

## TABLE OF CONTENTS

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

Tap	21
T-Slotter	22
Multi Diameter Drill	23
Two Sides Chamfering Tool	24
Boring and Chamfering Tool	25
Conical Mill	26
Thread Mill	27
Barrel Mill	28
Milling Directions	29
Conventional Milling	29
Climb Milling	30
Prismatic Machining Workbench	31
Specifications Tree	31
Prismatic Machining Workbench	34
Preparing to Machine	45
Part Design Review	45
Measurement Review	47
Assembly Review	51
Part Operation Setup	57
Configuring NC Resources	58
Defining the Part Operation	69
Machining Axis Definition	73
Geometry Definition	79
Position Definition	84
Resource Creation	86
Replaying	91
Design Part Comparison	107
Replacing a Machine	116
Simulation Scenario	122
Machining	135
Facing	136
Facing Geometry tab	139
Edge Selection	144
By Belt of Faces	145
By Boundary of Faces	145
Tool tab	148
Facing Strategy Parameters tab	150
Feedrate and Spindle Speed tab	170
Macros tab	173
Profile Contour Milling	191
Profile Contouring Geometry Parameters	192
Profile Contouring Modes	195
Profile Contouring Strategy Parameters	205
Profile Contouring Feedrate and Spindle Speed Parameters	216

Groove Milling .....	235
Groove Milling Strategy Parameters .....	236
Trochoid Milling .....	243
Trochoid Milling Strategy Parameters .....	245
Pocketing .....	249
Pocketing Strategy Parameters .....	255
Prismatic Roughing .....	285
Plunge Milling .....	299
Curve Following .....	337
Point to Point .....	344
Axial Machining .....	353
Spot Drilling .....	355
Geometry Tab .....	356
Drilling .....	370
Drilling Dwell Delay .....	373
Drilling Deep Hole .....	373
Drilling Break Chips .....	374
Tapping & Reverse Tapping .....	374
Thread Without Tap Head .....	375
Thread Milling .....	375
Boring .....	375
Boring and Chamfering .....	376
Boring Spindle Stop .....	377
Counter Boring .....	377
Back Boring .....	378
Reaming .....	378
Counter Sinking .....	378
Chamfering 2 Sides .....	378
T-Slotting .....	379
Circular Milling .....	380
Sequential Axial .....	382
Sequential Groove .....	387
Patterns .....	391
Part Design Patterns .....	391
Machining Patterns .....	392
Multiple Part Operations .....	401
Single Part, Two Assemblies .....	402
Multiple Parts, Single Assembly .....	415
Fixtures .....	429
Clamps .....	429
Tabs .....	442
Horizontal Tabs .....	442
Vertical Tabs .....	457

---

Transformations .....	469
Copy-Transformation .....	470
COPY Operator Instruction .....	479
TRACUT Operator Instruction .....	490
Opposite Hand Machining .....	494
Post Processor Instructions .....	500
Manual Tool Changes .....	505
NC Code Generation .....	523
Practice Problems .....	531
Appendix A .....	545
Machining - General .....	545
Machining - Resources .....	547
Machining - Operation .....	549
Machining - Output .....	550
Machining - Program .....	551
Machining - Material Removal .....	552
Machining - Simulation .....	554

## **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

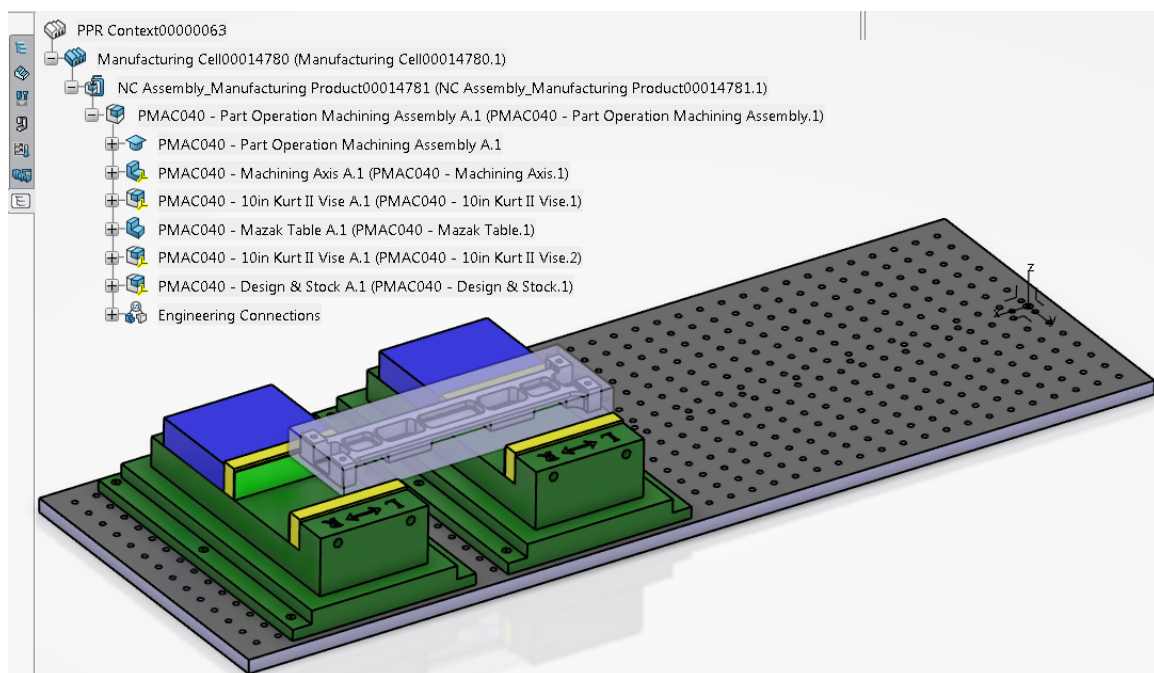
This section will investigate how to invoke the Prismatic Machining workbench and prepare your part for machining operations. Defining the part operation is a critical step for each machining process you start. Every time you prepare to machine a part, you must define the part operation.

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 *My Simulation Apps (South)* quadrant of the compass and selecting *Prismatic Machining*.


Now you are ready to begin defining the Part Operation.

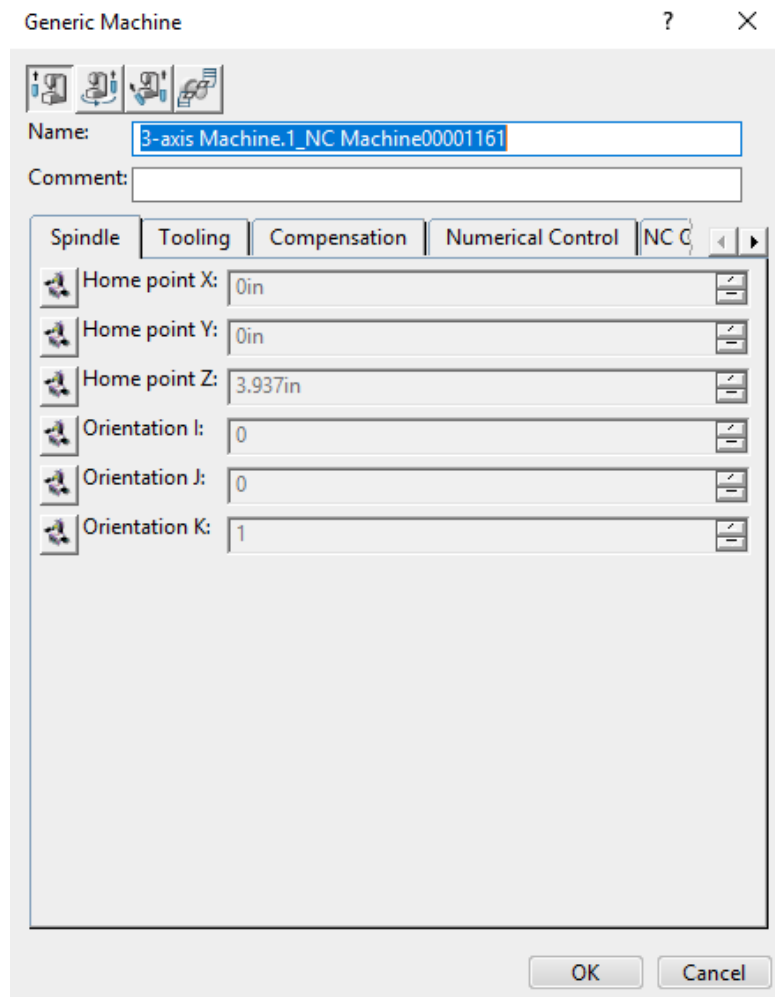


If you remember from the introduction, the Prismatic Machining workbench utilizes the PPR tree, or the Process Product Resource tree. Refer back to the introduction for full details on the PPR tree.

## 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.

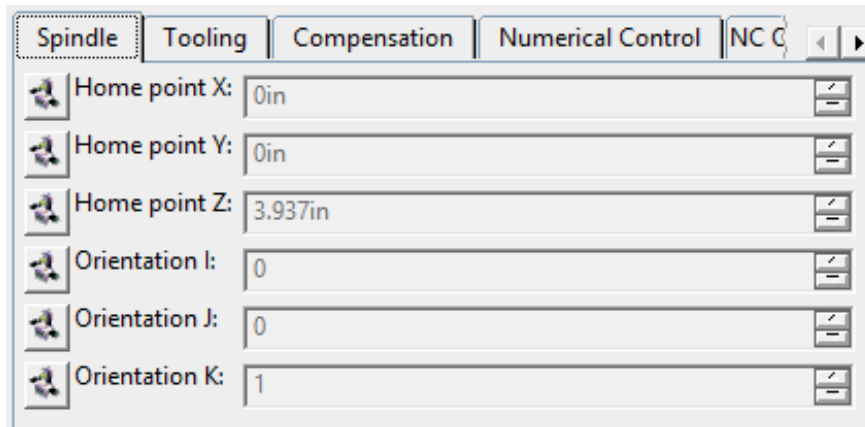
**Select the 3-axis Machine icon.**  This is the default option and may already be selected.

**Change the *Name of the machine* to 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.



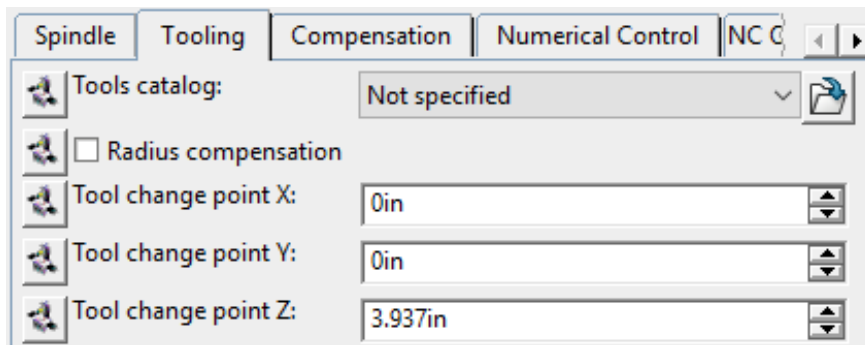
Spindle



*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

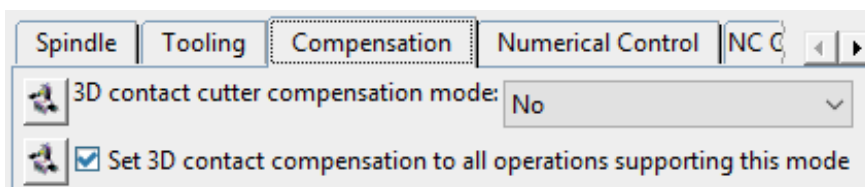


*Tools catalog* Defines what tool catalog you will be using

*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

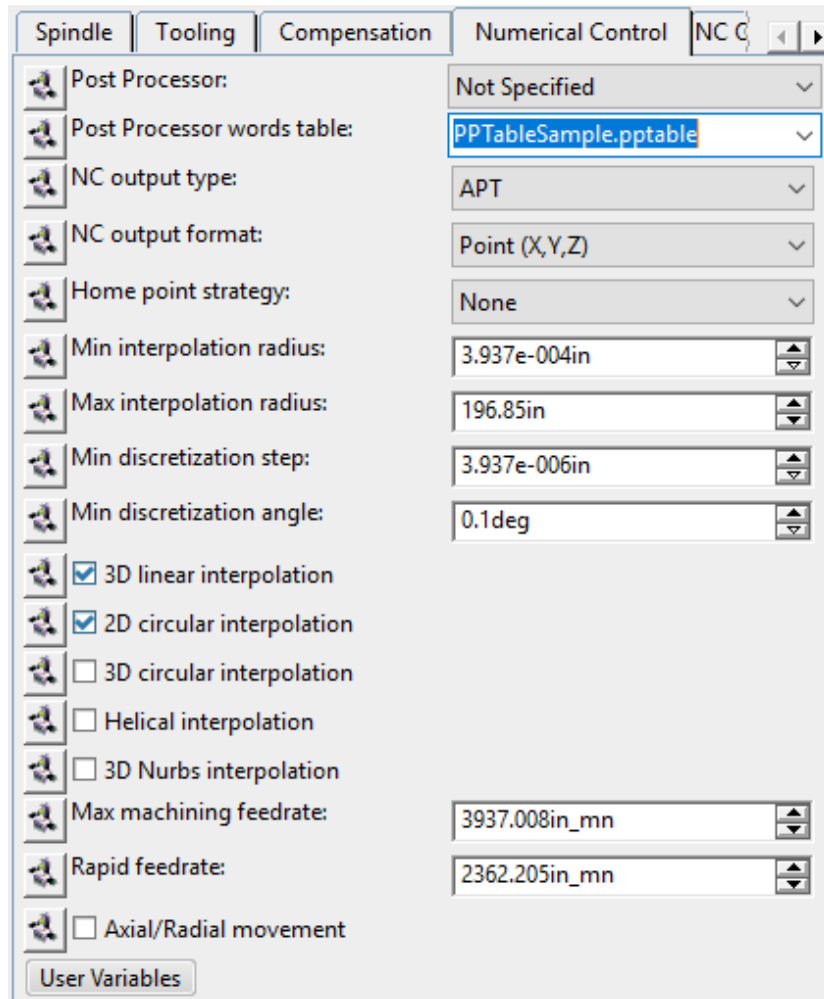
Compensation



*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.

Numerical Control



*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.

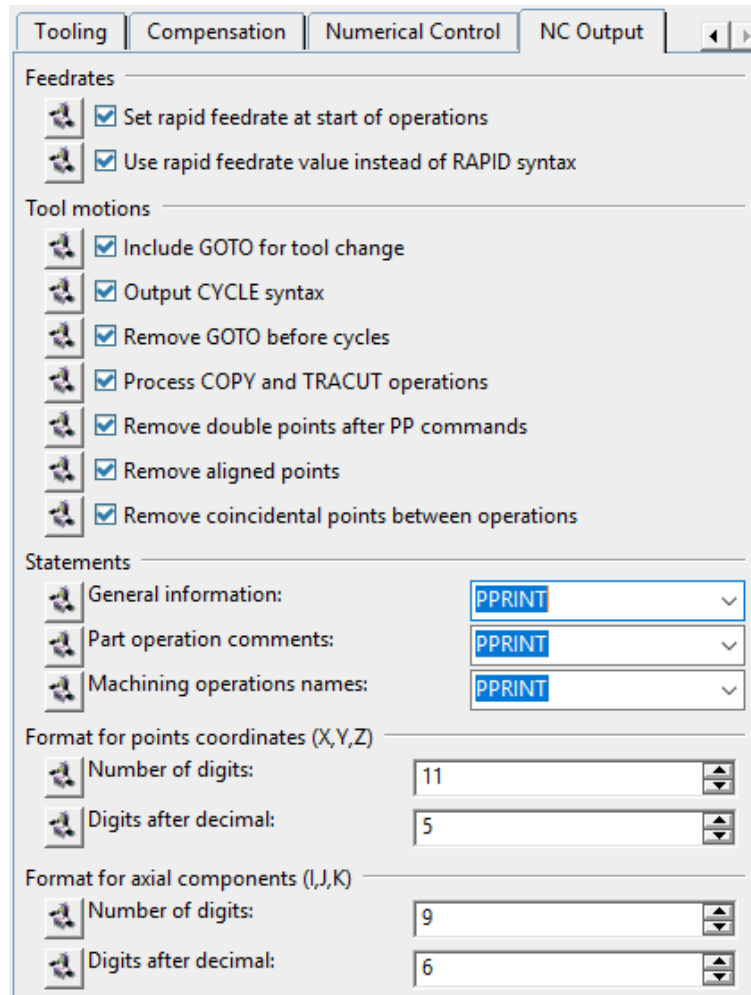
*Home point strategy* Indicates if the home point is the current tool location (*From*) or if it needs to move to the location (*Goto*)

<i>Min &amp; Max interpol. radius</i>	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.
<i>Min discretization step/angle</i>	Defines the minimum motion distance and minimum angle that will generate a GOTO statement in the APT code
<i>3D linear interpol.</i>	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.
<i>2D &amp; 3D circular interpol.</i>	When checked, either 2D or 3D circles will be interpolated, or both.
<i>Helical interpolation</i>	Specifies the ability to make a helical interpolation between two consecutive points.
<i>3D Nurbs interpolation</i>	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.
<i>Max machining feedrate</i>	Defines the maximum machining feedrate that will be allowed
<i>Rapid feedrate</i>	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.*

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

NC Output



*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.

<i>Remove GOTO before...</i>	Specifies points added by clearance approach distance or jump distance will be removed
<i>Process COPY and...</i>	Specifies copy and/or tracut instructions will be processed and not included in the generated ATP source
<i>Remove double points...</i>	Specifies the first point after PP command is removed if the previous point is coincident
<i>Remove aligned points</i>	Specifies points that are aligned will not be output
<i>Remove coincidental...</i>	Specifies coincident points will not be output
<i>Statements</i>	
<i>General information</i>	Specifies how information such as tool names and operation sequence numbers will be generated
<i>None</i>	Nothing will be generated
<i>PPRINT</i>	Generated with PPRINT word
<i>\$\$</i>	Generated as a comment
<i>Part operation comments</i>	Defines how part operation comments will be generated
<i>Machining operations...</i>	Defines how machining operation names will be generated
<i>Format for points coord... (X,Y,Z)</i>	Allows you to define other formats for NC data statements allowing better accuracy for large parts
<i>Number of digits</i>	Specifies the total number of digits for each point coordinate
<i>Digits after decimal</i>	Specifies the number of digits after the decimal point for each point coordinate
<i>Format for axial components (I,J,K)</i>	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 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.

Post Processor:	Not Specified
Post Processor words table:	ICAM_INCH.pptable
NC output type:	APT
NC output format:	Point (X,Y,Z)
Home point strategy:	Goto
Min interpolation radius:	0.005in
Max interpolation radius:	100in

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

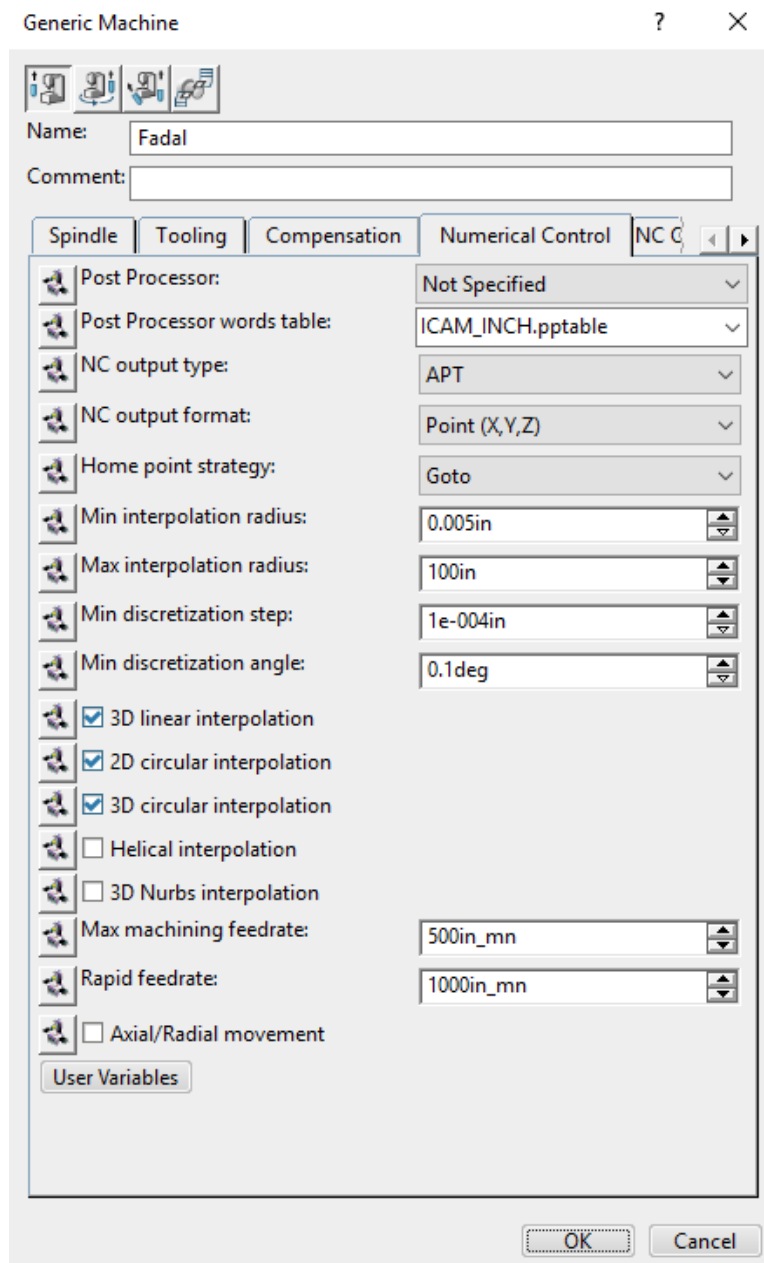
**Turn on *3D circular interpol*.** This will indicate that when a circle is made that is not normal to the tool axis that the APT generator should still output a circular statement rather than a series of GOTO points.

**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.

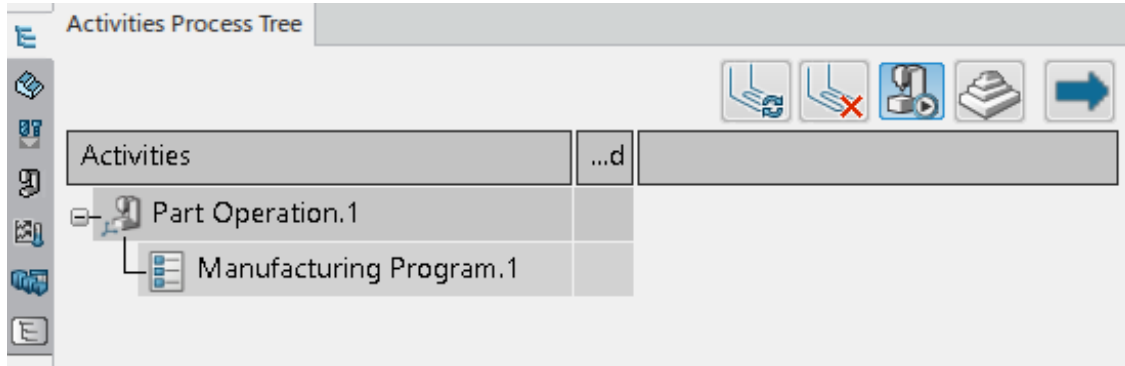
**Set the Rapid Feedrate to 1000in\_mn.** Even though the APT generator will output a RAPID statement, this will provide CATIA with a method of calculating the amount of time it takes to move from one point to another while in rapid.

**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.

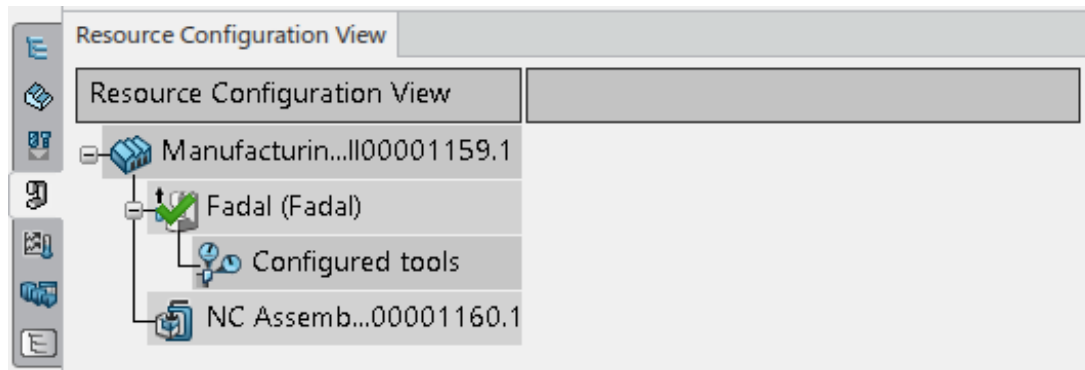
At this point, you have all the machine parameters completed.



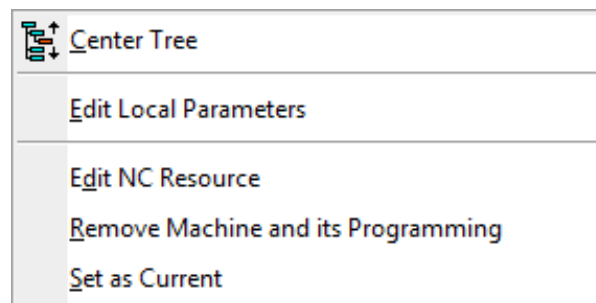
Select **OK** when done. This will have the machine set up. A part operation will appear in the tree.




Select the **Resource Configuration View** tab.



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

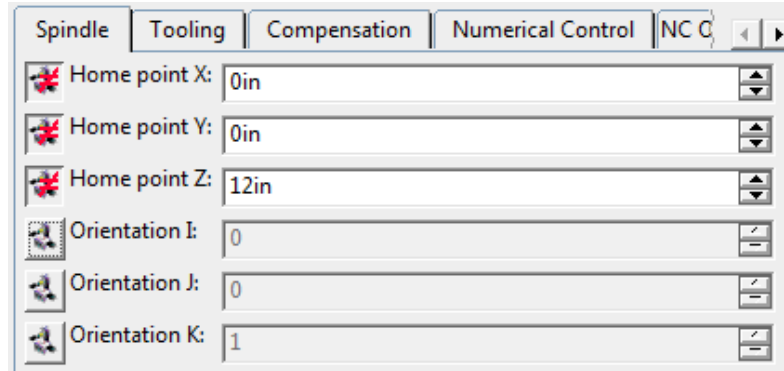








Select **Edit Local Parameters**. The *Local Modifications* of the machine window appears.

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 **(0,0,12in)**.  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.

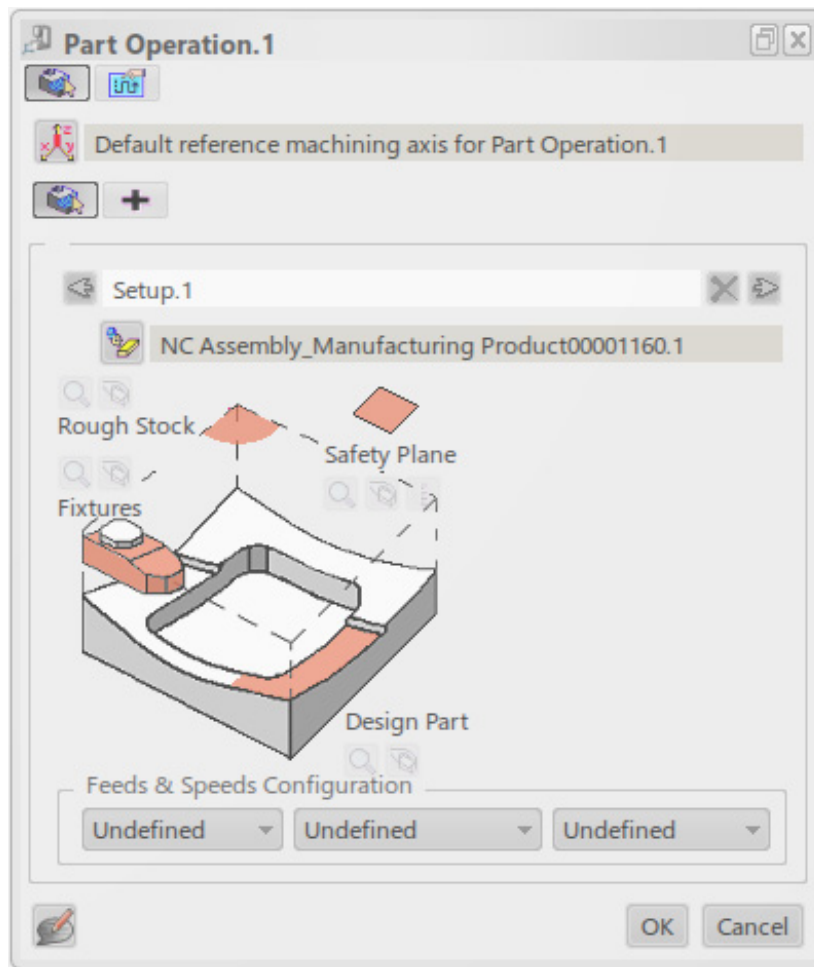


Spindle	Tooling	Compensation	Numerical Control	NC C
	Home point X:	0in		
	Home point Y:	0in		
	Home point Z:	12in		
	Orientation I:	0		
	Orientation J:	0		
	Orientation K:	1		


**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

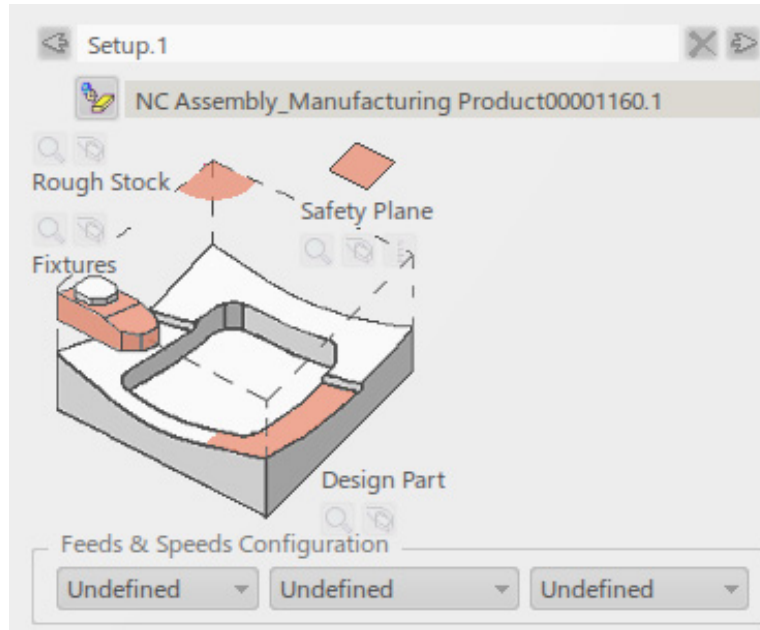
Switch to the *Activities Process Tree* tab and double select on *Part Operation.1*. This will display the *Part Operation* window.



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.

 **Setup** Defines the machining axis, stock, safety plane, fixtures and design part for the part operation

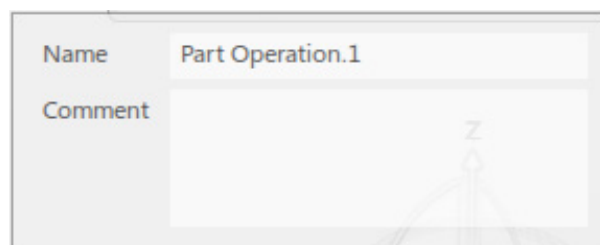
 Defines the location and orientation of the machining




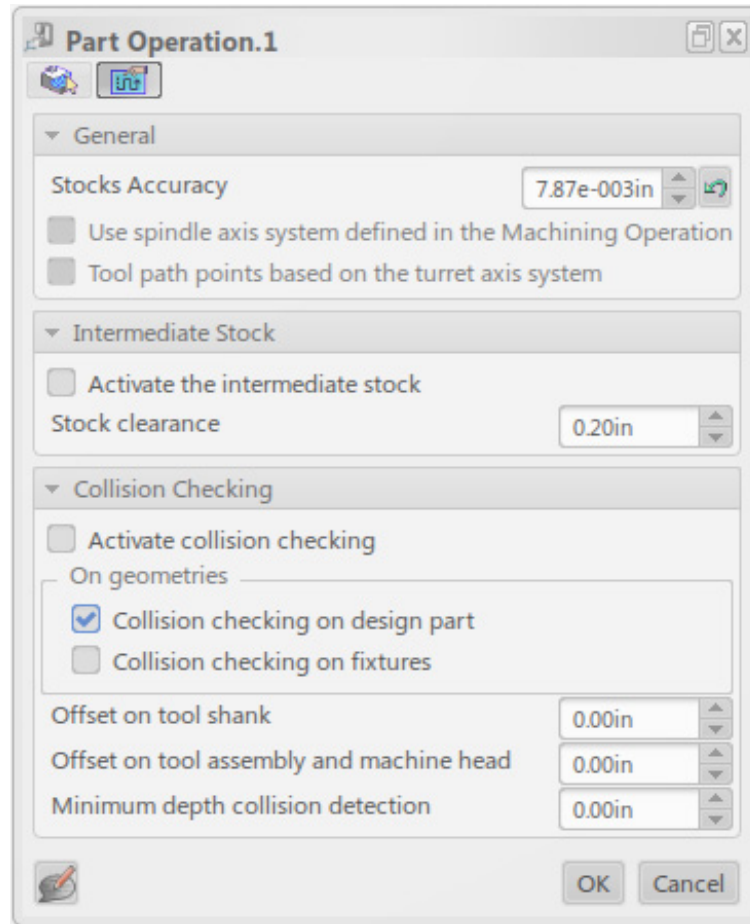
- Design Part* Defines the design part for use in the material removal analyses
- Rough Stock* Defines the stock part around the design part. If a stock part is not selected, a significantly bigger rectangular block will be assumed.
- Fixtures* Allows you to define any fixtures around the part. During material removal simulation, machining the fixture parts will display red areas to indicate crashes.
- Safety Plane* Defines the safety plane for the part operation
- Feeds & Speeds...* Allows you to configure the feeds and speeds for the part operation



Allows you to rename or enter any information about the program that you wish. The comments can be displayed in the documentation generated for the part operation.



 Options 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.

*Intermediate Stock* Allows the intermediate stock to be automatically computed and taken into account for the tool path computation

*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

<i>Offset on tool...</i>	Sets the offset on the tool shank
<i>Offset on tool...</i>	Sets the offset on the tool holder
<i>Minimum depth...</i>	Sets the minimum depth for collision detection

Select the **Rename and Comment icon**.



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

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

Select the **Rename and Comment icon to close the input.**

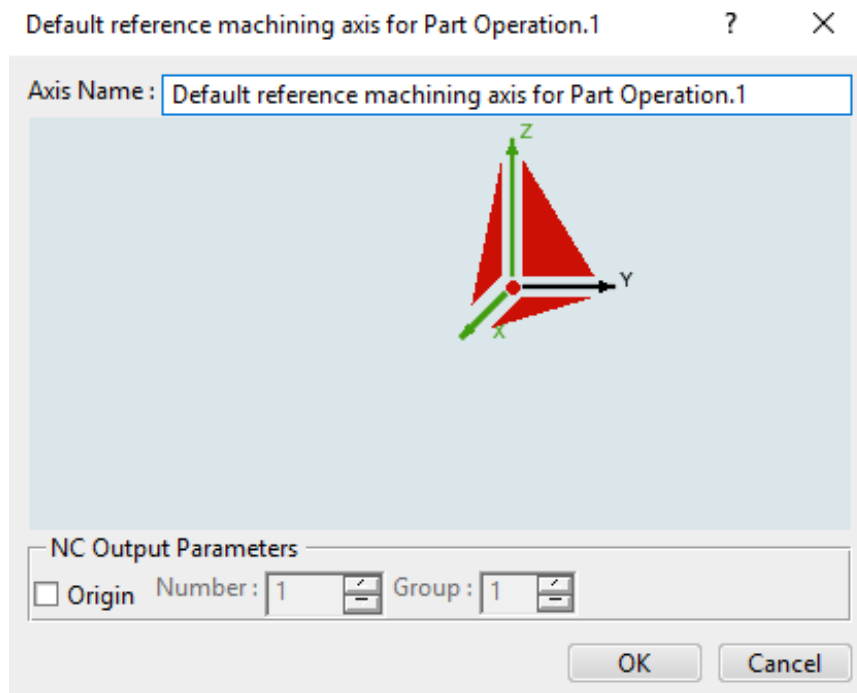


At this point, you are ready to begin defining other aspects of the part operation. The first definition that needs to be made is to define a machine.

## Machining Axis Definition

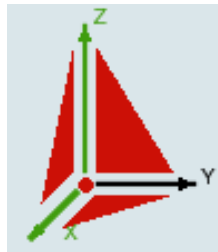
The machining axis is an important part of the NC Setup. The machining axis can generally be placed anywhere on the model that you want, however, there are a few locations that are better than others. You rarely want to have the machining axis buried or placed inside the stock material. You generally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



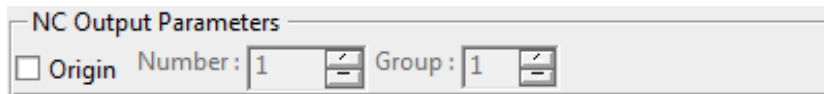
Take a moment to go over the various areas of the machining axis system window.

*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.

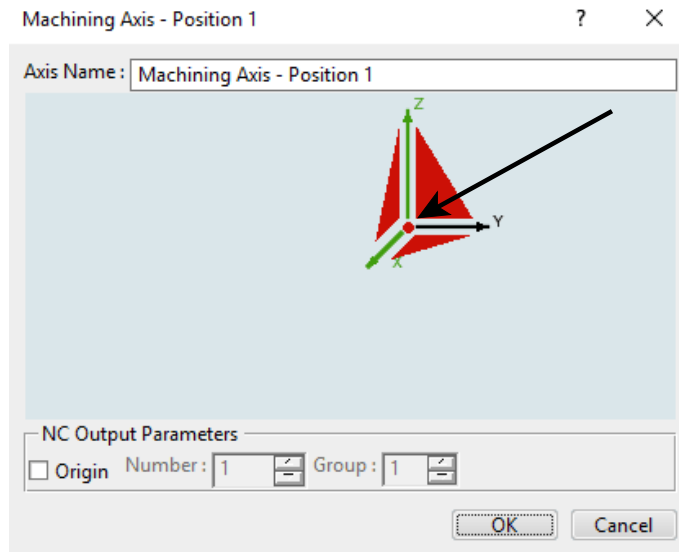
Now you are ready to define the axis location a few different ways.

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



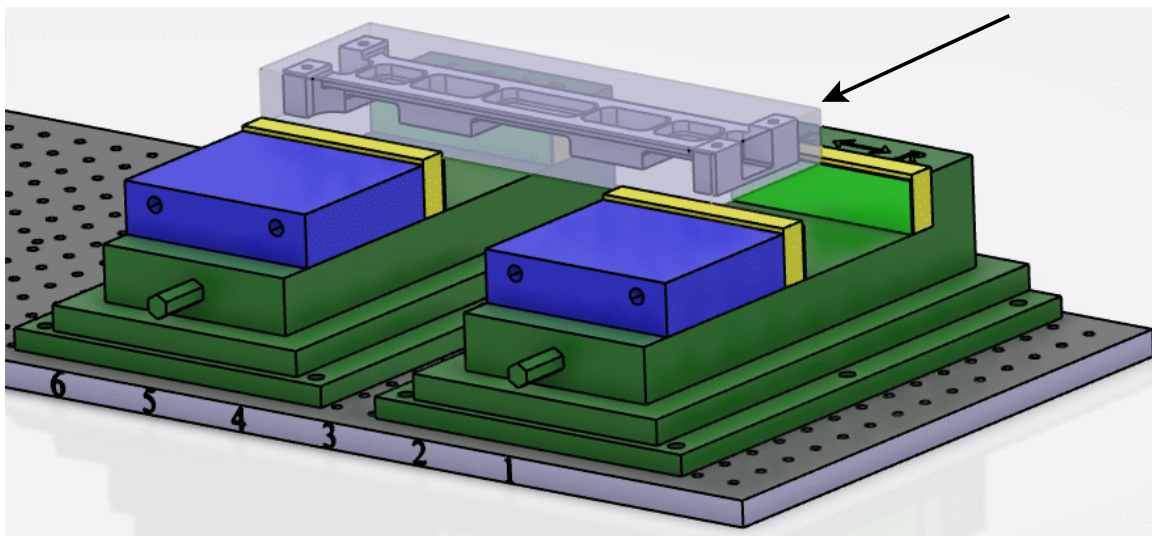
Properly identifying the axis systems will allow you to keep track of which axis systems are used for which part operations.

**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.

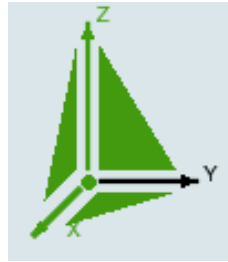


The machining axis system will move to the corner.



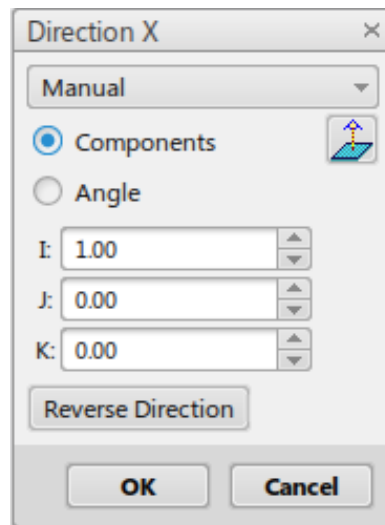


Notice the axis system in the machining axis window turns green. This denotes that a new axis location has been defined.



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.

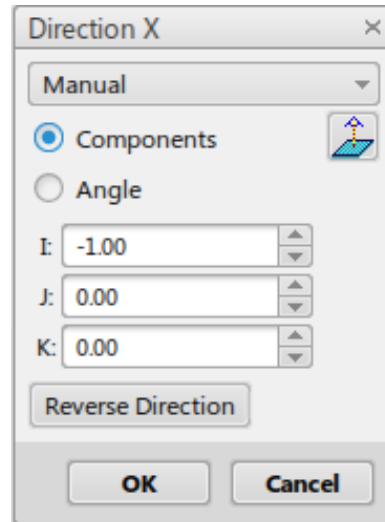
**Select the X axis arrow from the machining axis system window.** This will allow you to move the X axis. This will also display the *Direction* window.



There are three methods to define the axis direction.

<i>Selection</i>	This allows you to select an edge or line to define the axis direction
<i>Manual</i>	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.
<i>Points in the View</i>	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.

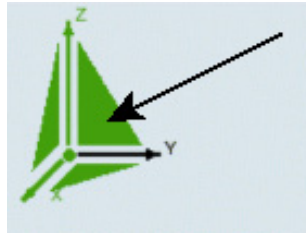
**Select *OK* to the *Direction* window.** This will display the machining axis window again.

**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.

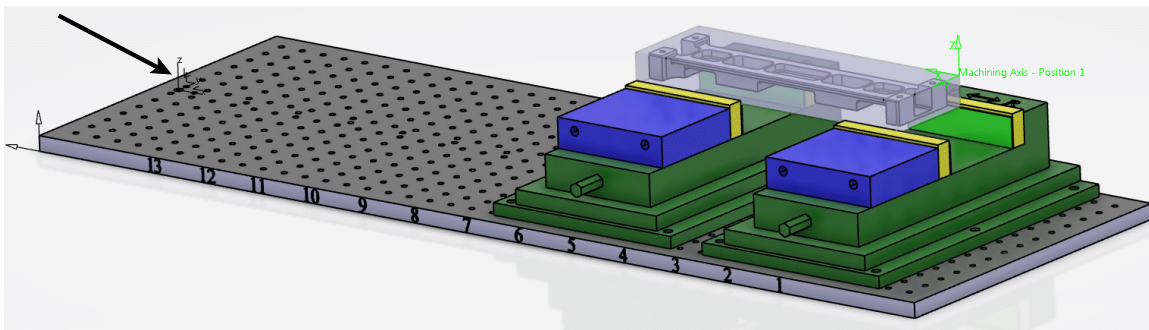
Select one of the planes of the axis system definition window. The planes are shown here.



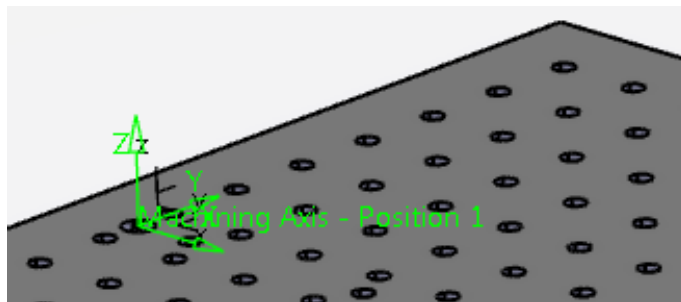
When you select the planes, the window will again disappear while CATIA waits for you to select an axis.

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.



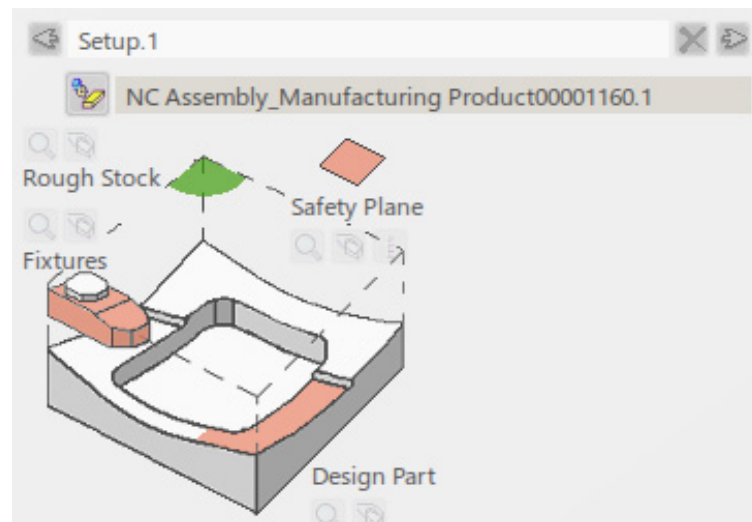
Select **OK** to the machining axis window. For now this will serve as the machining axis. Many times machine seeds will have an axis location defined that can serve as the machining axis.


## 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* selection area.** 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.

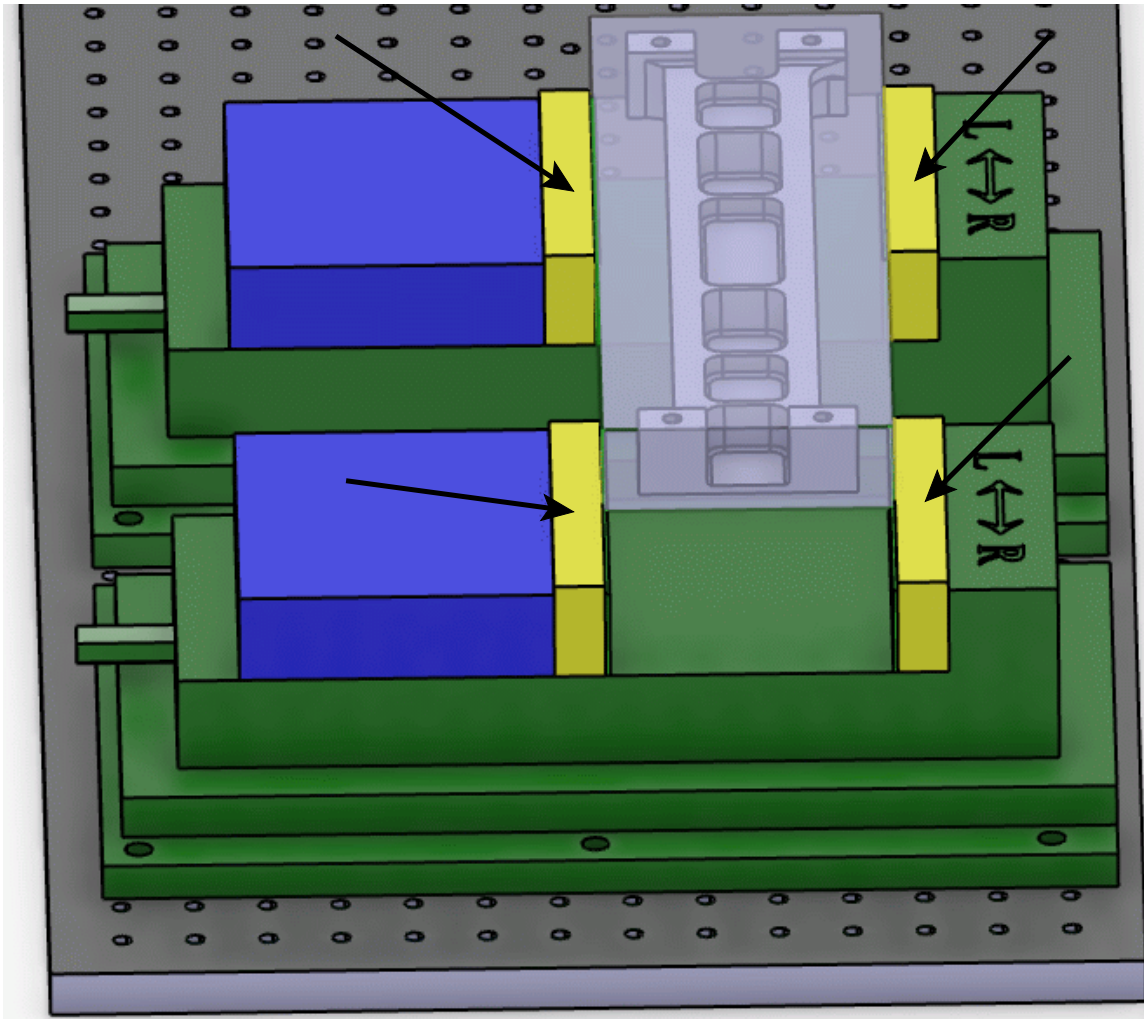
**Select the *Design Part* selection area.** This will define the final, as designed, part of the machining process.

**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* selection area.** 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.



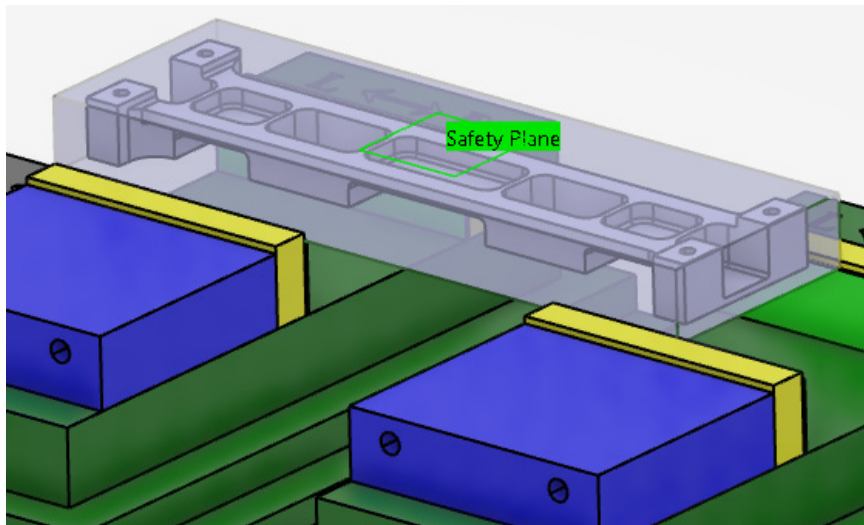
**Double select in the workspace when done.** You do not need to select on any part, just double select in space. You can define as many parts as fixture parts as you want. This will have four fixtures defined.

The last definition is the safety plane.

**Select the *Safety Plane* selection area.** This will allow you to define the safety plane or clearance plane that will be used by default in the machining operations.

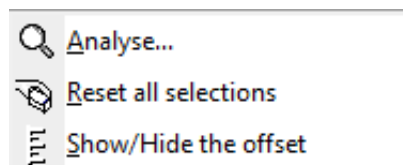
The safety plane is going to indicate a height above which there will be no obstacles. At this height, the machine is free to make any necessary movements around the part without fear of running into any sort of fixture, part, or other elements.

**Select the top of the stock material.** The top of the stock material will highlight green and show a virtual green plane.

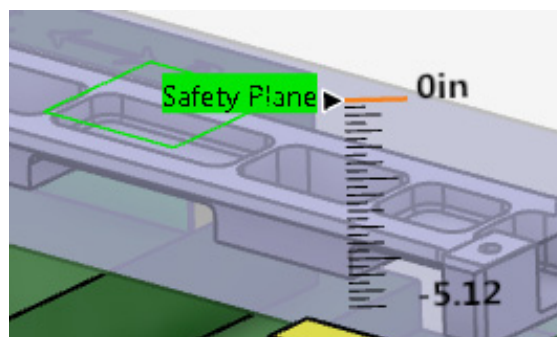


Granted, the top of the stock material is not a good location for the safety plane, however, the top of the stock material is the highest location on the assembly, therefore, it makes a good place to start from.

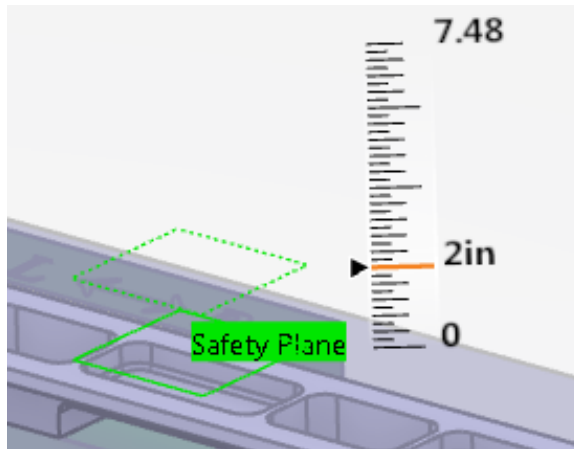
**With the third mouse button, select on the *Safety Plane* selection area in the window.** This will display the contextual menu for the safety plane.



**Select *Show/Hide the offset* from the contextual menu.** The offset appears in the display. This will allow you to offset the safety plane.



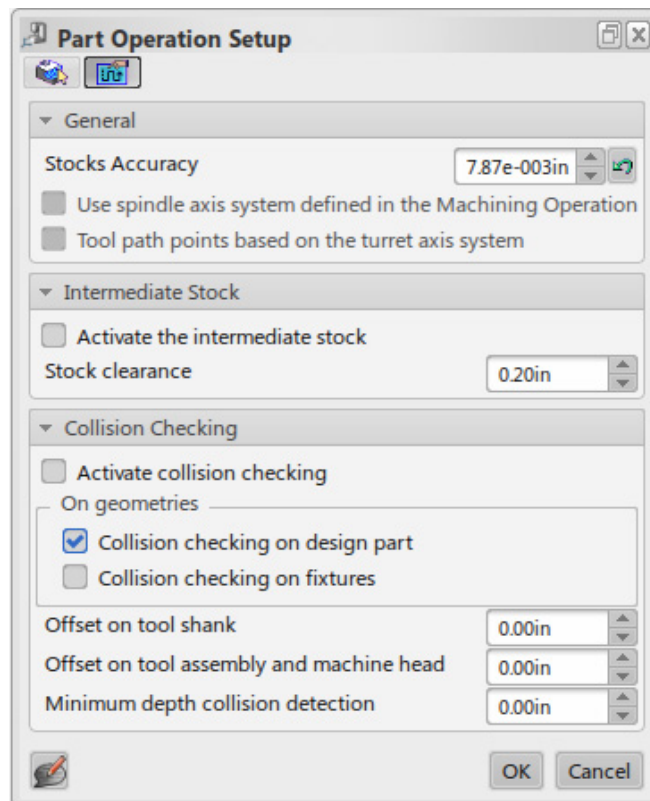
**Drag the slider to 2.0.** You can also select on the value and key in a specific number. By offsetting the safety plane 2 inches above the top of the stock part, you will be more assured that the machine will not collide with anything. You should notice the plane moves two inches above the stock part.



**Right select on the *Safety Plane* selection area in the window and select *Show/Hide the offset*.**

This will be the last of the geometry that you will be defining. Since you do not have a machine defined that was built by the machine builder, you will not need to define the transition or traverse planes.

**Select the Options icon.** 



**Change the *Stocks Accuracy* to 0.005in.** This will set the stock accuracy slightly higher to show a cleaner simulation. If you were machining this part, and began to have video replay issues, or were not satisfied with the performance, you could decrease the stock accuracy. Unfortunately, decreasing the stock accuracy will also decrease the quality and accuracy of video analysis options.

At this point, you are done defining the part operation. If desired you can activate collisions checking for the part and fixtures under the *Collisions Checking* section.

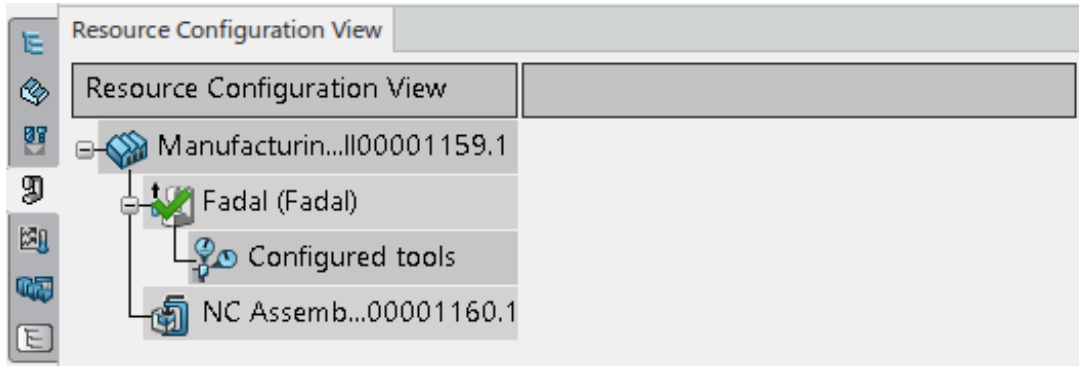
**Select *OK* to the *Part Operation Setup* window.**



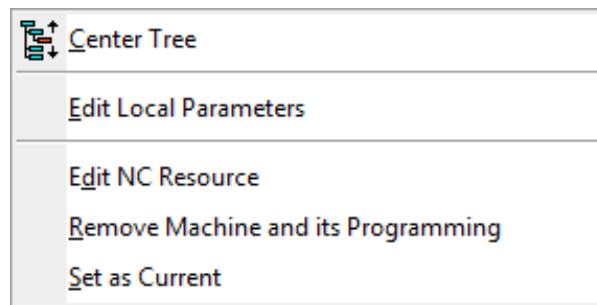
## Position Definition

It is important that you define the tool change point, and home point in the machine definition.

Select the *Resource Configuration View* tab.



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

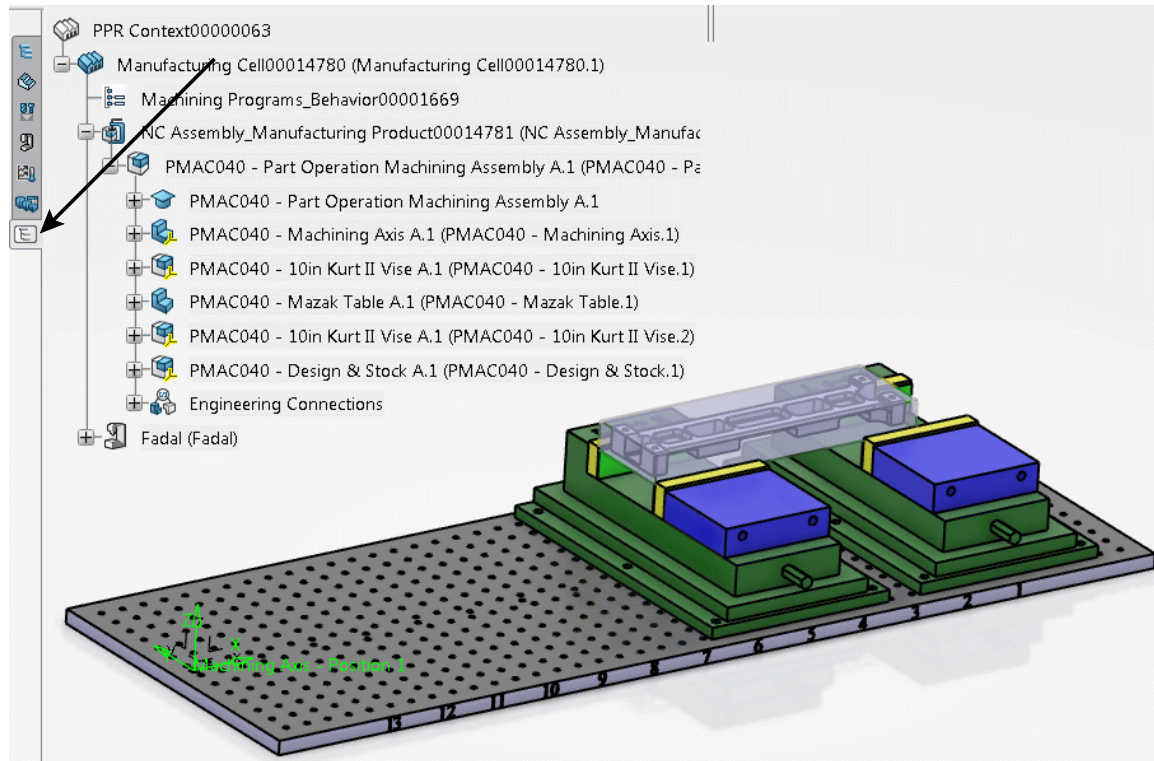


Select *Edit NC Resource*. The *Reference Parameters* of the machine window appears.

Select the *Tooling* tab.

Change the *Tool Change Point* to be **(0in,0in,30in)**. This will set the tool change point to be 30in above the machine axis.

Select *OK*.

**Select the *Tree* tab.**

At this point, you are ready to begin defining machining operations to machine the part. Although this may seem like a long drawn out process, it is a mandatory sequence to go through each time you start a program. Failure to completely define the part operation will result in operations and macros having issues as you define them.

This completes the definition of your part operation. Next you will look at resource creation.