TABLE OF CONTENTS

© Wichita State University

 ©Wichita State University

© Wichita State University

 ©Wichita State University

© Wichita State University

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

© Wichita State University

Part Operation Setup

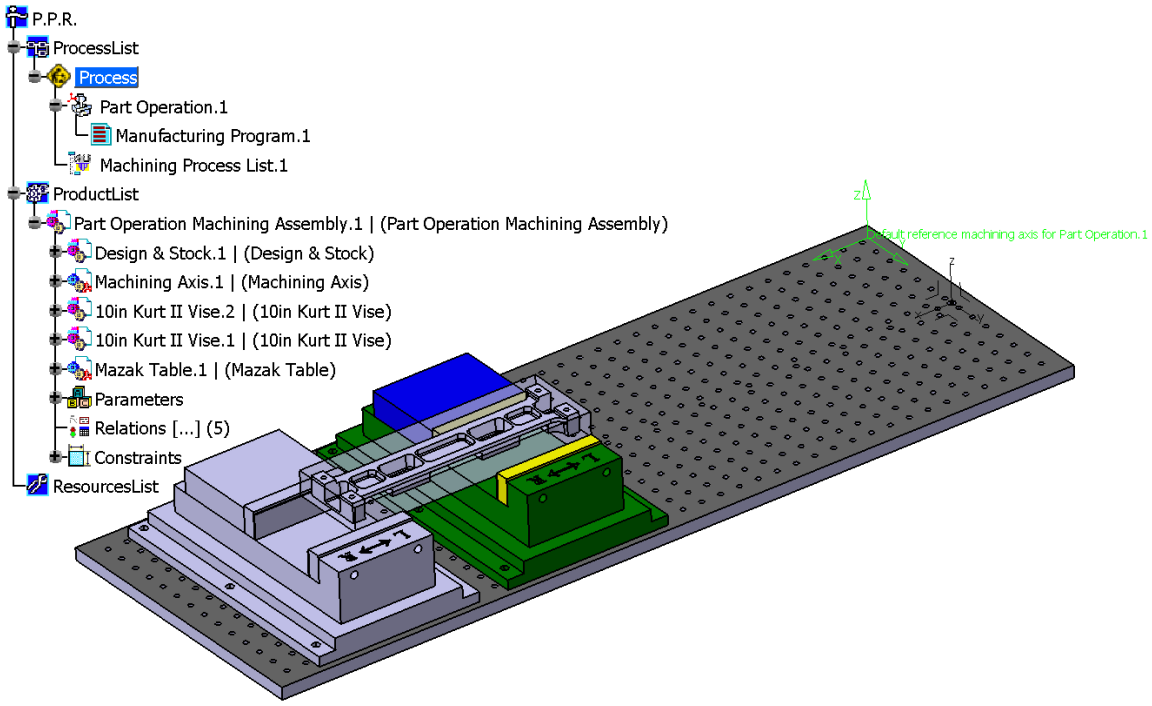
This section will investigate how to invoke the Prismatic Machining workbench and prepare your part for machining operations. Defining the part operation is a critical step for each machining process you start. Every time you prepare to machine a part, you must define the part operation.

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.

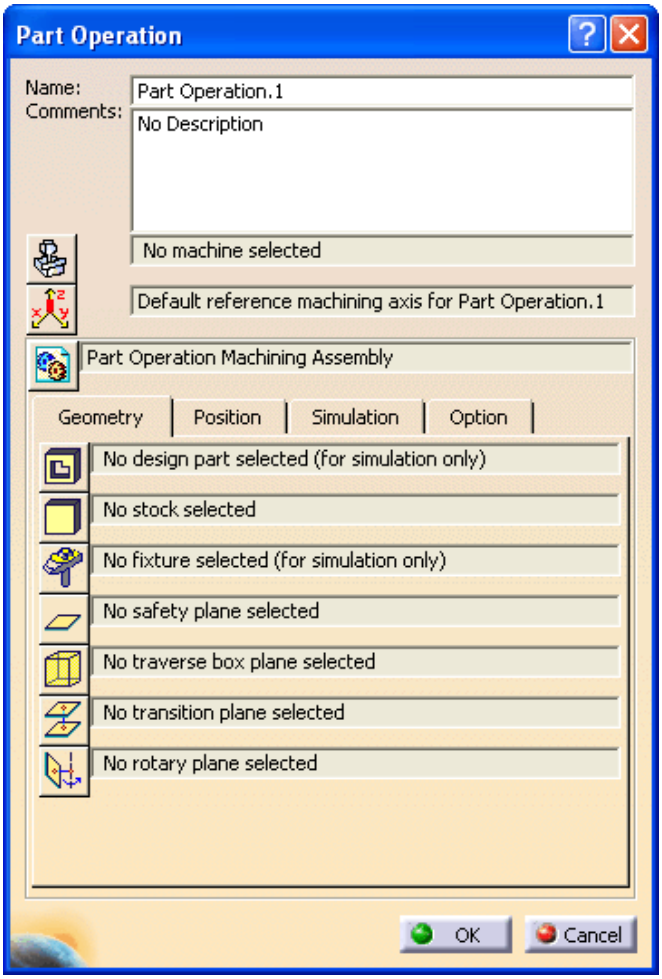
Now you are ready to begin defining the Part Operation.



If you remember from the introduction, the prismatic machining workbench utilizes the PPR tree, the Process Product Resource tree. Refer back to the introduction for full details as to 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.

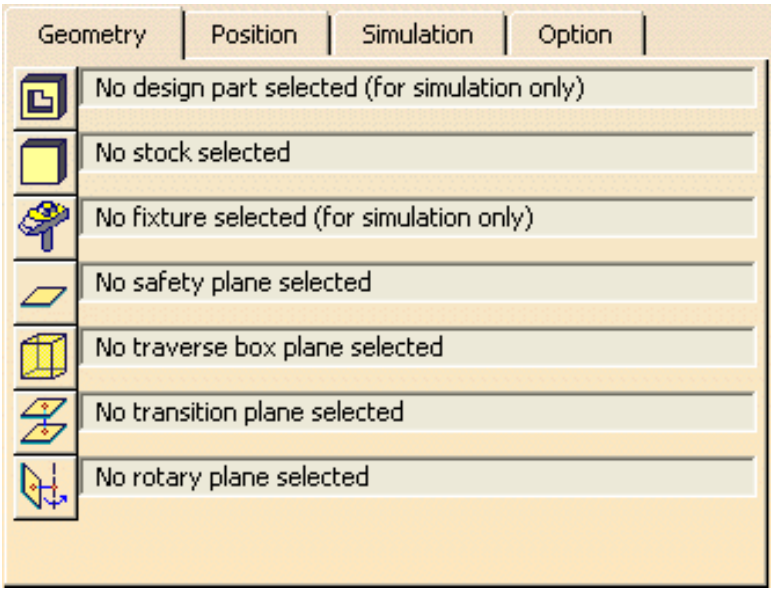
 *Machine Definition* — Defines the characteristics of the machine that you will be using

 *Machining Axis* — Defines the location and orientation of the machining axis

 *Part or Product* — Allows you to import a part or product into the process

 ©Wichita State University

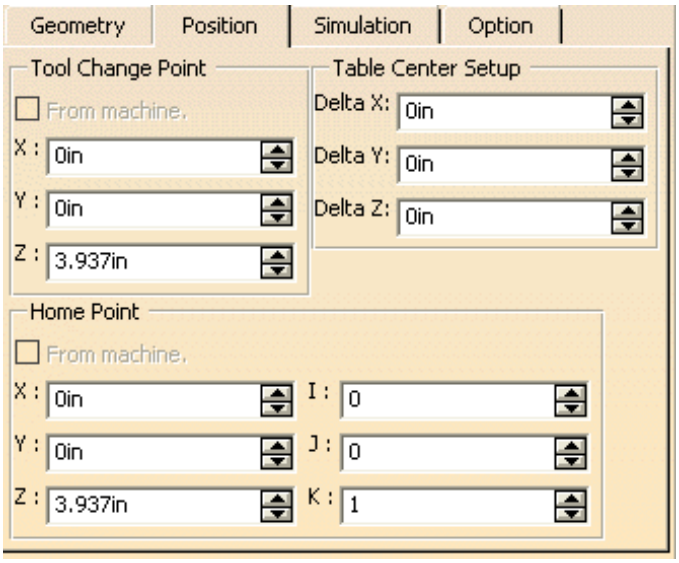
Geometry



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

© Wichita State University

Position

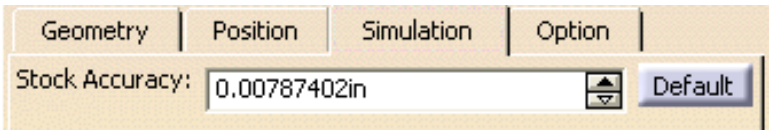


Tool Change Point Defines the X,Y,Z coordinates for the tool change point. This point should be away from the part, otherwise the tool will crash into the part to issue the tool change.

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

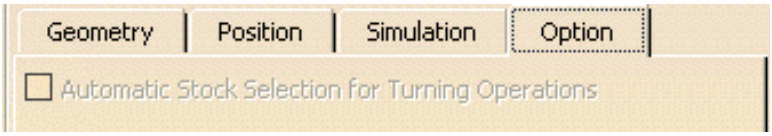
Home Point If not defined in the machine definition, the home point location of the machine must be defined here. Again, this point needs to be away from the part and stock, otherwise a crash will occur.

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



Automatic Stock... Allows the stock material to be obtained from the stock definition when working with lathe machining.

 ©Wichita State University

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.

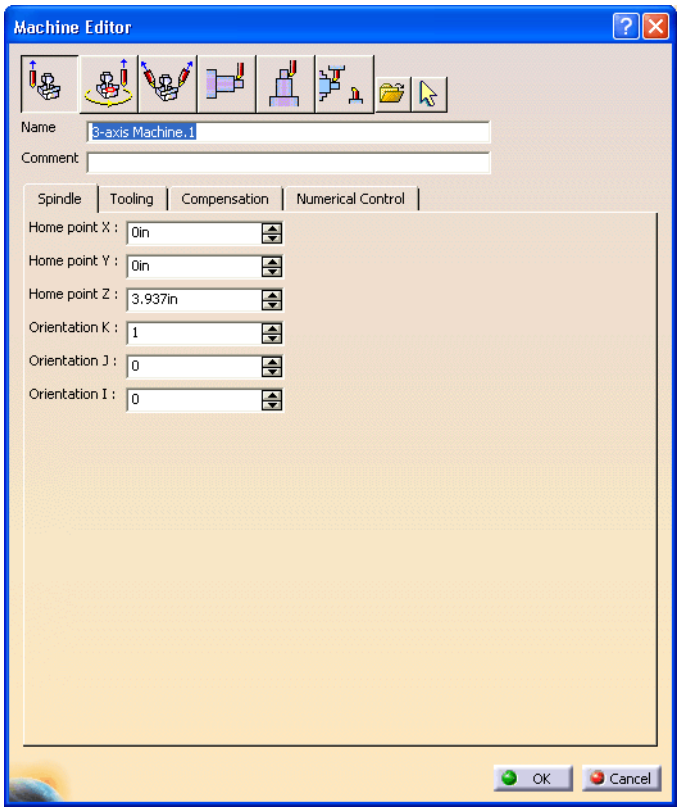
Change the *Comments* field to read Your Name and Today's Date. The comments field is very useful to insert specific information about the part operation. In this case, you are just going to state who you are, but you could add information about the part, information about the use of the part or anything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

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, as well as other aspects such as the tooling catalog, and NC code output parameters. It is best to always start with the machine definition before cointinuing.

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

 Multi-slide Lathe Machine

You also have the ability to assign a machine that would 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.

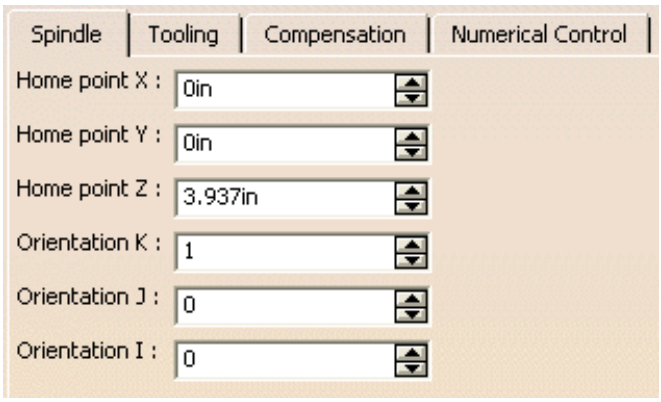
Select the 3-axis Machine icon.  This is the default option and may already be selected.

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.

 ©Wichita State University

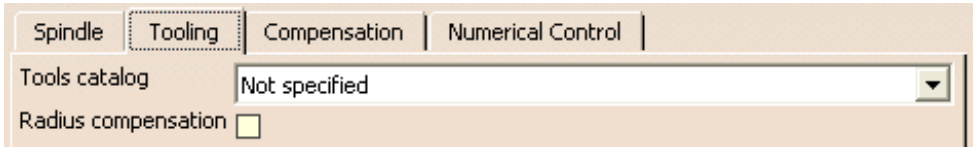
Spindle



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

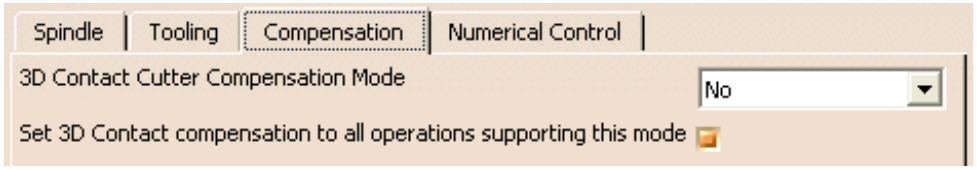
Tooling



Tools catalog Defines what tool catalog you will be using

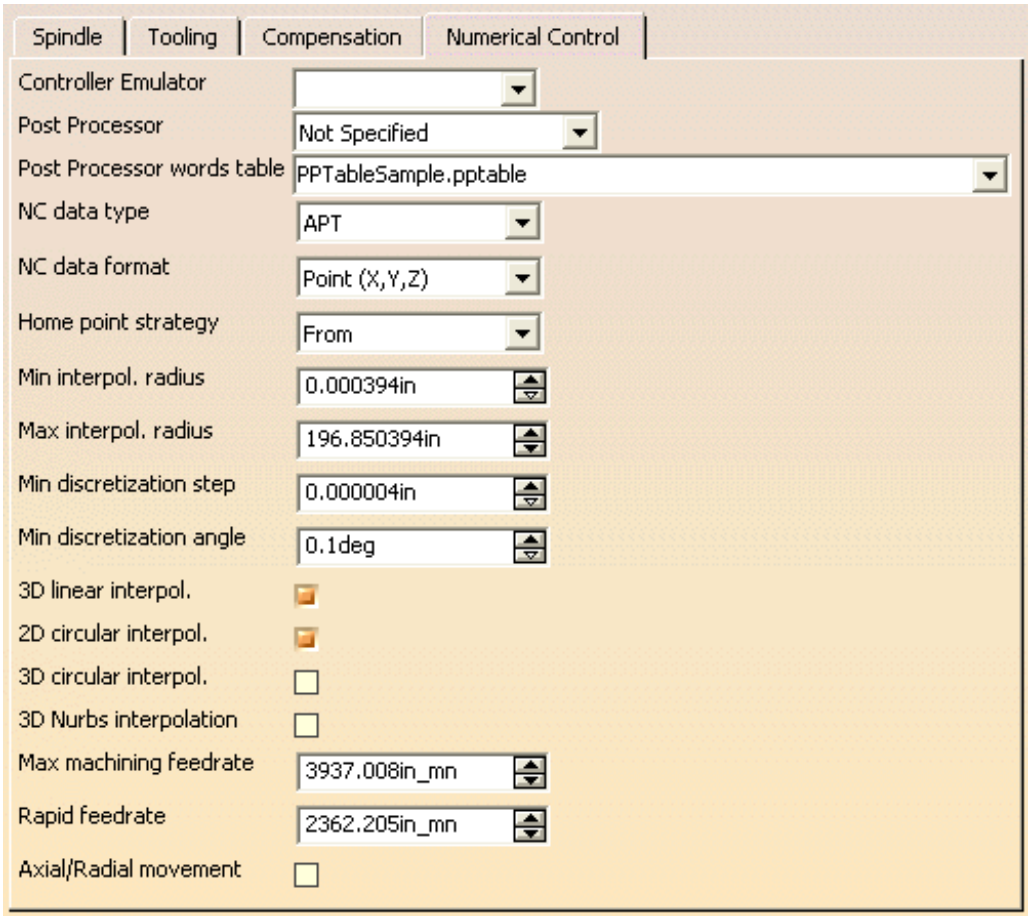
Radius compensation Toggles the radius compensation on or off for each tool

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.



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 outputted. This will automatically change based on the type of machine selected.

 ©Wichita State University

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.
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. CAITA 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)

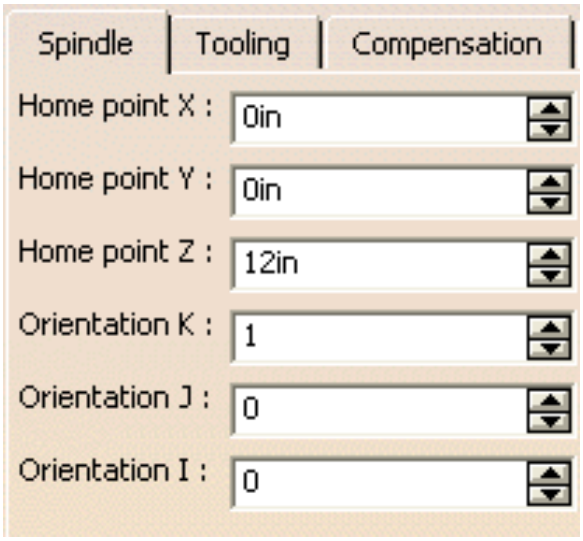
Due to the sheer number of options, you will often find that many companies will define machine process seeds that define all of these options ahead of time, just like the machine setup. This will allow for machine programmers to utilize the proper settings for the machine each time a program is started.

Since you are not starting with a machine seed, you will want to 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.

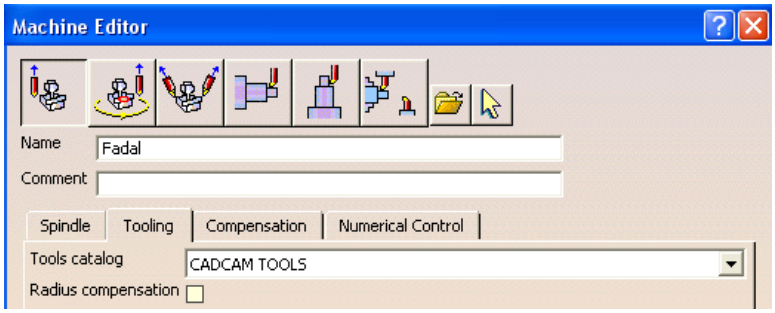
Change the *Home point* to be set to (0,0,12in). That is, make the X value 0 inches, Y value 0 inches and the Z value 12 inches. All coordinates will be relative to the machining axis system for this part operation. Even though the machine axis has not been defined, you will set the home point first.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

 ©Wichita State University

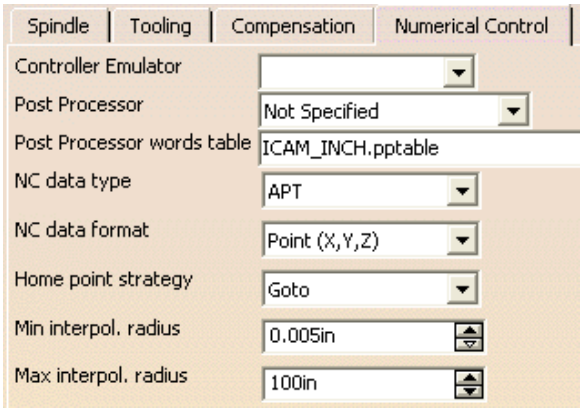
Switch to the *Numerical Control* tab, change the *Post Processor words table* to *ICAM_Inch.pptable*. This will allow you to utilize the ICAM post processor and insure that the output code will be in the proper format for the ICAM post processor. This will also insure that the output is in inches rather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

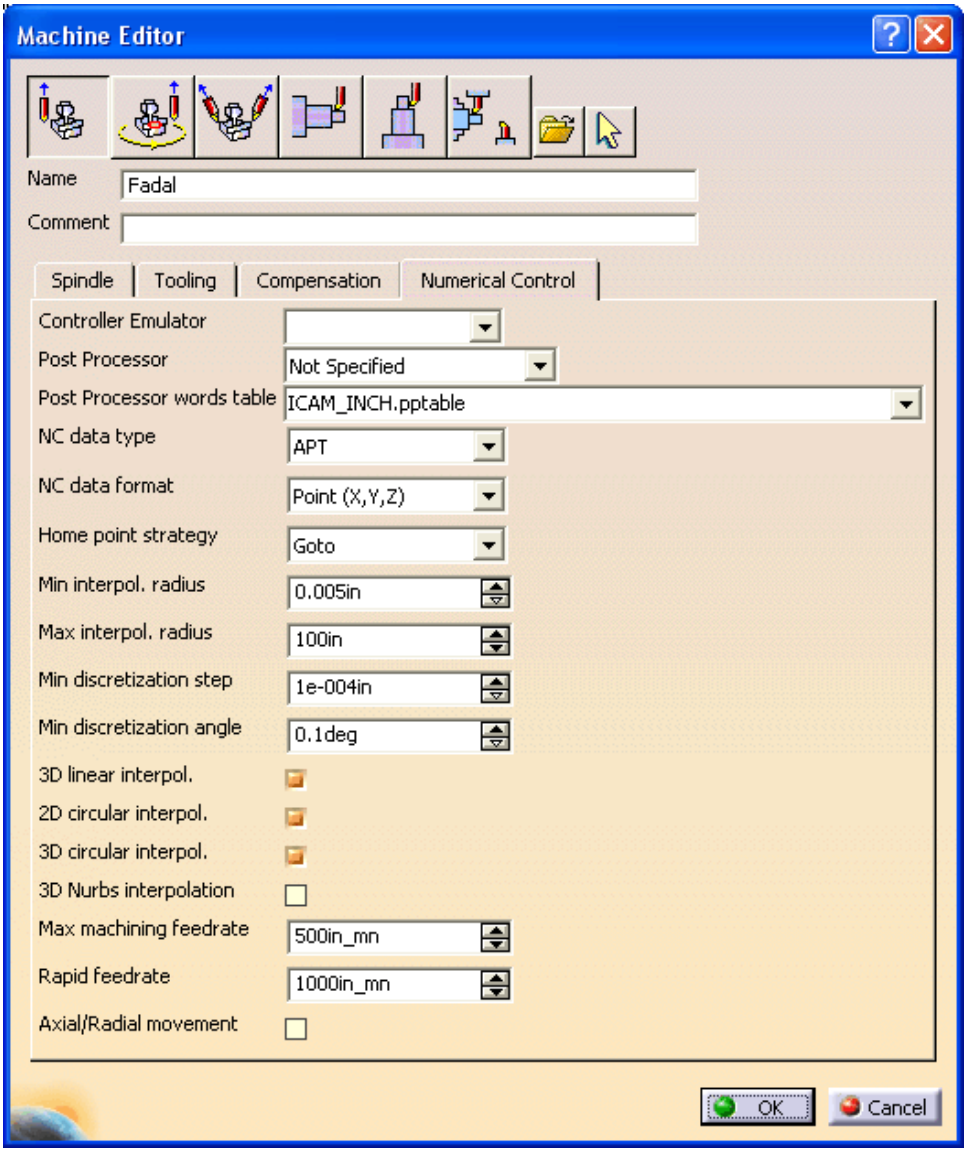
Set the *Max machining feedrate* to 500in_mn. This will indicate that the maximum machining feedrate allowed for this machine is 500 inches per minute.

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.

© Wichita State University

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



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

You will always want to make sure that 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.

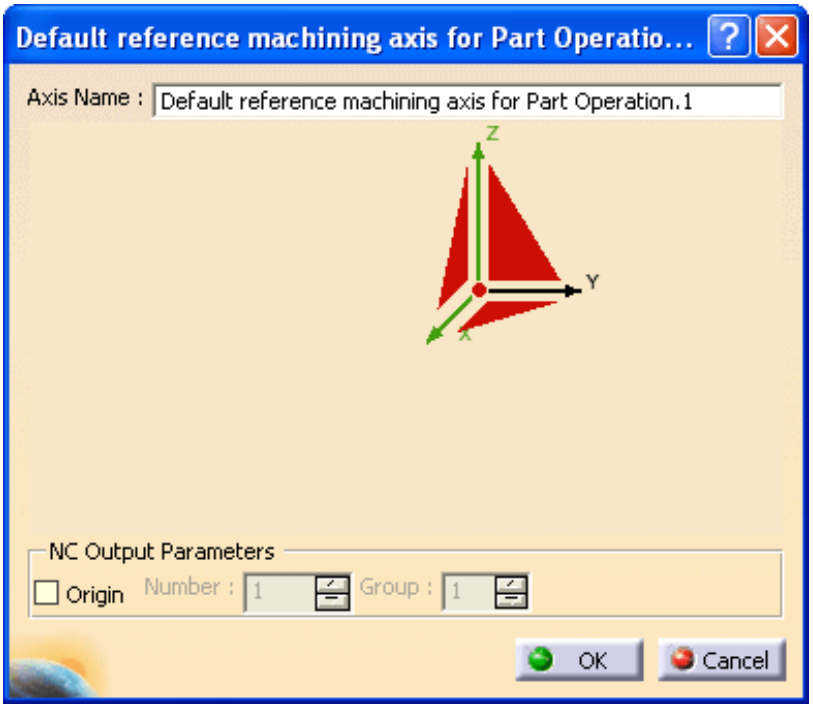
The next important step is to define the location of the Machining Axis.

 ©Wichita State University

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.

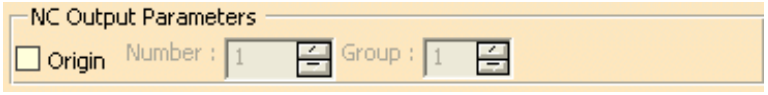
Select the Reference Machining Axis System icon.  This will display the *Machining Axis System* window.



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.



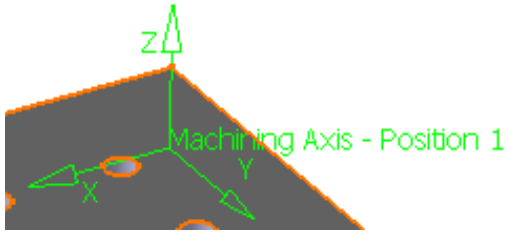
NC Output Parameters

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.

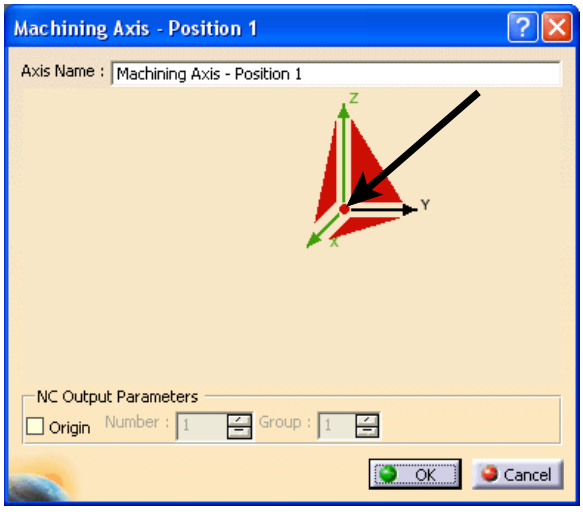
Change the *Axis Name* to <u>Machining Axis - Position 1</u>. This will give the axis a decent name. You will also notice that the axis changes names in the graphical workspace.



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

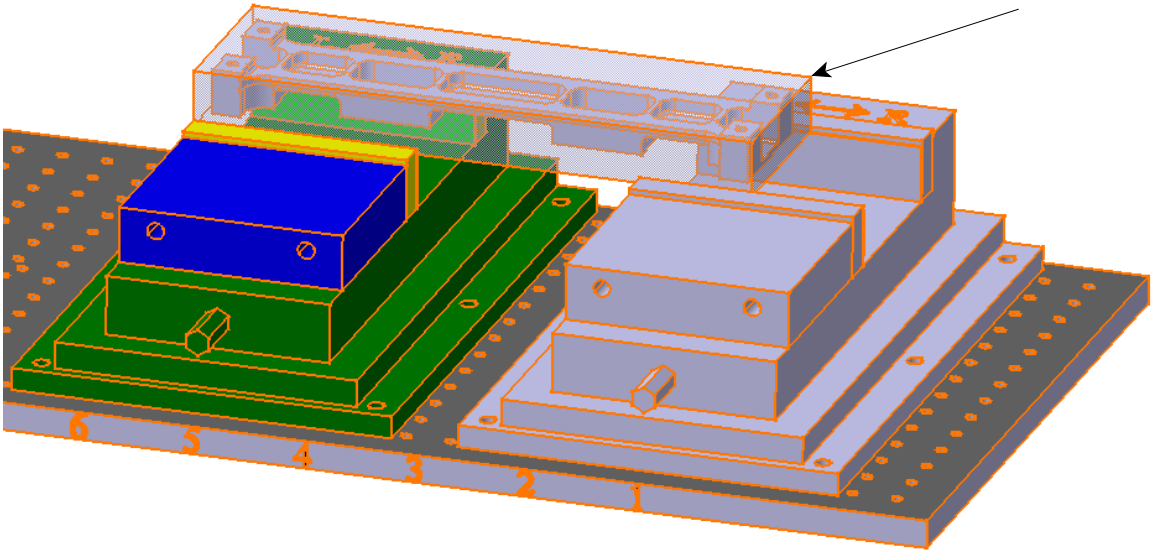
 ©Wichita State University

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

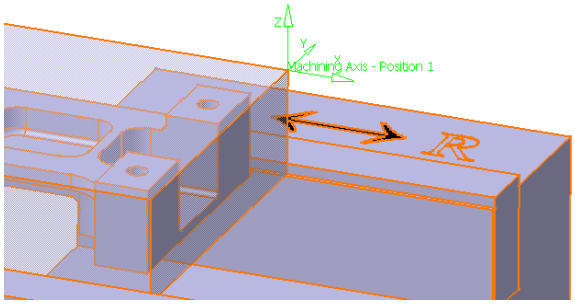


The center red dot will allow you to move the entire axis system from one location to another. The *Machining Axis System* window will disappear while CATIA waits for you to select a point or vertex to be the new center of the axis system.

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



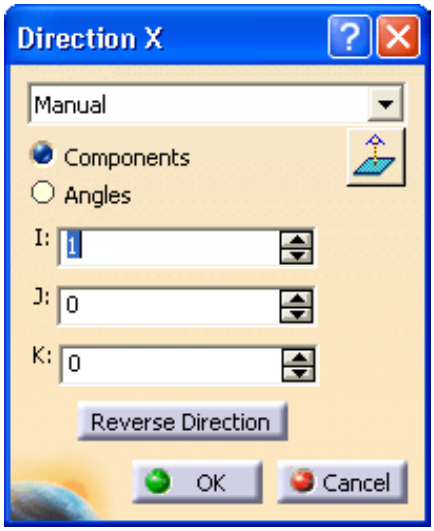
The machining axis system will move to the corner.



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.

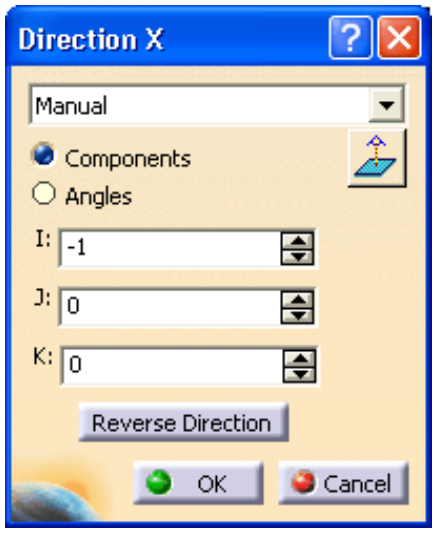


There are three methods to define the axis direction.

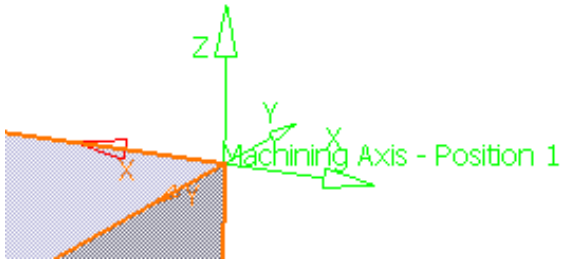
Selection	This allows you to select an edge or line to define the axis direction
Manual	This allows you to key I, J, and K directions to control the axis direction. Manual also allows for reversing the direction of the axis.
Points in the View	Allows for selection of two real points to define the axis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

 ©Wichita State University



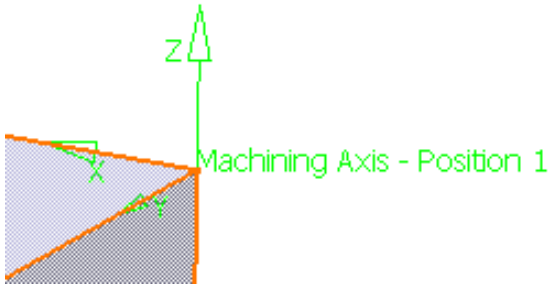
Notice that the axis does not appear to move.



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.

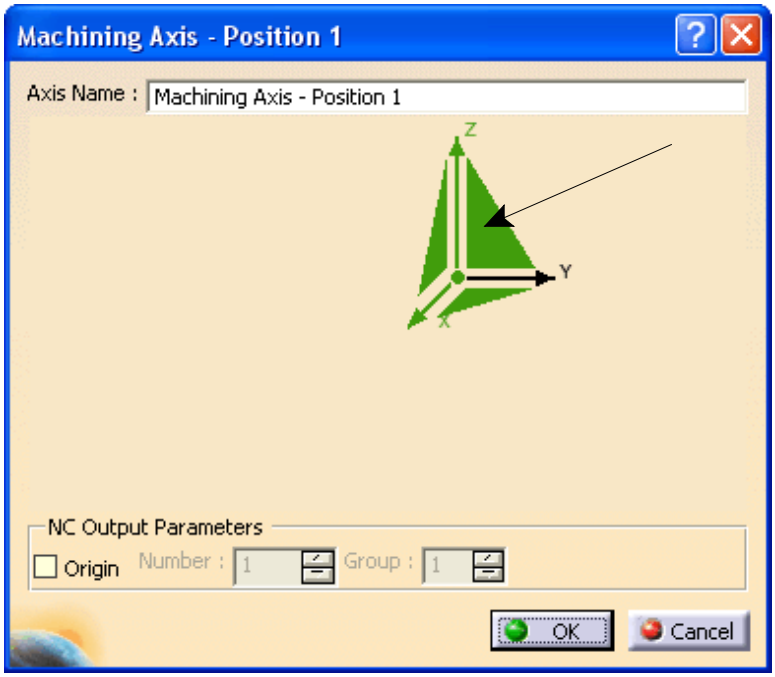
Select *OK* to the *Direction* window. This will display the machining axis window again.

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.

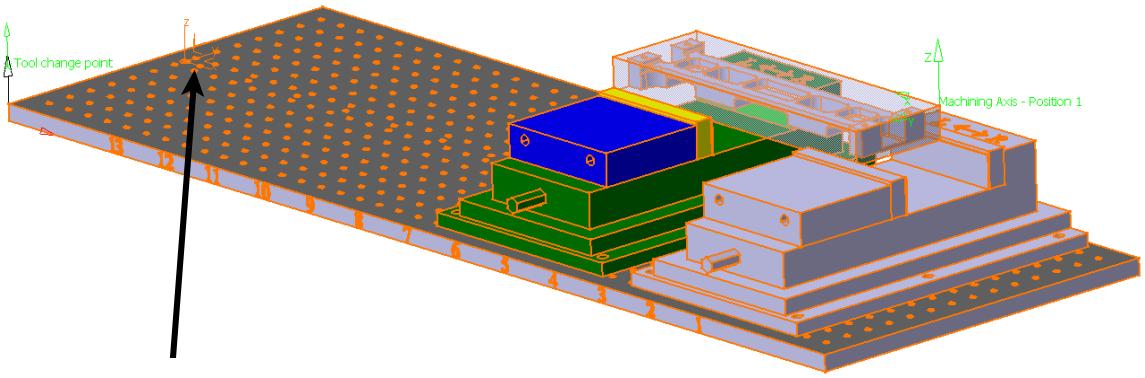
Select one of the planes of the axis system definition window. The planes are shown here.



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

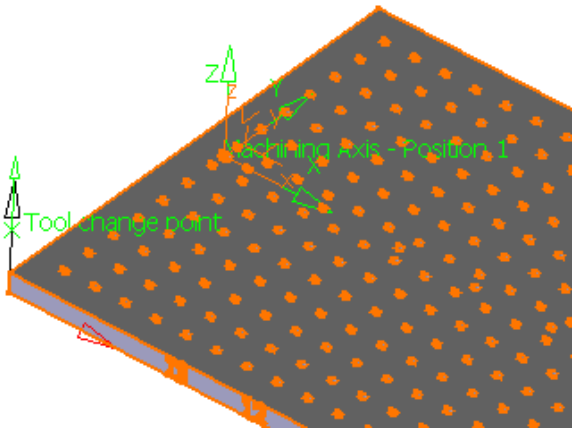
At the other end of the table, there is an axis system. You will set the axis system to be the same as this axis.

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.

 ©Wichita State University



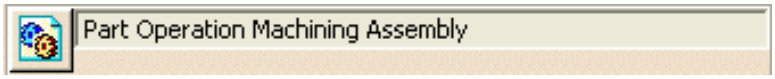
Select *OK* to the machining axis window. For now this will serve as the machining axis. Many times machine seeds will have an axis location defined that can serve as the machining axis.

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.

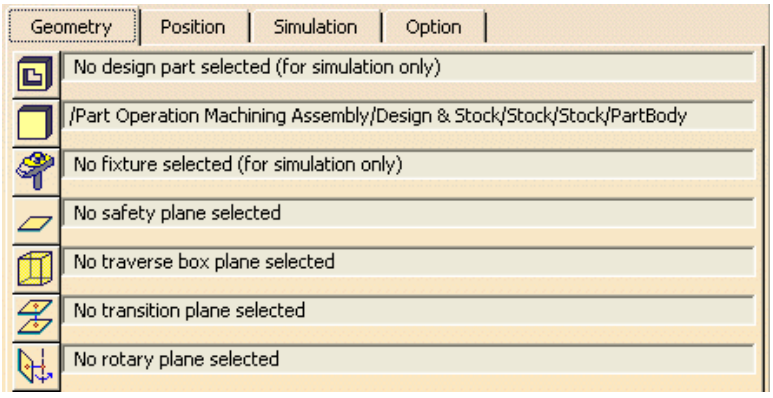
© Wichita State University

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. Even though it notes that the design part is for the simulation, the design part definition will allow

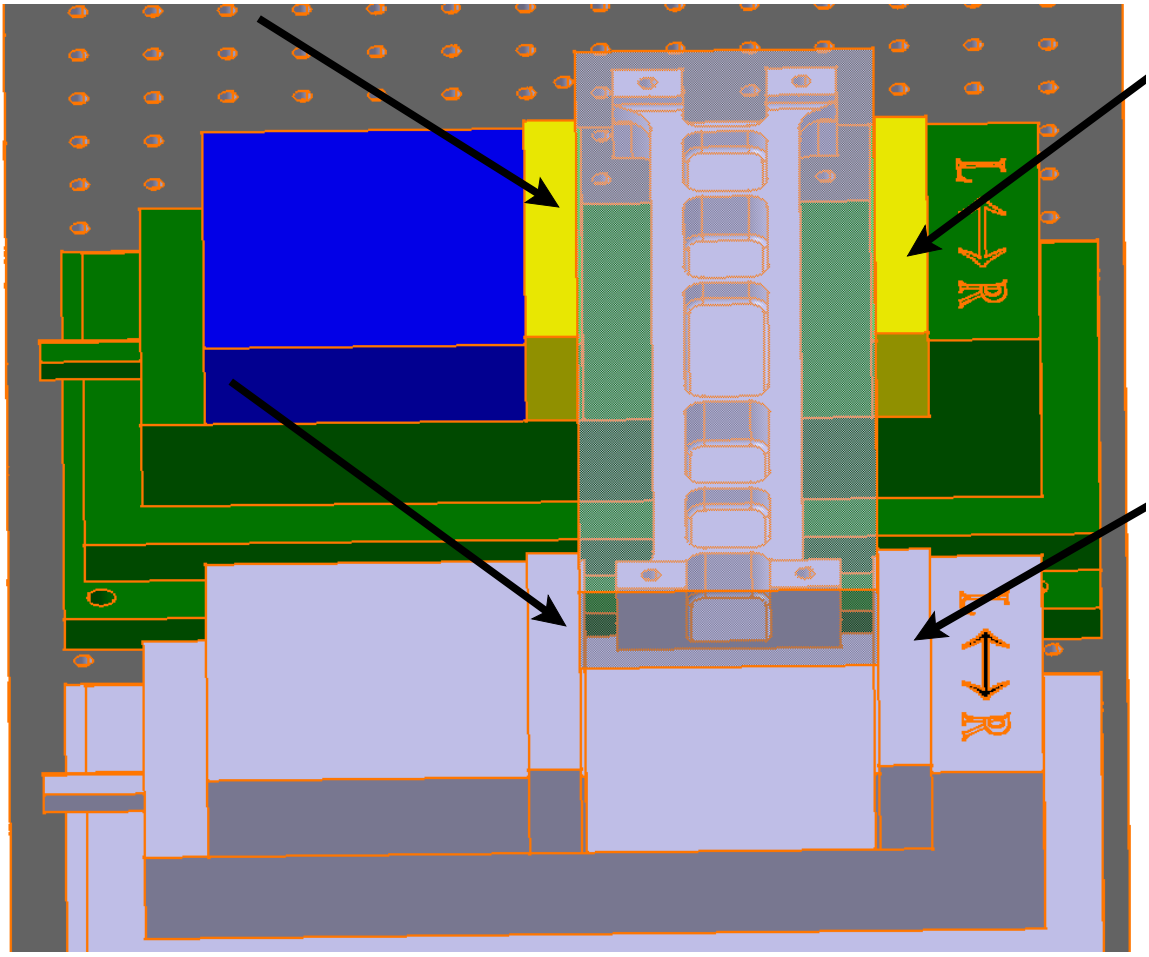
Select the design part, then double click in space. Unhide the stock when done. You will be needing the stock part again, so it is best to unhide it.

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, close fixture elements. Defining a lot of complex fixtures will require an excessive amount of video memory, and can cause CATIA to crash.

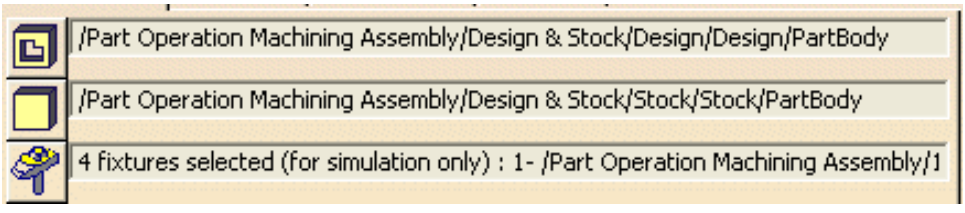
 ©Wichita State University

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 vises. 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.

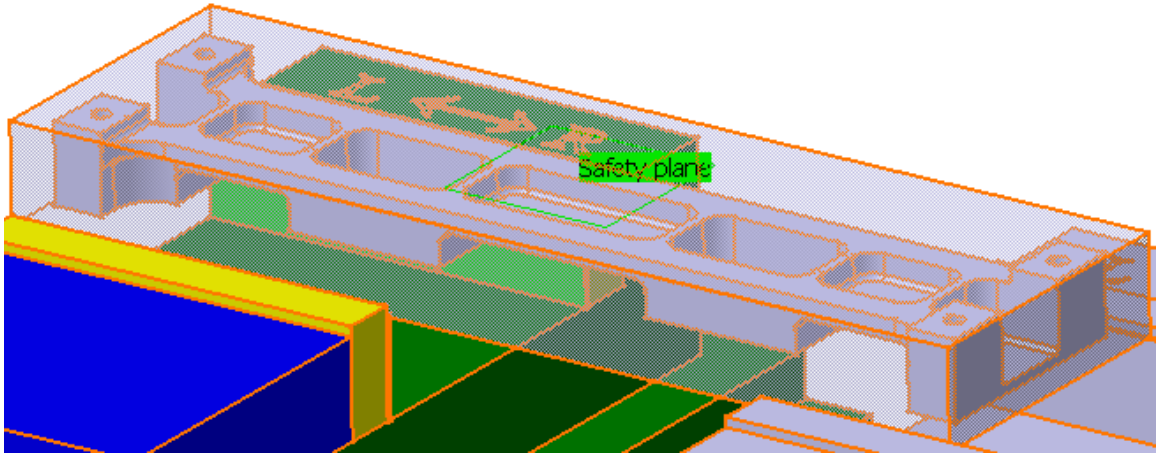


The last definition is the safety plane.

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

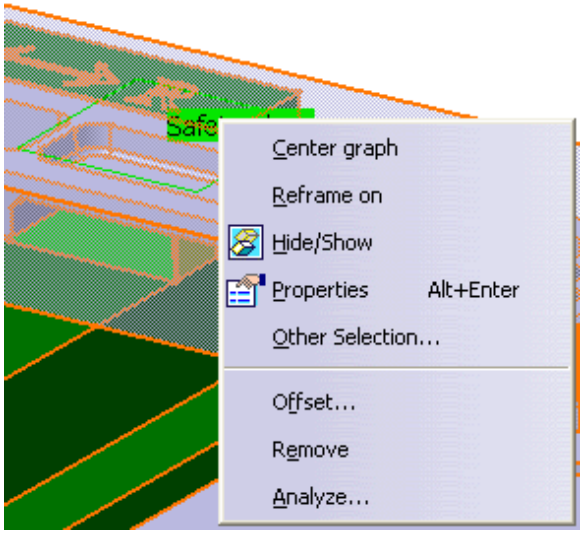
The safety plane is going to indicate a height above which there will be no obstacles. At this height, the machine is free to make any necessary movements around the part without fear of running into any sort of fixture, part, or other elements.

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



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.

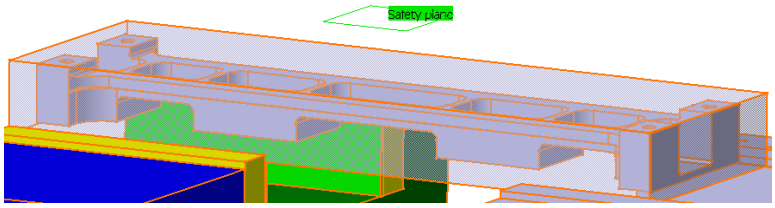
With the third mouse button, select on the safety plane text from the graphical workspace. This will display the contextual menu for the safety plane.



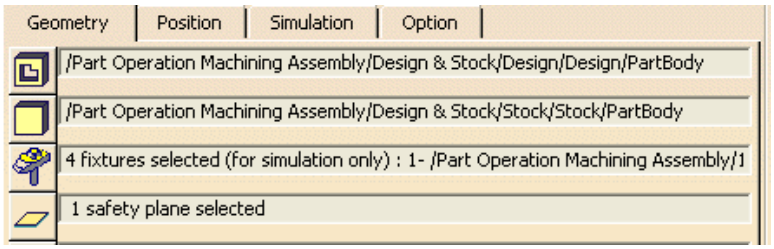
Select *Offset* from the contextual menu. The *Edit Parameter* window appears. This will allow you to offset the safety plane.

 ©Wichita State University

Change the *Thickness* to 2.0in and select *OK*. 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 moving two inches above the stock part.



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

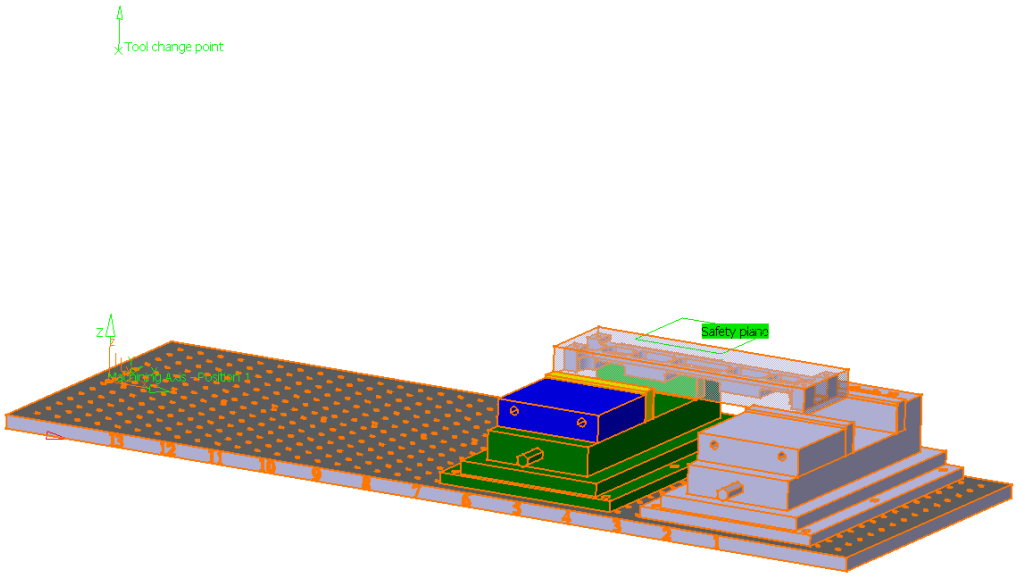


Position Definition

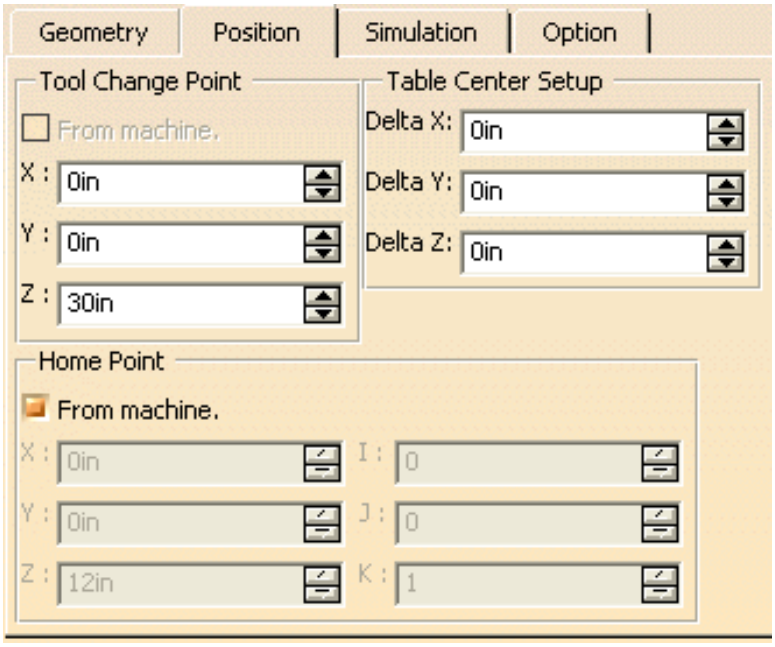
The position tab allows you to control various tool and table positions. It is important that you switch to the position tab to define the tool change point, and home point if it is not defined in the machine definition.

Select the *Position* tab. Keep in mind, all coordinates you define will be relative to the machining axis definition. If you move the machining axis, the various points will automatically update.

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



Since you defined a home point with the machine definition, you will not need to define the home point here. Had you not defined a machine for the part operation, you would want to be sure to specify a home point and orientation.



Simulation Accuracy Definition

The simulation accuracy helps control how accurate the cut stock is shown versus the amount of processing power required to show the replay. Many times, you can decrease the simulation stock accuracy to help fix video simulation issues.

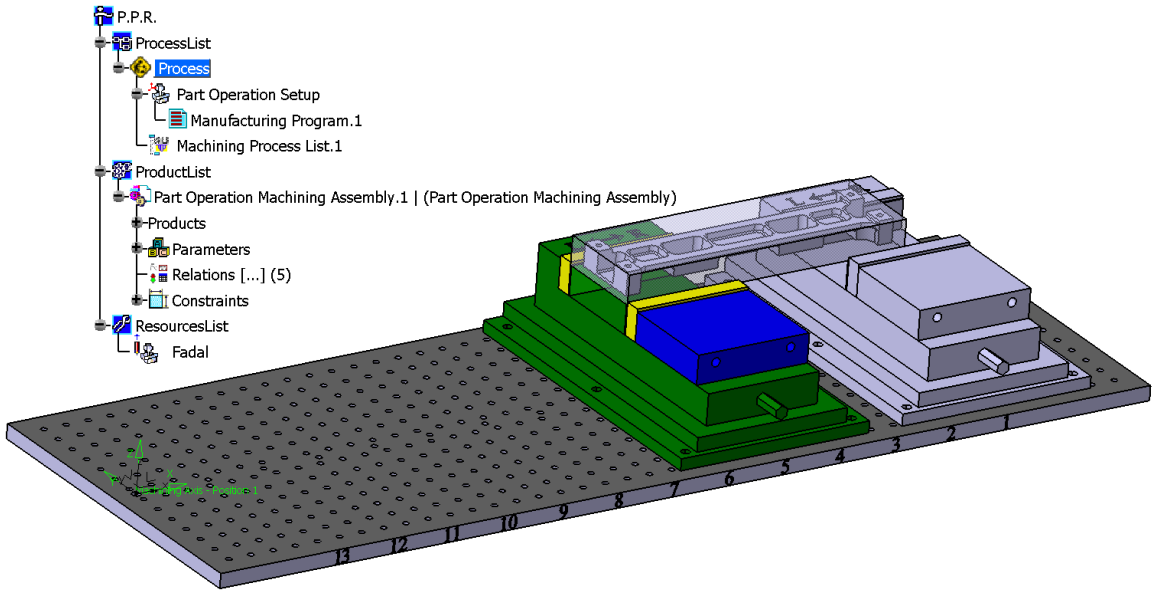
Switch to the *Simulation* tab. There is only one option with the simulation tab, the stock accuracy.

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

At this point, you are done defining the part operation. Since this is not a lathe machine, you do not have any other options to define in the *Options* tab.

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

 ©Wichita State University



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.

Save and close your document.