TABLE OF CONTENTS

Introduction

CATIA Version 5 Prismatic Machining

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

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

Part Operation Setup

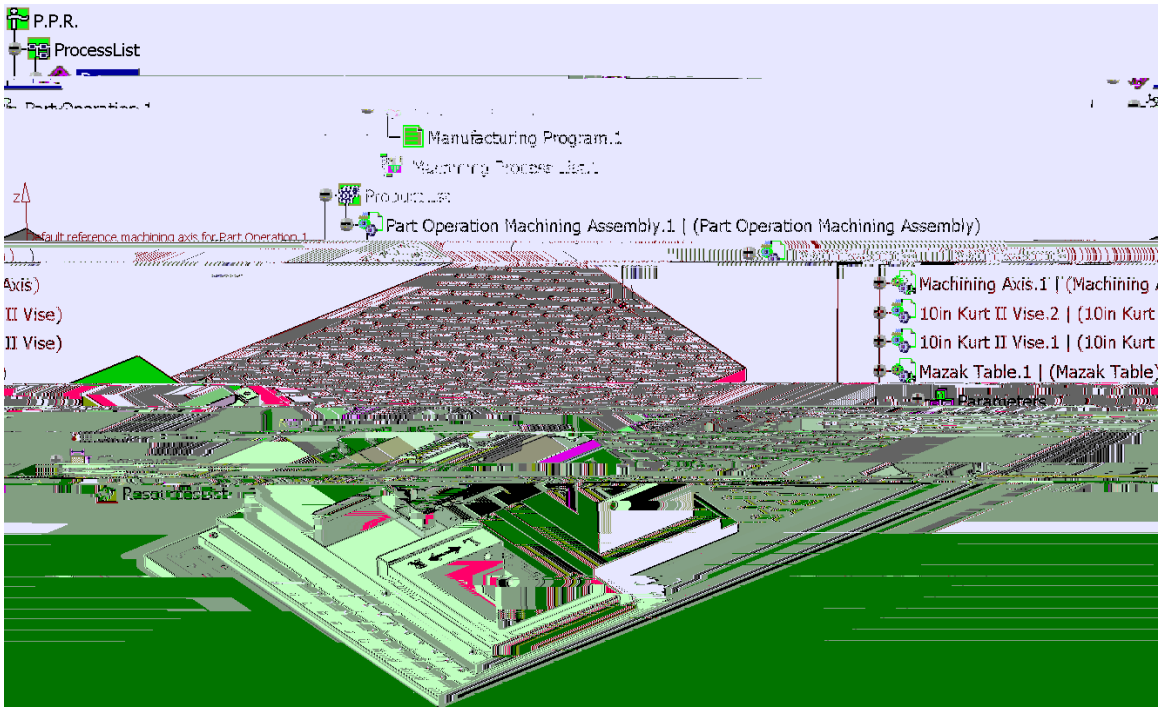
This section will investigate how to invoke the Prismatic Machining workbench and prepare your part for machining operations. Defining the part operation is a critical step for each machining process you start. Every time you prepare to machine a part, you must define the part operation.

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

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

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

Now you are ready to begin defining the Part Operation.

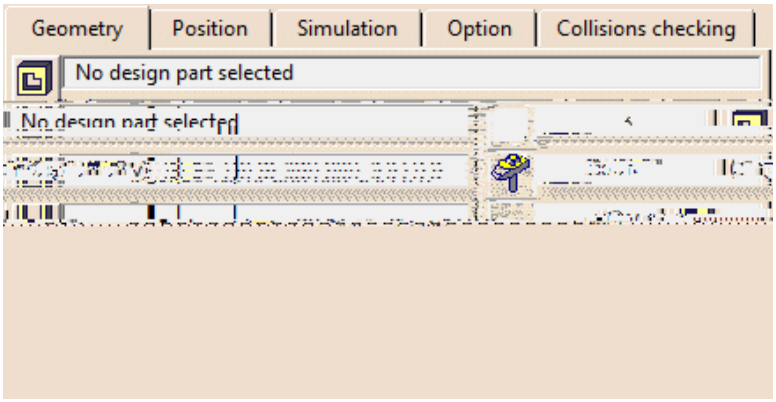


If you remember from the introduction, the Prismatic Machining workbench utilizes the PPR tree, or the Process Product Resource tree. Refer back to the introduction for full details on the PPR tree.

Defining the Part Operation

Double select the *Part Operation.1* branch in the

Geometry



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

Position

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

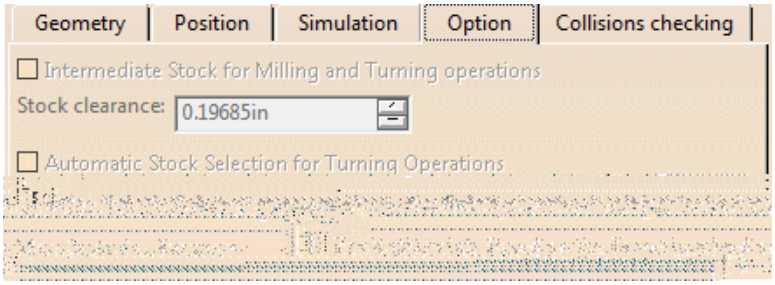
Table Center Setup Defines the offsets from the machining axis to the center of rotation for a rotary table machine

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

Simulation

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

Option



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

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

Use Spindle Axis... Allows the tool tip points to be computed based on the spindle rather than the default reference machining axis system

Toolpath Points... Allows the toolpath points to be computed based on the turret axis system rather than the part operation axis system

Collisions checking

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

On geometries

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

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

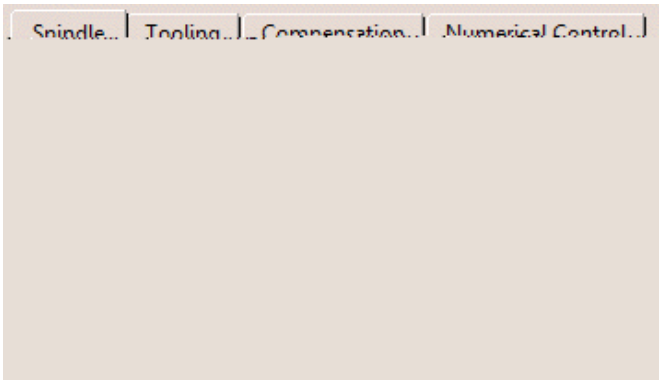
Offset on tool... Sets the offset on the tool holder

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

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

At this point, you are ready to be

Spindle



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

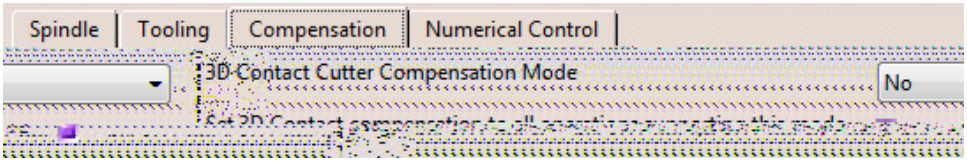
Orientation I, J, K Defines the initial orientation of the tool

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

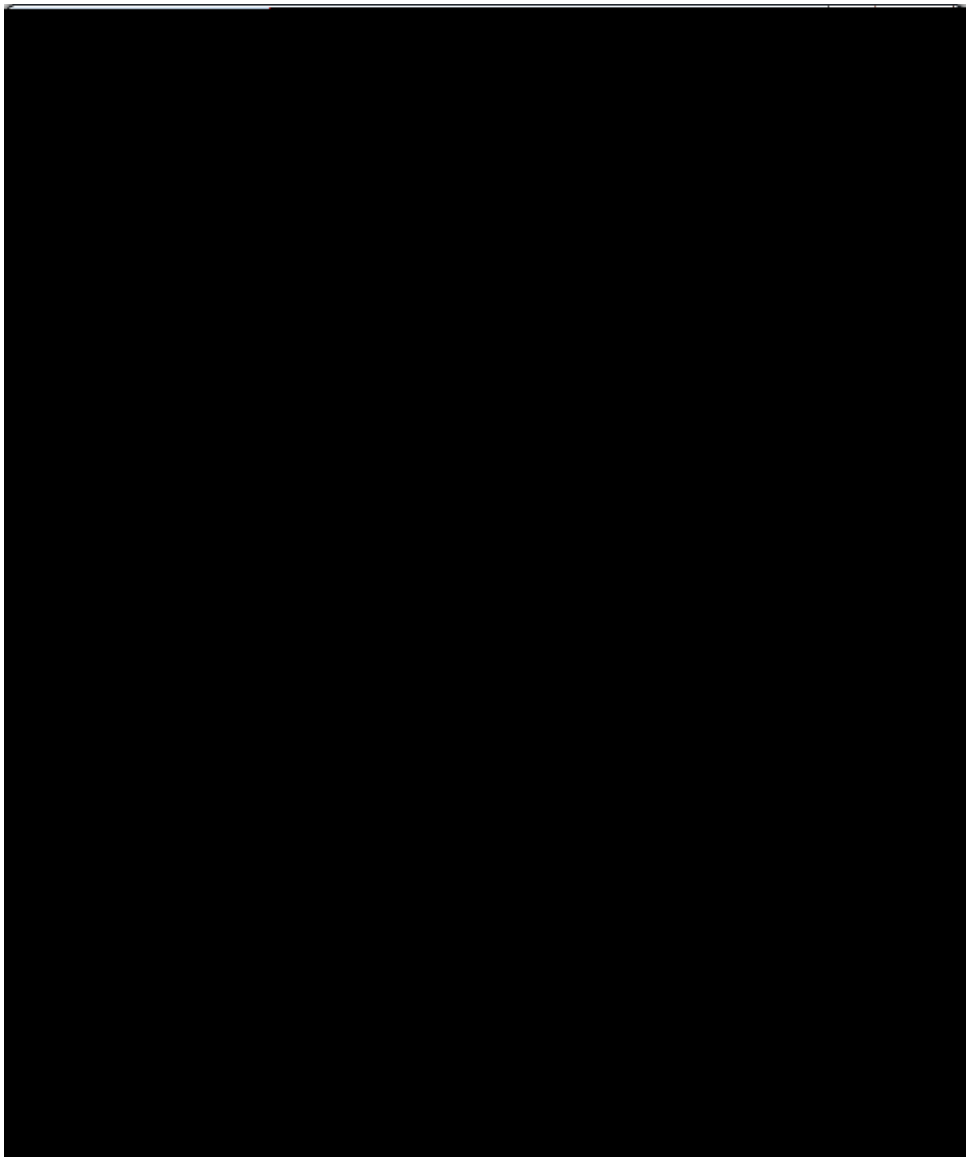
Since you are not starting with a machine seed, you will need go through and set all the options.

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

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

Switch to the *Numerical Control* tab, change the *Post Processor words table* to *ICAM_Inch.pptable*.

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



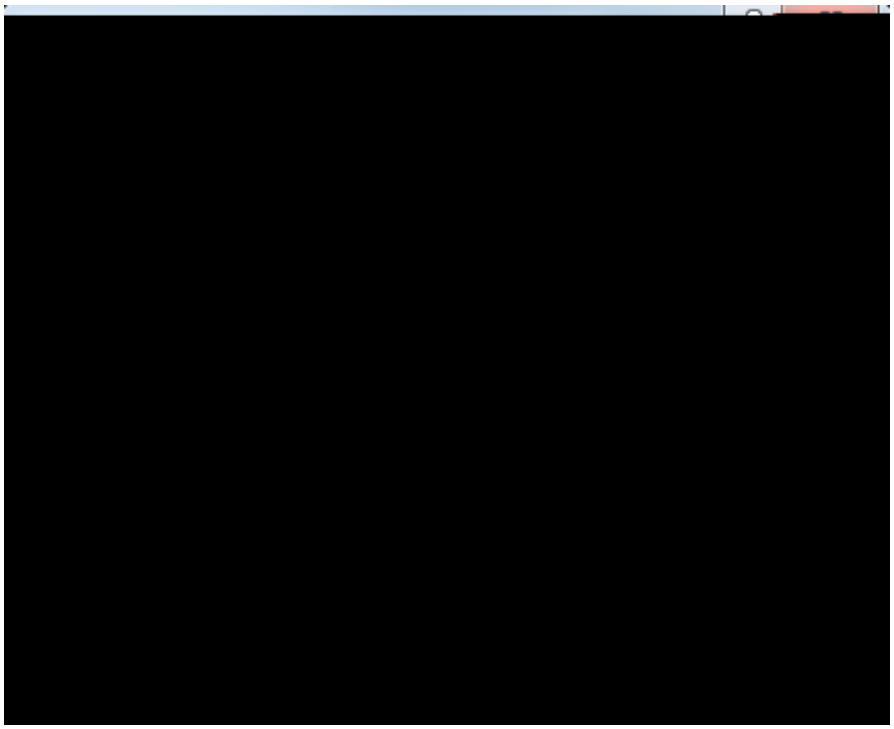
Select *OK* when done. This will have the machine set up.

Always be sure the machine parameters are set properly, otherwise you will get inaccurate output in the AP ѧ rl p- ъ ˬ Y Γ ɱ ˢ e Õ dnaccu

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.

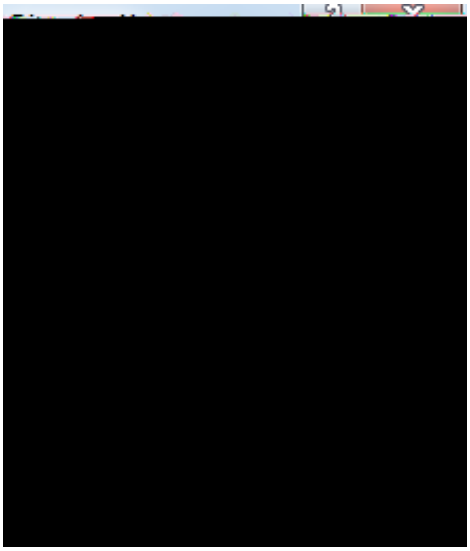
Axis Name This defines the name of the axis system, and the name that will be displayed in the graphical workspace

Machine Axis This sensitive area allows you t

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.

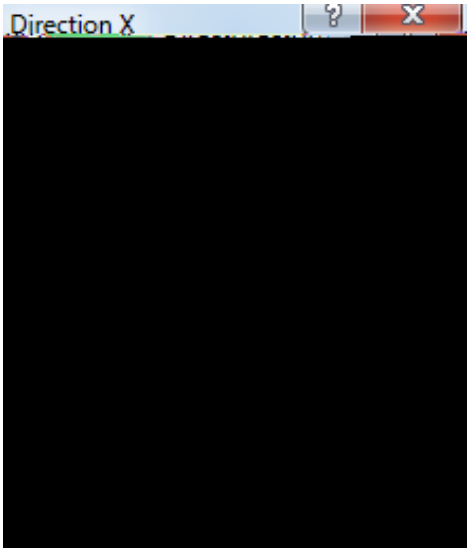
Select the X axis arrow from the machining axis system window. This will allow you to move the X axis. This will also display the *Direction* window.



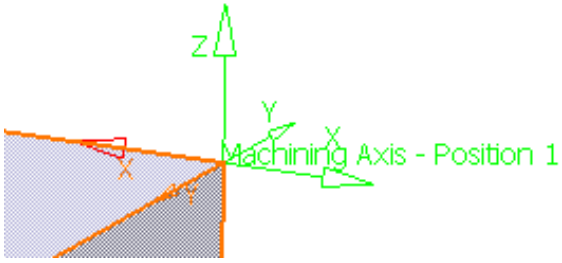
There are three methods to define the axis direction.

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 l ; ȳ o я f ise t h` = w̄ T e



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 0 D F K L Q L Q c n a g i n .V

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

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

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

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

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

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

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

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

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

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

Product or Part Definition

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

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

Had you started with a blank process, or a new part operation, you would find that the field would be blank, no product would be shown, and you would need to select the 3URGXFW RU 3DUW window in order to define the product. You will use this icon in more depth when you start defining multiple part operations.

Geometry Definition

The geometry definition is another important area for defining all the necessary geometry that you will be machining. If you remember from earlier, you have options to define the design part, stock part, fixtures, and safety planes. You shou

Select the

Replaying

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

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

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

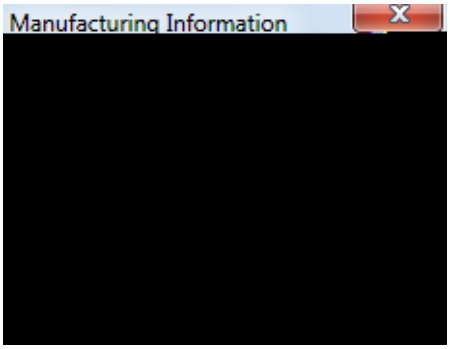
Since none of the machining opera

Start video simulation

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

Starts Machine Simulation

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

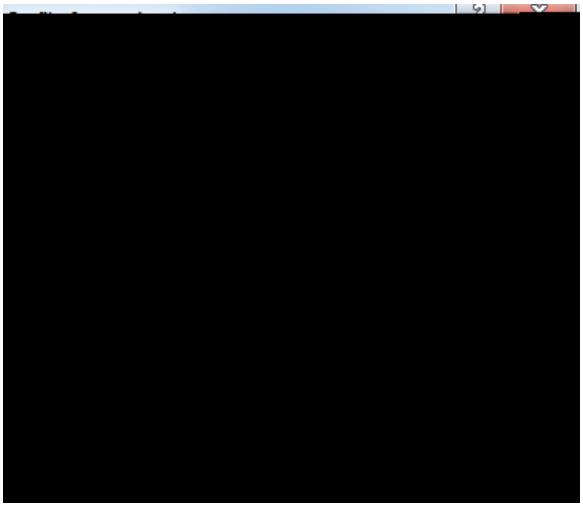


Select *OK* to the *Manufacturing Information* window. Now the tool path has been computed.

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

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

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



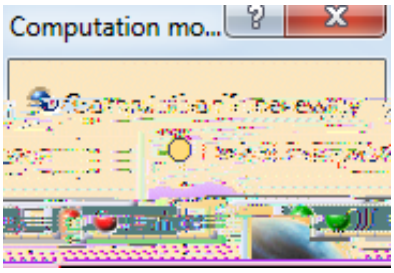
You will investigate the various aspects of the replay window, shortly.

Select *OK* to the replay window.

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

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

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



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

This will now go through and compute the remainder of the tool paths. For long programs this may take some time. Once done, the *Manufacturing Information* window will display indicat***OK*** ! ! " Á o z

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

Skip to Beginning (F5)	Skips the tool to be beginning of the tool path
Play Backwards (F6)	Replays the tool paths backwards
Pause	Pauses the tool path replay
Play Forwards (F7)	Plays the tool path forwards
Skip to End (F8)	Jumps the tool path to the end
Speed	Allows you to speed up or slow down the tool path animation replay. *Note: In Prismatic Machining, many of the tool paths run fast, and cannot be slowed down. The speed control does become more useful when working with Surface Machining operations.*

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

Start	Defines the point where the replay will begin
Current	Specifies what point the replay is at currently
End	Defines the end point of the replay

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

Feedrate Indicates the feedrate at any given point for the motion

X= Y= Z= Displays the current X, Y

Sectioning of a Tool Path 6SHFLILHV WKLV WRRO SDWK DW HDI
OHYHO 7KLV SURJUHVVLRQ ZLOO J
JRLQJ IRUZDUG RU EDFNZDUG DW W
EXWWRQ

Visualization Mode &RQWUROV KRZ WKH WRRO LV YLVXDOL
UHSOD\

Last tool position 6KRZV WKH ODVW WRRO SRVLWLRQ

Tool axis 6KRZV WKH ODVW WRRO SRVLWLRQ
D[LV YHFWRU WDQHDFRQJRWKH UHSO
SDWK :KHQ LQ VXUIDFH PDFKLQLQ
DEOH WR YLHZPDRWLRQRVDVWH\ DU
PDGH

All tool positions 6KRZV DOO WRRO ORFDWLRQV 7KL
\RX Z@ W•À— IRDV V™”fP „€0™”dORE 0

Contact Point Display Mode $ O O R Z V V X U I D F H P D F K L Q L Q J R S H U D W
W K H W R R O F W R Q W R O H U R Q W D F W H S R L Q W Z
V X U I D F H) R U W K L V F O D V V W K L V R S
D Y D L O D E O H R U X V H G

TRACUT Display Mode & R Q W U R O V Z K H W K H U 7 5 $ V S O D \ H G R U

No TRACUT Displayed 7 5 $ & 8 7 V D U H Q R W V K R Z Q L Q W K H

TRACUT Displayed 7 5 $ & 8 7 V Z L O O E H G L V S O D \ H G

Holde

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

Associate video results... Allows you to save the video results with the operation. Only one video result can be assocbe

Analyze	Allows you to analyze the part for gouges, remaining material, and clashes once the tool paths are complete
 *Analyze*	Displays graphical color gradients of remaining material and gouges
 *Video Measure*	Allows you to measure video results for accuracy
 *Measure*	Shows the cut part, and then show a series of measurement tools that will allow you to manually check the part for accuracy. This option is for use with photo results
 *Remove Chunks*	Allows you to remove material from the CGR results that would n. A γ f

Select the Forward Replay icon.

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

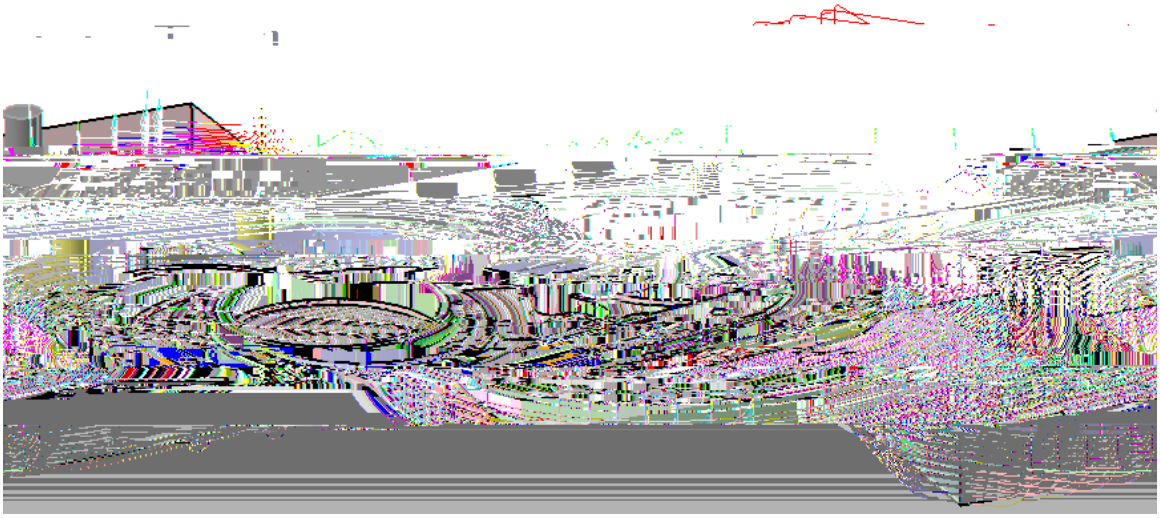
Select *OK* to exit the replay.

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

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

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

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



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

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

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

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

