TABLE OF CONTENTS

Introduction

CATIA Version 5 Prismatic Machining

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

Part Operation Setup

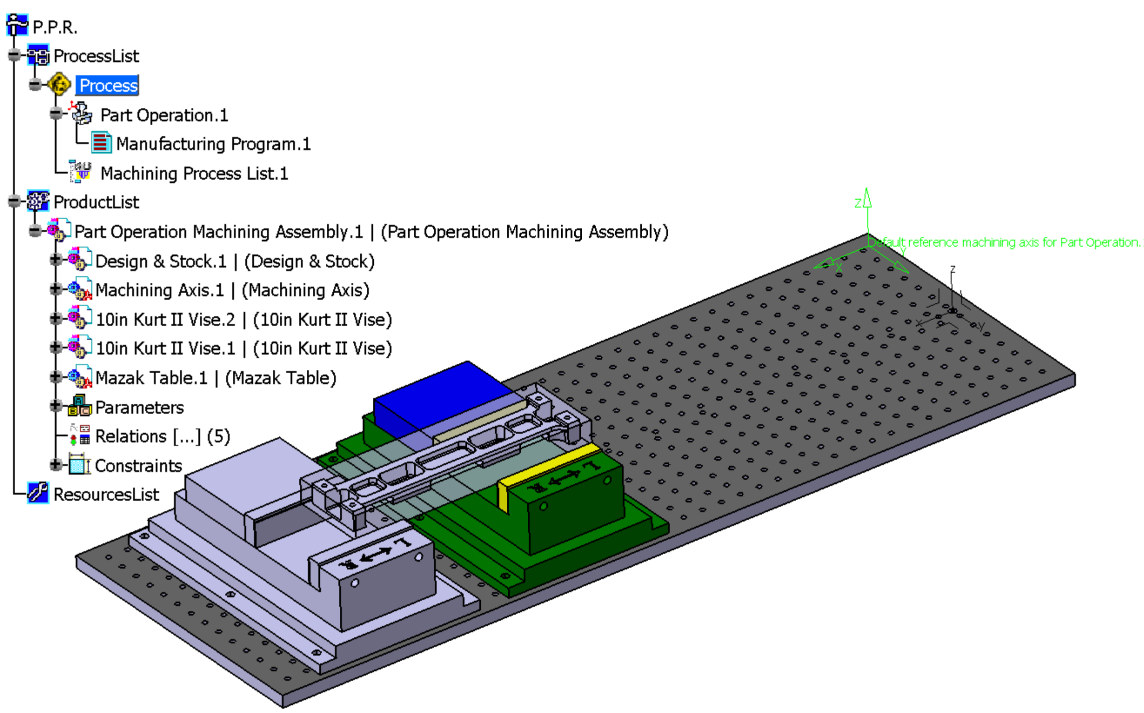
This section will investigate how to invoke the Prismatic Machining workbench and prepare your part for machining operations. Defining the part operation is a critical step for each machining process you start. Every time you prepare to machine a part, you must define the part operation.

There are two methods to start a new prismatic machining program. You can either start with the assembly open, then go to the prismatic machining workbench, or you can start with a blank prismatic machining process, then import the assembly into the process. Many times it will be easier to start with an assembly open, then switch to prismatic machining. This will be the method used here. You will use the other method later when working with multiple part operations.

Open the Part Operation Machining Assembly document from the *Part Operation* directory. By opening the assembly first, then switching to the prismatic machining workbench, you save the extra step of having to import the assembly.

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

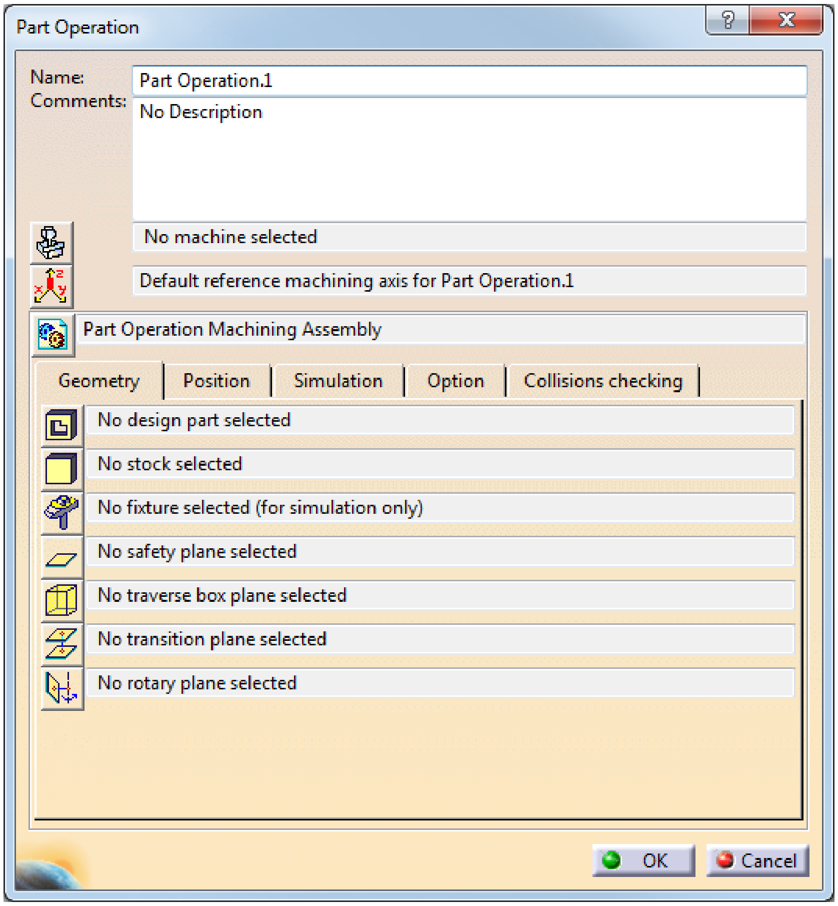
Now you are ready to begin defining the Part Operation.



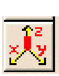
If you remember from the introduction, the Prismatic Machining workbench utilizes the PPR tree, or the Process Product Resource tree. Refer back to the introduction for full details on the PPR tree.

Defining the Part Operation

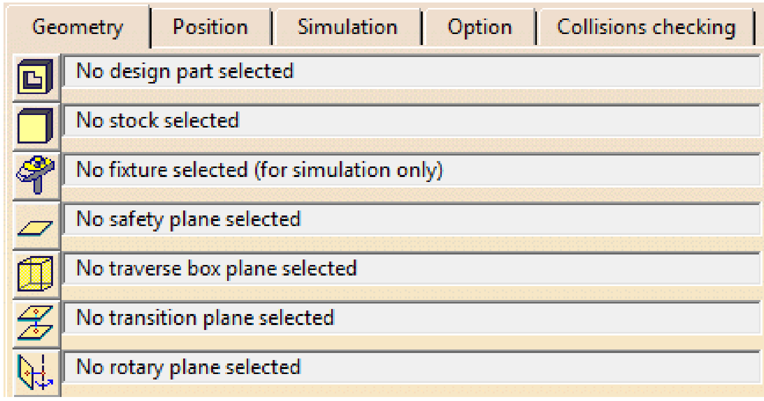
Double select the *Part Operation.1* branch in the *PPR* tree. This will display the *Part Operation* window.

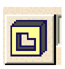
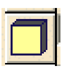
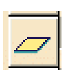
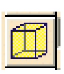
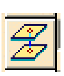
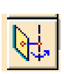


There are several different fields that will need to be defined before you begin machining. These steps will be very common for all parts in most situations. Take a quick look at the different areas of the part operation.

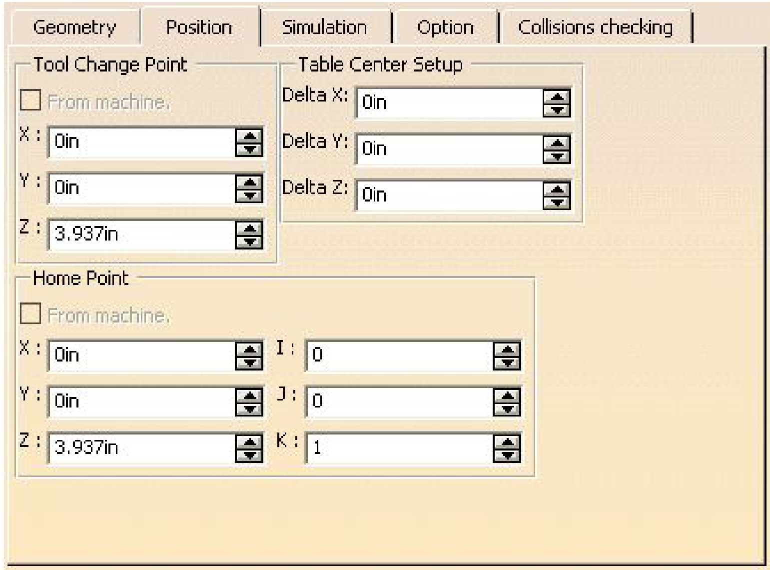
Name	Defines the name of the Part Operation
Comments	Allows you to enter any information about the program that you wish. The comments can be displayed in the documentation generated for the part operation.
 *Machine*	Defines the characteristics of the machine that you will be using
 *Reference Machining...*	Defines the location and orientation of the machining axis
 *Product or Part*	Allows you to import a part or product into the process

Geometry



	*Design Part for Simulation*	Defines the design part for use in the material removal analyses
	*Stock*	Defines the stock part around the design part. If a stock part is not selected, a significantly bigger rectangular block will be assumed.
	*Fixtures for Simulation*	Allows you to define any fixtures around the part. During material removal simulation, machining the fixture parts will display red areas to indicate crashes.
	*Safety plane*	Defines the safety plane for the part operation
	*Traverse Box Planes*	Allows safety planes to be defined on all six sides of the part
	*Transition Planes*	Defines additional traverse planes not at right angles to each other
	*Rotary Plane*	Defines a radial safety plane for working with rotary axis machines

Position

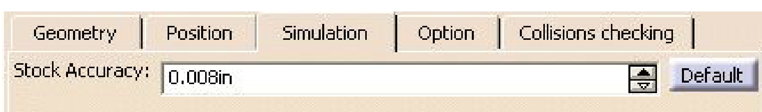


Tool Change Point Defines the X,Y,Z coordinates for the tool change point. This point should be away from the part, otherwise the tool will crash into the part to issue the tool change.

Table Center Setup Defines the offsets from the machining axis to the center of rotation for a rotary table machine

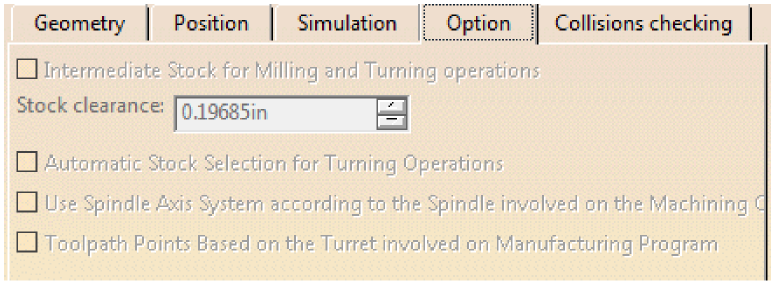
Home Point If not defined in the machine definition, the home point location of the machine must be defined here. Again, this point must provide clearance from the part and stock, otherwise a crash will occur.

Simulation



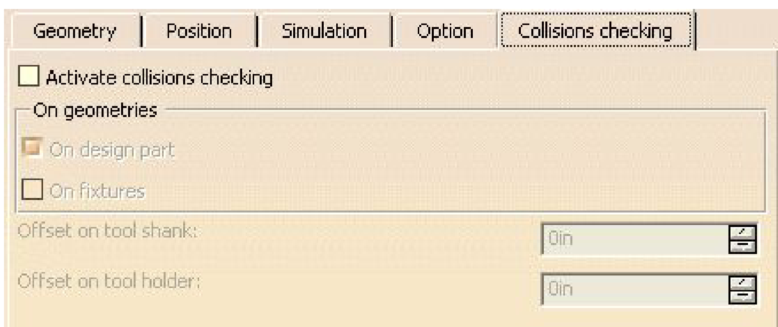
Stock Accuracy Defines the accuracy of the machine simulation stock material. The smaller the number, the higher the accuracy, however, the slower the simulation will run.

Option



Intermediate Stock...	Allows the intermediate stock to be automatically computed and taken into account for the tool path computation
Automatic Stock...	Allows the stock material to be obtained from the stock definition when working with lathe machining.
Use Spindle Axis...	Allows the tool tip points to be computed based on the spindle rather than the default reference machining axis system
Toolpath Points...	Allows the toolpath points to be computed based on the turret axis system rather than the part operation axis system

Collisions checking



Activate collisions... Allows for quick feedback about collisions during the tool path replay

On geometries

On design part	Detects collisions on tool/tool holder and design part
On fixtures	Detects collisions on tool/tool holder and fixtures

Offset on tool... Sets the offset on the tool shank

Offset on tool... Sets the offset on the tool holder

Change the *Name* to <u>Part Operation Setup</u>. 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.

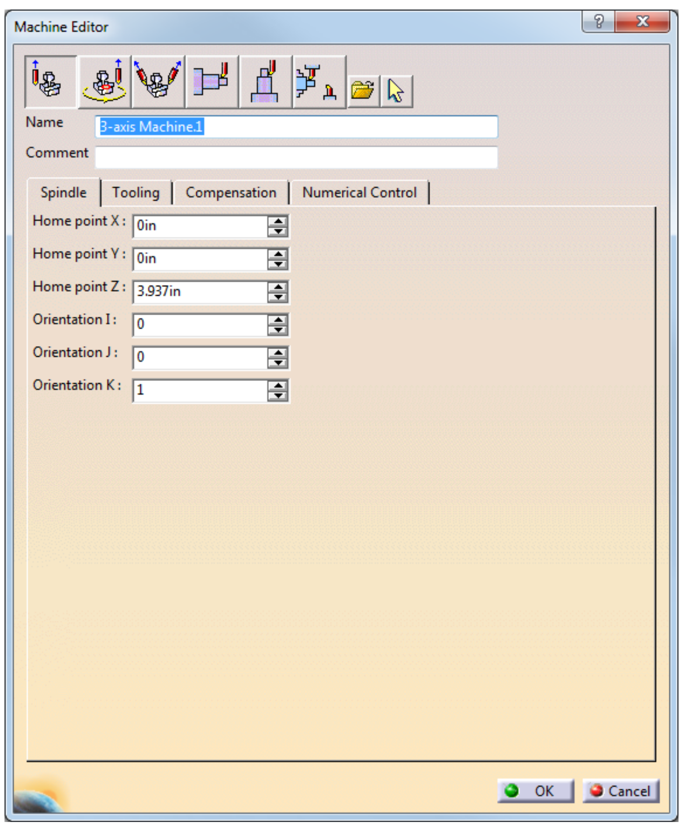
Change the *Comments* field to read <u>Your Name</u> and <u>Today's Date</u>. The comments field is very useful to insert specific information about the part operation. In this case, you are just going to state who you are, but you could add information about the part, information about the use of the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

At this point, you are ready to begin defining other aspects of the part operation. The first definition that needs to be made is to define a machine.

Machine Definition

There are a number of various machining parameters that can be set. It is very important that you define the machine. The machine definition gives CATIA information about the home point, orientation of the spindle, and other aspects such as the tooling catalog, and NC code output parameters. It is best to always start with the machine definition before continuing.

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.



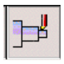
The first set of icons across the top allow you to define the specific type of machine you are going to be using.

The machine types are as follows:

 3-Axis Machine

 3-Axis with Rotary Table Machine

 5-Axis Machine

 Horizontal Lathe Machine

 Vertical Lathe Machine Deprecated

 Multi-slide Lathe Machine

You also have the ability to assign a machine that would be built with the Machine Builder workbench, or DELMIA applications.

 Opens a machine from file selection

 Assigns a machine based on selection from the resource list

Name Specifies the name of the machine

Comment Allows you to enter comments

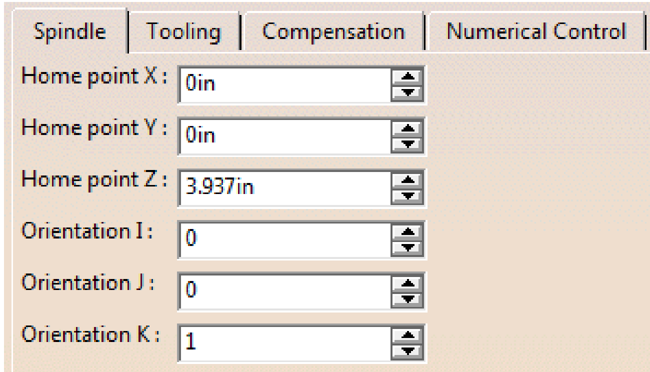
Based on the type of machine defined or selection, you will get various tabs to define the machine parameters. You will investigate the machine parameters for a simple 3-Axis machine.

Select the 3-axis Machine icon.  This is the default option and may already be selected.

Change the *Name* of the machine to <u>Fadal</u>. The name of the machine is not extremely important. The most important part is that the machine parameters are defined.

Take a moment to go over the various machine parameter tabs for the 3-axis machine.

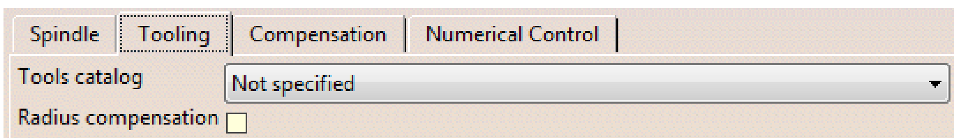
Spindle



Home point X, Y, Z Defines the X, Y, and Z coordinates of the tool home point

Orientation I, J, K Defines the initial orientation of the tool

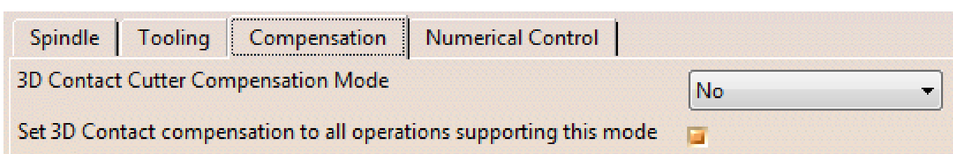
Tooling



Tools catalog Defines what tool catalog you will be using

Radius compensation Toggles the radius compensation on or off for each tool

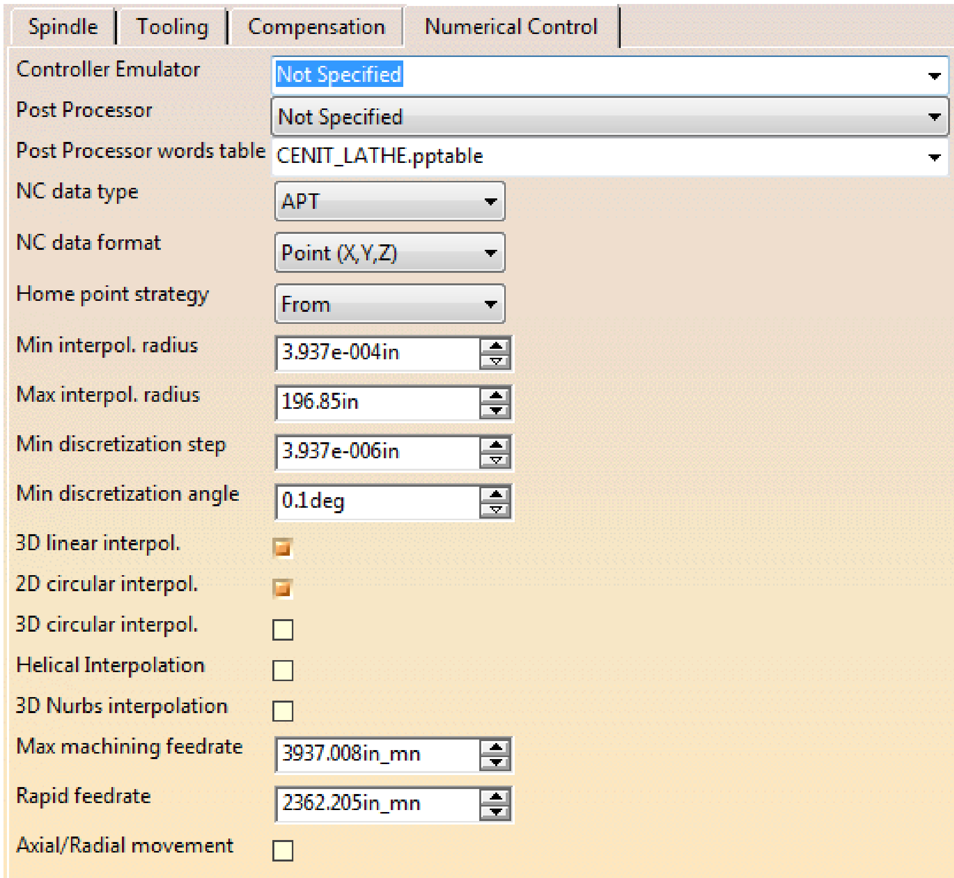
Compensation



3D Contact Cutter... Defines the cutter compensation mode for cutting in 3D space

Set 3D Contact... Allows you to turn on the cutter compensation for all supporting 3D cutting modes.

Numerical Control



Controller Emulator	Defines a controller emulator. The controller emulator allows machine simulation based on the post processed code.
Post Processor	Allows you to define the specific post processor database to use. You must have a post processor vendor selected under *Tools*, then the *Machining* branch, and the *Output* tab.
Post Processor words table	Defines what post processor word table to use. The post processor words table defines the specific output format for the post processor such that all the commands are generated properly.
NC Data Type	Allows you to choose from APT, CLF, or ISO data types
NC Data Format	Allows you to define if XYZ coordinates, or XYZ and IJK axis locations are output. This will automatically change based on the type of machine selected.

Home Point Strategy — Indicates if the home point is the current tool location (*From*) or if it needs to move to the location (*Goto*)

Min & Max interpol. radius — Defines the minimum and maximum circle that will output as a circular motion. Circular shapes outside of this range will output a series of GOTO statements in the APT code.

Min discretization step/angle — Defines the minimum motion distance and minimum angle that will generate a GOTO statement in the APT code

3D linear interpol. — When checked, a single linear GOTO statement will be issued when moving in a diagonal direction. If unchecked, a series of points will be generated based on the machining tolerance.

2D & 3D circular interpol. — When checked, either 2D or 3D circles will be interpolated, or both.

Helical Interpolation — Specifies the ability to make a helical interpolation between two consecutive points.

3D Nurbs interpol. — Outputs the NURBS curve to allow the controller to machine the curve directly. If unchecked, a series of GOTO points will be generated to define the curve.

Max machining feedrate — Defines the maximum machining feedrate that will be allowed

Rapid Feedrate — Defines the estimated rapid feedrate for the machine used. CATIA will always generate a RAPID statement to move rapid, however, this feedrate will allow for more accurate time calculations when the machine makes rapid movements.

Note: Acceleration and deceleration time is not taken into account unless a machine has been selected that has accelerations defined.

Axial/Radial movement — When checked, the tool will only make axial and radial movements, and not a combination of both (3D diagonal motions)

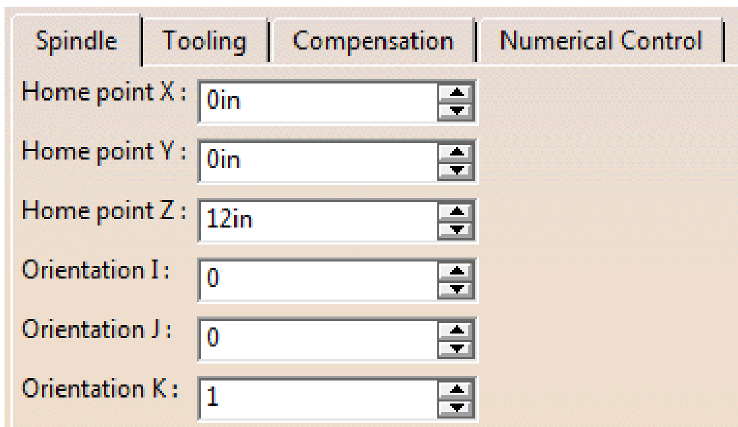
Due to the sheer number of options, you will often find that many companies will define machine process seeds that define all of these options ahead of time, just like the machine setup. This will allow for machine programmers to utilize the proper settings for the machine each time a program is started.

Since you are not starting with a machine seed, you will need go through and set all the options.

Switch to the *Spindle* tab if not already there. It is very important to set the home point and orientation for the machine.

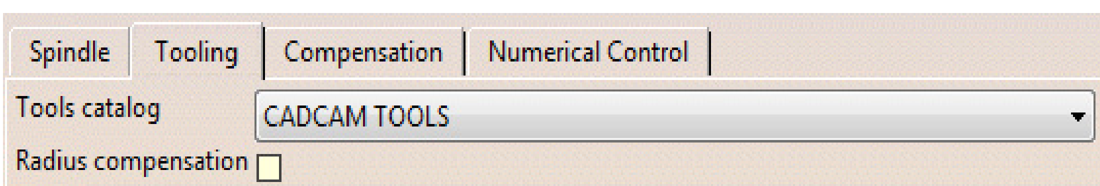
Change the *Home point* to be set to (0,0,12in). That is, make the X value 0 inches, Y value 0 inches and the Z value 12 inches. All coordinates will be relative to the machining axis system for this part operation. Even though the machine axis has not been defined, you will set the home point first. The machine axis will be moved after the machine definition is made.

Leave the *Orientation* set at 0,0,1. This will set the axis to be along the K direction, or parallel to the Z axis.



Switch to the *Tooling* tab. Remember, this tab allows you to define the tooling catalog for the machine. Many times you may have a global tool catalog for all available tools, or you may have a tool catalog per machine. Tooling catalogs per machine are often found when a common set of tooling is always loaded in the machine.

Change the *Tools catalog* to *CADCAM TOOLS*.



This will define a general tool catalog for the machine.

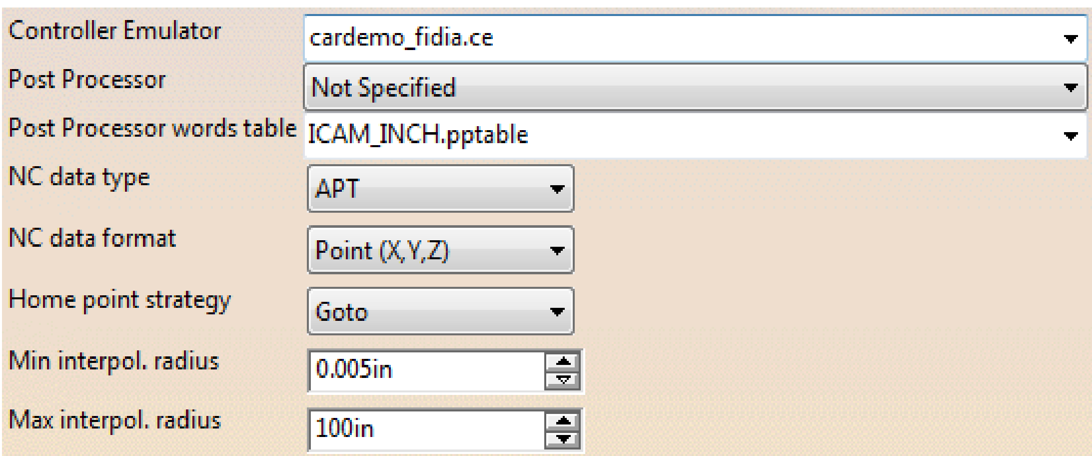
Switch to the *Numerical Control* tab, change the *Post Processor words table* to *ICAM_Inch.pptable*. This will allow you to utilize the ICAM post processor and insure that the output code will be in the proper format for the ICAM post processor. This will also insure that the output is in inches rather than millimeters.

Switch the *Home point strategy* to *Goto*. Since the exact location of the tool's parking place is unknown, it is best to set the *Home point strategy* to *Goto*. This will output a GOTO statement at the beginning of the program.

Change the *Min interpol. radius* to 0.005in. This will cause the machine to output circle statements for any circle greater than 0.005in.

Change the *Max interpol. radius* to 100in. This will cause CATIA to output circle statements for any circle less than a 100in radius.

By defining the minimum and maximum interpolation radii, you are essentially defining a range for the APT generator to define the circular motions.



Change the *Min discretization step* to 0.0001in. This will indicate to CATIA that the minimum distance between steps will be 0.0001in.

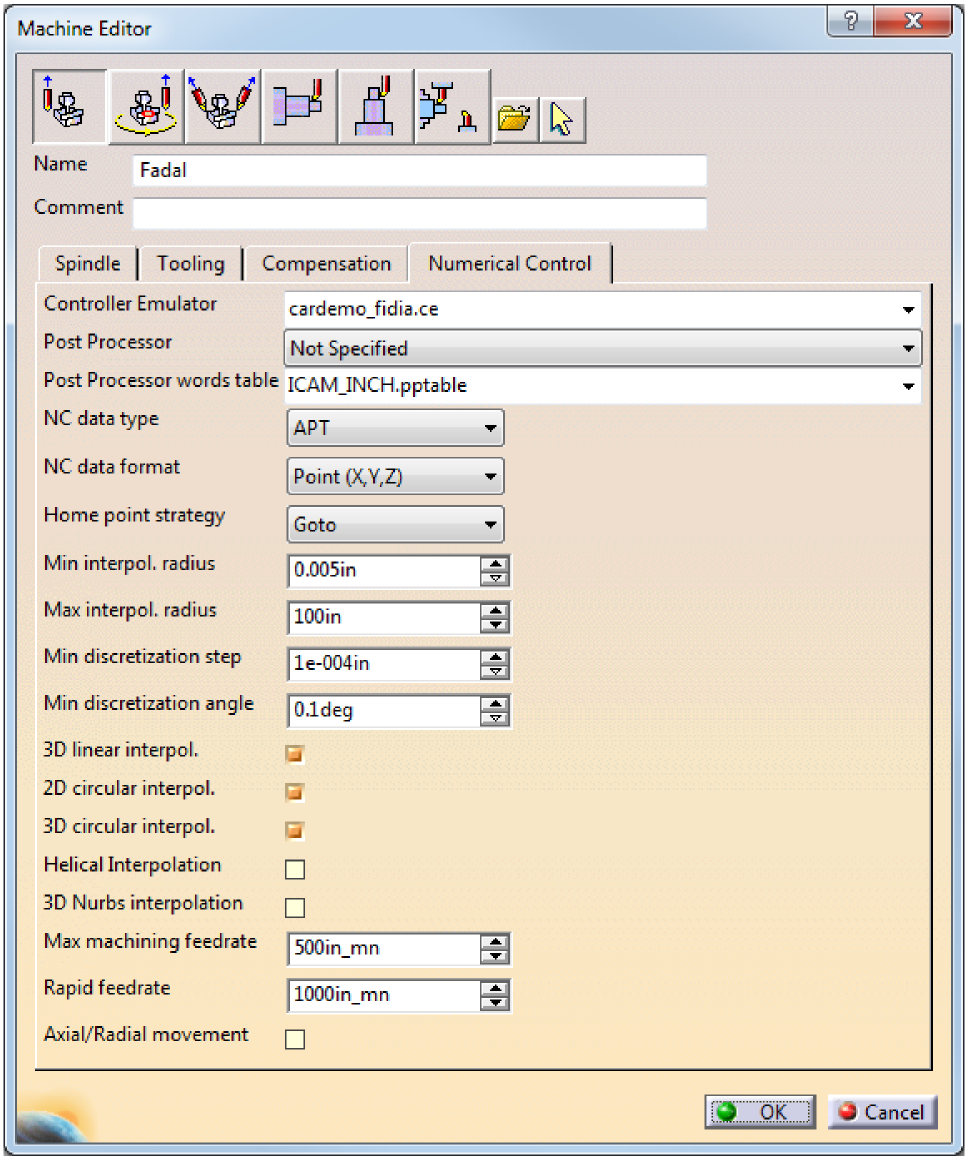
Turn on *3D circular interpol.* This will indicate that when a circle is made that is not normal to the tool axis that the APT generator should still output a circular statement rather than a series of GOTO points.

Set the *Max machining feedrate* to 500in_mn. This will indicate that the maximum machining feedrate allowed for this machine is 500 inches per minute.

Set the *Rapid Feedrate* to 1000in_mn. Even though the APT generator will output a RAPID statement, this will provide CATIA with a method of calculating the amount of time it takes to move from one point to another while in rapid.

Leave *Axial / Radial movement* off. If you remember, by turning this on, you will be indicating that CATIA should perform a best guess at moving in an axial and radial motion when requested to move diagonally. You are best to leave this off and control the movements via macros and other controllable methods.

At this point, you have all the machine parameters completed.



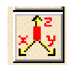
Select *OK* when done. This will have the machine set up.

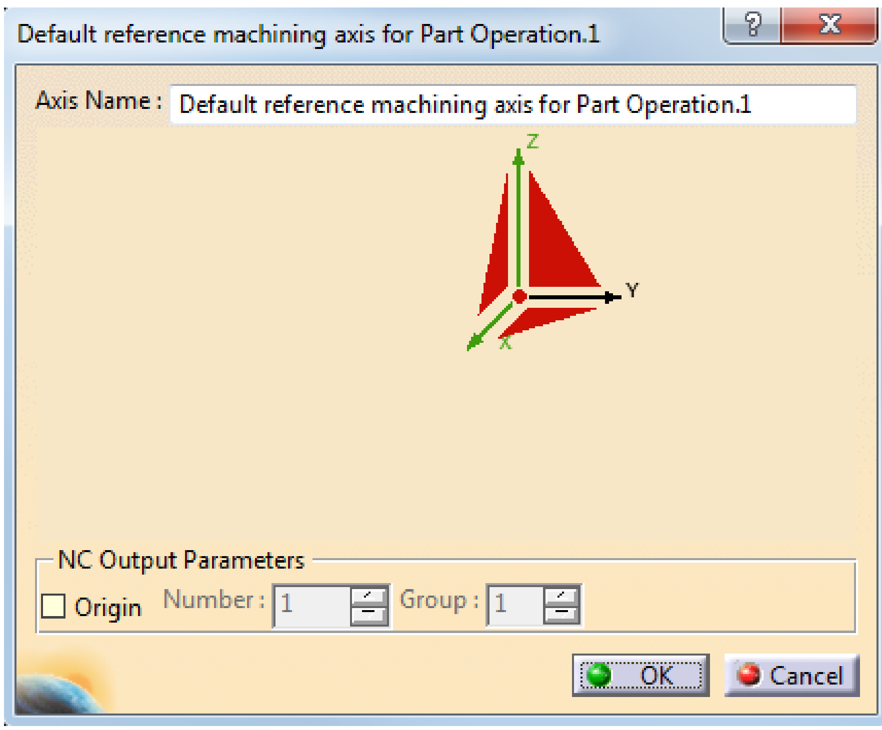
Always be sure the machine parameters are set properly, otherwise you will get inaccurate output in the APT generator, and thus you will get incorrect output in the final machine code.

The next important step is to define the location of the Machining Axis.

Machining Axis Definition

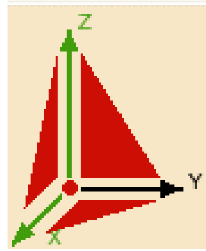
The machining axis is an important part of the NC Setup. The machining axis can generally be placed anywhere on the model that you want, however, there are a few locations that are better than others. You rarely want to have the machining axis buried or placed inside the stock material. You generally will want the machining axis to be based off of a corner of the stock material, that way it is much easier for the machinist to mount the stock material to the table. If you are simulating the entire table, as you are in this exercise, you may want to place the machining axis at the machine's specific machining axis location. If the machining axis is determined to be in a bad location, you can always move the axis. Moving the axis will cause the tool paths to automatically recompute to the new axis coordinates.

Select the Reference Machining Axis System icon.  This will display the *Machining Axis System* window.

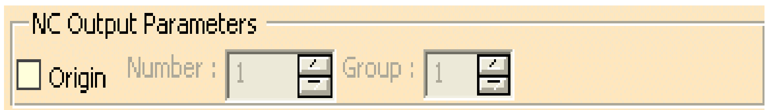


Take a moment to go over the various areas of the machining axis system window.

Axis Name This defines the name of the axis system, and the name that will be displayed in the graphical workspace



Machine Axis This sensitive area allows you to define the axis. The center dot allows for axis positioning, used in conjunction with the X and Z sensitive axes for orientation. The planes on the sides allow for axis selection, axis positioning and orientation is determined by the selected axis.



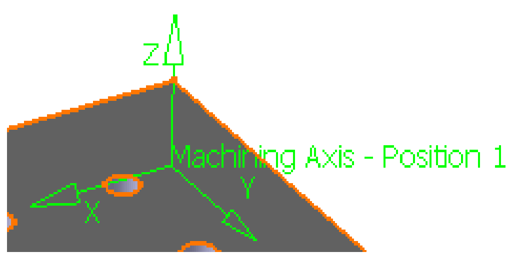
NC Output Parameters

Origin Defines if an origin identifier is generated with the axis definition in the APT code

You will notice the Y axis is not selection sensitive. This is due to the fact that all machining axis systems are right handed axis systems. The Y axis will always adjust based on the X and Z axis directions to maintain a right handed axis.

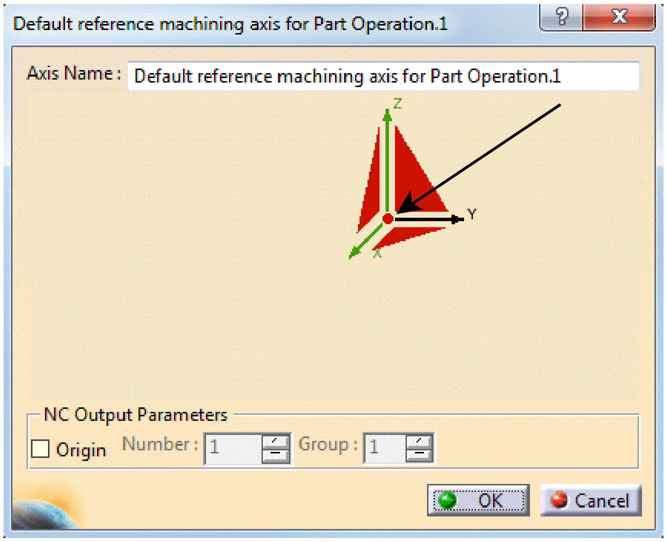
Now you are ready to define the axis location a few different ways.

Change the *Axis Name* to <u>Machining Axis - Position 1</u>. This will give the axis a decent name. You will also notice that the axis changes names in the graphical workspace.



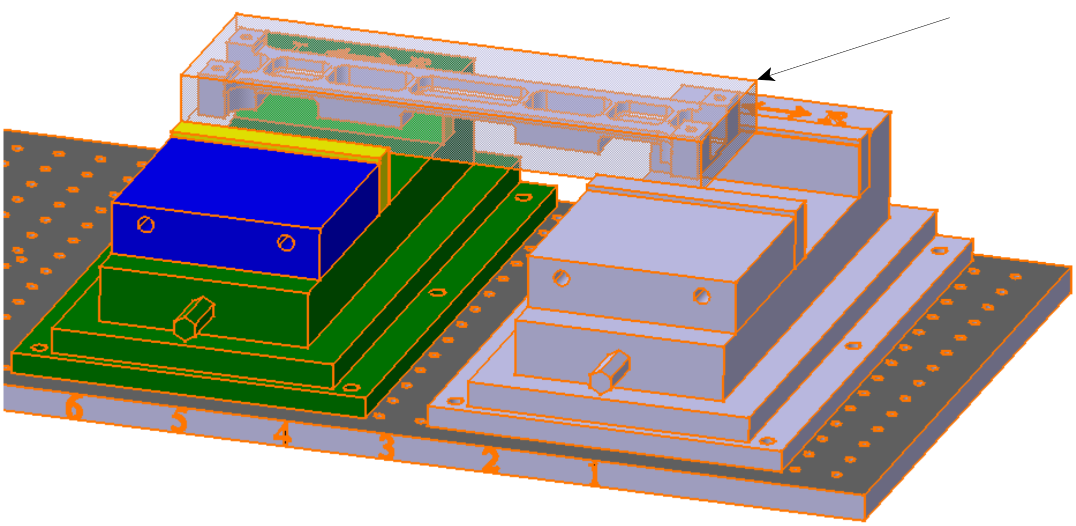
Properly identifying the axis systems will allow you to keep track of which axis systems are used for which part operations.

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

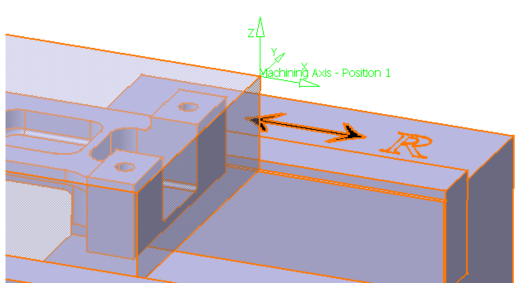


The center red dot will allow you to move the entire axis system from one location to another. The *Machining Axis System* window will disappear while CATIA waits for you to select a point or vertex to be the new center of the axis system.

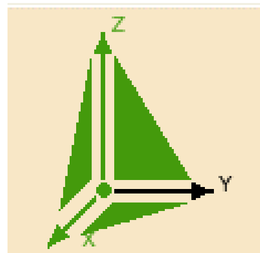
Select the top right corner of the stock part as shown. This will define the new center of the axis.



The machining axis system will move to the corner.

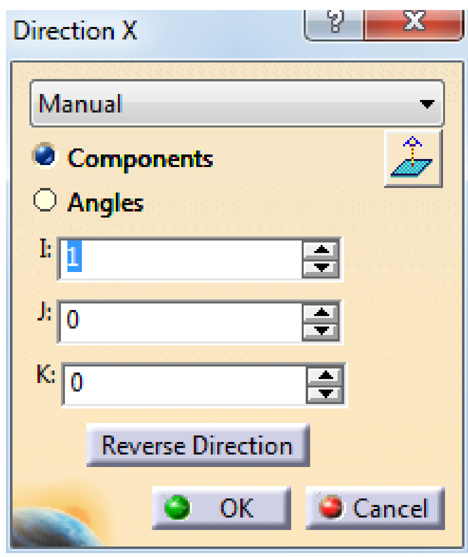


Notice the axis system in the machining axis window turns green. This denotes that a new axis location has been defined.



Now to adjust the axis directions. Assume in this case, you want the axis system pointing towards the part. This means that you want to reverse the X and Y axis directions such that they both point towards the stock part. Since you cannot change the Y axis, you will have to adjust the X axis direction.

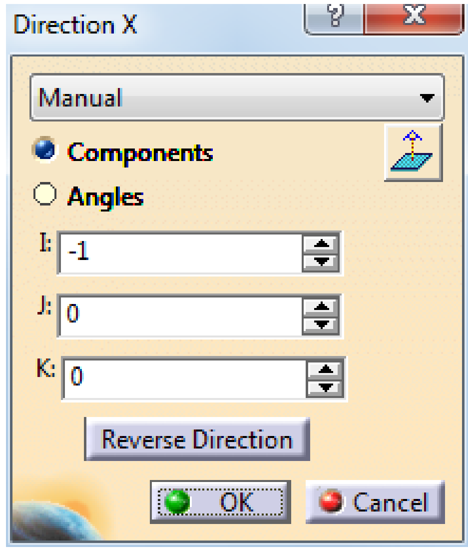
Select the X axis arrow from the machining axis system window. This will allow you to move the X axis. This will also display the *Direction* window.



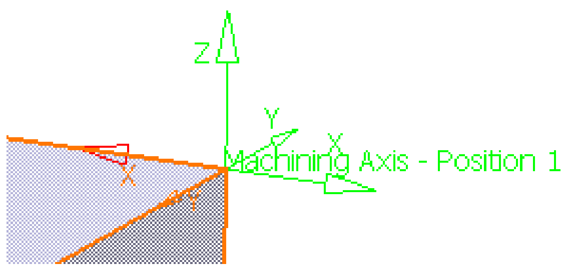
There are three methods to define the axis direction.

Selection	This allows you to select an edge or line to define the axis direction
Manual	This allows you to key I, J, and K directions to control the axis direction. Manual also allows for reversing the direction of the axis.
Points in the View	Allows for selection of two real points to define the axis direction

With the selection mode set to *Manual*, select *Reverse Direction*. This will reverse the I direction of the axis, hence changing the direction of the X axis.



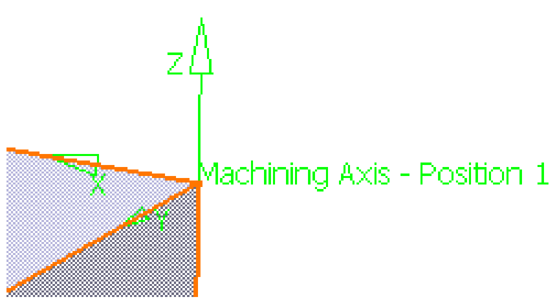
Notice that the axis does not appear to move.

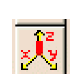


If you look very closely, you will notice faint red axis directions that indicate the new directions. Unfortunately, the green machining axis does not actually update until you complete the axis definition. You will find that many times you will need to complete the axis definition to insure the axis is correct. If it is not correct, then you will simply need to go back to the axis definition by selecting on the Machining Axis icon again.

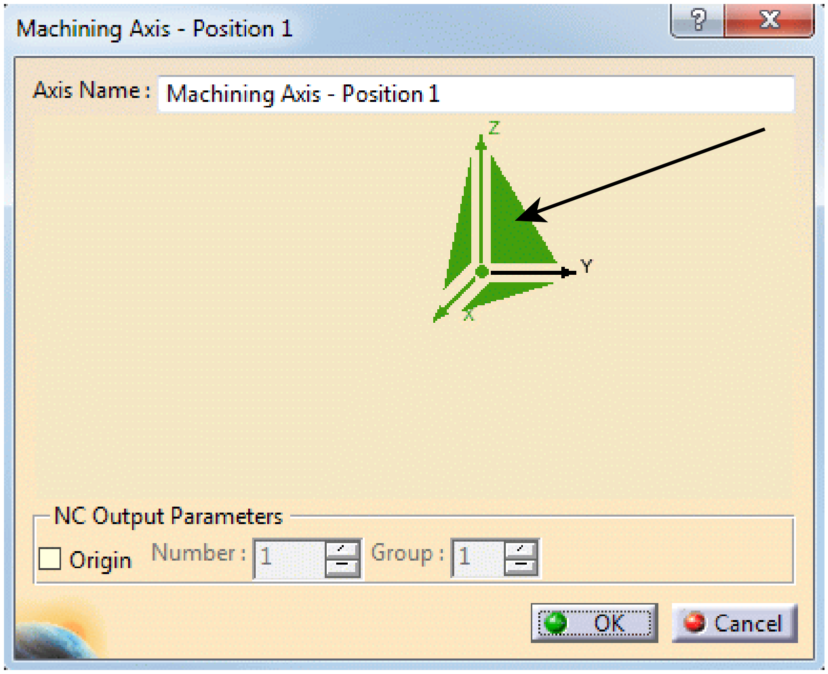
Select *OK* to the *Direction* window. This will display the machining axis window again.

Select *OK* to the machining axis window. This will take you back to the *Part Operation* window. Notice the axis system now changes and updates to show the new position.



Select the Reference Machining Axis System icon again.  This is going to take you back to the machining axis definition so that you can relocate the axis system.

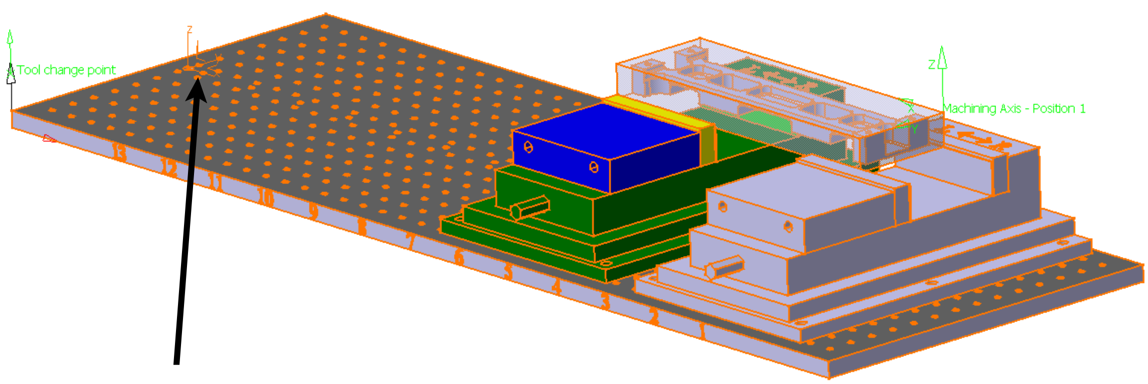
Select one of the planes of the axis system definition window. The planes are shown here.



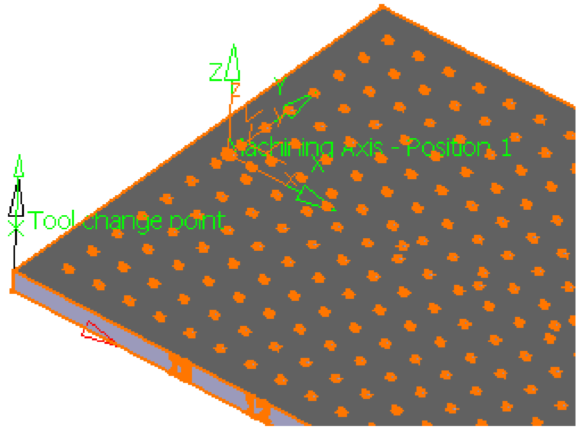
When you select the planes, the window will again disappear while CATIA waits for you to select an axis.

At the other end of the table, there is an axis system. You will set the axis system to be the same as this axis.

Select the axis system located at the end of the table. This axis system is shown here.



The machining axis will move and rotate to match the axis defined in the detail part.



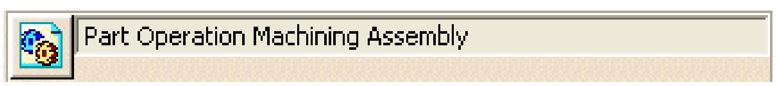
Select *OK* to the machining axis window. For now this will serve as the machining axis. Many times machine seeds will have an axis location defined that can serve as the machining axis.

At this point, you now have the machine defined, and the machining axis defined and located. The *Product or Part* icon is the next in the line of icons.

Product or Part Definition

The product or part definition allows you to define the part or assembly that you will be machining. Generally, you will always want to machine an assembly. By machining an assembly, you have the ability to show fixtures, tooling, stock, design, etc.

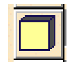
Since you started the machining process with a product open, you will notice the field is already defined.



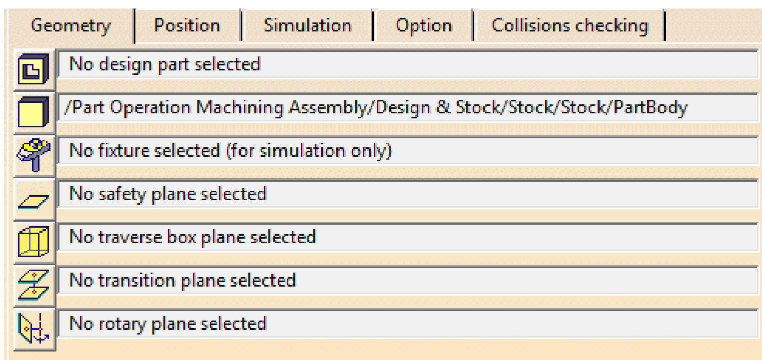
Had you started with a blank process, or a new part operation, you would find that the field would be blank, no product would be shown, and you would need to select the Product or Part icon in order to define the product. You will use this icon in more depth when you start defining multiple part operations.

Geometry Definition

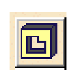
The geometry definition is another important area for defining all the necessary geometry that you will be machining. If you remember from earlier, you have options to define the design part, stock part, fixtures, and safety planes. You should always define as much geometry as you can. Defining all the geometry allows for better visual replays and analysis, as well as aiding in macro definitions.

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

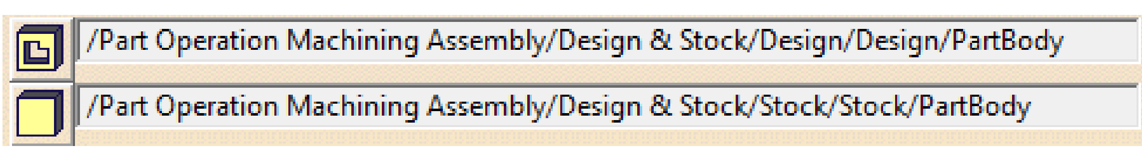
Select the stock part from the graphical workspace, then double click in space. This will define the stock model. As mentioned earlier, if you fail to define the stock model here, the system will assume a large, rectangular stock around the design part. This assumed stock part will be significantly bigger than the design part. You should notice that the stock is now defined.



Hide the stock part. With the stock hidden, you can select the design part much easier. If you did not want to hide the stock, you could have also expanded the specification tree until you can access the part body of the design part.

Select the Design Part for Simulation icon.  This will define the final, as designed, part of the machining process.

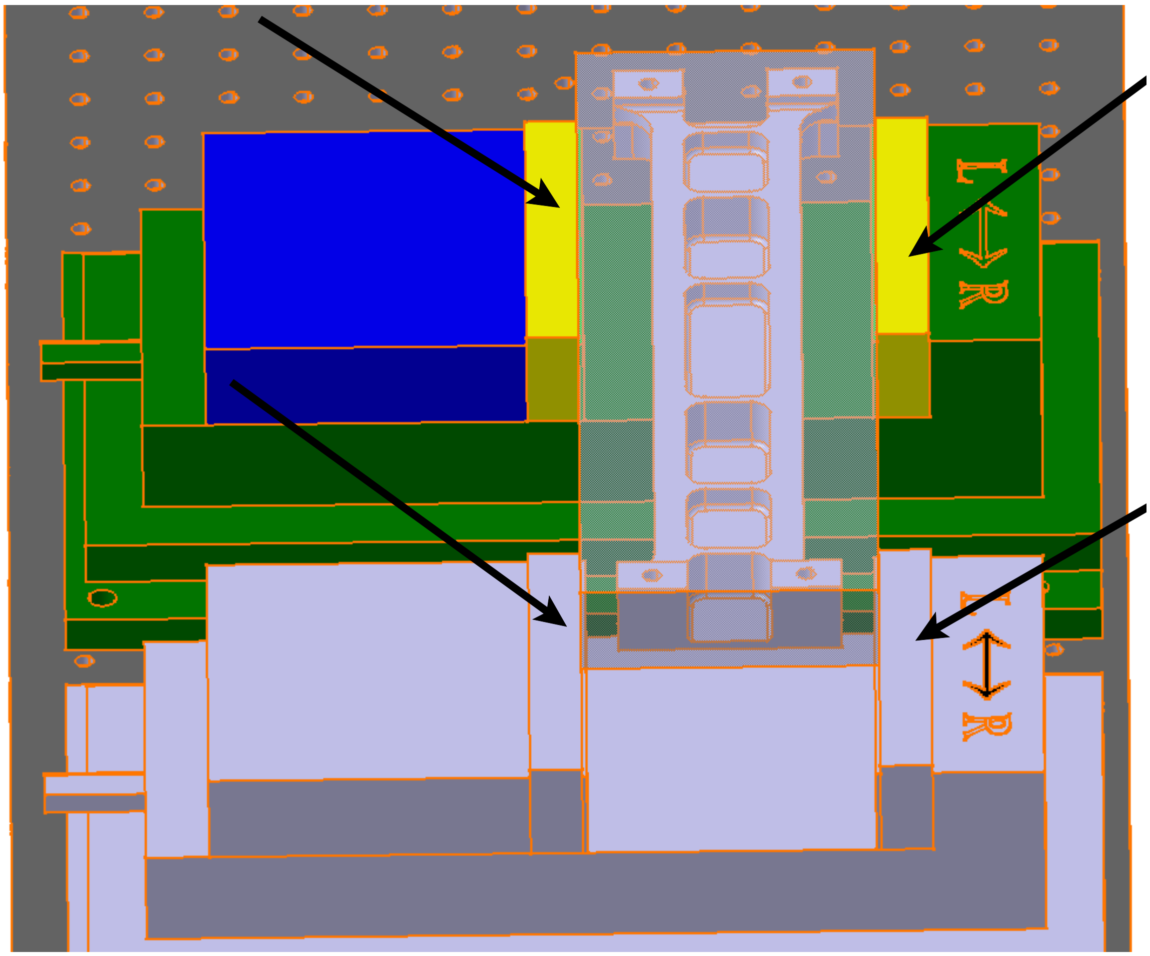
Select the design part, then double click in space. Unhide the stock when done. You will be needing the stock part again, so it is best to unhide it.



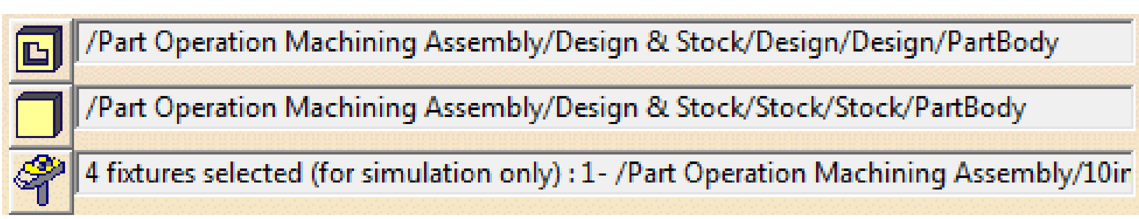
The next definition is the fixtures that you want to view in the replay. Generally, you do not define the entire assembly as the fixtures, but instead define the critical, specific fixture elements. Defining a lot of complex fixtures will require an excessive amount of video memory, and can cause CATIA to crash.

Select the Fixtures for Simulation icon.  Since you want to keep the number of fixtures defined to a minimum, you will just want to select the jaws of the vise. If you are concerned with collisions with other parts of the vise, table, or related fixtures, you would go ahead and define them as well.

Select the vise jaws from the graphical workspace as shown. Just select each part once. If you select it twice, it will un-select the part. Unfortunately, with the part already highlighted, it is difficult to determine if the jaws are selected or not.



Double select in the workspace when done. You do not need to select on any part, just double select in space. You can define as many parts as fixture parts as you want. This will have four fixtures defined.

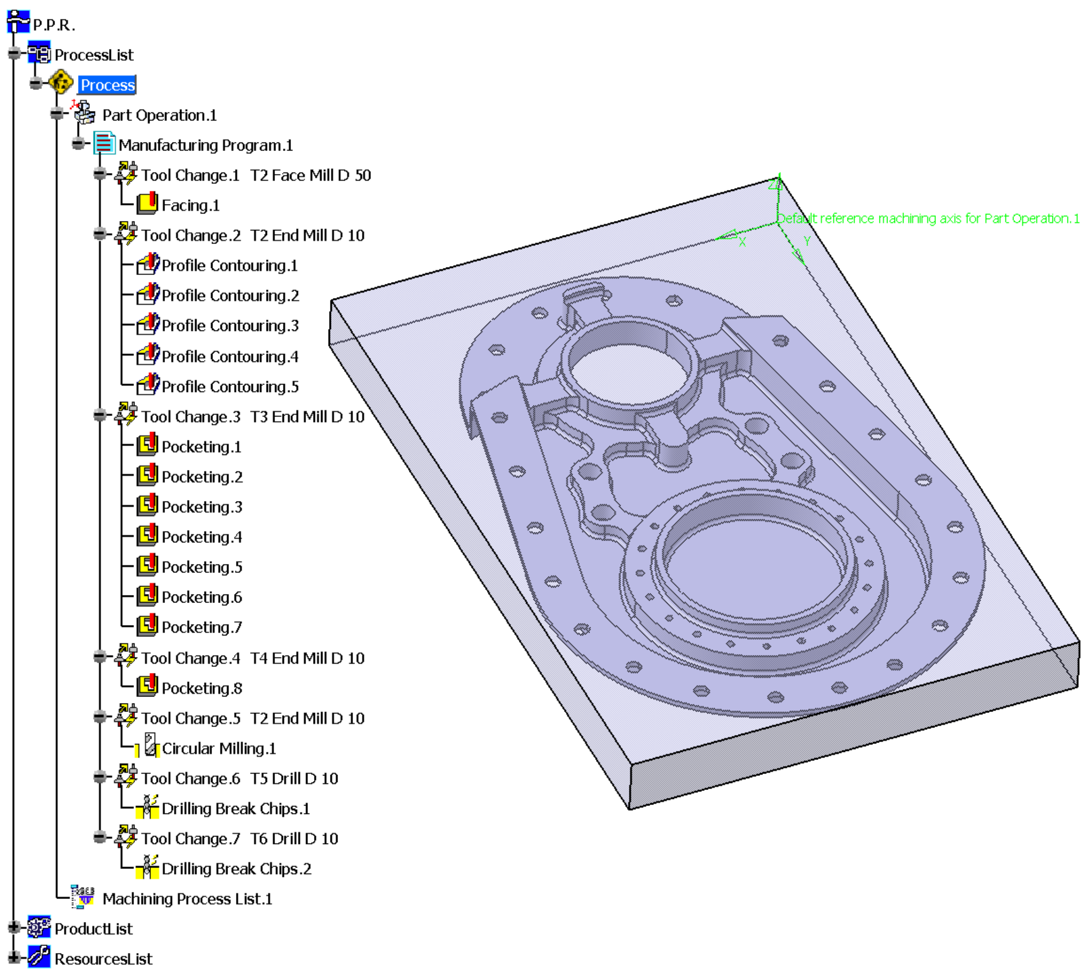


The last definition is the safety plane.

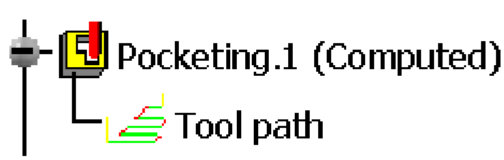
Replaying

Before you can fully understand what the various machining options allow you to do, you must first be familiar with replaying your tool paths. Replaying is the most important part of verifying whether the tool is accurately cutting the part. Viewing the replay helps to insure the correctness of the program overall.

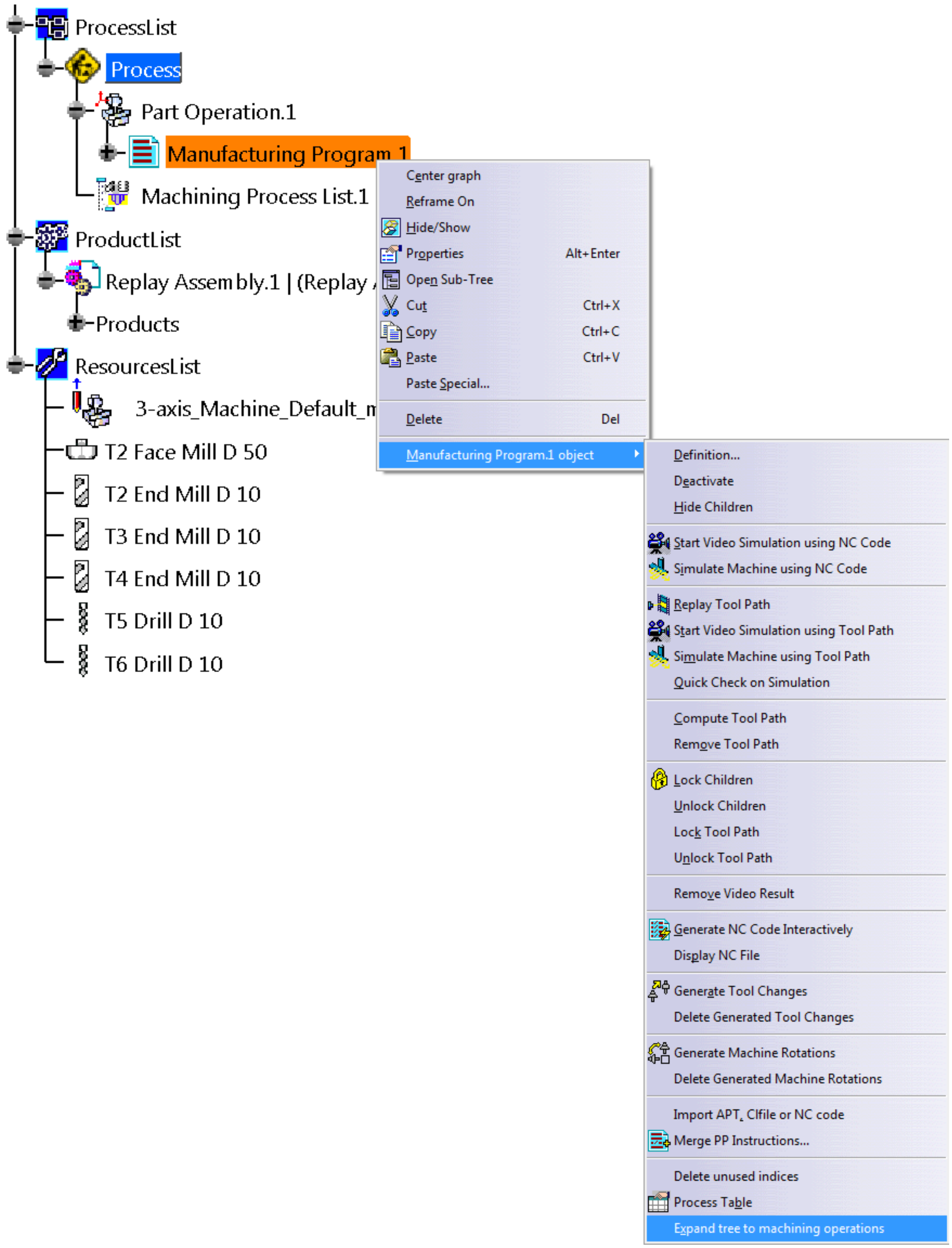
Open the Replay document from the *Replay* directory. This machining process already has machining operations applied to it. The various machining operations will allow you to become familiar with replaying your machining processes.



Machining operations can have two states. One state is to have the tool paths computed. In this state, the tool paths are available for the machining operation, and can be replayed instantly. When the tool paths are computed, a *(Computed)* text will appear next to the operation, and an additional branch will show up below the machining operation.



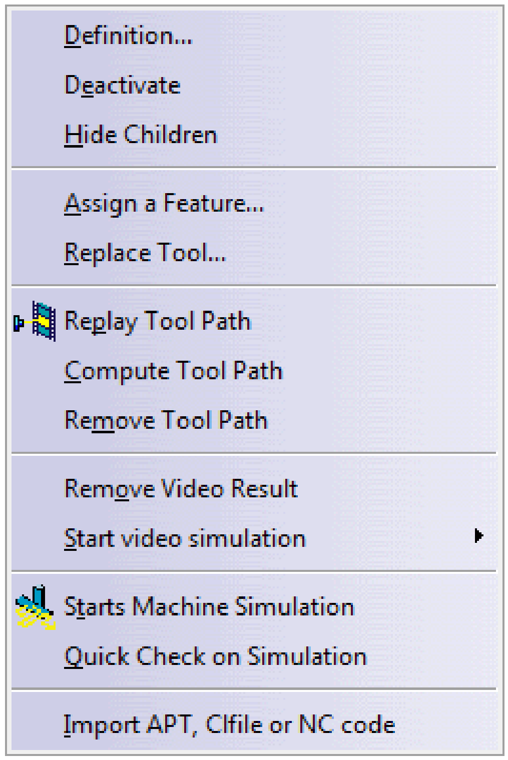
With the right mouse button, select on the *Manufacturing Program.1*, then select the *Manufacturing Program.1 object* and choose *Expand Tree to Machining Operations*.



This will insure that the tree is completely expanded to view all the machining operations. Alternatively, you could always select the plus next to each of the operations in the specification tree and expand it, however, if there are a lot of operations, this can be a time consuming process.

Since none of the machining operations have the Tool Path branch, and the *(Computed)* text is not present, the tool paths are not computed. The first step before replaying will be to compute the tool paths. Computing the tool paths can be done a number of different ways.

With the right mouse button, select on *Facing.1* from the tree and select *Facing.1 object* from the bottom of the contextual list. This will display the object properties for the facing operation. The most important options to you, while replaying, are covered below.



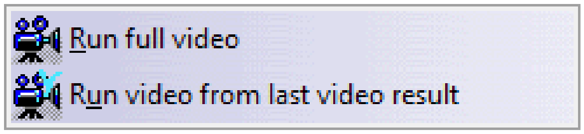
Replay Tool Path — Replays the currently selected machining operation. This can also be accomplished by selecting the Replay icon in the left hand toolbar. If a tool path is not computed, replaying the tool path will automatically compute it.

Compute Tool Path — Allows you to manually compute the tool path. When you compute the tool path, you also have the option to force the tool path computation.

Remove Tool Path — Allows you to remove or strip the tool path out of the process. By removing the tool path, you will dramatically reduce the size of the file saved.

Remove Video Result — When a video result is created, a temporary image is stored in your profile. This temporary image is generally deleted when you log off of your computer, hence breaking the link with CATIA. Generally it is a good idea to remove the video result when saving the file.

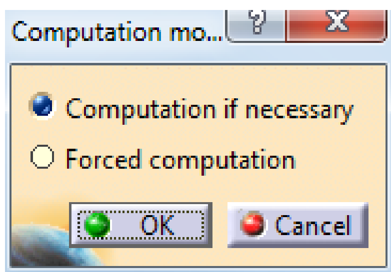
Start video simulation



Run Full Video	Allows you to run the full length video
Run video from last saved...	Allows you to run the video from the last saved result

Starts Machine Simulation	Allows you to visualize the actual machine cutting the part rather than just the tool. This will only work if a machine is defined and available for the visualization.
Quick Check on Simulation	Performs a quick check on the simulation
Import APT or NC code file	Import existing code from another document

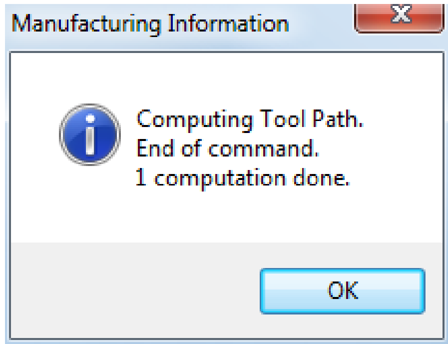
Select *Compute Tool Path*. This will display the *Computation* window that asks if you want to compute if necessary or force the computation.



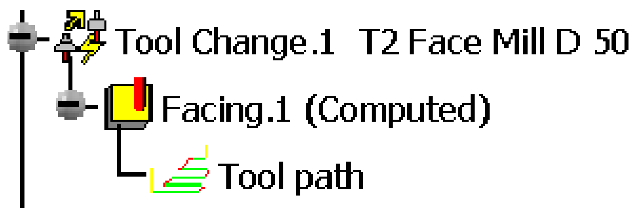
Computation if necessary	This will compute the tool paths if the status of the tool path is non-computed. Any tool paths with the *(Computed)* flag will not be re-computed.
Forced computation	This mode will recompute all tool paths, regardless of the status of the tool path. Tool paths denoted as *(Computed)* will also be re-computed.

Note: Many times, if you are getting unusual tool paths or are unsure if you are viewing accurate tool paths, it will be necessary to recompute the tool paths and force the computation. Forcing the computation will insure that all tool paths are up to date and computed properly.

Select *Computation if necessary*, then click *OK*. Since the tool paths were not computed, they will automatically compute. An information window displays the number of tool paths computed.



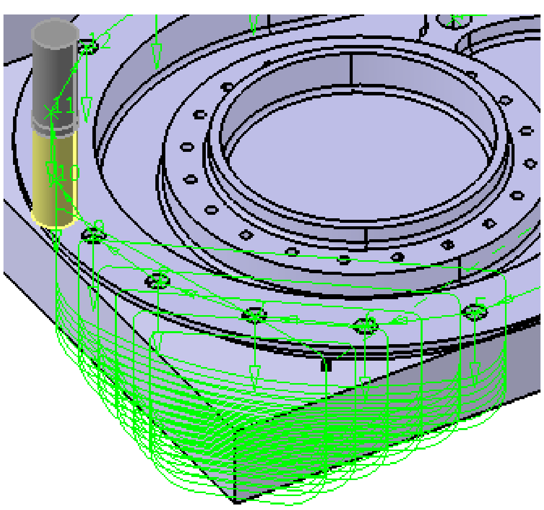
Select *OK* to the *Manufacturing Information* window. Now the tool path has been computed.

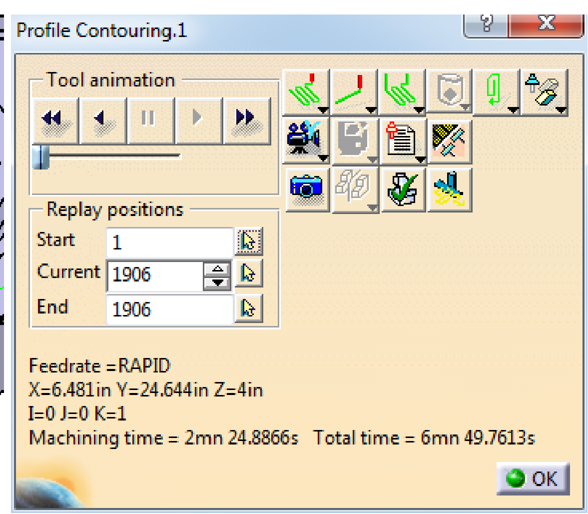


Note: If the operation shows as computed, yet the tool paths are not visible, you can turn them on by going to Tools, Options, Machining, Output Tab, and turning on Edit Tool Path is available.

You can also compute the tool paths by replaying the operation.

With the right mouse button, select on *Profile Contouring.1* from the tree and select the *Profile Contouring.1 object* from the bottom of the contextual list, then select *Replay Tool Path*. The *Tool Path Computation* window will show while the tool path is computed. This will display the tool paths on the screen, as well as show the replay window.





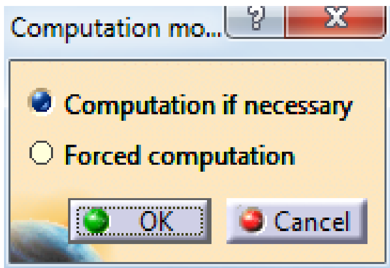
You will investigate the various aspects of the replay window, shortly.

Select *OK* to the replay window.

Computing the tool paths one at a time can become rather time consuming, especially when you have a lot of machining operations. Fortunately, you can also compute the tool paths by Manufacturing Program.

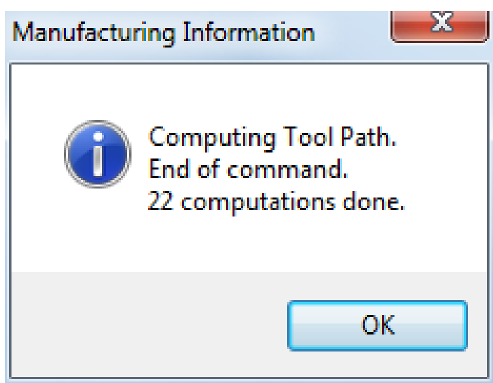
With the third mouse button, select on *Manufacturing Program.1* from the tree, then select on the *Manufacturing Program.1 object*. A number of new options appear, however, you will still find the ability to compute, remove, and replay the tool paths.

Select *Compute Tool Path*. This will display the *Computation* window again.

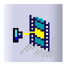


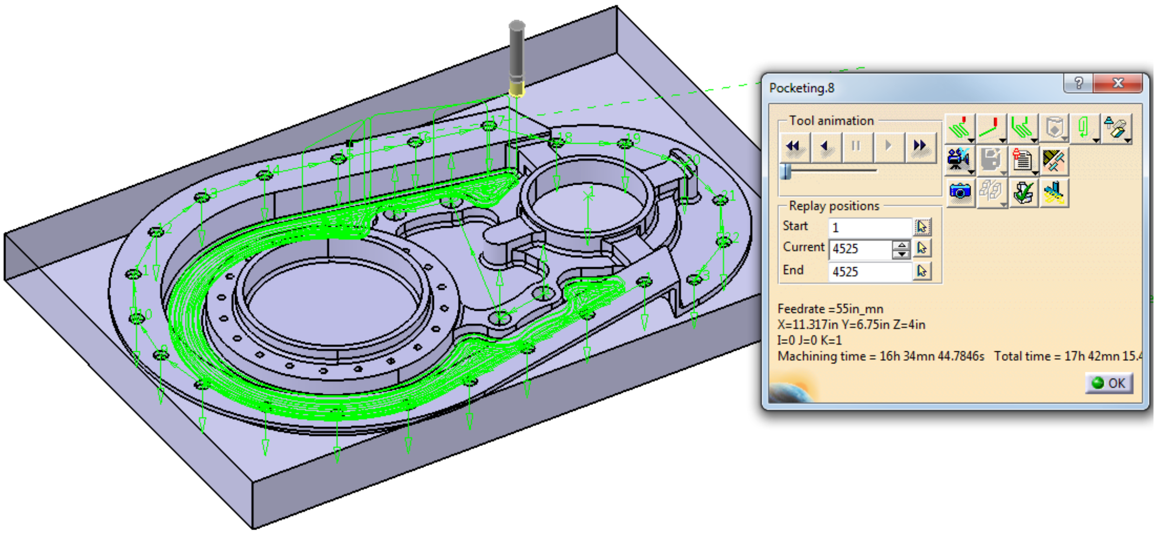
Be sure the *Computation if necessary* option is selected and select *OK*. The first two tool paths have already been computed, hence they will not get re-computed in this case. If you had selected on *Forced computation*, the first two tool paths would get re-computed.

This will now go through and compute the remainder of the tool paths. For long programs this may take some time. Once done, the *Manufacturing Information* window will display indicating the number of tool paths computed.



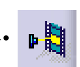
Select *OK* to the information window, then select in space. Selecting in space will insure that no machining operations or manufacturing programs are selected.

With no machining operations selected, select the Replay Tool Path icon, then select *Pocketing.8*.  Notice the tool paths immediately show up. Since the tool paths were already computed, there is no delay for tool path computation.

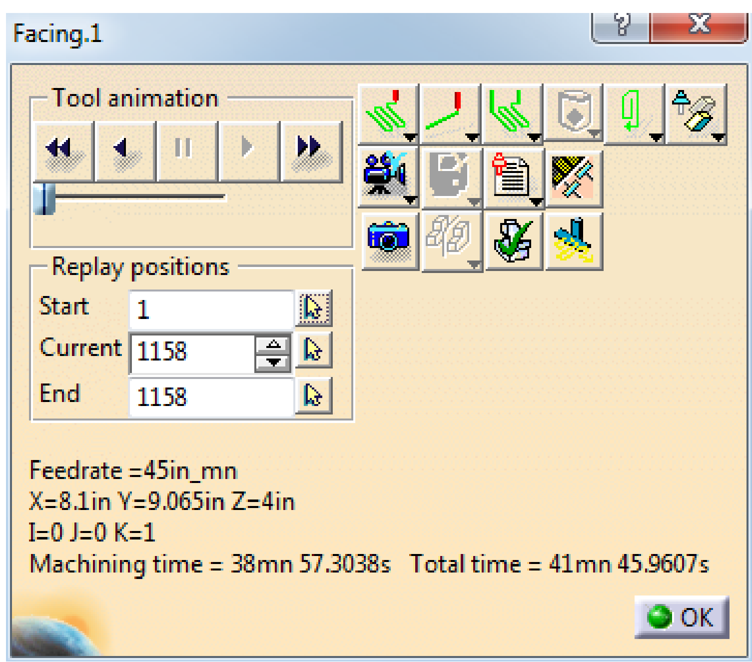


Select *OK* to the replay window.

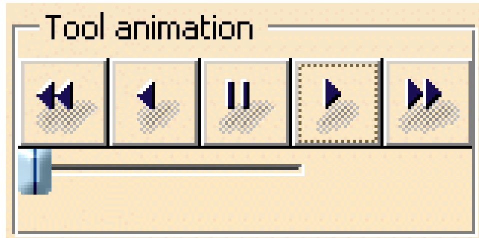
Now it is time to investigate the replay window a bit closer.

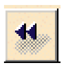
Select the Replay Tool Path icon, then select the *Facing.1* machining operation. 

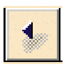
The replay window is shown.

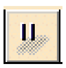


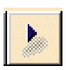
Tool Animation This frame houses the player controls. The controls work similar to VCR controls.



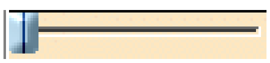
 *Skip to Beginning* (F5) — Skips the tool to be beginning of the tool path

 *Play Backwards* (F6) — Replays the tool paths backwards

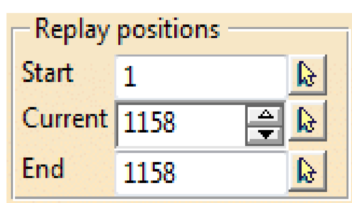
 *Pause* — Pauses the tool path replay

 *Play Forwards* (F7) — Plays the tool path forwards

 *Skip to End* (F8) — Jumps the tool path to the end

 *Speed* — Allows you to speed up or slow down the tool path animation replay. *Note: In Prismatic Machining, many of the tool paths run fast, and cannot be slowed down. The speed control does become more useful when working with Surface Machining operations.*

Replay positions Allows you to manipulate the exact position within the replay



Start — Defines the point where the replay will begin

Current — Specifies what point the replay is at currently

End — Defines the end point of the replay

Information block Gives information about the various aspects of the tool and the replay motion

Feedrate =45in_mn
X=8.1in Y=9.065in Z=4in
I=0 J=0 K=1
Machining time = 38mn 57.3038s Total time = 41mn 45.9607s

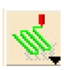
Feedrate Indicates the feedrate at any given point for the motion

X= Y= Z= Displays the current X, Y, and Z location of the tool

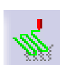
I= J= K= Denotes the I, J, and K components of the tool axis

Machining Time Indicates the time that the tool is cutting

Total Time Specifies the overall time of the process. This time represents the machining time, as well as any air time that the tool makes.

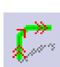


Replay Mode Controls how the tool paths are replayed. There are five different choices for the replay mode.

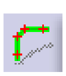


Continuous Replays from the current position and will not stop until it reaches the end of the operation

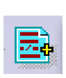
Plane by Plane Stops each time it moves to a different axial location. This option is best suited to allow you to visually inspect each axial pass of the tool.



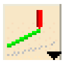
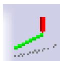
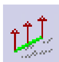
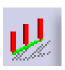
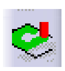
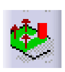
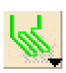
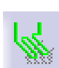
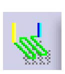
Feedrate by Feedrate Stops each time the feedrate is changed. This will allow you to inspect the tool motions without macro movements.



Point by Point Stops at each point along the tool path. This option is only useful when you want to carefully inspect each and every motion the tool makes.



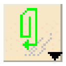
Post Processor Instruction Stops each time a new post processor instruction is found. If no post processor instructions are entered into the macros or operation, the tool will not stop.

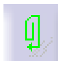
	*Sectioning of a Tool Path*	Specifies the tool path is sectioned at each level. This progression will go level by level going forward or backward at the click of a button.
	*Visualization Mode*	Controls how the tool is visualized throughout the replay
	*Last tool position*	Shows the last tool position replayed
	*Tool axis*	Shows the last tool position, as well as a tool axis vector at each location along the replay path. When in surface machining, you will be able to view all tool path motions as they are made
	*All tool positions*	Shows all tool locations. This mode is good if you want to inspect any particular location along the toolpaths for clashes
	*Machined Surface*	Specifies the machined surface is displayed in the sectioning mode
	*With all tool axes*	Specifies the tool axes are displayed as well as the machined surface in sectioning mode
	*Color Mode*	Controls the coloring of the toolpaths
	*Same Color*	Displays all toolpaths in the same color
	*Different Color*	Displays toolpaths in different colors for different feedrates. The different colors shown are as follows:

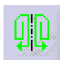
Yellow	Approach Feedrate
Green	Machining Feedrate
Blue	Retract Feedrate
Red	Rapid Feedrate
White	Local Feedrate
Light Blue	Finishing Feedrate
Pink	Plunge Feedrate

Note: These will also be the same color codes as what is used with the macro definition. Not all feedrates or colors are available for all operations.

 *Contact Point Display Mode* Allows surface machining operations to show either the tool center, or the tool contact point with the surface. For this class, this option will not be available or used.

 *TRACUT Display Mode* Controls whether TRACUTS are displayed or not

 *No TRACUT Displayed* TRACUTs are not shown in the replay

 *TRACUT Displayed* TRACUTs will be displayed

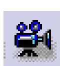
 *Holder Visibility Options* Allows you to specifies whether the holder is visible or not

 *Displays the holder* Holder is displayed in the replay

 *Hides the holder* Holder is hidden in the replay

 *Video Mode* Video mode is a method of replay that shows material removal as the tool makes passes. There are three different video modes.

 *Replay from last saved...* Displays the last saved video results, then replays the tool passes from that point

 *Full Video* Shows the entire simulation from start to finish within the part operation

 *Photo/Video Mode* Takes a photo mode snapshot of the part at the previous operation. This snapshot is then used as the initial starting point for the video removal simulation.

 *Save/Associate Video...*

Allows you to capture the video results at a particular moment in the replay, or at the end of the replay. There are two methods.

 *Associate video results...*

Allows you to save the video results with the operation. Only one video result can be associated with an operation at a time. When replaying from the last saved video result, this representation will be the starting point of the material removal.

 *Save video results in a...*

Saves a CGR (CATIA Graphics Render) image of the video results and a product containing the CGR. This CGR image can then be imported back into the assembly and used as stock material for another part operation.

 *Report Type*

Specifies the type of report

 *Video Collision Report*

Displays a report of all collisions from a video replay

 *Save Video Collision Report*

Saves the video collision reports to an external file

 *Video Parameters*

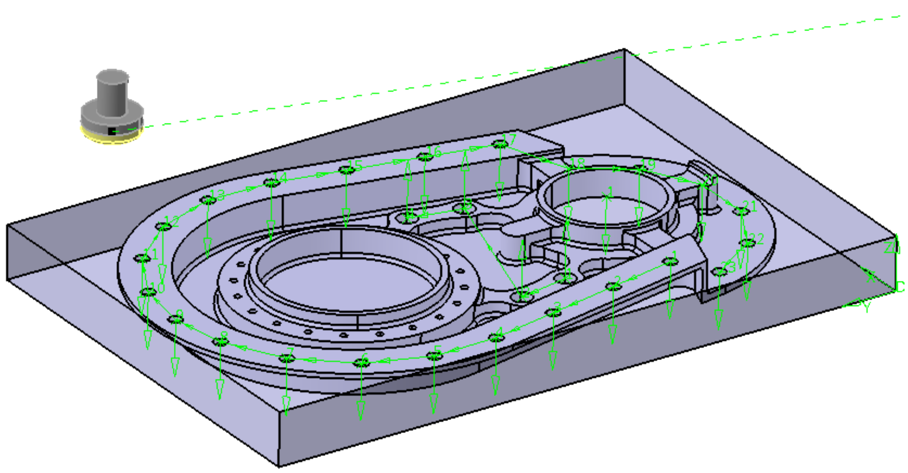
Allows you to adjust a few parameters for the video replay

 *Photo mode*

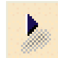
Displays the finished part after tool operations have been applied. This only shows the finished tool passes, not the intermediate steps.

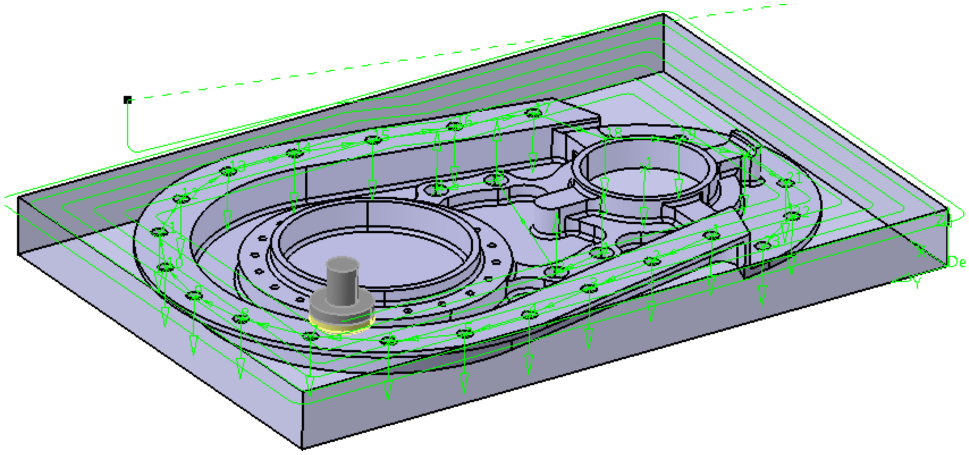
Analyze	Allows you to analyze the part for gouges, remaining material, and clashes once the tool paths are complete
Analyze	Displays graphical color gradients of remaining material and gouges
Video Measure	Allows you to measure video results for accuracy
Measure	Shows the cut part, and then show a series of measurement tools that will allow you to manually check the part for accuracy. This option is for use with photo results
Remove Chunks	Allows you to remove material from the CGR results that would normally fall off of the part
Check Reachability	Checks to insure that the machine has enough travel and in the case of a multi axis machine, enough rotation angles. This icon only works when Virtual NC, a product made by DELMIA, has been installed.
Starts Machine Simulation	Allows you to visualize the actual machine cutting the part rather than just the tool. This will only work if a machine is defined and available for the visualization.

Select the Start of Tool Path icon. When you first start a replay operation, the tool paths are shown. This will rewind and remove the tool visualizations.



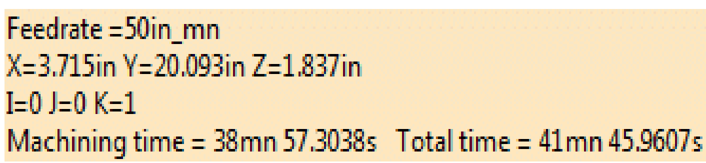
The only tool path displayed is the movement of the tool from the tool change point to the start of the tool paths. The transition motions are always shown in a dashed motion.

Select the Forward Replay icon.  The tool will pass over the part showing the tool path replay.

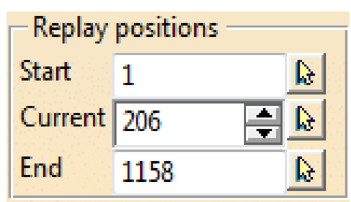


You will notice a number of things happening while the tool replays over the part.

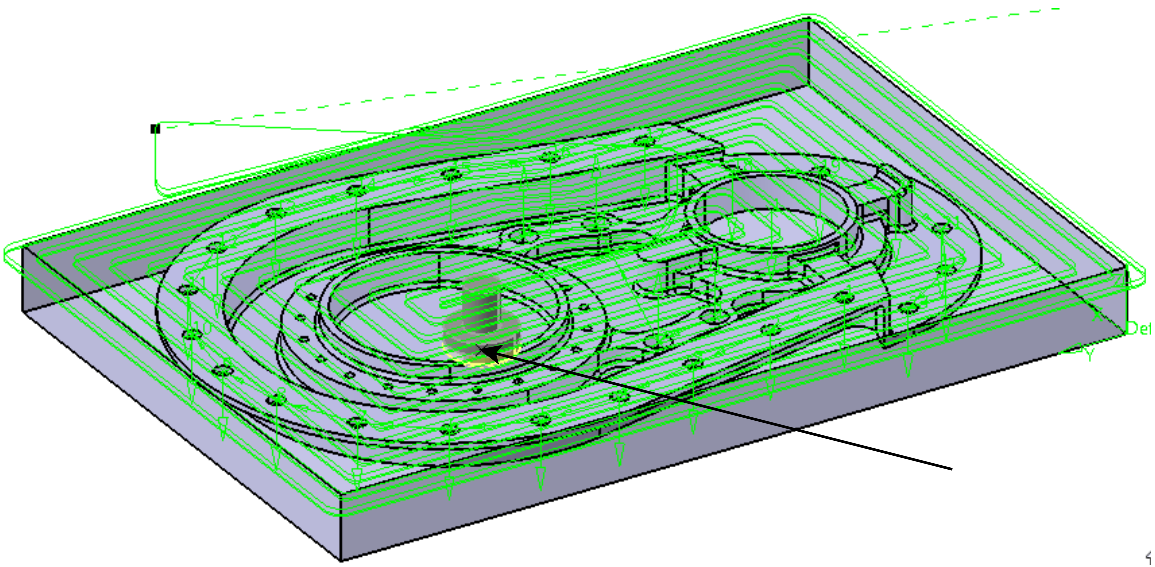
If you watch the replay window, you will notice the feedrates and coordinates change dynamically as the tool moves.



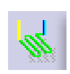
These values update contiguously until the end of the program. You will also notice the *Replay Position* updates constantly until the end of the program.

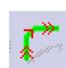


Once the tool is finished replaying, click your cursor on a portion of the tool path. Notice the tool shows at that location. This is a good method for you to visually inspect tool locations along the tool path. If the tool does not display near the point, you can turn the option on by going to *Tools*, *Options*, *Machining*, *General*, *Tool Path Replay*, and selecting *Display tool near cursor position on tool path*.



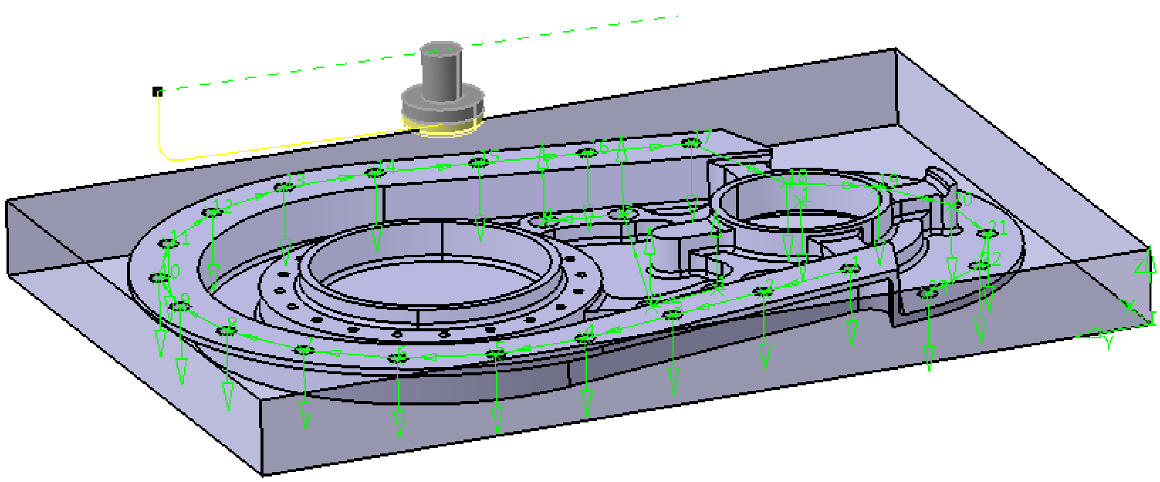
Having the tool follow the cursor can be beneficial to inspect tool position relative to the design part in a variety of different positions.

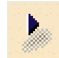
Set the color mode to Different Colors.  Notice the tool paths change to different colors. The various colors represent the different feedrates.

Set the replay mode to Feedrate by Feedrate.  This will make it so that the tool stops each time the feedrate is changed.

Press F5 on the keyboard. This will rewind the tool paths back to the beginning.

Press F7 to play forward. Notice the tool only moves a short distance, then stops. This is where the feedrate changes from one setting to another.



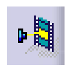
Select the Forward Replay icon.  This time the tool makes the complete level pass.

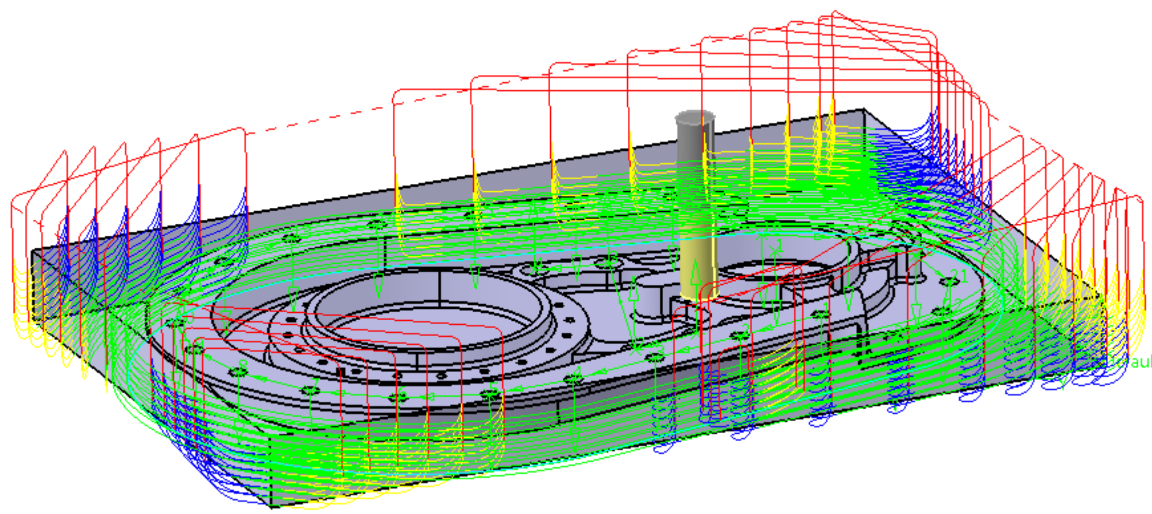
Select *OK* to exit the replay.

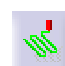
Playing your program feedrate by feedrate is a good way to check and insure the proper feedrates are used.

Expand the specifications tree so you can see the entire manufacturing program. You are going to want to be able to get to all of the operations.

While holding down the CTRL key, select all of the profile contouring machining operations. By selecting multiple machining operations at once, you can replay all of the tool paths at once.

Select the Replay Tool Path icon.  Take note of all of the tool paths.



Set the replay mode to Continuous, then rewind and replay the operations.  This will give you a good idea of the entire tool path process for the program.

While the tool paths are replaying, you can use the speed slider to increase the replay speed.

Replaying the tool paths is one thing, however, there will be many times when you want to see the resulting machined part. This is where some of the other replay modes come into play.

Select *OK* to the replay window. This will allow you to move on to the next replay mode.