TABLE OF CONTENTS

Introduction

DELMIA 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

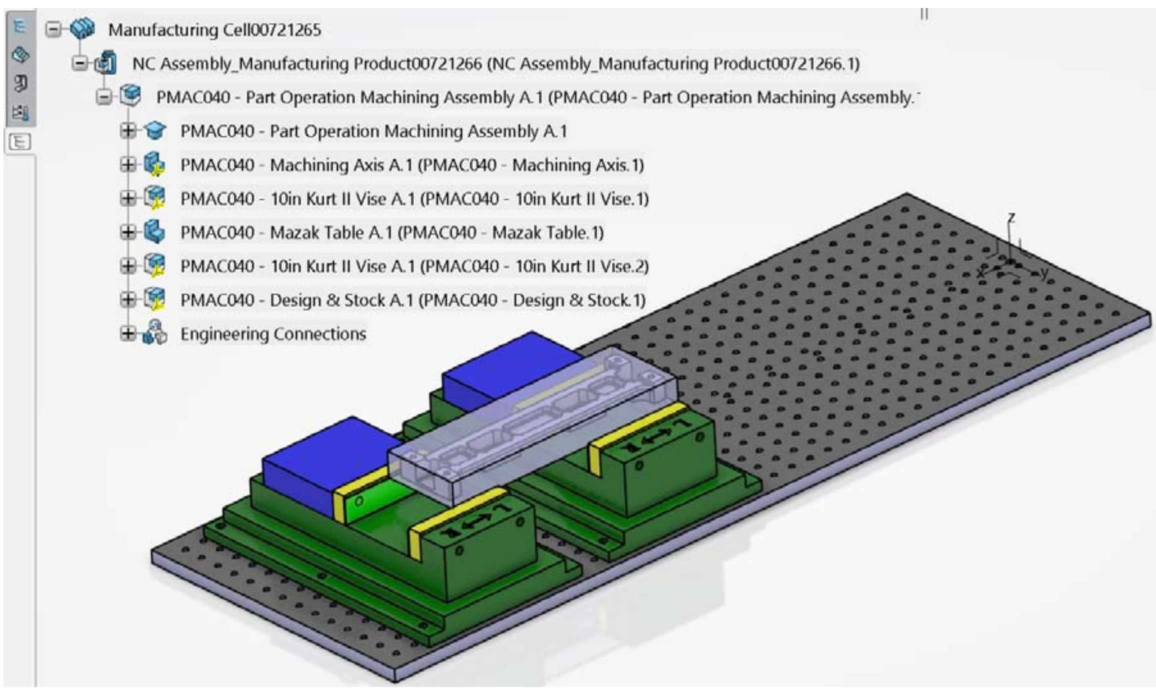
This section will investigate how to invoke the Prismatic Machining workbench and prepare your part for machining operations. Defining the part operation is a critical step for each machining process you start. Every time you prepare to machine a part, you must define the part operation.

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 PMAC040 - Part Operation Machining Assembly. By opening the assembly first, then switching to the prismatic machining workbench, you save the extra step of having to import the assembly.

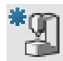
Switch to the Prismatic Machining workbench. This can be done by selecting the compass and selecting *Prismatic Machining*.

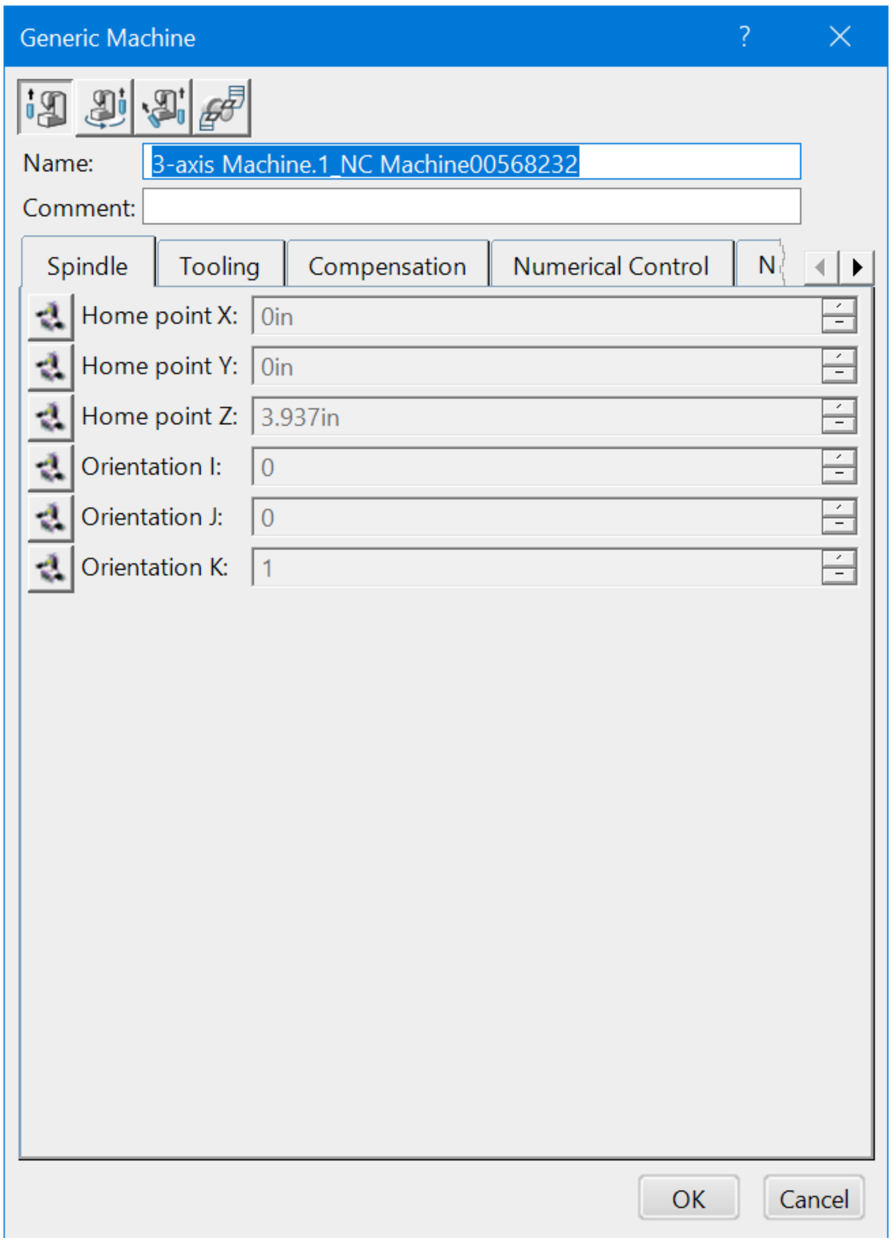
Now you are ready to begin defining the Part Operation.



Configuring NC Resources

There are a number of various machining parameters that can be set. It is very important that you define the machine. The machine definition specifies 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 Generic Machine icon.  The *Generic Machine* window appears. 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

 Mil-Turn Machine

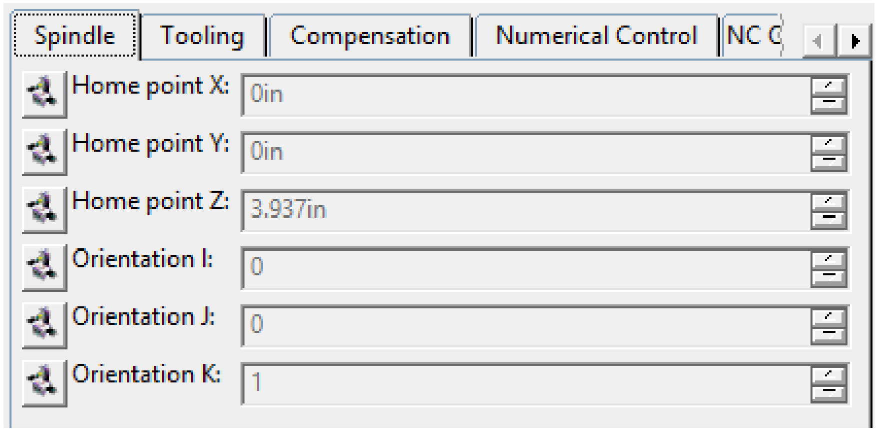
Based on the type of machine defined or selection, you will get various tabs to define the machine parameters. You will investigate the machine parameters for a simple 3-Axis machine.

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

Change the *Name* of the machine to PMAC040 - Fadal. The name of the machine is not extremely important. The most important part is that the machine parameters are defined.

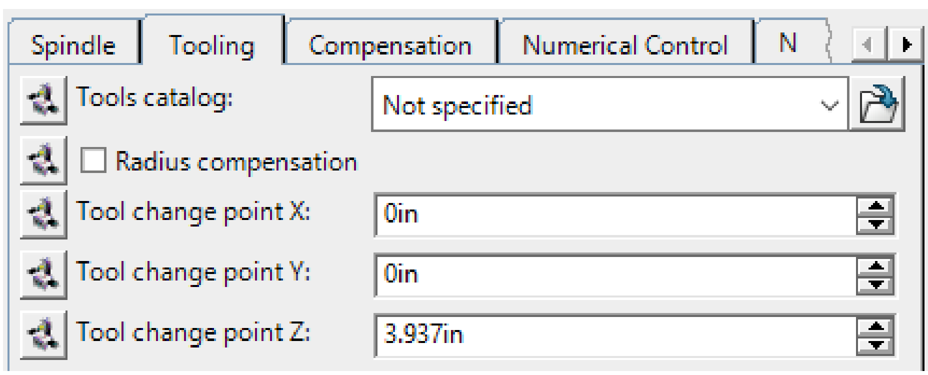
Take a moment to go over the various machine parameter tabs for the 3-axis machine.

Spindle



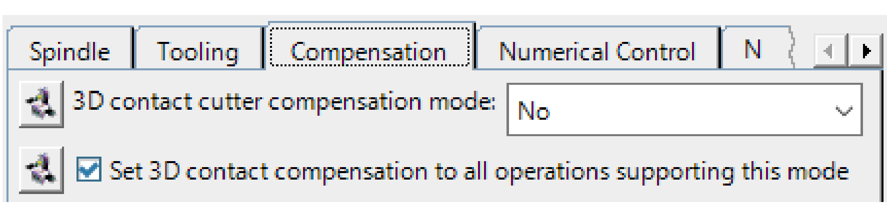
Home point X, Y, Z	Defines the X, Y, and Z coordinates of the tool home point
Orientation I, J, K	Defines the initial orientation of the tool

Tooling

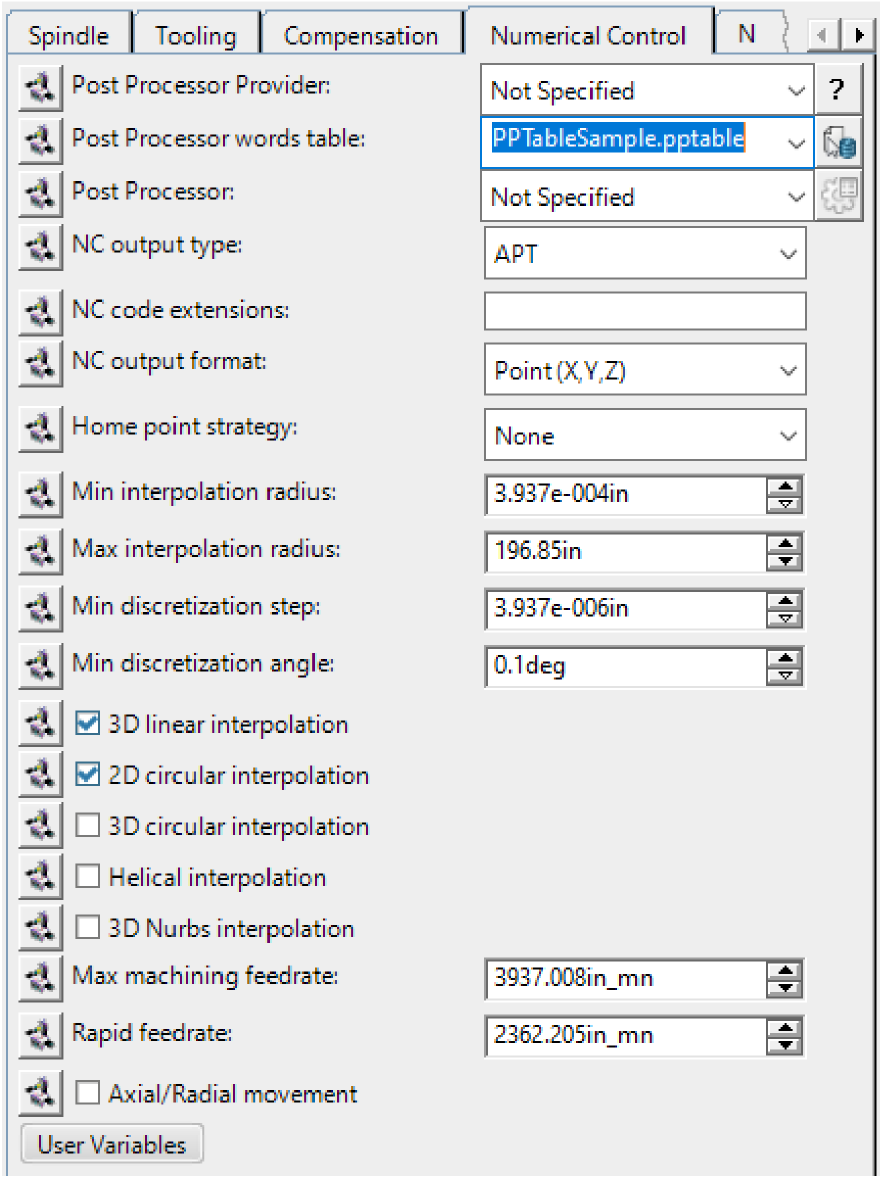


Tools catalog	Defines what tool catalog you will be using
Radius compensation	Toggles the radius compensation on or off for each tool
Tool change point	Defines the X, Y, and Z coordinates of the tool change point

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.

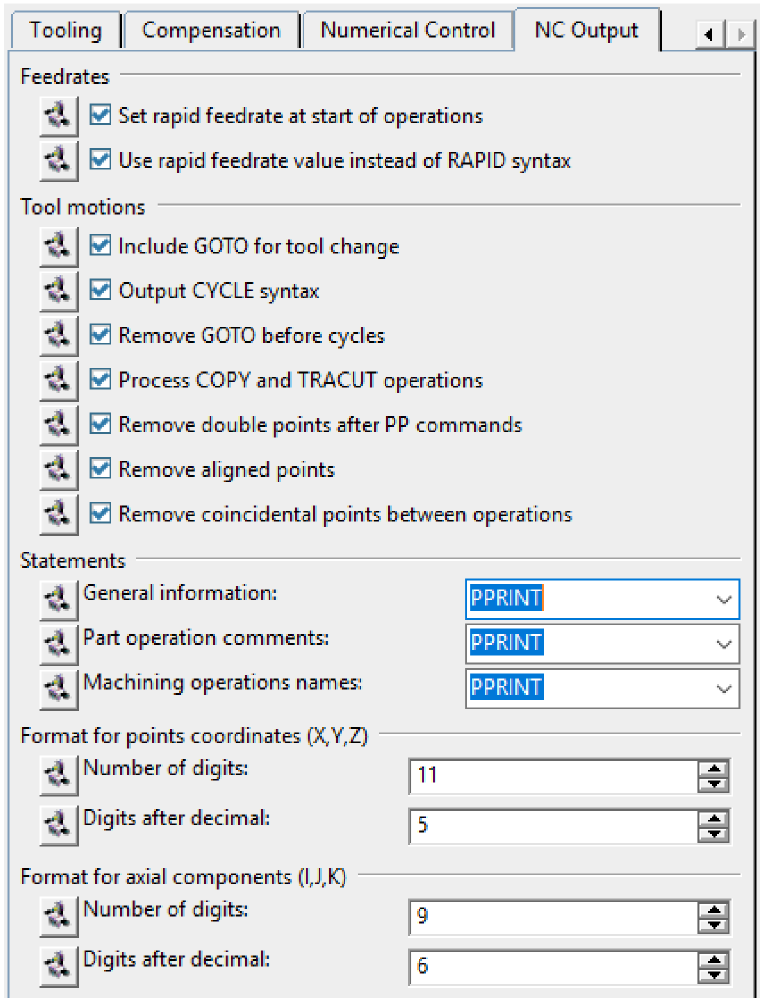


Post Processor	Allows you to define the specific post processor database to use. You must have a post processor vendor selected under *Tools*, then the *Machining* branch, and the *Output* tab.
Post Processor words table	Defines what post processor word table to use. The post processor words table defines the specific output format for the post processor such that all the commands are generated properly.
NC output type	Allows you to choose from APT, CLF, or ISO data types

NC output 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 interpolation	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)



Feedrates

Set rapid feedrate... Specifies whether a rapid feedrate will be included at the start of each operation. If a clearance macro is defined, it will take precedence.

Use rapid feedrate... Specifies the machine rapid feedrate will be used instead of the RAPID syntax

Tool motions

Include GOTO for... Specifies a GOTO statement will be included before each tool change

Output CYCLE syntax Specifies the PP word syntax in the PP word table will be output for axial machining operations. If the option is off, GOTO statements will be generated in the NC data output.

Remove GOTO before...	Specifies points added by clearance approach distance or jump distance will be removed
Process COPY and...	Specifies copy and/or tracut instructions will be processed and not included in the generated ATP source
Remove double points...	Specifies the first point after PP command is removed if the previous point is coincident
Remove aligned points	Specifies points that are aligned will not be output
Remove coincidental...	Specifies coincident points will not be output

Statements

General information	Specifies how information such as tool names and operation sequence numbers will be generated
None	Nothing will be generated
PPRINT	Generated with PPRINT word
$$	Generated as a comment
Part operation comments	Defines how part operation comments will be generated
Machining operations...	Defines how machining operation names will be generated
Format for points coord... (X,Y,Z)	Allows you to define other formats for NC data statements allowing better accuracy for large parts
Number of digits	Specifies the total number of digits for each point coordinate
Digits after decimal	Specifies the number of digits after the decimal point for each point coordinate
Format for axial components (I,J,K)	Allows you to define other formats for NC data statements allowing better accuracy for large parts

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. Notice the Home point parameters cannot be edited. You will have to modify these after the machine is created.

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.

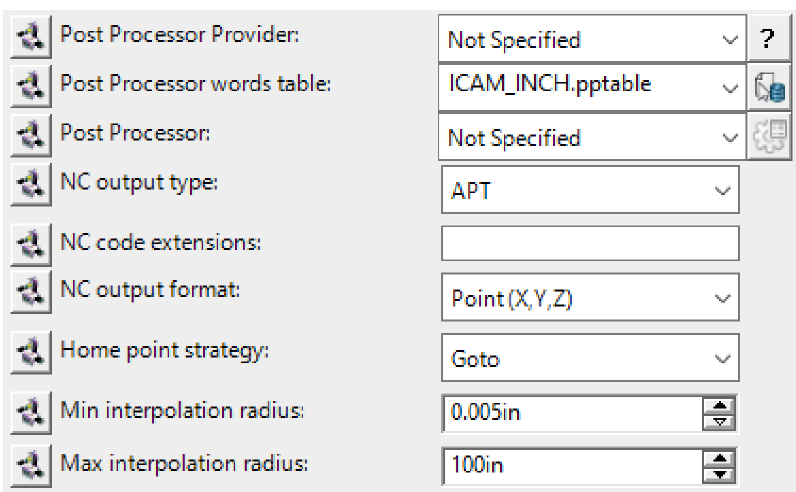
Switch to the *Numerical Control* tab, change the *Post Processor words table* to *ICAM_INCH.pptable*. This will allow you to utilize the ICAM post processor and insure that the output code will be in the proper format for the ICAM post processor. This will also insure that the output is in inches rather 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 interpolation radius* to 0.005in. This will cause the machine to output circle statements for any circle greater than 0.005in.

Change the *Max interpolation 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.

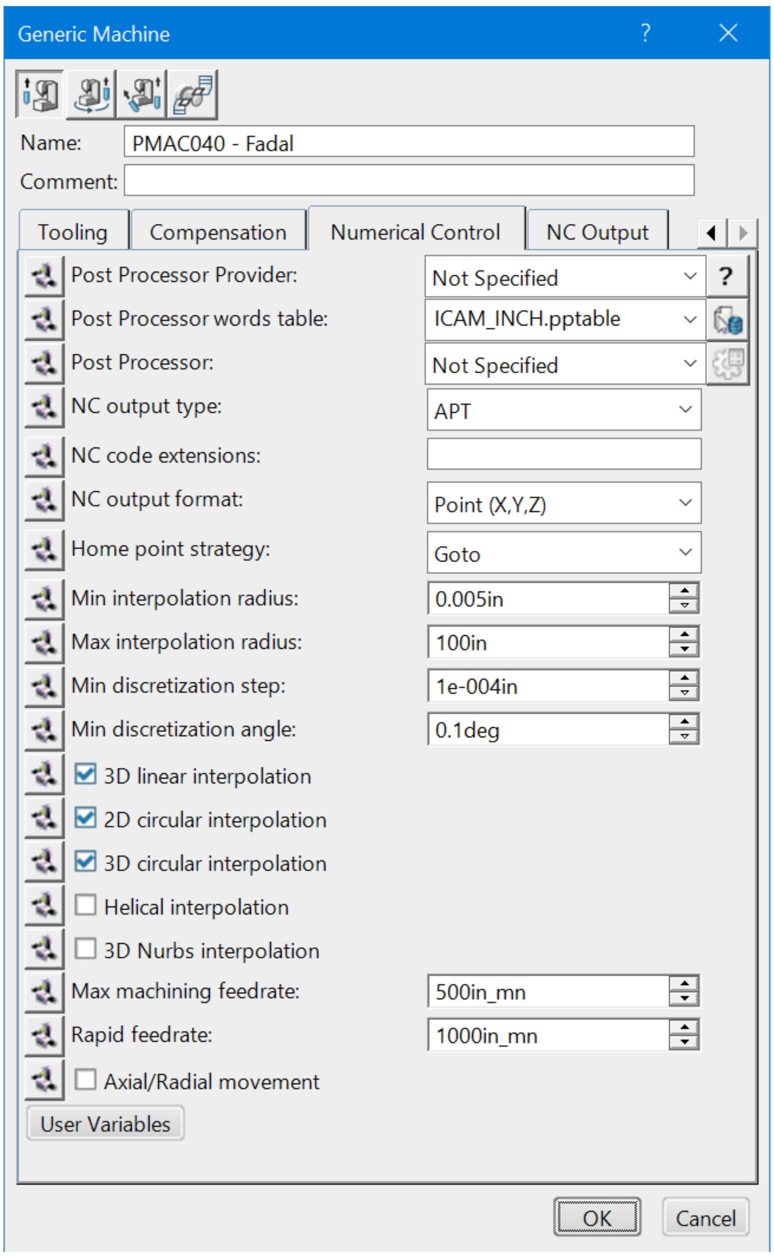
Turn on *3D circular interpolation*. This will indicate that when a circle is made that is not normal to the tool axis that the APT generator should still output a circular statement rather than a series of GOTO points.

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.

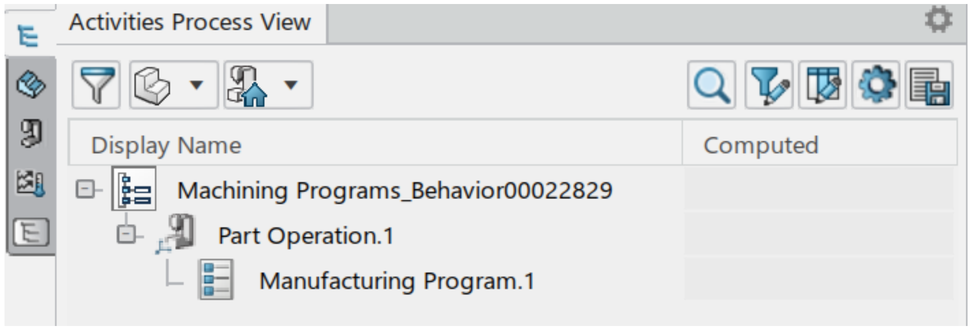
Set the *Rapid Feedrate* to 1000in_mn. Even though the APT generator will output a RAPID statement, this will provide CATIA with a method of calculating the amount of time it takes to move from one point to another while in rapid.

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.

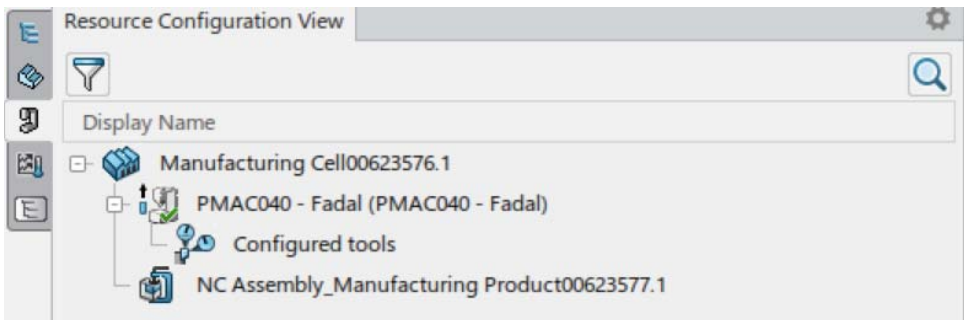
At this point, you have all the machine parameters completed.



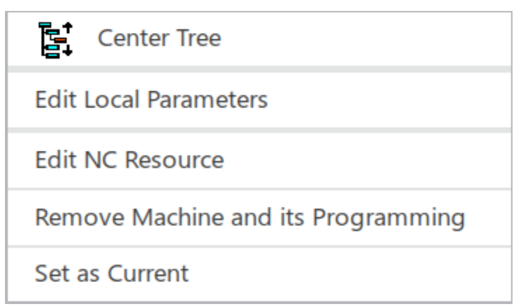
Select *OK* when done. This will have the machine set up. A part operation will appear in the tree.



Select the *Resource Configuration View* tab.



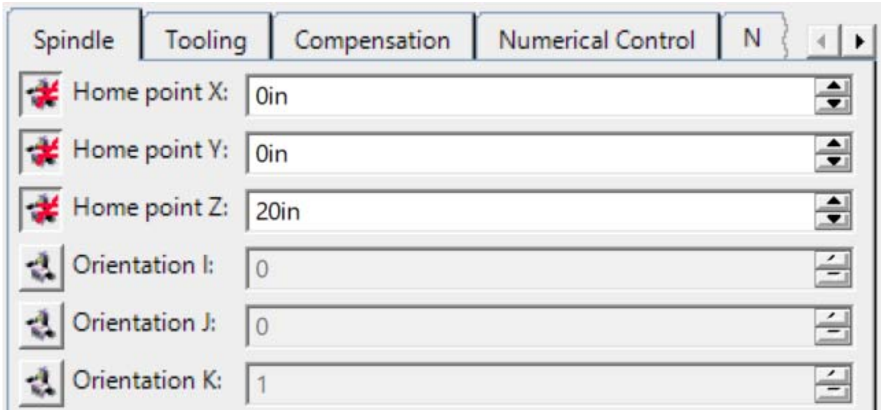
Right select on the *PMAC040 - Fadal* machine you created earlier. A contextual menu appears.



Select *Edit Local Parameters*. The *Local Modifications* of the machine window appears.

On the *Spindle* tab, select the Modify icon next to each of the Home point parameters and change the *Home point* to be set to (0,0,20in).  That is, make the X value 0 inches, Y value 0 inches and the Z value 20 inches. All coordinates will be relative to the machining axis system for this part operation. This is the location of the Spindle Bottom Center Point. 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.

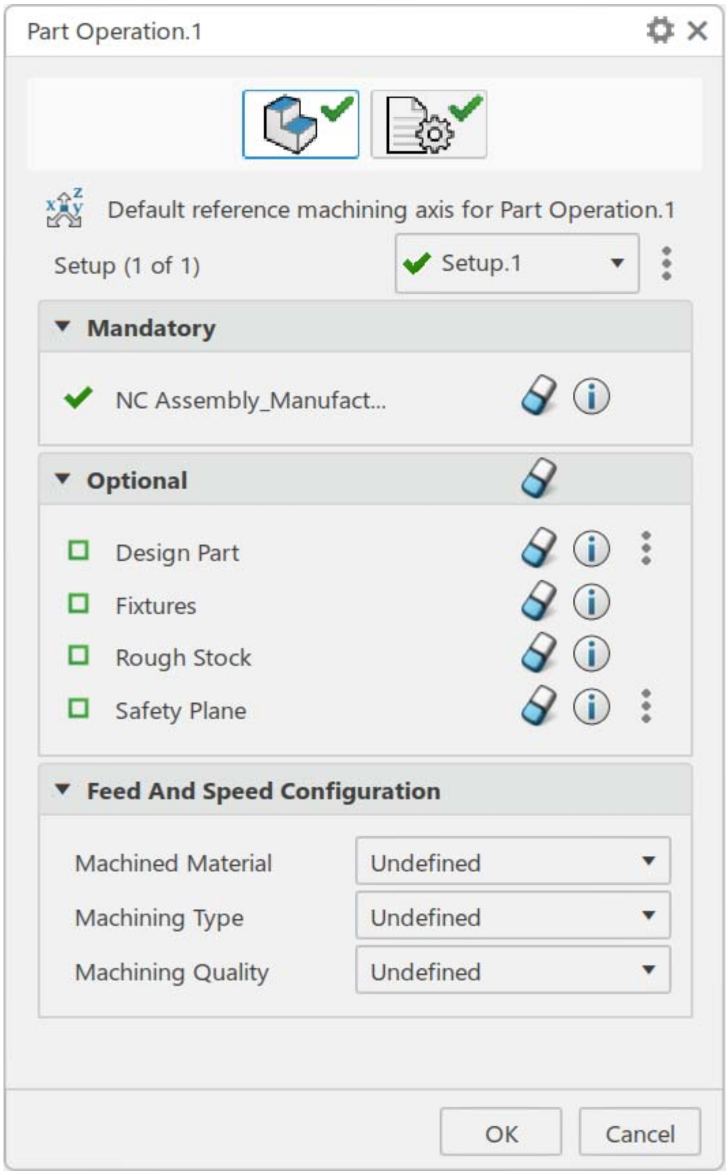
Leave the *Orientation* set at 0,0,1. This will set the axis to be along the K direction, or parallel to the Z axis.



Select *OK*. 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.

Defining the Part Operation

Switch to the *Activities Process Tree* tab and double select on *Part Operation.1*. 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.

 Geometry — Defines the machining axis, stock, safety plane, fixtures and design part for the part operation

 Defines the location and orientation of the machining axis

Mandatory

NC Assembly — Specifies the NC assembly

Optional

Design Part — Defines the design part for use in the material removal analyses

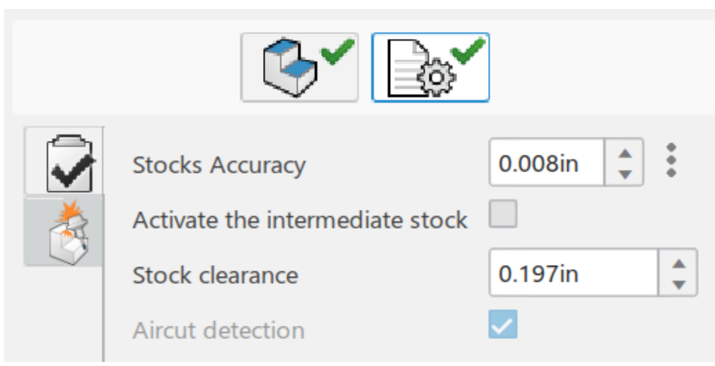
Fixtures — Allows you to define any fixtures around the part. During material removal simulation, machining the fixture parts will display red areas to indicate crashes.

Rough Stock — Defines the stock part around the design part. If a stock part is not selected, a significantly bigger rectangular block will be assumed.

Safety Plane — Defines the safety plane for the part operation

Feeds & Speeds... — Allows you to configure the feeds and speeds for the part operation

Strategy — Defines the options for the part operation



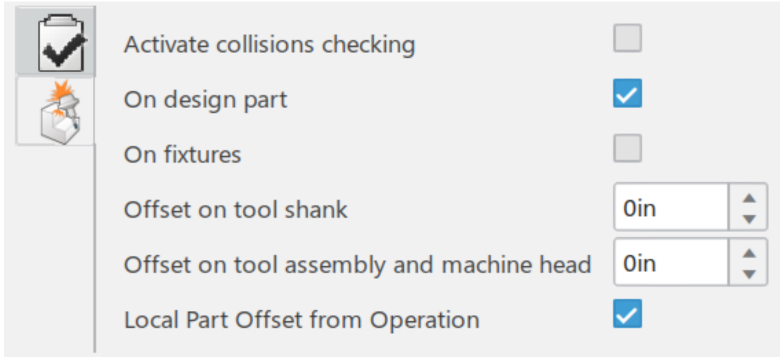
General

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.

Activate the inter... — Allows the intermediate stock to be automatically computed and taken into account for the tool path computation

Stock clearance — Specifies the default clearance value for the stock

Aircut detection — Specifies cutting air will be detected



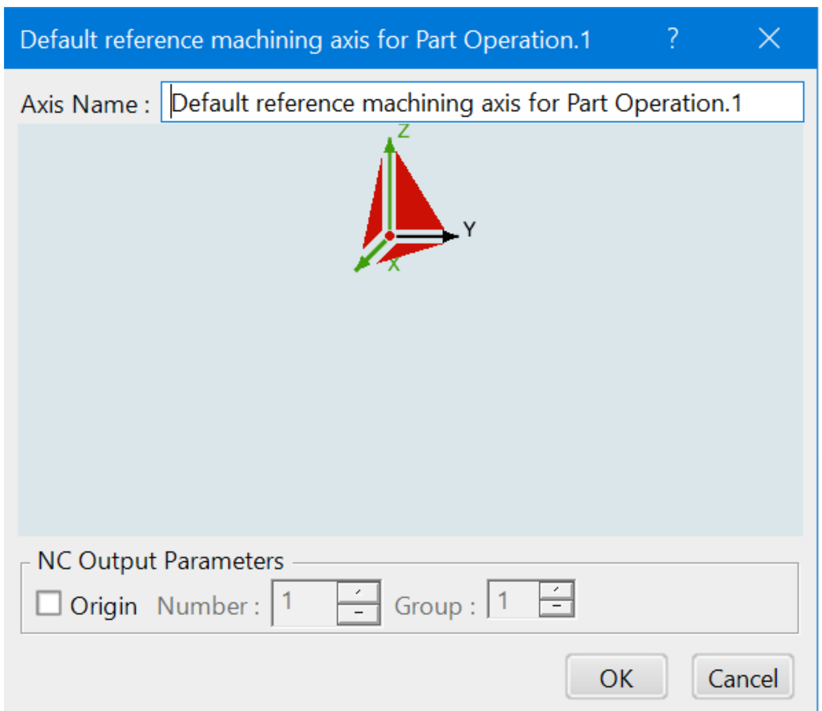
Collisions Parameters

Activate collisions...	Allows for quick feedback about collisions during the tool path replay
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
Local Part Offset...	Specifies a local part offset will be included

Machining Axis Definition

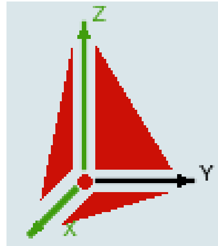
The machining axis is an important part of the NC Setup. The machining axis can generally be placed anywhere on the model that you want, however, there are a few locations that are better than others. You rarely want to have the machining axis buried or placed inside the stock material. You generally 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.

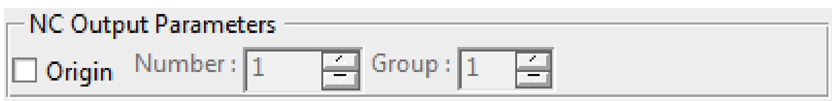


Take a moment to go over the various areas of the machining axis system window.

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. Keep in mind this is the location where you want to recognize the start of your first defined machining operation. It is where you will zero out the x, y, z location. Typical locations would be the table's zero axis, a corner of the fixture or a corner of the stock.



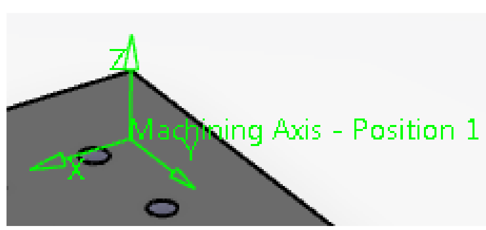
NC Output Parameters

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.

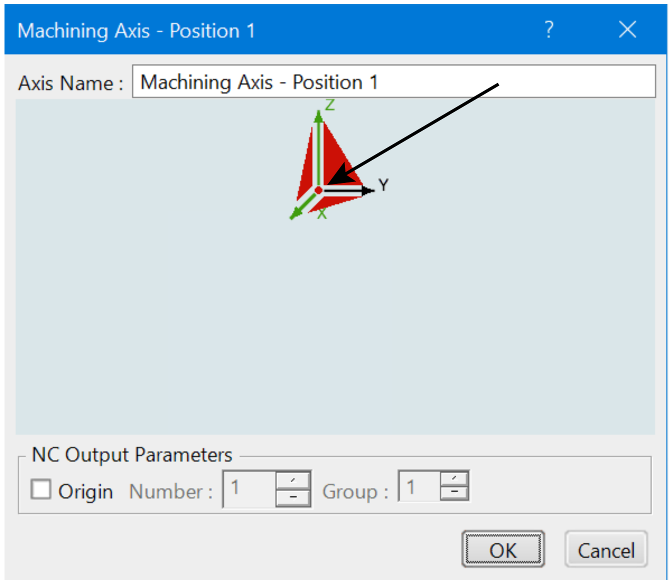
Now you are ready to define the axis location a few different ways.

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.



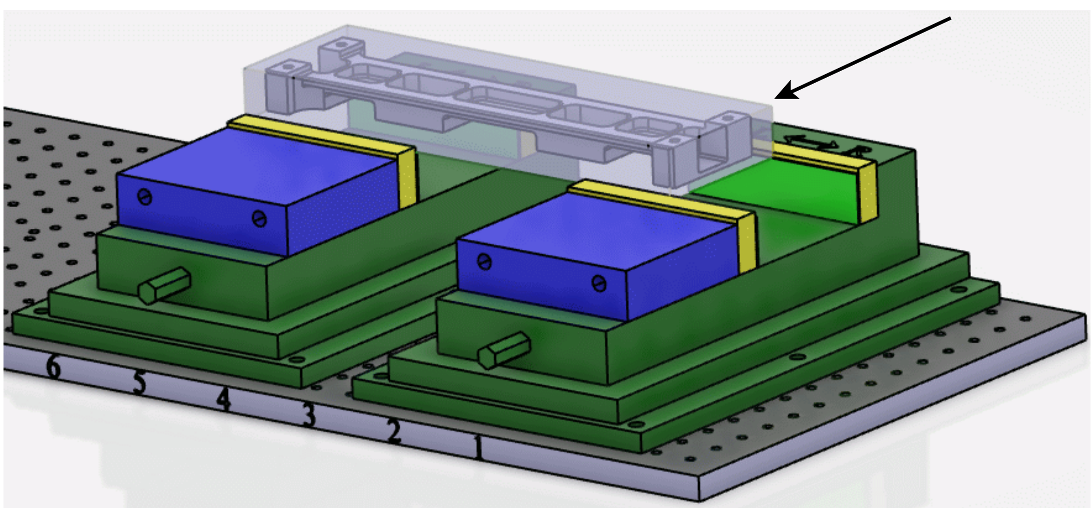
Properly identifying the axis systems will allow you to keep track of which axis systems are used for which part operations.

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



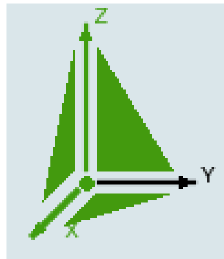
The center red dot will allow you to move the entire axis system from one location to another. The *Machining Axis System* window will disappear while CATIA waits for you to select a point or vertex to be the new center of the axis system.

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



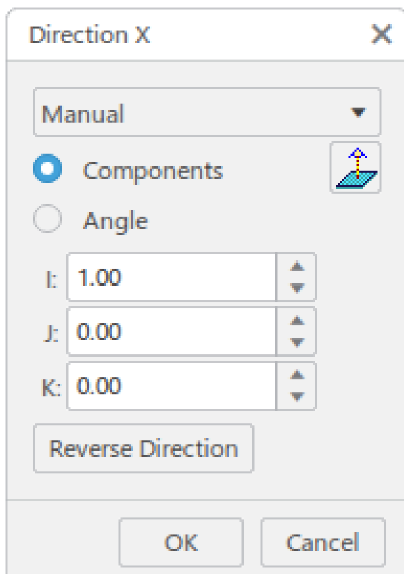
The machining axis system will move to the corner.

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.

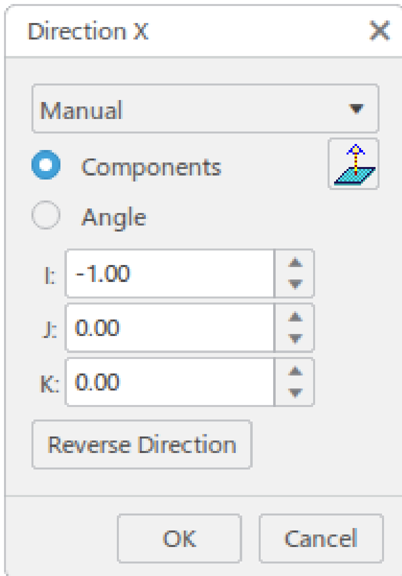
Select the X axis arrow from the machining axis system window. This will allow you to move the X axis. This will also display the *Direction* window.



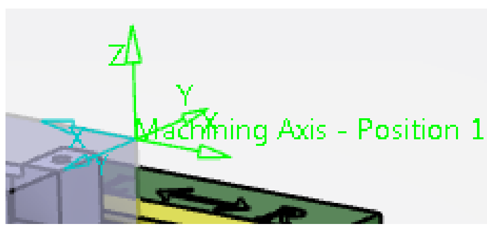
There are three methods to define the axis direction.

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.



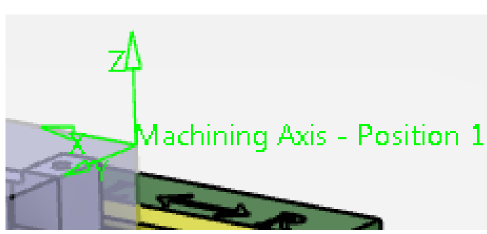
Notice that the axis does not appear to move.



If you look closely, you will notice blue 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.

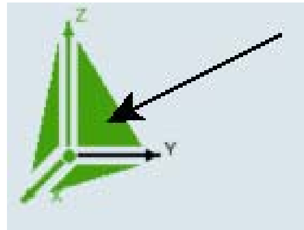
Select *OK* to the *Direction* window. This will display the machining axis window again.

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.

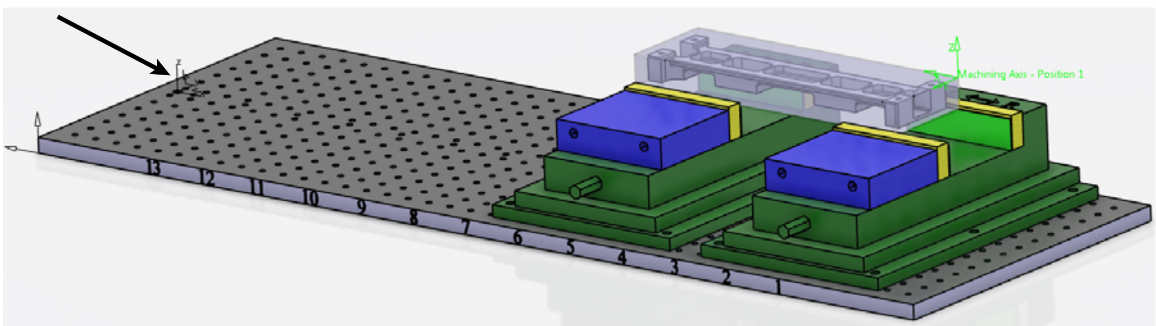
Select one of the planes of the axis system definition window. The planes are shown here.



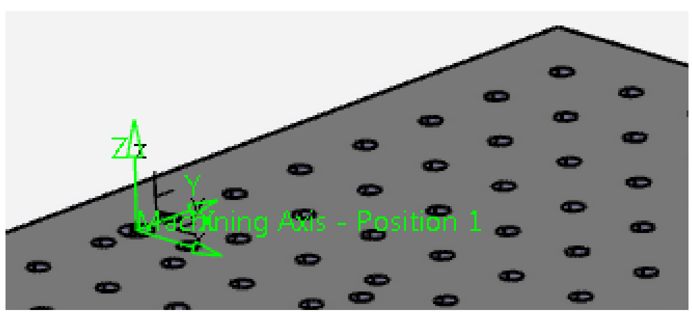
When you select the planes, the window will again disappear while CATIA waits for you to select an axis.

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.



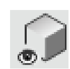
Select *OK* to the machining axis window. For now this will serve as the machining axis. Many times machine seeds will have an axis location defined that can serve as the machining axis.

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 *Rough Stock* option. 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. You can use the Hide/Show Stock icon.  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.

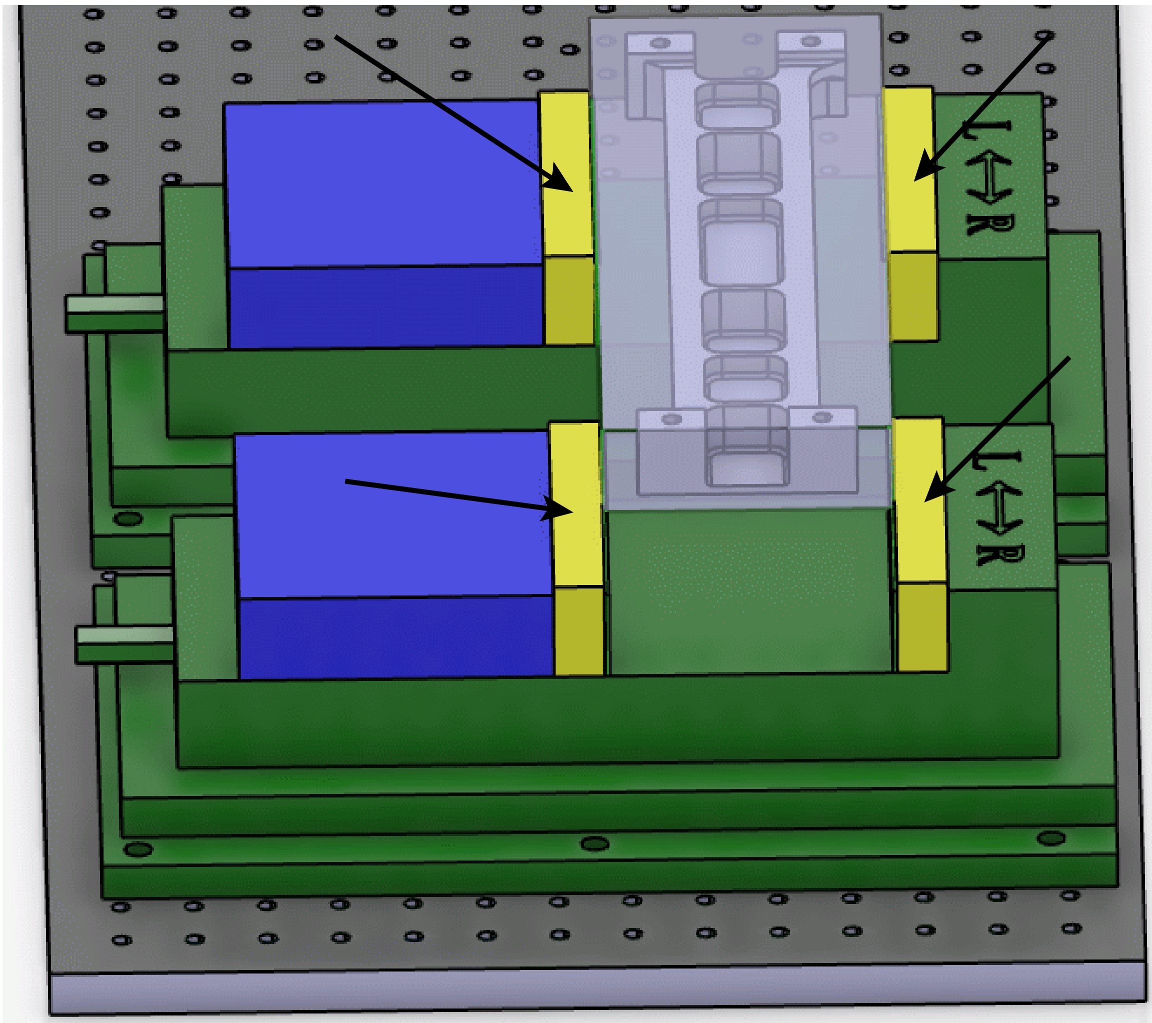
Select the *Design Part* option. This will define the final, as designed, part of the machining process.

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* option. 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.



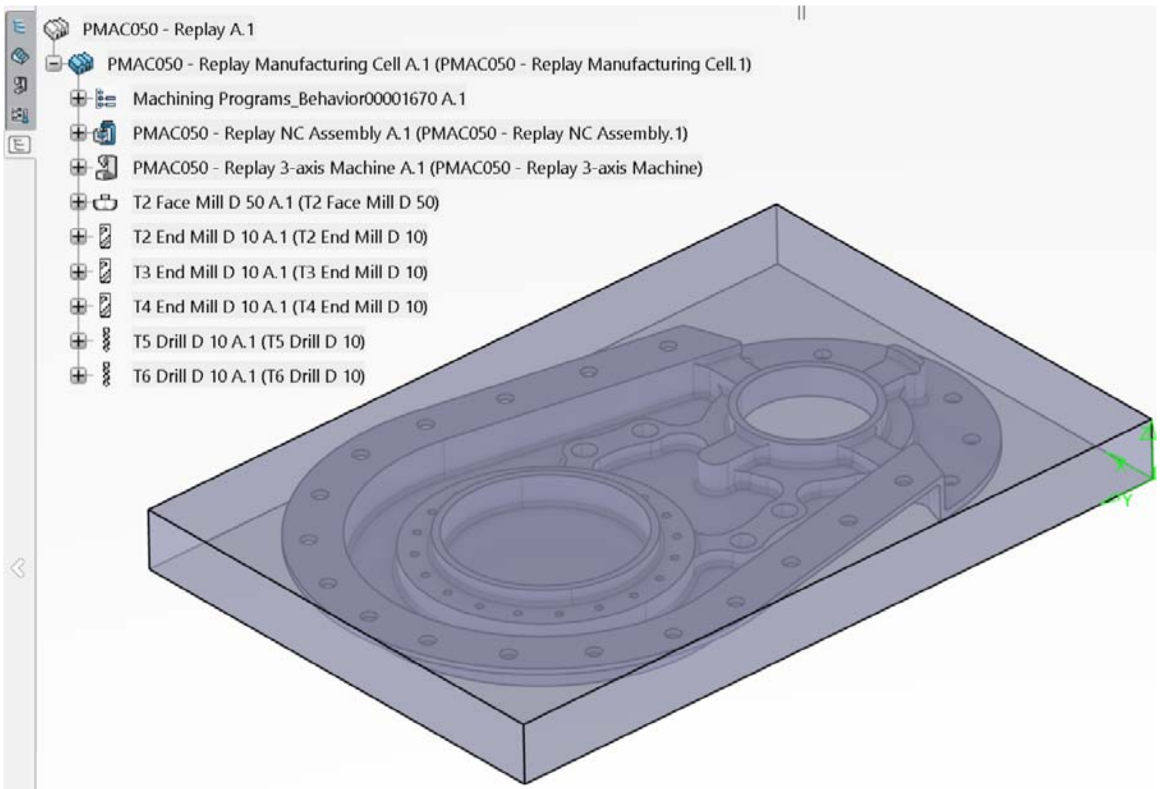
Double select in the workspace when done. You do not need to select on any part, just double select in space. You can define as many parts as fixture parts as you want. This will have four fixtures defined.

The last definition is the safety plane.

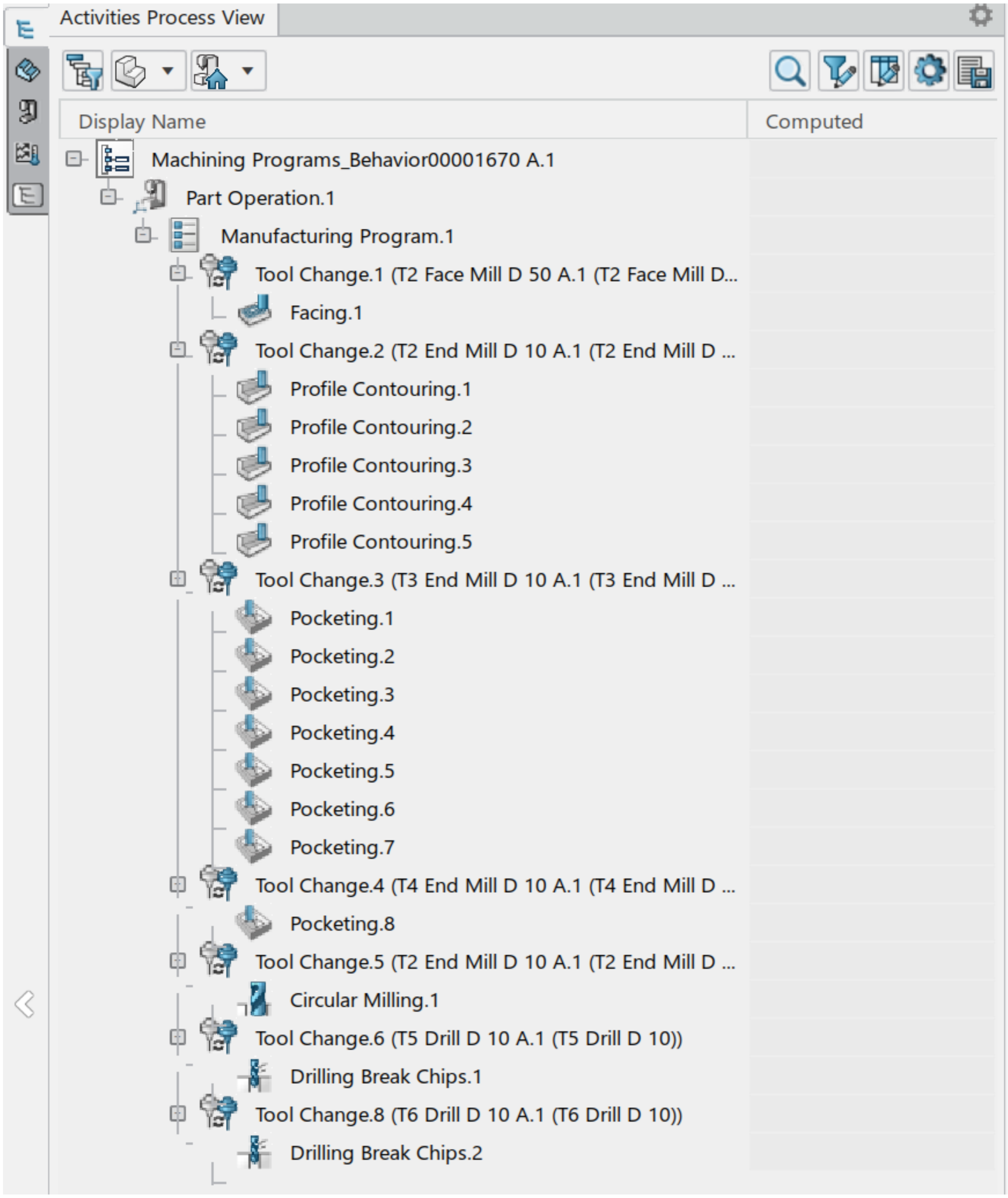
Replaying

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 PMAC050 - Replay document. 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.



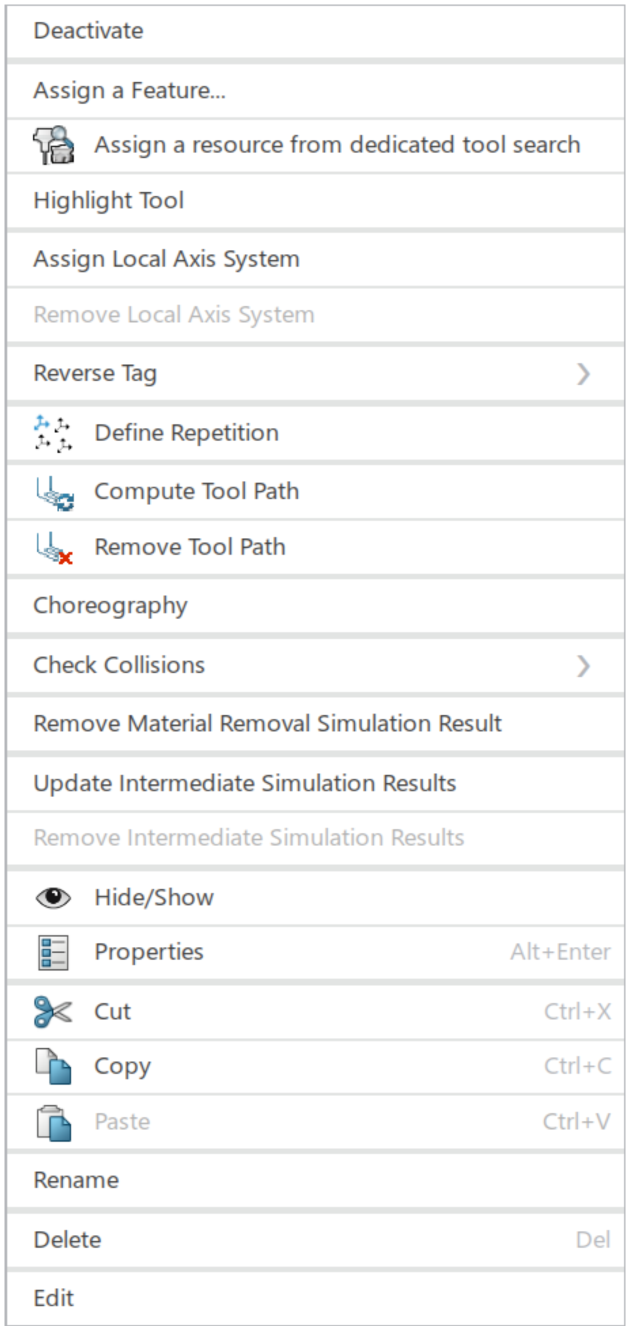
Select the *Activities Process Tree* tab. It should appear as shown.



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 the green computed symbol will show up in the *Computed* column.

Since none of the machining operations have the computed symbol, 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. This will display the contextual menu. The most important options to you, while replaying, are covered below.



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.

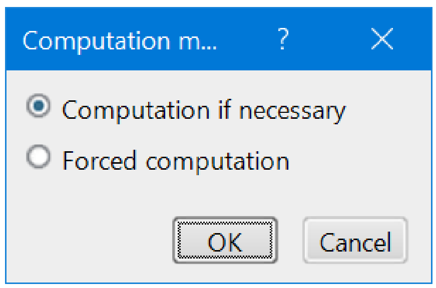
Remove Tool Path — Allows you to remove or strip the tool path out of the process. By removing the tool path, you will dramatically reduce the size of the file saved.

Remove Material Sim... — 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.

Update Intermediate... — Allows the intermediate simulation results to be computed

Remove Intermediate... — Allows the intermediate simulation results to be removed

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

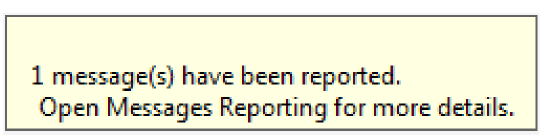


Computation if necessary — This will compute the tool paths if the status of the tool path is non-computed. Any tool paths with the *(Computed)* flag will not be re-computed.

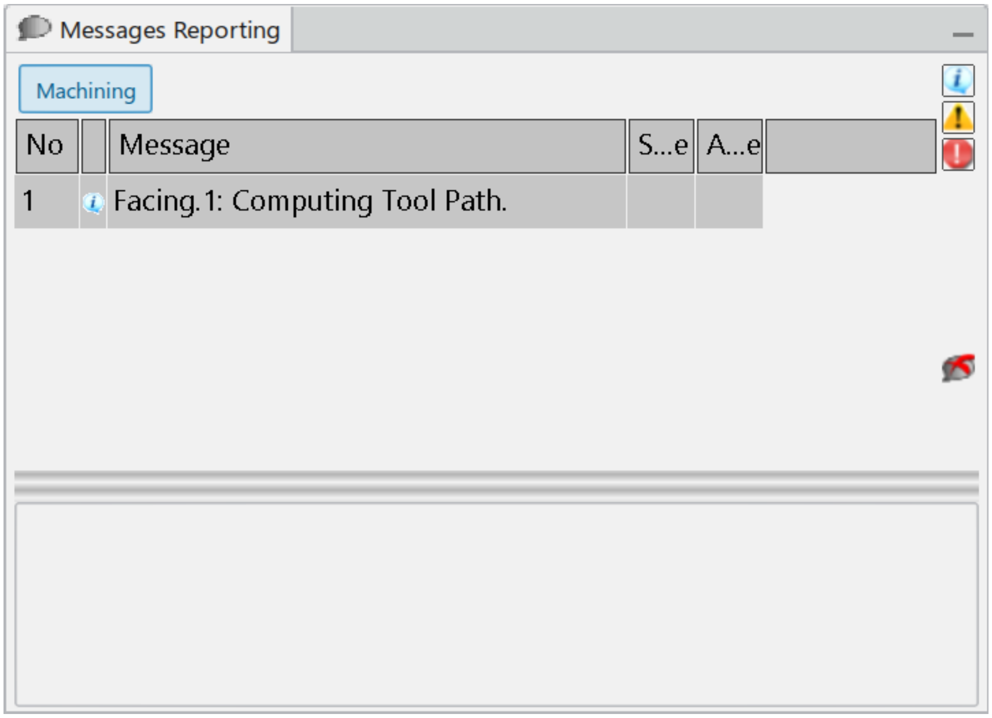
Forced computation — This mode will recompute all tool paths, regardless of the status of the tool path. Tool paths denoted as *(Computed)* will also be re-computed.

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.

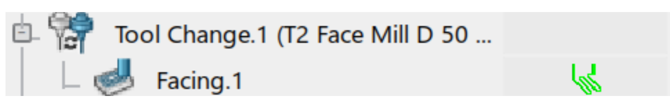
Select *Computation if necessary*, then click *OK*. Since the tool paths were not computed, they will automatically compute. An information window appears stating a message has been reported.



Select the Message Reporting icon on the right side of the display.  The *Messages Reporting* pane appears.

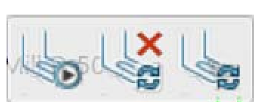


You may leave the *Messages Reporting* pane expanded if you like or you may close it. The icon will always be there to bring it back. Now the tool path has been computed.



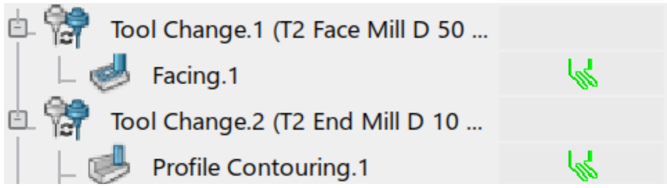
Note: If the operation shows as computed, yet the tool paths are not visible, you can turn them on by going to Preferences, Legacy Preferences, Machining, Output tab, and turning on the Tool Path Edition option.

Select on *Profile Contouring.1* in the list. A computation toolbar appears in the display.



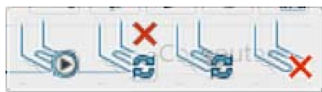
	Compute and Check Tool Path	Computes and displays the tool paths of the selected operations
	Force Compute Tool Paths	Forces computation of the tool paths
	Compute Tool Path if Necessary	Computes the tool paths of the selected operations if not already computed

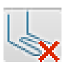
Select the Compute Tool Path if Necessary icon.  The tool path is computed.



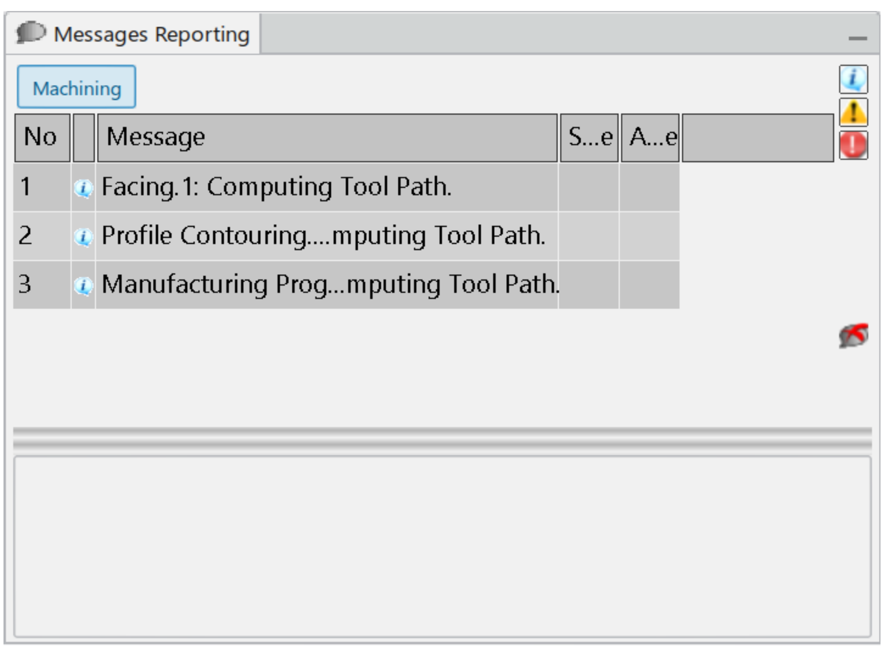
Computing the tool paths one at a time can become rather time consuming, especially when you have a lot of machining operations. Fortunately, you can also compute the tool paths by Manufacturing Program.

Select on *Manufacturing Program.1* from the tree. The computation toolbar appears again. This time there is an extra icon to remove tool paths as well.



 Remove Tool Path — Removes the computed tool paths of selected operations

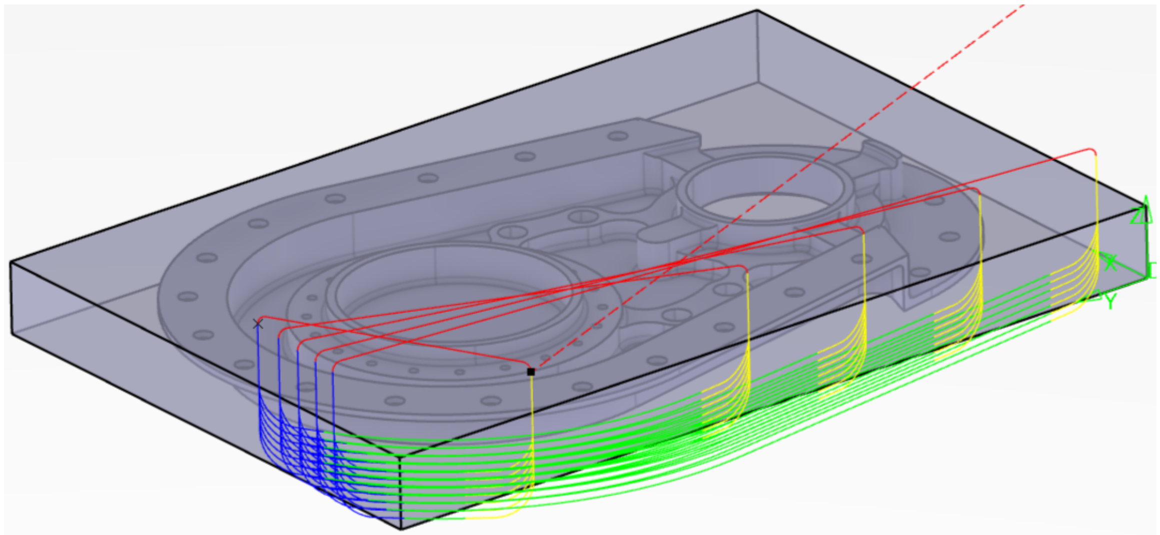
Select the Force Compute Tool Path icon.  This will now go through and compute all of the tool paths. For long programs this may take some time. Once done, the *Message Reporting* pane will display indicating the number of tool paths computed. Keep in mind you will have to expand it by selecting the Message Reporting icon on the right side of the display if you had minimized it before. 



Now the tool paths can be simulated.

Select *Profile Contouring.1* from the tree and select the Compute and Check Tool Path icon from the pop up toolbar.  This will display the tool paths on the screen, as well as show the *Play* toolbar at the bottom of the screen.

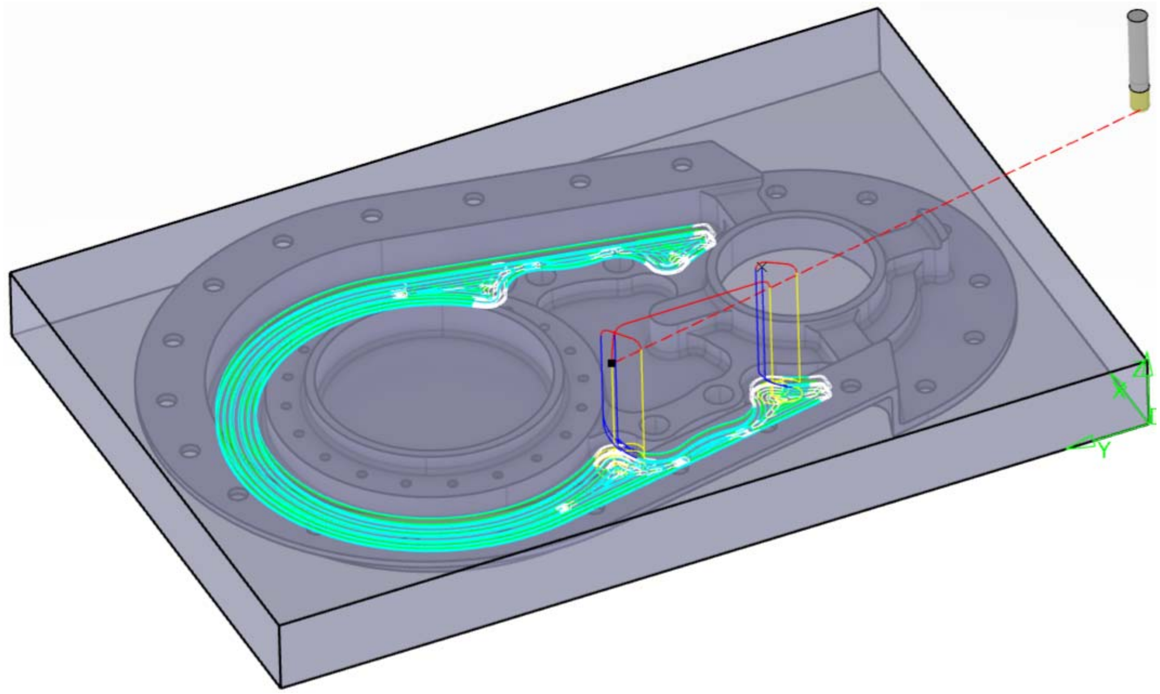
Note: If the tool paths aren't visible during the simulation, you can turn them on by going to Preferences, Legacy Preferences, Machining, Simulation tab and turning on the Tool Path Display option.



You will investigate the various aspects of the *Play* toolbar, shortly.

Select the Exit icon in the *Play* toolbar.  The toolbar closes and the tool paths disappear.

Select *Pocketing.8* and select the Compute and Check Tool Path icon.  The tool paths appear.



Select the Exit icon in the *Play* toolbar.  The toolbar closes and the tool paths disappear.

Now it is time to investigate the Play options a bit closer.

Select *Facing.1* and select the Compute and Check Tool Path icon.  The tool paths appear along with the *Play* toolbar.