Pro/E Wildfire, Tutorial_3

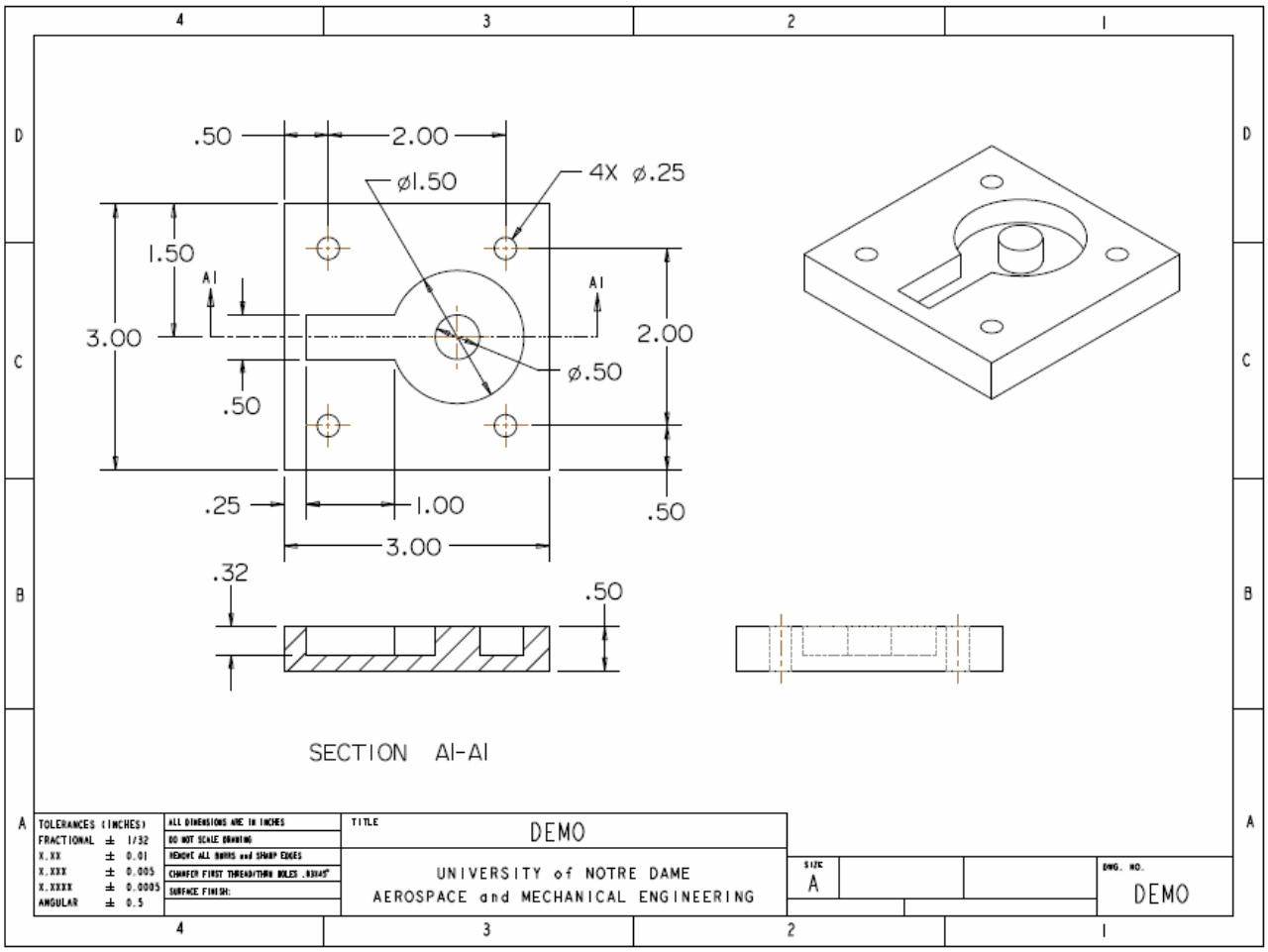


Figure 1: Part to Machine

Creating the Part

In this tutorial we will create and machine the part shown in Figure 1. The first step of this tutorial will be to create the part (demo.prt).

Step 1: Start a new part model using Pro/ENGINEER's default template.

Name the part … demo.

Step 2: Model the part as shown in Figure 1.

Step 3: Save your part model.

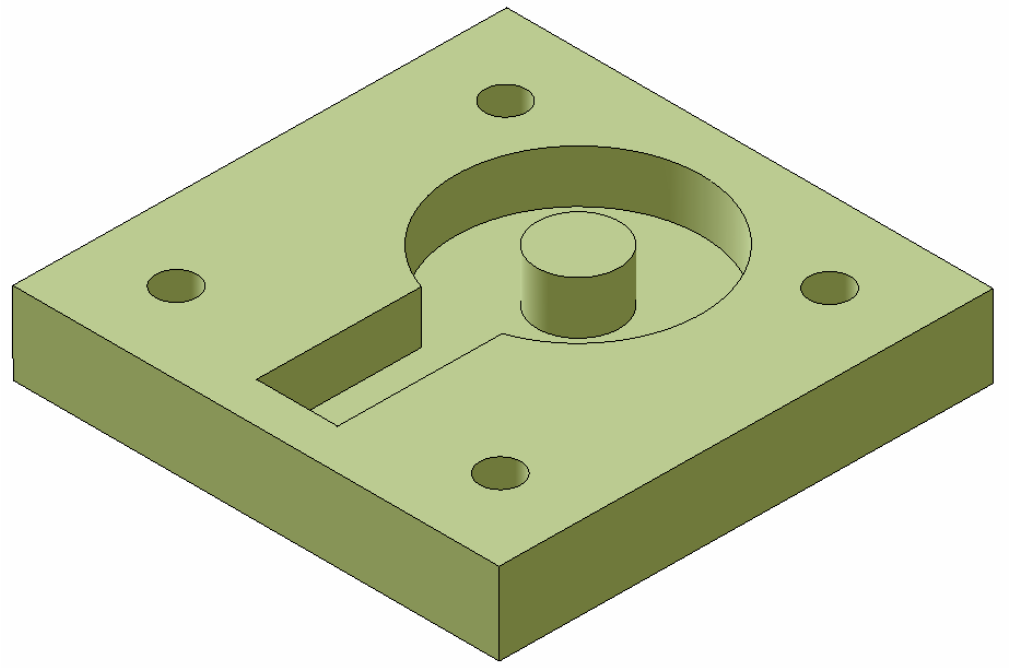


Figure 2: Finished part

Establishing a Manufacturing Model

This segment of the tutorial will establish the manufacturing models used within this tutorial. Two models are utilized: design part and manufacturing. The design part model is the part that is to be produced by the manufacturing code generated in this tutorial. The manufacturing model is the collection of the design part, assembly model and the manufacturing processes generated within the manufacturing object file.

Step 1: Select **FILE >> NEW**.

Step 2: Select Manufacturing Type on the New dialog box, then select **NC ASSEMBLY** as the Sub-type, **Name >> Demo_mfg** (Figure 3a)

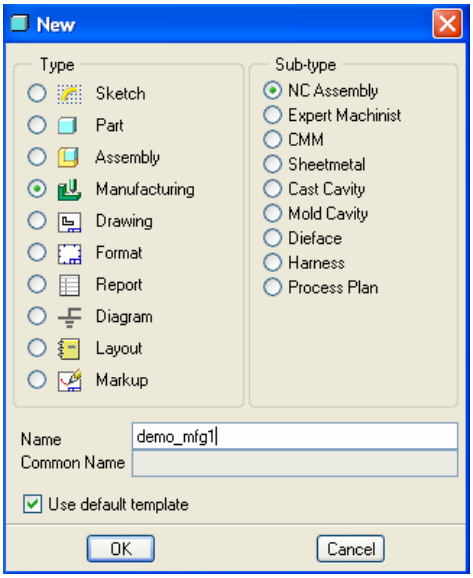


Figure 3a: New File Dialog Box

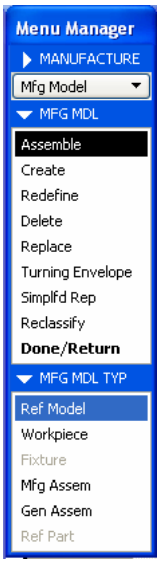


Figure 3b: Reference Model

Step 3: Select **MFG MODEL >> ASSEMBLE >> Ref MODEL** on the Menu Manager. (Figure 3b)

Step 4: Open Demo.prt as the part to manufacture.

Your block part represents the design model. It will be used as geometry to define tool paths within your manufacturing model.

Step 5: Using the **Assemble Component at Default Location** Constraint Type, constrain the location of the reference model. Choose **OK** to exit the Component Placement menu.

Step 6: Select **DONE/RETURN** to exit the Manufacturing Model (MFG MDL) menu.

Setting the Manufacturing Environment

In this section of the tutorial, you will establish the manufacturing environment. Manufacturing mode provides the Manufacturing Setup (MFG SETUP) menu to establish specific settings for your model. Within this menu, examples of items that can be set include the machining workcell, tooling, the machine coordinate system, and fixtures. You must define a workcell and a coordinate system before you can start creating NC sequences.

Step 1: Select MFG SETUP on the Menu Manager.

Pro/ENGINEER will launch the **Operation Setup** dialog box. An operation is one specific setup of a machine tool for the manufacturing of a design. It can consist of multiple NC Sequences. Within any operation, the minimum required setup includes a machine work cell and a machine coordinate system.

The Operation Setup dialog box (Figure 4a) contains the following elements:

Operation Name >> The operation name identifies the operation within the manufacturing process. The default operation names have the format OP010, OP020, where the number gets automatically incremented by the system. You can type any name.

NC Machine >> The name of the machine tool (workcell) used to perform the operation. If you have set up some machine tools prior to creating the operation, their names appear in the NC Machine drop-down list.

Fixture Setup >> This section contains the icons for creating, modifying, and deleting fixture setups. The drop-down list contains the names of all the fixture setups defined for the operation, with the name of the currently active setup displayed in the list box.

Machine Zero >> Select or create the Program Zero coordinate system, to be used for NC output and for other machining references.

The Retract group box >> Specify how the tool retracts between the cuts.

Surface >> Set up the retract surface.

Tolerance >> Controls maximum deviation of the tool when it moves along a non-planar retract surface. The default is 0.1" (in English units) or 1 mm (in metric units). You can type any value.

Stock Material >> Select a name of the stock material.

Step 2:  Select the Machine Tool icon on the Operation Setup dialog box.

Step 3: On the Machine Tool Setup dialog box, enter the parameters shown in Figure 4a.

Enter the following parameters:

- **Machine Name:** 3-axis-mill
- **Machine Type:** Mill
- **Number of Axes:** 3 Axis

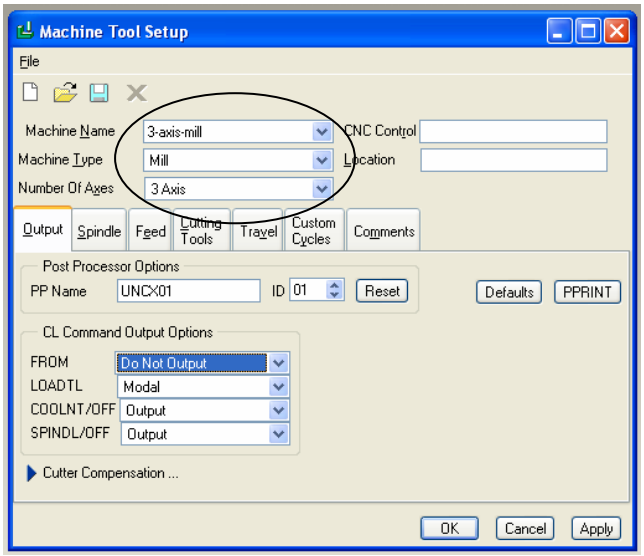


Figure 4a: Machine Tool Setup

Step 4: Select **OK** to exit the Machine Tool Setup dialog box.

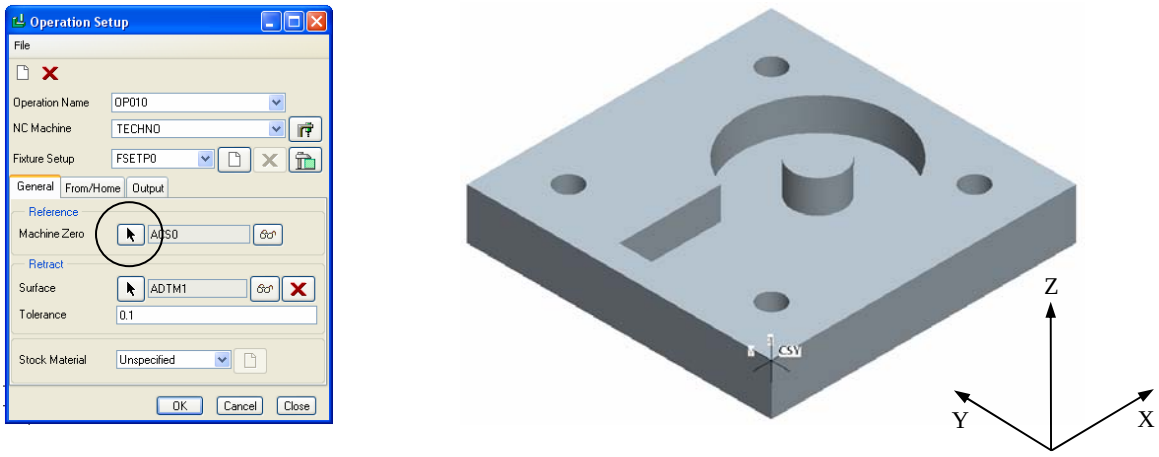


Figure 4b: Coordinate System Definition

Step 5: On the Operation Setup dialog box, select the **Machine Zero** pick icon (Figure 4b).

Step 6: Select/create the coordinate system on the model. (Z pointing up)

Step 7: Select the Retract Selection box. Select **ALONG Z AXIS** a value of 0.75.

Figure 8: Retract Selection dialog box

Step 8: Select **Apply** and **OK** to exit the Operation Setup dialog box.

Step 9: Select **DONE/RETURN** to exit the Manufacturing (MFG) Setup menu.

Step 10: Save your manufacturing model

Step 11: Select **Apply** and **OK** to exit the Operation Setup dialog box.

Step 12: Select **DONE/RETURN** to exit the Manufacturing (MFG) Setup menu.

Step 13: Save your manufacturing model.

Facing Sequence

Within this segment of the tutorial you will define a facing operation to mill the top surface of the part to ensure that the surface is level. In order, the parameters that you will set include the cutting tool, the tool's parameters, the retract depth, and the surface to face.

Step 1: Select MACHINING on the Menu Manager.

Step 2: Select NC SEQUENCE on the Machining menu.

Step 3: Select MACHINING >> FACE >> DONE.

Step 4: Check the Machining Parameters shown in Figure 6:

Pro/ENGINEER will check the **minimum** parameters required for a specific machining operation. Notice in Figure 6 the parameters that are checked and the parameters that are unchecked. Many of these selections, such as name and comments, are optional. Others, such as Tool and coordinate system (Coord Sys), are required. The Coordinate System option is not checked since it was defined in the previous segment of this tutorial.

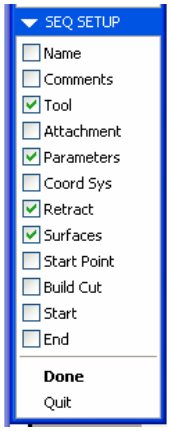


Figure 6: Sequence Setup

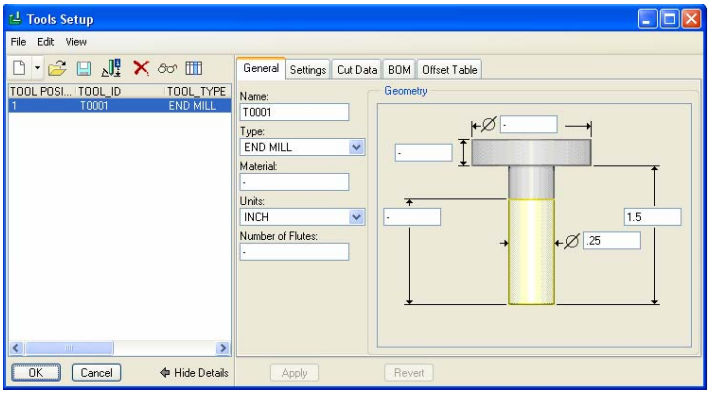


Figure 6a: Tool Setup Dialog Box

Step 5: Select DONE on the Sequence Setup menu (Figure 6)

Step 6: Enter the tool parameters shown in Figure 6a.

- **Name:** T0001
- **Tool Type:** End Mill
- **Diam:** 0.25
- **Length:** 1.5

Step 7: Select **APPLY** to create the tool, Select **OK**.

Step 8: Select **SET** on the Manufacturing Parameters menu.

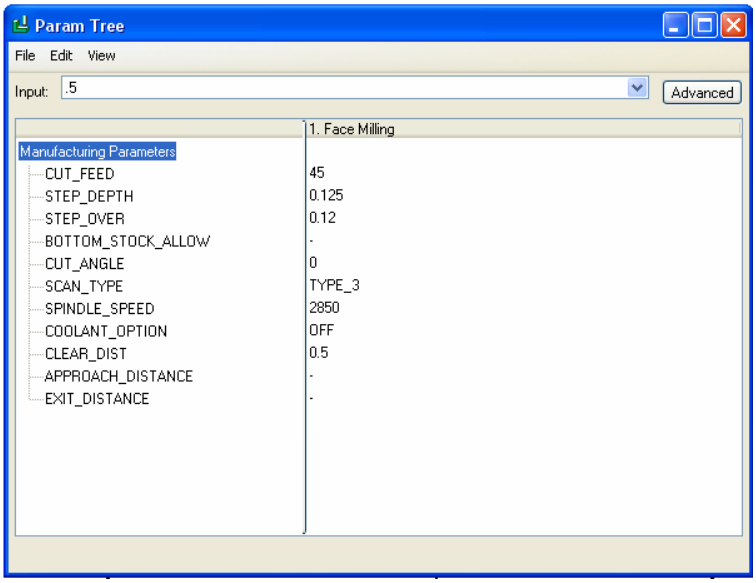


Figure 6c: Manufacturing Tool Parameters

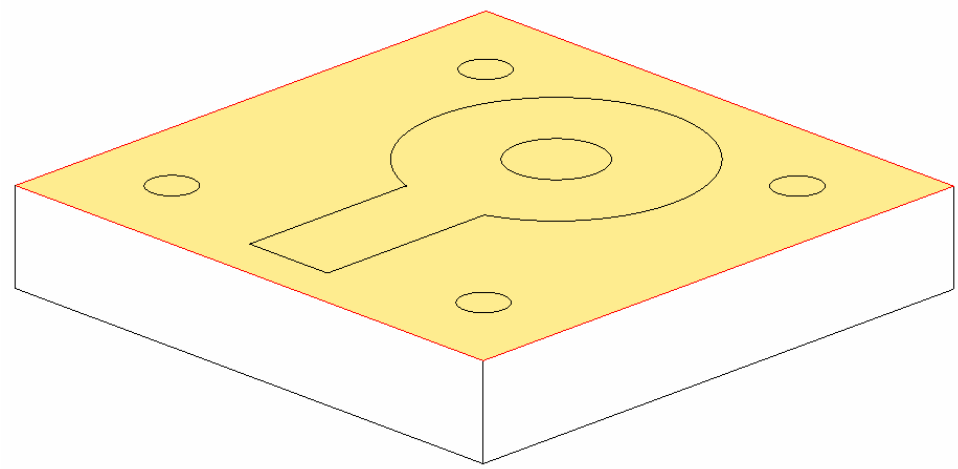
Step 9: Enter the tool parameters shown in Figure 6c.

Step 10: After the values in Figure 6c are set, select FILE >> EXIT to save and exit the Parameter Tree dialog box.

Step 11: Select **DONE** on the Manufacturing Parameters menu.

Step 12: Select **Mill Surface>> DONE >>**  on the menu bar.

The surface to be milled in this NC Sequence is the "complete" top surface of our part. Therefore, we need to create a surface which covers the top of our part.



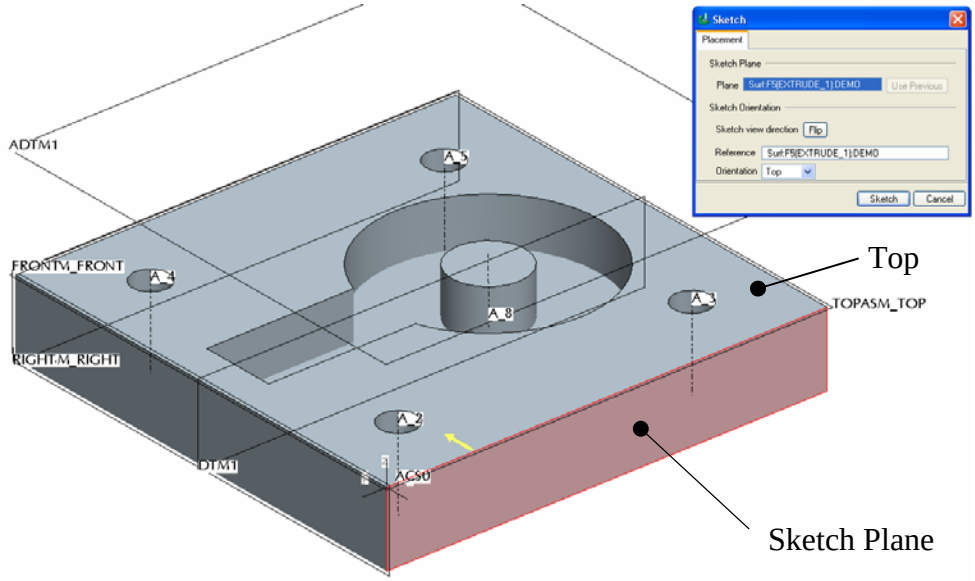


Figure 9a: Extrude Orientation.

Step13: Select  Extrude and choose the side surface shown in Figure 9a as a sketch plane and orient the sketcher using the "Top" surface as the top.

Step 14: Select the  "use edge" and select the top edge to extrude. Check out of the sketch environment. Use the  UP TO SURFACE depth option and select the "opposite" edge of the part! (Figure 9b)

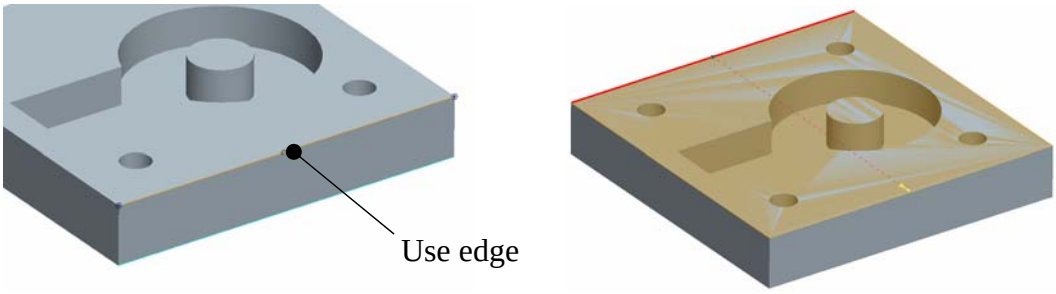
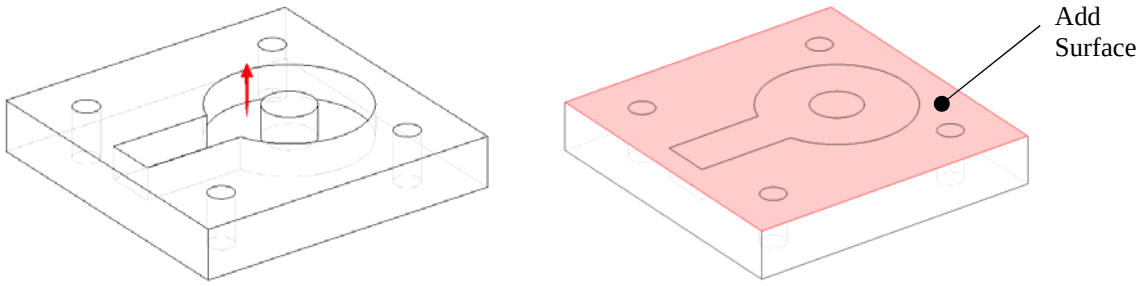


Figure 9b: Sketcher

Step 15: Select  to apply and make changes and  to check out.

Step 16: Make sure the arrow faces upward for the surface to be machined >> **Okay**. (flip)



Now you must "Add" the surface you created to the sequence.

Step 17: Select the surface you just created, choose **OK** on the Select dialog box and **Done/Return** on the Add Surfaces menu.

Step 18: Select **PLAY PATH** on the **NC SEQUENCE** main menu.

Step 19: Select **SCREEN PLAY** on the **Play path** menu.

Step 20: Use the Play Path dialog box to play the NC Sequence.

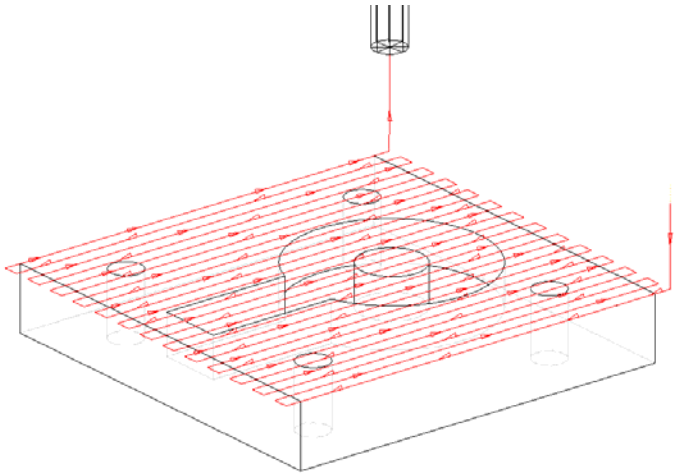


Figure 9d: Play Path

Step 21: Close the Play Path dialog box.

Step 22: Select **DONE SEQ** on the NC Sequence menu.

Step 23: Save your manufacturing model.

Create a Mill Volume

This new sequence will mill the volume shown in Figure 10. We will use the same tool as the previous sequence, an end mill, .25 dia.

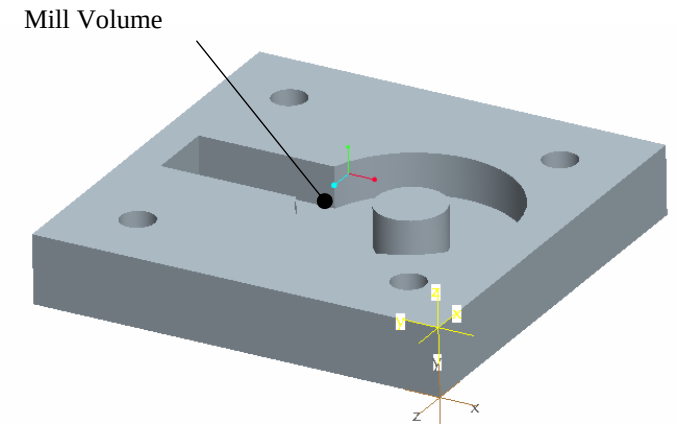


Figure 10: Volume Milling

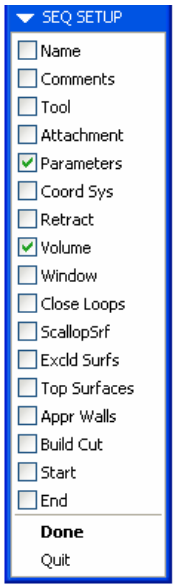


Figure 11: Sequence Setup Parameters

Step 1: On the Machining menu, select the NC SEQUENCE >> NEW SEQUENCE option.

This specific NC sequence will consist of a volume milling operation.

Step 2: Select MACHINING >> VOLUME >> DONE.

Step 3: Select the parameters shown in Figure 11 to define for this specific NC Sequence, select DONE.

In order, the settings that will be defined include: the tool parameters and the mill volume. We will use the same tool and retract plane as we defined in the previous sequence.

Step 4: Select **SET** on the Manufacturing parameters menu.
The next requirement is to define specific tool settings. Examples of tool settings include cutting speed and spindle speed.

Step 5: Enter the tool parameters shown in Figure 12. They are the same as was used before. You could also have selected Use Previous.

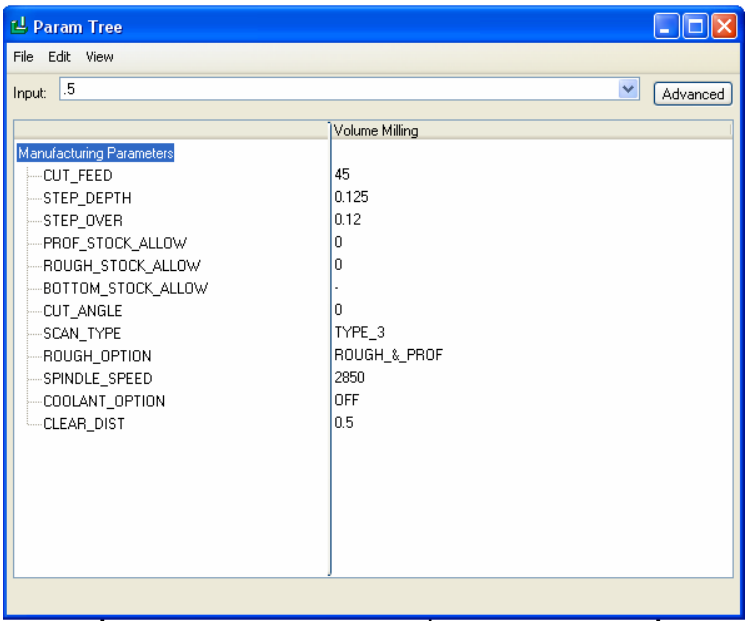


Figure 12: Manufacturing Tool Parameters

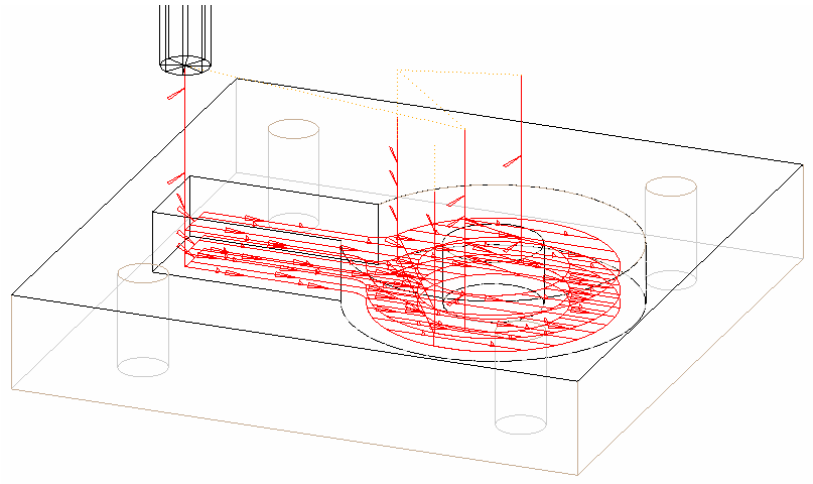
Step 6: Select FILE >> EXIT on the Parameter Tree dialog box.

Step 7: Select **DONE** on the Manufacturing Parameters menu.

Step 8: Select  the Mill Volume Tool icon and then select  the sketch tool.

You will define this mill volume by sketching the area to mill. You will sketch this area as a rectangle and extrude the section the depth of the pocket (0.32 inch). The actual mill volume will be defined by trimming empty space from this sketched volume.

You finish this sequence …



Save your manufacturing model.

Holemaking Sequence

Within this segment of the tutorial, you will create an NC sequence that will machine/drill the part's holes.

Step 1: Select NC SEQUENCE on the Machining menu.

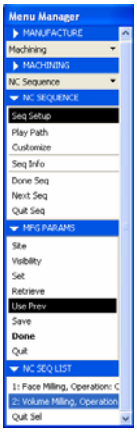
For this tutorial, the machining operations (facing, volume milling, and drilling) will be performed on one workcell (machine tool) with one setup. Due to this, the four sequences defined within this tutorial will be defined under one operation. A new operation can be established with the Operation option.

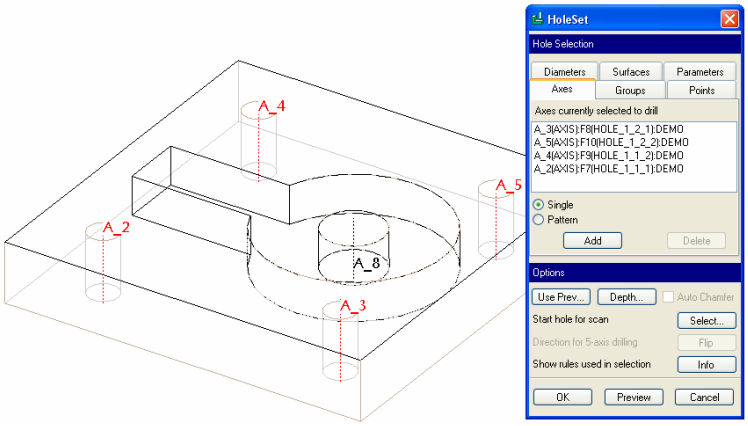
Step 2: Select NEW SEQUENCE.

Step 3: Select MACHING >> HOLEMAKING >> DONE >> Standard >> Done.
(profile will not work with .25 dia tool)

You finish this sequence …

Select USE PREVIOUS on the Manufacturing parameters menu and VOLUME MILLING from the NC Sequence List menu as shown below.





Save your manufacturing model.

Profiling Sequence

Within this section you will create an NC sequence to profile around the part.

Step 1: Select NC SEQUENCE on the Machining Menu.

Step 2: Select NEW SEQUENCE

Step 3: Select MACHINING >> PROFILE >> DONE.

Step 4: Check the following setup operations: Parameters and Surfaces.

Step 5: Select DONE.

Step 6: Choose USE PREVIOUS on the Manufacturing Parameters menu and select the volume sequence.

Step 7: Select DONE on the Manufacturing Parameters menu.

Step 9: On the Ncseq Surfs menu, choose SELECT SURFACES << MODEL << DONE.

Step 10: Choose Surface on the Surf/Loop menu.

Step 11: Choose the four surfaces shown in Figure 17. Use Ctrl to select multiple surfaces.

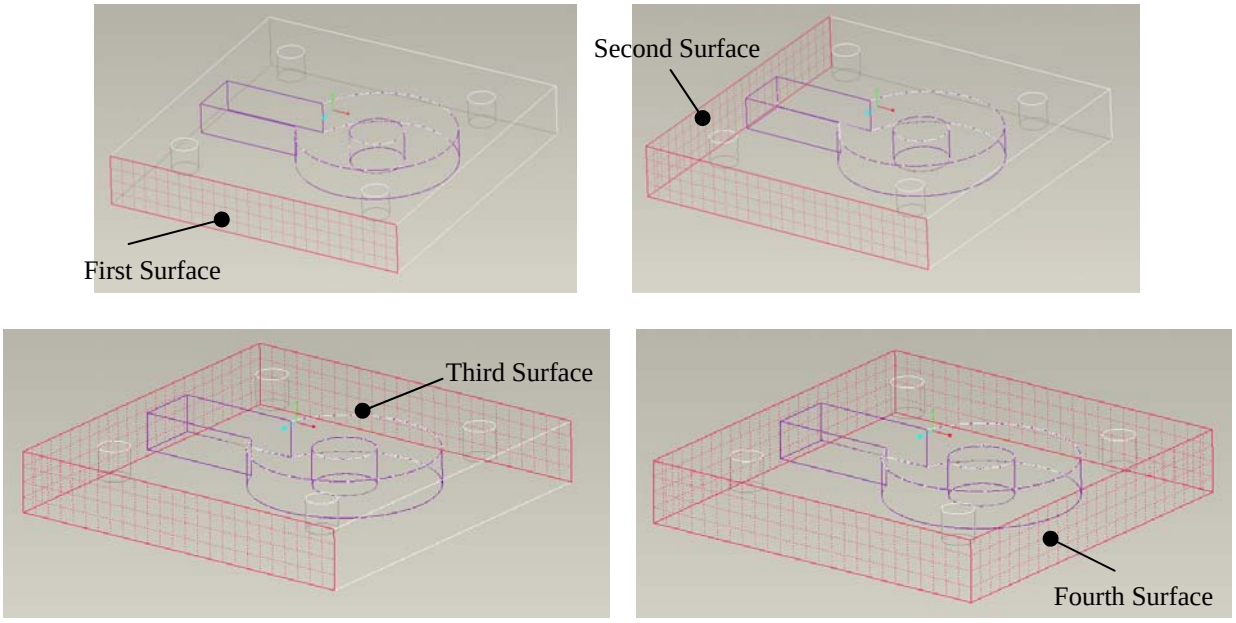


Figure 17: Choose the Surfaces to Profile.

Step 12: Choose DONE RETURN >> DONE RETURN >> DONE RETURN.

Step 13: Choose PLAYPATH >> SCREENPLAY and see that the tool profiles around the edges selected.

Step 14: Select DONE SEQ on the NC Sequence menu and DONE RETURN to exit the Machining Menu.

Step 15: Save your manufacturing model.

Outputting the Centerline (CL) Data for Individual Sequences

Within this segment of the tutorial, you output the CL data for each sequence individually. We will begin by outputting the G-code for the Facing sequence.

Step 1: Select **CL DATA** on the Menu Manager.

Step 2: Select **OUTPUT >> NC SEQUENCE**.

Step 3: Select the Facing sequence from the NC SEQ LIST.

Step 4: Select **Display** as the location of the output.

Step 5: Select **Done** on the Play Path menu. The speed of your display can be slowed or increased with the Time increment option. This should now show the tool path for the facing sequence.

Step 6: Select **File** on the Path menu.

Next, you will post-process the CL data to a specific machine tool.

Step 7: On the Output Type menu, be sure the CL FILE, INTERACTIVE and COMPUTE CL options are selected.

Step 8: Select **Done** on the Output Type menu.

Step 9: On the Save As dialog box, enter Face as the name for the CL file, then select OK.

Step 10: Select **Done Output** on the Path menu and then select **Post Process** on the CL DATA menu. Select the file Face.ncl from the Open Dialog box, select **Open** to close the window. In the PP Options menu make sure the Verbose and Trace options are selected.

Step 11: Select **Done** on the PP OPTIONS menu and a list of post-processors appears.

The specific machine tool we have in-house is:

uncx01.p20 Fanuc 16M controller (Milltronics)

Choose the UNCX01.p20 Option.

IMPORTANT: Only the Windows computers in the CAD Lab (303 Cushing), Learning Center, and Fitzpatrick Cluster have the proper post-processor for the Techno cells in B19. All files that you intend to machine MUST be post-processed using these computers. All other work can be done on Linux machines, but post-processing MUST be completed on one of the above mentioned Windows computers.

Step 12: Close the information window and Select **DONE/RETURN** from the CL DATA menu.

Step 13: Save your manufacturing file.

Step 14: Repeat the above process for each of the individual sequences you created, name the files Holes and Profile.

Step 15: Be sure to save the files in a directory you will have access to in B19 >> N drive >> AFS space