modes. Curve following mode drives the tool along any given curve in the workspace. The curve does not have to lie on a support of any kind, and can either be in a sketch, or wireframe geometry. The most simple type of curve following is point to point. This is a separate icon but works in a similar fashion. After points are defined then the tool will make straight paths between the points.

Axial Modes



Drilling

Drilling is the most basic of the axial modes. Drilling makes the machine act as though it were a large, automatic drill press. Drilling is used for holes that vary from very small, through a moderate size. If a large hole (several inches in diameter) is desired, a circular milling motion or pocket operation is utilized instead.



Spot Drilling

Spot drilling is usually used before a drilling operation is performed. Spot drilling creates a small hole in the center of the desired hole. This keeps the tool from “walking” away from the center of the hole.



Drilling Dwell Delay

Drilling dwell delay will drill a hole in the same fashion as a standard drilling operation but will delay or stop when it is inside the hole. This allows the tool time to completely finish a hole, before retracting and starting a new one. A delay at the bottom of the hole generally results in a smoother hole cut than a standard drilling motion.



Drilling Deep Hole

Drilling deep hole is used when a large, deep hole is desired. The tool is drilled into the material a set distance, a dwell time can be added, then the drill is completely retracted. The drill is then re-inserted into the hole, drilled a bit further. The process is repeated until the hole is drilled to the bottom or drilled clear through.



Drilling Break Chips

During a drilling break chips operation, the drill bit is drilled partially into the material, then it is reversed and then drilled further. This allows the chips bound in the drill bit to be removed, thus breaking away any excess chips. This keeps the drill from overheating and keeps the chips from binding around the drill bit.



Tapping

Tapping is the process where threads are cut into a hole. Generally a tapping motion is for holes that are not too excessive in size. Large holes have a different method of creating threads. A tapped hole allows for bolts or pipes to be screwed into the part.



Reverse Threading

Reverse threading is the same as a tapping motion, with exception that the threads are cut by an opposite handed cutter.



Thread Without Tap Head

Threading without a tap head is generally used for a larger hole or where the threads are not a common pitch angle. Many times a tap will not be big enough for a hole, so a tool that has a single tooth will be used. The tool is spun around, cutting the thread as it is being pressed through.



Boring

This is the standard boring operation. Many times a hole needs to be perfectly round and straight, therefore a boring bar will be ran though the hole to insure that it is straight and round.



Boring and Chamfering

In a boring and chamfering operation, a specialized boring and chamfering tool will be used. The operation will create a hole with a chamfer at the top of the hole. This type of the hole will generally be used for a flush mount bolt or screw.



Boring Spindle Stop

A boring spindle stop operation is the same type of operation as a boring operation, but the spindle will stop when it reaches the bottom of the hole.



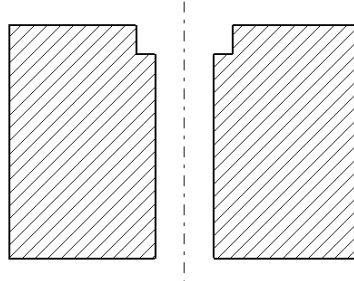
Reaming

A reaming operation is a method of finishing a hole. A reaming operation will remove any burrs or chatter marks from a hole.



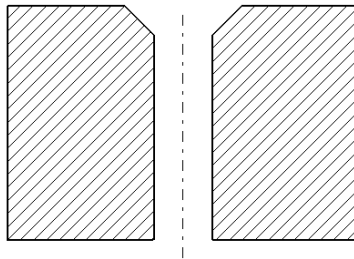
Counter Boring

A counter boring operation is an operation designed to drill out counter bored holes. A larger tool is used to drill down part way in a pre-defined hole, allowing for bolts to set below the surface of the part. The bottom of a counterbore is generally flat.



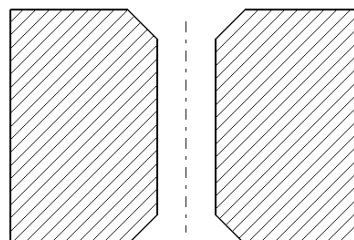
Counter Sinking

A counter sinking operation will use a specialized tool to angle, or chamfer a hole to allow for a screw or rivet to set below the surface. The tool is drilled partially into the material and then removed to create the counter sink.



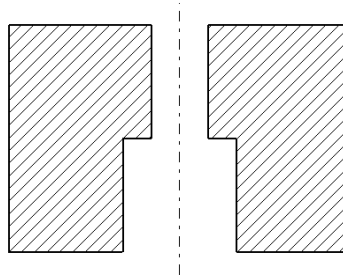
Chamfering 2 Sides

A chamfering two sides operation is another specialized chamfering operation. The chamfer 2 sides hole is similar to a counter sunk hole, with exception that the chamfer is on both sides instead of just one. A two sided chamfering tool is used to chamfer the top of the hole, then the spindle is spun at a different rate, or stopped, as the tool is pushed through the hole, not cutting the material. When through the hole, the tool is then spun again and the other side is chamfered.

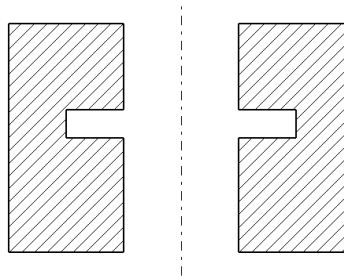


 **Back Boring**

A back boring operation is similar to that of a boring operation, with the exception that the tool is pushed through the hole, and then the boring begins from the bottom of the hole. Many times the back boring hole will not be milled completely through the part.

 **T-Slotting**

A T-Slot is a specialized motion where a t-slotting tool (looks similar to a small saw blade) is put in a hole, and then a circular motion is made with the t-slotted cutting. This creates a slot inside a hole.

 **Circular Milling**

Circular milling is used for very large holes. A circular milling operation is very similar to that of a pocket operation, but is specialized for a circular hole. Generally an end mill will be used to circular mill a large hole.

 **Thread Milling**

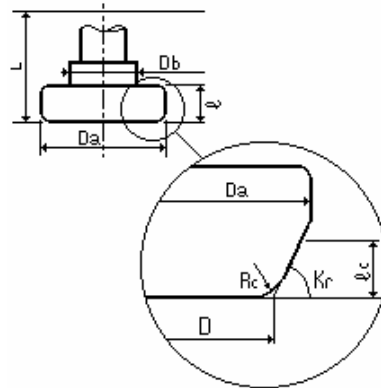
Thread milling is used for large holes that need to be threaded. A cutting tool will have a finger, or series of fingers on the side that spin at a high RPM, and then the tool is moved in a downward helical motion to cut all of the threads in the large hole.

NC Tools

The following will look at the various tools that can be used in the previous milling modes. Not all tools are available for all different modes.

Facing Tool

A facing tool is used for cutting a large amount of material fast. Face mills are generally only available when creating a facing operation.

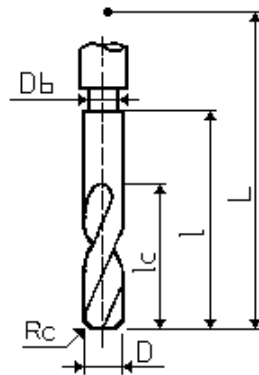


Where:

- D nominal diameter of the cutter
- D_a outside diameter of the cutter
- l length of the base of the cutter
- l_c cutting length of the cutter
- R_c fillet radius around the bottom of the cutter
- K_r cut angle of the cutter
- D_b body diameter of the cutter
- L overall length of the cutter

End Mills

End mills are used for most milling applications. End mills have several different styles. One style of end mill is a flat end mill. In this type of end mill, the bottom of the cutter is flat and generally has cutters. The second type of end mill is a filleted end mill. This type of end mill also has a flat bottom but is filleted along the bottom edge instead of being sharp. This allows for better and faster milling of semi-flat surfaces. The third type of end mill is a ball nose end mill. Ball nose end mills are used for surfaces that have a lot of contour. A ball nose end mill is completely rounded on the end with no flat surfaces.

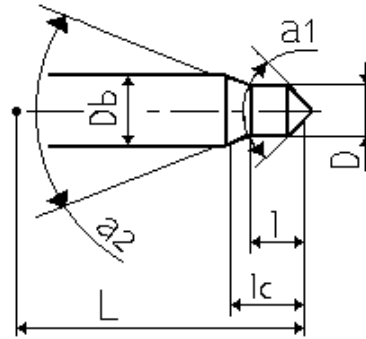


Where:

- D cutter nominal diameter
- L overall length of the tool
- l length of the body of the cutter
- l_c length of the cutting surface of the mill
- R_c corner radius
- D_b diameter of the shank

Center Drills

Center drills are used for piloting or creating a small center hole at the center of a larger hole. This will keep the larger drill from walking away from the center of the hole, thus creating a bad hole.

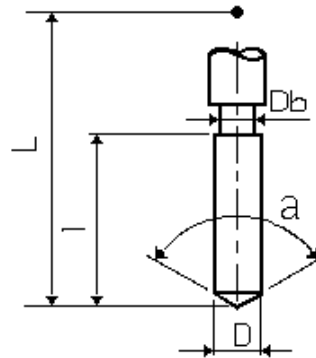


Where:

- D cutter nominal diameter
- L overall length of the tool
- l length of the body of the cutter
- l_c length of the cutting surface of the drill
- a_1 cut angle
- a_2 taper angle

Spot Drills

A spot drill is used for the same reason as a center drill. A spot drill will create a small hole to pilot the larger drill.

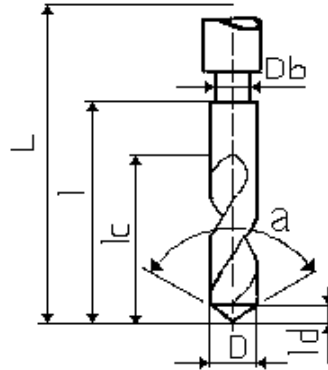


Where:

- D cutter nominal diameter
- L overall length of the tool
- l length of the body of the cutter
- Db body diameter
- a cut angle

Drill

A drill is used to cut a hole of various sizes. Drills have a number of different sizes and are used in a wide range of holes. Generally drills are used when a hole does not exceed an inch or two.

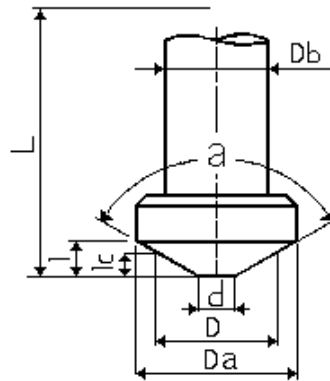


Where:

- D cutter nominal diameter
- L overall length of the tool
- l length of the body of the cutter
- Db body diameter
- a cut angle
- lc cut length
- ld tip length

Countersink

A counter sink tool is used for a counter sinking operation. The counter sink tool is drilled into the material a short distance, allowing for a counter sink screw to be used in the hole. This will keep the surface of the tool smooth and flush.

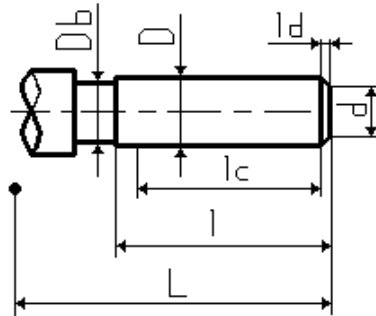


Where:

- D cutter nominal diameter
- D_a outside diameter
- D_b body diameter
- L overall length of the tool
- l length of the body of the cutter
- a cut angle
- l_c cut length
- d entry diameter

Reamer

A reamer is used for finishing out a hole. Many times the hole may be slightly out of round or possibly needs smoothed. This is where the reaming tool is used. The tool is ran through the pre-existing hole, cutting it smooth and clean.

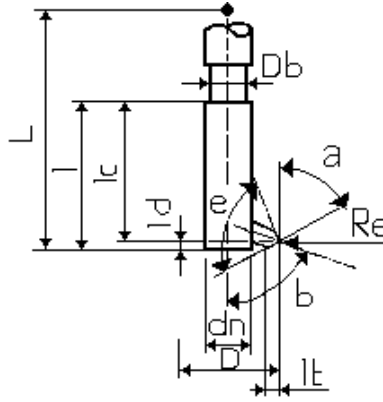


Where:

- D cutter nominal diameter
- D_b body diameter
- L overall length of the tool
- l length of the body of the cutter
- l_d tip length
- l_c cut length
- d entry diameter

Boring Bar

A boring bar is used when a hole needs to be a very exact and straight diameter. Boring bars are also used for larger or deep holes. There are quite a few parameters involved in the definition of the boring bar. Many of the parameters define the cutting tool at the tip of the cutter.

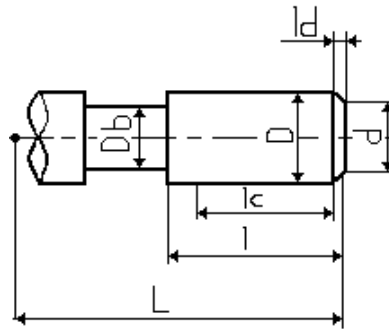


Where:

- D cutter nominal diameter
- Db body diameter
- L overall length of the tool
- l length of the body
- a cut angle
- lc cut length
- dn non cutting diameter
- lt tip length
- e tip angle
- Re tip radius
- b tool angle
- ld tool tip length

Tap

A tap is used for cutting threads in a smaller diameter hole. When the hole is less than an inch or two, it is threaded with a tap. If a larger hole is to be tapped, a different tool is utilized.

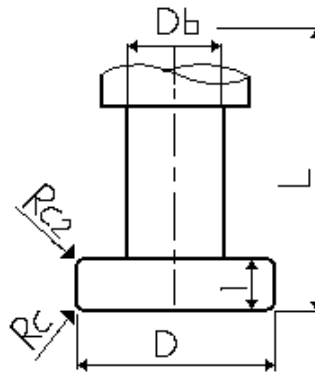


Where:

- D cutter nominal diameter
- D_b body diameter
- L overall length of the tool
- l length of the body of the cutter
- l_c cut length
- d entry diameter
- l_d tool tip length

T-Slotter

A t-slotting tool is used to cut a ring or a slot inside a hole. The t-slotter is inserted into the hole, then it is ran around the inside of the hole.

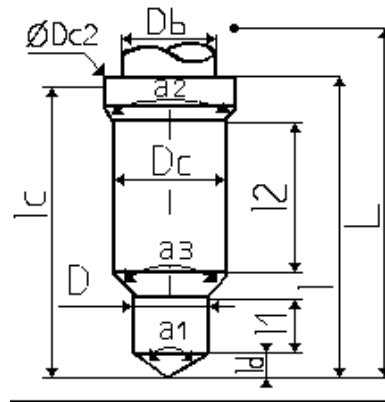


Where:

- D cutter nominal diameter
- D_b body diameter
- L overall length of the tool
- l length of the body of the cutter
- R_c bottom cutting radius
- R_{c2} top cutting radius

Multi-Diameter Drill

A multi diameter drill is a mix bread of a center drill with a standard drill. There is a smaller portion of the drill at the tip to drill the initial center of the hole and then behind the smaller drill is a larger drill portion that is used for drilling the remainder size of the hole. A yet larger drill section is at the top sometimes to allow for stepping up the size of the hole yet larger.

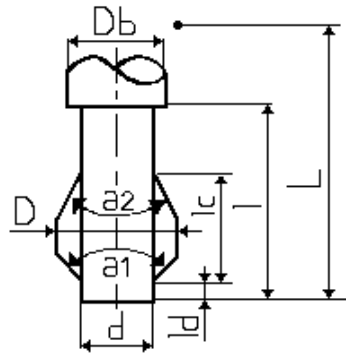


Where:

- D cutter diameter at the first stage
- D_c cutter diameter at the second stage
- D_{c2} cutter diameter at the third stage
- D_b body diameter
- L overall length of the tool
- l length of the body of the cutter
- l_c cut length
- l_d tool tip length
- l_1 length of the first stage
- l_2 length of the second stage
- a_1 tip cut angle
- a_2 top angle
- a_3 taper angle

Two Sides Chamfering Tool

A two sided chamfering tool is used specifically with the two sided axial motion.

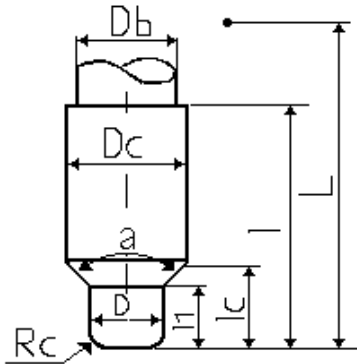


Where:

- D cutter diameter of the outside of the chamfer cutters
- Db body diameter
- d entry diameter
- L overall length of the tool
- l length of the body of the cutter
- lc cut length of the cutters
- ld tool tip length
- a1 lower chamfer angle
- a2 upper chamfer angle

Boring and Chamfering Tool

A boring and chamfering tool is a combination of a boring bar along with a chamfering cutter. The tool base is similar to that of a boring bar with exception of the cutting blade. The tool then has a chamfering angle that will cut the chamfer at the top of the bore.

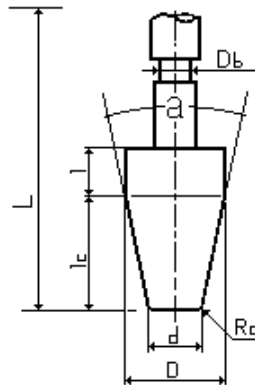


Where:

- D cutter diameter at the first stage
- D_c cutter diameter at the second stage
- D_b body diameter
- L overall length of the tool
- l length of the body of the cutter
- l_c cut length
- l_l boring bar length
- R_c corner radius on the boring bar
- a taper angle of the chamfer portion of the tool

Conical Mill

A conical mill is used when a drafting angle is desired on a part or pocket, yet only a three axis machine is used. A conical mill is tapered at a given angle with a given length.

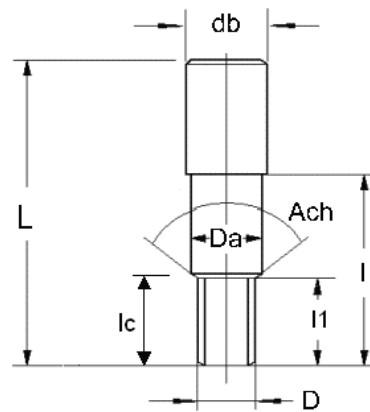


Where:

- D cutter diameter at the first stage
- d entry or bottom diameter
- D_b body diameter
- L overall length of the tool
- l length of the body of the cutter
- l_c cut length
- R_c corner radius
- a taper angle

Thread Mill

A thread mill is used when a large hole is needing to be threaded. The tool is spun about its axis and then swept in a circular motion around the hole to cut the threads.



Where:

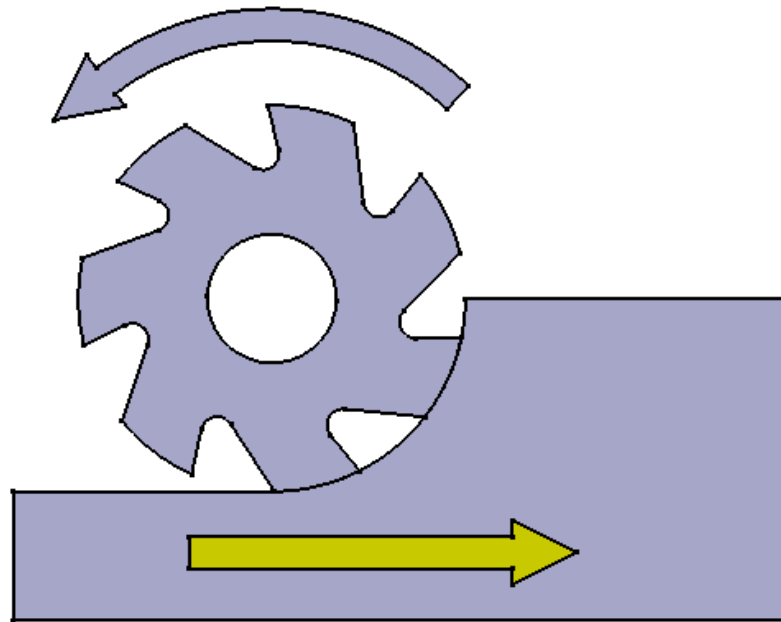
- D diameter of the cutters
- L overall length of the tool
- l length of the body of the tool
- l_c cut length of the tool
- Da outside diameter of the cutter
- db body diameter
- l_1 length of the thread cutters
- Ach taper angle from the cutters to the body of the tool

Milling Directions

You can mill in two different directions. The directions are defined as conventional and climb milling. The two different options will be discussed. A rule of thumb to determine if you are climb or conventional milling is to think of driving. As you drive along side the road, the curb is to your right. Think of the curb as your stock material. As the tool moves away from you (drives away from you) your stock material is on the right hand side of the tool. If you are climb milling the tool should be on the left of the stock as it moves away, providing you have a right handed cutter that is spinning clockwise.

Conventional Milling

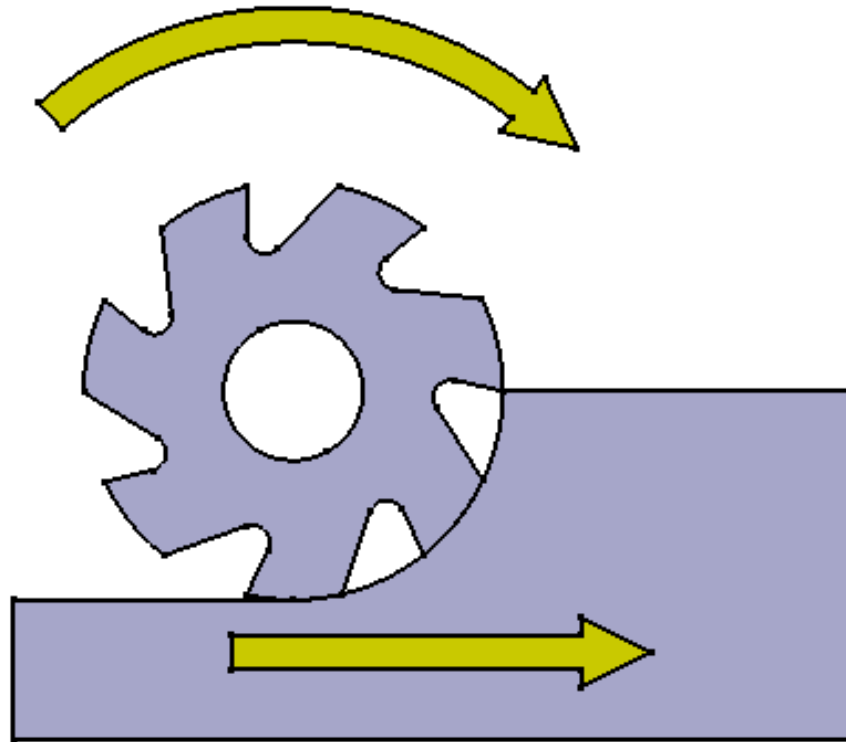
With conventional milling, chip thickness and thus cutting pressure gradually increases at the tooth as it engages and reaches a maximum prior to the tooth leaving the material. When the tooth leaves the material, the cutting pressure is suddenly eliminated so that the cutter "jumps" forward and the next tooth penetrates the material with a jerking action. This generally results in "chatter" marks. Conventional milling is however desirable when a large amount of material is desired to be removed quickly.



Conventional Milling

Climb Milling

With climb milling, the chip is formed in reverse sequence. When the tooth penetrates, the chip thickness and thus the cutting pressure is at a maximum. When the cutter leaves the work piece, the chip is at its thinnest and the cutting pressure is at minimum. This produces less "chatter" and a better finish. Climb milling requires less drive power but a more rigid table that is free of backlash. Because most of the newer CNC machines are more rigid and can control backlash, climb milling is preferable.



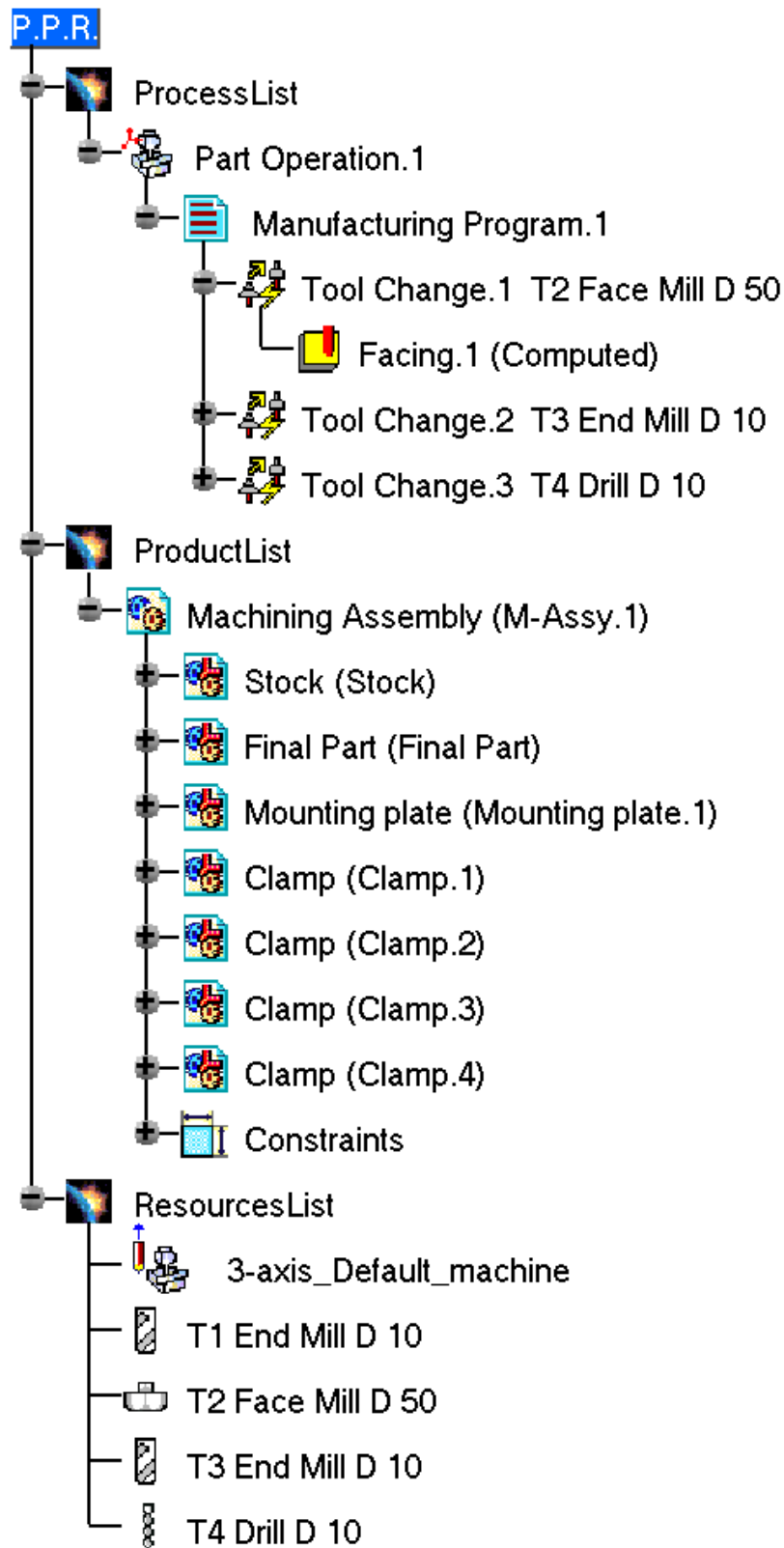
Climb Milling

Prismatic Machining Workbench





















Specifications Tree

The specification tree for Prismatic Machining is a bit different than what you have seen prior to this class. This does not imply that the tree is specific to the Prismatic Machining workbench only. The PPR style tree is applicable to several other workbenches within CATIA.

- P.P.R This is the Process Product Resource branch. This is the upper level branch in the tree and everything is held within this master branch.
- ProcessList This is where all processes are held in the PPR. For prismatic machining, you will only be concerned with part operations. There can be several part operations within one process list.
- Part Operation This is where the information about the part operation is held. Machine information, clearance plane, home point, stock and final part, as well as several other variables are defined in this tab. Be sure to set all settings within this tab before beginning to machine.
- Manufacturing Program This is the branch that handles all machining operations. There can be several manufacturing programs within each part operation. Within the manufacturing program, tool changes, machine rotations and machining operations are all defined.
- ProductList This is where the product or products that make up the process are kept. For prismatic machining, an assembly will be pre-defined that contains the stock material, final part, mounting plate, and any clamps necessary. This assembly will be the only product in the product list.
- ResourceList This is where any resources utilized in the process are defined. Any tools, machines and tool assemblies used in any of the processes defined above will be displayed here to give a visual reference as to the resources necessary to define the process.



Toolbars

		Inserts a new part operation in the Process List		Allows you to change the machine axis or origin
		Inserts a new manufacturing program into a part operation		Allows you to insert post processor commands directly in the part operation
		Inserts an axial motion into the part operation. See previous section for a complete list of axial motions.		Allows a translation or rotation copy to be made
		Inserts a pocket operation		Replays the part operation
		Inserts a facing operation		Allows generation of NC Code either interactively or in a batch mode
		Inserts a contouring operation		Generates documentation
		Inserts a curve following operation		Runs a macro to output the machining program to Virtual NC, a higher level NC verification tool
		Inserts a point to point operation		Replaces a tool already defined
		Inserts a tool change to a particular tool		
		Inserts a machine rotation		

This page is intentionally left blank.

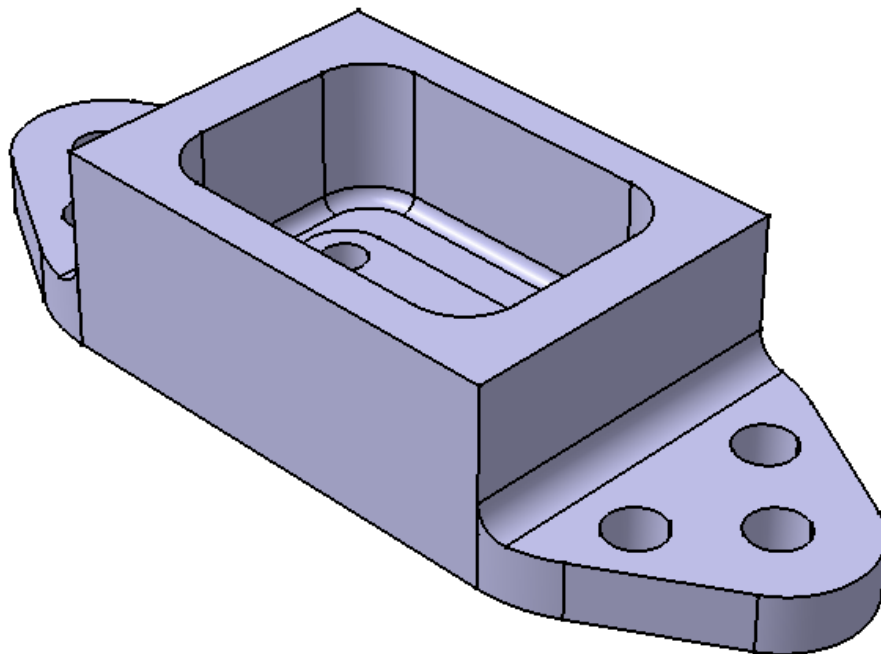
Preparing to Machine

The following sections will demonstrate how to prepare a finished part for machining. When a part is to be machined, it is held somehow in the machine. Very rarely, if ever, will the part be directly fixed to the table of the machine. Usually the part is held in a vise, locked down by clamps, or at least be bonded to a mounting plate and then that mount of some sorts will be fixed to the machining table. Regardless, there will be some type of fixture necessary to hold the part down. Not only will the part need to be held in place but stock material will also need to be assigned to the part. Sometimes the rough part will come from a mold, only being slightly larger than the finished part, while other times the stock material will be in the shape of a cube. Either way, it is very advisable to define stock material for your finished part. CATIA V5 Prismatic Machining has facilities to view material removal and therefore by defining the stock material, you can observe the material being removed to make sure you did not miss machining a specific feature.

Part Design Review

This section will serve two purposes. The first purpose is to serve as a Part Design and Sketcher review. If you feel your part design skills are not what they should be, review your Part Design and Sketcher book to refresh yourself. The second purpose of this exercise is to demonstrate how to draw stock material for a part. The finished part is pre-defined and is available in the *Mount* folder.

Open the Mount part from the *Mount* directory. You are going to build a square stock material for this very simple mount bracket.

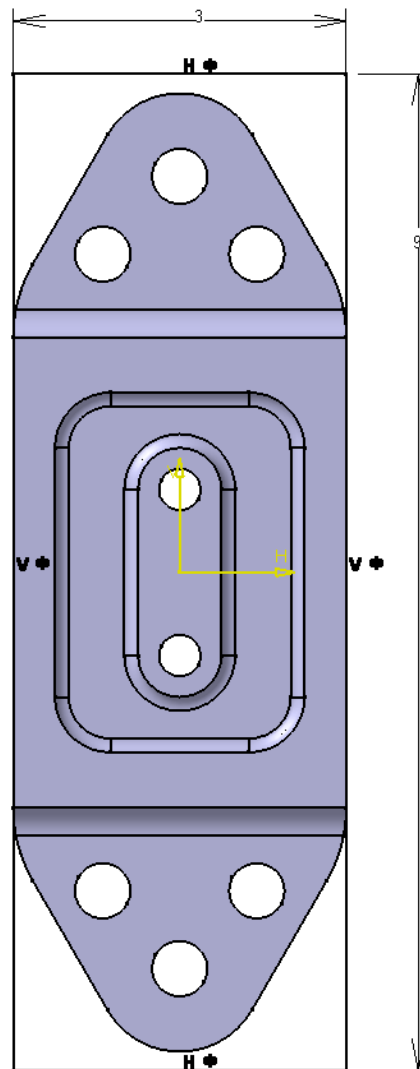


When building the stock material for a part, it is very important to get the orientation

correct. This is not so important for the assembly but for the simulation of material removal, the stock material and the final part need to be oriented properly. This is why you opened the final part.

Create a sketch on the XY plane. It is very important that the sketch be on the XY plane and not the bottom of the part. This sketch is only temporary. You will be copying and pasting this sketch into another part to build the stock material.

Create a rectangle around the part that is 3" X 9" symmetrical about the H and V axis system. This will represent the square stock that will be used. *Note: You do NOT want to constrain the rectangle against the part.* Again, this sketch is only temporary and therefore will not have the supporting elements when it is moved.




Exit the sketch and create a new part. Remember, use pull down menu *File, New* to create a new part. Now that you have a new part, it is time to cut and paste the sketch into the new part.

Tile the windows vertically. Use pull down menu *Window, Tile Vertically* to accomplish this. This will make both windows visible at the same time.

Cut the sketch you just created. This can be done by selecting the sketch, pressing button three on your mouse, then selecting cut, or by using pull down menu *Edit, Cut*. The sketch will disappear from the mount part. This has not deleted the sketch, it has just put it into memory.

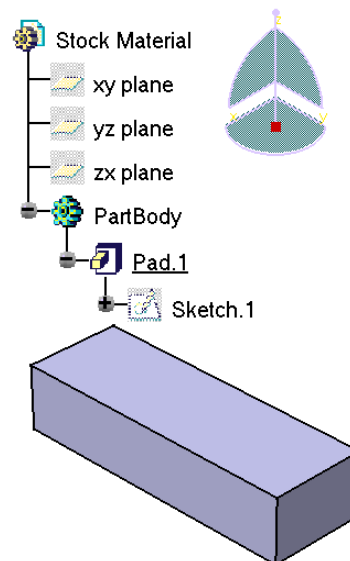
Paste the sketch into your new part. This can be done by pressing button three while on the *PartBody* of your new part and selecting paste. The sketch will re-appear in the new part as well as be re-numbered to Sketch.1.

Measure the height of the part. This can be done by selecting the measure between icon  and then selecting the top and bottom faces of the part. Take note the height of the part is 1.75". Assuming you can only get stock material in inch increments, your stock material will have to be 2" tall.

Pad the new sketch to be 2" tall. This will successfully define the stock material. Now you have just a few housekeeping measures to preform before calling it completely done.


Rename the stock part to Stock Material. Remember, when you assemble parts together, it is very important that everything have a unique identifier.

Hide the three Principle Planes. This step is not completely necessary, however it does help when you assemble your product. This keeps the assembly cleaner and easier to view.



Now a safety or clearance plane needs to be defined. The clearance plane can be defined in another part but it is unnecessary.

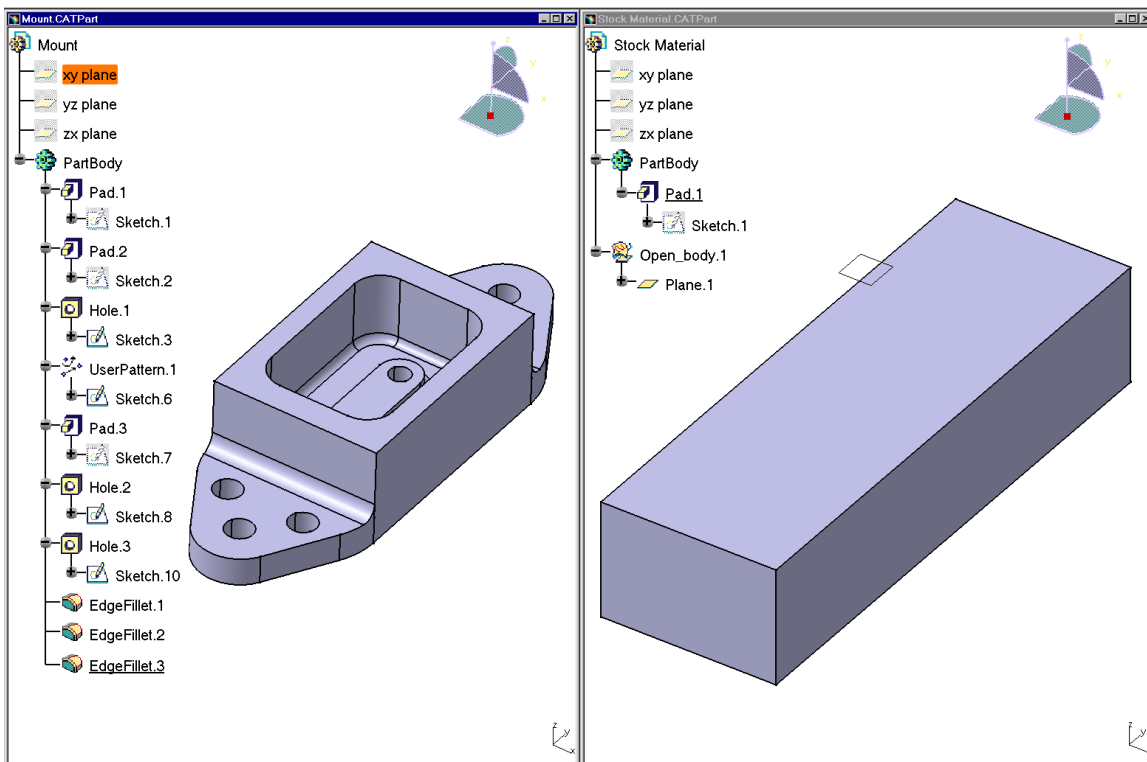
A **clearance plane** is a plane at which you guarantee that there will be nothing in the way. The clearance plane is usually defined a few inches above the stock material or any mounting clamps that are present.

Select the plane icon.  This will allow us to create a simple plane above the stock material.

Select the top of the stock material. CATIA now needs an offset value to define how high above the stock you want the plane.

Change the offset value to 2in. and select OK. This will define your clearance plane that will be used later.


Save your document. Next, an assembly design review will take place. This is where the stock material, final part and any mounting brackets will be defined. When you close your finished part, CATIA will ask about saving it. It is not necessary to save it because you did not make any changes to it. The reason it prompts you is because you created a sketch in the part thus changing it, but then you cut the sketch, returning it to the original part, although CATIA sees it differently.



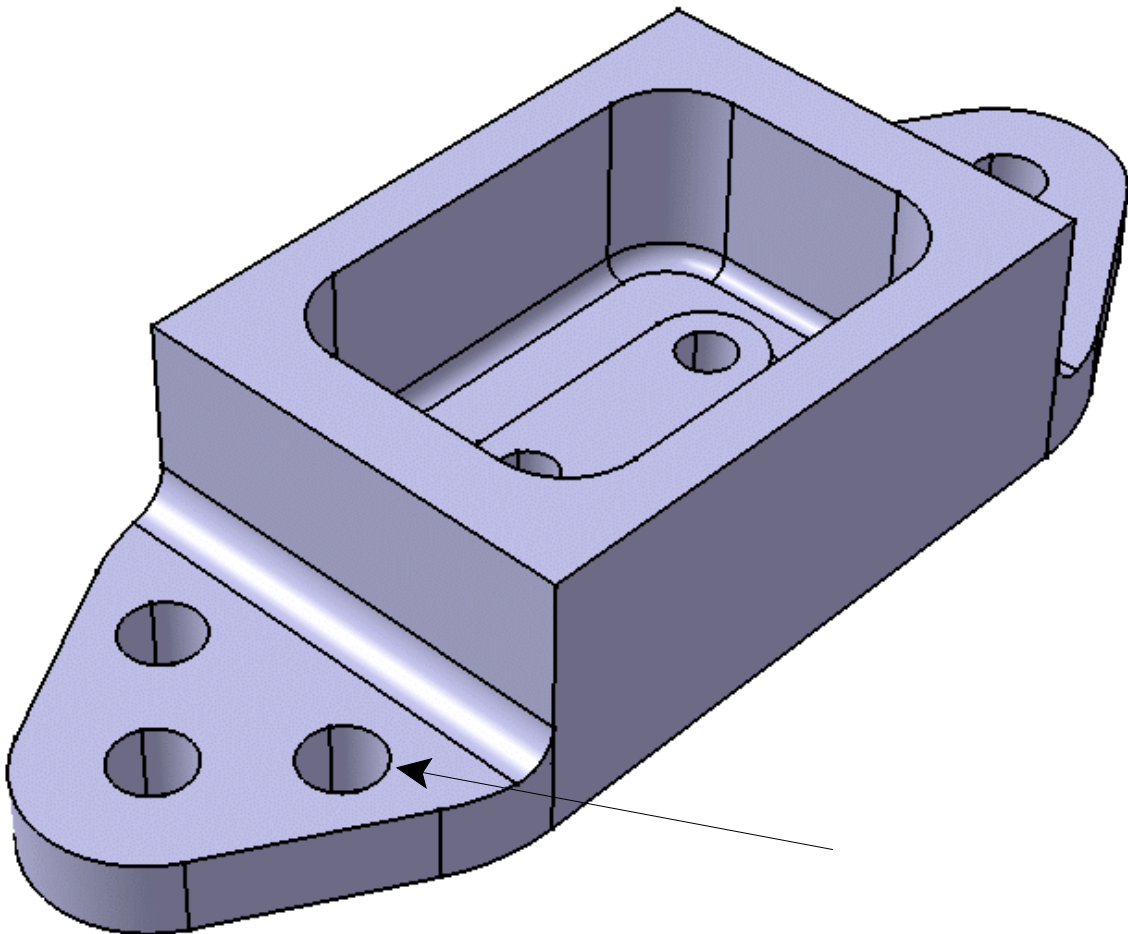
Measurement Review

Anytime a part is to be machined, it is always necessary to know some information about the part. Things such as fillet radii, corner radii, as well as hole diameters are all necessary information that you need to know before getting started. This section will review how to create the necessary measurements, as well as keep the measurements with the model for future reference.

If not already open, open the Mount part so measurement information can be added.

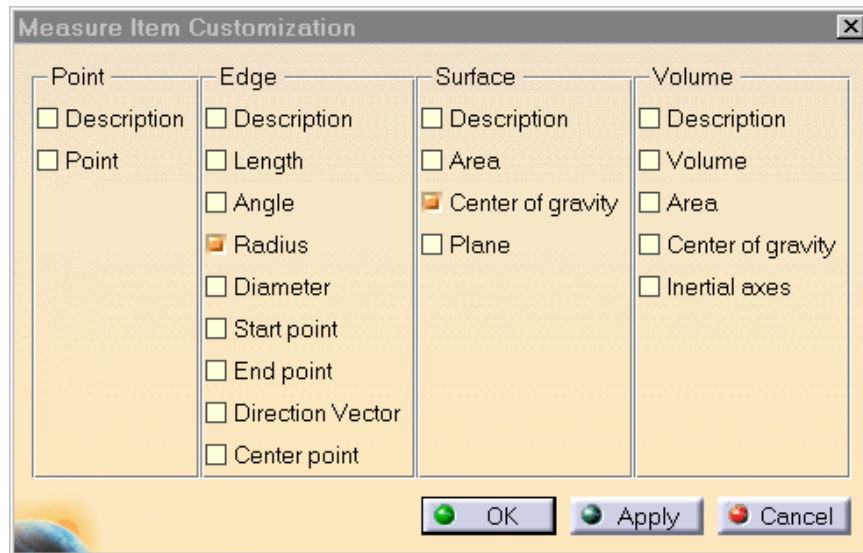
Select the measure icon.  This will allow you to create measurement information on single elements. The *Measure Item* window displays.

Select one of the outside flange holes along the edge. The edge is shown below, but it is not necessary to pick that exact hole.




You will get a lot of information when you select the hole. A lot of this information is unnecessary and useless for your purposes. Customize the output to refine what you are after.

Select the **Customize...** button at the bottom of the **Measure Item** window. A series of check boxes will be displayed. You will want to turn all the check boxes off with exception of the edge radius and the surface center of gravity.



Select **OK**. This will change the results to only show the radius of the edge.

Select the **Keep Measure** check box on the **Measure Item** window. Select **Close** when done. This will keep the measurement and store it in the tree as well as show the measurement on the part.

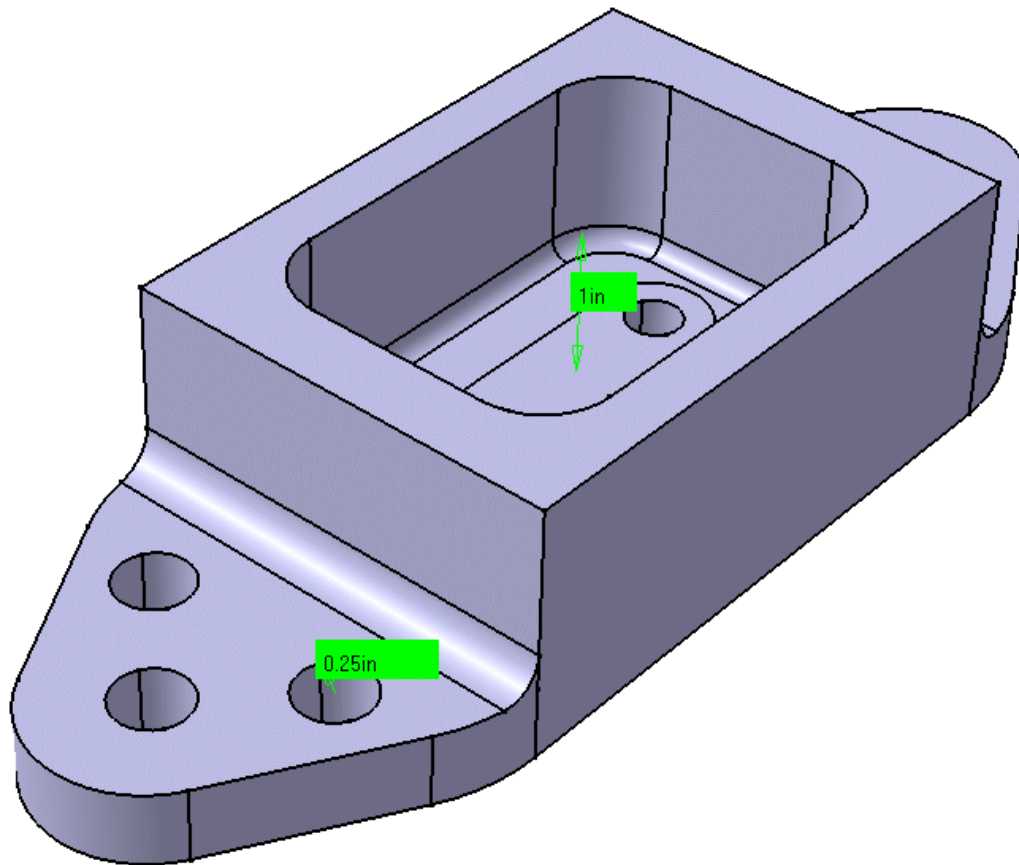
Select the **measure between** icon.  This icon will allow you to measure the distance between or along any two elements.

Be sure the **Reference mode** and **Target mode** are set to **Any geometry, infinite**. This will make sure that you get a normal direction measurement. Again, customize the distance measurement information to show only what is interesting.

Select **Customize** and **uncheck** all boxes except **Minimum Distance**, select **OK**. This will only show the minimum distance between two elements that you select. Usually this is used to measure the distance between planes.

Select the **top of the island** and the **top of the part**. A measurement is made and placed on the model.

Be sure the **Keep Measure** checkbox is on and select **Close** when done. Again, the keep measure will add a measurement branch to your specification tree keeping it there for later reference.



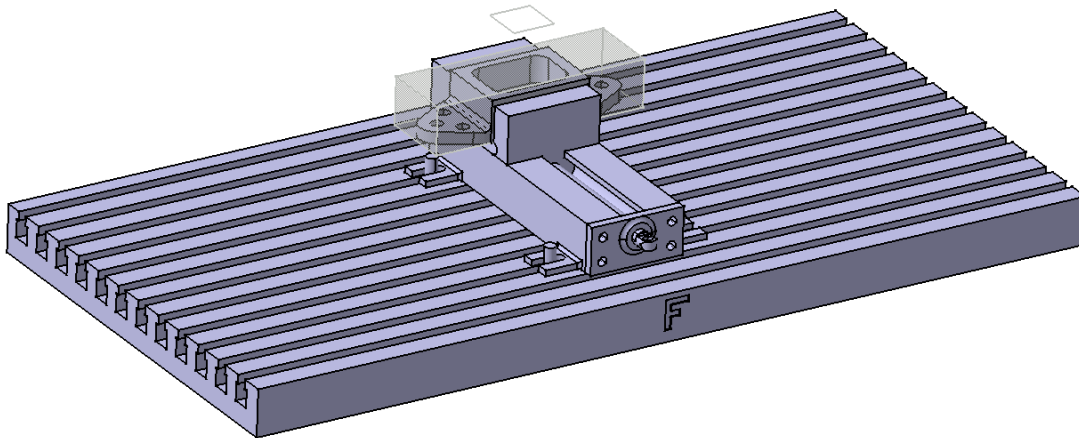
Create other measurements as necessary. Measurements such as fillet radii and corner radii are most useful. Depth of the pocket and height of the part is also good information to extract.


Save your document when done.

For clarity, the measurements have been hidden from now on. You can hide your measurements if desired.

Assembly Review

For the assembly review, several things are going to be assembled. The assembly is going to consist of a machining table, vise, finished part, and stock material.




Create a new assembly, and insert the small table. To create a new assembly, select pull down menu *File, New*, and then *Product*. To insert the small table into the assembly select the existing component icon , *Product1*, and then find the **Small Table** document in the *Machining Equipment* directory.

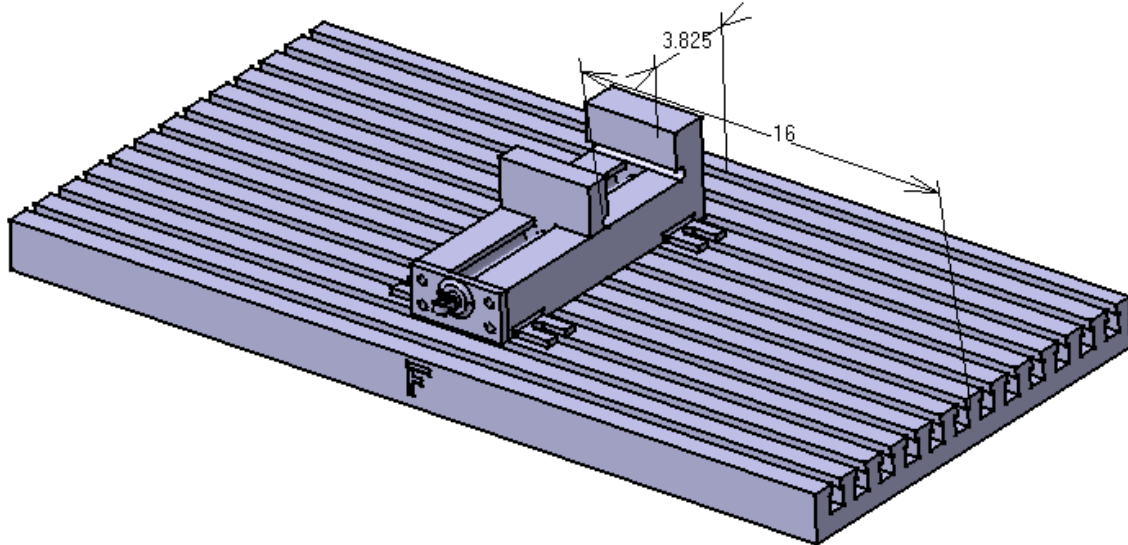
Apply a fixed constraint to the table.  This will hold the table in place.

Insert the Vise into the assembly. The vise is pre-assembled. There will be no need to assemble the vise before putting it in the assembly. It is located in the *Vise* directory under the *Machining Equipment* directory.

Apply a contact constraint between the bottom of the vise and the top of the table.



Position the vise and apply two offset constraints from the edges of the table to the vise.  The two constraint distances are shown below. The vise should be 3.825" from the back and 16" from the side.



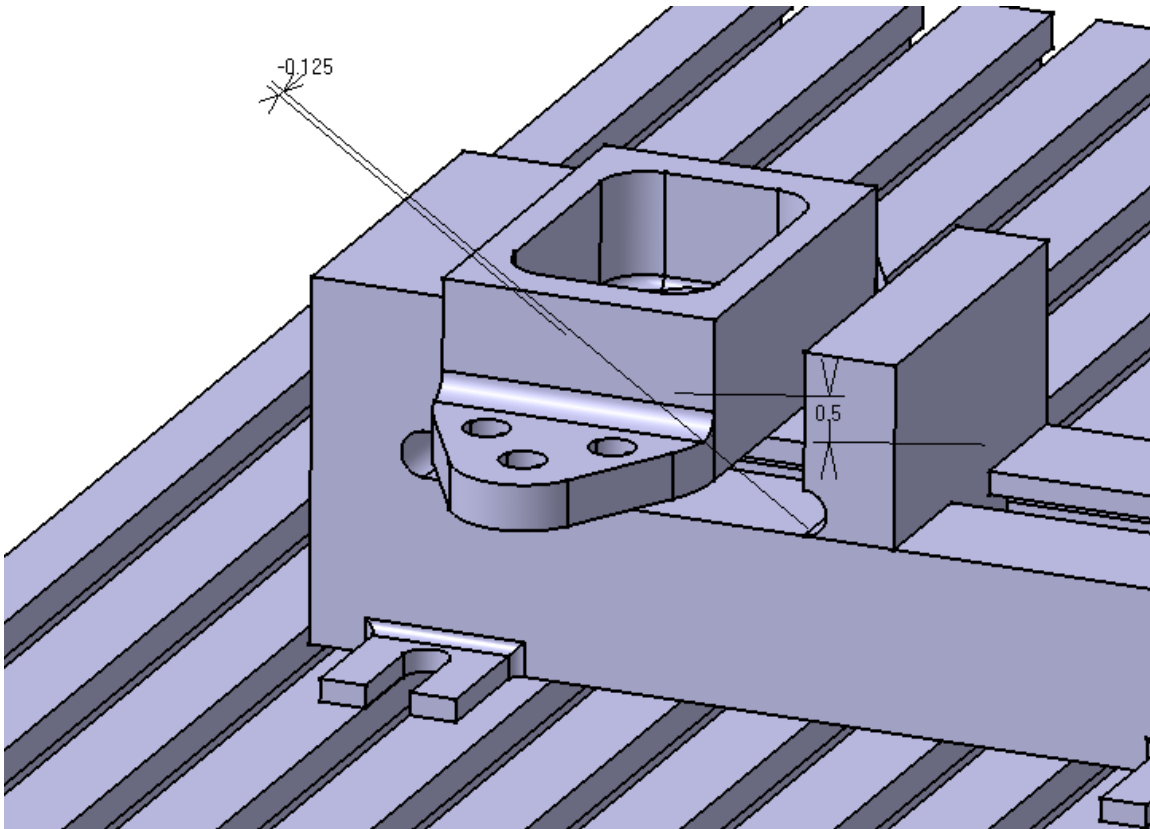
Bolts to hold the vise down are available in the **Clamps** folder. It is left to you if you want to insert and constrain the clamps. In this case, the vise is tall enough that the bolts will not be getting in the way.

Insert your Mount into the assembly. The mount is going to be affixed into the jaws of the vise.


Apply a coincidence constraint between one side of the mount and the back jaw of the vise. You most likely will have to reverse the direction of the arrows to make the part fit inside the jaws.

Apply an offset constraint between the bottom of the mount and the inside bottom of the jaws. The offset distance is 0.5in. Watch the part to make sure the orientation does not need to be changed.

Apply an offset constraint between the outside pocket wall of the mount and the side of the vise. The offset distance should be **.125in**. This should constrain the mount to be located correctly in the vise. The assembly should look something like the picture shown below. *Not all constraints are shown.*



To get an accurate representation of how the vise is going to hold the part, you need to close the jaws of the vise. Since the vise is a separate assembly, by default nothing on it can move independently of each other. You need to make the vise a flexible sub-assembly.

Change the Vise to a flexible sub-assembly.  This can be accomplished by selecting the flexible/rigid sub-assembly icon and then selecting the Vise from the specifications tree. This will unhide the constraints for the sub-assembly and make the vise flexible meaning that the components can move independently of each other.

Apply a contact constraint between the other side of the mount and the movable jaw. The jaws will close together on the mount.

Now there is just one last component to install into the assembly, the stock material that you created earlier.

Insert the stock material into the assembly. The stock material will be placed directly over the final part to represent how the stock will be loaded into the vise on the machine.

In order for both parts to be visible at the same time, you will need to make the stock material transparent.

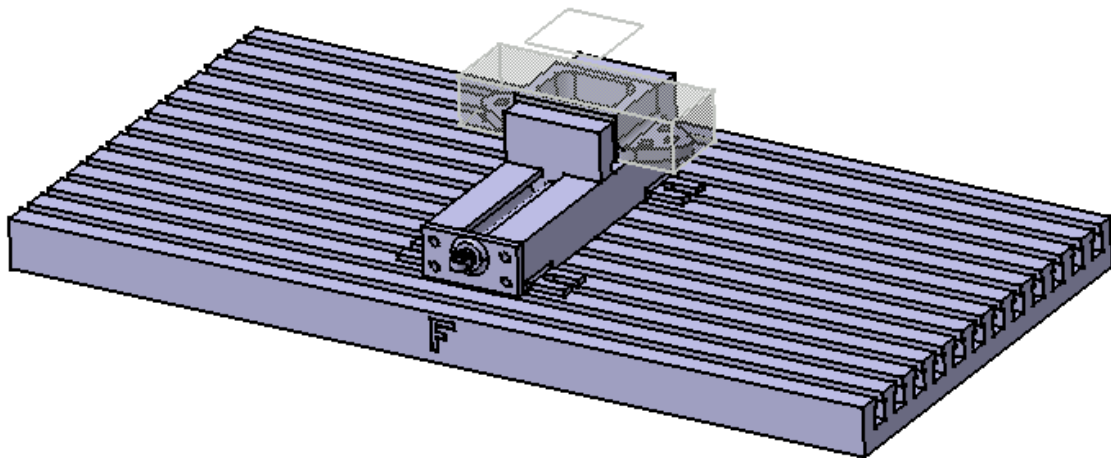
Using the third mouse button, select on the *Stock Material* in the specification tree. This will bring up the contextual menu for the stock material component.

Select *Properties* and then the *Graphic* tab. Adjust the *Transparency* to approximately 125 and select *OK*. This will make the block semi-transparent.

Apply a coincidence constraint between the side of the stock and the side of the mount. This should move the stock material to the side of the final part.

Apply a coincidence constraint between the bottom of the stock material and the bottom of the mount. This will move the stock down to the level of the final part. The last step is to position is properly over the final part.

Apply an offset constraint between the side of the stock and the side of the vise. The distance should be 2.5in. This finalizes the assembly.



Be sure to rename the product to something more descriptive than *Product1* and save your assembly. Next, this same assembly will be utilized in the Prismatic Machining workbench. You will define the initial setup conditions of the machining process.

This page is intentionally left blank.

Part Operation Setup

This section will investigate how to invoke the Prismatic Machining workbench and prepare your part for machining operations. There are several steps beyond building stock material and assembling all of the components: stock, finished part, and any mounting brackets.

Open the assembly you created in the assembly review section. When you are ready to begin machining a part, you should have it opened first.

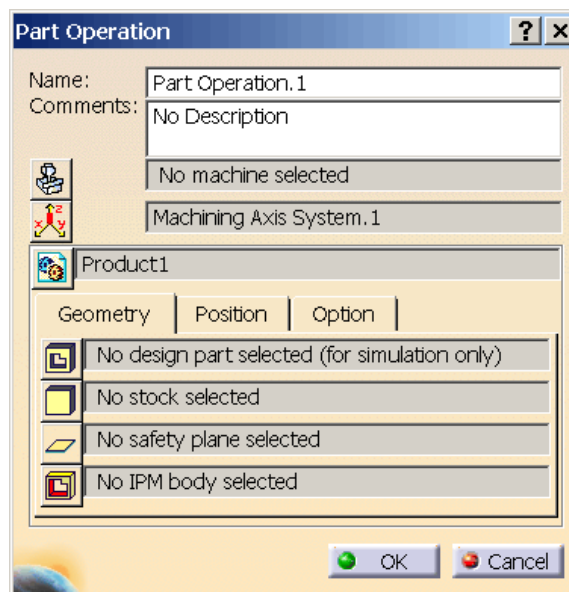
Change the workbench to the Prismatic Machining workbench. This can be done by either selecting pull down menu *Start* and then finding Prismatic Machining, or by selecting the change workbench icon and then the Prismatic Machining workbench.

It is not mandatory that you have the assembly open that you are going to be machining. There is an option that will allow you to associate a product with your part operation, it is just easier to have the assembly open before hand.

Now you are ready to begin defining the Part Operation.

Defining the Part Operation


Double select the *Part Operation.1* branch in the *PPR* tree. This will display the *Part Operation* window.

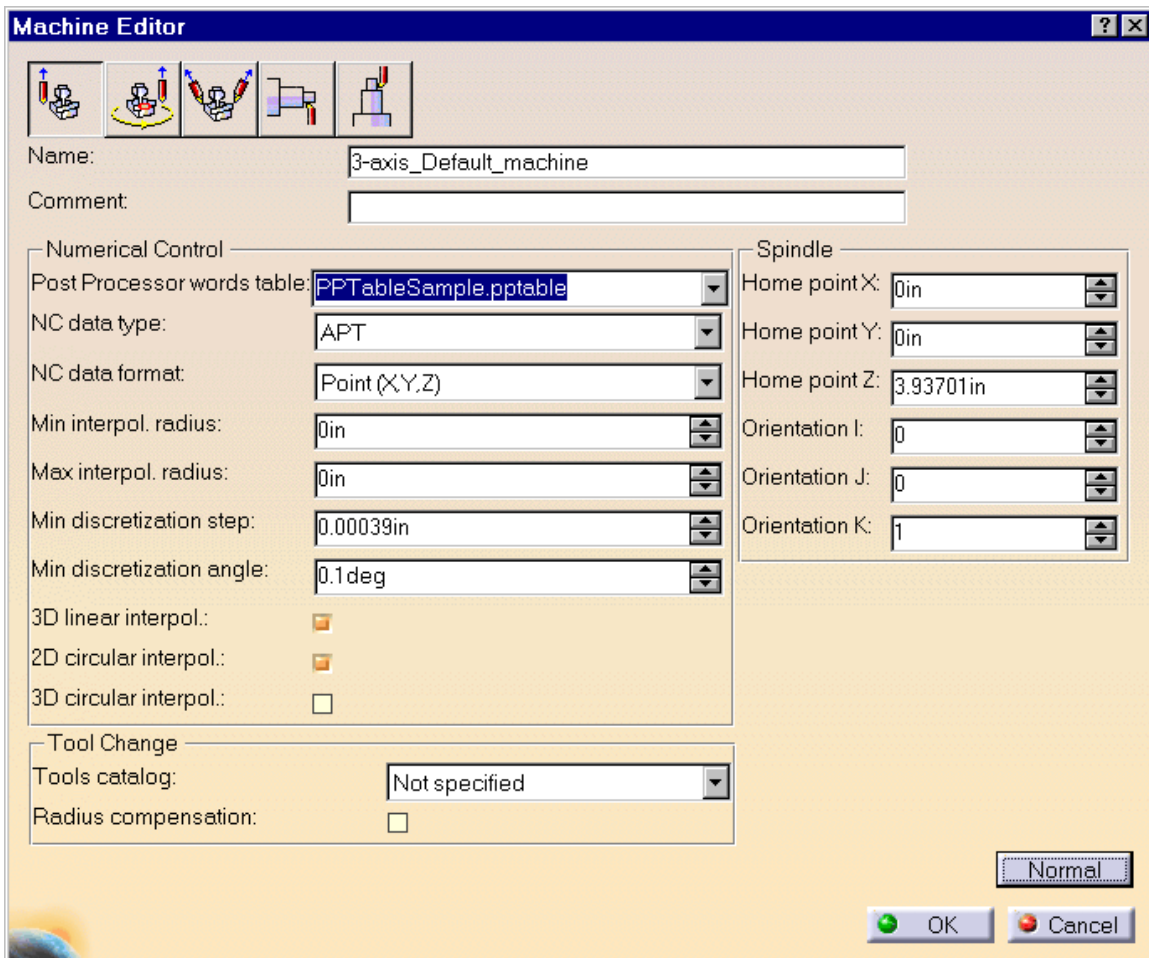


There are several different fields that will need to be defined before you begin machining. These steps will be very common for all parts in most situations.

Change the *Name* to *Mount Bracket Milling*. As with most everything else, it is a good idea to get in the habit of naming your part operations, that way they will remain clear as to what they are.

Change the Comments field to read Your Name. The comments field is very useful to insert specific information about the part operation. In this case, you are just going to state who you are, but you could add information about the part, information about the use of the part or anything else that you want.

Select the machine icon.  Selecting this icon will display the *Machine Editor* window. Within this window you can define the machine that you will be working with for your part operation.



The first set of icons across the top allow you to define the specific type of machine you are going to be using. The machine types are as follows:



Simple 3-Axis machine



3-Axis machine with a Rotary Table



5-Axis machine



Horizontal lathe for machining



Vertical lathe for machining

After the series of machine types, you can name the machine. A comment field is also allowed for additional comments on the machine.

The **Numerical Control** section of the *Machine Editor* window defines how the NC code is written for the machine. You need to define a **Post Processor words table** for your machine. These tables will hold the machine specific language commands for things such as turning the coolant on and off, drill motion commands, etc. The **NC data type** defines what type of NC data you are creating. Usually this will be left on *APT*, but the option to change it to something else is there. The next field defines the **NC data format**. The NC data format field allows you to change the type of output from simple X,Y,Z coordinates, to X,Y,Z coordinates as well as I,J,K axis directions. For 3-Axis machines, X,Y,Z, is sufficient. If a 5-Axis machine is to be used, you would also need to output I,J,K axis directions. The next series of fields define a few parameters about the exactness of the NC data. Unless there is a specific need to change these values, it is advisable to leave them at the default settings.

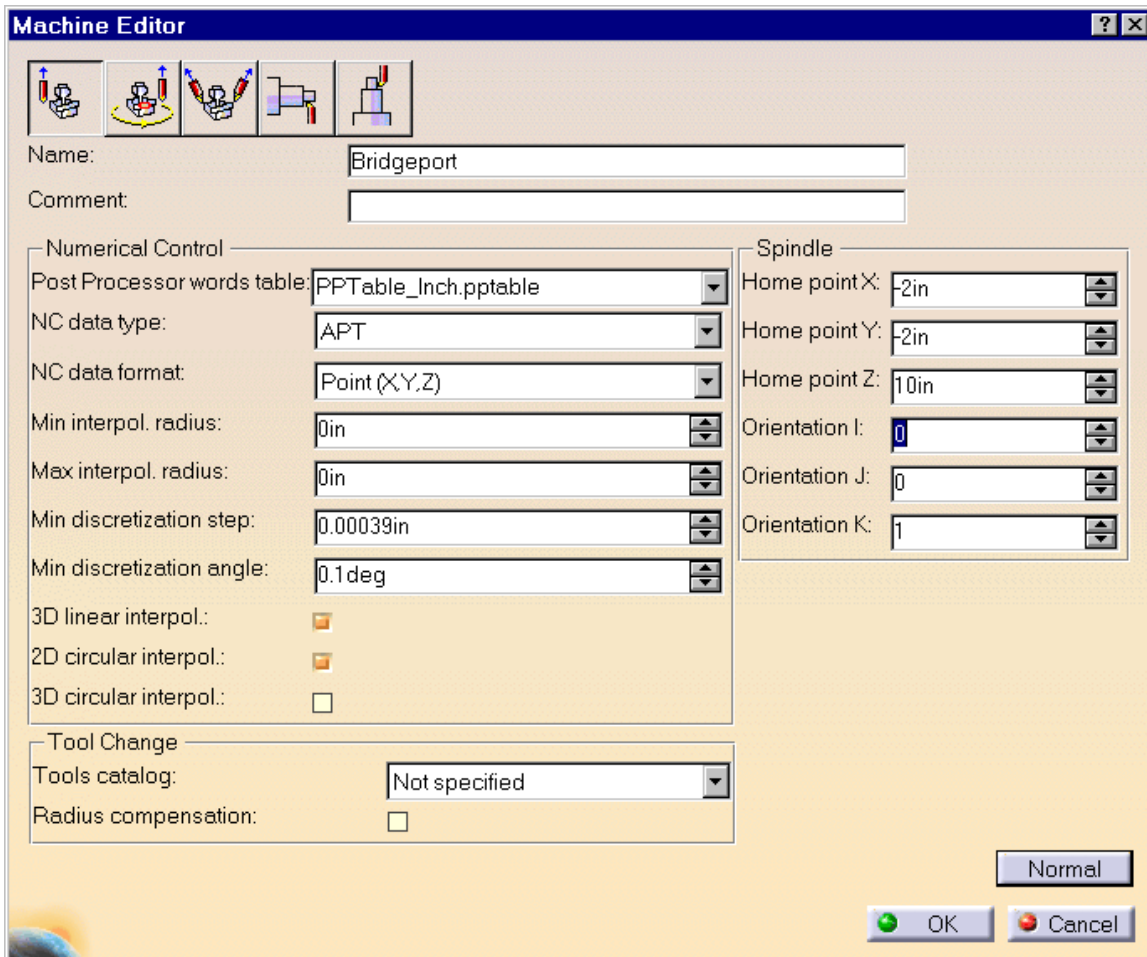
The **Spindle** section of the *Machine Editor* window defines several characteristics of the spindle. The home point is defined within this section, where the home point is relative to the machining axis. The orientation of I, J, and K will define the spindle direction. It is in this field that you could define whether you have a horizontal or vertical 3-Axis machine.

The last field found from the *Advanced* options is the **Tool Change** field. The tool change defines what tool change catalog you will be using. If no tool catalog is defined, then there is no default database that will be searched when requesting a tool. There is also a checkbox for defining a radius compensation. Radius compensation is useful when a tool has been re-sharpened and is slightly smaller than the original. By defining radius compensation, otherwise known as cutter compensation, the system will automatically adjust the tool path.


Change the Name of the machine to Bridgeport. You will be programming for a simple 3-Axis Bridgeport machine.

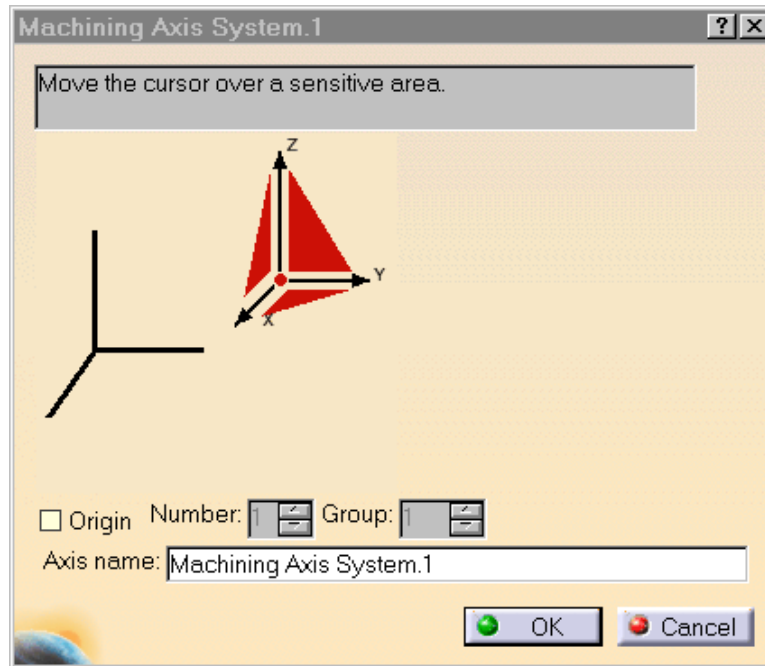
Change the Post Processor words table to PPTableSample_Inch.pptable. This will insure that the output for the Bridgeport will be in inch standards.

Change the *Home point* to be set to **(-2,-2,10)**. That is, make the X value -2 inches, Y value -2 inches and the Z value 10 inches.



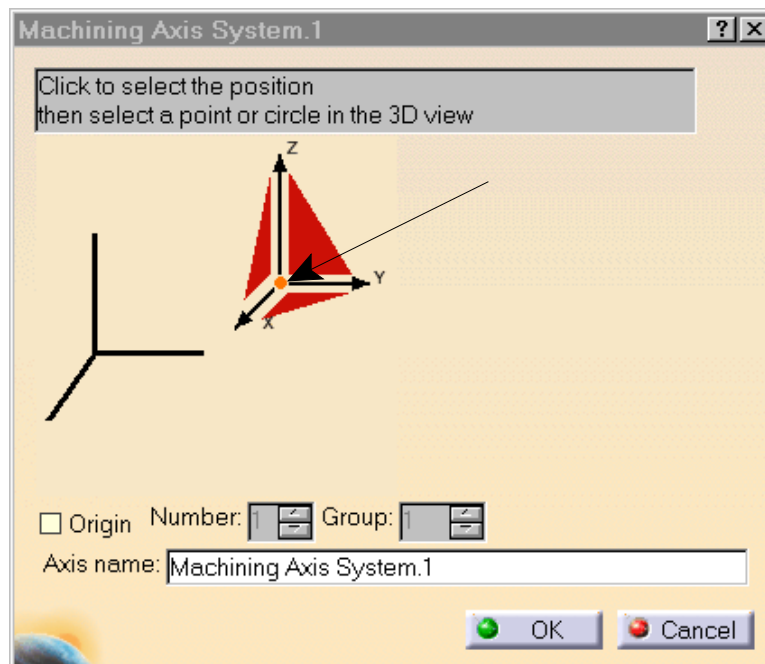
Select OK when done. This will take you back to the *Part Operation* window. Next, the machining axis system will be defined.

Select the machining axis system icon.  This will display the *Machining Axis System* window.



There will be a red set of axes and planes in the center of the window. This axis system is how you define a new machining axis. By default, the machining axis appears at the central axis system for the assembly. Many times this will not be the right location. The axis will be moved to the back right corner of the table.

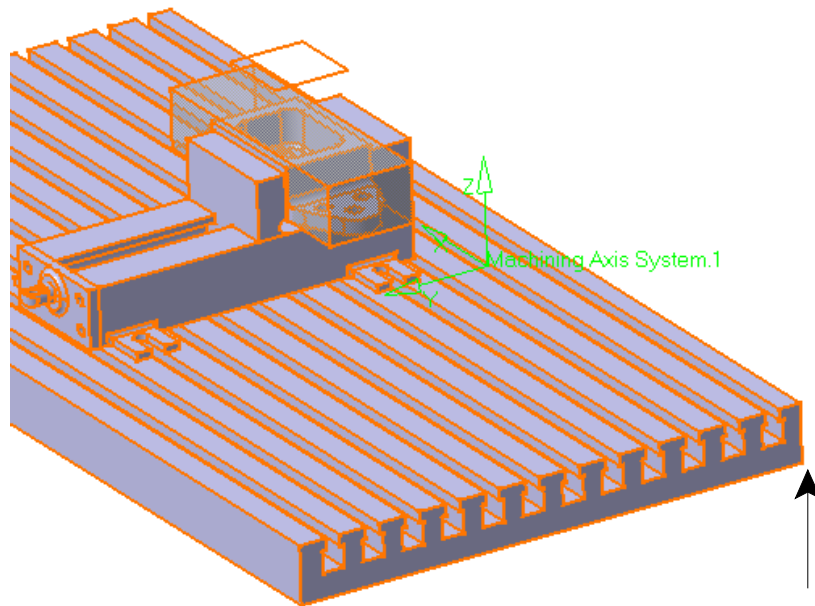
Select the center point of the axis system. This will be the small red dot in the center of the axis as shown below.



The center red dot will allow you to move the entire axis system from one location to

another. The *Machining Axis System* window will disappear and CATIA is waiting for you to select a corner vertex to be the new center of the axis system.


Select the back right corner of the table. This will define the new center of the axis. Select the corner shown below.




The *Machining Axis System.1* axis will move to the corner. Notice the axis system turns green. This denotes that a new axis location has been defined. If the orientation of the axis is incorrect, you can select one of the axes and then an edge to define a new direction for the axis.

Select OK. This will define the new machining axis system. Next, the geometry will be defined.


The part or product icon that is next in line is where you would define the assembly that you are using. Since the assembly was already open, it is unnecessary to define the assembly.

Select the design part for simulation icon.  This will define the final part of the machining process. It is not necessary to define this design part because it will only be used for simulation purposes.

Double select the Mount part. This can most easily be done by selecting it from the specifications tree.


Select the stock icon.  This will define the stock material that you will start machining from. This will be used for other purposes than just simulations.

Double select the *Stock Material*. Again, this may most easily be selected from the specifications tree.

Select the safety plane icon.  This will allow you to define the safety plane or clearance plane that will be used by default in the machining operations.

Select the plane you created in your *Stock Material* part. This is probably easier to select from the working area itself as opposed to the specifications tree.

There is one icon left in the list that will not be used for this class. It is the IPM body for DPM Powertrain icon.

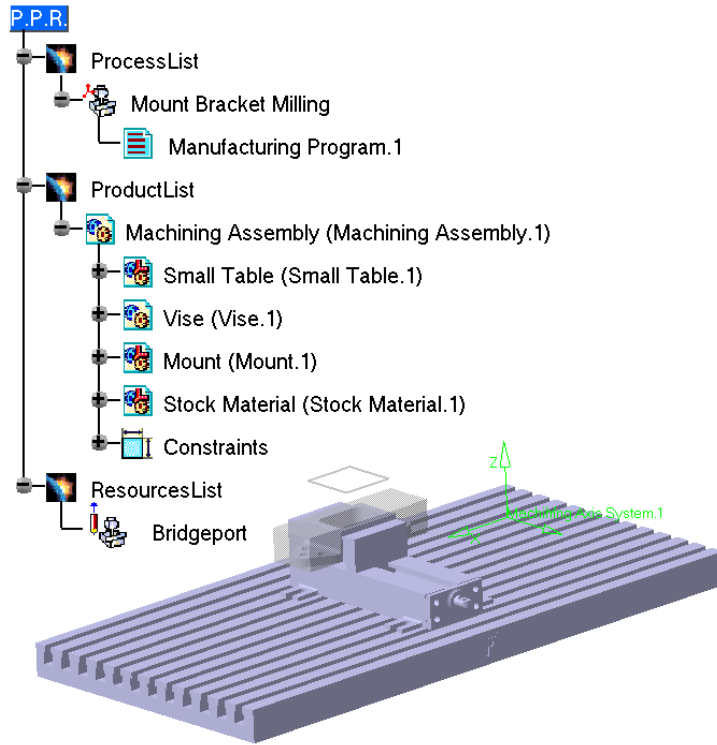
 IPM body for DPM Powertrain – This icon allows users who use the Digital Product Management Powertrain to define an In Process Model. Since the Digital Product Management Powertrain is beyond the scope of this class, it will not be discussed further. Defining a body for this field is the exact same as defining a body for the design part, or stock material.

Now all of the geometry has been defined, you can move on to other options. It is always advisable to define all the *Geometry* fields, although it is not mandatory. By defining the stock and design part, you will be able to see simulation removal of the actual stock part that you are working with. The design part definition will allow you to check deviations from the final cut part against the design part. The safety plane definition will allow for simpler, and faster macro creations.

Select the *Position* tab. The position information allows you to define the exact location of a tool change as well as the center of the table.

Change the *Tool Change Point* to be -1,-1, 10. This will set where the tool change happens.

Select *OK*. This will finalize the Part Operation. There have been no machining operations created, but everything is set up and ready.

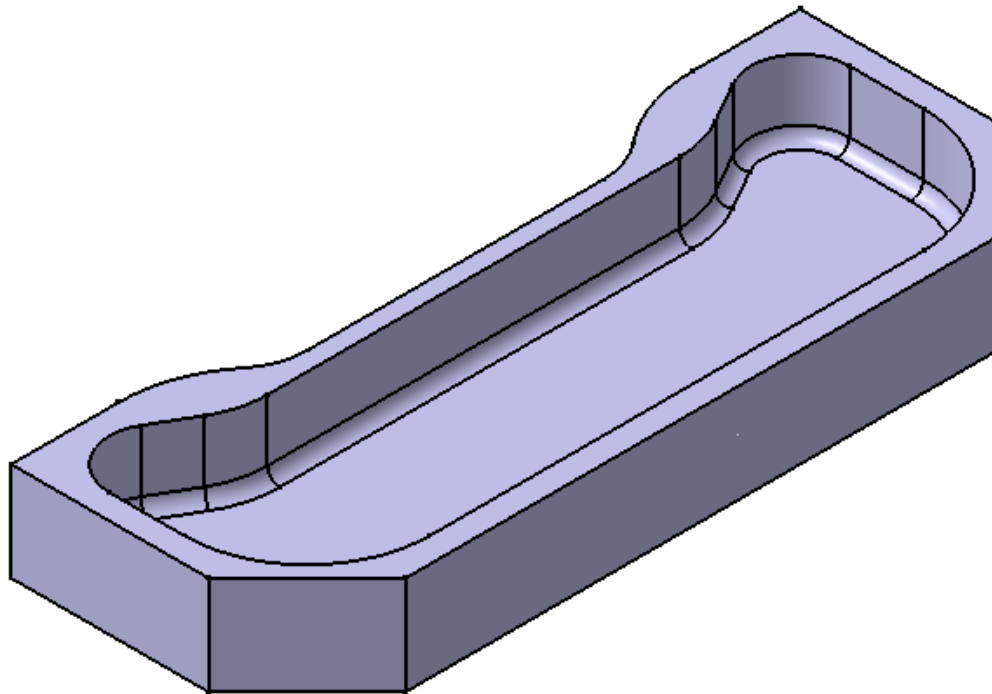


Be sure to save your machining process.

Basic Machining

This will be the first section involving machining of a part. The order of the basic milling modes will be the order most commonly followed with any given part. First the part will be faced off or the excess material on the top will be removed. Secondly, the outside of the part will be contoured, removing any excess material from the outside. Afterwards, the pocket of the part will be milled.

Open the Basic Machining Milling document from the *Basic Machining* folder. This will be the first part that you machine. You will start off by facing the part off.



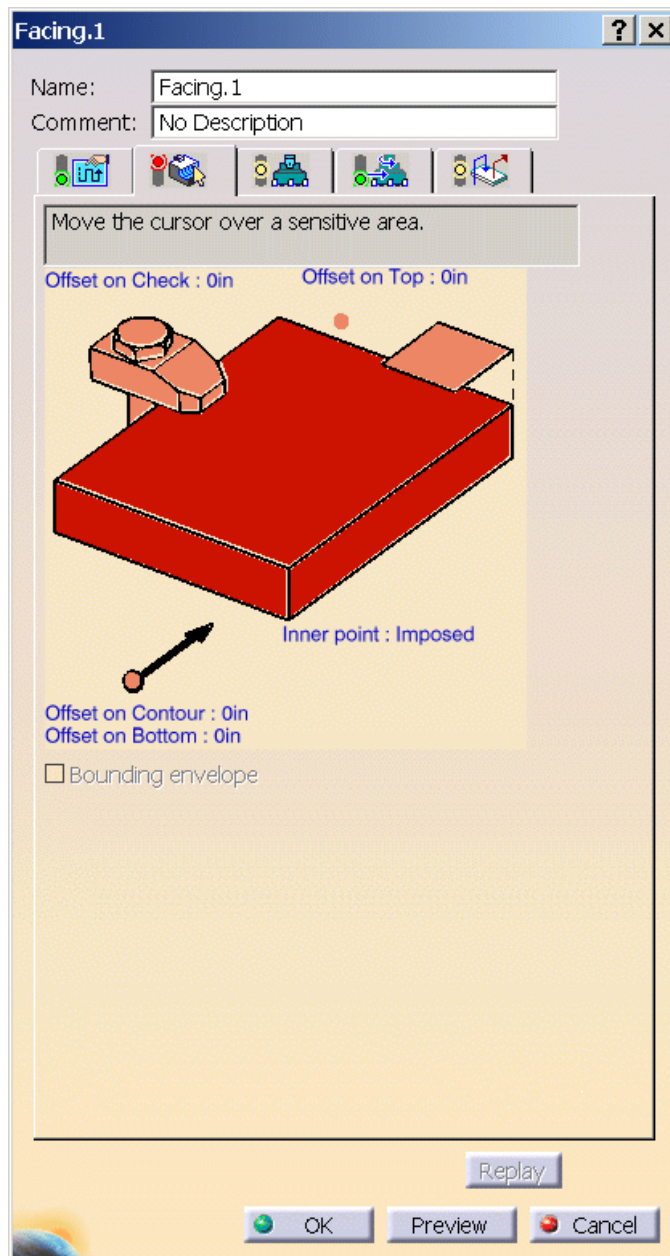
Facing

Select the facing icon.



The icon will highlight, but nothing will happen. This is because CATIA is waiting for the insertion point of the facing operation to be defined.

Select **Manufacturing Program.1** from the specifications tree. This will insert the facing operation and tool change operation into the specification tree just after the manufacturing program. Now the *Facing* window displays.



This is the basic layout of all of the windows for all machining operations. The first field is to define a more descriptive name for the operation. The second field is to add comment descriptions with the operation. Just below the comment field there are several tab settings.

Each tab also has a set of “lights” associated with them at the side of the icons. A red light indicates that information needs to be entered. This denotes that CATIA does not have enough information to complete the tab. A yellow light indicates that the user has not defined any information but there is enough default information to complete the tab. You should always check these tabs to make sure everything is correct. A green light indicates that all information is entered and is valid.



This tab defines the machining operation parameters. Settings such as machining direction, various machining strategies, and high speed milling options are kept under this tab.



This tab defines the geometry to be milled. This panel is very dynamic for all different machining modes. Each machining mode geometry panel will be discussed with the machining mode.



This tab defines the tool to be used. For each type of operation, there are only a few different tools that can be used, so this panel will display only the types of tools that are permissible to use.



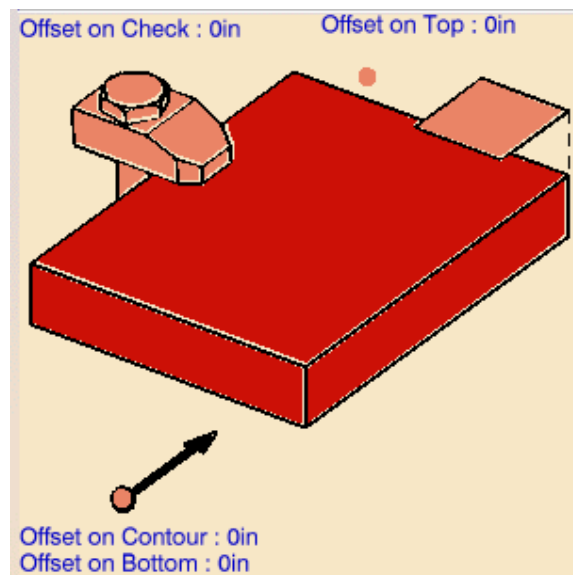
This is the speeds and feeds tab. All feedrates and spindle speeds are defined here.



The last tab defines the macros to be used for the machining operation. Various macros can be defined and utilized.

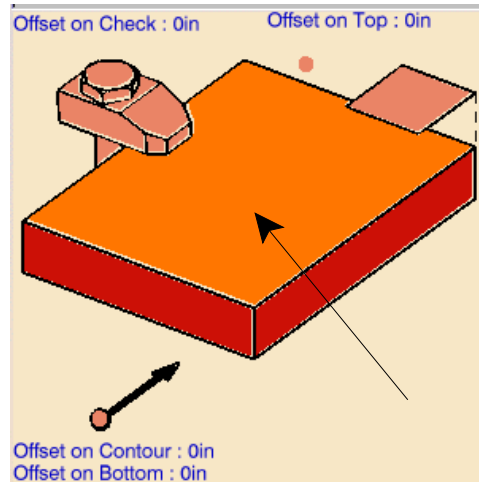
Geometry tab

Go back and take a closer look at the geometry tab for facing operations.



This is the sensitive area of the geometry tab. Any areas in red are areas that have to be defined. When an area turns green, the information is defined and the user can move on. Orange or peach colored areas are those that are optional to define. Things such as check surfaces and starting points are optional. Around the outside is several sets of blue text. This text is also modifiable and can be set to any distance necessary. All areas of this sensitive area is contextual menu sensitive, meaning that by pressing button three on the mouse, you get alternate selection options.

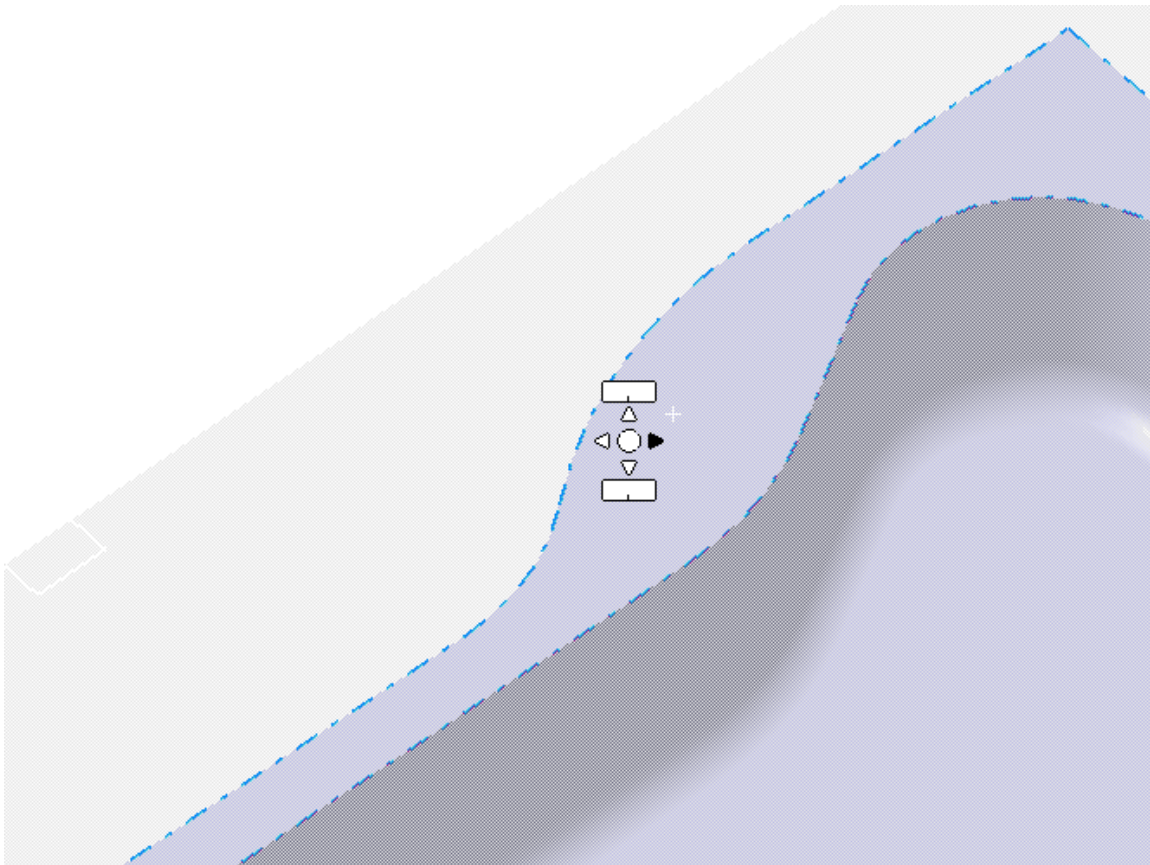
Select the top of the facing geometry. This will define the top of the design part that is to be created by the facing operation. The *Facing* window will disappear and allow for room to select the top of the design part.



With the stock material shown, it will be difficult to select the top of the design part.

Zoom in and place the cursor over an area that you want to select for the top of the final design part. Below is shown a good area to concentrate on.

Press the up arrow on the keyboard. The peel selection will display. This is a utility that allows you to select faces, edges or bodies that are not on top.



The peel utility has several uses. The up and down arrows allow you to scan through the different layers of geometry that can be selected. The center round dot allows you to select a highlighted layer.

Since the top face of the final part is the second level down, the face highlights, preparing you to select.

Select the top of the final part face. That is, select the round dot in the center of the peel selection utility. The *Facing* window displays again, this time with the top and sides of the part turned green. By default, the boundaries of the top surface will be used as the boundaries of the facing operation. This is not the case since you have the stock material defined.

There are four different options when defining the perimeter of the geometry to machine.

Edge Selection

Edge selection selects the boundaries of a facing operation by the edges of the parts. Edge selection is available by selecting the side of the facing cube.



Navigates tangentially around the edges of the part.



Closes a contour with a straight line.



Inserts a straight line between gaps.



Resets all selections. This is common to all selection modes.

The *OK* and *Cancel* icons either confirm or cease the edge selections.

By Belt of Faces

This method of selection allows you to select all of the exterior faces around a part that you want to mill. This method of selection is available by selecting button three on the boundary of the facing cube and then selecting *By Belt of Faces*.



This navigates the faces tangentially.



This displays a preview of the contour that you have selected.



This allows faces to be selected by trapping them with an indicated polygon.



After selecting one face, this icon will select all the faces of the same color.



This will allow faces to be selected by a selection set that is pre-defined.

By Boundary of Faces

This method of selection selects the boundaries of a facing operation by selecting the boundaries of a selected face. For this, you would usually select the top face of the stock material and then the boundaries of the top of the face will be used as the boundaries of the facing operation. This method of selection is also available by selecting button three on the side of the facing cube.

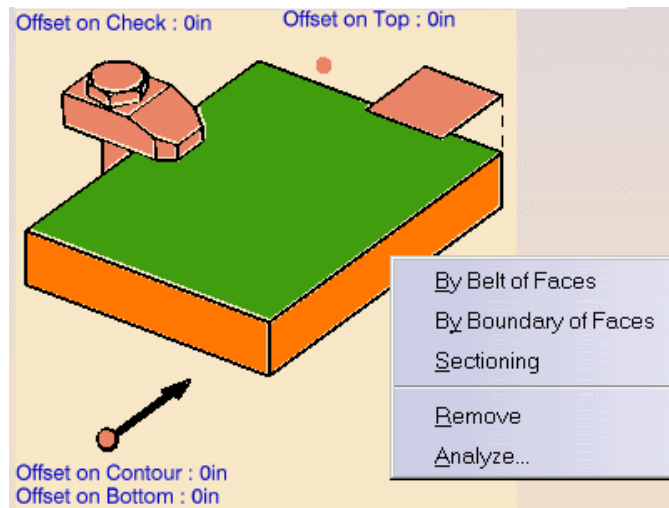


All icons in this panel have already been covered.

Sectioning

Sectioning is the last method of selection. This method allows you to define boundaries by generating a section cut and then selecting a profile of the section cut. This method of boundary selection will not be covered.

Button three select on the sides of the facing cube. This will bring up the contextual menu for the different select options on the boundaries.

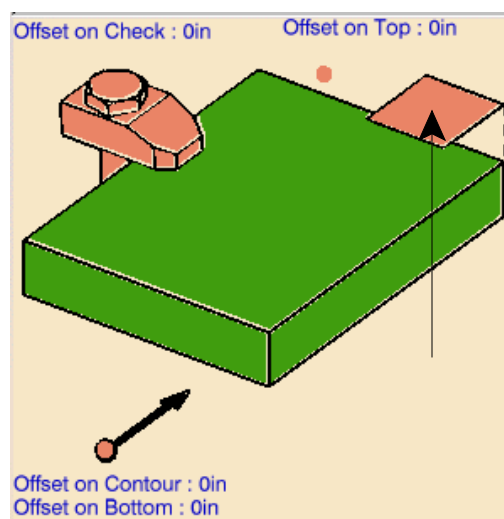


Select *By Boundary of Faces*. This will allow you to just select the top face of the stock material.

Select the top of the stock material. When done, select *OK* on the face selection toolbar. This will define the boundary of the facing operation. There is still some geometry that needs to be defined. You still need to define how deep the facing operation is going to cut. CATIA knows where the bottom of the facing operation is but it is still oblivious as to where the top of the material begins.

You'll note that even though information is missing, the light on the geometry tab is green. This is because all of the necessary information has been entered. Defining the depth of the facing operation is extraneous information and is considered unnecessary.

Select the plane in the contextual area. This plane allows you to define the top of the stock material.



Select the top of the stock material. The plane will turn green when the *Facing* window shows again. There are several options that have yet to be touched.

Selecting the clamp object will allow you to define any fixture check bodies or check surfaces. You will investigate these later. There is also two points in the contextual area. The point with the arrow is the start point icon. This will allow you to specifically choose a start point, where the other point allows you to specify an ending point.

There are also a few textual options that can be defined.


Offset on Check This defines the distance away from clamps, fixtures and any other check surfaces that you select.

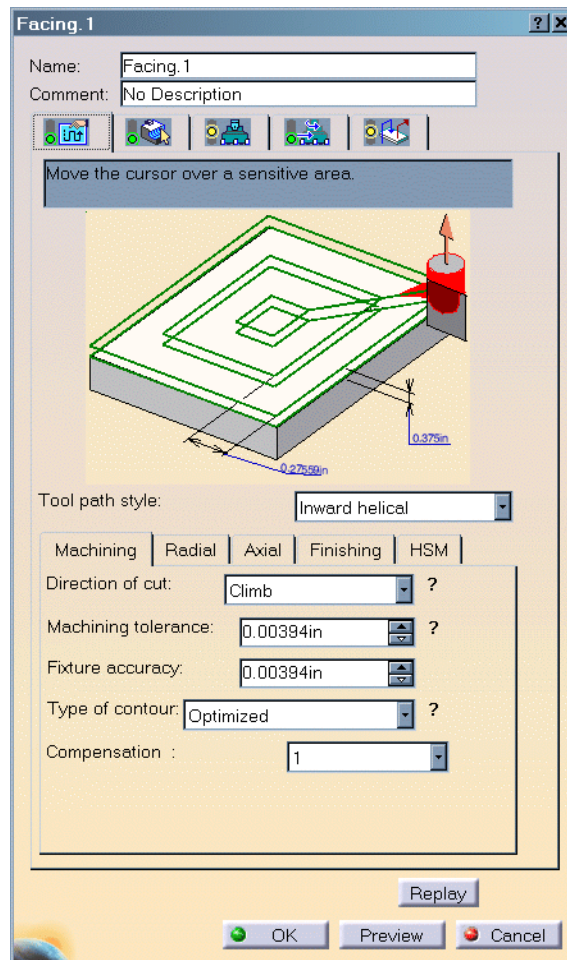
Offset on Top This defines the distance above the top selection that the tool should approach at.

Offset on Contour and Bottom These options allow you to define a distance that the tool should stay away from the sides and the bottom selection. A bottom offset would be most useful if a finish pass was desired.

It is now time to move on to another tab.

Machining operation parameters tab

Select the machining operation parameters tab.  This tab will be found for all machining modes. After setting the parameters for the first machining operation for a particular mode, the settings will remain persistent for the next similar machining operation. Even though the parameters will be the same, it is always a good idea to check them, watching for any new parameters or parameters needing to be changed.



Take a look at all the different parameters that can be set. Notice next to some of the parameters is a question mark (?), this denotes an image is available to aid you in the definition. Also note the different tabs available. Each tab will be investigated in turn.

Tool Path Style Defines how the tool moves around the part

Inward Helical Helical motion that starts from the outside and works its way in

Back and Forth Runs side to side across the part. *Back and Forth* also has one additional option under the geometry definition, *Bounding envelope*. This makes the back and forth motion run a square area, and not an odd shape.

One Way Only goes in one direction across the part

<i>Machining</i>	Defines the overall machining tolerances
<i>Direction of Cut</i>	Defines if you are climb milling or conventional milling
<i>Machining tolerance</i>	Defines to what degree of accuracy the machine can cut
<i>Fixture accuracy</i>	Defines to what degree of accuracy that the fixture is located where it is drawn
<i>Type of Contour</i>	Defines how CATIA maneuvers around corners. There are four different options that you can view by selecting the option type, and then selecting the question mark (?) next to the drop down menu. They are <i>Circular</i> , <i>Angular</i> , <i>Optimized</i> and <i>Forced circular</i> .
<i>Compensation</i>	Allows you to define the cutter compensation number to be applied to the tool and tool path
<i>Radial</i>	Defines how the tool moves across the part
<i>Mode</i>	Determines how the tool calculates its next path
<i>Maximum distance</i>	Uses a maximum distance between paths
<i>Tool diameter ratio</i>	Moves the tool over a percentage until fully milled
<i>Distance between paths</i>	Defines how far apart each path is
<i>Percentage of tool diameter</i>	Defines what percentage of the tool is moved over each pass
<i>End of Path</i>	For back and forth and one way motions, this defines where the tool ends up, whether it is inside the boundaries or outside the boundaries.
<i>Overhang</i>	Defines how far over the boundary the tool is allowed to move. This value is in percentage tool diameter.
<i>Tool side approach clearance</i>	Defines how far away from the side the tool remains when the end of path setting is on out.

Axial Controls the motions that the tool makes in the direction of the tool axis. This controls how deep each cut is and if there are multiple levels to be cut into the material.

Mode Defines how the axial strategy is calculated. There are three options:
Maximum depth of cut, *Number of levels* and *Number of levels without top*.

Maximum depth of cut Defines how deep the cutter is allowed to cut each pass. This depth will be cut each pass until the cutter reaches the bottom of the geometry defined.

Number of levels Splits the thickness of the geometry defined into a particular number of levels.

Finishing Used to move the tool one last time across a surface to insure that it is clean, smooth and to the exact size necessary.

Mode Controls whether or not there is a finish path. In other operations, this would control if there was a finish path and if so, where it is located. There are two options: *No finish path* and *At bottom*.

Bottom finish thickness Defines the thickness that is left on the bottom before the finish pass is made

The *HSM* tab will be covered later, but in short, this tab controls the machining methods when a high speed machine is to be used.


You are now ready to set the appropriate settings to continue.


Change the following settings:

<i>Tool Path Style</i>	Inward Helical
<i>Machining</i>	
<i>Direction of Cut</i>	Climb
<i>Type of contour</i>	Optimized
<i>Radial</i>	
<i>Mode</i>	Tool Diameter Ratio
<i>Percentage of tool diameter</i>	30%
<i>Overhang</i>	75%
<i>Axial</i>	
<i>Mode</i>	Maximum depth of cut
<i>Maximum depth of cut</i>	.125 in

This will define all the different machining parameters that is desired to be set for the facing operation. Next, a tool will be defined.

Tool tab

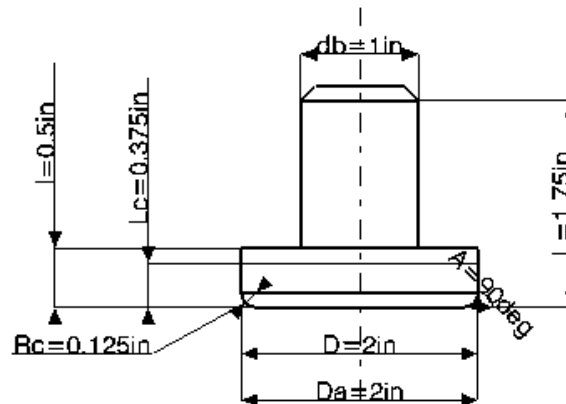
Select the tool tab.  If you remember, this tab allows you to define the tool that you want to use. Start by defining a facing cutter instead of an end mill.

Select the facing cutter icon.  The image in the contact sensitive window will change to a facing cutter. If you are unsure of the parameters that you can define, refer to the introduction where all the facing cutter parameters are defined.


Double select on the Nominal Diameter (D) parameter and change it to 2in. This will make your face be 2 inches in diameter.

Change the following parameters: Changing these parameters will work in the same fashion as changing the diameter parameter.

$$db = 1\text{in}, lc = .375\text{in}, l = .5\text{in}, Rc = .125\text{in}, L = 1.75\text{in}, Da = 2\text{in}$$



Speeds and Feeds tab


Select the speeds and feeds tab.  This will allow you to change the feedrate for the facing operation as well as change the spindle speed.

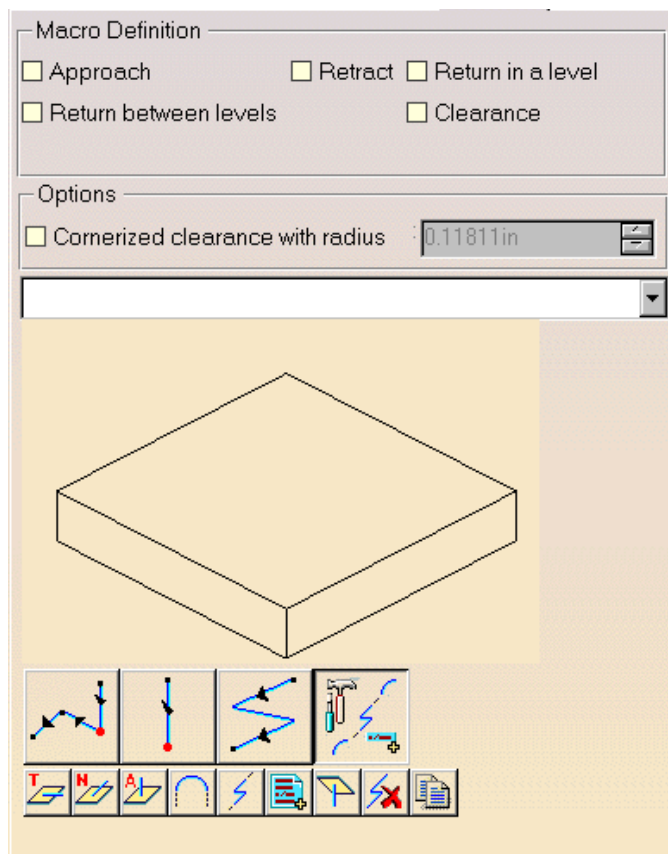
Change the following parameters:

Approach: 15in_mn, Machining: 25in_mn, Retract: 5in_mn

Since there is not a finish path defined, it is unnecessary to define a finish path feedrate. There is just one set of options left to define, the macros.

Macros tab










Select the macros tab.  There are several parameters available in the macros panel. Take a closer look at some of the different options.



The macro definition area defines what macros are going to be applied and what macros are not. By selecting and turning on different check boxes, you can control the different macros that will be applied.

The series of large icons at the bottom allow you to use pre-defined macros or to build your own. Most macros that you will use will be user built macros. Take a close look at the user macro definition icons.




-  Applies a tangential motion approaching or leaving the part
-  Applies a normal motion away from or approaching the part
-  Inserts an axial motion a given distance
-  Creates a circular motion approaching or retracting from the part
-  Creates a ramping or helical motion into a part
-  Inserts Post Processor words directly into the macro
-  Inserts an axial motion to a plane
-  Removes all motions in the macro
-  Copies a macro to all macro motions of that type. For example, if you build a specific approach macro, it will copy that approach to all approach motions of the return and linking macros.

Turn on the *Approach* and *Retract* macros. This will tell CATIA that you are going to define both an approach and a retract macro in your machining operation.

Use the drop down and select the *Approach* macro. This will define what macro you are going to build. When the macro has a green checkmark beside it, it means that this macro has been defined. When the macro has a red “X” beside it, then the macro has not been defined.

Select the user defined macro icon.  This will bring up the previous list of icons that allow you to build your own macro.

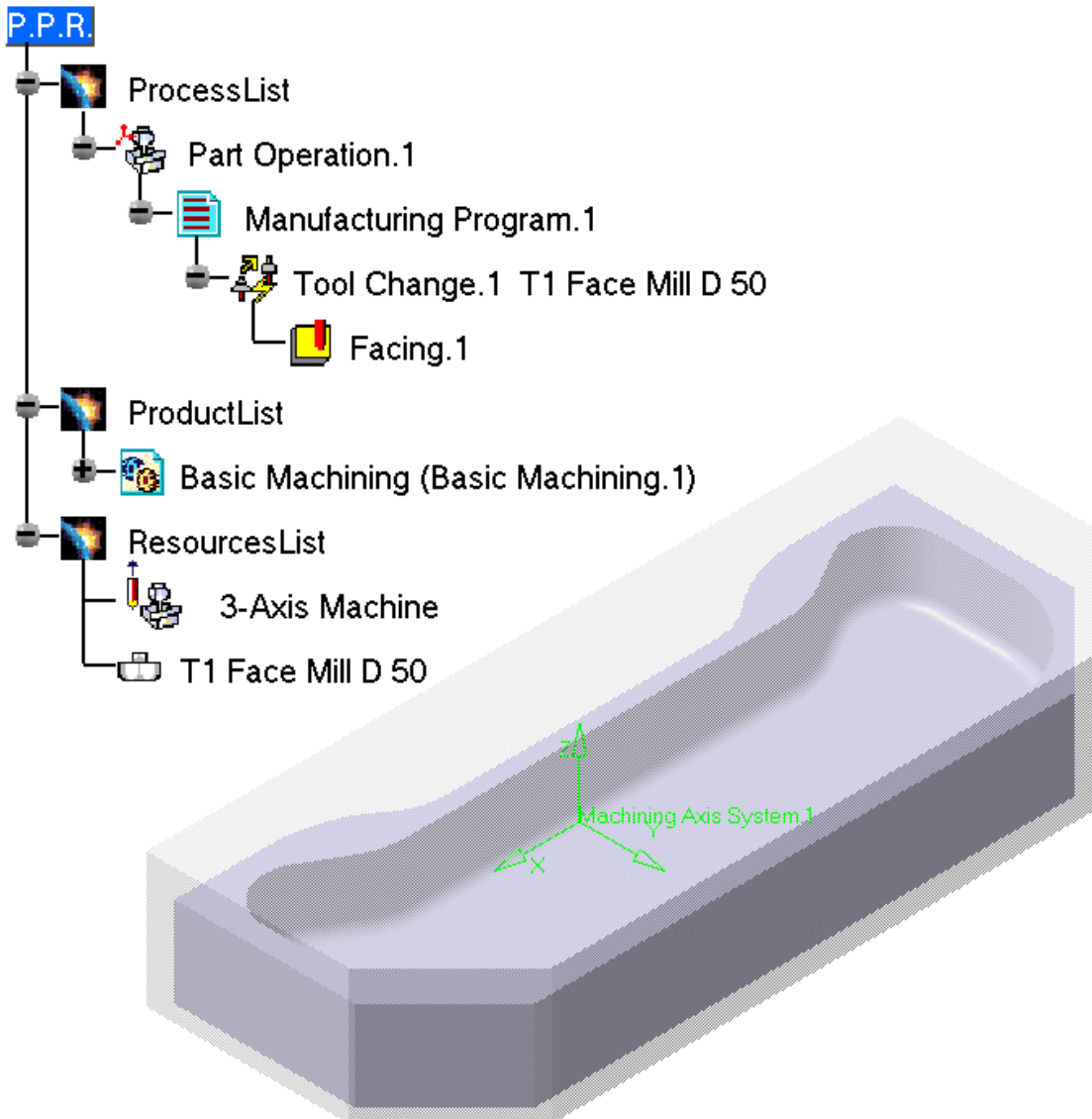
Select the axis to a plane icon.  Notice the plane is orange/ peach colored. This means that selecting a plane by hand is not necessary. Since you defined a safety plane in the part operation, that plane will be used as the axial motion plane.

Create the same macro for the retract motion. The steps for this will be the same. Use the drop down to select the retract macro. Select the user defined macro icon and then select the axis to a plane icon to create the motion.

Select **OK to the Facing panel**. Typically replaying your operations would be done within the machining operation panel, but for now, close it so you can save your document.

It is finally time to replay your machining operation. Replaying your operations can become very involved.

Save your document before moving on.



Replaying


Replaying your machining operations is a very important part of this class and in industry. You must make sure your tool paths are what you expect them to be before you send your program to the machine. A mistake in tool paths in CATIA costs only a little time to fix. A mistake in tool paths at the machine could cost much more due to machine breakage, tool breakage or other various problems.

Before beginning to replay, make sure a particular setting is set in the options panel.

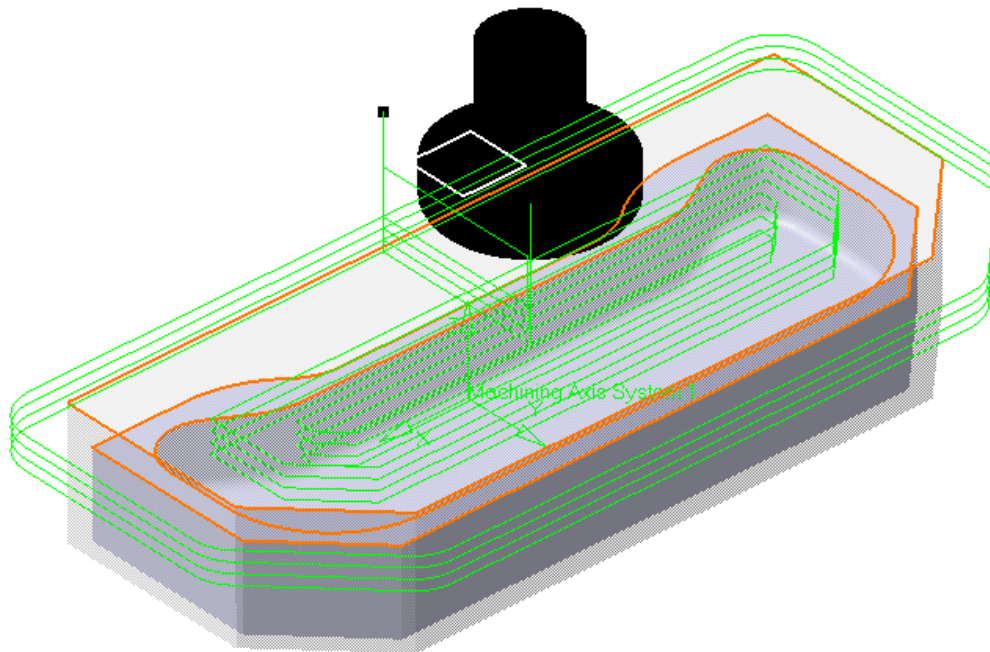
Select pull down menu *Tools, Options, NC Manufacturing, Display tab*. This is where the customization options for NC Manufacturing are held.

Turn on the check box for *Display tool near cursor position on tool path*. This option will show the tool wherever your cursor is along the replayed tool path. Although not necessary, this is a nice feature to have turned on.

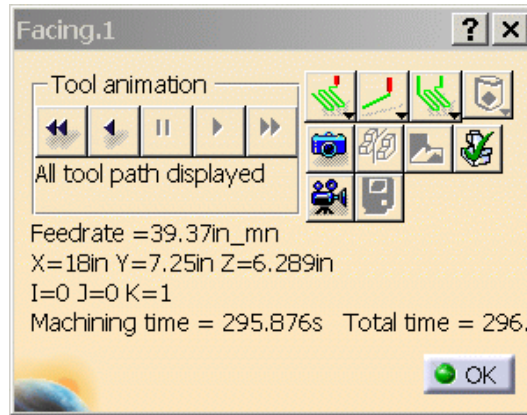
Select *OK* when done. This will save the new customization setting.

Select the tool path replay icon.  Again, when you select this icon, nothing happens. CATIA is waiting for you to select what machining operation you wish to replay.

Select the *Facing.1* machining operation. A tool path computation dialog box will display. This is where CATIA is computing the tool path that the tool will take. As soon as the computation is over, the tool path and the tool is viewable.



The replay window is also shown. Take a closer look at this window.



Under the *Tool animation* area is a series of play controls. These work similar to VCR play controls and will show the tool path being played forward, backwards or paused.



Controls the tool path replay steps



Continuous replay, the tool will not stop until it reaches the end of the operation



A plane by plane replay, the tool stops each time it moves to a different axial location



Feedrate change replay, the tool stops each time the feedrate is changed



Point by point replay, each visualization step that is made the tool stops



Controls the tool visualization



Shows only the last tool position



Shows only the tool axis vector



Shows all tool positions



Controls the color mode



Keeps all tool path colors the same



Shows the different operations with a different color of tool path



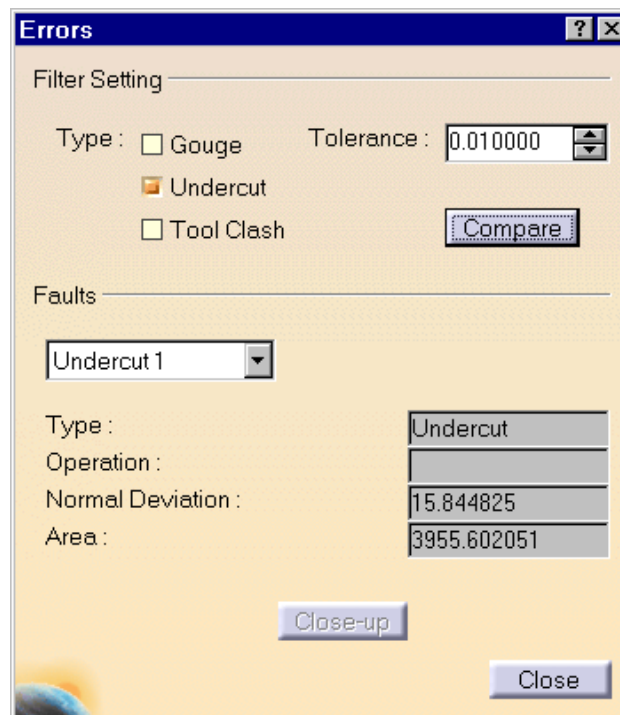
When surface machining, this allows the tool path to be shown at the tool center or where the tool contacts the surface.



Photo mode, shows a picture of the stock material with the machined material removed. If your tool path missed a spot, then the spot would show up in the picture mode. While in photo mode there are two icons that become available.



Analyze photo mode, this brings up another window for checking for errors. With the *Errors* window you can check for gouges, undercuts and tool clashes.



The analyze photo mode utilizes the final design part that you specified and compares it to the part that was milled. If there are deviations based on a specified tolerance, then it will be reported here.



Adjusts the settings of the photo mode, this will control the different color settings as well as the resolution settings for the photo.



Checks to insure that the machine programmed for has enough travel and in the case of a multi axis machine, enough rotation angles. This icon only works when Virtual NC has been installed.



Simulation mode icon, shows the tool making passes around the part and shows the material being removed as it makes its passes.



Save video result, allows you to save a CATIA Geometric Representation (CGR) file of the milled part. This CGR file can then be imported back into the assembly if desired.

Select the beginning of tool path icon.



This will remove the visualization of the tool path already on the part.

Select the play icon.



The tool will pass over the part showing the tool path replay.

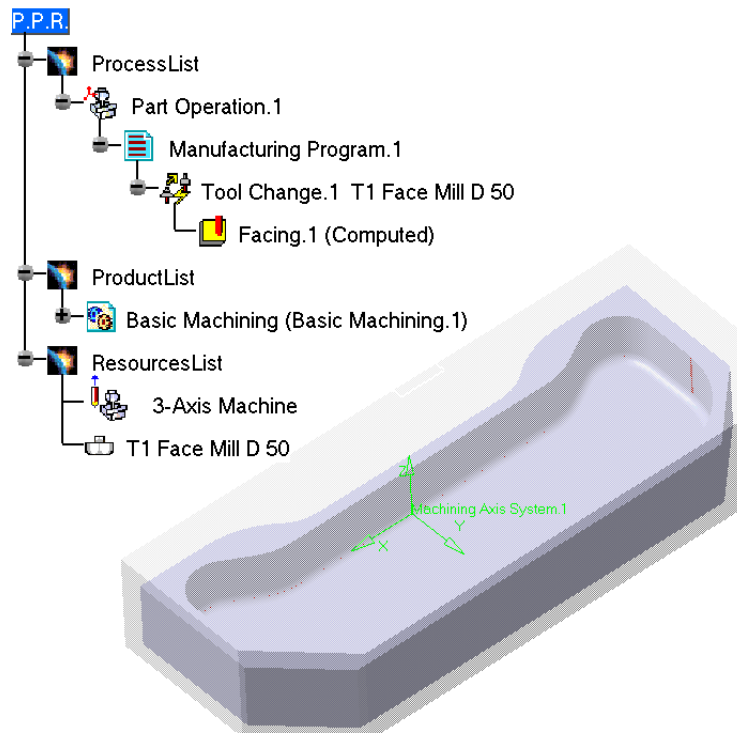
To see what the final milled stock material is going to look like, use the photo mode.

Select the photo mode icon.



CATIA will calculate for a second and then display an image of the final milled stock material. Notice the top of the part is a dark green color. This denotes where the stock was milled.

Select OK. This will take you back to the machining process. It is now time to continue machining.



Contour Milling

Contour milling will allow you to create a machining operation that follows along the outside of a part. You will use contour milling to mill around the outside of the stock material until it is at the final part thickness.

Before beginning to contour mill, it is strongly advisable to hide the stock material. The section from here on will assume that you have the stock material hidden and out of the way.

Select the profile contour milling icon.  Nothing will happen until you select where you want to insert the milling operation.

Select the *Facing.1* machining operation. A *Profile Contouring.1* machining operation is added to the tree under the *Tool Change* operation.

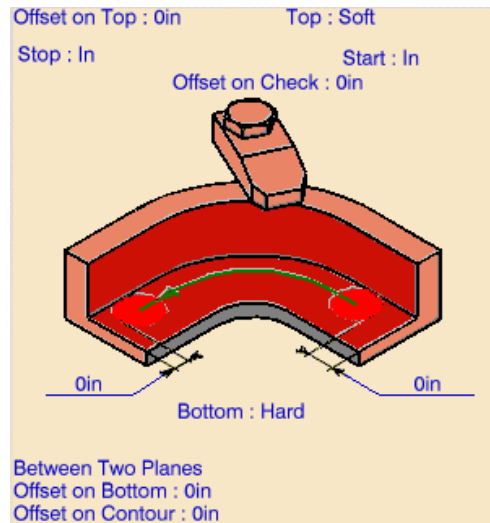
The *Profile Contouring* window displays. Notice all the icons are the same. You will find the same six tabs for all the different machining operations.

Profile Contouring Methods

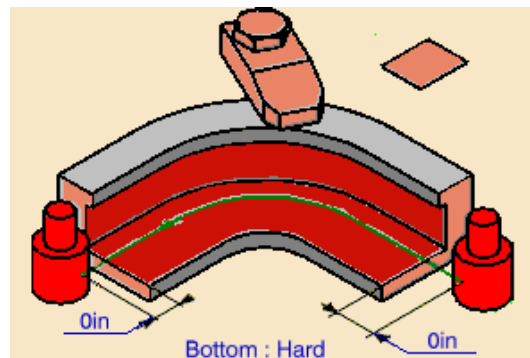
There is quite a range of methods to machine the profile of a part. In fact, there are enough methods that a separate section discussing them is necessary. The four main different types of profile milling are *Between Two Planes*, *Between Two Curves*, *Between Curve and Surfaces*, and *By Flank Contouring*. Each of these types will be investigated further. In order to change the type you must press the third mouse button while on the current contouring method.

Between Two Planes

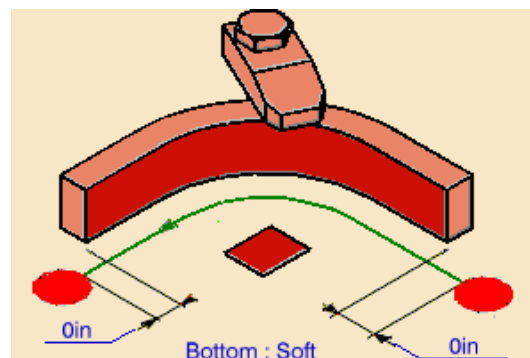
Milling between two planes is the most common profile contouring method.



This is the default method of milling between two planes. There are two options that are of major concern, the top and bottom conditions. Right now the top is set to a soft condition, meaning that the tool is free to pass above it. The bottom is hard, denoting that the tool cannot pass beyond a selected base. Changing the top to a hard condition will impose an upper limit on the tool as shown below.

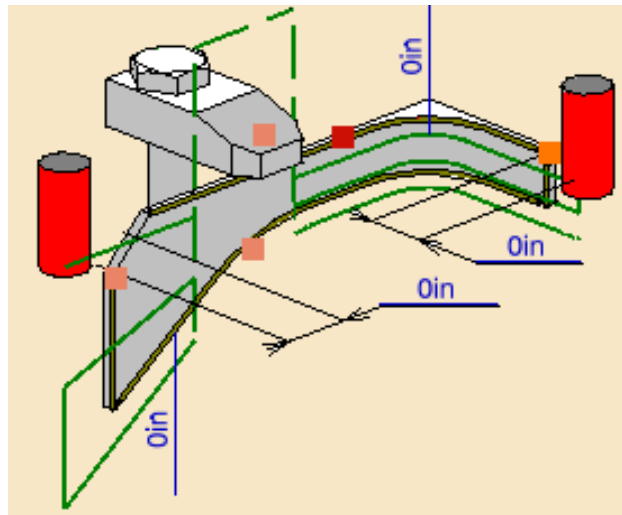


A soft bottom will allow the tool to pass beyond the bottom of the part. For many cases of milling around the outside profile of a part you would want the top and bottom to be soft, resulting in a profile such as the one shown below.



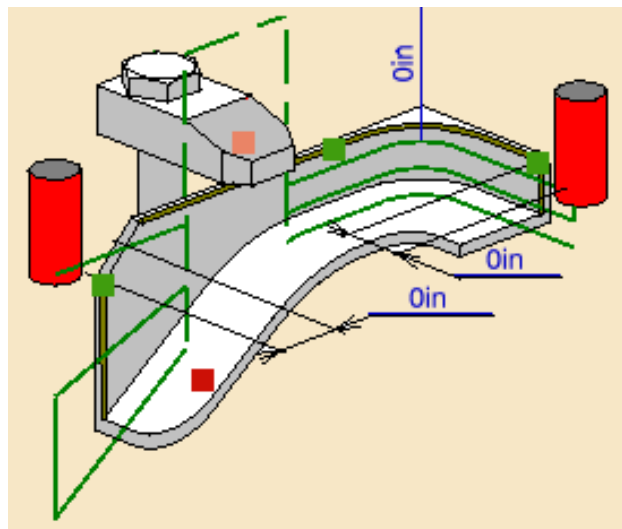
Between Two Curves

Machining between two curves is similar to machining by planes. With machining by curves however, it is not necessary to have completely planer surfaces to machine. The geometry can be defined by curves as well as surfaces and faces, but most generally you will want to define the geometry by curves or edges of your part. This option is specifically useful when you have a non-planar contour along the top of your part. The tool will follow that contour along the top and then begin to flatten out as it makes passes towards the bottom.



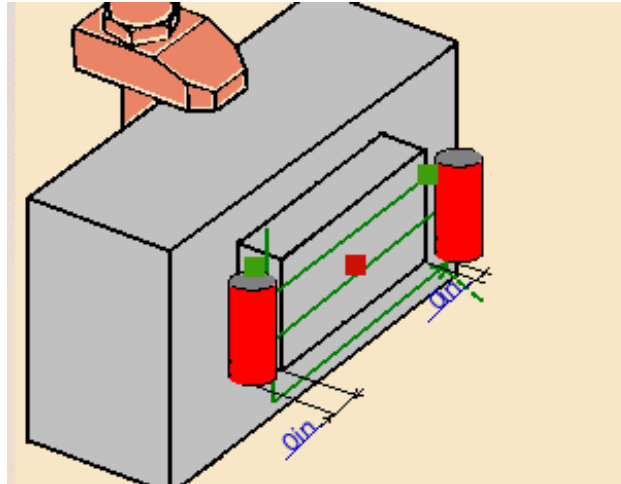
Between Curve and Surfaces

This option works the exact same way as the between two curves option. The only difference between the two options is the ability to define a hard limiting surface along the bottom of your profile.



By Flank Contouring

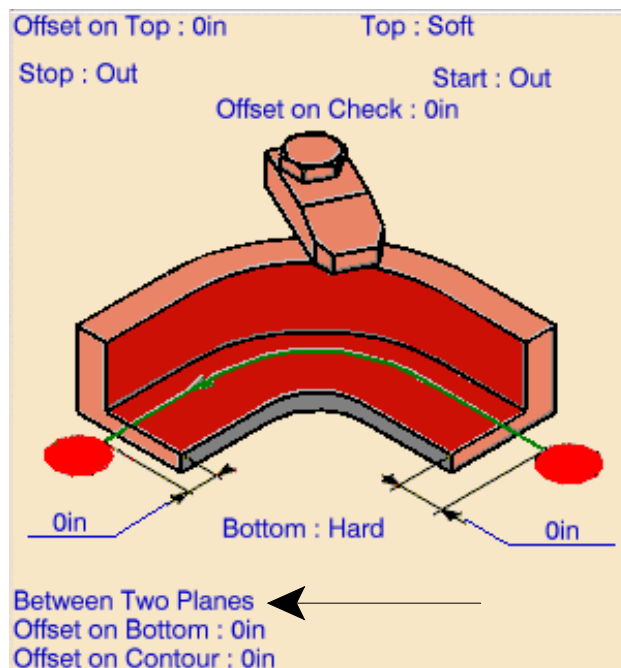
This is the most basic of profile contouring. Flank contouring only mills straight sides. If you have a profile with a curve or rounded edge of any kind, this is not the option to use. Flank contouring also only mills one element. If a profile has several segments, again, this will not mill it.



Back to milling your part.

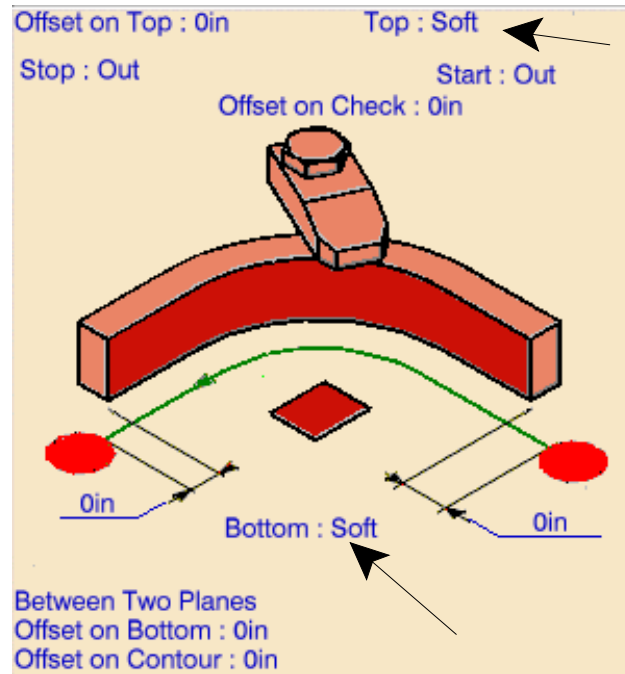
Since the top is planar, it is unnecessary to use the curve milling options. The profile is not simple enough for flank contouring either, so the between two planes option will be utilized.

Be sure the profile contouring method is set to *Between Two Planes*. This option is the bottom left corner of the sensitive area as shown below.



There is no upper limit or lower limit on your part, so the top and bottom limits need to be set to soft boundaries.

Set the top and bottom to soft. The soft boundaries will allow the tool to pass above and/or below the contour if necessary. There are a few more options to define before defining the geometry.



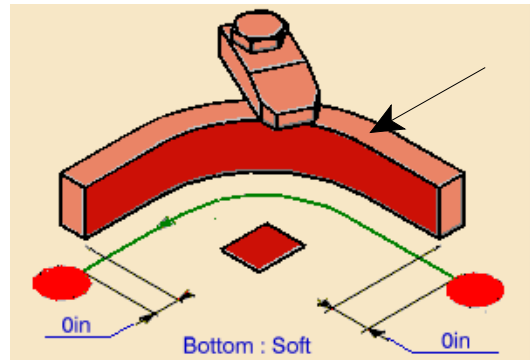
Change the *Offset on Bottom* to be $-.020\text{in}$. This can be accomplished by double selecting the *Offset on Bottom* parameter and then keying in the $-.020$ distance.

Why did you make the tool go below the bottom plane by twenty thousandths? The reason you did this was to set the tool below the bottom contour. When an end mill is used, the corner on the edge may dull slightly, thus creating a small radius around the bottom. If you run the tool exactly along the bottom of the profile, a small burr, or edge will be left behind. This part will then have to be taken to get de-burred. By offsetting the tool $-.020$ along the bottom you avoid having that burr and you end up with a nice sharp straight edge when the cutter is done. A negative offset of $.020$ is only a rule of thumb. Sometimes it may be necessary to offset the cutter a little more, sometimes a little less.

The start and stop area definition is unnecessary. Since you are milling a closed profile, the difference will not be noticeable. But, if you want to change the conditions, using the third mouse button select on the stop or start condition and change it.

Now it is time to define some geometry. First, the top plane will be defined.

Select the top of the sensitive area part. This will define the top of the part. Select the area shown below.



Select the top of the part. This will define the top limit of the profile. The top of the part is not necessary, but is useful in determining the number of axial steps to make. The top of the sensitive profile should turn green and the *Profile Contouring* window should display again.

Using the third mouse button select on the side of the profile part. This will bring up the different selection options. The selection options that you see here are the same selection options that were discussed with face milling.

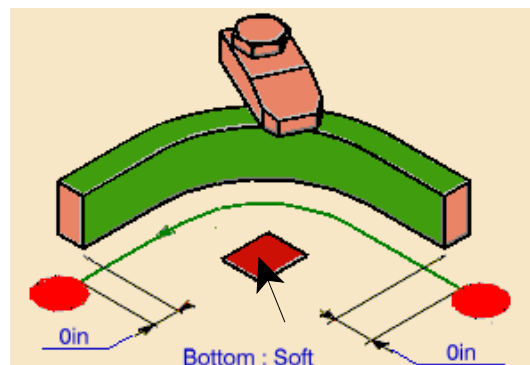
Select *By Belt of Faces*. This will allow you to define your profile by a series of faces.

Select all of the exterior faces around the part. If you want, after you select a face on the back side, you can select the navigate on faces icon on the *Face Selection* toolbar. This will select all of the faces around the back of the part automatically.

Be sure the guide arrow points towards the outside of the part and select *OK*. The guide arrow is the side of the profile that will be milled. If the arrow is pointing inwards, then the inside of the part will be machined.

The last thing that needs to be defined is the bottom of the part.

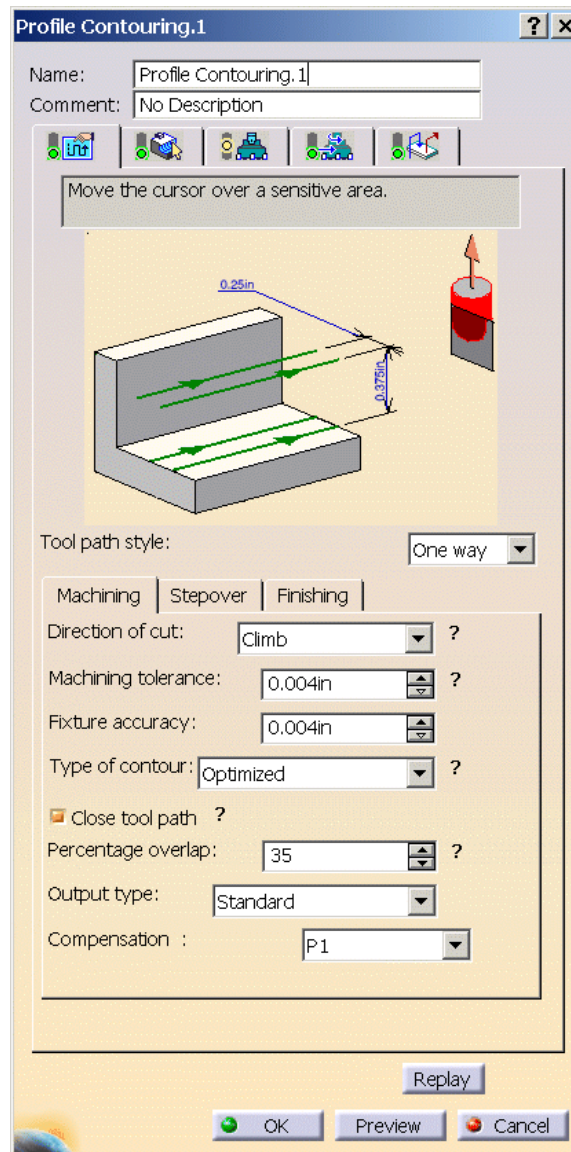
Select the bottom plane definition from the sensitive area. This will be the red plane that is in the center of the profile as shown below.



Select the bottom of the part. This will define the bottom of the profile that will be cut. The geometry definition is now complete. If an open profile were to be milled, you could define start and ending limits and if a clamp or other holding fixture was present you could define check curves to avoid them.

Change to the machining operation parameters tab. This will be the first tab in the series.

There are a few different options in the machining parameters panel.



Tool Path Style

- Zig Zag* Runs the tool back and forth across the profile
- One Way* Makes one directional cutting

Machining

Close tool path Checkbox to select if the tool path is on a closed contour

Stepover


Sequencing controls whether the radial passes get made first or the axial paths

Automatic draft angle Puts a draft angle on your part as it is being machined

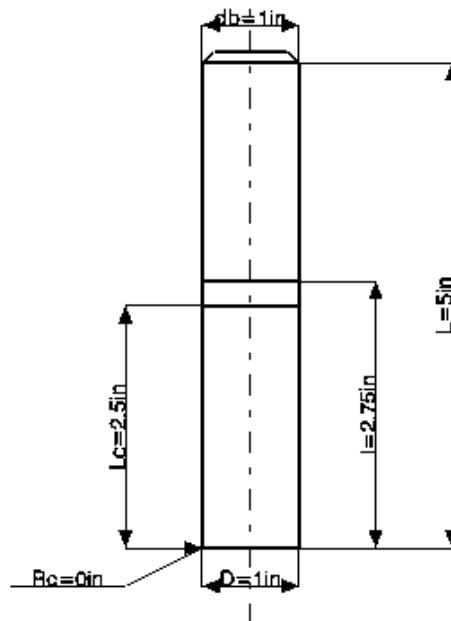
Change the following settings:

This will set all the settings to something more desirable. Next, the tool needs to be defined.

Change to the tool tab. You'll note the face mill is still defined. That is because you put the profile contouring operation after the facing operation that was under the face mill tool change. This will not hurt anything. You are going to change the tool to an end mill, and a tool change will automatically be inserted.

Change the tool type to End Mill.  You'll notice in the tree a new tool change operation has been added and the profile contouring operation was moved under that tool change. Now it is time to define the tool.

Change the tool parameters to those shown below.



$db = 1\text{in}$, $L = 5\text{in}$, $l = 2.75\text{in}$, $Lc = 2.5\text{in}$, $D = 1\text{in}$ and $Rc = 0\text{in}$

The speeds and feeds that you used for the face milling operation will still be present in the feedrate options. However, it is still a good idea to take a look at them to make sure they are right.

Change to the feeds and speeds tab. A few values will be changed here. For example, the finishing feedrate is somewhat slow. This could be sped up some as well as the retracting feedrate.

Change the following parameters for the speeds and feeds:



Retract 20 in_mn

Finishing 12 in_mn

The macros that you used in the facing operation are still the same as those used here. Check the macro tab to make sure the macros are right but you will find that you do not have to re-define any of them.


Select the *Replay* button at the bottom of the *Profile Contouring* window. This will

display the replay control window.

Rewind and replay the tool path. This can be done by selecting the rewind icon () and then selecting the play icon ().

Simulating the Replay

This time you are going to simulate the replay. This will show actual material removal from the stock part.

Select the simulation mode icon.  When the mode loads you will get a window similar to that of the picture mode.

Select the play icon. Watch the material on the screen. Notice that as the tool passes over the part the material is being removed. This will turn out to be the most effective method of replaying your manufacturing programs.


After the replay ends, the part is shown as it would end up on the machine. If there were missed spots, they would be identifiable by the simulation replay.

Select *OK* in the replay window and the *Profile Contouring* window. This will finish the replay and the profile contouring.

Next, it is time to look at the last of the basic milling operations, pocketing. This would be a good time to save your document before continuing.

Manually Defining Tool Changes


Sometimes it is desirable to manually insert a tool change. Maybe you want to be able to measure the part without getting out of the machining operation definition panel or for some other reason. By manually defining a tool change, you can also define a tool assembly or the tool holder that goes with the tool. Defining the tool holder can be very important in the replay or validation process in order to determine how much clearance there is between the tool and the part. Take a look at inserting a tool change before creating the pocket operation.

Select the tool change drop down.  To access the drop down, select the small black triangle on the bottom right corner of the icon.

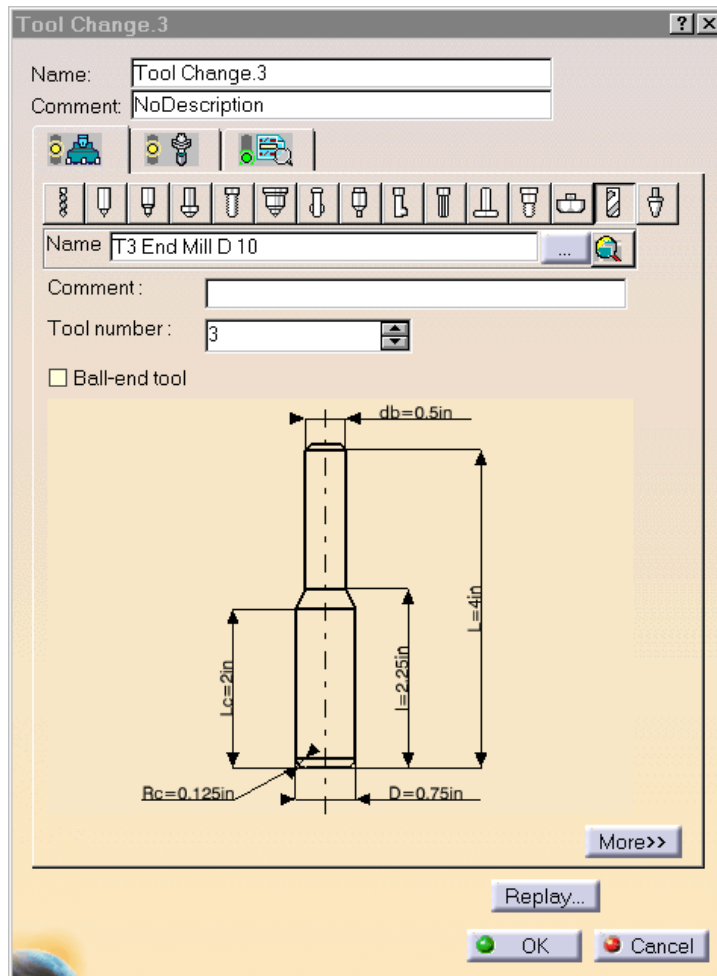
The tool change series of icons will display.



Each tool shown here was discussed in the introduction of this book. You want to change the tool to a different end mill than the one used to profile the part.

Select the end mill tool change icon.  You will see a new tool being added to the resources list. Note that the tool change is not inserted into the program though. The tool change is not inserted because CATIA does not know where to place the tool change.

Select the *Profile Contouring.1* machining operation. A tool change is added to the manufacturing program. The tool change window is displayed.



There are three tabs across the top of the tool change window. The first tab defines the tool, the second tab allows you to define the tool assembly and the last tab allows you to add additional post processor words to the tool change command. The last tab will not be modified.

Selecting the *More*>> button at the bottom of the window shows additional information that can be entered about the tool.

The image displays two screenshots of the CATIA tool parameter dialog box. The left screenshot shows the 'Geometry' tab with the following parameters:

- Nominal diameter (D): 0.3937in
- Corner radius (Rc): 0.07874in
- Overall length (L): 3.93701in
- Cutting length (Lc): 1.9685in
- Length (l): 2.3622in
- Body diameter (db): 0.59055in

The right screenshot shows the 'Technology' tab with the following parameters:

- Number of flutes: 4
- Way of rotation: Right hand
- Machining quality: Either
- Composition: One piece
- Tooth material: High speed steel
- Tooth description: [Empty field]
- Tooth material desc.: [Empty field]
- Tool rake angle: 0deg
- Max plunge angle: 120deg
- Max machining length: 0in
- Max life time: 0s
- Coolant syntax: [Empty field]
- Weight syntax: [Empty field]

The image displays the 'Compensation' tab in the CATIA tool parameter dialog box. It contains the following parameters:

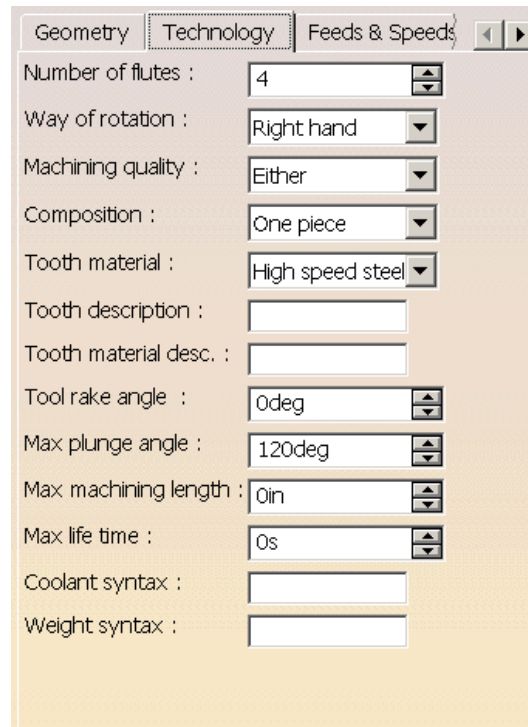
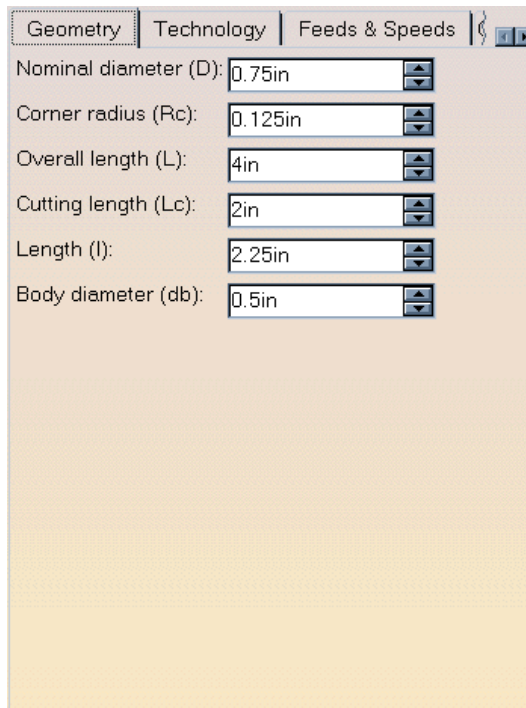
- Finishing cutting speed: 0in_mn
- Finishing feed/tooth: 0in_turn
- Axial depth of cut for finish: 0in
- Radial depth of cut for finish: 0in
- Roughing cutting speed: 0in_mn
- Roughing feed/tooth: 0in_turn
- Axial depth of cut for rough: 0in
- Radial depth of cut for rough: 0in

The *Geometry* tab (shown on the top left) allows you to enter the data about the tool without double selecting on the tool parameters. The *Technology* tab (shown on the top right) allows you to add additional information about the tool. The *Feeds & Speeds* tab (shown on the bottom) allows you to define specific speeds and feedrate information for the tool. The *Compensation* tab (not shown) allows you to define cutter compensation values for the tool. Cutter compensation is necessary when a tool has been re-sharpened and is slightly smaller than the original size. For example, a 1" diameter end mill may get re-sharpened and after re-sharpening the tool diameter is .975". The .025" difference is defined in the compensation tab to correct for the re-sharpening.

Take a closer look at the *Technology* tab.

<i>Number of Flutes</i>	Defines the total number of flutes on the tool
<i>Way of Rotation</i>	Defines the direction that the tool should be spun
<i>Machining Quality</i>	Defines the finished cut of the pass. You have three choices: <i>Rough</i> defines a rough edge that will need smoothing, <i>Finish</i> defines that you will end up with a smooth, finished cut, and <i>Either</i> means that it can be used for roughening or finishing.
<i>Composition</i>	Defines if the tool can be used alone (one piece) or if it requires an insert holder
<i>Tooth Material</i>	Defines the material that the teeth of the cutter are made of
<i>Tooth Description</i>	Allows you to enter any additional description of the tool
<i>Tooth Material Description</i>	If the tool is of a special material, a brief description of the material can be made here
<i>Tool Rake angle</i>	Allows you to define the rake angle that the tool will run at
<i>Max Plunge Angle</i>	Defines the maximum plunge angle that the tool can handle
<i>Max Machining Length</i>	Defines the maximum depth that the tool can cut in one pass
<i>Max Life Time</i>	Defines the maximum time that the tool can be used in seconds
<i>Coolant and Weight Syntax</i>	Allows you to add special syntax words or commands about the coolant or the tool weight

Change the following *Geometry and Technology* information:



This will fully define the tool itself. Next, define a tool assembly.

Select the tool assembly tab.

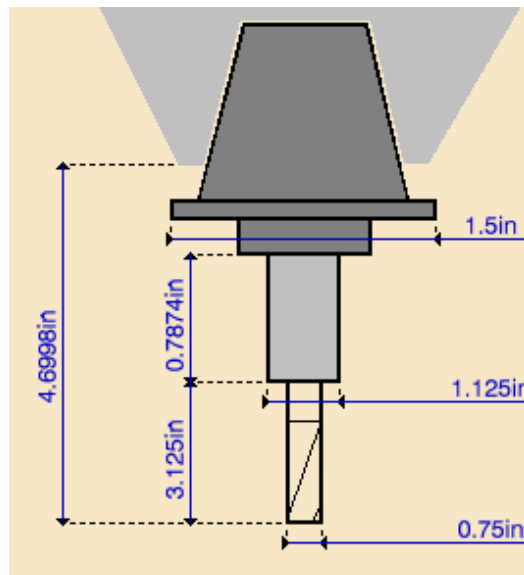


This tab will allow you to define a tool assembly.

Since no database is implemented yet, the first question that CATIA will inquire about is the name of the assembly.

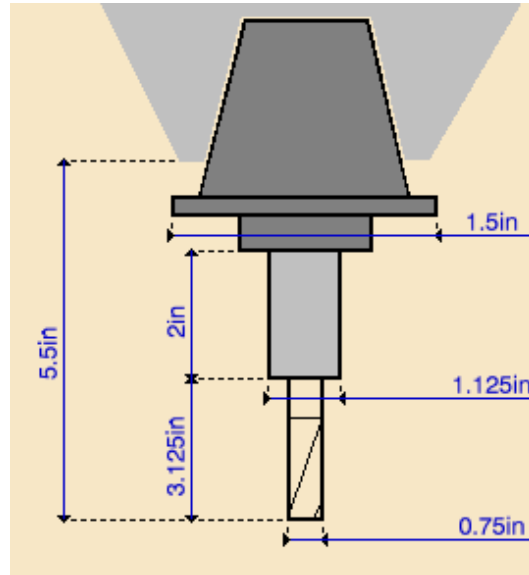
Change the *Name* to Tool Assembly 1.

Do not forget to press Enter after you key in the name. This will tell CATIA that you are going to be making a tool assembly to hold the tool. Another sensitive area displays.



This is a simple representation of the tool holder as well as the tool itself. Note some of the information is already entered about the tool. Things such as the tool diameter and the tool length is already an even number.

Change the parameters to match the following:




This will define the tool assembly that will be needed for this tool.

Select OK. This inserts the tool change operation into your manufacturing program.

Pocketing

Pocketing cuts out the inside area of a part. Pocketing operations can also clear out an area, other wise known as an open pocket but that will be investigated later.

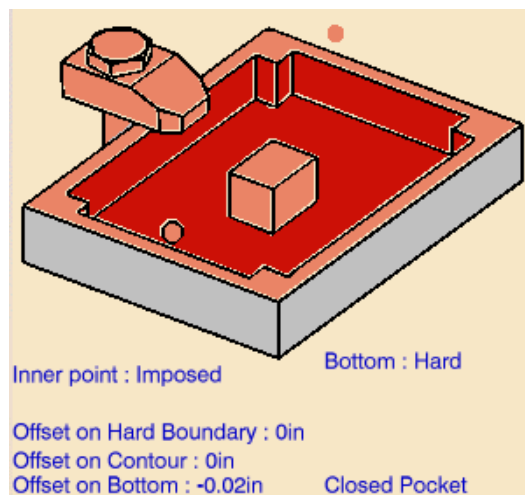
Select the pocketing icon.  As usual, CATIA needs a location for the machining operation.

Select *Tool Change.3* to insert the pocket operation after. The pocket operation will be inserted and then the *Pocketing* window will display.

The sensitive area shown is similar to the ones you have seen before. There are some areas in red, like the bottom of the pocket and the boundary, and there are some areas that are a peach, or pink color.

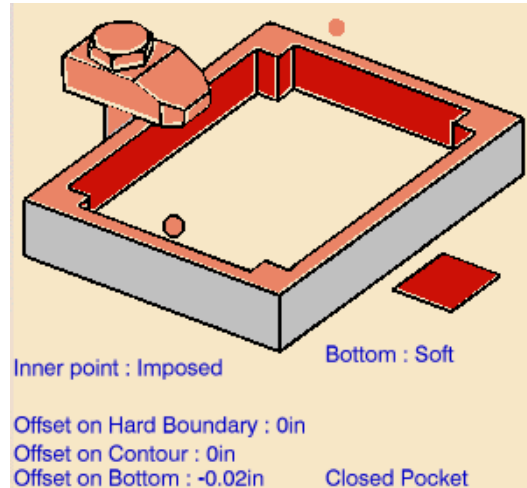
There are several pocket types that can be made. Take a quick look at the differences between them.

Hard Bottom, Closed Pocket



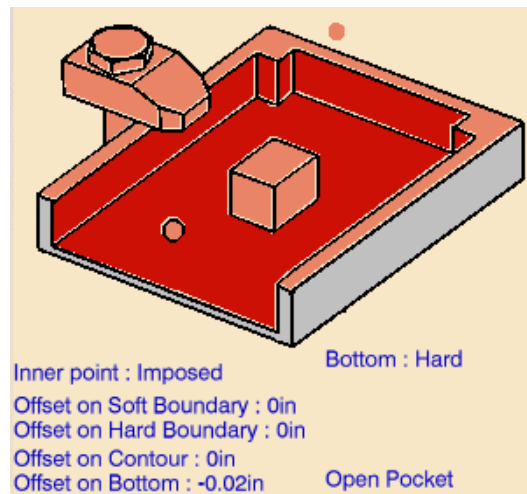
This pocket is denoted by having a solid bottom and has boundaries on all sides. This is your most common type of pocket. Note the optional selectable cube in the center of the pocket. This allows you to define islands or areas that the tool should not cut inside a pocket.

Soft Bottom, Closed Pocket



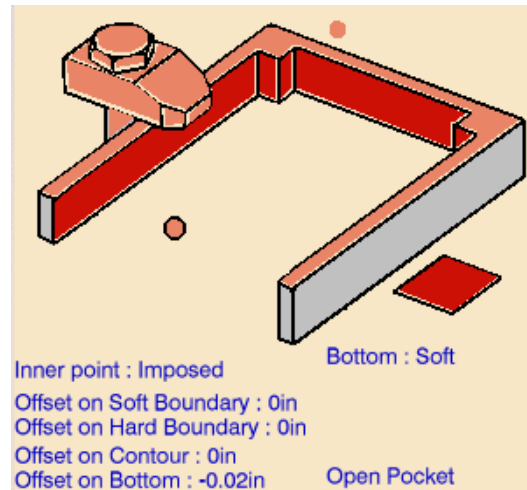
In this instance, the bottom is considered non-existent. The tool is allowed to cut all the way through the pocket, thus creating a large open hole. With this type of pocket, you would most commonly want to set a bottom offset to drop the tool slightly below the bottom edge of the pocket.

Hard Bottom, Open Pocket



Open pockets work in a similar fashion that the closed pockets do. The major exception is the pocket does not need a full boundary to complete the operation. The tool is also allowed to pass beyond the open end of the pocket.

Soft Bottom, Open Pocket




This type of pocket makes the pocket open on a side as well as open through the bottom.

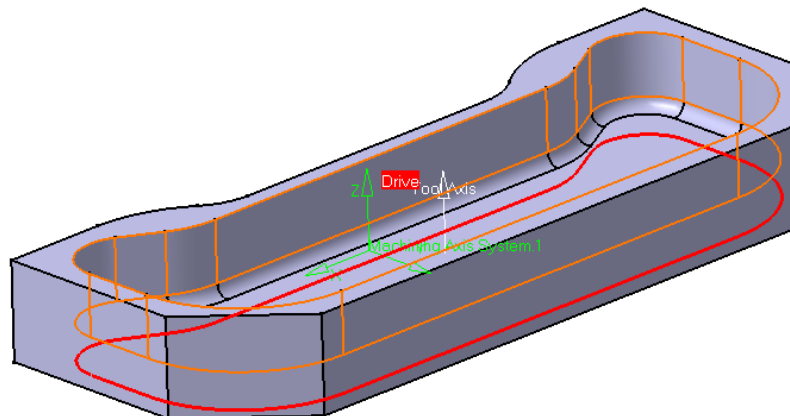
Be sure the *Bottom* is set to *Hard* and the pocket is closed. Given the geometry that is present in the part as well as the different options given for pockets, a hard bottom and closed pocket is the pocketing method of choice.

Using the third mouse button select on a sidewall of the pocket. This will show the alternate selection options.

Select *By Belt of Faces*. Remember, *By Belt of Faces* will allow you to select the boundary by the faces themselves and not just the edges.

Select the inside walls of the pocket. This will define the boundary drive elements for the pocket. Do not press the navigate icon. Selecting this icon will make the face selection navigate around the bottom of the sidewalls as well as the sides. This is not what you want.

Select the preview icon.  The preview icon will show a heavy red curve at the bottom of the part where the boundaries of the pocket are.



Select OK when done. This will define the outside perimeter of the pocket. Next, the bottom will be defined.

Select the bottom of the pocket from the sensitive area. This will wait for you to select the bottom of the pocket from the geometry.

Select the inside bottom of the pocket. This will define the bottom of the pocket. The top of the pocket is the last thing that will be defined.

Select the top edge of the pocket from the sensitive area and then select the top of the part. This finishes the geometry definition for the pocket.

Before continuing notice the *Offset on Bottom* has a negative offset. This will undercut the pocket. The -0.02in was a carry-over from the profile operation. Since the parameter is the same between the two operations the values carry over.

Change the *Offset on Bottom* to be 0in. This will keep you from undercutting the pocket operation.

Switch to the machine operation parameters tab. You will note that the values are again carried over from the profile operation.

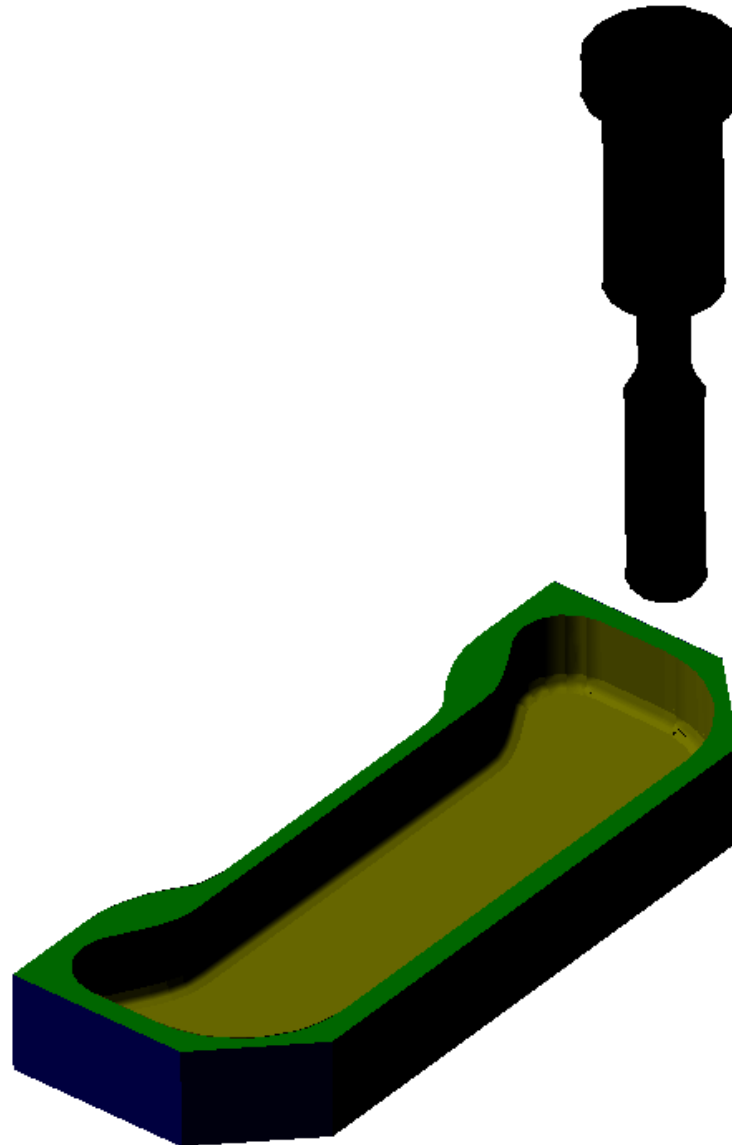
Add a complete finish path. A complete finish pass adds a finish path on the side as well as the bottom.

Change the *Side finish thickness* to .05, the *Bottom finish thickness* to .05 and turn the *Avoid scallops on bottom* checkbox on. *Avoid scallops on bottom* will make the cutter cut in passes that will minimize the number and height of the scallops found along the bottom of the pocket.

All of the rest of the information for the pocket has been defined. Normally, you would have to define the tool but since the tool was defined with the tool change it is unnecessary to define it here. You can check the different tabs to make sure everything looks in order, but you will find that everything is good to go.

Select OK. This would make a good point to save and replay your program.

Notice the tool holder only shows for the last tool change. The other tool change operations do not have a tool assembly defined with them.

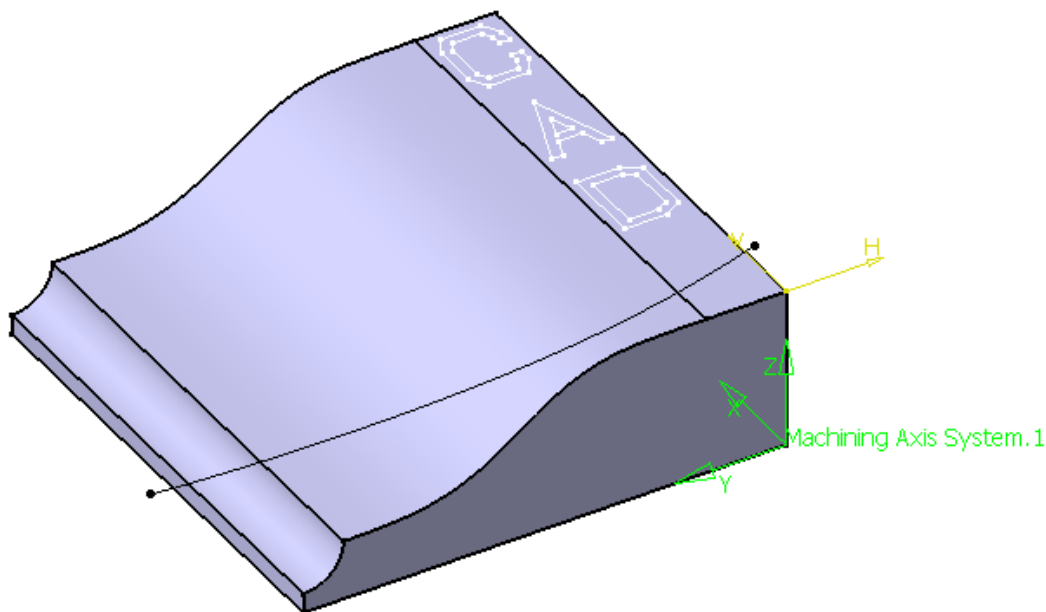


Save and close your document.


Curve Following

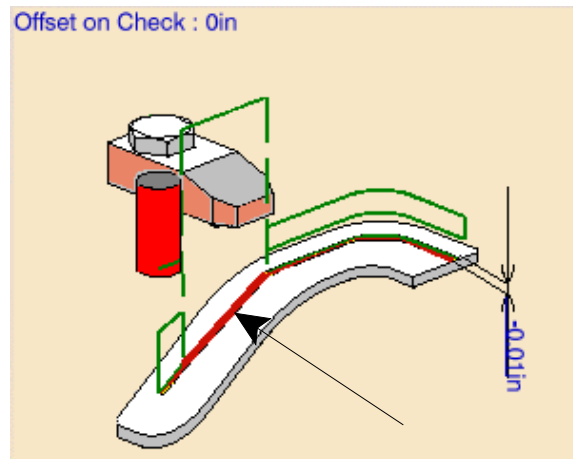
Curve following is one of the more simple machining operations. A curve of any type, or series of curves, needs to be defined. This can either be done by a sketch, or through wireframe techniques. The tool is then instructed to follow along the curve, at a given distance below the curve. Note the word *below* was used. CATIA assumes that the curve or sketch that will be used is placed above, or on top of the area to be machined. If the sketch is below the machine area, a few tricks have to be applied, which will become obvious as you work through the exercise.

Open the Curve Following Machining process. This is a machining process that has already been defined and started for you. This can be found in the *Curve Following* folder.



This document has a few features that will be machined. First, the 3-D curve that flows through the model will be machined. Secondly, the letters C, A and D will be scribed into the block. The last step will be to mill the front ledge. The front ledge will be machined with the point-to-point icon, instead of the curve following icon.

Select the curve following icon.  This will bring up the *Curve Following* window. The sensitive area for this option only has a few options to it. *Remember, you need to select where you want to insert the curve following.*



Select the heavy curve following line from the sensitive area. This is indicated above. This is the guide curve you select when you want to define the curve that will be followed.

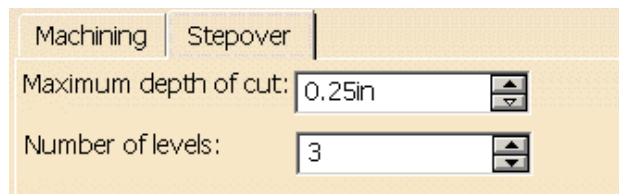
Notice the edge selection toolbar appears. This is because edges can also be selected as lines and curves for the tool to follow along.

Select the 3-D curve that flows through the part and select *OK*. This will define the curve that the tool will follow. If there are several curves, they can be selected sequentially.

Change the axial offset to 0in. This will make sure the line itself gets machine. The offset can either be positive or negative.

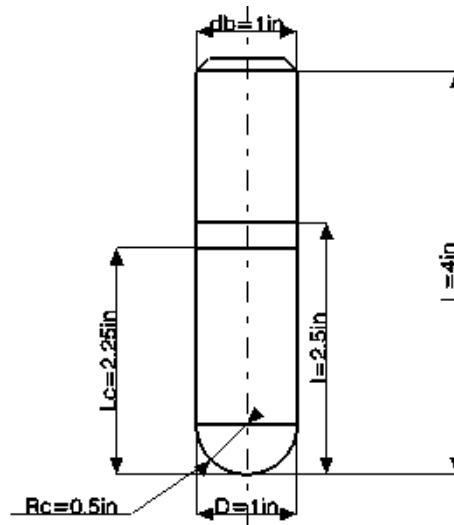
Change to the machining operation parameters tab. There are only two tabs within the machining operation parameters. The *Machining* tab holds information about the machine tolerance and fixture accuracy and the *Stepover* tab controls how many levels are going to be cut before the selected curve is cut.

Select the *Stepover* tab and change the options to match what is shown below. This will make the tool cut three passes, each .25" deep. The final pass will be on the same level as the curve.



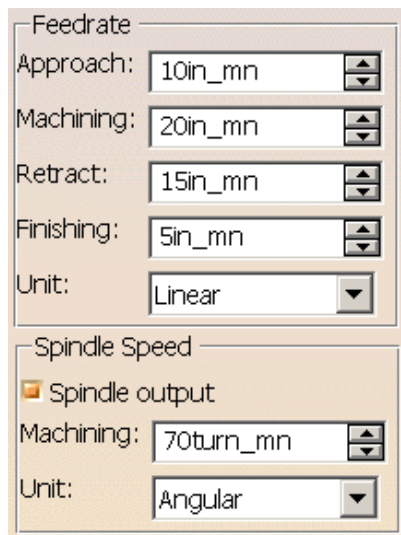
Change the *Tool path style* to *One Way*. This will make the tool only cut in one direction. Since a one way path is being used return macros will be necessary.

Create the following end mill to cut the curve.



Rc=0.5in, D=1in, Lc=2.25in, l=2.5in, L=4in and db=1in

Set the following feedrates and spindle speed.



The last step is to define the macros.

Define the following macros, all with a straight retract to the clearance plane.

Approach


Retract

Return in a level

Return between levels

Select **OK** when done. Don't forget to replay the machining operation.

Next, the letters C, A, and D will all be scribed into the block.

Select the curve following icon. Define the curve following process after the previous curve following process.  This time the entire sketch will need to be selected.

Expand the *Curve Following Assembly* and the *Curve Following* part until you find the *C-Sketch*. Remember, to expand the tree, all that needs to be done is to select the plus (+) beside the branch.

Select the red guiding element from the sensitive area. This time the entire sketch will be machined.


Select the *C-Sketch* from the specification tree and select *OK* when done. The entire sketch will be defined to be the machining elements.

Since the letters are going to be scribed into the part a negative offset will be necessary.

Change the offset to be *-.050in*. A depth of .050 will be deep enough to recognize.

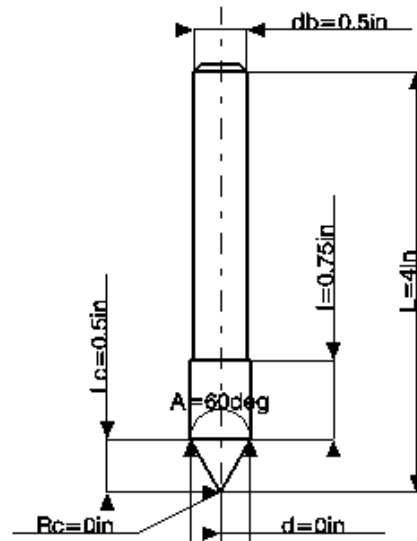
In the machining operation parameters, change the maximum depth of cut to be *.025in* and the number of levels to be *2*. This will make the scribe cut the letter twice, each at .025" deep per pass.

Since there is not a scribe tool icon, a specialized canonical mill will have to suffice.

Change to the tool tab and select the canonical mill icon.  If you remember from the introduction, canonical mills are cutters that taper to the end. The mill that is going to be used here is going to taper to a end radius of zero.

Change the comments field to read *Scribe tool*. This added bit of information will help you remember what type of tool you are modeling.

Change the tool parameters to the following. Note: the tool diameter (D) has been omitted. This value will be calculated based on other defined values.



d=1in, Rc=0.5in, A=60deg, Lc=0.5in, l=0.75in, L=4in and db=0.5in

The feedrates and spindle speed are still set from the last curve following machining process that was used, hence they do not need to be changed. However, it is still a good idea to take a quick look at them to make sure they are right. The same goes for the macro.

Select OK when done. As always, don't forget to replay the operation before calling it done.

Create a curve following machining process for the A and D. The same series of steps apply and remember, you will want to select the A and D from the specifications tree.

Point to Point

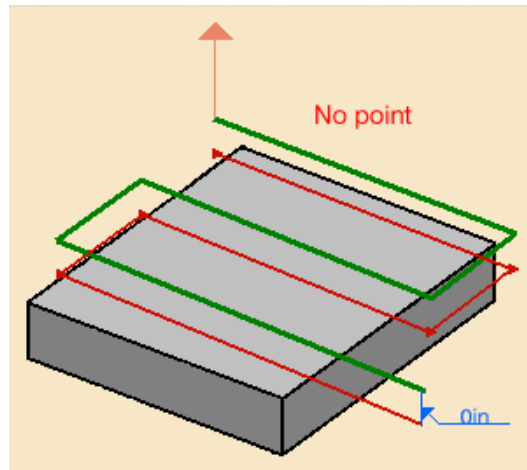
Point to point curve following is by far the easiest type of machining process that can be made. All that needs to be defined is a series of points and then the tool will go from one point to another in a connect-the-dots fashion.

Define a point to point machining process after the last curve following operation.



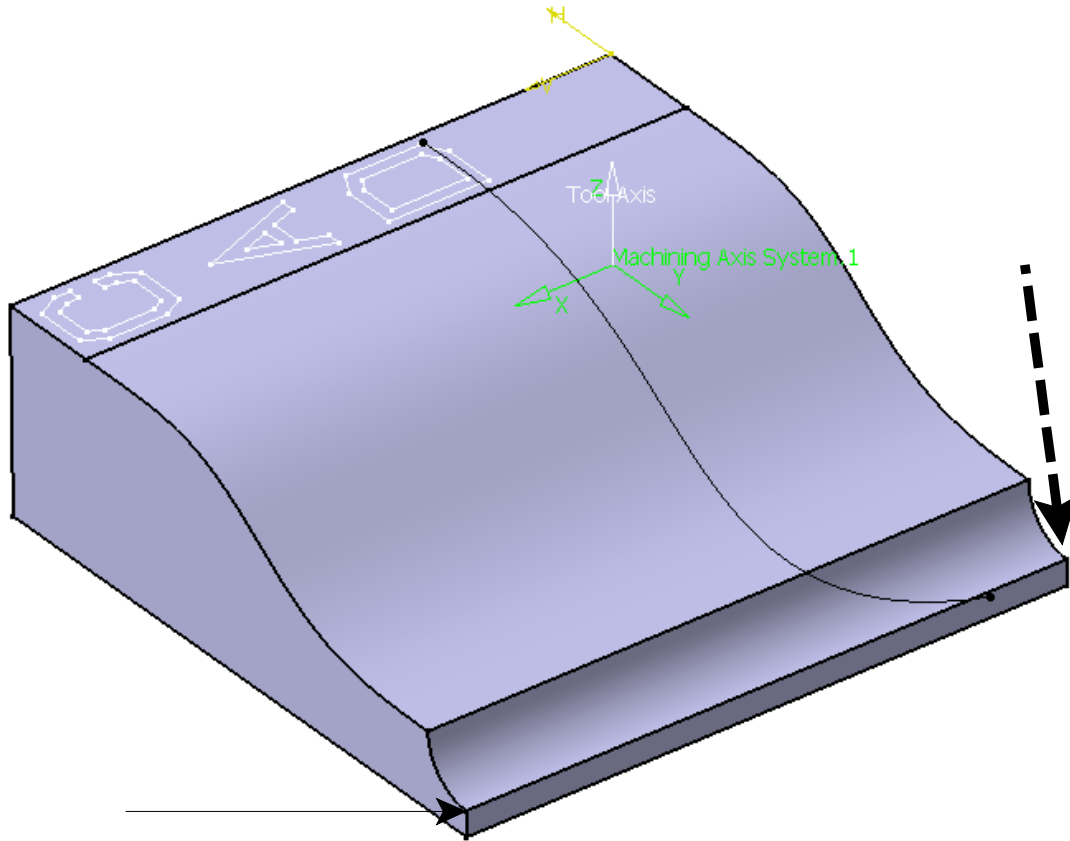
Remember, you have to select where you want the operation to appear.

The sensitive area for a point to point operation is very simple. The heavy red line below the green line is the positions line. This is what you select to define and add points to the point to point operation.



Select the positions line. The window disappears and waits for you to select points.

Select the two vertex points of the front slot in a climb milling direction. The vertex points are indicated below. You will want to select where the heavy, dashed arrow is first, then the other vertex point. The order they are selected is the order in which they get machined.

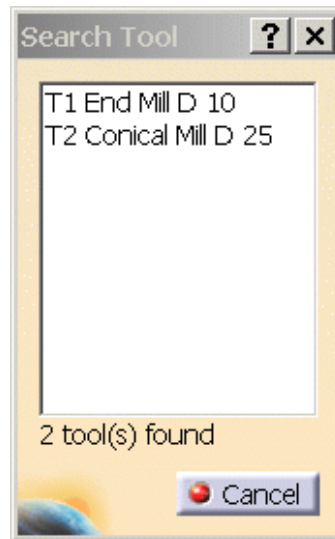


When done, double select in space. This will finish the point selection.

If you check the machining operation parameters tab, you will note there is nothing there but a cutter compensation. As was said earlier, the point to point operation is very simplistic, and does not have a lot of fancy options. The tool does need to be changed though.

Select the tool tab. The tool used for the first curve following operation will work here, but instead of re-creating a new tool, you are going to use the original tool.

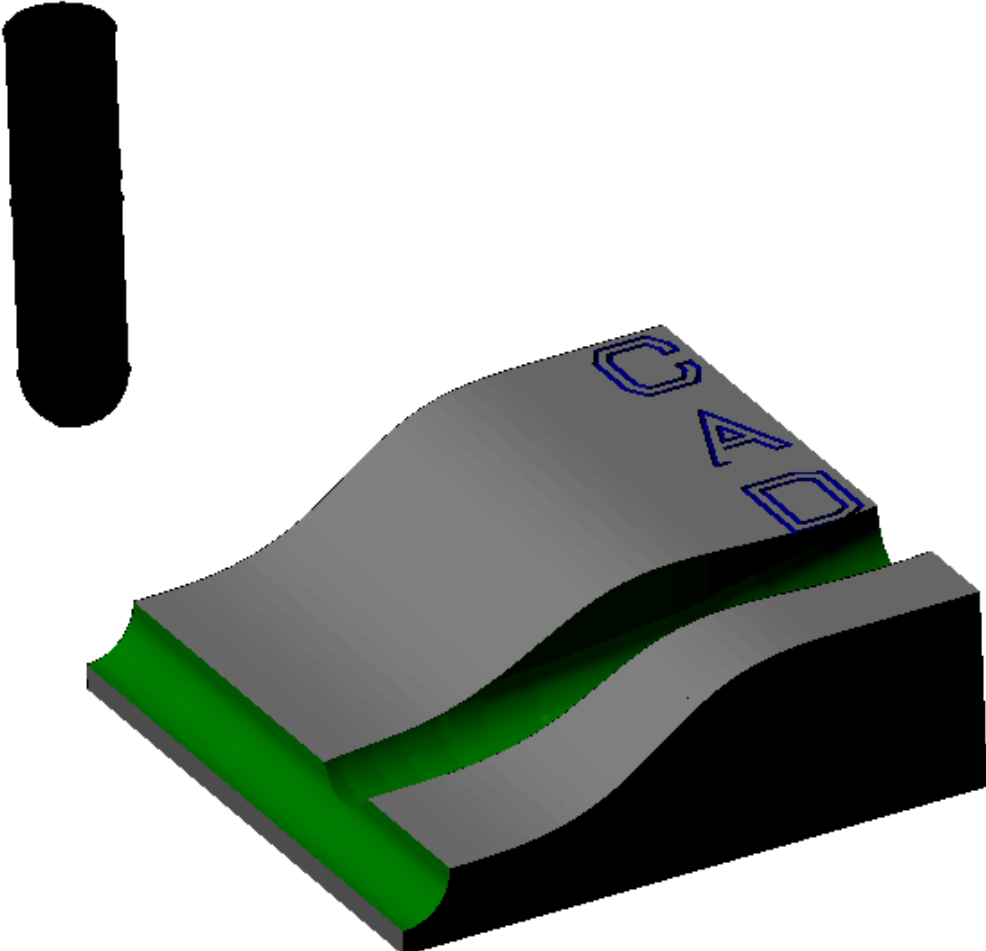
Select the three dots just to the right of the name field on the tool.  This is going to bring up the *Search Tool* window.



Any tools that you create in a V5 machining process will be part of the machining process. This dialog shows you all the tools that you have created and allows you to select one to re-use.

Select the *T1 End Mill D10*. The tool and its parameters will be shown in the tool sensitive area. Notice the line of icons still has the canonical mill depressed. This is not a problem. Since you used a tool already created the selection at the top has no impact.

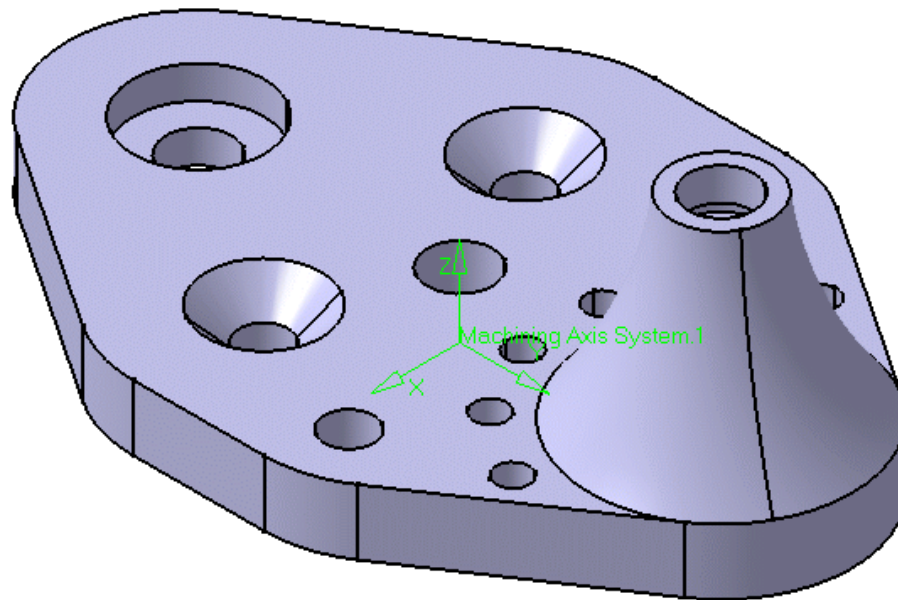
Select *OK* when done. Do not forget to check the macros, feedrates, spindle speed and most importantly the replay.



Axial Machining

Axial machining involves all circular drilling type motions. In this exercise, the machining that you utilized in the prior section will not be necessary. The part just needs to be drilled, using several different drilling tools. The order in which holes are drilled is not as important as the order that a part gets machined. Usually drilling operations get performed last.

Open Axial Machining from the *Axial Motions* folder. This part already has the *Part Operation* defined for you. Do not forget to save the document into your directory.



When drilling a hole, especially if the hole is somewhat large ($\frac{1}{2}'' +$), the hole should be spot drilled or center drilled first. Spot drilling a hole does one of two things. First, the hole has a small drill location in the center of the hole. This keeps the larger drill bit from “walking” or working its way off-center. The second reason to spot drill a hole before starting in on a larger cutter is to keep the tip of the larger drill bit from wearing down quickly. Starting a new large hole with the large drill bit can cut away at the tip, so by creating a small hole for the tip to set into, the tip is saved from having to initially start the hole.

All drilling icons are kept under one icon.

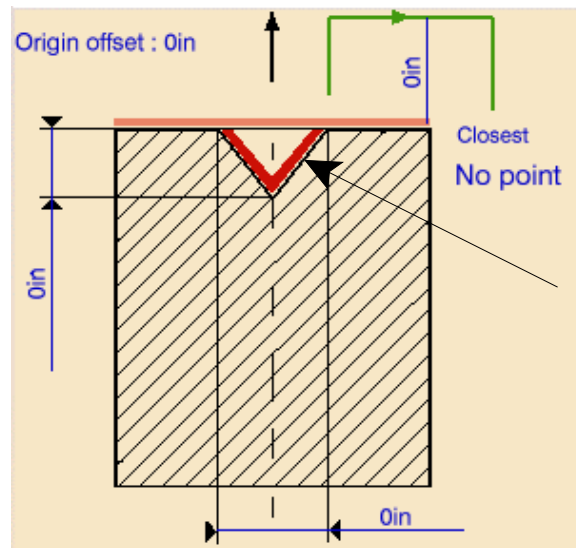


Spot Drilling

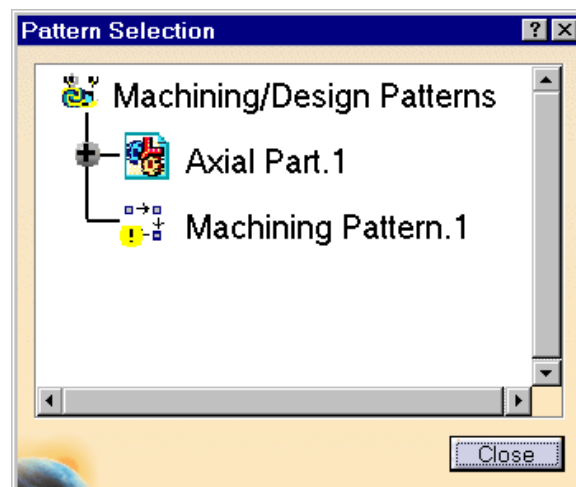
Select the spot drill icon and then select *Manufacturing Program.1*.  You have to select *Manufacturing Program.1* before CATIA knows where to put the spot drilling motion. The *Spot Drilling* window will display.

Notice from the sensitive area that there is only one bit of information that is mandatory, the hole location.

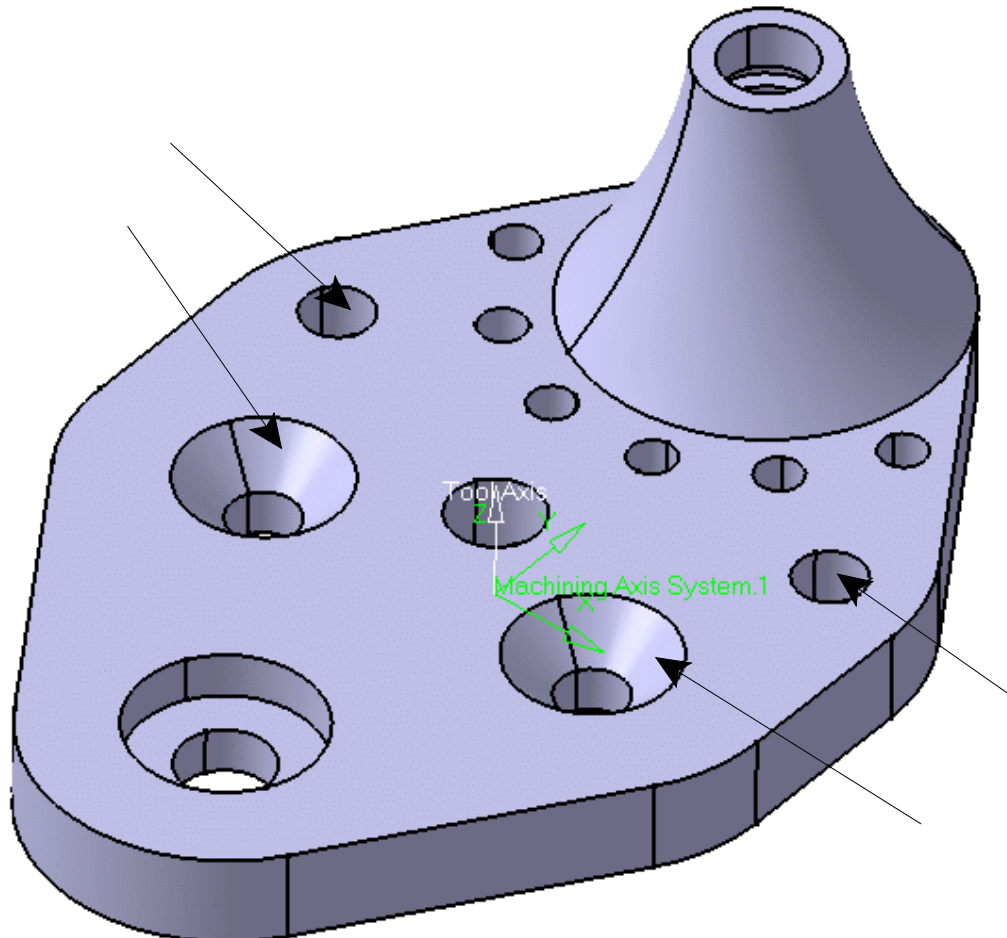
Select the hole location sensitive area. This will allow you to define what holes are going to be spot drilled.



This will display the *Pattern Selection* window. The pattern selection allows you to re-use machining patterns or re-use design patterns that you created when drawing the part. The pattern that you are creating is going to contain the four larger holes on the main body of the part.



Select the four holes shown below. When selecting the holes you will not get a pre-selection showing you that you are going to be selecting the hole. You also do not want to try to select the edges of the holes, but instead, you want to select the inside faces of the holes. The order that you select the holes is irrelevant but will determine the order of the holes.



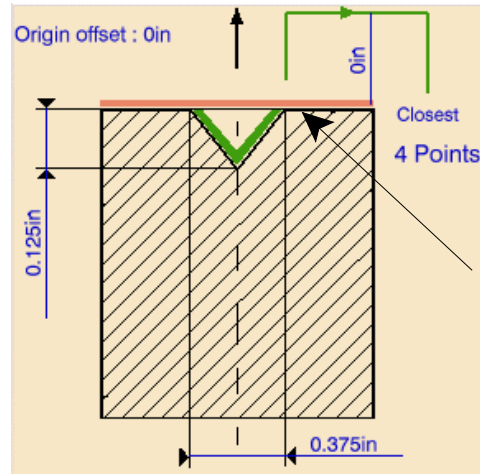
Double select in space. You have to double select on nothing to tell CATIA that you are done selecting points.

Notice the diameter of the hole is automatically inserted into the sensitive area. This is common for drilling motions. When you select a hole of any type, the size of the hole is automatically determined as well as the depth and is entered into the appropriate fields. Notice the depth of the spot drill is still zero however. This is because CATIA is unsure how deep you want to spot drill the holes.

Change the depth of the hole to be 0.125in. This can be accomplished by double selecting the value and then keying in 0.125.

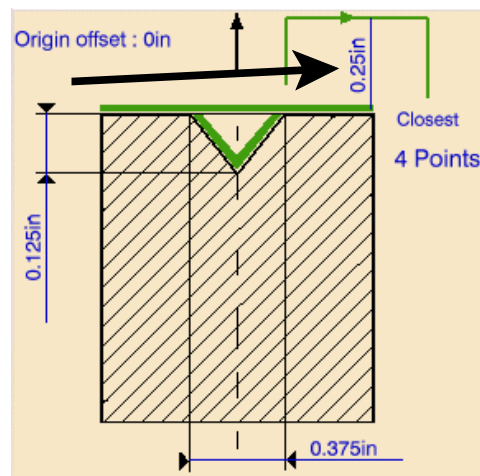
There are still a few more pieces of information that you will want to add to the spot drill sensitive area. Next, you will want to define the top of the holes.

Select the top of the spot drill sensitive area. This will be the heavy pink line across the top. This will define the top of the holes. This is an optional selection because CATIA will assume that the top of the hole is the top of the material. Defining the top in this manner is especially important when the top of the hole is not the top of the material, as in the case where you are drilling holes before facing off a part.

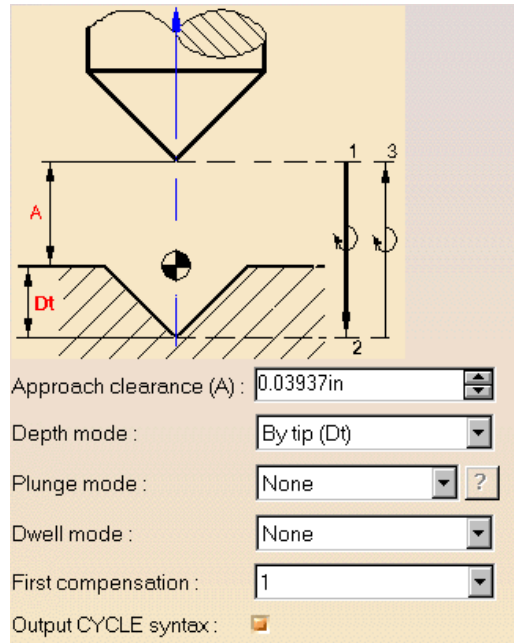


Select the top of the main body of the part. This will define the top of the holes. The last thing that you will want to define is the jump height. Notice the small green line that has the arrow along the top. This represents the default linking macro that will be applied to the spot drilling operations. You will want to make the height of the jump or link between the holes something larger than zero. Leaving the height at zero will drag the tool along the top of the part causing scratches or worse, broken tools.

Change the jump height to 0.25in. This will make the tool jump 1/4" between each hole.



Change to the machining operation parameters tab. This will be the first tab in the series of tabs across the top. There are only a few parameters that can be set. Take a closer look at them.



Approach clearance Defines the height above the point that the drill starts to spin and begin drilling. Usually you will want this to be a non-zero height but it does not need to be too high.

Depth mode Controls how CATIA measures the depth of the hole. There are two options: *By Tip* and *By Diameter*. *By Tip* measures the depth of the hole. *By Diameter* drills the hole until the hole is a particular diameter.

Plunge Mode Allows you to define an axial plunge determined by the depth or by the diameter of the hole. Most times, it is advisable to leave this option turned off.

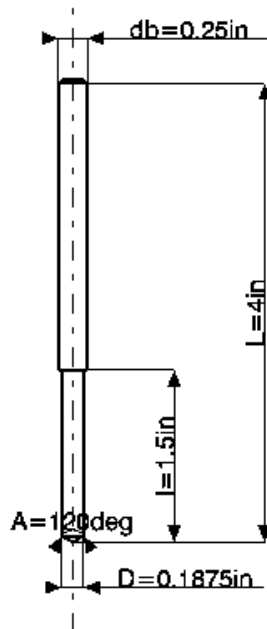
Dwell Mode Controls the dwelling of the tool. When turned off the tool drills the hole and then immediately retracts. There are two options that allow you to control the dwell: *By revolutions* and *By time units*. *By revolutions* allows you to dwell the tool a particular number of spindle revolutions while *By time units* allows you to dwell the tool for a certain number of seconds.

First compensation Controls the cutter compensation which has to be pre-set with the tool in the tool change operation.

Output CYCLE syntax Determines if a specific CYCLE/DRILL syntax will be used or if simple goto statements will be used to define the drill motion. If you are unsure of the machine that the program will be ran on you would be better off with goto statements.

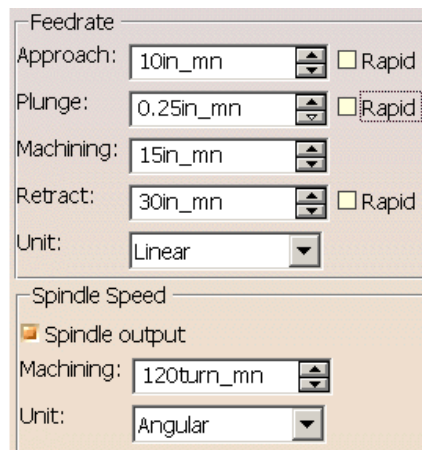
Set the **Approach clearance to 0.0625in**, the **Dwell mode to By revolutions** and the **Revolutions to 10**. This will make the tool start drilling 1/16" above the part and the cutter will dwell at the bottom of the cut for 10 revolutions.

Change to the tool tab and set the following parameters for a spot drill. This will make a small hole in each of the holes that you defined to spot drill.



db=0.25in, L=4in, l=1.5in, D=0.1875in and A=120deg


Change to the speeds and feeds tab and change the following feedrates and spindle speed. One last thing to do, define macros.



Change to the macros tab and add *Approach* and *Retract* macros that go to the safety plane. Remember, to do this, change to the tab, turn on the *Approach* and *Retract* check boxes and then select the add motion to a plane icon for both.

Select OK. This will finalize the spot drilling motion. Feel free to replay the operation if desired.

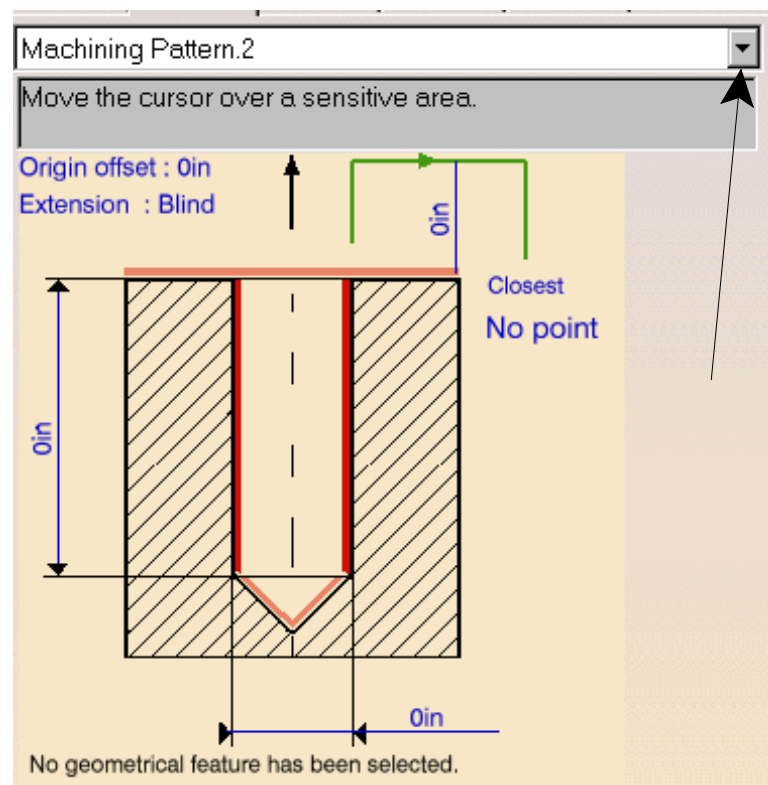
Drilling

Select the drilling icon and then select the *Spot Drilling.1* operation in the specifications tree.  Drilling is your basic drill operation that will be used to drill the same holes that were spot drilled.

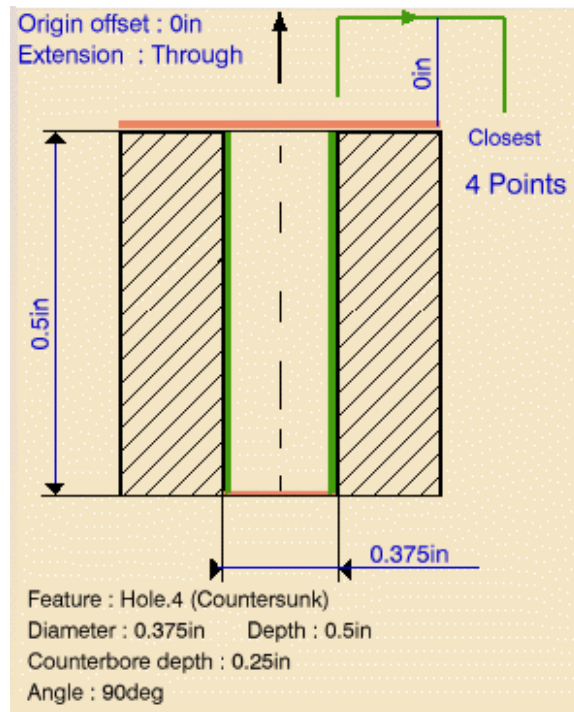
The sensitive area for the drilling motion will look very similar to that for the spot drilling operation. The one main difference is the *Extension* option. There are two settings for this option. *Blind* creates a hole that stops before going all the way through. *Through* drills the hole all the way through the selected part. By default, the extension is set to *Blind* but by selecting a hole that is all the way through a part the extension will change automatically.

Usually, you would select the side walls of the hole to define the machining pattern. However, since you already defined the pattern for the spot drilling motions you do not need to re-define the pattern.

Select the machining pattern drop down and select *Machining Pattern.1*. As soon as you select the *Machining Pattern.1* the menu will snap back to *Machining Pattern.2*. You do not have to worry, it is just duplicating *Machining Pattern.1* to *Machining Pattern.2*.



Notice the sensitive window changes and fills out with all of the necessary information. The diameter as well as the depth and extension change to match the holes exactly.



There is, however, still some information that needs to be defined before moving on.

Define the top and bottom of the part. This can be done by selecting the top heavy pink line and then selecting the top of the part, and then selecting the bottom pink line and selecting the bottom of the part.

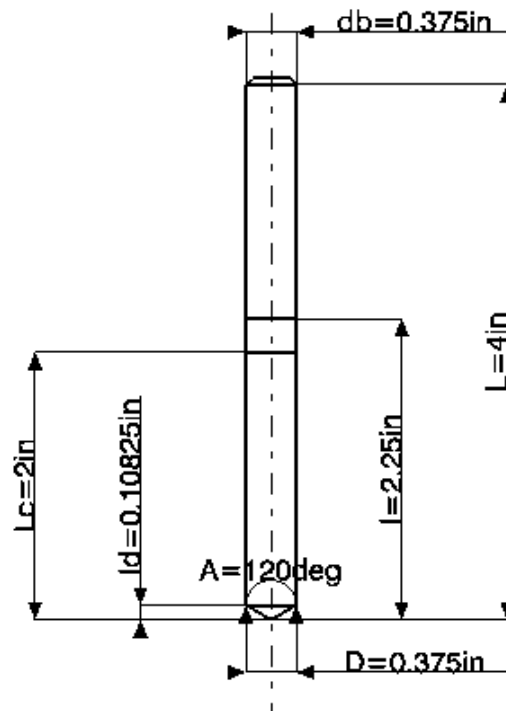
Change the jump height to 0.25in. Again, you want the tool to pick up off of the part between holes. This sets all of the geometric options. You will now set the machining operation parameters.

Change to the machining operation parameters tab. You should note that the approach clearance is still the same as what was set with the spot drill operation. You do have a new option, *Breakthrough*. *Breakthrough* is the distance that the tool should go through the part. Since you are drilling all the way through the hole it is advisable to have a breakthrough value.

Change the *Breakthrough* to 0.125in. This will make the tool drop 1/8" below the bottom of the part.

Next, the tool needs to be defined.

Change to the tool tab and set the following parameters. This will define the right tool for drilling the holes.



$db=0.375\text{in}$, $L=4\text{in}$, $l=2.25\text{in}$, $D=0.375\text{in}$, $A=120\text{deg}$, $ld=0.10825\text{in}$ and $Lc=2\text{in}$

The speeds and feeds for the tool will be the same as the spot drilling operation so there will be no changes necessary.

Add Approach and Retract macros. Just add motions to a plane for the macros. This will be sufficient.

Select OK to finish the drilling operation. Again, do not forget to replay your operation to make sure it performed as you expected.

Drilling Pre-Defined Patterns

A lot of times a series of holes will be pre-defined in a design pattern or a pattern that was created when designing the part. A spot drill motion will be added to the program where the first spot drilling operation is located in the specifications tree.

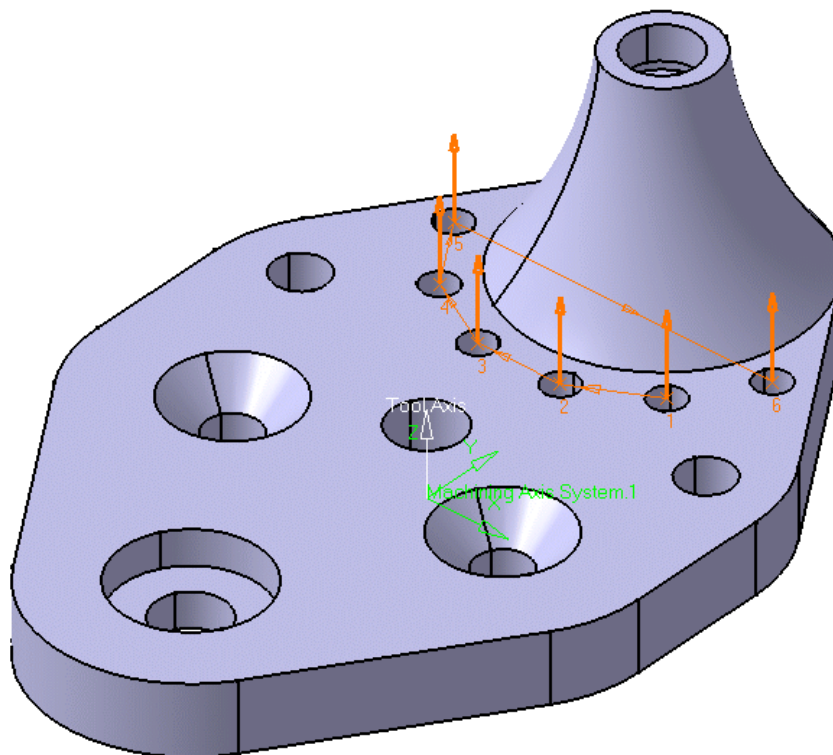
Select the spot drill icon and then select *Spot Drilling.1* from the specifications tree.

Notice the second spot drilling operation appears below the first. This is how you insert operations in other places in the tree rather than just at the bottom each time.

Select the hole definition area on the sensitive area for the spot drill. This is the red V at the bottom of the hole in the sensitive area. The *Pattern Selection* window displays again.

Expand the *Axial Part.1* in the *Pattern Selection* window. This can be accomplished by selecting the plus (+) on the left side of the tree. Notice the *CircPattern.1*. This is the pattern that was created for the holes around the center of the boss.

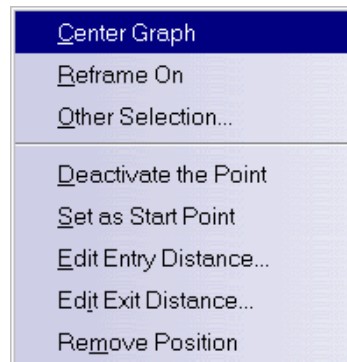
Double select *CircPattern.1*. Notice the hole diameter as well as the number of points change to reflect the hole pattern. Also take note to the order of the hole pattern. Sometimes the pattern may not be as desired. What you want is for the pattern to start on one side, then drill each hole until it is on the other side. The pattern that you get by default may be similar to the one shown below and is not what you want, if the pattern is in the proper order then you can skip the next couple of steps. Make sure you review the steps in order to know how to fix a problem if it occurs.



In the sensitive area there is a word that the jump points to: *Closest*. This denotes the order that the holes get drilled, however, if a pre-defined pattern is used it may not drill the holes in the correct order.

To fix the problem, you just need to define where the start point of the pattern is and then CATIA will re-number the holes according to what hole is closest.

Using the third mouse button select on hole number six from the pattern. This will bring up the contextual menu for the hole pattern. *Note: You will want to select towards the tip of the arrow, and not at the base.* This may not be necessary for you to do if your pattern is correct already.



Deactivate the Point

Allows you to keep a point in a machining pattern but just not drill the hole for this operation. This is useful if you want to have just a few machining patterns. You can drill only a few holes with the specific drills, all the while using the same pattern.

Set as Start Point

Defines where the pattern begins.

Edit Entry and Exit Distance These options allow you to manually define the distance that the drill enters and exits from the hole.

Remove Position

Allows you to completely remove a hole location from the machining pattern.

Select *Set as Start Point*. The pattern will update, reordering with this hole as hole one.

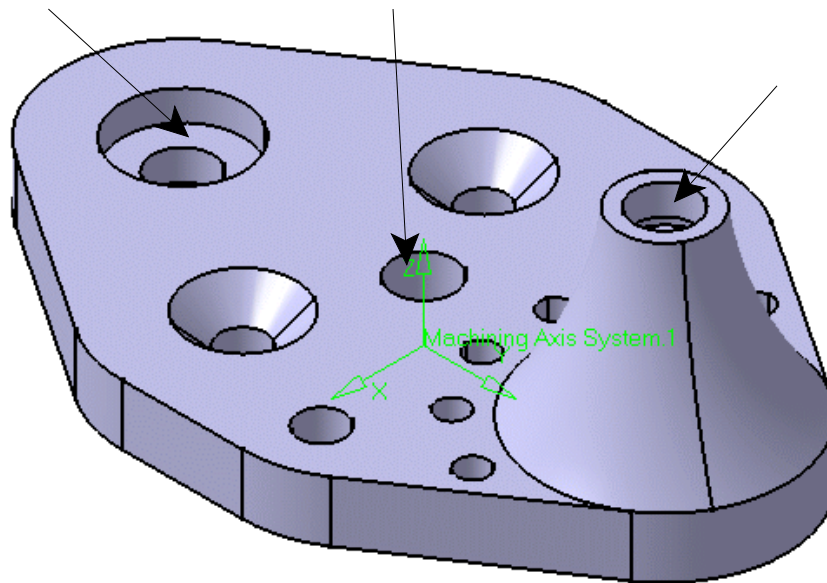
Define the top of the hole. Remember, this can be done by selecting the heavy pink line and then selecting the top of the part.

Change the depth of the hole to 0.0625in and the jump distance to be 0.25in. This will finish setting the geometry options.

Since the spot drilling operation is inserted just after the original spot drill operation, all the machining operation parameters, macros, and speeds and feeds will remain the same.

Select *OK* to finish the spot drill operation. Now it will be up to you to create a new spot drill operation.

Create a spot drill operation after the second spot drilling operation. Define the three holes as shown below to drill. There will be a few different options to set since the holes are not at the same level.



Do NOT define a top surface for the holes. Defining a top surface will make the tool drill at too high or too low of a level for all the points.

Make sure the jump height is zero. This is important. Since the holes are at such a different height you will need a linking macro. The linking macro will not work if there is a jump height set.

Make sure the depth is defined to be 0.125in. Since you are working with larger holes it is a good idea to go ahead and drill them a bit deeper.

Under the macros tab, turn the *Linking macro* on. A linking macro has two separate moves to it, a linking approach and a linking retract.

Add motions to a plane for the *Linking Approach* and the *Linking Retract*. Again, defining a plane is unnecessary since a safety plane was defined in the part operation.


Select *OK* to the spot drill operation. This has finished all of the spot drilling operations.

Replay and save your program.

This covered the basics of axial motions. Next, you will go through a few more exercises on more specific drilling motions. Although they will be drilling motions of a different type, they still work in the same manner as the basic drilling motions. The only thing that differs between the different drilling motions is the information that needs to be defined.

Drilling Deep Hole

The drilling deep hole option will be used for the hole on top of the boss. If you remember from the introduction, drilling deep hole will drill part ways and then completely retract the tool to allow all the chips to clear from the drill bit.

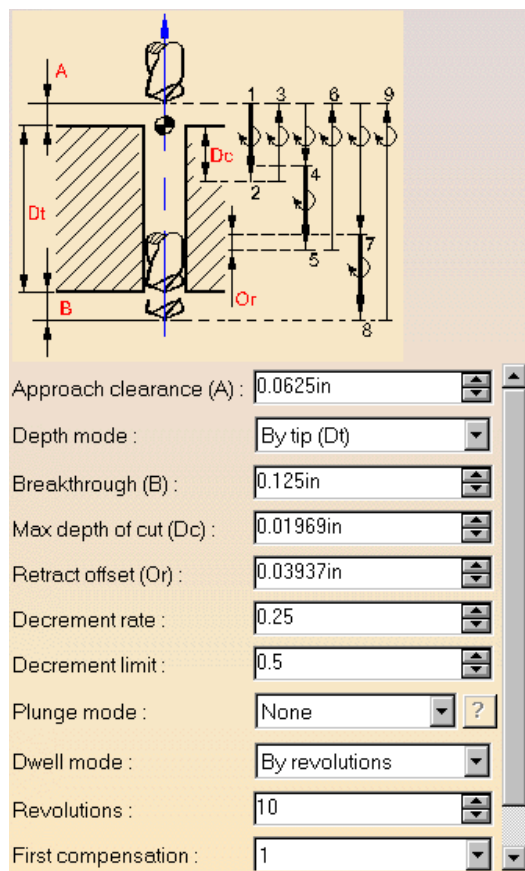
Select the drilling deep hole icon and then select *Drilling.1*.  If you remember from the introduction to this book, a drilling deep hole operation will make the tool drill a distance and then completely remove the tool and then drill a bit deeper. This allows all chips formed and bound around the tool to be removed before drilling a bit deeper.

Select the position definition from the sensitive area. This will be the side of the hole.

Select the inside of the hole at the top of the boss. This hole also has a T-Slot in it. Do not worry about this now. You will investigate t-slotting later. The depth and diameter of the hole is defined automatically.

Define the top and bottom of the hole. Make sure you select the top of the hole as the top of the boss and not the top of the rest of the part. A jump height is not necessary because you are only drilling one hole.

Change to the machining operation parameters tab. Notice there are a lot of new options. Take a look at the new options.



The first three options you already know about.

- Max depth of cut* Defines the maximum depth that the drill will cut until it begins a retract motion to pull the tool back out of the hole.
- Retract offset* Defines how high above the previous drill peck that the next drill cycle begins.
- Decrement rate* This is the rate at which the maximum depth of cut will be reduced as the drill goes deeper and deeper into the part.
- Decrement limit* This is the maximum decrement percentage that will be applied.

The rest of the options are the same as those found in the spot drilling motions.

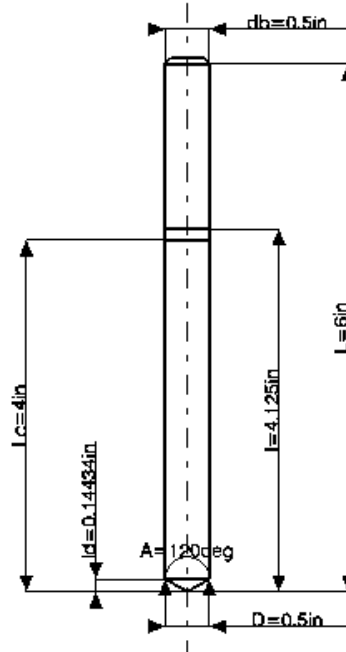
Set the following parameters. This will define the deep hole drilling operation. Next, a tool needs to be defined.

Approach clearance (A) :	0.0625in
Depth mode :	By tip (Dt)
Breakthrough (B) :	0.125in
Max depth of cut (Dc) :	0.25in
Retract offset (Or) :	0.0625in
Decrement rate :	0.1
Decrement limit :	0.75
Plunge mode :	None ?
Dwell mode :	By revolutions
Revolutions :	5
First compensation :	1

Change to the tool tab. Try changing a parameter. None of the parameters are modifiable. This is because the deep hole is defined after the drilling operation which already is using a drill bit. However, the 3/8" drill bit is not the right size. You need a 1/2" drill to drill the hole with. This means you will have to force a tool change.

Select the multi diameter drill icon.  There is no intention in using a multi diameter drill but this will force the tool change operation.

Select the drill icon again and define the following drill bit. This will change the tool change operation back to a drill tool change.



$db=0.5in$, $L=6in$, $l=4.125in$, $D=0.5in$, $Lc=4in$, $A=120deg$ and $ld=0.14434in$

The last thing that needs to be added is the macros.

Add approach and retract macros to the deep hole operation. Select *OK* when done.
This will be the last thing that needs to be defined for the operation.

Do not forget to replay the operation when you are done.

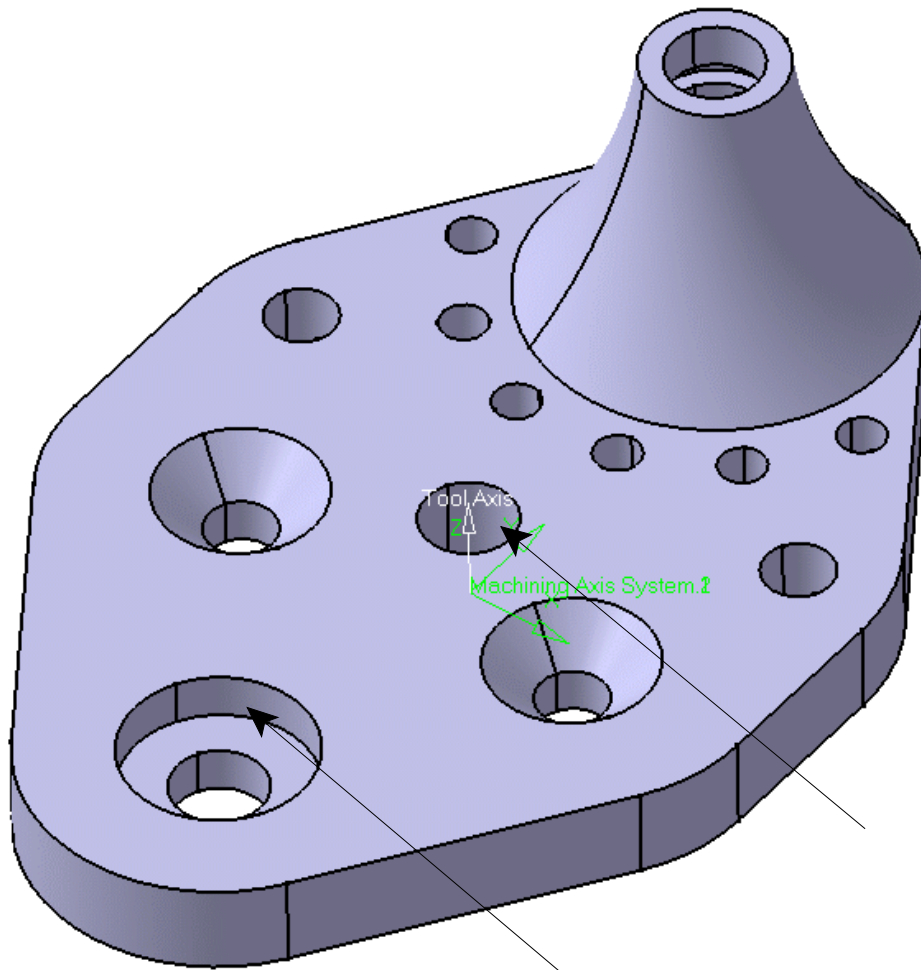
Drilling Break Chips

Drilling break chips will be used to drill the other two half inch holes down the center of the part. These holes are not actually deep enough to need a drilling break chips operation but this will work as an example to understand how the drilling break chips operation works.

Select the drilling break chips icon and then select *Drilling Deep Hole.1*. 

Remember, the drilling break chips operation drills the tool a distance into the part, then retracts a given distance and then continues to drill the hole. This is similar to a deep hole drilling operation except the tool is not completely retracted from the material.

Define the two center holes to drill. This can be done by selecting the side wall from the sensitive area, then selecting the two holes. Do not forget that you have to double select in space to finish the hole definition.



Define the top, bottom and the jump height to be 0.25in. This will define the geometry that is necessary for the operation.

The machining operation parameters tab has the same information to be defined that you have seen and set before. Notice the similarities between the drilling deep hole and the drilling break chips.


Add macros and select *OK* when done. Just an approach and retract macro is going to be necessary. The linking macro is defined by the jump height.

You will now create a drilling dwell delay operation on your own. You should use your experience from the previous exercises to perform this operation.

Create a drilling dwell delay to drill the six holes around the boss. For this you will have to force a tool change to get the tool to be the right size. That can be done by selecting a different tool and then selecting the drill icon again. The pattern held under the axial part branch can be used for the pattern. Do not forget to watch the order of the holes. You do not want the tool crashing through the boss. The machining parameters will already be defined and will not need changing.

Countersinking

Since the two holes on either side of the part have already been drilled, they just need a countersinking operation performed on them. If the holes were not drilled just yet, you would need to use a boring and chamfering operation to perform both operations at the same time.

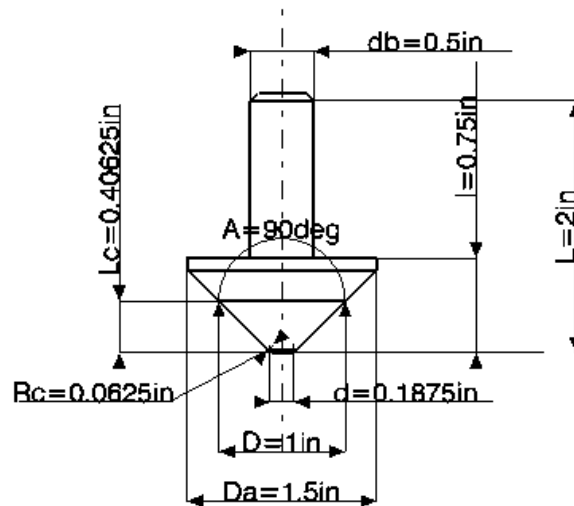
Select the countersinking icon.  You will want to insert the countersinking operation after *Drilling Dwell Delay.1*.

Define the two holes to countersink as well as the top of the part. Select the side wall of the hole and then select the holes from the geometry.

Set the jump height to 0.25in. Again, you want the tool to jump above the part as it goes hole to hole. Just dragging the tool along the part is no good.

Change to the machining operation parameters tab. Everything here you have seen before.

Change to the tool tab and set the following parameters for the countersink tool. You will want to make sure you change the to the chamfer tool.




$db=0.5in$, $L=2in$, $l=0.75in$, $d=.1875in$, $D=1in$, $Da=1.5in$, $Rc=.0625in$, $A=90deg$ and $Lc=0.40625in$

Change to the feeds and speeds tab and change the plunge feedrate to 3in_mn and the finishing feedrate to .5in_mn. Since the hole is already drilled, you can speed up the plunge feedrate.

Make sure approach and retract macros are defined. Select OK when done. This will complete the countersinking operation. Next, you will counter bore the hole opposite of the boss.

Counterboring

Select the counterboring icon.  You will want to insert the counterbore operation after *Counter Sinking.1*. This will be the next to last operation that you will be creating. To counterbore the hole you will be using an end mill rather than a standard drill bit. Drills have a taper angle at the tip of them and you do not want a taper so a flat end mill will do the trick.

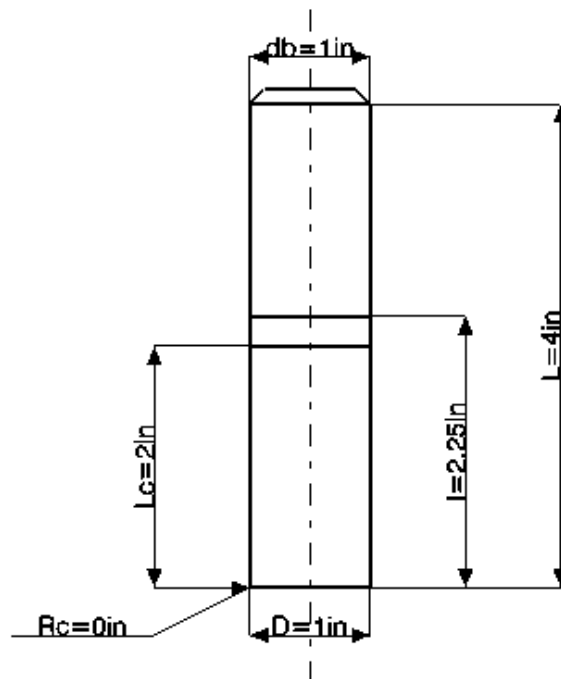
Define the point to drill. The depth of the hole as well as the size of the counterbore is defined.

Define the top and bottom of the hole. The only new thing here is the bottom definition. Instead of defining the underneath side of the part you will want to define the bottom of the counterbore.

The machine operation parameters are all set correctly. Again, the values from the previous axial machining operations carry over.

Change to the tool tab and change the tool type to an end mill. Remember, drills have a taper at the end of the tool so you will have to use a flat end mill to counterbore the hole.

Change the tool parameters to the following:




$db=1in$, $L=4in$, $l=2.25in$, $D=1in$, $Rc=0in$ and $Lc=2in$

Add approach and retract macros to the operation and select OK when done. This will bore the hole out. There is just one operation left to do, the t-slotting operation.

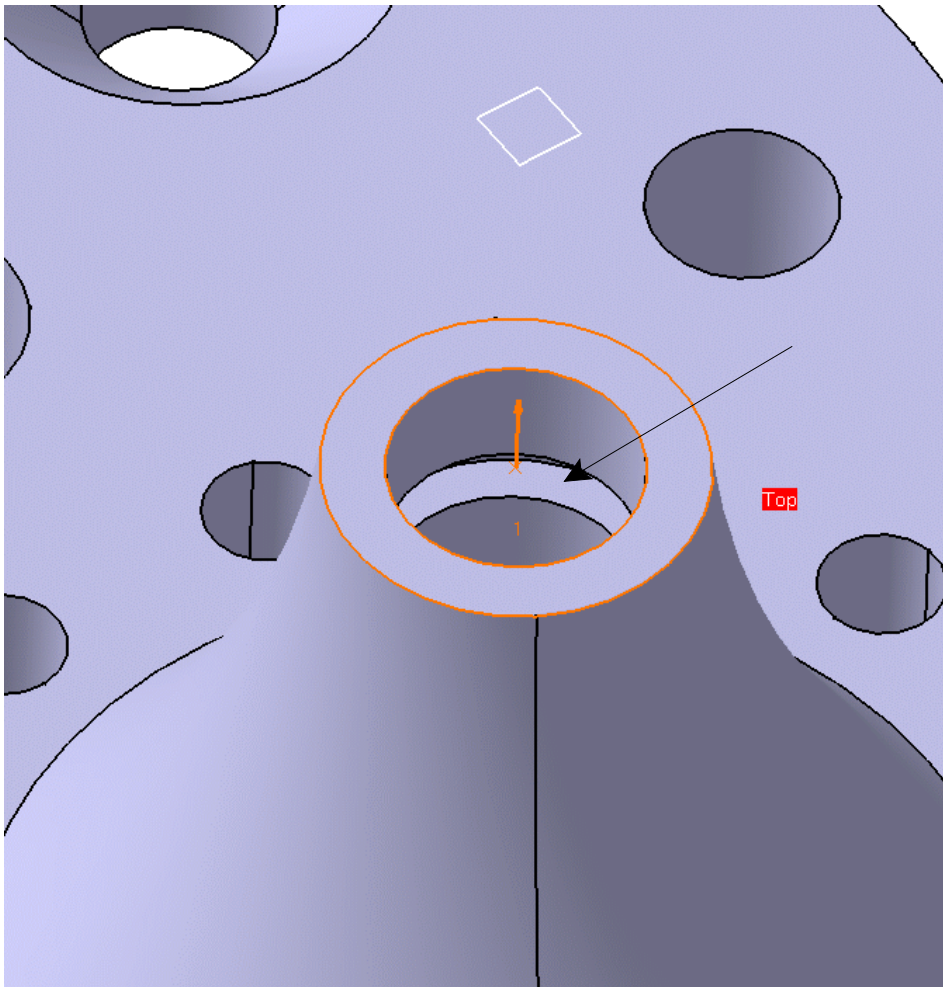
T-Slotting

This will be the last speciality axial motion investigated.

Select the T-slotting icon.  You will want to insert the T-slotting operation after *Counter Boring.1*. This will work very similar to all the rest of the axial motion operations that you have created thus far.

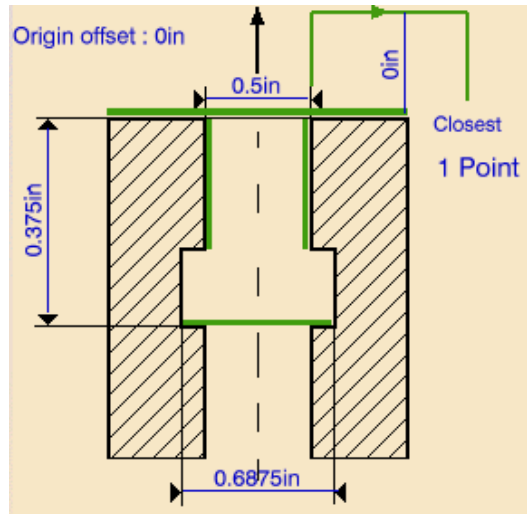
Define the hole and the top of the hole. This will be the same as everything else. Next, you will want to set the bottom of the t-slot.

Define the bottom of the t-slot in the hole. This can be done by selecting the bottom of the t-slot from the sensitive area and then defining the bottom of the t-slot as shown below.



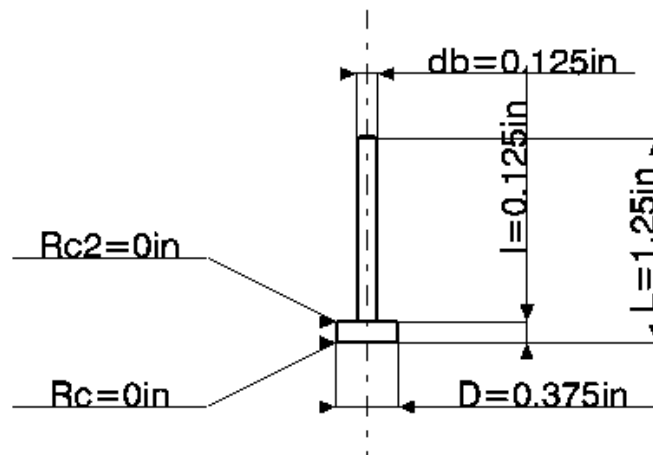
Notice however, that the distance to the bottom of the t-slot did not change. This and the diameter of the t-slot will have to be entered manually. Manually measuring the distance between the bottom of the t-slot and the top of the boss will yield $.375''$. The diameter of the t-slotted hole is found to be $.6875''$.

Be sure the following parameters are set.



Change the dwell mode under the machining parameters to none. Since you are working with a t-slotted there is no reason to dwell at the bottom of the hole.

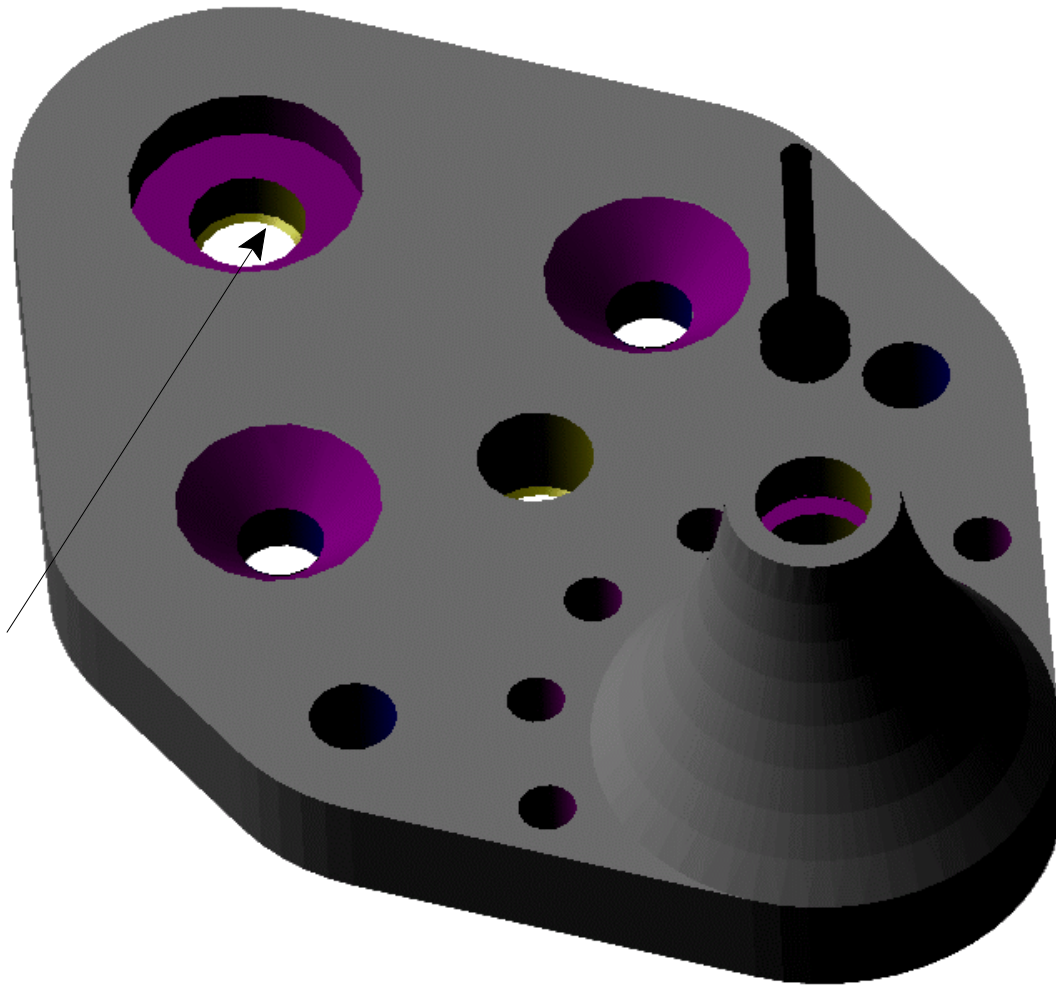
Define the following tool:



$db=0.125in$, $L=1.25in$, $l=0.125in$, $D=0.375in$, $Rc=0in$ and $Rc2=0in$

Add approach and retract macros. Select *OK* when done. This will complete the t-slotting motion.

Replay the manufacturing program in video mode to view all of the machining operations. Do you notice something wrong? Note the center 1/2" holes. There is a burr left at the bottom of the holes that you did not intend to leave. This means that the breakthrough value needs to be increased. It will not be changed for this exercise, but modifying the geometry and the machining operations will be investigated later.

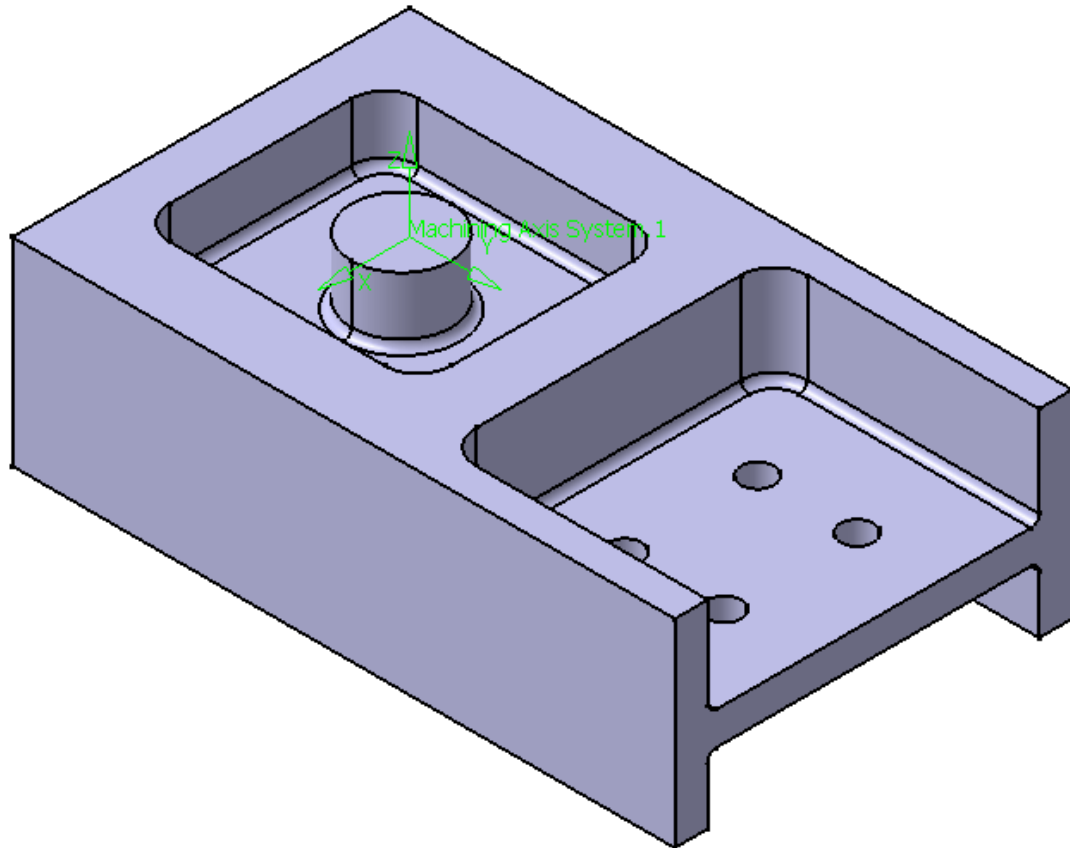


Advanced Machining

This section will investigate some of the advanced machining options available in prismatic machining. First, open pockets as well as pockets with islands will be investigated. Then multiple part operations, machining axes and manufacturing programs will be looked at.

Open the Advanced Machining process found in the *Advanced Machining* folder.

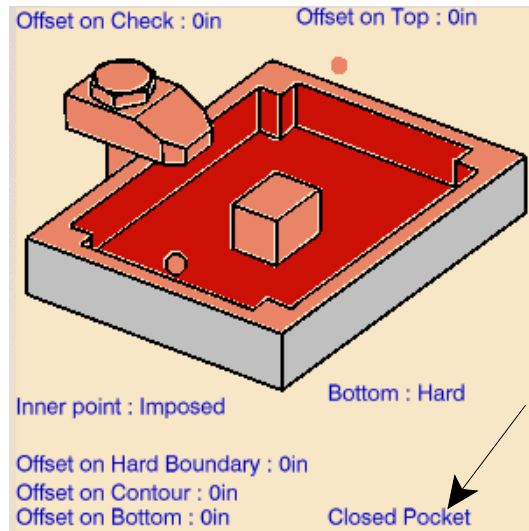
This machining process already has the part operation and the tool change added.



This part has four pockets that will be machined. Each pocket poses a slightly different challenge that will be investigated one at a time. Also, multiple part operations will be generated to handle the different part positions.

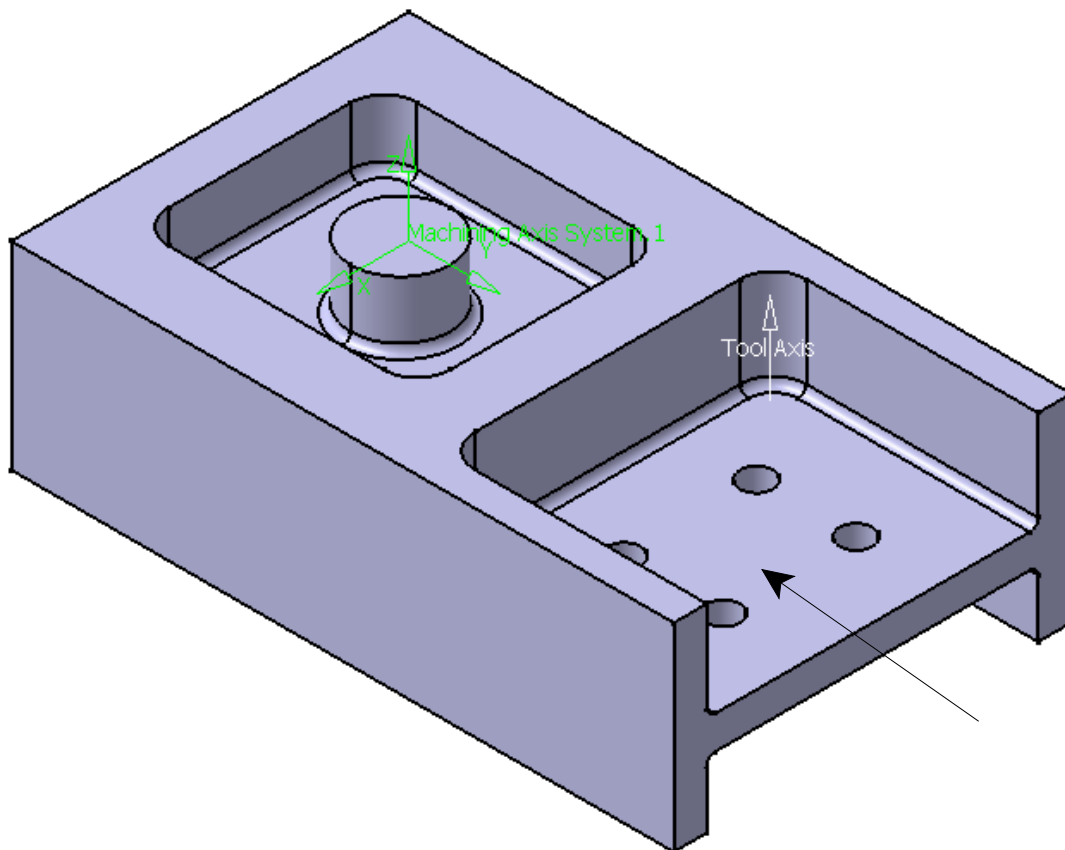
Select the pocket icon and then select *Tool Change.1*. The pockets will be milled one at a time starting with the open pocket on the front.

Change the pocket type to an *Open Pocket*. This can be done by selecting the Pocket Style option at the bottom of the sensitive area of the *Pocketing* window as shown below.



This will change the pocket to an open pocket. Next, the bottom of the pocket will be defined.

Define the bottom of the pocket. Remember, this can be done by selecting the bottom of the pocket from the sensitive area and then selecting the bottom of the open pocket on the top side of the part as shown below.



Notice the island in the middle of the sensitive area turns green. This is because the four holes defined on the bottom of the pocket were defined as islands. This is not what you want.

Using the third mouse button select on the island in the center of the pocket in the sensitive area. This will bring up the contextual menu for the island selections.

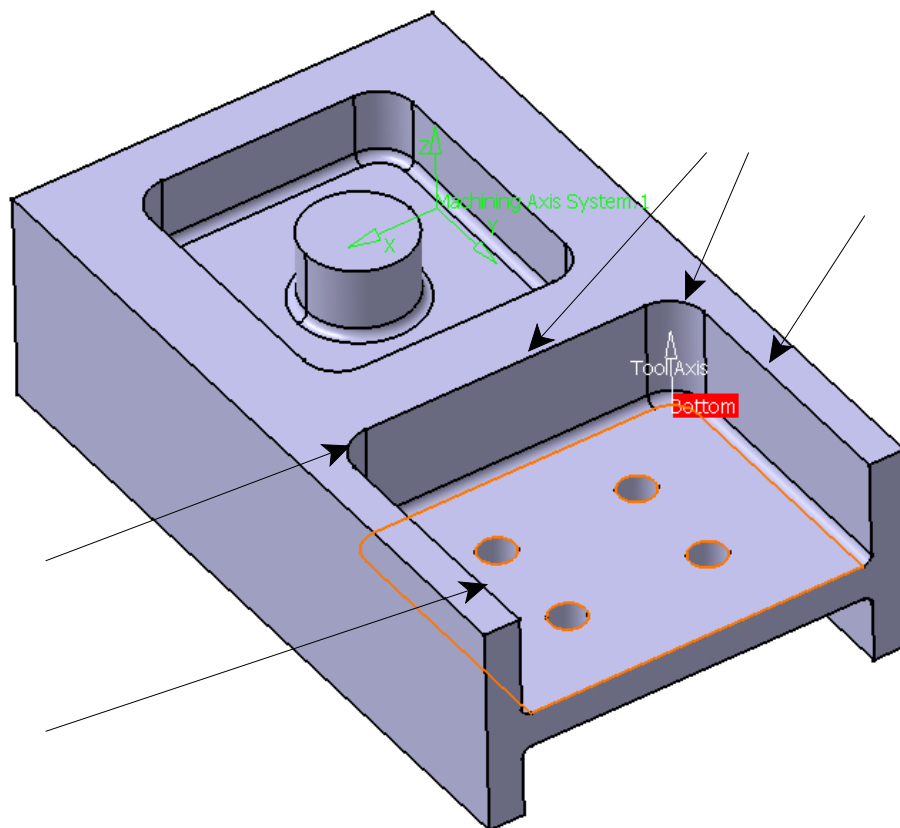
Select *Remove All Islands*. This will remove all islands defined by the bottom selection.

Also note the boundary of the pocket was also automatically selected. The boundaries of the face along the bottom of the pocket were used as the boundaries. You will want to remove the boundary and then re-define the boundary.


Using the third mouse button select on the boundary in the sensitive area and select *Remove*. The boundary will turn red thus displaying that it needs to be defined.

Now the boundary is ready to be defined. Since you are working with an open pocket, you will note that there is no option to select the boundary by a face boundary or belt of faces. This leaves edge selection as the last choice.

Select the boundary of the pocket from the sensitive area. Select all of the hard boundaries of the open pocket. The boundaries are indicated below.



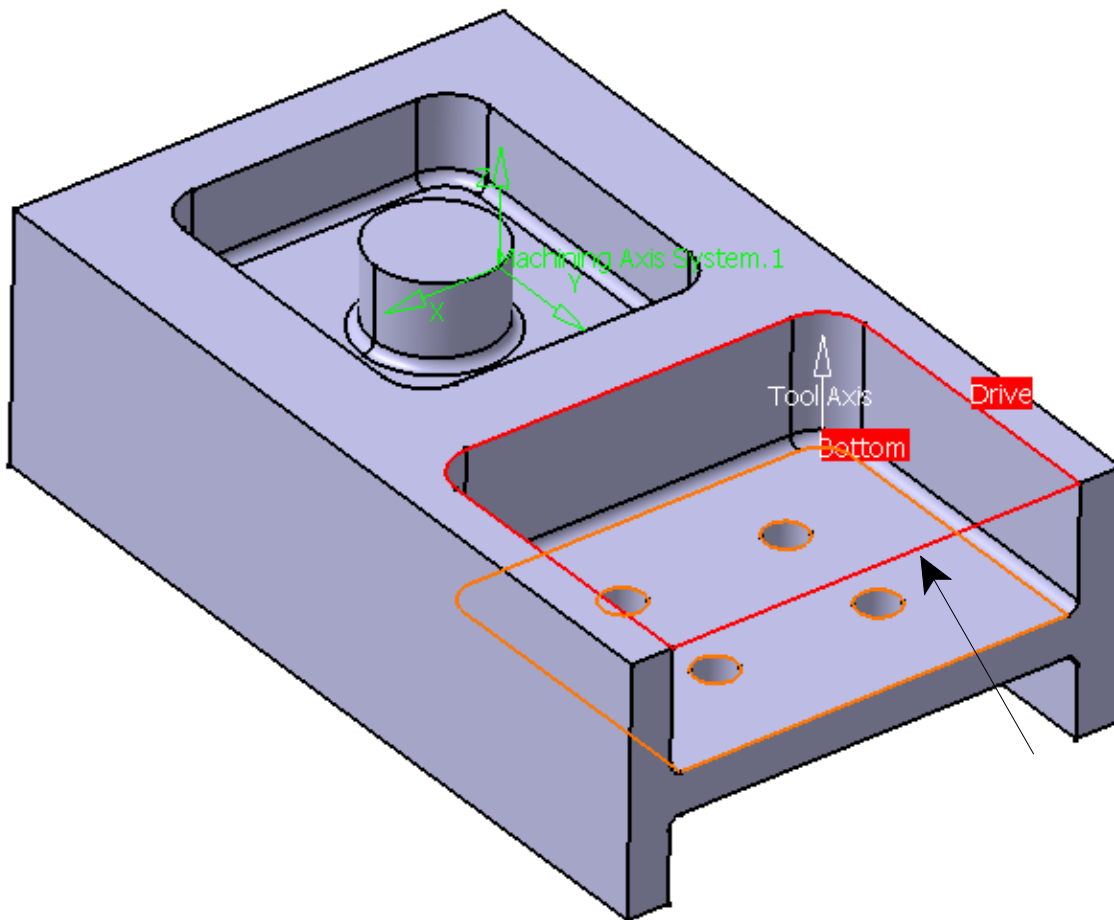
Before you select *OK* you have to make a closed boundary loop. This can be done by inserting a line into the boundary.

Select the close contour with a line icon from the *Edge Selection* toolbar.  This will insert a line between the two ends of the boundary.

Select *OK*. This will return you to the *Pocketing* window. You will now change one of the boundaries to be a soft boundary.

This brings up a new term, hard versus soft boundaries. Hard boundaries are defined as a solid line where soft boundaries are dashed. A hard boundary defines a wall or barrier that the tool cannot cross. Hard boundaries are used to define walls of pockets. Soft boundaries on the other hand are boundaries that are only there for pocket computation. A soft boundary defines a non-existent wall that the tool is free to pass beyond.

Select the boundary on the open side. The boundary needing to be selected is shown below. This will make the boundary a soft boundary.



Define the top of the part. This is done the same as all other pockets.

Change the following machining operation parameters:*Tool Path Style: Outward Helical*

Machining	Radial	Axial	Finishing	HSM
Direction of cut:	Climb			
Machining tolerance:	0.005in			
Fixture accuracy:	0.01in			
Type of contour:	Optimized			
Compensation :	P1			

Machining	Radial	Axial	Finishing	HSM
Mode:	Tool diameter ratio			
Distance between paths:	0.188in			
Percentage of tool diameter:	75			
Overhang:	100			
Center path overlap:	50			
Reduction percentage:	100			
Pocket Navigation				
<input type="checkbox"/> Always stay on bottom				
<input type="checkbox"/> Inward/outward mix				

Machining	Radial	Axial	Finishing	HSM
Mode:	Maximum depth of cut			
Maximum depth of cut:	0.5in			
Number of levels:	3			
Automatic draft angle:	0deg			

Machining	Radial	Axial	Finishing	HSM
Mode:	Complete			
Side finish thickness:	0.02in			
Side finish thickness on bottom:	0in			
Bottom finish thickness:	0.05in			
<input type="checkbox"/> Spring pass				
<input checked="" type="checkbox"/> Avoid scallops on bottom				

Add approach and retract macros and replay your operation. Select *OK* when done. This will finish the first open pocket. Make sure you replay the operation to make sure it performed accordingly.

Next, a closed pocket with an island will be created.

Create a pocket operation after *Pocketing.1*. Change the pocket type to a closed pocket. Remember, to change the pocket style just select the blue text where it says *Open Pocket* and it will change to *Closed Pocket*.

Using the third mouse button select on the boundary and select *By Belt of Faces*. Define the boundary around the back pocket, selecting *OK* when done. This will define the boundary of the pocket. Since you are creating a closed pocket you can define the boundary by a belt of faces instead of just the edges. Next, the island will be defined.

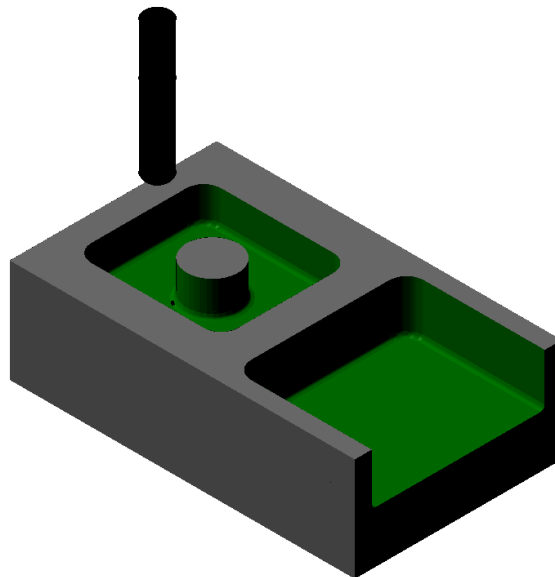
Instead of defining the bottom first you are defining the boundary, then the island and then the bottom. The reason is, if you define the bottom first, the island and boundary will be automatically selected. Since there is a fillet around the bottom of the boundary and the island the edge of the part will be off by the amount of the fillet radius. This is not what you want. By defining the boundary and island first you avoid this problem of automatic selection. It is suggested that you always define the boundary and islands before defining the bottom.

Using the third mouse button select on the island and select *By Boundary of Faces*. Select the top of the island and then *OK* when done. By choosing *Boundary of Faces*, the face boundary around the top of the island will be used as the boundary for the island. This will create an island in the pocket operation.

Define the bottom and top of the pocket. Since you defined the bottom last, the boundaries and islands are not automatically selected. This saves you the step of having to delete the boundaries and islands and then re-create them.


Make sure the machining parameters are the same as the previous pocket operation and that there are approach and return macros. Select *OK* when finished. This will complete the closed pocket with an island.

Do not forget to replay your operations to make sure everything looks good.



Multiple Part Operations

There are several reasons that you might need several part operations. One example will be demonstrated. When you want to change the position of the part on the machine, for example, turn the part over, you will want a separate part operation. Another reason you might want a second part operation would be to change machines or change fixture locations.

Select the part operation icon.  This will tell CATIA that you are going to be inserting a new part operation. CATIA wants to know where you are going to put the part operation.

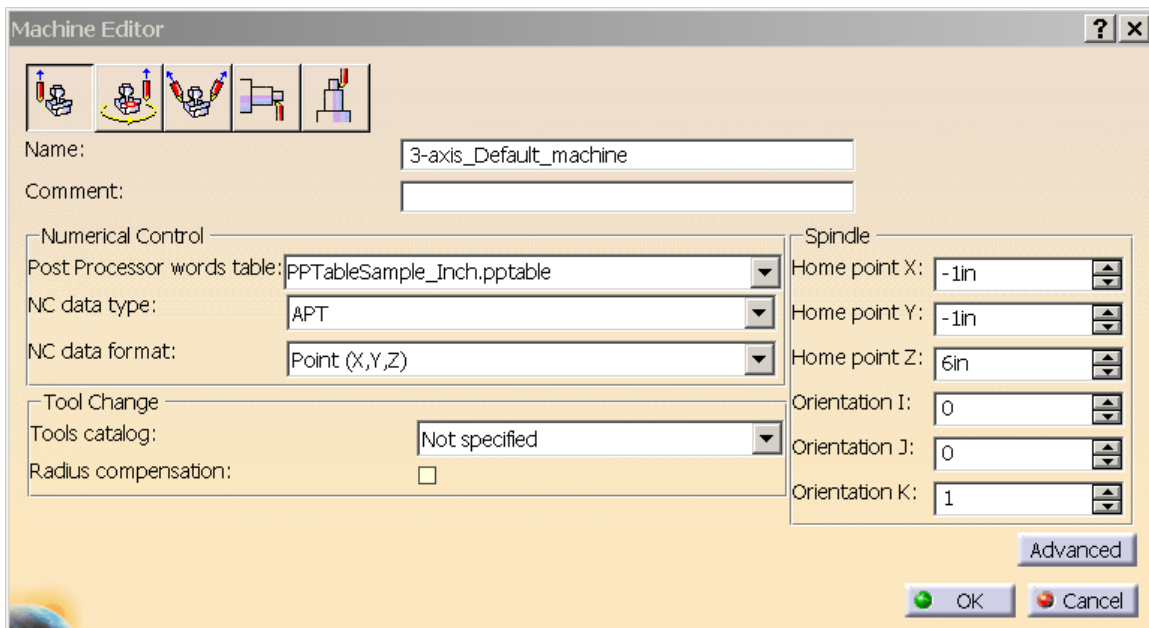
Select Part Operation.1. A new part operation, *Part Operation.2* is inserted into the program after *Part Operation.1*. Notice that there is not a manufacturing program below the new part operation. This does not mean that you cannot insert machining operations after the part operation.

Double Select Part Operation.2. If you remember, this will bring up the *Part Operation* window that will allow you to change the various information about the part.

The machining axis for the other side will need to be modified. Right now the machine axis is inverted and in a poor location for the flipped over part.

Select the product or part icon and select Adv. Pocket Assy from the *Advanced Machining* directory.

Define the following machine. This will define the machine that is going to be used. Do not forget, to define the machine you have to first select the machine icon. Do not forget to select *OK* when you are done.




The image shows the 'Machine Editor' dialog box in CATIA. It contains the following fields and controls:

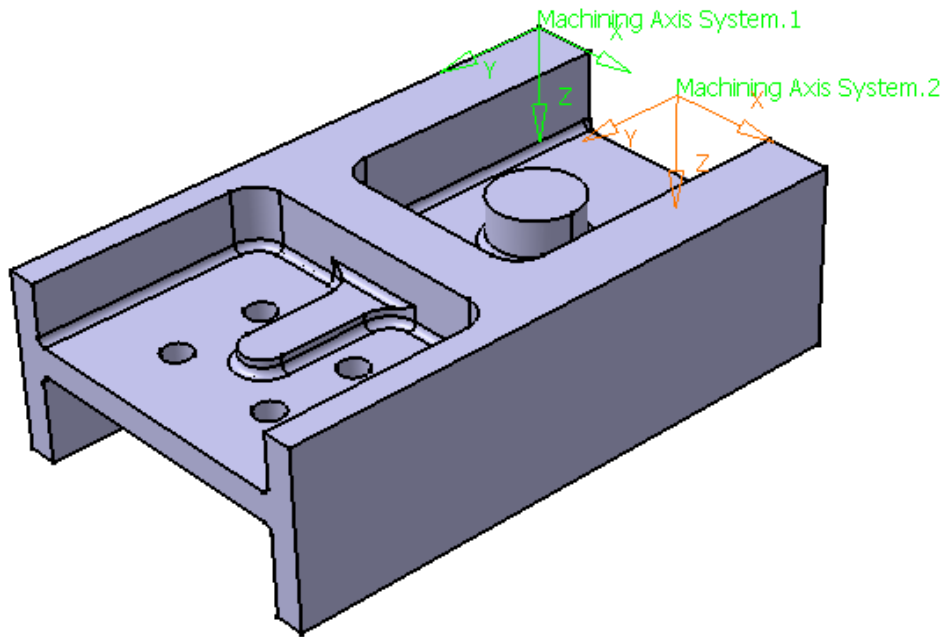
- Name:** 3-axis_Default_machine
- Comment:** (empty)
- Numerical Control:**
 - Post Processor words table: PPTableSample_Inch.pptable
 - NC data type: APT
 - NC data format: Point (X,Y,Z)
- Tool Change:**
 - Tools catalog: Not specified
 - Radius compensation:
- Spindle:**
 - Home point X: -1in
 - Home point Y: -1in
 - Home point Z: 6in
 - Orientation I: 0
 - Orientation J: 0
 - Orientation K: 1

Buttons: Advanced, OK, Cancel

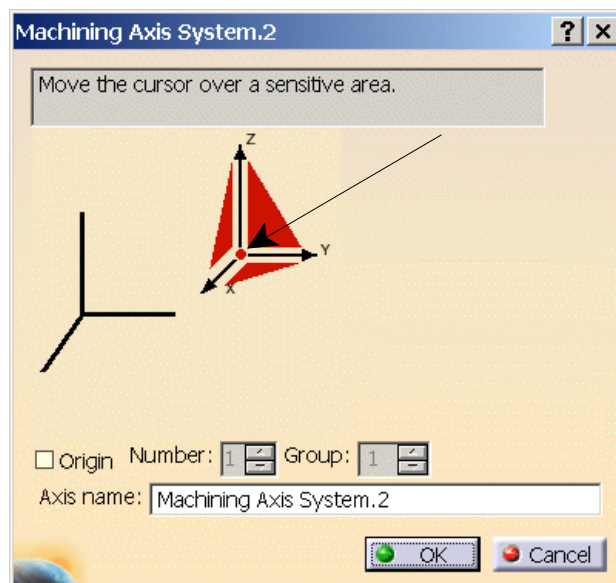
Next, the machining axis will be defined.

Select the machining axis icon.  You may remember from the beginning of this book that you can move the machining axis around with this icon. This time, not only are you going to need to move the machining axis but you will also need to flip it over.

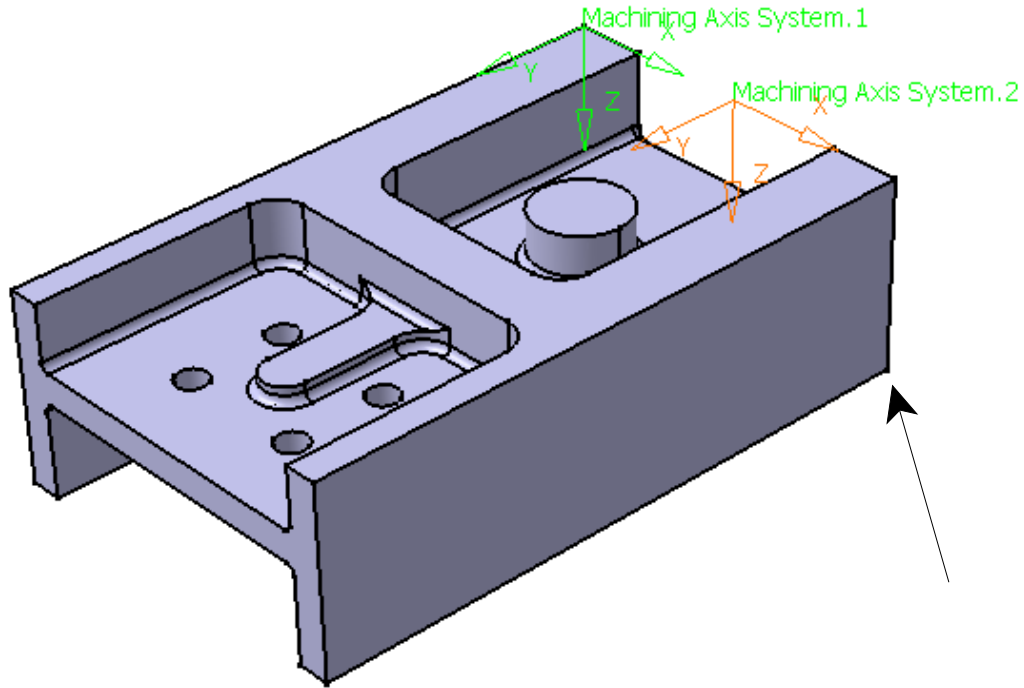
Orient your part as shown below. This will help you define and position the machining axis properly.



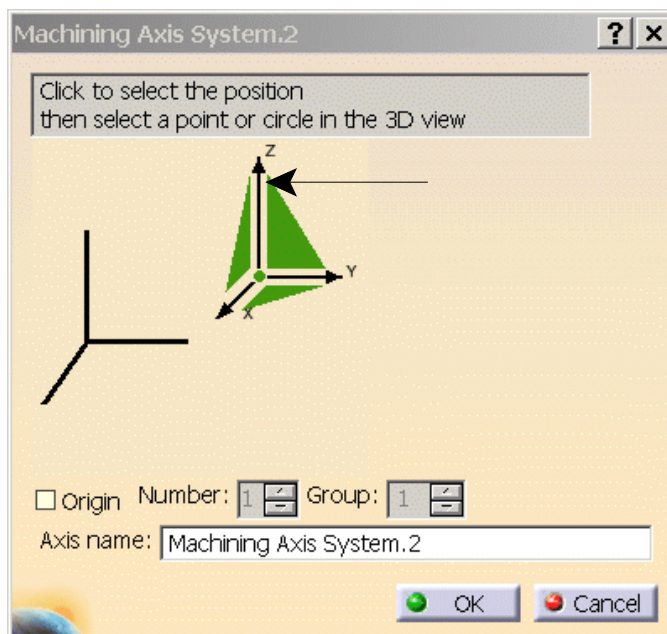
Select the center of the sensitive axis area. The center dot is indicated below. This dot allows you to move and position the axis around the part.



Select the bottom right corner of the part. Again, this corner is indicated below. Make sure you select the vertex or corner of the part and not along the side. This will position the axis with the Z axis pointing the wrong way. The axis will be flipped next.



Select the Z axis from the axis sensitive area. You will want to move the cursor to the point that it is on the Z axis but is not highlighting any of the planes around the Z axis.



This will bring up the *Direction Z* window. There are two different methods of defining the axis. First, you can select a direction. This is done by selecting an edge along your part. The second method is by defining the X,Y,Z directions manually. This is a more complicated method of defining the directions. Since the Z axis is already oriented in the correct direction but just needs reversed, you can select the *Reverse Direction* button in the *Direction Z* window and the Z axis will be corrected.

Select *Reverse Direction* from the *Direction Z* window. This will flip not only the Z axis but the Y axis also. The Y axis was flipped to keep the axis a right hand axis.

Select *OK* on the *Direction Z* window. This will bring you back to the axis definition window.

Select the X axis from the axis definition window. This time a *Direction X* window appears. This will be the same window you worked with on the Z axis direction.

Select *Reverse Direction* from the *Direction X* window. Again, this will reverse the X and Y axes to keep the axis system a right hand axis.


Select *OK* from the *Direction X* and the axis system definition windows. This will set the axis in a proper orientation.

Define all of the *Geometry* options. You may need to bring the stock material out of hiding so that you can select the safety plane.

Change the tool change point to be (0,0,6). This will define the tool change location to be at the same coordinates as the previous part operations. Remember, this is under the *Position* tab.

Select *OK* when finished. This will define the new part operation.

Now that you have a new part operation defined, it is time to define a manufacturing program in the part operation.

Select the manufacturing program icon and then select *Part Operation.2*.  This will insert a second manufacturing program into the manufacturing process.

You are going to want to use the same tool and tool change information for the second manufacturing.

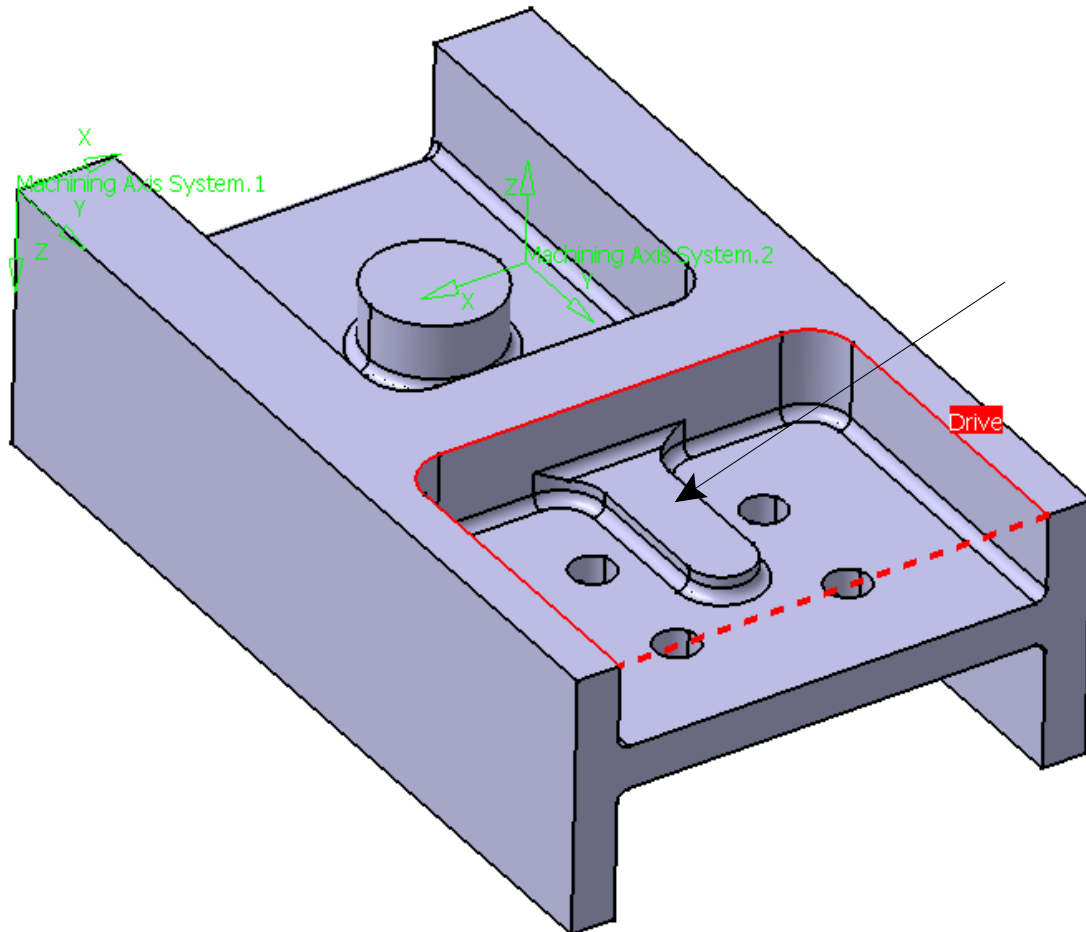
Using the third mouse button select on *Tool Change.1*. This will bring up the contextual menu for the tool change operation.

Select *Copy*. Using the third mouse button select on *Manufacturing Program.2* and select *Paste*. This will put the exact same tool change operation into the second manufacturing program. Now it will be time to start machining.

Take a close look at both pockets. Notice that they will have to be machined in two different operations for each one. The front open pocket will have to be machined down to the level of the top of the web sticking out from the center, then a new boundary will have to be defined to machine the rest. For the back pocket, one pocket motion will have to be performed to machine down to the level of the island and then another motion will have to be made to the bottom of the part.

Define a pocket operation after *Tool Change.2*. Make the pocket an open pocket and define the outside boundary around the pocket making the open end of the boundary a soft boundary. Remember, you can only select the boundary by edges when you are working with an open pocket. To make a side of an open pocket soft, all you need to do is select the heavy red line.

Define the bottom of the pocket. Remember, you are going to have to mill the pocket in two steps, so make sure you define the face shown below and not the face with the holes in it.



Define the top of the pocket. This will complete the geometry definition for the pocket.

Change the following machining operation parameters.

Tool Path Style: Outward helical

<p>Machining Radial Axial Finishing HSM</p> <p>Direction of cut: Climb ?</p> <p>Machining tolerance: 0.002in ?</p> <p>Fixture accuracy: 0.01in ?</p> <p>Type of contour: Optimized ?</p> <p>Compensation : P1</p>	<p>Machining Radial Axial Finishing HSM</p> <p>Mode: Tool diameter ratio ?</p> <p>Distance between paths: 0.487in ?</p> <p>Percentage of tool diameter: 35 ?</p> <p>Overhang: 90 ?</p> <p>Center path overlap: 75 ?</p> <p>Reduction percentage: 100 ?</p> <p>Pocket Navigation</p> <p><input type="checkbox"/> Always stay on bottom <input type="checkbox"/> Inward/outward mix</p>
<p>Machining Radial Axial Finishing HSM</p> <p>Mode: Maximum depth of cut ?</p> <p>Maximum depth of cut: 0.25in ?</p> <p>Number of levels: 1 ?</p> <p>Automatic draft angle: 0deg ?</p>	<p>Machining Radial Axial Finishing HSM</p> <p>Mode: No finish path ?</p> <p>Side finish thickness: 0in ?</p> <p>Side finish thickness on bottom: 0in ?</p> <p>Bottom finish thickness: 0in ?</p> <p><input type="checkbox"/> Spring pass <input type="checkbox"/> Avoid scallops on bottom</p>

Change the following feedrates and spindle speed.

Feedrate	
Approach:	15in_mn
Machining:	30in_mn
Retract:	5in_mn
Finishing:	10in_mn
Unit:	Linear
Spindle Speed	
<input checked="" type="checkbox"/> Spindle output	
Machining:	120turn_mn
Unit:	Angular

Check to make sure you have approach and retract macros and replay your operation. Select OK when done.

Define the pocket motion for the back pocket, machining the pocket down to the top of the island. The steps to accomplish this are the same as what you performed earlier. There will be no difference between the steps. Do not forget to make the back of the pocket open.

Select the manufacturing program icon and then select *Manufacturing Program.2*.

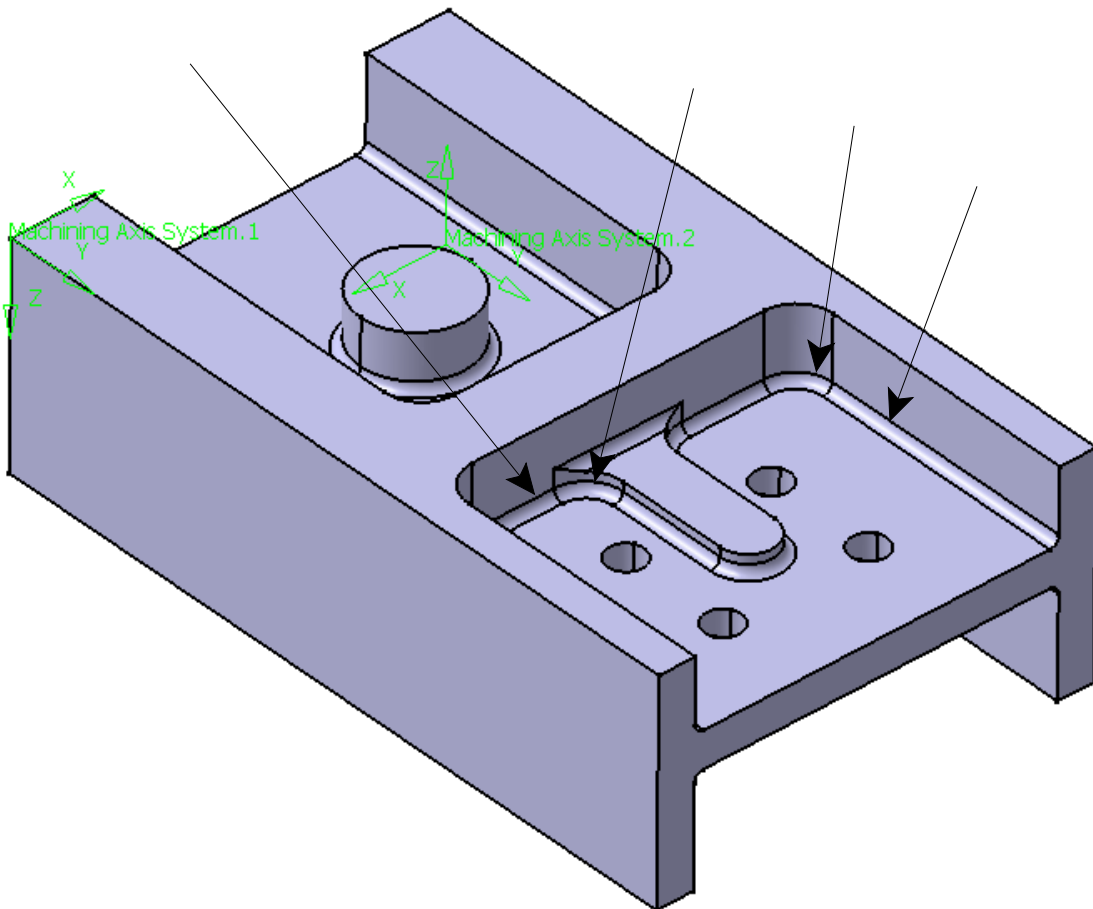
This will create another manufacturing program below the second part operation. You will stick the second level of machining in this second manufacturing program.

Copy and Paste the tool change operation into the new manufacturing program. This will put the exact same tool change into the new manufacturing program.

Define a pocket motion after Tool Change.3. This pocket motion will finish the open pocket on the side with the holes.

Select the boundary from the sensitive area on the pocket. This time, to define the boundary you are going to want to select the top of the fillet for the boundary. The fillet boundary will need to be selected so that you will not need to add a boundary offset.

Select the top edge of the fillet around the bottom of the part. Be sure to close the boundary and select *OK* when finished. Make the open side a soft boundary. A few boundary lines are indicated below.



Define the top and the bottom of the pocket. Be sure when you define the top you define the top of the tab and not the top of the entire part. Remember, you have already milled down to the level of the tab.

All of the machining parameters and macros have already been defined. You can double check them if you desire.

Select *OK* when done. This will finish the pocket operation. Do not forget to replay your operation when done.

Create a pocket operation to finish the back pocket. This pocket can be handled in the same manner as the previous pocket operation. The only difference is you need to define the island in the middle.

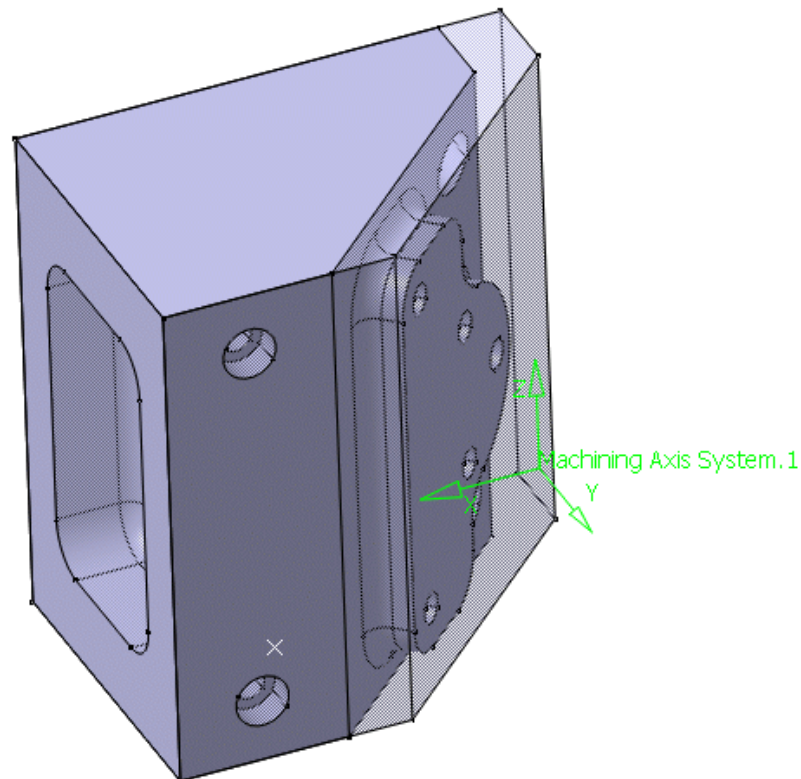
You can define the drilling operation for the holes if you want to but it is not required.

Notice when you replay the operations, you only view one manufacturing program at a time. This can be annoying to say the least and is the reason to have as few manufacturing programs as possible.

Machining Axes

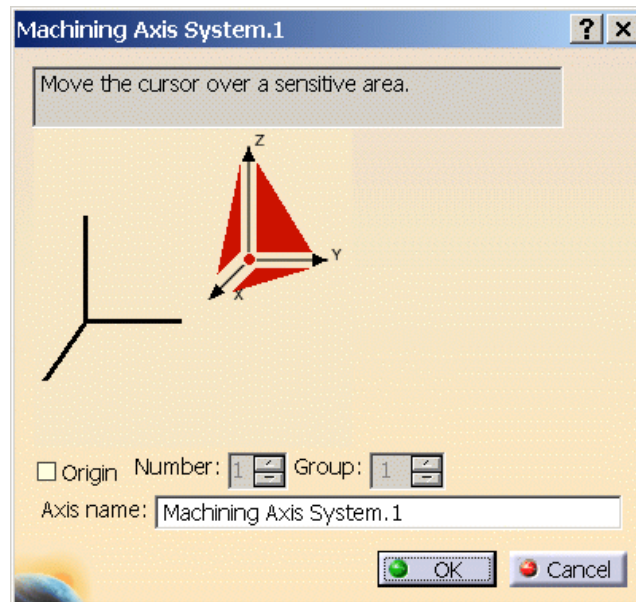
In many cases, the default machining axis is not going to be a proper machining axis. This is why it is important to know how to set the axis to a new location and orient it properly.

Open the Mounting Block Machining process found in the *Mounting Block* folder. This part will serve two uses. First, multiple different machining axes are going to need to be made since this part will be machined on two different machines. Second, the part will be used on a horizontal machine with a rotary axis.



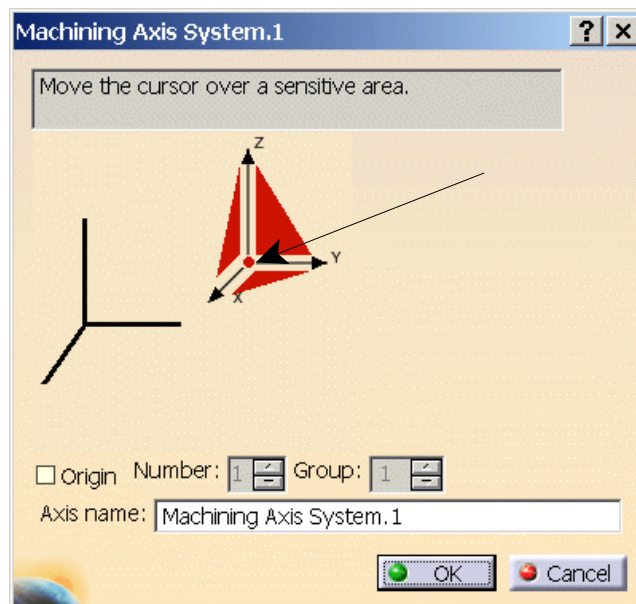
On a horizontal machine, the Y axis usually points upwards and the Z axis points along the tool axis directions. This is why it will be necessary to modify the machining axis system already provided. The next thing that can be noticed about the part is the machining axis is not at the center or close to the center of the part. Note the point near the center though. In order to define an axis at a location you must have a point at that location. If you don't use a point, a vertex will have to be used.

Double select the machining axis system from the workspace. This will display the *Machining Axis System* window.



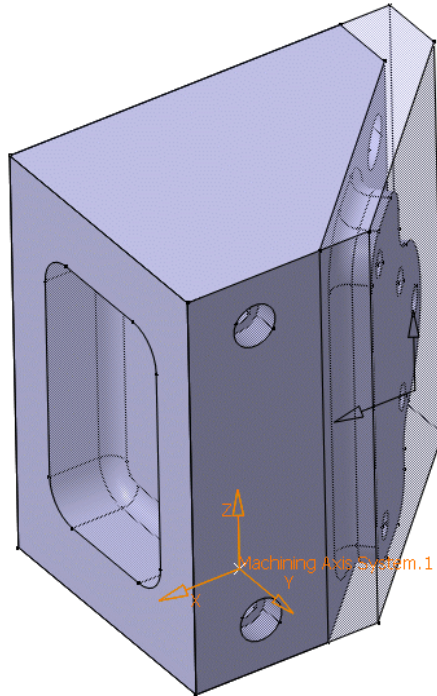
Notice this window is the exact same as what was discussed in the part operation setup. It will be used in the same manner with the exception that this time, the axes will also be modified.

Select the center dot of the axes as shown below. This will allow you to re-define the center or origin of the axes system.



The window will disappear.

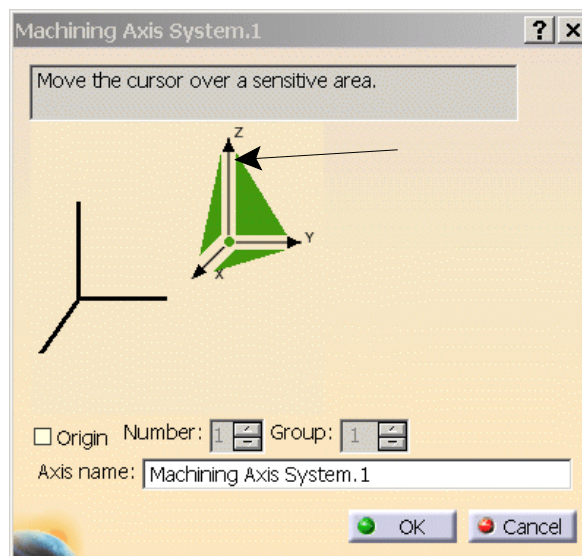
Select the point in the center bottom of the part. The axis system will re-locate to that point.



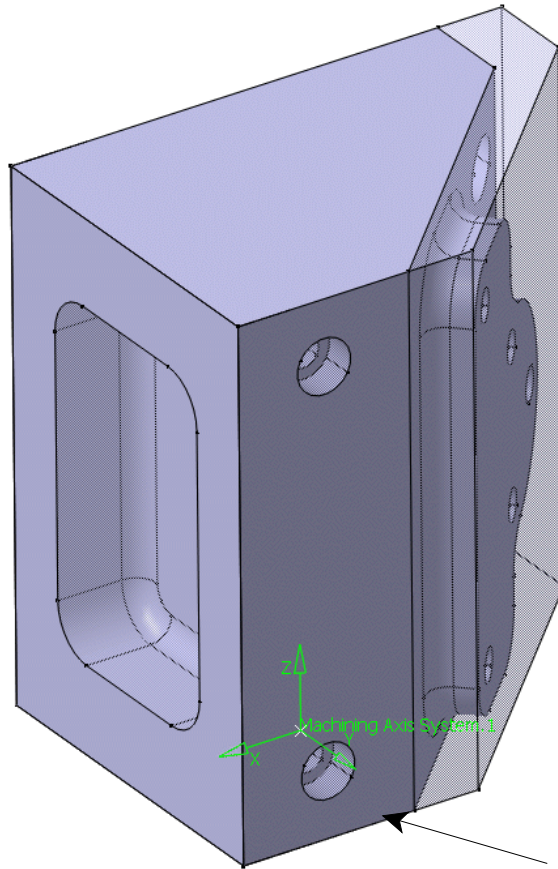
Now, the next step is to move the axes and set them to a horizontal machine axis system.

Both the X and Z axes on the machining axis dialog box are sensitive and can be selected to modify their orientation. This is not the case for the Y axis.

Select the Z axis from the sensitive axes area as shown below. This will bring up the *Direction Z* window. The window is labeled *Direction Z* due to the Z axis being selected. Selecting the X axis would yield the exact same window but with a different label.



Select the front edge running along the top of the part as shown. This will define the direction of the Z axis.




The Z axis runs along the positive direction of the edge, but it might put the Y axis in the wrong direction.

Select **Reverse Direction** from the **Direction Z** window if the **Y axis points away from the part**. This will reverse the direction of the Z axis and thus reverse the direction of the Y axis. You should notice your newly oriented axis is stacked on top of the original axis making it difficult to make out the direction of the axes. This is not a problem. When you are done defining the axis the old orientation will disappear.

Select **OK** on the **Direction Z** and the **Machining Axis System** window. This finalizes the new axis setup and erases the old orientation.

Now it is time to define and setup the part operation.

Double select *Part Operation.1* and then select the machine icon. This will be where the horizontal machine with a rotary axis will be defined.

Select the 3-Axis with rotary table icon for the machine specification.  This will define the rotary table machine as well as display a new frame for the *Rotary Table* information. Take a closer look at the *Rotary Table* frame.

Rotary Table	
Center point X:	0in
Center point Y:	0in
Center point Z:	0in
Initial position:	0deg
Rotary angle:	0deg
Rotary axis:	A
Rotary direction:	Clockwise
Rotary type:	Absolute

Center point

Defines where the center of the table is or where the table rotates about. The X, Y and Z coordinates are all relative to the machining axis defined. Since the axis was moved to the center of the part it will be unnecessary to define a new set of coordinates.

Initial position

Defines where the table currently is located. Many times the table will be reset to zero therefore this angle will be at zero degrees. But, if several part operations were in the same process list then the final position of the last machining operation would need to be inserted here.

Rotary angle

Defines the maximum angle the table could turn. Setting this to 0 or 360 deg sets the maximum angle the table can turn from zero. If the contextual menu is brought up for the rotary angle an option is available to define a maximum and minimum angle.

Rotary axis

Defines the axis that the table will rotate about. Since the Y axis is in the vertical direction then the rotation axis will be the B axis. Refer to the introduction if you don't remember all the rotary axes.

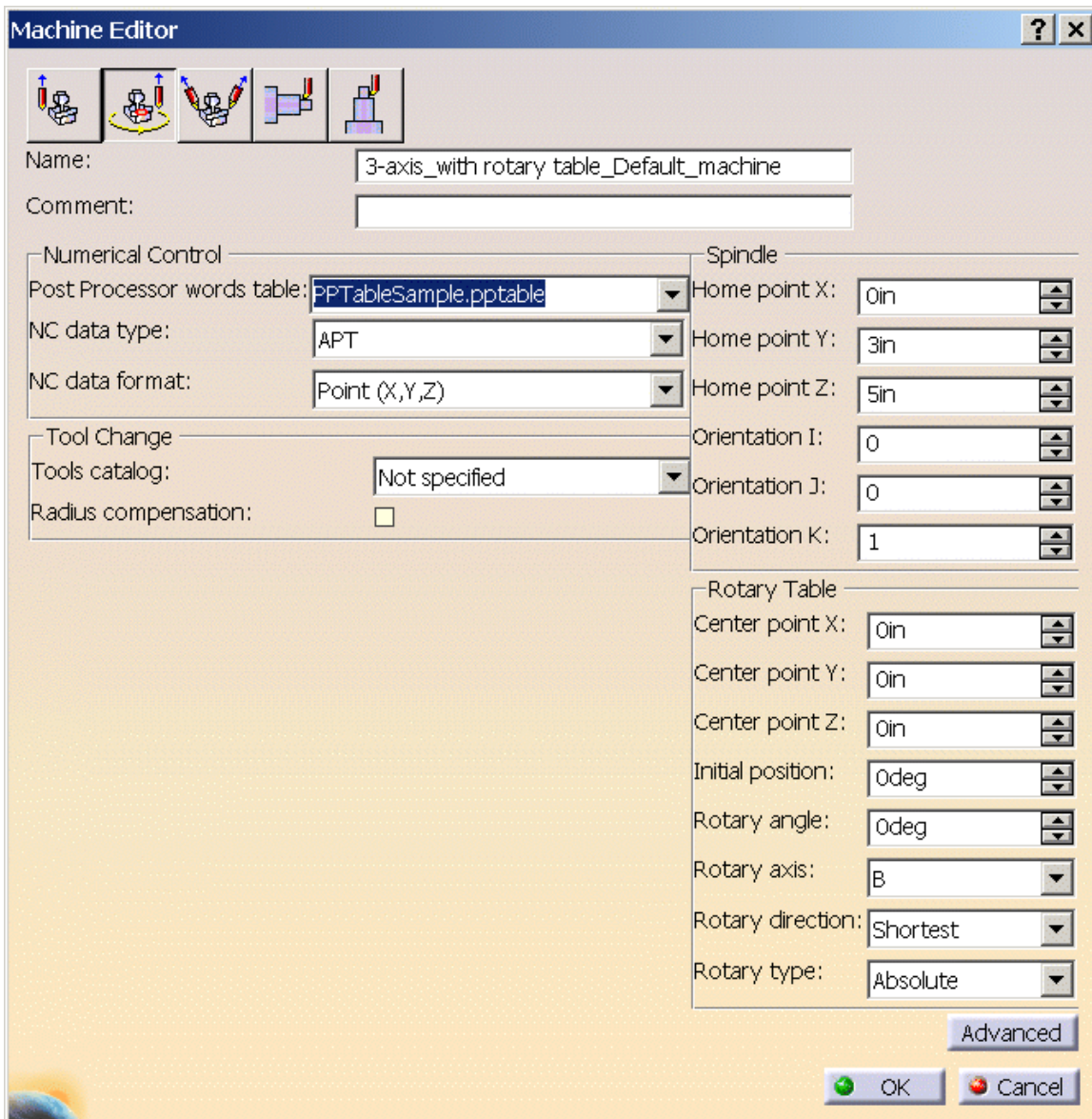
Rotary direction

Defines the direction that the table rotates. There are three options: *Clockwise*, *Counter-clockwise* and *Shortest*. *Shortest* will issue a clockwise or counter-clockwise motion depending on the shortest path to the new angle.

Rotary type

Defines how the angle is specified to the machine.

Change the following information per the picture below. The only thing to change is the spindle home point as well as the rotary axis and rotary direction.

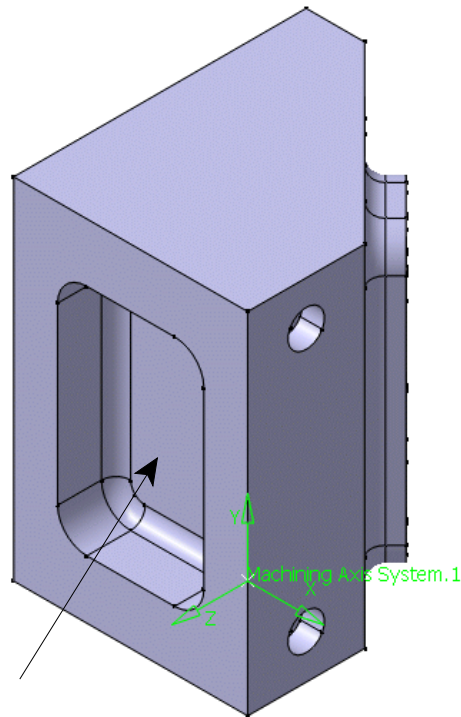


Select OK when done making changes. This will define the machine that will be used to machine the part. Take note of the orientation fields. This defines the initial orientation of the tool axis. Right now it is set to (0,0,1). The I, J, K values depict what orientation along the X, Y and Z axes respectively the tool will be set to. Since the I, J, K values are set to (0,0,1) then the tool will be oriented straight along the Z axis.

Define the stock, final part and set the Tool Change Point to be (0,5,10). Select OK when done. Note: no safety plane was defined. This is due to the machine rotation. A safety plane for one angle is not going to be the safety plane for another angle.

Now the part is ready to be machined with the horizontal machine.

Define a pocket motion to machine the back pocket as shown below. Notice that when the bottom of the pocket is defined the tool axis displays normal to the plane.



Define appropriate machining operation parameters and a tool. The tool is going to need to be a 1" diameter with a .25" corner radius.


Define approach and return macros. Be sure to manually select the clearance plane for each macro. If the plane is not selected then the macro will not know where the plane is supposed to be.

Select OK when done. This will finish the pocket for the back side. Now it is time to machine the pocket on the bottom side.

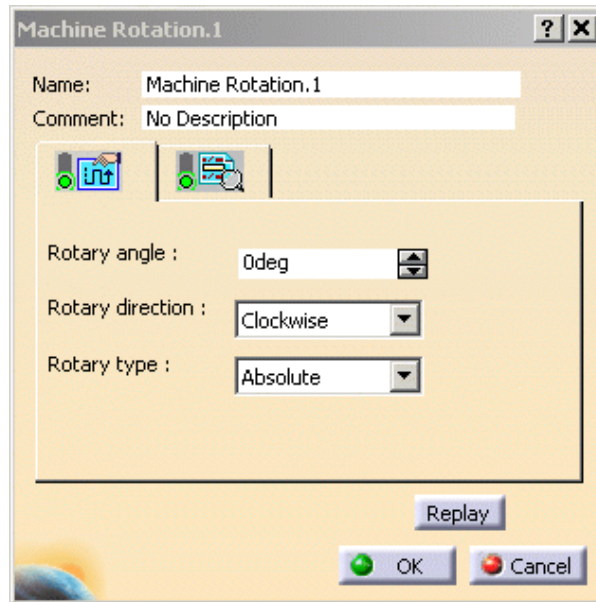
There are two methods for defining a machine rotation. One method is to use the machine rotation icon. The second method is to automatically define the machine rotation. Both will be investigated.

Next the bottom pocket will be machined. The machining axis will be handled exactly the same way as the previous pocket with the exception that the machine rotation command will be given.

Machine Rotation

Select the Machine Rotation icon.  Like all other machining operations you will have to specify where you want the machine rotation to occur.

Select *Pocketing.1*. This will bring up the *Machine Rotation* window.



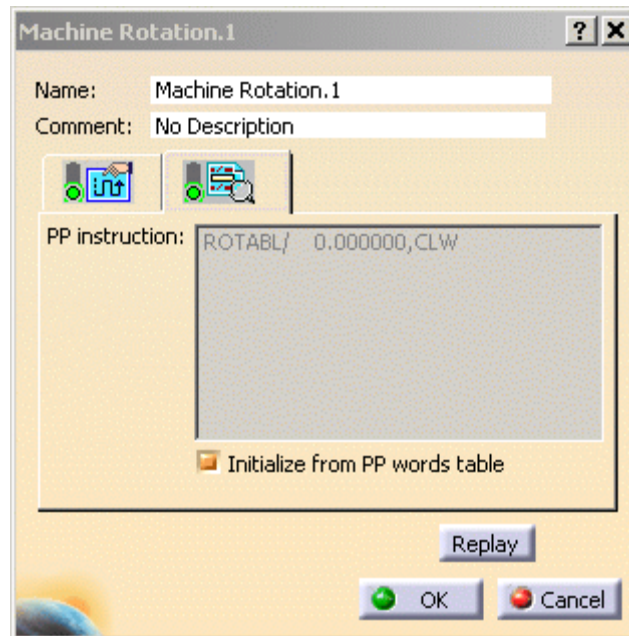
Machine operation parameters tab

Rotary angle Defines the angle at which you are rotating the machine.

Rotary Direction Defines the direction that the machine will rotate.

Rotary Type Defines if the rotation angle is absolute or incremental.

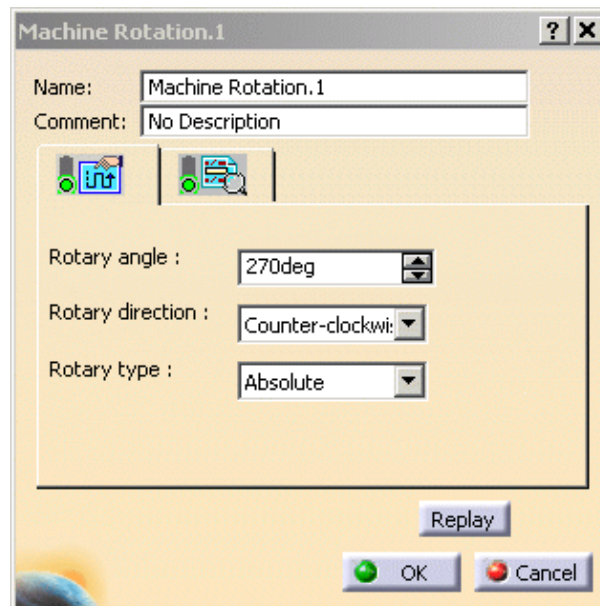
Post Processor instruction tab



The Post Processor instruction tab allows the addition of other Post Processor words. If the *Initialize from PP words table* is turned on, whatever post processor instruction that is defined in the table will be used.

Change the Rotary angle to 270deg. Remember, the rotary type is absolute so an angle of 270° will need to be used.

Change the Rotary direction to Counter-clockwise. This will rotate the part in a counter clockwise direction to the back pocket.

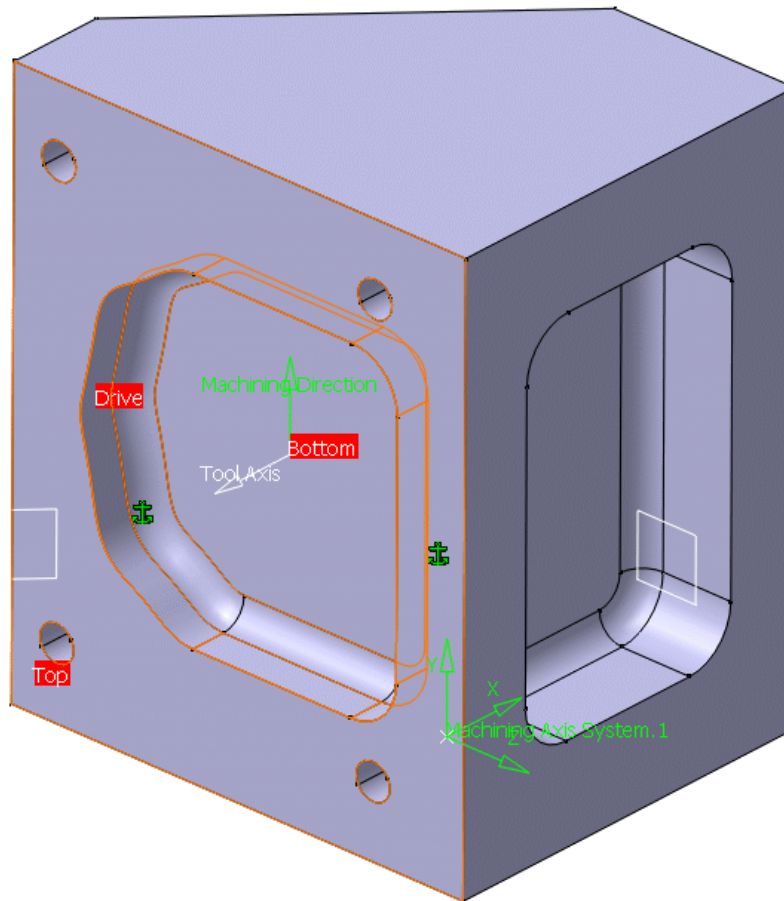


You can press the replay button if you want to see the rotation.

Select *OK* when done. This will insert the machine rotation into the machining process.

Define a pocket operation after *Machine Rotation.1*. Notice the tool axis is not changed. This is not a concern since it will be changed when the bottom of the pocket is defined.

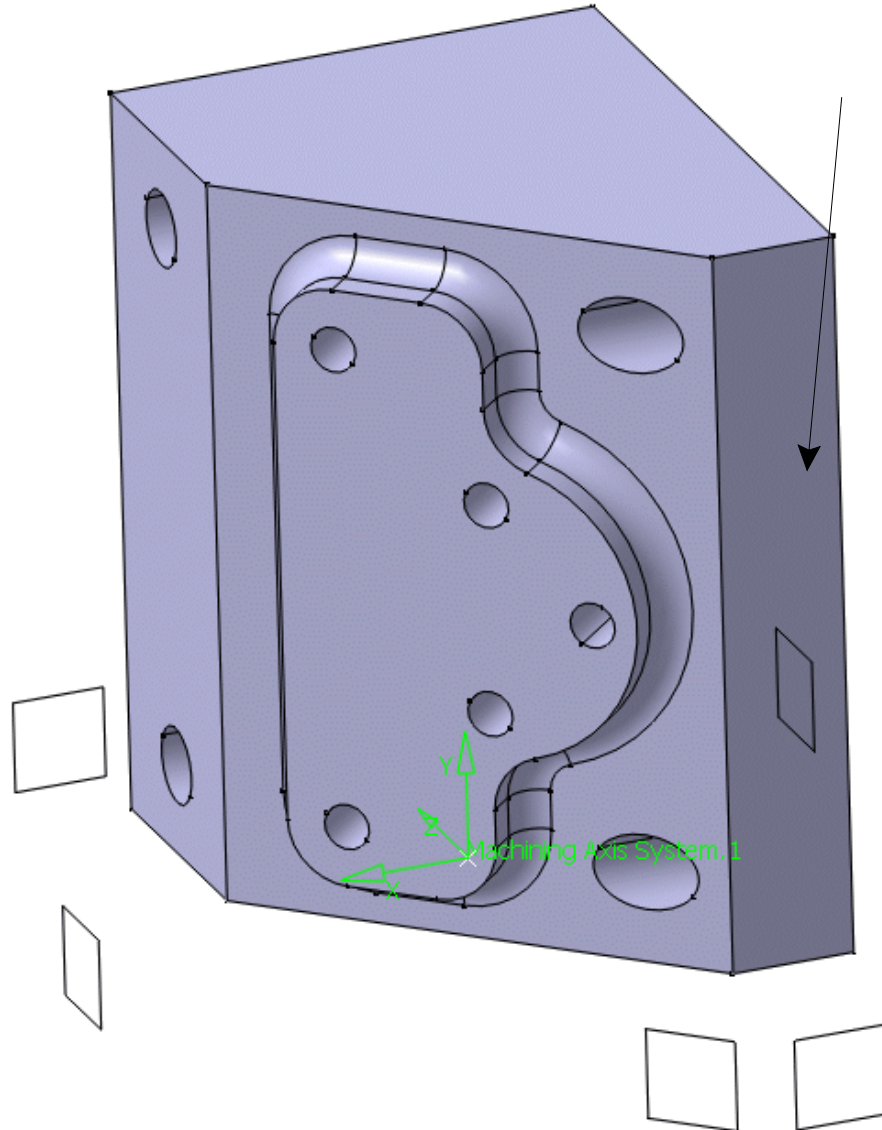
Define the geometry for the pocket. Notice when you define the bottom of the pocket the tool axis changes.



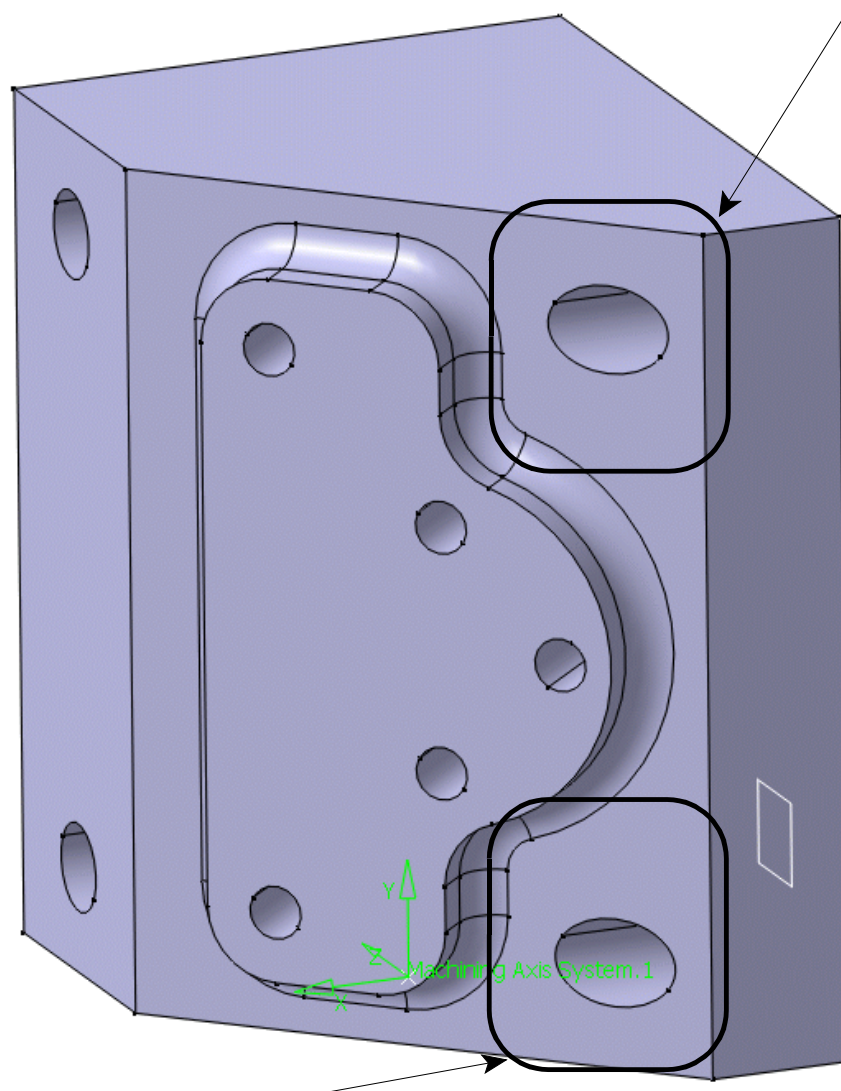
Define the machining operation parameters, check the tool and feedrates and define the macros. The macros will look like they are defined. Be sure to re-define them though. They were defined using the original plane on the wrong side of the part.

Replay and select *OK* when done. This will complete the pocket motion for the bottom of the block. If the replay is viewed in video mode, it can be seen that it crashes with the part. This is not what is desired, however, leave the mistake alone it will be fixed later.

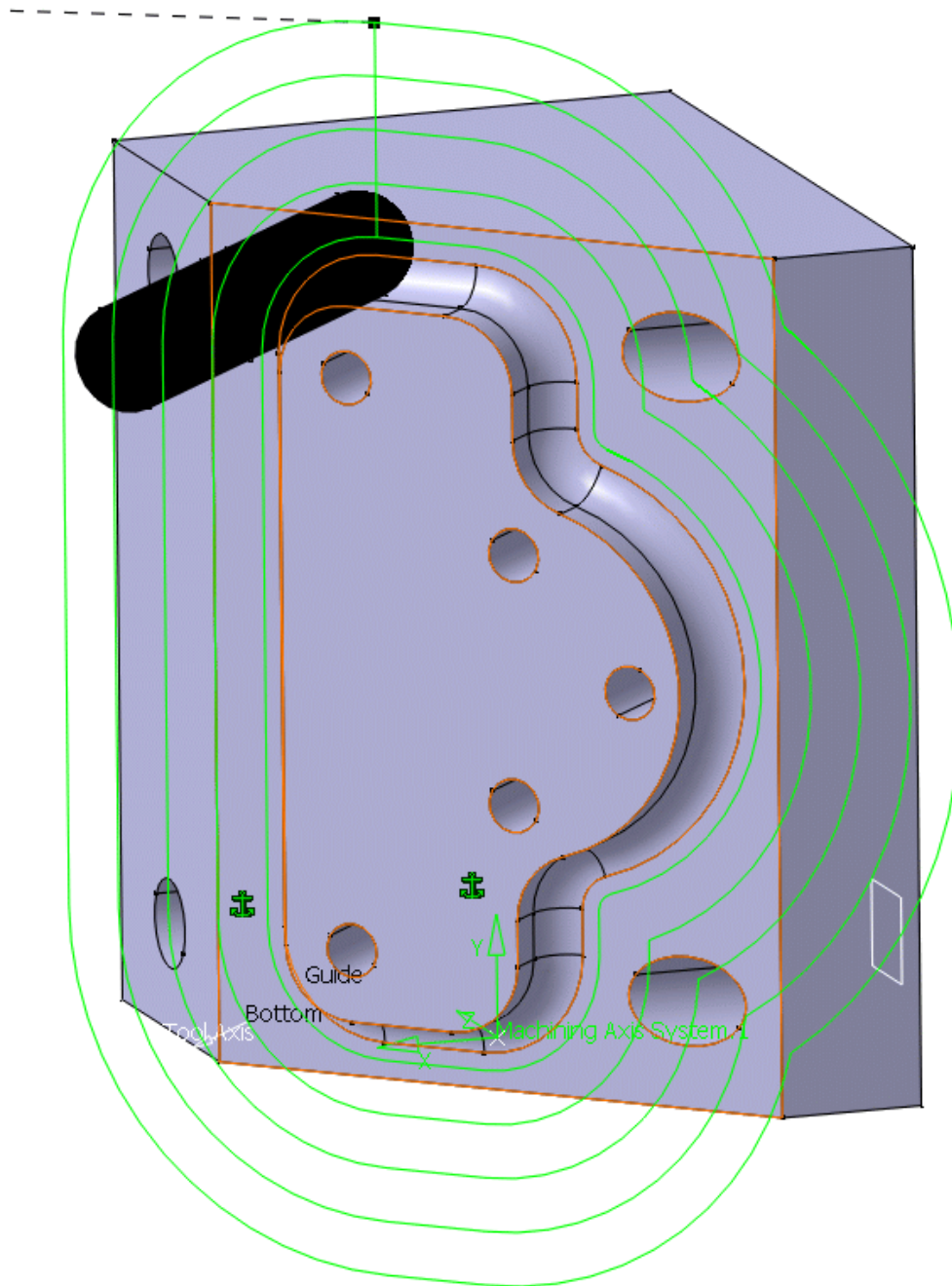
Using the same tool, define a facing motion on the narrow edge of the part. The facing operation will be sure to take care of the entire area. It is not necessary to change the tool since it will work just as well as any other tool and it will save a tool change. Be sure to watch the tool axis making sure the tool axis points away from the part. Don't worry about the machine rotation that will be taken care of later.



Now it has come time to machine the front of the mounting block. The first thing to notice is the large areas, indicated below, that are going to require extra machining.



There are a few options to make sure the entire area gets machine. One option is to add enough radial steps to make sure the area is cleared.

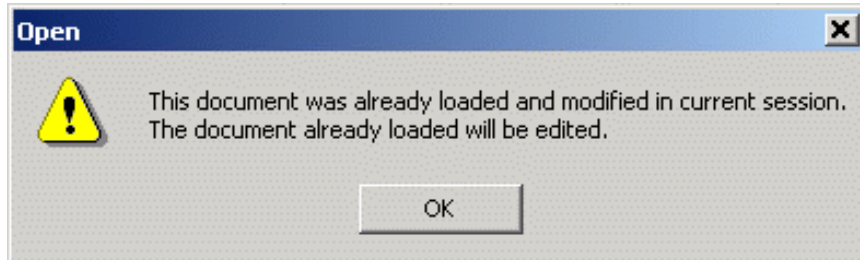


Take note as to how much the tool is machining air. This is very costly and wastes time. There is also the added danger that the machine bed or clamps are near the part and adding enough radial steps to cover the missing area will only cause the tool to crash with other parts.

The second option is to create sketches that define where you want extra machining to take place and machine those areas separately. This will give the added advantage of machining the smaller areas, then making a couple of finishing passes around the part.

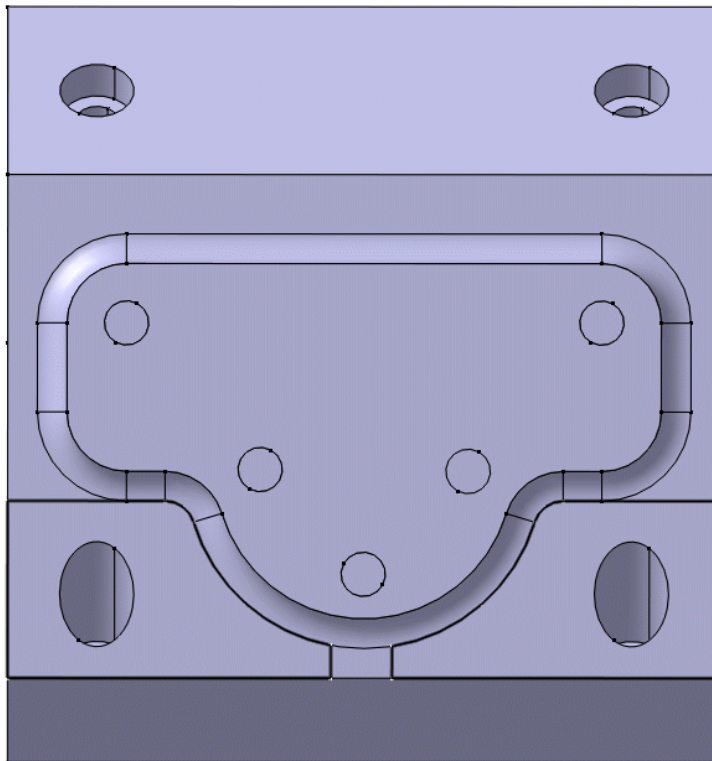
With the machining process open, open the Mounting Block document. The reason it is imperative that the machining process be open is the fact that the link between the Mounting Block CATPart and the machining process will be updated. Otherwise the link will not be updated and it will be lost when you open and save the new part.

A message about the document being already loaded and will be edited will appear.

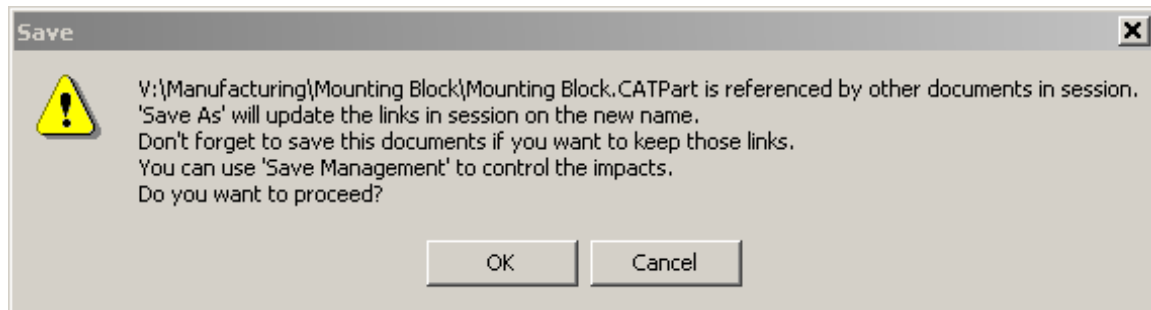


Select OK. This will confirm the opening of the document already loaded.

Create two separate sketches as shown below. The sketches are shown in dark heavy lines. The sketches do not need to be at any exact location. The idea is to create an area to machine separately so the tool is not making a bunch of passes in space. Be sure that two separate sketches are used and both sketches are not put into one.



Select PDM File, Save As and save the document in your folder. Another warning window will display, explaining that the link for the machining process will be updated with the new document location.



Select OK and close the window. This will confirm the new document location and save the file. Closing the window will bring the machining process back to the front.

Notice the two pocketing operations need updated even though the part was not modified. Since the date on the part is newer, any machining operations will need to be updated to be sure nothing has been modified. Updating the operation is as simple as replaying it.

Now that the larger areas are defined by a sketch it is time to machine the top.

Define a pocketing operation after *Facing.1*. This will be one of the operations used to machine the larger areas.

Define an open pocket using the top sketch that you created for the boundaries of the pocket. Make the two sides open on the pocket. The open pocket will allow for the tool to pass beyond the edge of the side of the part. This is a good idea so the tool will completely cut along the top of the outside of the part.

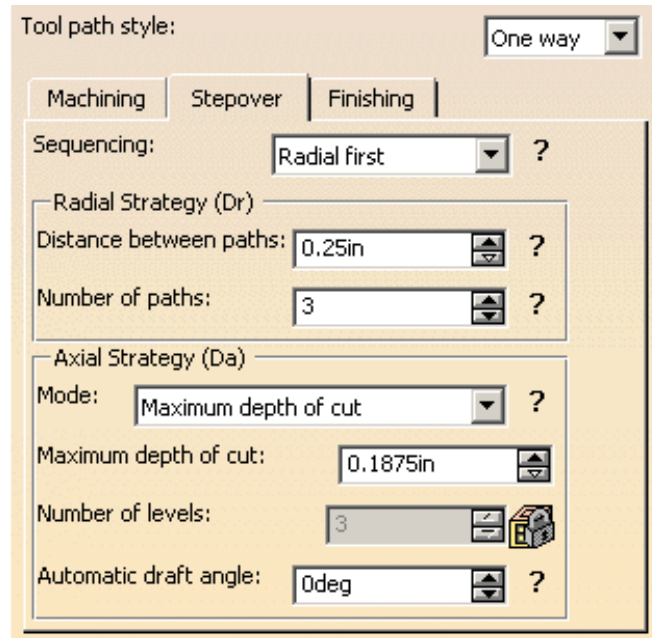
Be sure to define proper machining parameters and accurate approach and retract macros. Remember, the approach and retract macros use the original planes and not the proper safety plane.

Define another pocketing operation for the bottom sketch. The process will be the same as with the top pocket.

With the larger areas machined it is now time to finish the contour motions around the step. By machining the larger areas first, the passes required around the part will be reduced significantly.

Define a profile contour operation after the last pocketing operation. Define the geometry necessary for the profile operation.

For machining parameters, define the following:



This will define enough radial steps to cover the corners missed.

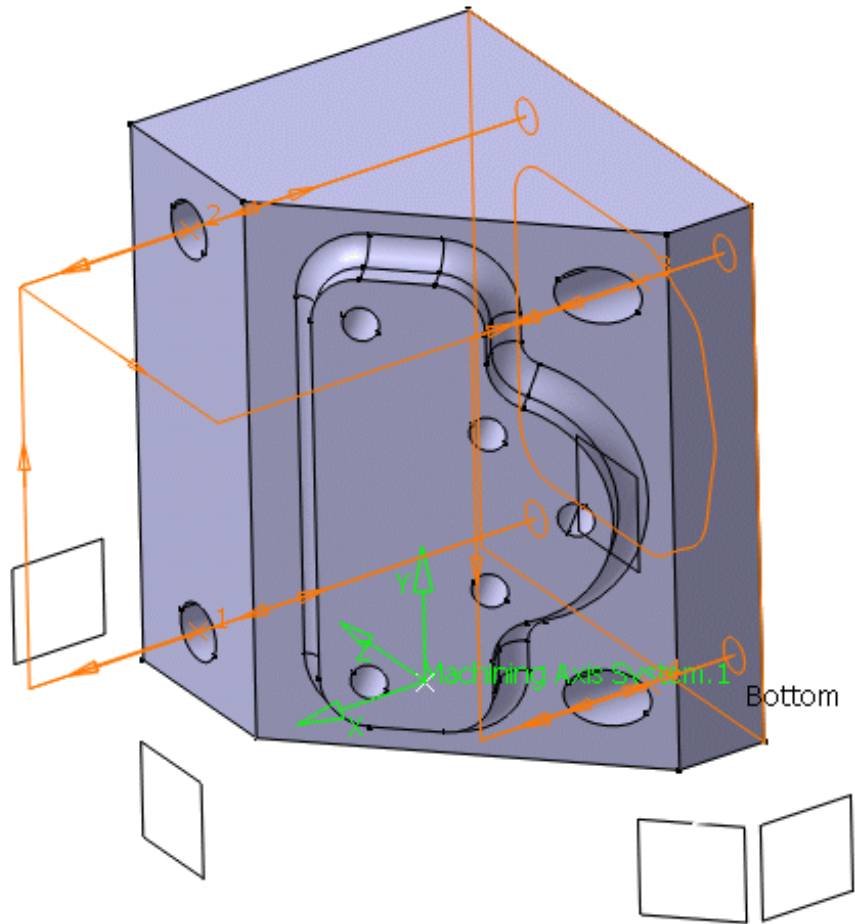
Define proper feedrates and macros. Select OK when done. Do not forget to replay your machining operation before calling it finished.

The last machining operation to define is the drilling motions for the counterbores and the hole on the step.

Define a deep hole axial operation after the profile contouring motion. For the geometry definition the four holes will be used.

Define the four holes parallel to the X axis. Define the bottom of the holes. Do not define the top of the holes. If the top of the holes are defined then each operation will start drilling from the same level. This is not what is desired. Instead an offset will be defined.

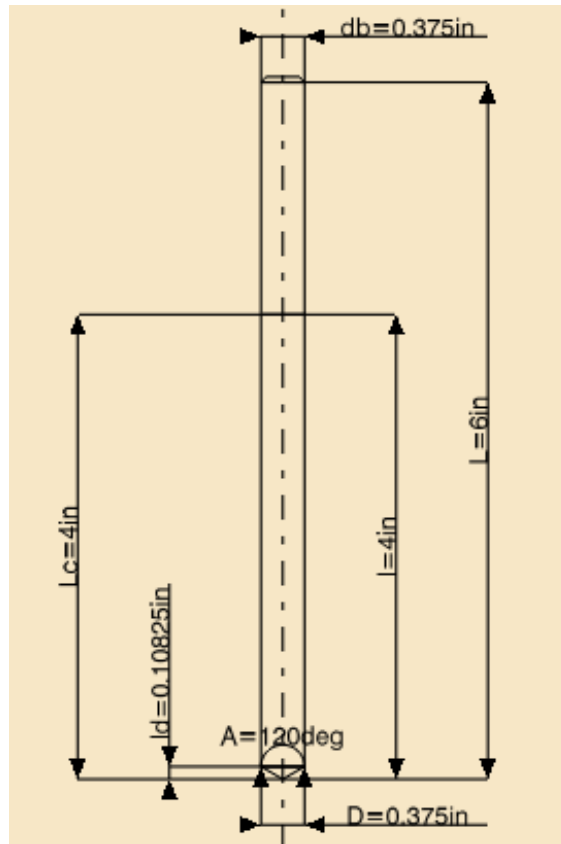
Define a jump height of 2". This will cause the tool to retract 2" between drilling operations.



Define the following axial motion parameters.

Approach clearance (A) :	0.125in
Depth mode :	By tip (Dt)
Breakthrough (B) :	0.25in
Max depth of cut (Dc) :	0.25in
Retract offset (Or) :	0.0625in
Decrement rate :	0.25
Decrement limit :	0.5
Plunge mode :	None ?
Dwell mode :	None
First compensation :	1
Output CYCLE syntax :	

Define a drill tool with the following parameters.



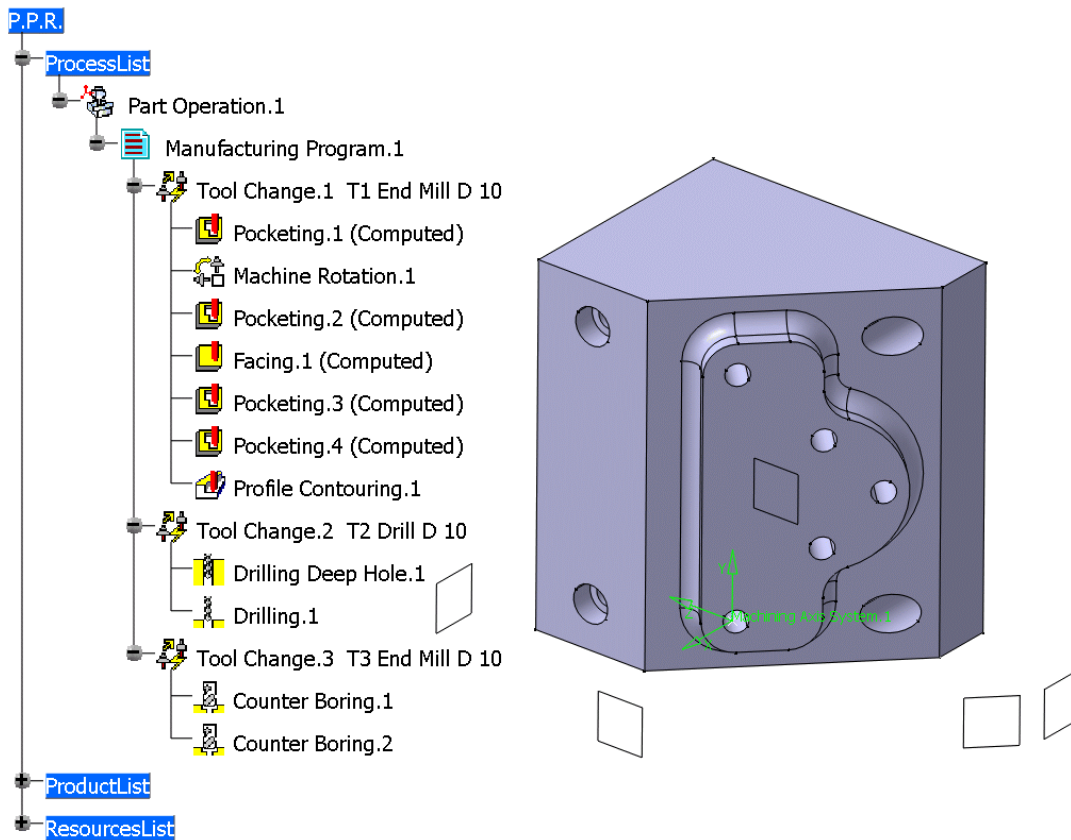
$db=0.375\text{in}$, $L=6\text{in}$, $l=4\text{in}$, $D=0.375\text{in}$, $Lc=4\text{in}$, $A=120\text{deg}$ and $ld=0.10825\text{in}$

Define the feedrates as well as approach and retract macros being sure to use the correct plane. Select *OK* when finished. As always, it is a good idea to replay the machining operation to make sure everything looks correct.

Define a drilling motion for the five holes on the step. Do not forget to define approach and retract macros, as well as other machining parameters.

Define a counterboring operation defining the two holes on the flat area as one operation and the two holes on the slant as another. For the first operation, you will want to make sure you change the tool to a flat end mill, 5/8" diameter. For the first operation a jump height of .5" will be required while a 2" jump height will be needed for the holes on the slant.

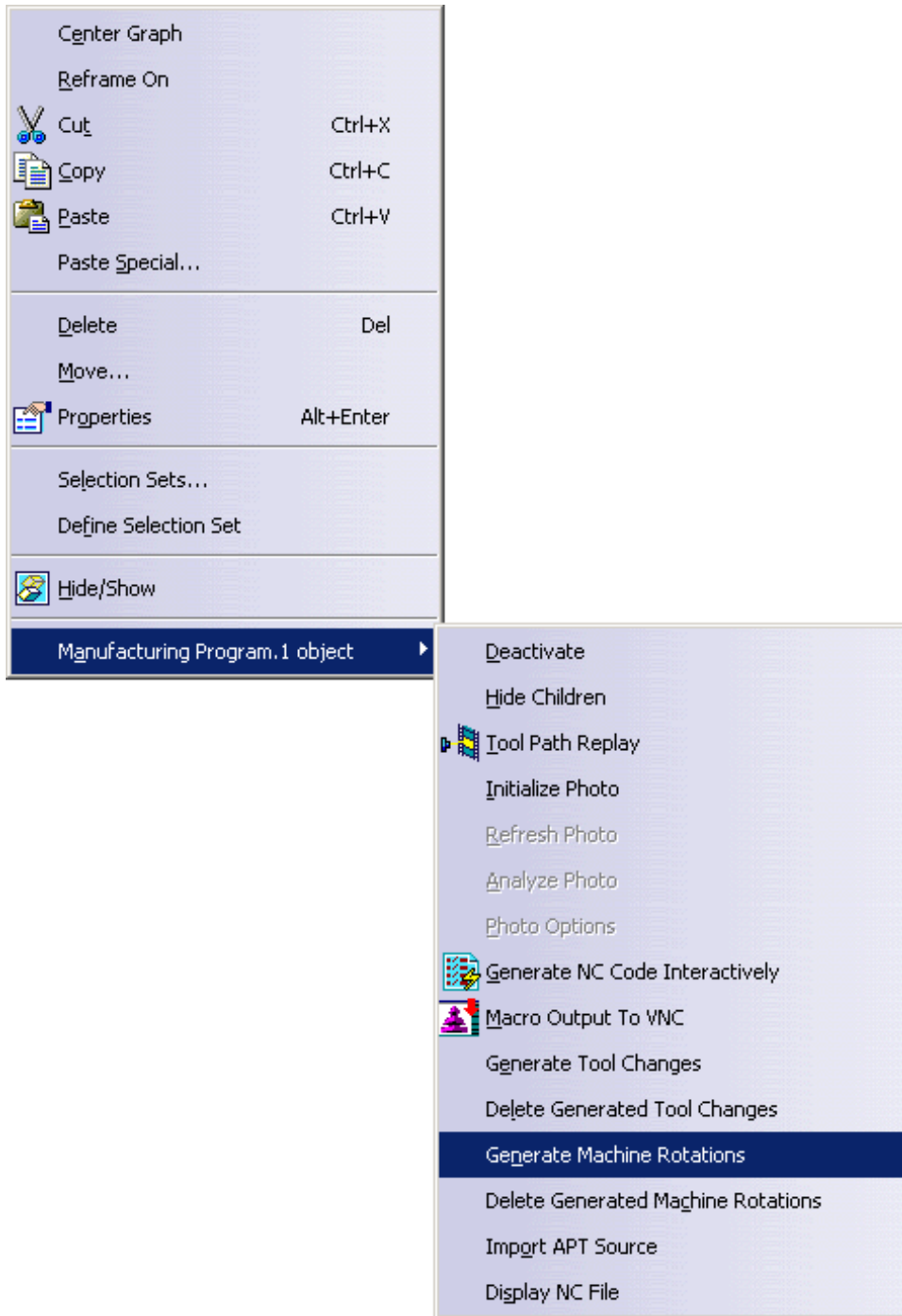
Your PPR Tree should look something like the following.

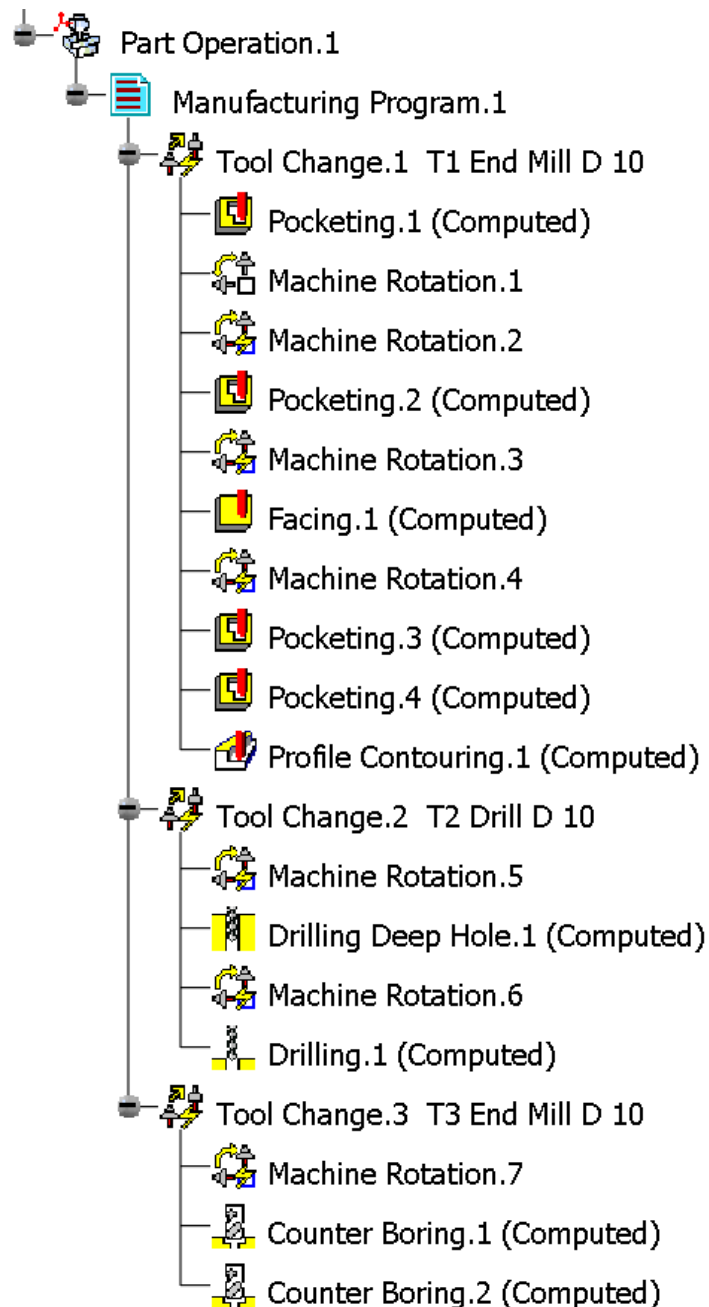


Now, all of the machining operations have been defined, it is time to finish defining machine rotations. This time all rotations will be made automatically.

Using the third mouse button select on *Manufacturing Program.1*. This will bring up the contextual menu for the manufacturing program.

Select **Manufacturing Program.1** object and then select **Generate Machine Rotations**.
 The machine rotations necessary will automatically be generated.





Notice the difference in the icon between *Machine Rotation.1* and *Machine Rotation.2*. The generated machine rotations have a small lightning bolt in the box where the manually defined machine rotations do not. Also note that there are two machine rotations right after one another. This is because *Machine Rotation.1* was generated by hand and *Machine Rotation.2* was automatically generated. For future reference all machine rotations should be either computer generated or hand generated. They should not be mixed and matched.

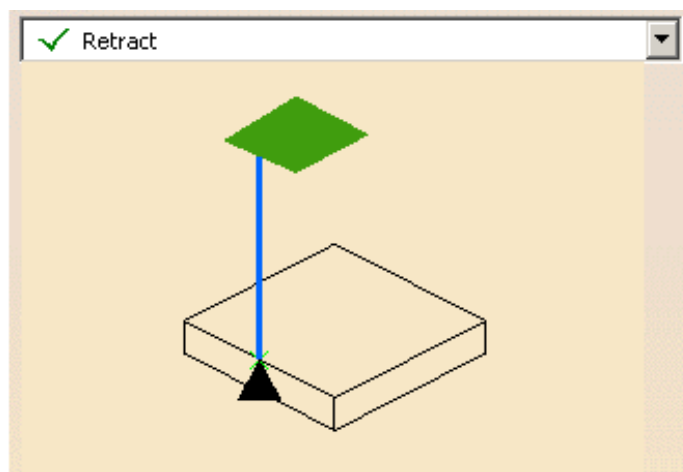
Delete *Machine Rotation.2*. Since *Machine Rotation.1* is exactly the same, it is unnecessary to have both machine rotations in the same location.

Before this exercise can be called done, there is a problem that needs to be fixed. Notice when the replay is in video mode, there is the crash with the tool at the back of the part. This is not good and will result in ruined parts and possibly a ruined machine. To fix this problem additional retract macro manipulation will take place.

Double select on the *Pocketing.1* operation and switch to the macros tab. In order to fix the crash additional macro maneuvers will need to be made.

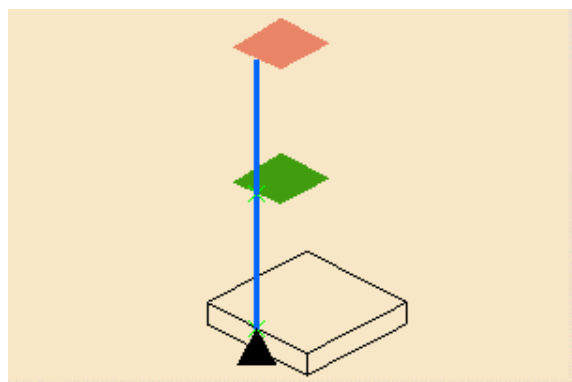
Switch to the retract macro motion if not already there. To switch to the macro all that needs to be done is to select it from the drop down options.

Currently, there is one single motion in the retract macro. Another motion will need to be made in order to keep from crashing with the part.



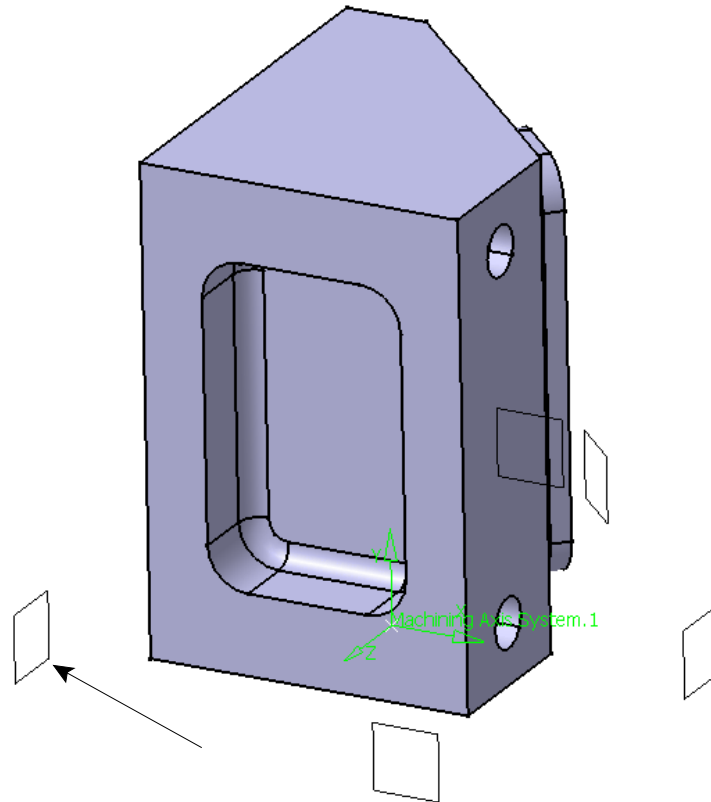
Select the user defined macros icon. This will be the icon to the far right of the line of macro options.

Select the add motion to plane icon.  This will stack another motion to a plane on top of the current plane definition.



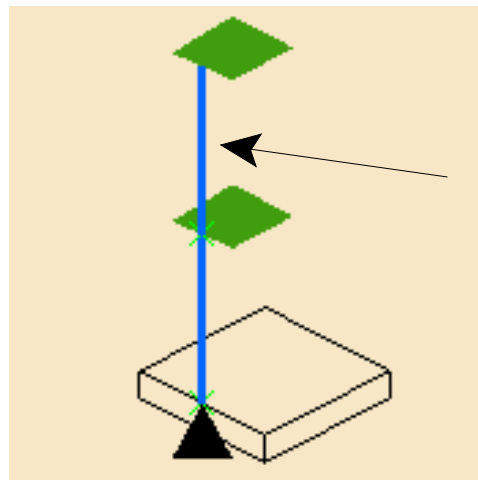
Select the top plane. This will be the orange plane, needing a definition.

Select the plane to the left of the Z axis as shown below. This will make the motion to a plane move sideways until it reaches the side plane.

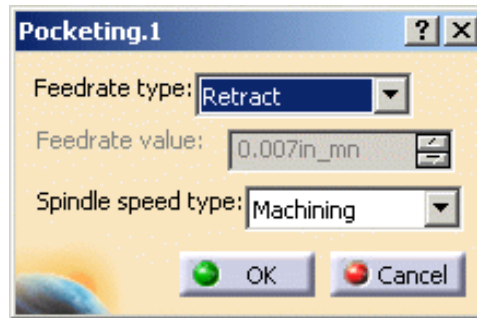


One thing to keep in mind. The tool is moving beyond the clearance plane therefore it is allowed to travel faster than the approach or machining feedrate. This is a good location or motion to make the tool travel at the rapid feedrate.

Using the third mouse button select on the second macro parameters line. The macro parameters lines are the heavy blue lines between the starting point and the plane. This will bring up the contextual menu for the motion.



Select *Feedrate*. This will bring up yet another window allowing you to change and control the feedrate for the motion.



Change the *Feedrate type* to be *RAPID*. This will make the tool travel at the rapid feedrate when going to the plane. Notice the blue line turns red. This indicates that the tool is making a rapid motion.

Select *OK* to the feedrate and pocketing operation when done. Now replay the operations in video mode. The tool will no longer collide with the part.

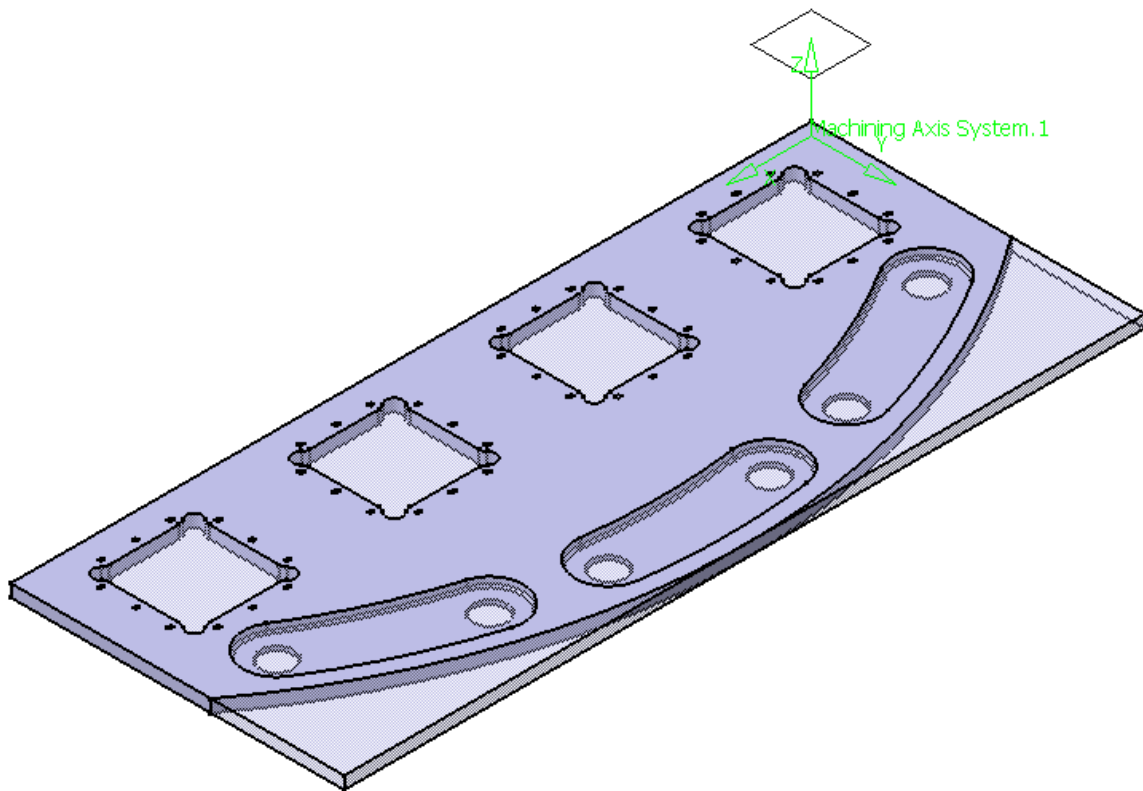
Advanced Machining Topics

Copy Transformations

Copy transformations allow one set of NC instructions to be created and then translated to multiple instances. There are two types of copy transformations: translation and rotation. Both will be investigated.

Start a new manufacturing process using the Instrument Panel Assembly. The assembly can be found in the *Instrument Panel* folder.

The stock, finished part and safety plane should be defined. The home point as well as the tool change location is (0,0,10).



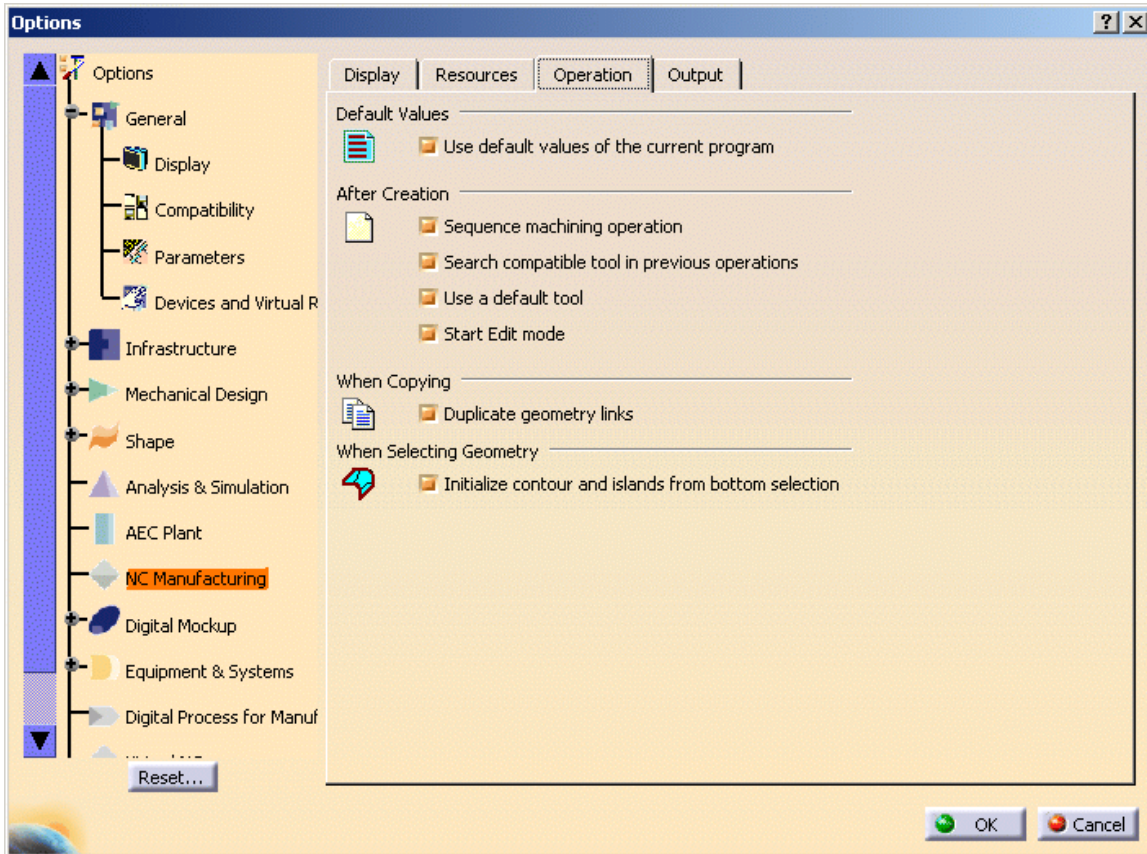
This model has a series of features that can be machined using copy transformations. Feel free to hide the stock material but not the clearance plane. The stock material will be used later when investigating knowledgware capabilities.

Start a pocketing operation after *Manufacturing Program.1*. This pocket will machine the cutout closest to the machining axis.

Change the pocket to an open bottom and define the bottom plane. Notice when you define the bottom of the part the boundaries are automatically determined. This, as in most cases results in incorrect boundary curves. There are, however, options to turn this off.

Cancel the pocketing operation and select pull down menu *Tools, Options*. Some of the NC Manufacturing options are going to be changed to keep the bottom definition from assuming boundaries.

Select *NC Manufacturing* and the *Operation* tab. This tab defines some of the automatic processes when defining operations.



The option that is of interest is the *When Selecting Geometry* option.

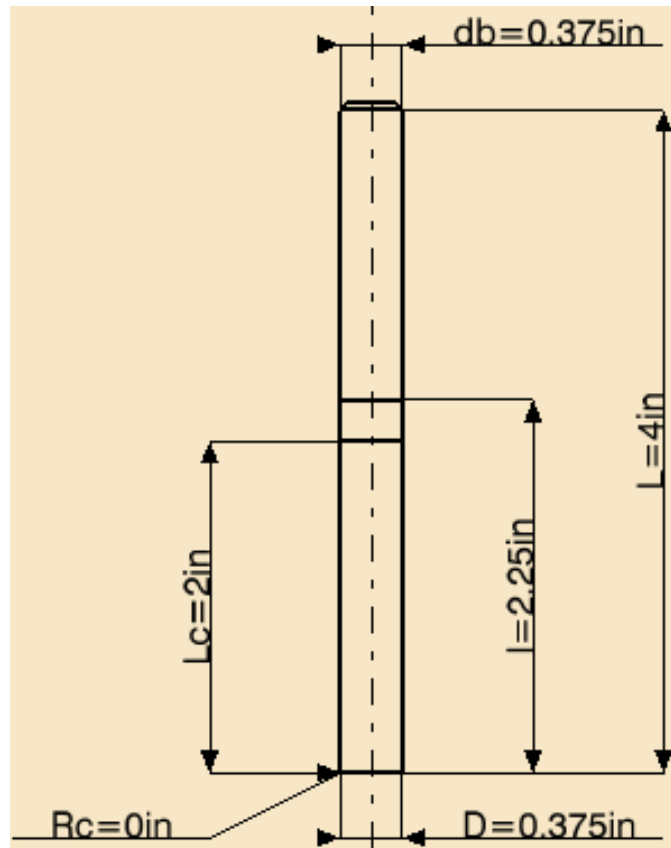
Turn the checkbox off for *Initialize contour and islands from bottom selection*. Select *OK* when finished. This will keep the bottom selection from trying to determine the islands and contour.

Notice *Tool Change.1* is still in the tree. Even when an operation is cancelled the tool changes remain.

Start a pocketing operation after *Tool Change.1*. Change the bottom mode to *Soft* and define the bottom of the pocket. Notice the contour of the part did not automatically turn green. This is because the option was turned off.

Finish defining the geometry necessary for the pocket operation. This will be the contour and top. Also be sure to define a negative offset on the bottom of the part (-.020 should suffice). This will help reduce the size of the burrs that will need to be cleaned out.

Define the following tool.

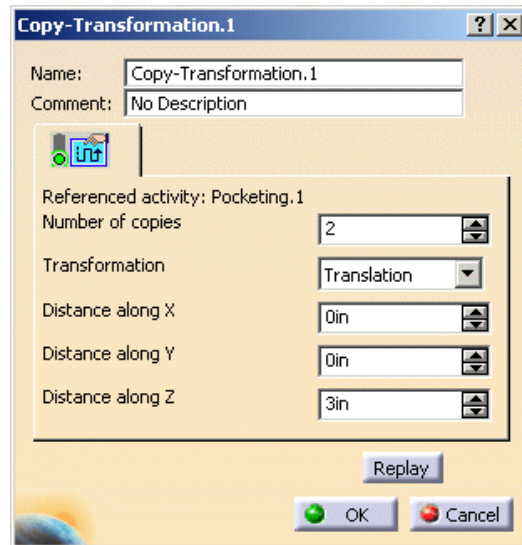


Define appropriate feedrates as well as approach and retract macros. Select *OK* when finished. This will complete this pocketing operation. This operation will now be used to translate to the other locations through a copy transformation.

Select the copy transformation instruction icon.  This icon like many others does not do anything until the operation has a defined location.

Select *Pocketing.1*. The first selection defines where the copy transformation is going to be located in the tree.

Select *Pocketing.1* again. This selection actually defines what is going to be copied and transformed. This will bring up the Copy-Transformation window.



There are a few basic options.

Number of copies Defines how many additional copies of the machining operation are to be created.

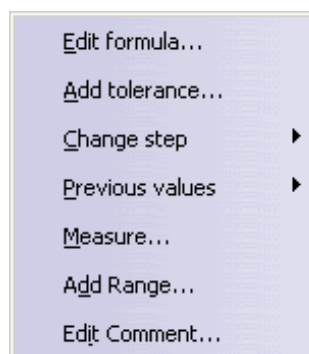
Transformation Defines what type of transformation is to take place. Currently, there are only two options: *Translation* and *Rotation*.

Distance along X, Y, Z Defines the transformation distances. The tool paths can be moved in all three directions at the same time if necessary. This option changes to *Axis* and *Angle* when a rotation is defined.

Change the number of copies to 3. Three additional copies are needed to machine all four pockets.

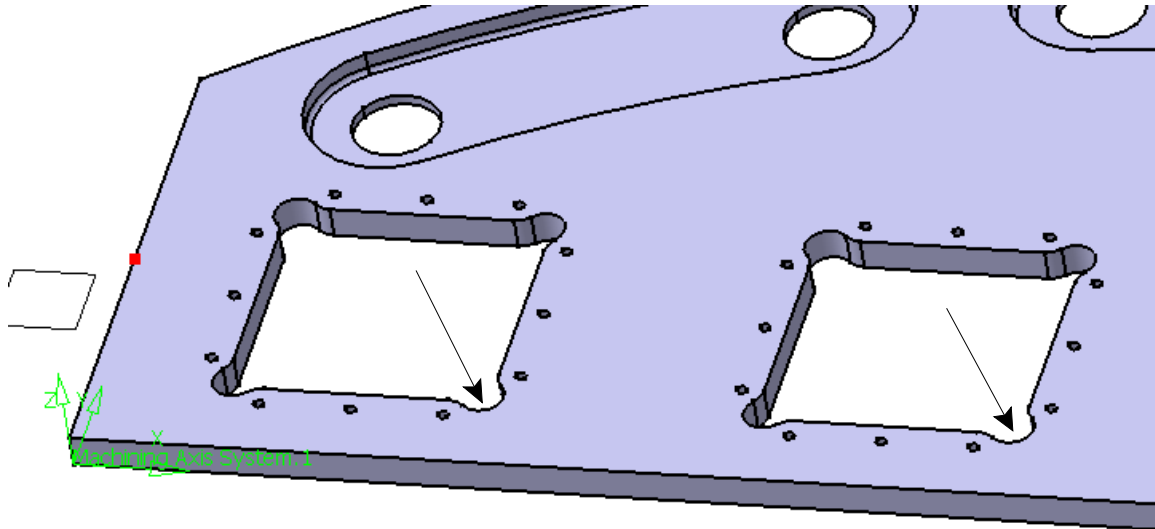
The distances are measured along the reference axis not the machining axis. The copy transformation is going to need to be translated along the X axis but at a distance that is unknown. This is not a problem.

Using the third mouse button select in the *Distance along X* field. This will bring up the contextual menu for the translations. If the right contextual menu does not appear the first time select button three again.



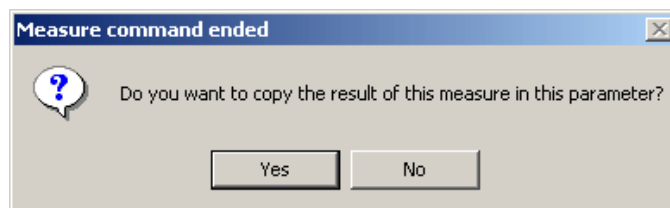
Select Measure. This will bring up the *Measure Between* window. This is going to allow for distances between measurements to be made then the results placed back into the distance field.

Change the Reference and Target mode to Arc Center. Select two corresponding arc centers to determine the distance between two pockets. Using the arc centers is not mandatory, any measurement between two elements that are the same in each will suffice.



The distance will be displayed. In this case the number is an even distance of 6" but sometimes this may not be the case.

Select Close in the Measure Between window. A *Measurement command ended* window will be displayed asking if the results of the measure between should be copied into this parameter.



Select Yes. Notice 6in was automatically inserted into the *Distance along X* field. All fields that require a distance to be inserted has the capabilities to allow the measurement of the part for information.

Make sure all other distance fields show 0in and select OK. Do not forget to replay the copy-transformation machining operation.

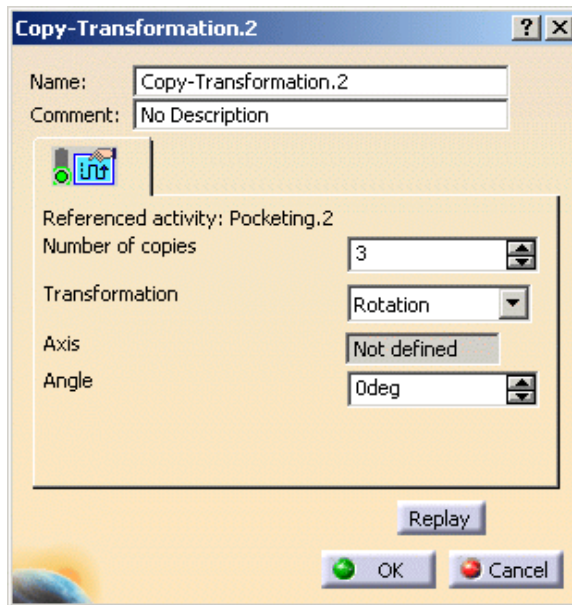
Notice when the copy-transformation is replayed, three tool path branches appear below it. These branches are the actual copies of the pocketing operation. The tool path is copied from the pocketing operation and then applied six inches along the X direction three times. The geometry does not necessarily need to be defined for a copy transformation operation to work.

Next a rotational copy transformation will be created.

Define a pocketing motion for the curved slot closest to the machining axis system. Do not forget to turn the -.020 bottom thickness off, as well as change the tool to have a .0625in corner radius. A tool diameter of .75" works good.

When done select the copy transformation icon and then select *Pocketing.2* twice. This will define a copy transformation after *Pocketing.2*.

Change the *Transformation* type to *Rotation*. Since the transformation for the three slots are in a circular fashion a rotation transformation is needed.



Select the *Axis* field. This field will make the *Copy-Transformation* window disappear.


Select the vertical line in space. This will define the rotation center for the copy transformation. The *Axis* field will change from *Not defined* to *Axis Defined*.

Set the *Angle* to 25deg. The slots are set to be 25° apart. This can be determined by checking the sketch.

Change the *Number of copies* to 2 and select *OK*. This will finish the copy transformations.

Next, the series of holes in the shallow pockets are going to be milled. Instead of using a pocket operation on the part, a circular milling operation will be used. Since this is an axial machining operation a copy transformation will not be applicable.

Circular Milling

Select the circular milling icon.  This can be found under the axial motions toolbar. Circular milling is a method to mill a hole in a part while running the tool in a circular, helical motion.

Define the circular milling motion after the Copy-Transformation.2 operation. Note the interface is similar to the other axial mode operations.

Define all six holes to machine. Change the *Extension* to *Through* and set the jump distance to .5in. Define the top and bottom of the hole with a -.020in *Offset on Bottom*. This will have the geometry defined. Next, the options for a circular milling operation need to be set.

Take a close look at some of the new options.

Machining mode :	Standard
Approach clearance (A) :	0.03937in
Distance between paths (Dp) :	0.25in
Number of paths (Np) :	3
Axial mode :	Number of levels
Number of levels (Nl) :	1
Sequencing mode :	Radial first
Machining tolerance :	0.00394in
Direction of cut :	Climb
Percentage overlap :	50
Automatic draft angle :	0deg
First compensation :	1
Output style :	Standard
Output CYCLE syntax :	<input type="checkbox"/>


Machining mode Defines what type of circular pattern milling is going to take place. *Standard* machining mode makes a circular pass, then lowers a level making another pass until the hole is machined. *Helical* spirals the tool downward along the outside of the hole until it reaches the bottom.

Distance between paths Controls the maximum distance between each circular pass in the hole.

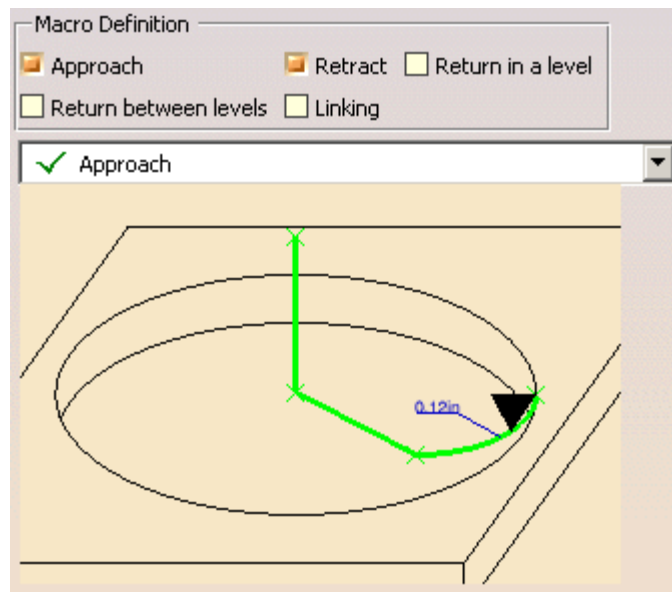
Number of paths Defines the maximum number of paths in a circular milling operation. Neither *Distance between paths* or *Number of paths* control the exact number of paths. Instead they work together to calculate the number of paths to create in the hole without violating the constraints.

Set the following machining parameters:

Machining mode :	Standard
Approach clearance (A) :	0.25in
Distance between paths (Dp) :	0.125in
Number of paths (Np) :	5
Axial mode :	Number of levels
Number of levels (Nl) :	3
Sequencing mode :	Radial first
Machining tolerance :	0.002in
Direction of cut :	Climb
Percentage overlap :	75
Automatic draft angle :	0deg
First compensation :	1
Output style :	Standard
Output CYCLE syntax :	<input type="checkbox"/>

Change the tool back to T1 End Mill D10. Do not create a new tool. This is unnecessary. Remember from earlier sections, select the three dots button () next to the tool name, then select T1 End Mill D10 (assuming the default tool name was kept).


Switch to the macros tab. Notice the Approach and Retract macros are already turned on. This is because they cannot be turned off. Each helical motion uses an approach and retract macro to approach into the part. For this, a helical approach and retract is used.



For each approach and retract there is a straight motion down, then a movement towards the side of the hole, then a helical motion downward. The only parameters that can be changed for the motions is the feedrate (through the contextual menu) and the radius of the helical motion.

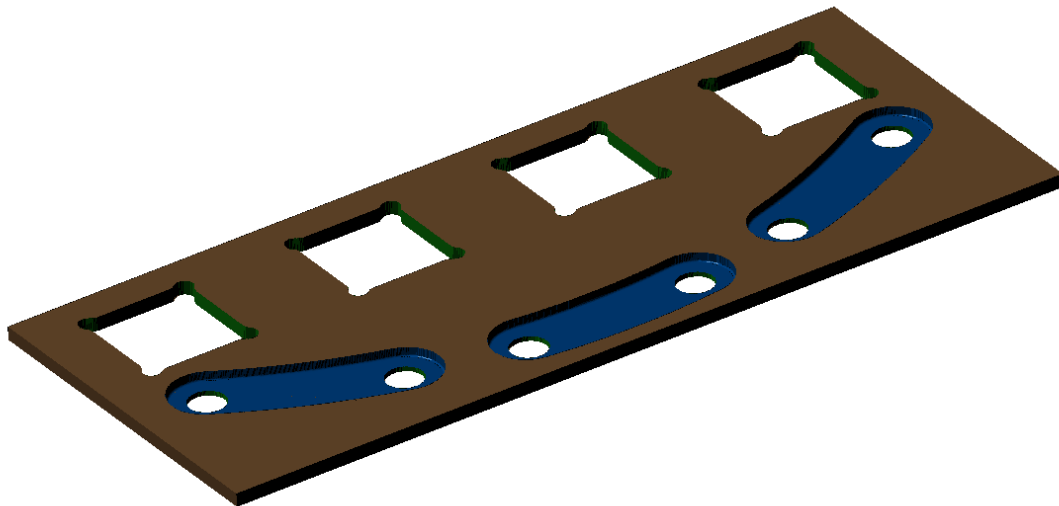
Change the radius of the helical motions for the approach and retract macros to be .125in. Notice there are *Global Approach* and *Global Retract* macros available. These will be defined too.

Switch to the *Global Approach* macro. Notice the toolbar icons below the sensitive macro area are now present. The global approach is similar to the standard approach macro for other operations.

Select the motion to plane icon.  This will make the motion move to the safety plane.

Define a motion to plane macro for the global retract. The steps to define the macro are the same as what was just performed.

Select *OK* when finished. This will finish the circular machining operations. The last thing to machine is the flanges, but for this a little knowledgeware will be used.



Manufacturing Knowledgware

As with many other workbenches in CATIA V5, knowledgware can be utilized with the manufacturing aspects. In this example a brief introduction to knowledgware applications to manufacturing will be investigated. There are two areas that need contour milling on the Instrument Panel part. The areas will be machined using a simple contour motion.

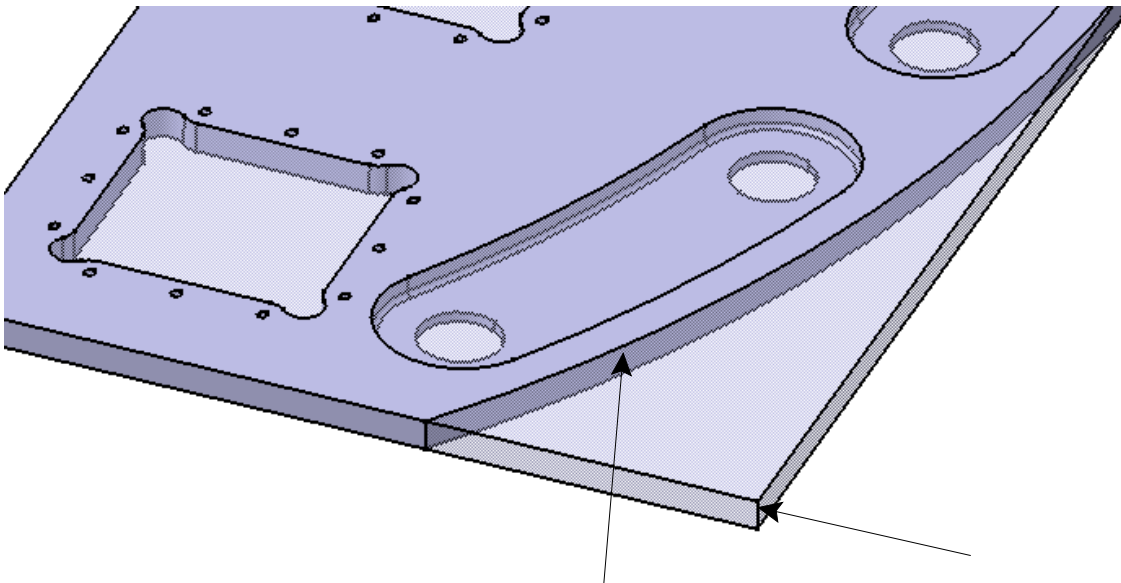
Knowledgware will then be applied to the number of paths to determine how many paths are needed given the distance between paths.

Open your Instrument Panel manufacturing process. The manufacturing exercise may already be open from the previous exercise.

Before going too far a measurement and some additional geometry should be created first.

Create a measurement between the outside corner and the circular instrument panel.

Select the measure between icon, then select the edge and the radius shown below. Make sure the *Keep Measure* option is turned on and close the window when done.



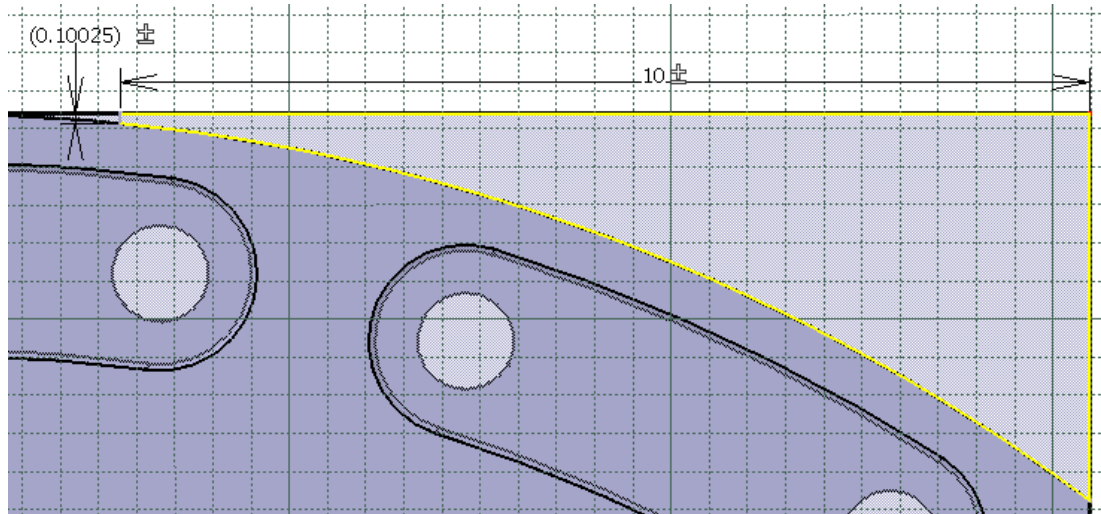
The order in which the two are selected is irrelevant.

Change the *Feature Name* of the distance to *Edge Distance*. To change the feature name, using the third mouse button select on the measurement then select *Properties*. From there, select *Feature Properties* and change the name.

Now that the distance has been created some additional geometry needs to be created. This problem poses an interesting problem for profile contouring. Since the part and the stock material becomes tangent at the top the profile operation has a hard time deciphering where the limits are. It is this reason that a sketch will need to be created that does not come tangent.

Double select on the *PartBody* for the *Stock*. It is always a good idea to keep any extra geometry necessary in the stock part and not in the finished part. Remember, the finished part will be used downstream by many other people and does not to be cluttered up with extra data.

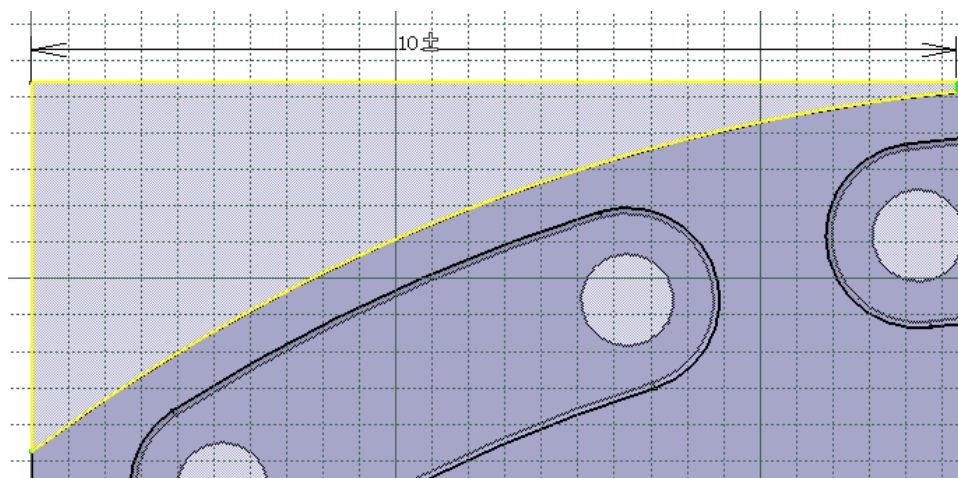
Create a sketch on the top of the stock material that is similar to the sketch below. The 10" constraint is not necessary, but it is always a good idea to have perfectly constrained sketches. The curve and two other lines work best as projections from the other geometry. The distance measurement was hidden for clarity. Be sure to add the referenced dimension on the short vertical piece.



Change the name of the referenced dimension to *Contour Thickness*. This will make the parameter easier to find when working with the formula editor. Also, keeping in mind of downstream users, it would be a good idea to label the parameter.

Using the third mouse button select on the referenced constraint and select *Contour Thickness object* then *Name/Value Display*. This will show both the name of the parameter as well as the distance associated with it.

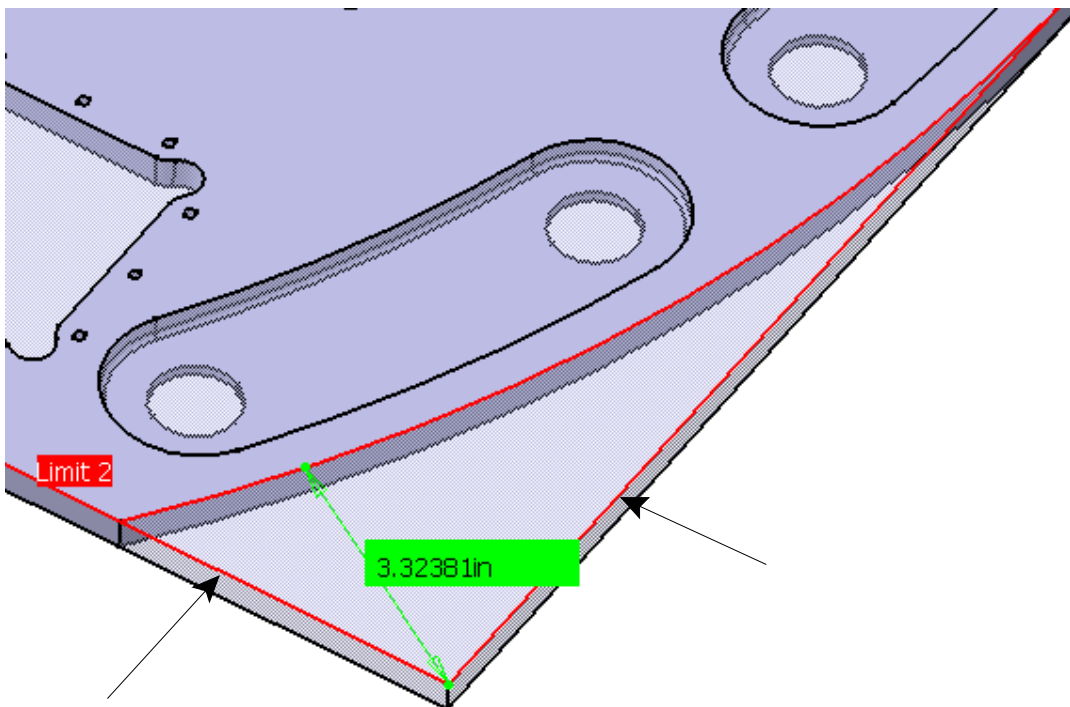
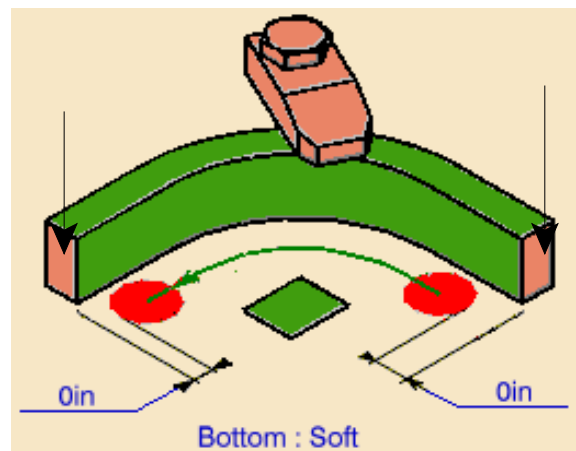
Create a similar sketch on the opposite side.



Start a profile contour operation after the *Circular Milling.1* operation. It will be through the machining parameters of the profile contour operation that the knowledgware will be applied.

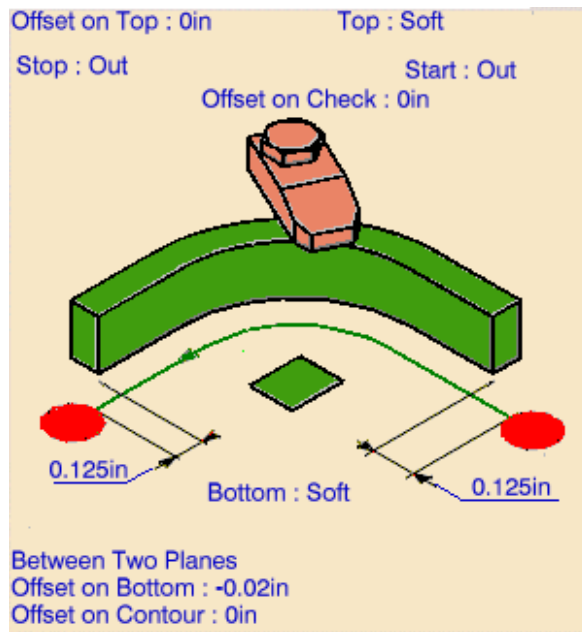
Change the *Bottom* to *Soft*. Define the bottom and top. Define the guiding element (the boundary curve) to be the edge of the instrument panel. Be sure to use the sketch edge and not the edge of the part. This will have most of the geometry defined. For this exercise the limits will also be defined, that way the whole top of the instrument panel is not machined only the triangular area.

Select one of the relimiting elements from the sensitive area then define the edge of the stock as one limit. Repeat this for the other relimiting element. All that you need to do is select the end of the sensitive area part as shown below then select the edge of the stock also shown below.



This will define the triangular area that is to be machined.

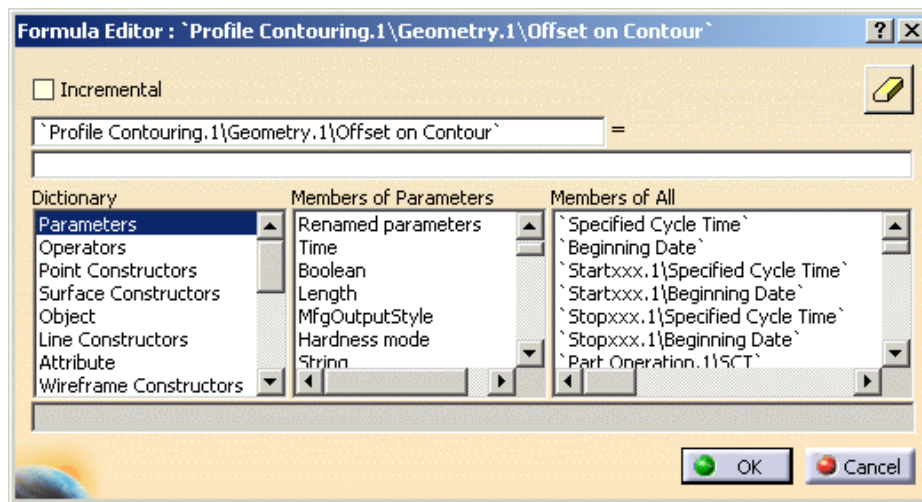
Set the *Start* and *Stop* positions to *Out* and then define a 0.125in edge distance at each end. Also define a -0.020in *Offset on the Bottom*. This will make the tool start and stop 1/8" from the relimiting edge. The offset on the bottom makes sure that most burrs are cut off.



Next the small contour thickness will be applied to the *Offset on Contour* distance.

Double select the *Offset on Contour* parameter. This will bring up the *Edit Parameter* window where a distance would typically be keyed.

Using the third mouse button select in the *Offset on Contour* field and select *Edit Formula*. This will bring up the *Formula Editor* window.

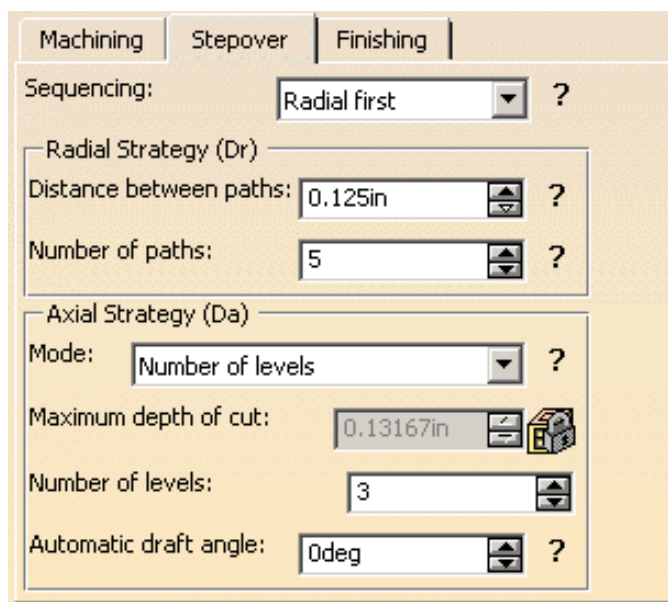


The difficulty with knowledgeware and formulas is deciphering what parameters are required. However, the difficulty in this is somewhat reduced when the elements or operations in question are selected.

Expand the *Stock* part in the specifications tree until the sketches created are available. Select *Sketch.2*. This is the sketch containing the referenced contour thickness dimension. Notice the number of parameters to choose from was greatly reduced.

Double select '*Stock\Partbody\Sketch.2\Contour Thickness\Length*'. Select *OK* when done. Select *OK* on the *Edit Parameter* window as well. This will make the offset thickness the same as the Contour offset thickness.

Switch to the manufacturing operation parameters tab and then the *Stepover* parameters. This will be the area where the knowledgeware is defined.



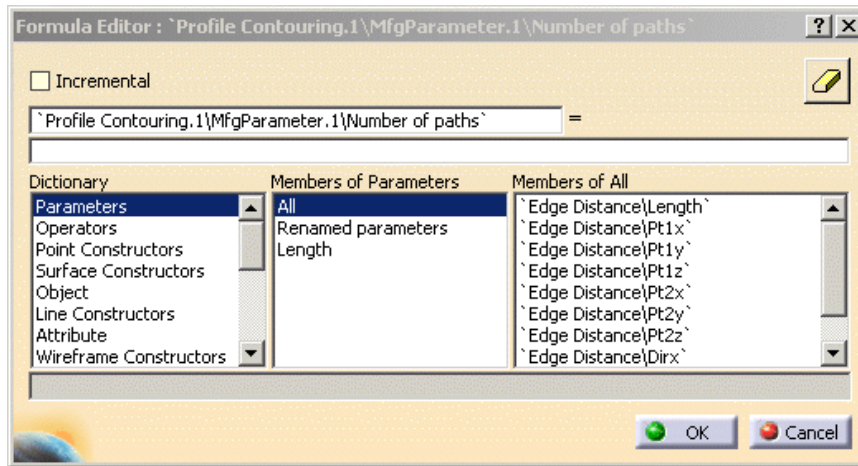
The radial strategy defines the number of passes that it takes to go from the outside edge to the final part. The problem is, only the number of paths and distance between paths can be defined. This means that there is no way to tell if the entire area will be milled.

Instead of calculating the distance and number of paths by hand, the number of paths will be calculated using knowledgeware depending on the distance between the paths.

Using the third mouse button select in the field for the *Number of paths*. This brings the contextual menu for the number of paths up.

Select *Edit Formula*. This will bring up the formula wizard. This contextual menu as well as the formula editor window is available for nearly all numerical parameters within CATIA V5.

Select the measurement created from the workspace. This will show only the parameters associated with the measurement. This method of determining formula parameters can prove quite useful when working with advanced models.



Double select ‘Edge Distance\Length’ from the Members of All window. Double selecting the parameter will put it into the equation window. Next, the distance between paths parameter is needed from the profile contouring operation.

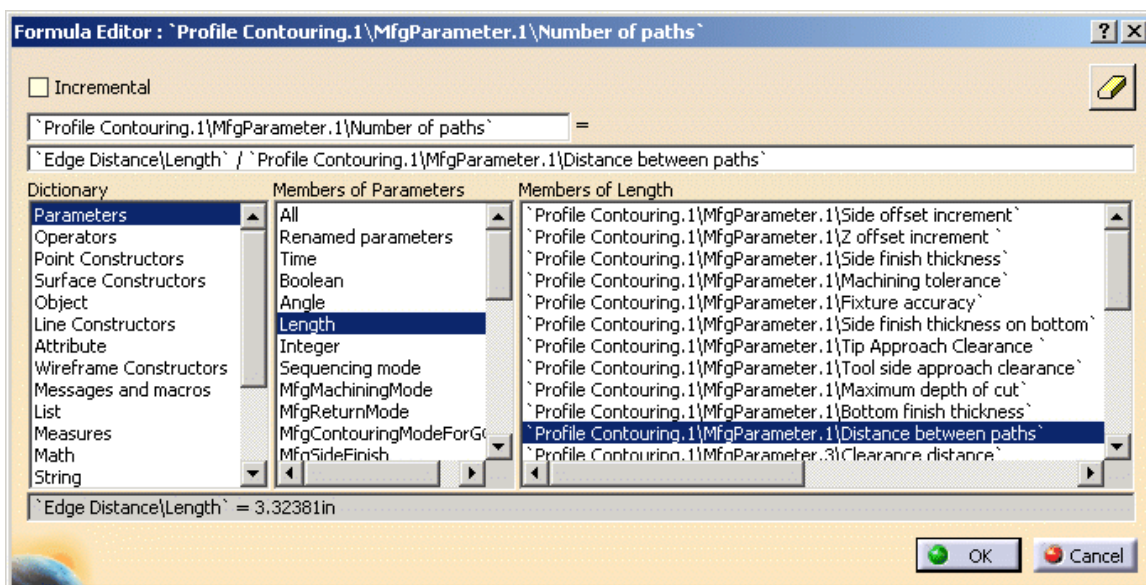
Key a forward slash (/) after ‘Edge Distance\Length’ in the equals (=) part of the formula. The forward slash is the divided by symbol.

Select the Profile Contouring.1 machining operation from the specifications tree. Again, the Members of All frame will modify to show all the available parameters for the profile contouring operation.

There are a lot of parameters associated with the profile contouring operation.

Select Length from the Members of Parameters frame. This will decrease the amount of information drastically.

Double select Distance between paths in the Members of Length area. The window may have to be scrolled to find it. The full name of the parameter is: ‘Profile Contoring.1\MfgParameter.1\Distance between paths’.



Select OK when done. After selecting *OK*, the *Profile Contouring* window will display again. Notice the number of paths has become greyed out with the formula symbol next to it. This denotes that the number can no longer be driven by hand and only by the formula.

Change the *Distance between paths* to .25in. Notice the number of paths calculates to 13.

Change the *Distance between paths* to .0625in. Now the number of paths re-calculates to 53. Granted, .0625" is a bit small but the point is made that the distance between the paths will control the number of paths.

Set the *Distance between paths* to .125in and the *Number of levels in Axial Strategy* to 3. Select OK when done. Do not forget to replay. Try going back and changing the distance between paths then replay again. Notice the number of paths are directly effected by the distance.

Start another profile contouring operation for the other side. Notice the Offset on Contour is already set. If the stepover values are checked the number of paths will already be defined also.

Define the geometry for the profile contouring operation on the other side. Since all of the formulas and constraints have been set it will not be necessary to define any other knowledgeware for the operation.

The last step to finish the part is to add one more profile contouring operation to make a few finish passes. This time the profile contour operation can pass all the way across the top of the part.

Start another profile contour operation after *Profile Contouring.2*. The knowledge applied to the other two contour operations will still be applied to this operation. It is imperative that this be turned off.

Define the guiding element to be the entire top arc and not parts of the sketches. Also define the top and bottom planes. Since the entire front of the part is to be machined, the limiting elements are not necessary.

Double select on the *Offset on Contour*. This will display the edit parameter window. Notice the area is greyed out but there is the formula icon next to it. To delete the formula, the formula editor must be used..

Select the formula icon next to the offset field. Delete the formula in the formula editor. Select OK when done. The formula will be the second field in the formula editor window. Clearing this out removes the formula from the parameter. After selecting *OK* the edit parameter window will display again.

Set the *Offset on Contour* to 0 in. This changes the offset to be net size. The same thing needs to be performed on the number of paths.

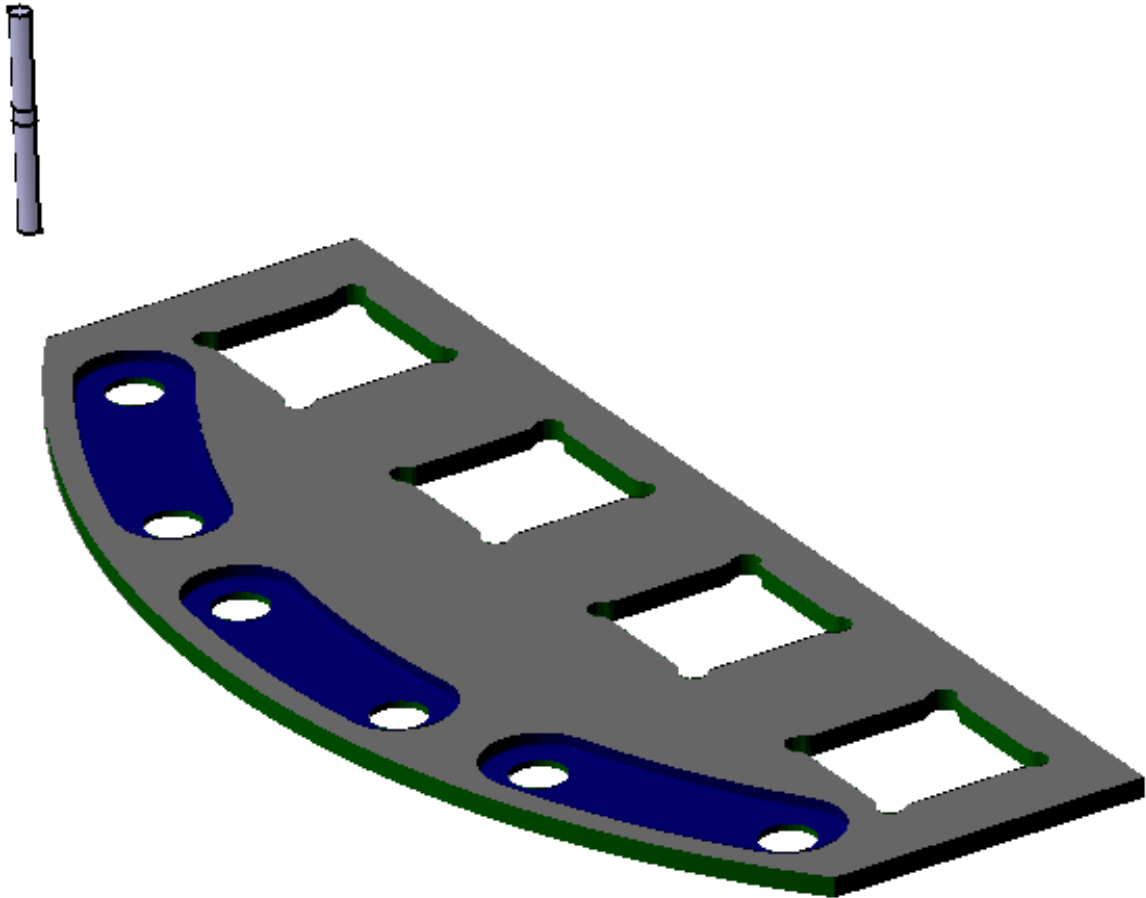
Delete the formula for the number of paths. The steps to do this are the same as before. Select the formula icon next to the number of paths and then delete the field for the formula.

Change the number of paths to 1. This one path will work as the finishing path for the operation.

Set the *Tool path style* to *One way* and define a return between levels macro that goes to the safety plane each time. The return macro will keep the tool from crashing with the part each time it returns for another pass.

Select *OK* and replay the program.

The last type of operations to create for the part are the drilling operations.



Post Processor Instructions

Post processor instructions are the machine code that gets directly interpreted by the post processor (the translator that goes from APT code to the machine specific language). One must exercise caution when inserting post processor words. By defining a specific syntax for a particular machine the program is then tied to that machine. This means that apt cannot be made and processed for another machine. If the program is to run on a separate machine then the post processor instruction will have to be changed. However, there are a number of post processor commands that are common throughout the different machines and can be used as necessary.

There are two methods for inserting Post-Processor instructions into the manufacturing process. One method is to insert the post processor instructions in the macros when defining the machining operation. The second method is to insert a post processor instruction into the manufacturing program. Both options will be investigated.

Start a drilling operation after Profile Contouring.3. Define the user pattern as the points to drill. When defining the points, the pattern selection will display, expand the *Instrument Panel.1* branch and select the user pattern from there.

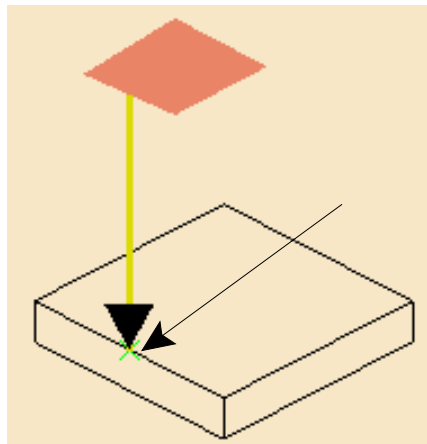
Define the top and the bottom of the holes. Do not define a jump distance. Instead, a linking macro will be used.

Define an *Approach clearance* of .125in and a *Breakthrough* value of .125in. This will have the machining parameters defined.


Define a drill bit as well as the feedrates and spindle speeds. This will have everything but the macros defined.

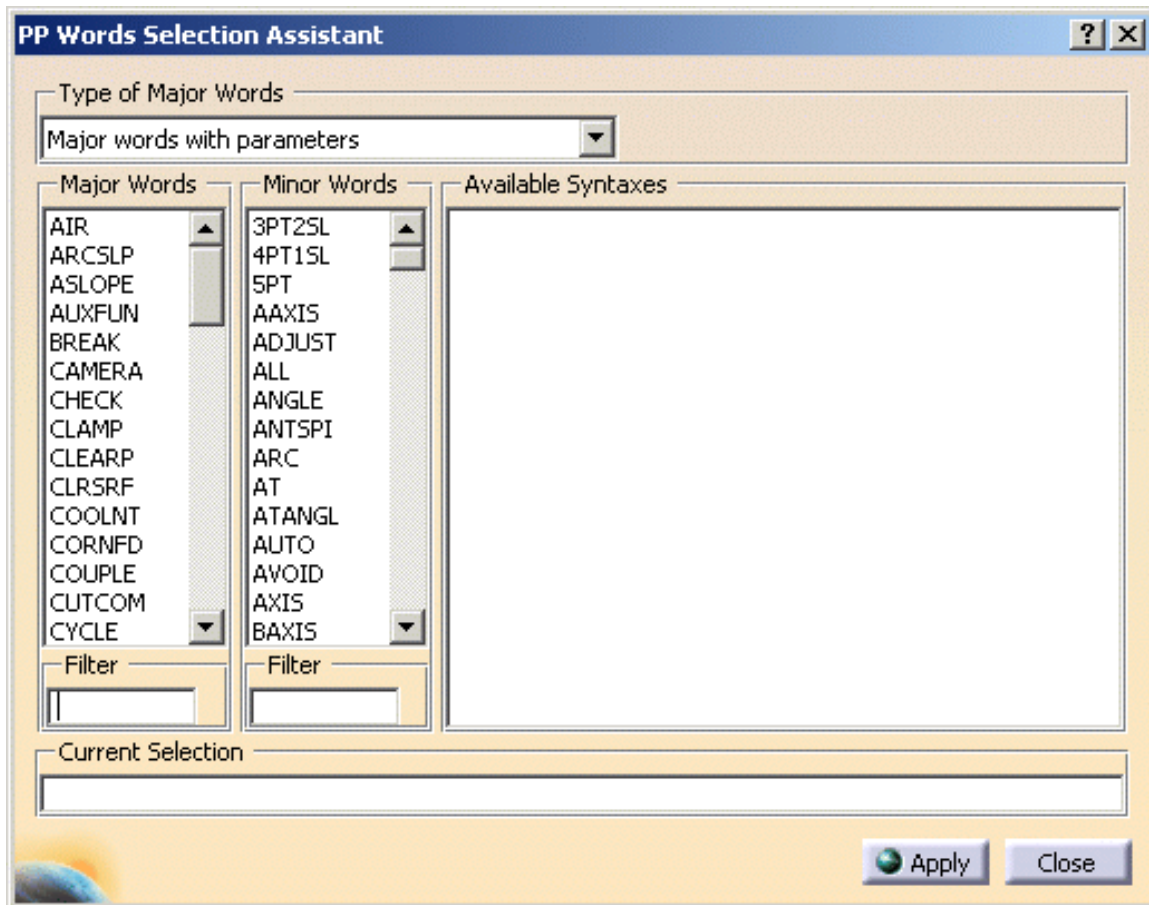
Turn all macros off. By default, there should not be any macros turned on but if this is not the case make sure all the macros are off.

Turn the *Approach* macro on and select the motion to a plane. There is a feature in the sensitive area that has not been covered yet. There is an “X” at the bottom of the approach motion. This is a point in which post processor words can be defined.



Using the third mouse button select on the point and then select *PPword list*. This will display the *PP Words Selection* window. This window allows post processor (PP) words to be defined. The words can be either keyed in or selected from a list.

Select the **PP Table access icon in the *PP Words Selection* window.**  This will display the *PP Words Selection Assistant* window. All available PP words are displayed in the window. The *PP Words Selection Assistant* helps define the PP words and parameters necessary, that way extra words or parameters are not forgotten.



Select ***COOLNT*** from the *Major Words* frame. As soon as the *COOLNT* command is selected a list of available syntaxes display.

Select ***COOLNT/ON*** from the *Available Syntaxes* frame. Notice the command appears in the current selection area at the bottom of the window.

Select ***Apply*** and then ***Close***. Select ***OK*** in the *PP Words Selection* window when done. Apply has to be selected before the coolant command is placed in the PP Words Selection area.


Notice the PP commands now appear in the sensitive area. Note the PP words icon available in the list of icons available for the macro definition. Adding PP words here inserts the PP command before the tool reaches the bottom.

Define a retract motion to a plane. Again, an “X” will appear at the bottom of the retract motion. This will be where the coolant is turned off.

Define a *COOLNT/OFF* command at the bottom of the point. This will turn the coolant off before the tool raises into the air. The steps to perform this operation are the same as before. Just use the third mouse button to select on the point, select PP words, use the PP word assistant and select the coolant/off command.

A linking macro will also be needed to go between the points.

Switch to the linking approach macro. If a motion exists, remove it. Select the add

axial motion icon.  This will move the tool along the axis a given distance. Again, notice the “X” at the bottom of the motion.

Set the distance to .5in. Add a *COOLNT/ON* command at the point. This will work practically the same as the approach macro.

Define a .5in axial motion with a *COOLNT/OFF* command for the linking retract macro. This will have all the macros and additional PP words applied to the drilling operation. When the replay is viewed there will be no notable difference with the PP words added.

Select *OK* to the drilling operation when done. This will finish machining the part. One more PP command will be added to the program at the end.

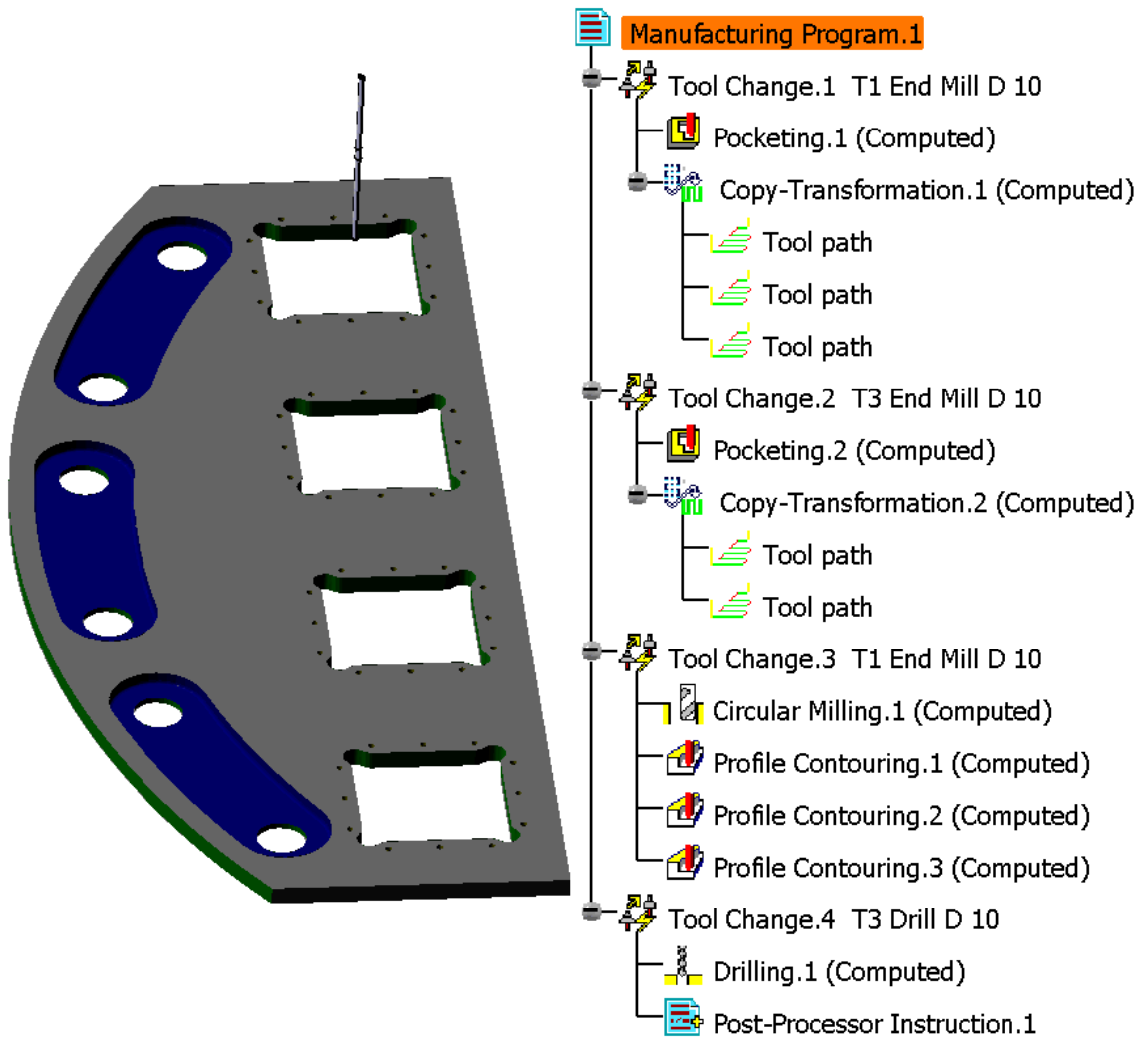
Select the post-processor instruction icon.  This icon will add a Post Processor command to the manufacturing program.

Select *Drilling.1*. This will define where the command will insert after. A similar window will show for the Post Processor instruction as was shown when defining PP words with the macros.

Key the PP command *PPRINT YOUR NAME* in the *PP Instruction* field. The word *PPRINT* can be found using the PP word assistant if you wish. *PPRINT* statements are used to define comments or additional text in the program.

Select *OK* when done. This adds the *PPRINT* comment command to the manufacturing program.

Feel free to define other PP instructions or commands throughout the program.



There are two more jobs to perform on this part. One is to create documentation which will happen next. The second will be to create the apt code and post processed code for the part.

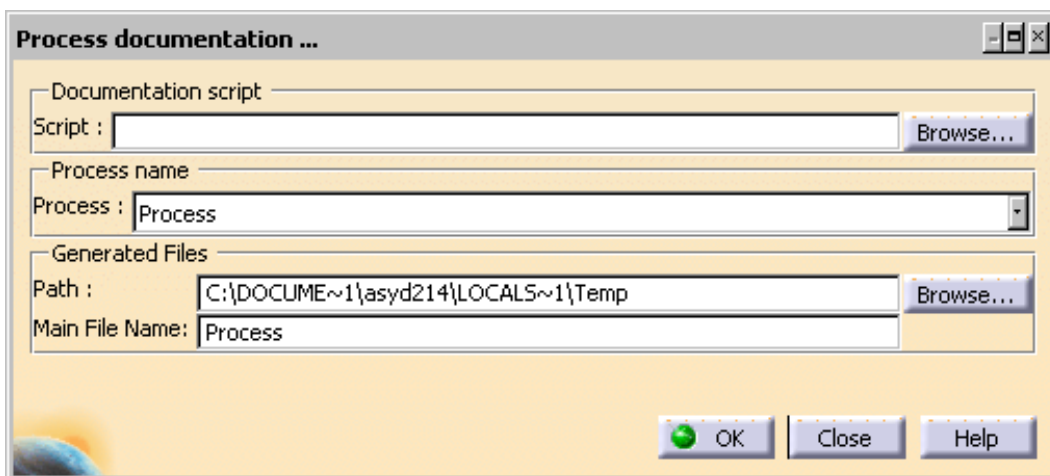
NC Documentation

NC Documentation is an important part to any manufacturing program. The machinist at the machine does not have a computer with CATIA V5 on it to view the manufacturing program. The NC Documentation is the only method that is available to relay the information

To create NC Documentation a CATScript has to be ran. This CATScript analyzes the manufacturing program, then generates HTML documentation accordingly. Also, there will be an image of the workspace placed on the document. This means that anything showing in the workspace window will be shown in the document. It is advisable to unhide all the necessary geometry (fixtures, stock material, machine tables, etc.) and hide all the extra geometry (construction elements, extra planes, etc.).

Prepare the workspace for documentation. Hide all unnecessary geometry, unhide all necessary parts. Position the part so that it is easy to view, determine the machining axis location, position, etc.

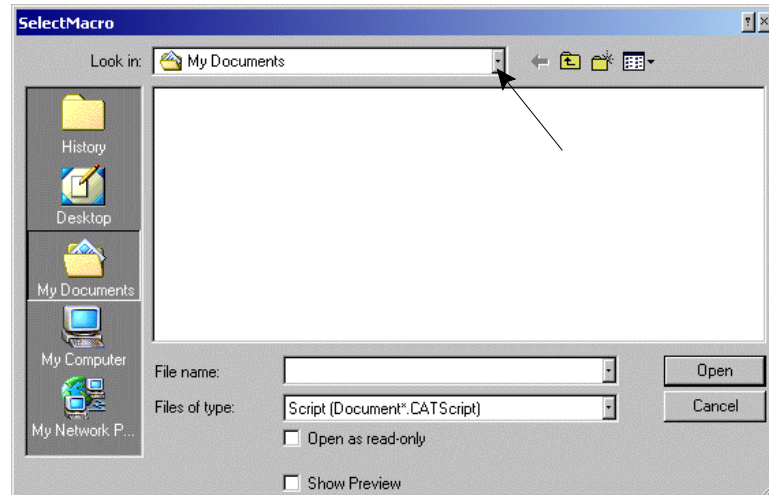
Select the generate documentation icon.  This icon will display the *Process documentation* window.



There are several options that have to be set in this window.

Select the *Browse* button next to the *Script* field. This will be used to find the CATScript that is ran to generate the documentation. The CATScript is in the Prismatic Machining folder on the Z drive.

Select the drop down as shown below. This drop down will allow the available drives to be displayed.

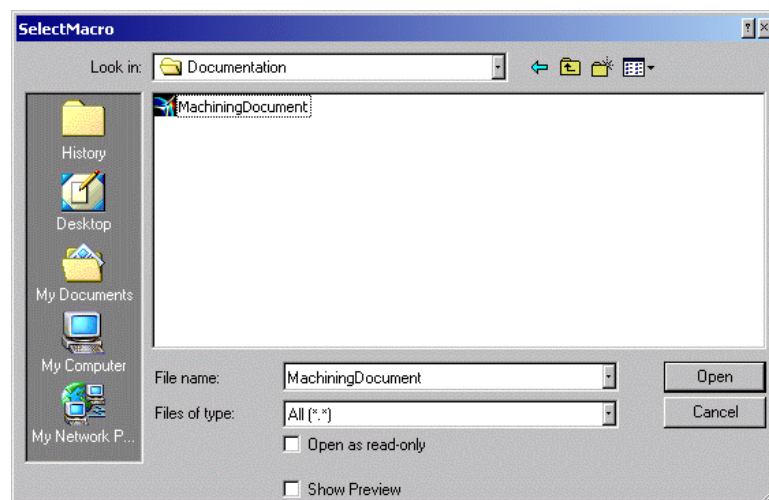


Select the last drive. This will be labeled as *{your username}* on 'nemesis' (Z:). All of the files contained within the drive will be shown.

Double select on *drawings-v5* and then double select on *Prismatic Machining*. Notice when you double select on the *drawings-v5* folder, all the CAD/CAM CATIA V5 class models are available as well as your *Save Your Models Here* folder. This is where all of the models for this class are stored. Within this folder there is another folder called *Documentation*. This is the folder containing the CATScript that is needed.

Double select on *Documentation*. Notice no files are displayed. This is not a problem. There is a filter set on the type of files that CATIA is searching for and the filter just needs to be adjusted.

Select the drop down for *Files of type* and select *All (*.*)*. Now the *MachiningDocument* CATScript file will be visible.




Select on the Machining Document and then select *Open*. This will place the full path of the documentation into the *Script* area.

The *Process name* field is where the name for the process will be placed, however in the current versions of CATIA the process name cannot be changed so it will keep the name of *Process* for now.

The *Generated Files* area defines where the HTML documentation is going to be stored. This path will need to be set to your *Save Your Models Here* folder.

Select *Browse* next to the *Path* field. Navigate your way to your *Save Your Models Here* folder. The steps to navigate to the folder is the same process as before, select the drop down, select the Z drive, drawings-v5, then save your models here.

Select the new folder icon. Name the folder *Documentation*.  This will create a new folder. Key *Documentation* immediately after selecting the icon to change its name.

Double select the *Documentation* folder, then change the file name to be *Instrument Panel*. Select *Save when done*. This defines the folder and filename for the documentation. The *Process documentation* window displays again showing the pieces of information in the appropriate fields.

Select *OK* when done. A *Documenting the process* window will display showing the steps that the script is going through.

Once the script is done, a window will display asking if you want to view the documentation now. Unfortunately, this does not work and the documentation will have to be opened manually.

Select *No* to the VB Script question. Select *OK* in the *Documenting the process* window and close the *Process documentation* window. Now it is time to view the documentation created.

Double select on the My Computer icon in the top left hand side of the desktop. This will show the Windows Explorer.

Double select on {username} on '*nemesis*' (Z:), then on *drawings-v5*, *Save Your Models Here*, *Documentation*, and finally *Instrument Panel*. There will be a few files and a folder in this directory. There will be one file associated with a web browser named *Instrument Panel*. This is the master HTML file that was defined and that will be ran to view the NC Documentation.

Double select on *Instrument Panel*. This will display the NC Documentation.

The information displayed here as well as in the rest of the document is customizable through the CATScript. Many companies will choose to create their own specialized CATScript to generate documentation in a particular method. Continue exploring through the document. Towards the bottom of the first page is a link (blue text) that goes to Part Operation.1.

Select the Part Operation link at the bottom of the main page.

SHOP FLOOR DOCUMENTATION Process1 bb.CATProcess

Part operation : Part Operation.1

Generation : 10/19/2001 at 6:11:58 PM

Part name : Instrument Panel Assembly

Machine name : [3-axis Default machine](#)

Machining axis system : Machining Axis System.1

[Tool List](#)

Programs :

- Program : [Manufacturing Program.1](#)

Comment :

No Description

Notice the information displayed here is information that was specified in the part operation. If comments were added to the part operation or any other machining operations they would also be displayed. It is for this reason that it is a good idea to comment and document anything special about the program or machining operation.

It is left to you to continue exploring the documentation. When finished close all web browser windows before continuing.


This finishes creating documentation. The next thing to do is to create the APT code.

NC Code

Generation of apt code is essential to get the CATIA manufacturing program to the machine. There will be two topics covered here. One will be to investigate creating APT code. The second will be to run the code through a post processor built into CATIA V5. The difference between APT code and post processed code is that the APT code is general. It can be processed for most machines by feeding it through a post processor. However, CATIA V5 has the capabilities to convert the manufacturing programs directly into the post processed form, skipping the step of creating APT code.

APT Code Generation


There are two methods for generating APT code. One method is to generate the apt interactively, or on-the-fly. This type of generation of APT does not require the model to be saved, and can take place at any time. The second method of generating APT code is to generate it in a batch mode. With batch mode generation, there are a lot more options to control the code generation, but there are a few more steps, as well as the document having to be saved.

With your Instrument Panel machining process open, select *Manufacturing Program.1* and select the generate NC code interactively icon.  This icon can be found under the generate NC code in batch mode icon. The save NC file window will display asking where the APT code should be saved. If the manufacturing program is not selected to begin with then the APT generation will not know what program to generate APT for.

Change the directory to your *Save Your Models Here* folder. In order to keep from cluttering this folder up too bad a new directory for your code will be created.

Create a new folder called Code. Go into the folder when done. Define the name of the file to be Instrument Panel and select *Save* when done. This will create the apt source file as well as a log file into the *Code* directory. It is suggested that if a manufacturing process has multiple manufacturing programs a separate directory within the *Code* directory should be created.

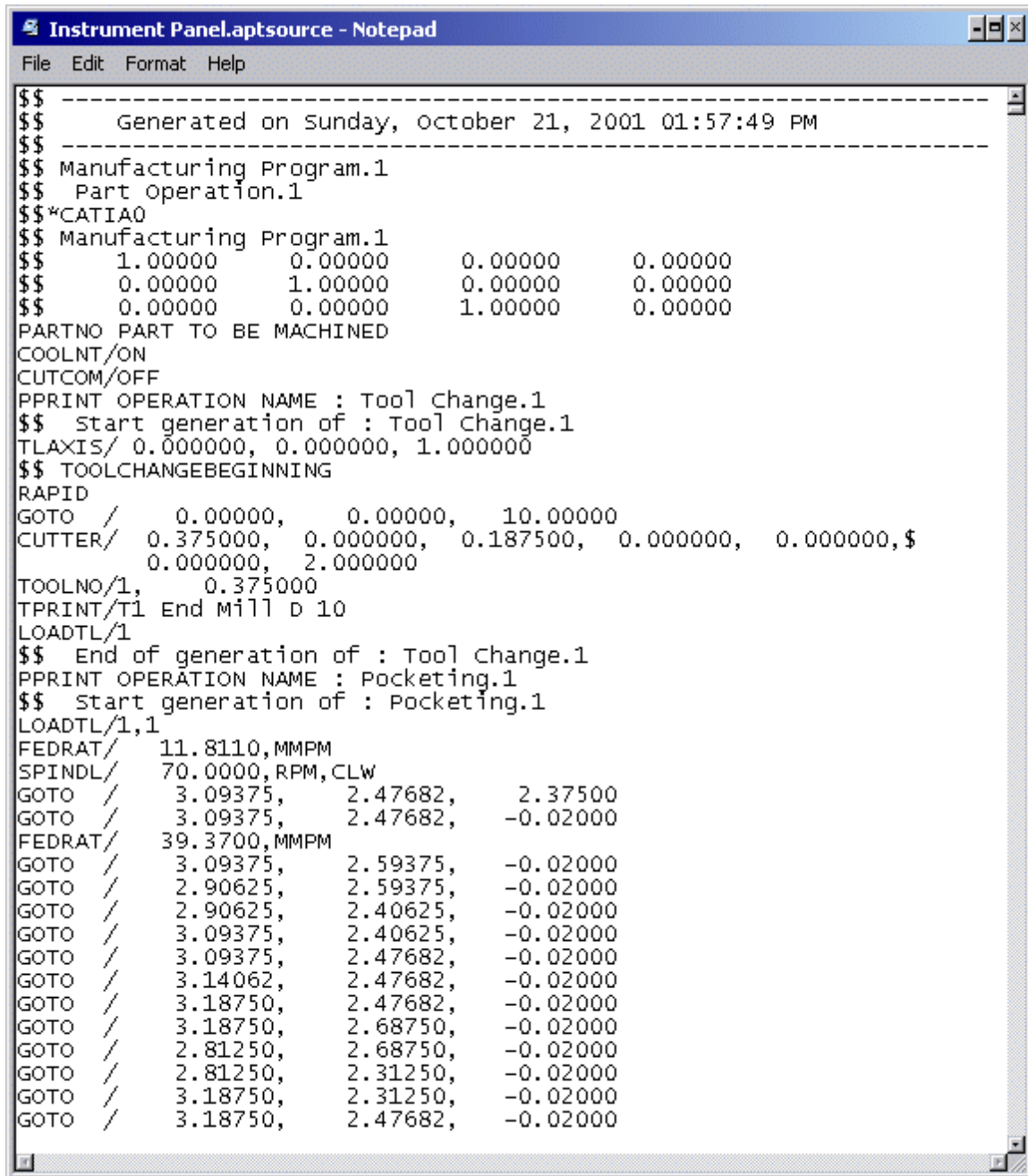
Now that the APT code has been generated, it is time to view the code. On the desktop there is an icon for Notepad, a generic, simple text editor. Notepad will be sufficient to view the APT code.

Double select the Notepad icon.  This will launch Notepad. Now the document will be opened and viewed.

Select pull down menu *File, Open*. Switch to the *Save Your Models Here* folder, then to the *Code* directory. *File Open* will bring up the standard *Open* window. Notice there appears to be no files in the *Code* folder. This is because notepad is only looking for text (.txt) files and not for APT source (.aptsource) files.

Change the Files of type to All Files. Two files will be displayed. One file will be the APT source file, the other will be the APT source log file. The log file will contain all of the processing that it took to generate the file. The log file is usually only necessary to determine what caused particular errors in APT source generation.

Select on the Instrument Panel.aptsource file and then select *Open*. This will open the apt source file.



```

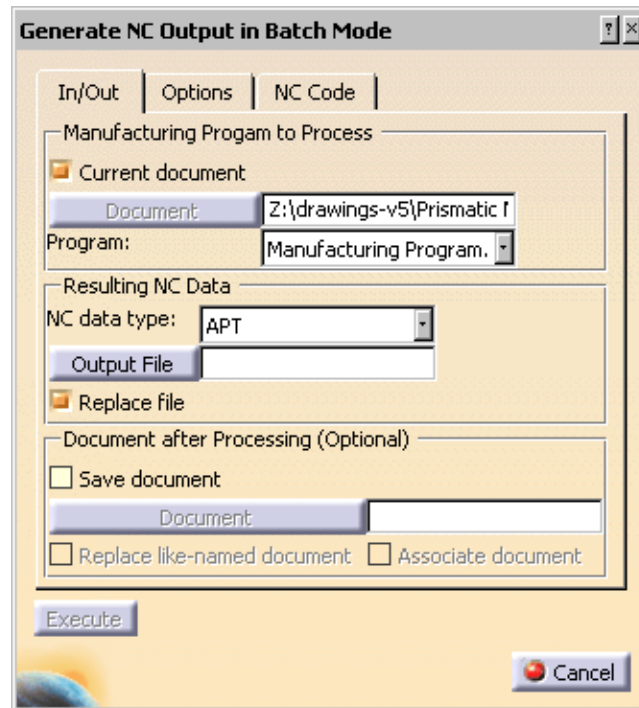
Instrument Panel.aptsource - Notepad
File Edit Format Help
-----
Generated on Sunday, October 21, 2001 01:57:49 PM
-----
Manufacturing Program.1
Part Operation.1
**CATIA0
Manufacturing Program.1
1.00000 0.00000 0.00000 0.00000
0.00000 1.00000 0.00000 0.00000
0.00000 0.00000 1.00000 0.00000
PARTNO PART TO BE MACHINED
COOLNT/ON
CUTCOM/OFF
PPRINT OPERATION NAME : Tool Change.1
Start generation of : Tool Change.1
TLAXIS/ 0.000000, 0.000000, 1.000000
TOOLCHANGEBEGINNING
RAPID
GOTO / 0.00000, 0.00000, 10.00000
CUTTER/ 0.375000, 0.000000, 0.187500, 0.000000, 0.000000,$
0.000000, 2.000000
TOOLNO/1, 0.375000
TPRINT/T1 End Mill D 10
LOADTL/1
End of generation of : Tool Change.1
PPRINT OPERATION NAME : Pocketing.1
Start generation of : Pocketing.1
LOADTL/1,1
FEDRAT/ 11.8110,MMPM
SPINDL/ 70.0000,RPM,CLW
GOTO / 3.09375, 2.47682, 2.37500
GOTO / 3.09375, 2.47682, -0.02000
FEDRAT/ 39.3700,MMPM
GOTO / 3.09375, 2.59375, -0.02000
GOTO / 2.90625, 2.59375, -0.02000
GOTO / 2.90625, 2.40625, -0.02000
GOTO / 3.09375, 2.40625, -0.02000
GOTO / 3.09375, 2.47682, -0.02000
GOTO / 3.14062, 2.47682, -0.02000
GOTO / 3.18750, 2.47682, -0.02000
GOTO / 3.18750, 2.68750, -0.02000
GOTO / 2.81250, 2.68750, -0.02000
GOTO / 2.81250, 2.31250, -0.02000
GOTO / 3.18750, 2.31250, -0.02000
GOTO / 3.18750, 2.47682, -0.02000

```

Feel free to look through the APT file as well as the log file. When done, close Notepad.

Save the manufacturing process. The next step would be to generate the APT source through batch mode. In order for batch mode to work the document has to be saved first.

Select the generate NC code in batch mode icon.  This will display the *Generate NC Output in Batch Mode* window.



There are several fields and options in this window. Take a closer look at them.

Manufacturing Program to Process Defines what document will be processed. Either the current document can be processed or a different document that is saved.

Resulting NC Data Defines what will be outputted in the generation process as well as where the output will go. There are four types of NC Data that can be created.

APT Code used for a variety of machines needing post processed before reaching the machine.

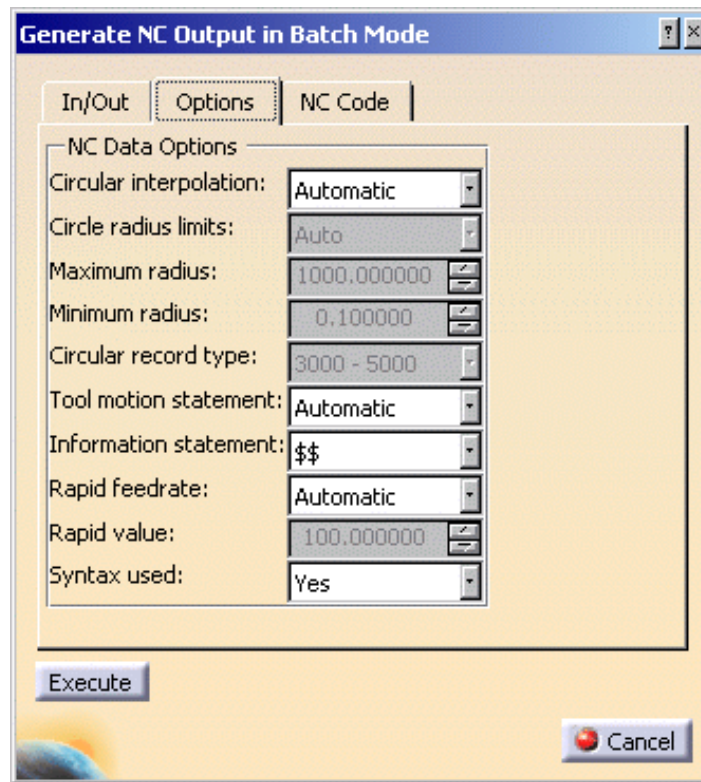
CLF Cutter location file that contains cutter coordinates. This type of code is similar to APT, however is used less.

NC Code Post Processed code if a post processor and controller has been defined.

In Process Model Creates a CGR file of what the model looks like after machining the current manufacturing program. Useful for multiple position parts. The CGR file can be used for the stock material of another Part Operation.

Document after Processing Allows the document to be saved immediately after it is processed.

The *Options* tab contains a lot of customizable information about the output of the code.



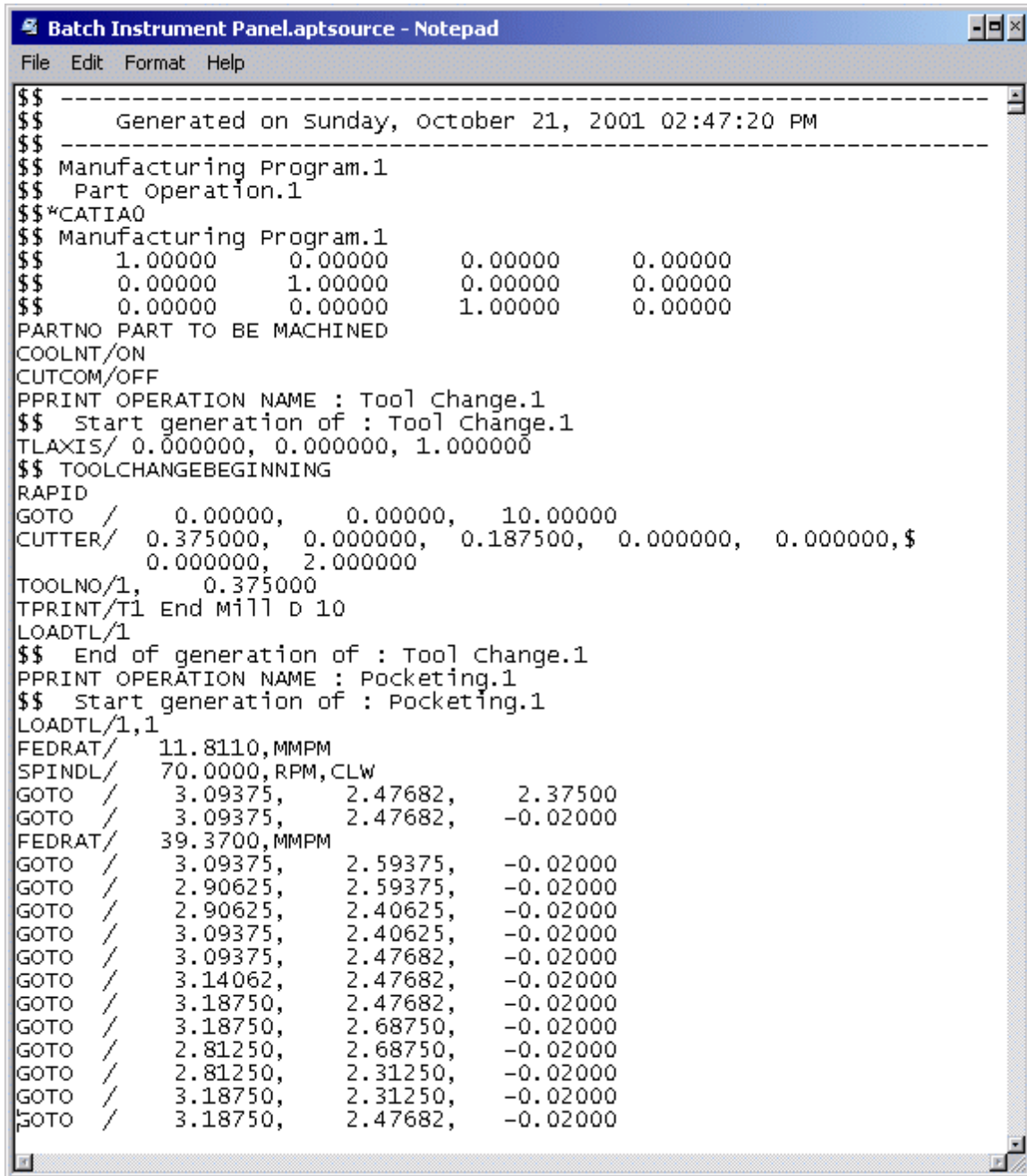
<i>Circular interpolation</i>	Defines how circles are looked at.
<i>Circle radius limits</i>	If the <i>Circular interpolation</i> is not set to automatic, then radius limits need set.
<i>Minimum and Maximum radius</i>	Defines the minimum and maximum radius limits if <i>Circle radius limits</i> is not set to automatic.
<i>Circular record type</i>	Defines the type of record for circles when a CLF file is requested.
<i>Tool motion statement</i>	Describes the format of the tool motions in the output.
<i>Information statement</i>	Defines what characters represent information or comment statements.
<i>Rapid feedrate</i>	Defines the rapid feedrate, hard coded or machine default.
<i>Rapid value</i>	Defines the rapid feedrate value if hard coded.
<i>Syntax used</i>	Defines if the default syntax should be used in the program.

The NC Code tab will be investigated later.

Make sure the *NC data type* is set to *APT* and the output file is the *Code* folder in the *Save Your Models Here* folder with a file name of **Batch Instrument Panel**. Calling the code Batch Instrument Panel is just so the two sets of APT Code do not get overwritten.

Select **Execute** at the bottom of the window. A command line window will display showing the statement that is being executed.

When the command window closes select **Cancel** in the *Generate NC Output in Batch Mode* window. Open the code in Notepad. The code will look exactly the same as the code generated interactively.



```

Batch Instrument Panel.aptsource - Notepad
File Edit Format Help
-----
Generated on Sunday, October 21, 2001 02:47:20 PM
-----
Manufacturing Program.1
Part Operation.1
**CATIA0
Manufacturing Program.1
1.00000 0.00000 0.00000 0.00000
0.00000 1.00000 0.00000 0.00000
0.00000 0.00000 1.00000 0.00000
PARTNO PART TO BE MACHINED
COOLNT/ON
CUTCOM/OFF
PPRINT OPERATION NAME : Tool Change.1
Start generation of : Tool Change.1
TLAXIS/ 0.000000, 0.000000, 1.000000
TOOLCHANGEBEGINNING
RAPID
GOTO / 0.00000, 0.00000, 10.00000
CUTTER/ 0.375000, 0.000000, 0.187500, 0.000000, 0.000000,$
0.000000, 2.000000
TOOLNO/1, 0.375000
TPRINT/T1 End Mill D 10
LOADTL/1
End of generation of : Tool Change.1
PPRINT OPERATION NAME : Pocketing.1
Start generation of : Pocketing.1
LOADTL/1,1
FEDRAT/ 11.8110,MPPM
SPINDL/ 70.0000,RPM,CLW
GOTO / 3.09375, 2.47682, 2.37500
GOTO / 3.09375, 2.47682, -0.02000
FEDRAT/ 39.3700,MPPM
GOTO / 3.09375, 2.59375, -0.02000
GOTO / 2.90625, 2.59375, -0.02000
GOTO / 2.90625, 2.40625, -0.02000
GOTO / 3.09375, 2.40625, -0.02000
GOTO / 3.09375, 2.47682, -0.02000
GOTO / 3.14062, 2.47682, -0.02000
GOTO / 3.18750, 2.47682, -0.02000
GOTO / 3.18750, 2.68750, -0.02000
GOTO / 2.81250, 2.68750, -0.02000
GOTO / 2.81250, 2.31250, -0.02000
GOTO / 3.18750, 2.31250, -0.02000
GOTO / 3.18750, 2.47682, -0.02000

```

The next step will be to create post processed code.

Generating Post Processed Code

Post Processed code is code that can be sent directly to the CNC Machine. This code does not have to go through another translation before the machine can read and understand it.

Before a manufacturing program can go through post processing, the post processor has to be defined.

Select the pull down menu *Tools, Options, NC Manufacturing, Output tab*. This will customize the output options for manufacturing.

Change the *Post Processor to Cenit* and select *OK*. The IMS Post Processor requires a license not included with CATIA. The IMS Post Processor will generate a partial post file that is not useful for much.

Select the generate NC code in batch mode icon. This time NC Code will be generated sending the file through the Cenit Post Processor.

Change the *NC data type* to be *NC Code* , and set the output file to be *Instrument Panel Code* in the *Code* directory. This will define where and what the name of the code will be.

Change to the NC Code tab. Now there will be an option available. There is a *Cenit Post-processor file* drop down that will allow the selection of a post processor. There are a number of post processors available. The post processor corresponds to the type of controller the machine has.

Select the *OKUMA_MX45VA_3X* post processor. Selecting the question mark next to the drop down field will display information about the specified post processor.

Select *Execute when done*. Another command window will display while the code is generated.

Open the *Instrument Panel Code.CATNCCode* file in Notepad. When opening the file there will be another file with the same name but different extension. The other file will be the log file associated with the NCCode file.

```

(=====)
(MODEL :N NAME : Tool Change.1)
(PART OP :)
(MACHINE :)
 (DATE :Sun Oct 21 15:21:58 2001
)
(=====)
VC1=-9999 (X SAFE POSITION)
VC2=9999 (Z SAFE POSITION)
G90 G17 G40 G21 G95
G40
G0 X0 Y0 Z10 M08
M09 M05
(===== TOOL CHANGE =====)
(TLDESC :)
NT1
G00 Z=VC2 ( Z SAFE POSITION )
G00 X=VC1 ( X SAFE POSITION )
TLFON
(CALL OCHEK)
T1
TLCO
M6
G90 G17 G21 G40 G95
T3
S70 M03
M08
G1 X3.094 Y2.477 Z2.375 G56 H1. F11.811
Z-0.02
Y2.594 F39.37
X2.906
Y2.406
X3.094
Y2.477
X3.188
Y2.688
X2.813
Y2.313
X3.188
Y2.477
X3.375
Y2.875
X2.625
Y2.125
X3.375

```

This is the actual code that the machine reads. This code is now ready to be sent to a machine using the controller specified in the post processor.

Select *Cancel* in the *Generate NC Output in Batch Mode* window and save your process.

There are two workbench options that were not covered in this course.



Macro Output to VNC

VNC (Virtual NC) is a machine simulation/verification tool made by Delmia, a subsidiary of Dassault Systèmes. VNC is a separate package that has to be purchased from Delmia and is not yet fully integrated into CATIA V5.



Replace Tools

This option allows tool representations to be changed. This option is only useful when a tool database is used that has an alternate tool representation defined.