Chapter 2. A Case Study for CAD/CAM/CAE

[illegible]tive of this chapter is to give a quick overview on how UG can [illegible]CAM/CAE. A simple part is used to take you through step-by-step from design, analysis, then to manufacturing of the part. The figure of the part appears in Figure 2.1. The part is made of stainless steel. One functional performance requirement of the part is that the maximum displacement of the part (i.e., deformation) is less than a threshold value, .01 inch, under the following two conditions. The first condition is that 10,000 lb of force is applied to the hole downward. The second condition is that the fixed constraint is applied to the back long rectangular face such that this face is not allowed to move.

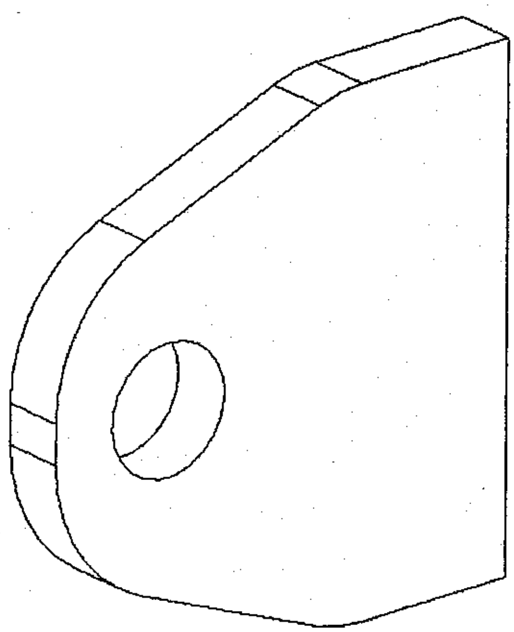
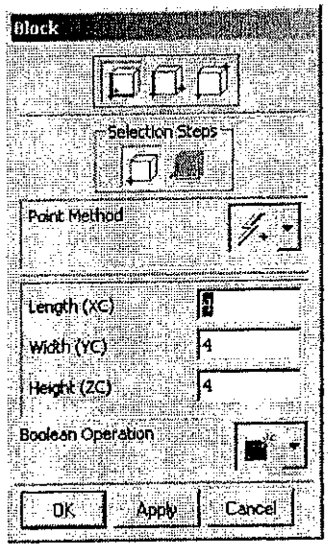


Figure 2.1 A case study part

2.1 CAD for 3-D solid modeling

You will first create a 3-D solid model for this part.



1.5 Click the left icon on the first row, named as "Origin, Edge Lengths"

1.6 Enter

Length (XC) = 0.5 (Use the Tab key to move to the next field)

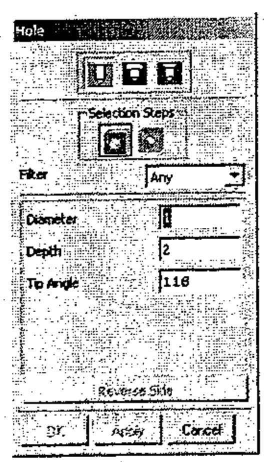
Width (YC) = 4

Height (ZC) = 4

1.7 Click "OK" or the middle mouse button (MB2). You will see the block created and located at the origin of Work Coordinate System (WCS).

Step 2. Create a thru hole in the block.

2.1 Insert → Form Feature → Hole or choose the Hole icon . A small dialog "Hole" opens.



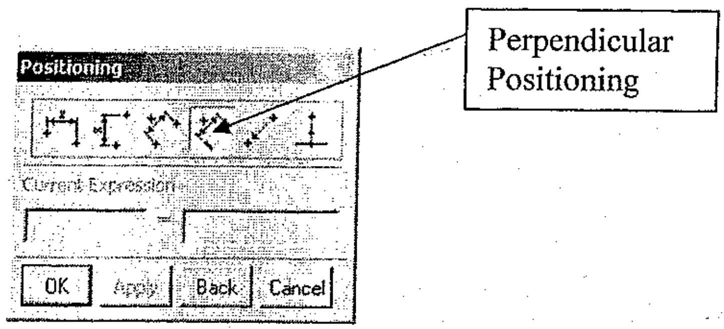
The default hole type (simple hole) and parameter values are already in place.

2.2 The Cue Line is prompting to select the planar placement face, so select the large square face of the block to place a hole on the face. As you click the face, you may see a small dialog pop out to show that there are two faces that can be selectable. If this is the case, click "1".

2.3 The Cue Line is prompting to select the thru face so select the other large square face to specify the thru face of the hole.

2.4 The hole parameters change to only requiring the diameter so enter Diameter = 1 and click "OK". A hole with diameter =1 is created.

2.5 Position the hole on the face. Dialog "Positioning" appears to guide where to place the hole on the face.



The hole center will be located 2 inch from the top edge and 1 inch from the left edge as shown below.

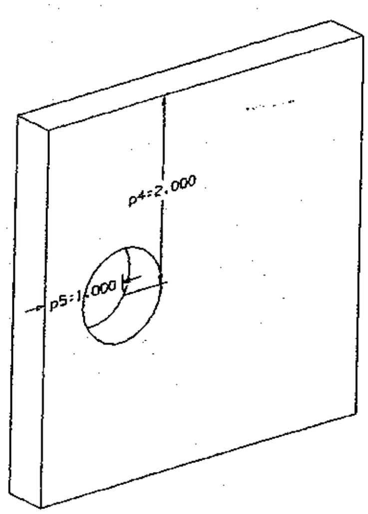


Figure 2.3 Positioning a hole on the block

You will use two perpendicular positions to locate the hole. Use the first perpendicular position by picking the fourth icon from the 1st row. Pick the upper edge of the face and enter 2 as shown in figure and click "Apply". Use the second perpendicular position by picking the same icon. This time pick the left edge of the face and enter 1 and click "OK".

Step 3. Apply a chamfer operation to two edges.

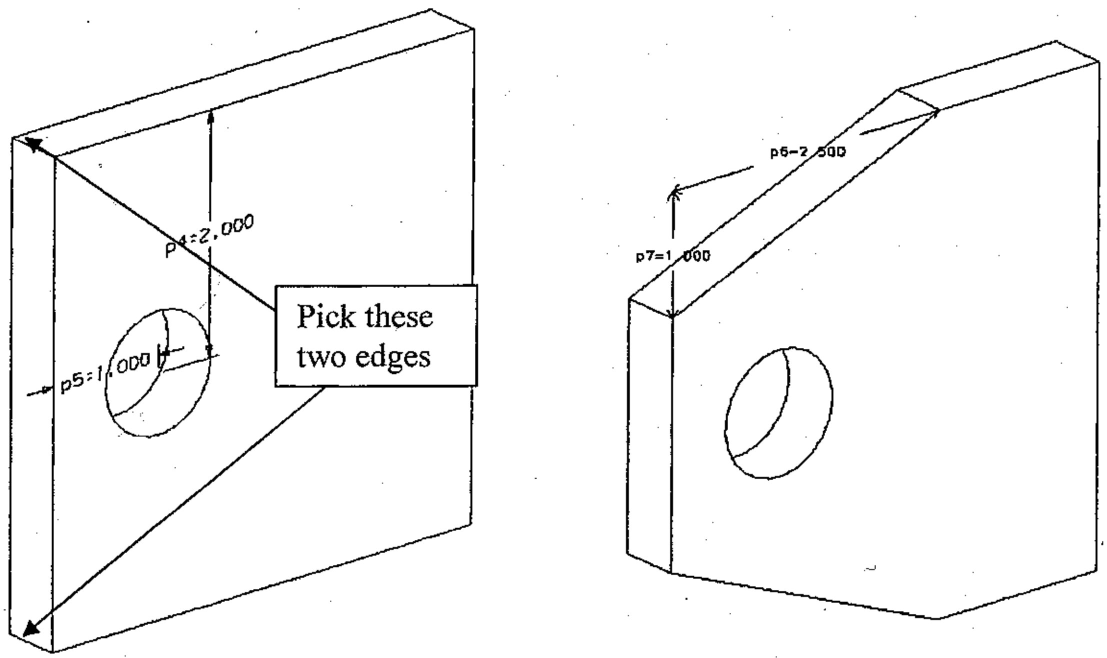
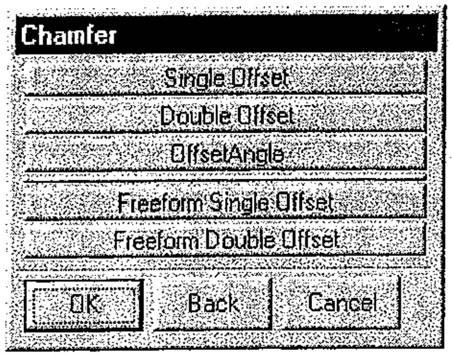


Figure 2.4 Before and after chamfering two left corner edges

3.1 Insert → Feature Operation → Chamfer or choose the chamfer icon, .

3.2 Chamfer dialog appears. Click "Double Offset".

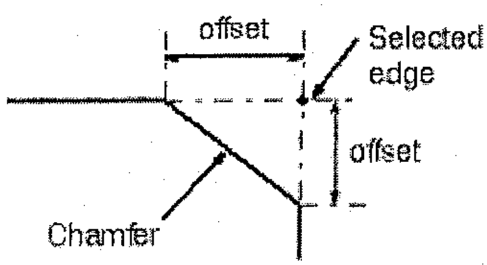


3.3 Pick two small edges that you want to chamfer, one upper left and the other bottom left as shown in Figure 2.4, and then click “OK”. As you click the edge, you may see another small dialog pop out to show that there is more than one object that can be selectable with your last click. Click among the selectable numbers one number that indicates the edge.

3.4 Enter the following chamfer offset distances and click “OK”.

First Offset = 2.5

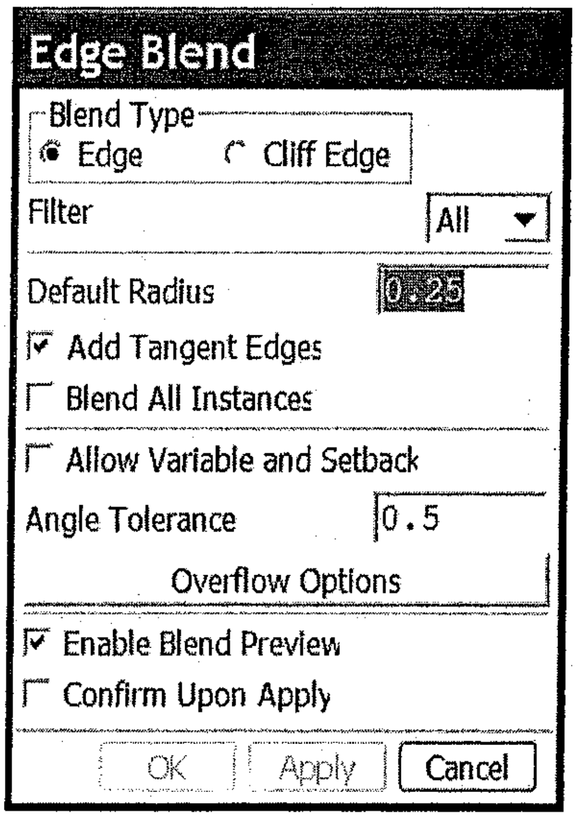
Second Offset = 1



3.5 Click “Cancel” to stop creating chamfers.

Step 4. Apply two edge blend operations to four small edges created from the chamfer operation.

4.1 Insert → Feature Operation → Edge Blend or choose the edge blend icon . The Edge Blend dialog appears.



4.2 Pick two small edges to blend, one leftmost upper and the other leftmost bottom as shown in Figure 2.5.

4.3 Enter Default Radius = 1.25 and click "Apply" on the bottom of the dialog. These two edges are blended and the Edge Blend dialog is still open.

4.4 Do the edge blend operation one more time. Pick another two small edges resulting from the chamfer operation: one small edge on the top face and the other on the bottom face.

4.5 Enter Default Radius= 1 and click "OK". Now the 3-D CAD model for this part is created.

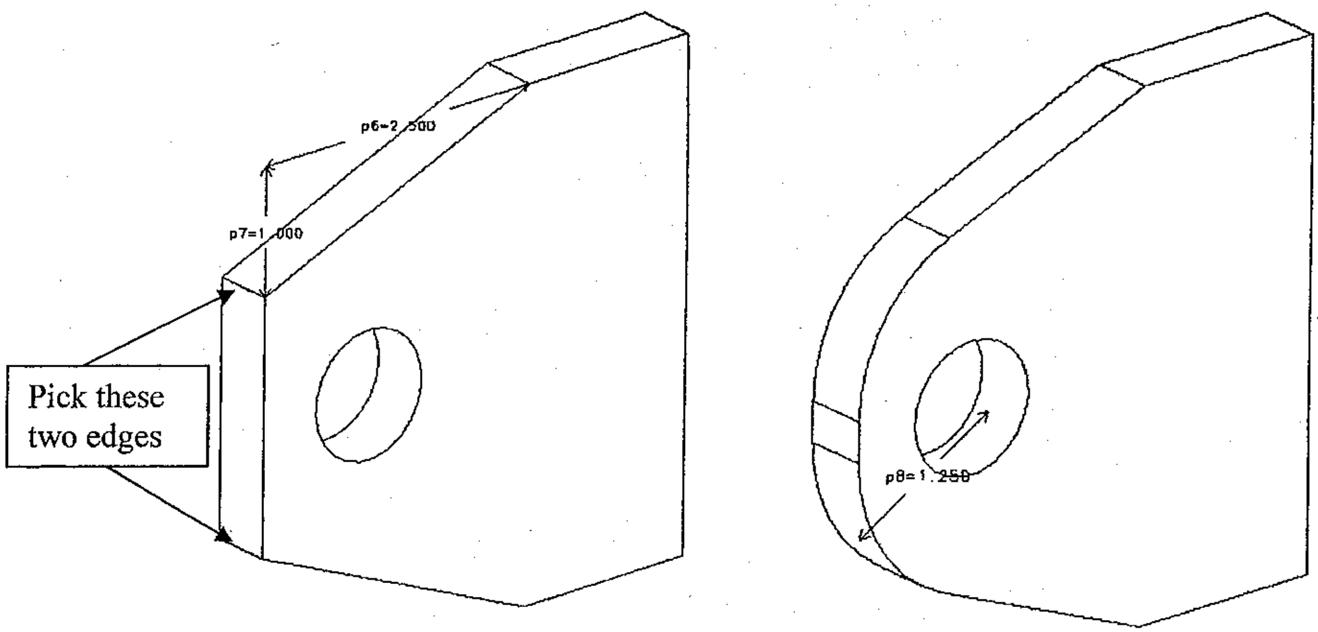
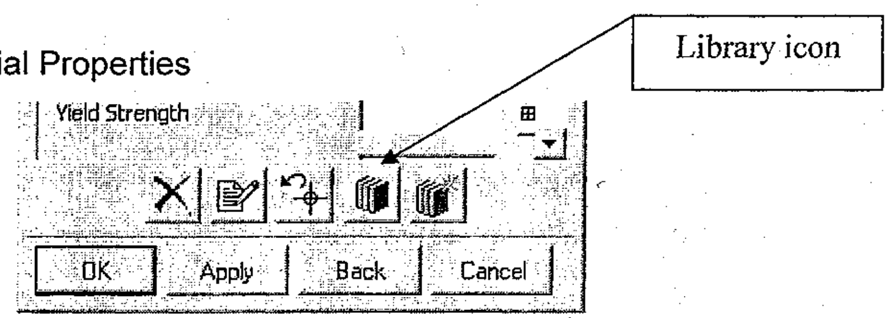


Figure 2.5 Before and after blending two left corner edges

Step 5. Assign material type, stainless steel, to this part.

5.1 Tools → Material Properties



5.2 Materials dialog appears. Click "Library" on the near bottom of the dialog.

5.3 Another dialog "Search Criteria" appears. Accept the default setting by just clicking "OK".

5.4 Pick Ref 15 "Stainless_Steel" among the list of metals found in the library and click "OK".

5.5 Pick the part in the display window and click "OK". The selected material, stainless steel, has been assigned to the part.

2.2 CAE for Design Analysis

You will find the part weight and conduct the structure analysis of the part.

Step 1. Save your model by pointing your cursor to File→Save.

Step 2. Find the Volume/Area/Weight of the block.

2.1. Choose Analysis → Mass using Solids

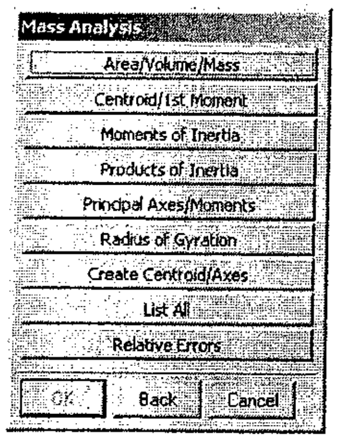
2.2. Choose OK on the dialog for Tolerances and Accuracy.

2.3. Choose OK to the menu that follows for Area = 1 and Volume = 1 as another measure of accuracy.

2.4. Next the system asks you to select the solid to analyze, so select the part in the graphics window.

2.5. Choose OK to confirm and tell the system you are finished selecting.

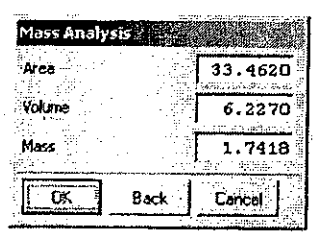
2.6. The next dialog is the menu to choose the analysis you wish to see.

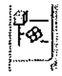


Also appearing in the graphics area are the orthographic vectors that identify the centroid and principle axes. You may choose from the list on the dialog or you may choose List All to see all of the results in a tabular form. We are interested the Area/Volume/Mass.

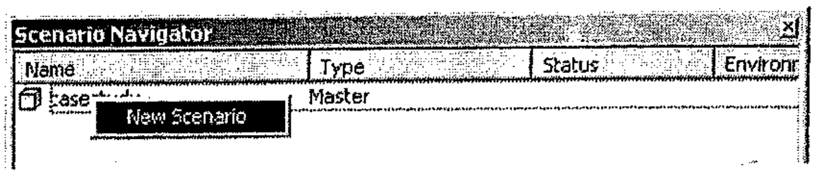
2.7. Choose the Area/Volume/Mass button.

The result is this dialog. The units are in lbs and inches.



Step 3. Select Application→Structures to enter the analysis application. If prompted with a message "...Do you want to save changes?", select Yes. And click the scenario navigator icon  located on the top of the right side column of the graphics screen (this right side column of icons is called the resource bar).

Step 4. A Scenario Navigator window comes up with our part name inside. Select the name of the part with MB1 and right click the name with MB3.

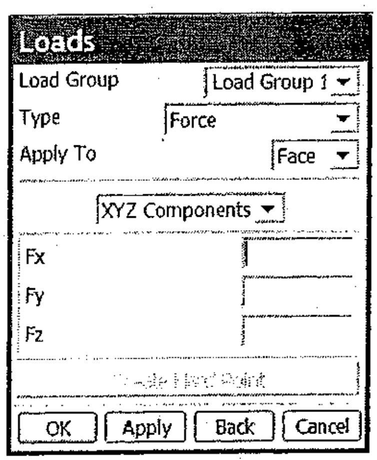


The Cue Line is prompting to create a new scenario or select an existing one so select window New Scenario as shown above.

Note the icons that are dedicated to structure analysis are added to the screen and we are ready to do the analysis.

Step 5. Apply a vertical load.

5.1. Select the Loads Icon  from the tool bar to bring up the following dialog.



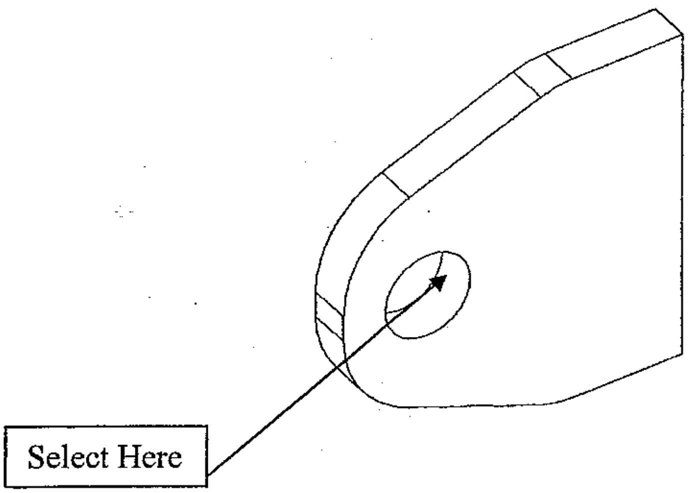
5.2 Make the following adjustments to the dialog if necessary (as reflected in the dialog shown).

5.2.1 Type→Force

5.2.2 Apply to→Face

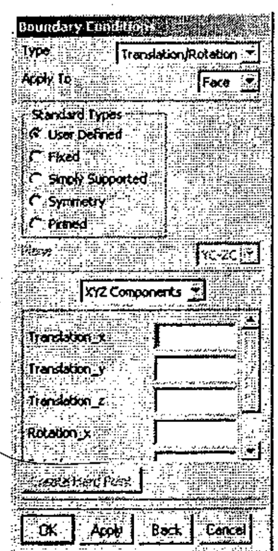
5.3 Enter in a load of –10000 pounds in the FZ field (note that this is in direct relation to the coordinate system shown on the graphics screen).

5.4 Select the face as shown below followed by OK.



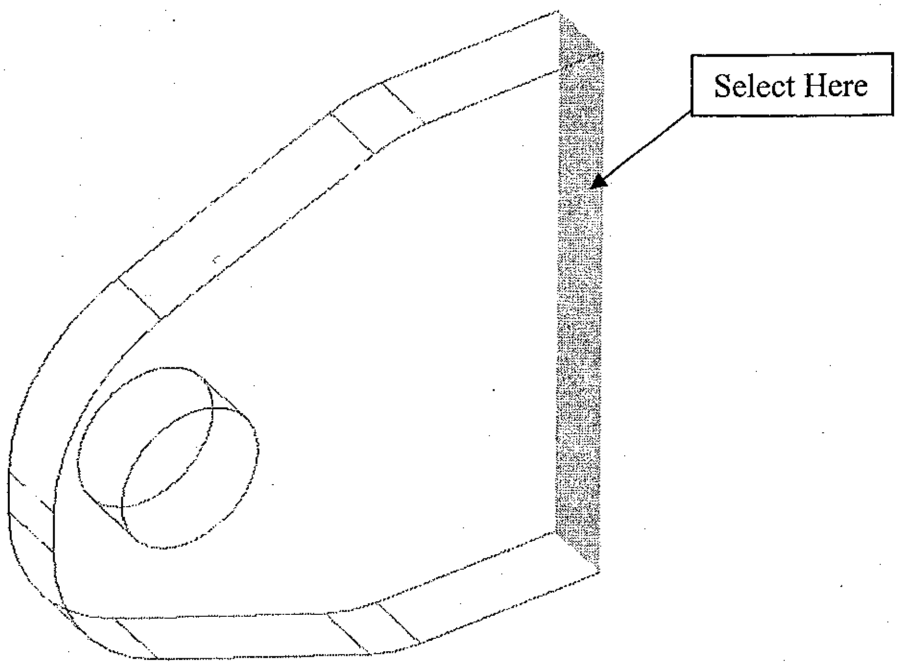
Step 6. Apply the Boundary Condition.

6.1. Select the Boundary Conditions Icon  to bring up the following dialog.

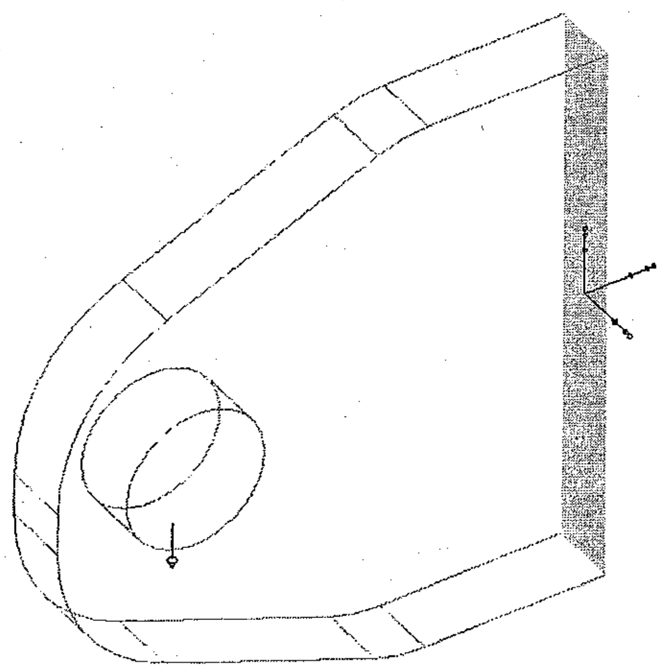


6.2. Select the Fixed radio button among 'Standard Types' options in the dialog.

6.3. Select the back face as shown below followed by OK.

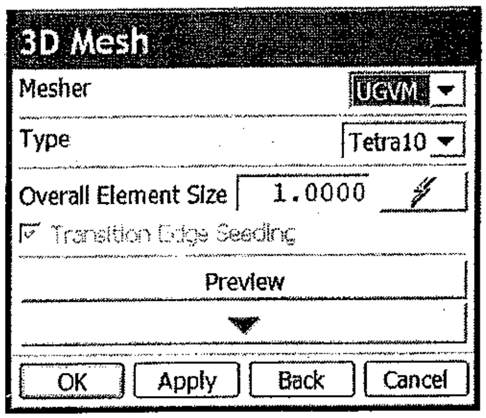


At this point, your graphics screen should look as shown below. Next, we will apply the solid mesh that will carry the load.

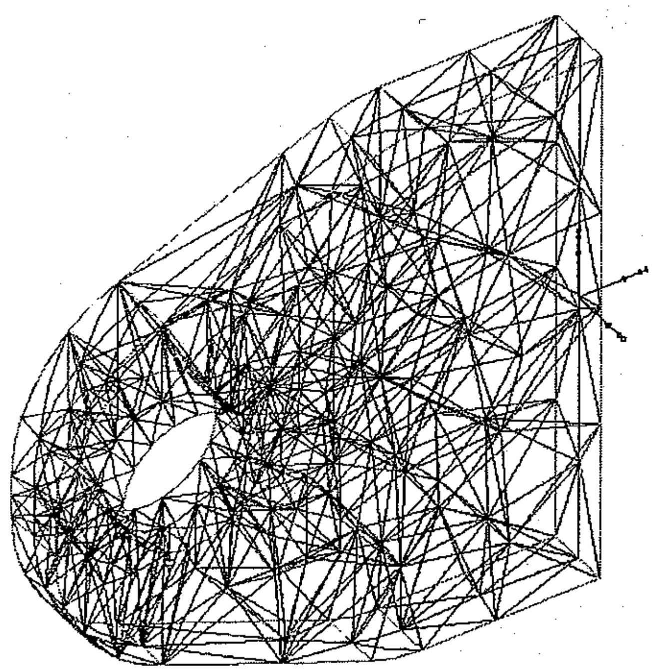


Step 7. Create the Mesh

7.1. Select the 3D Tetrahedral Mesh Icon  to bring up the following dialog.



7.2. Enter Overall Element Size=0.5 and select the solid body followed by OK to create the mesh. When complete, your model looks similar to one as shown below:



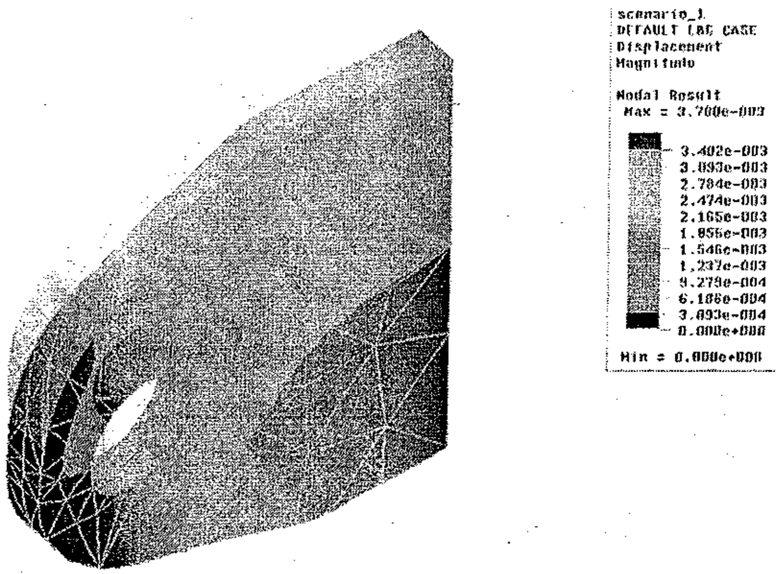
At this point, we are ready to run the analysis.

Step 8. Run the Analysis

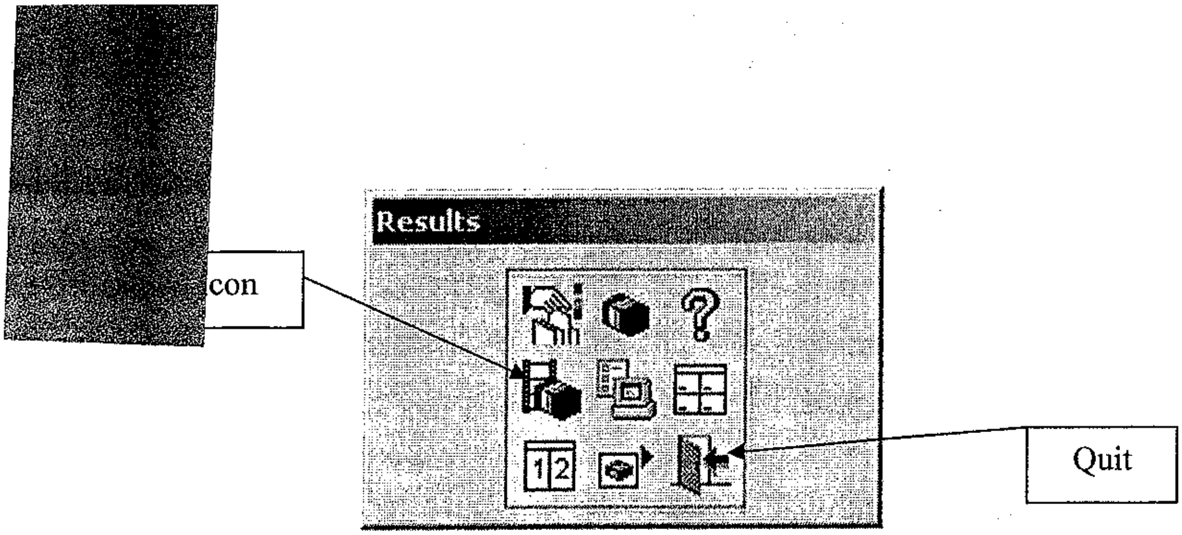
8.1 Select the Solve Icon  followed by OK to submit the analysis. A monitor window will come up so wait for it to notify complete.

Step 9. View the Results

9.1. Select the Results icon  to see the deflected shape as shown below.

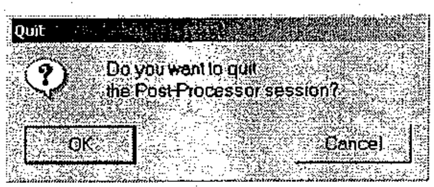


As you can see the graphical results above from the analysis, you can observe a couple of points. Relatively large displacement, colored in red, occurs around the hole where the force of 10,000 pound is applied as specified in Step 4. The maximum displacement estimated by this analysis is 3.78 x 10^{-3} inch (your number may slightly be different). This is less than the threshold value, .01 inch, thus meeting the functional performance requirements. Very little displacement, colored in blue, occurs in the back rectangular face, where the fixed boundary condition is applied in Step 5.



9.2. Animate the deflections by selecting the animation icon followed by selecting the play button at the bottom. When you are through viewing the animation, select Close.

9.3. Quit the Results processing by selecting the Quit Icon as shown above and confirming the message shown below.



2.3 CAM for Cutting Tool Path Generation

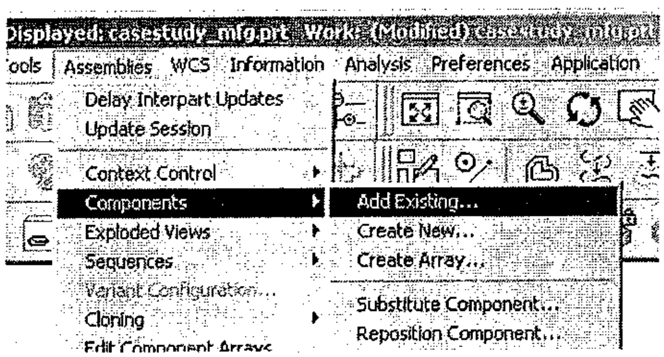
Step 1. Applications → Modeling. Create a new part file and name it as ***_casestudy_block.prt, where *** is your three letter name initials. Create a block and assign the same material type (i.e., stainless steel) to it by following Steps 1 and 5 of Section 2.1. This part serves as a blank part. Save and close this file. You will be guided to create a cutting tool path to cut this blank part into the final part shown in Figure 2.1.

Step 2. Create another new part and name it as ***_casestudy_mfg.prt. This part will contain the information required to generate the tool path.

Step 3. Add the two parts, the blank part (***_casestudy_block.prt) and the final part (***_casestudy.prt), into the manufacturing part (***_casestudy_mfg.prt).

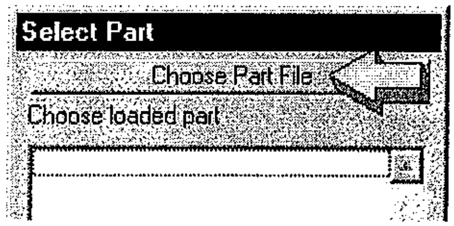
3.1 Application → Assemblies

3.2 Assemblies → Components



3.3 Slide out and click "Add Existing" button.

3.4 "Select Part" dialog appears. If you see ***_casestudy_block in the list of loaded parts, pick it and then click "OK". If not, click "Choose Part File" button and find the part file by navigating the file directories in the same manner as you do in "File → Open".



3.5 Accept the default setting on "Add Existing Part" dialog by clicking "OK".

3.6 "Point Constructor" dialog appears. Click "Reset" button and then click "OK". Now the blank part has been added.

3.7 Replace the current view with view TFR-TRI if necessary by clicking Right Mouse Button (MB3) on the graphics window → Replace View → TFR-TRI

3.8 Repeat 3.4 to 3.6 above except adding ***_casestudy.prt in place of ***_casestudy_block.prt.

3.9 Click "Cancel" to stop adding any more parts and save this file.

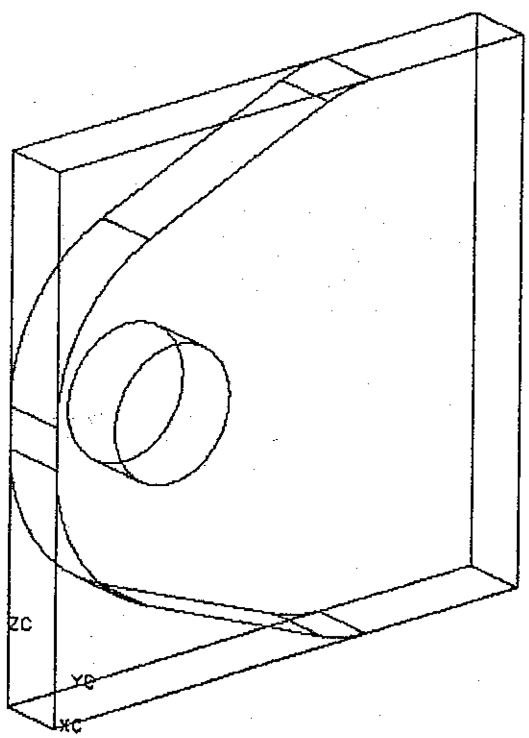
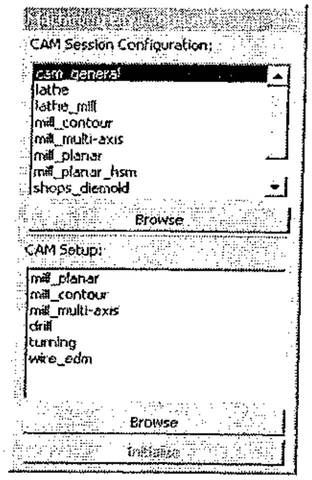


Figure 2.6 ***_casestudy_mfg.prt after adding the two parts in Step 3

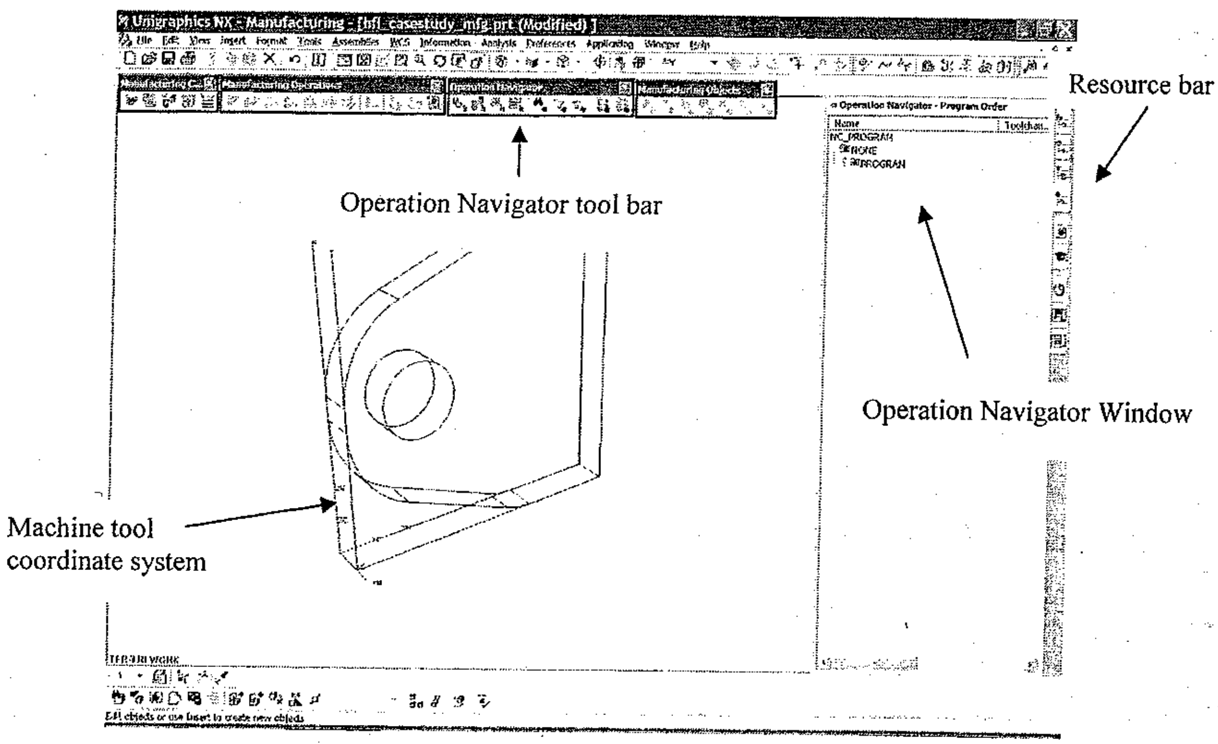
Step 4. Application → Manufacturing to enter the Manufacturing Application. "Machining Environment" dialog appears.



Step 5. Initialize the machining environment.

5.1 Pick "mill_contour" from a list of options available in the second menu "CAM Setup". Click "Initialize" from the bottom of the dialog. The system displays manufacturing-related tool bars including the Operation Navigator tool bar as shown in the below figure.

5.2 Click Operation Navigator icon from the resource bar on the right side of the screen. The Operation Navigator window opens.



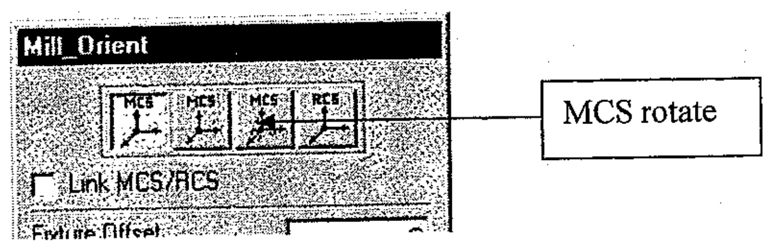
Click the pin button in the right upper corner of the Operation Navigator window to keep this window remaining open.

Step 6. Specify the correct machine tool coordinate system (MCS). The part display window shows that the current ZM direction, which is a cutting tool access direction to the part, points upward. It needs to rotate by 90 degrees such that it points right.

6.1 Select Tools → Operation Navigator → View → Geometry View or click the Geometry View icon located on the Operation Navigator toolbar, which changes window "Operation Navigator" into Geometry view.

6.2 In window "Operation Navigator – Geometry", double click "MCS_MILL" to edit MCS.

6.3 "Mill_Orient" dialog appears. Click the "MCS rotate" icon, which is the third one on the top as shown below.

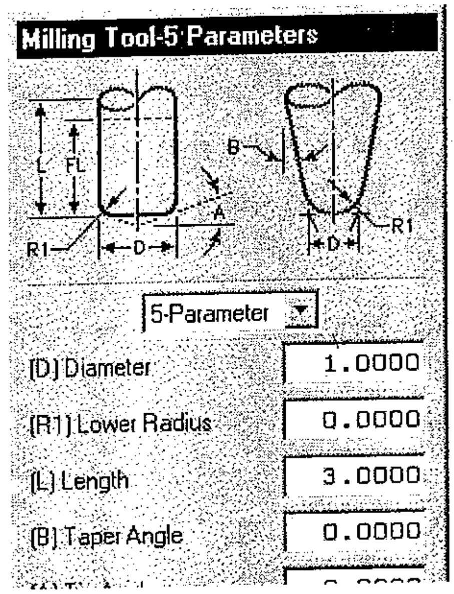


6.4 "Rotate MCS about" dialog appears. Click "+YM axis; ZM -> XM" and click "Apply" (accepting default angle 90°). Now the ZM direction points right. Click "Cancel", and then "OK" to close the "Mill_Orient" dialog.

Step 7. Specify a 0.5-inch end mill tool for the cutting tool.

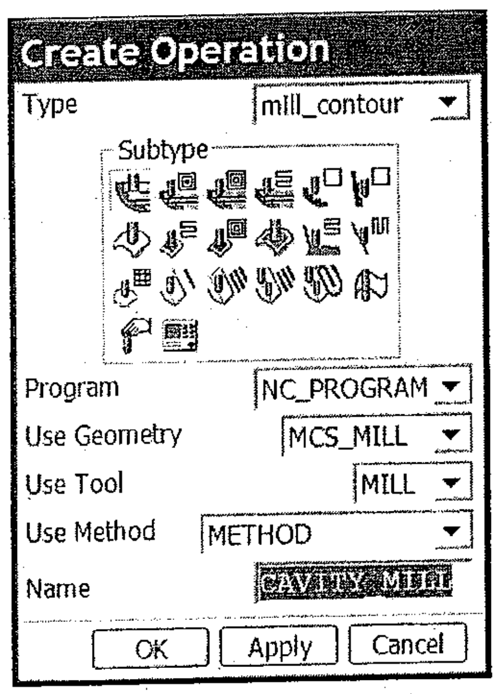
7.1 Select Insert → Tool.

7.2 Dialog "Create Tool" appears. The tool icon MILL is highlighted (a default tool) and click OK. Dialog Milling Tool-5 Parameters appears.



7.3 Enter (D) Diameter = .5 to cut the 1" diameter hole, and click "OK" by accepting the default values for other parameters.

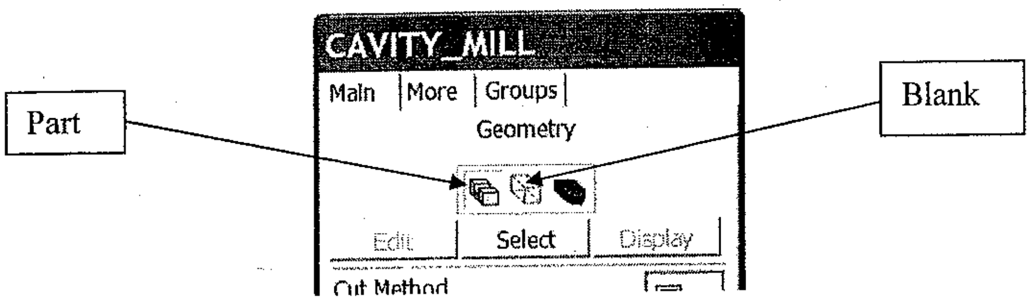
7.4 Select Insert → Operation. Dialog "Create Operation" appears.



Step 8. Setup the Cavity Mill operation.

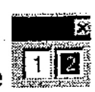
8.1 The default operation selected among a set of operations listed under "Subtype" is CAVITY_MILL (the upper left corner icon highlighted).

8.2 Choose OK. Cavity_Mill dialog appears.



Next, you will assign which part among the two in the graphics window is the Blank and the final Part, respectively.

8.3 Click "Part" icon  under title "Geometry". Click "Select" button on the next line.

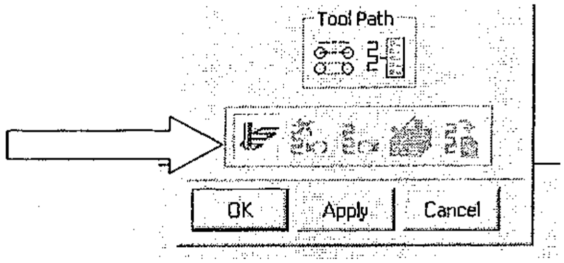
8.4 "Part Geometry" dialog appears. Select the final part in the graphics window. If more than one is selectable, you will see the selection dialogue looking like  appear on the graphics window. Click the correct number between the two to pick the final part, but not the blank part. Click "OK" to close the Part Geometry dialog.

8.5 Click "Blank" icon right next to "Part" icon as shown above. Click "Select" button.

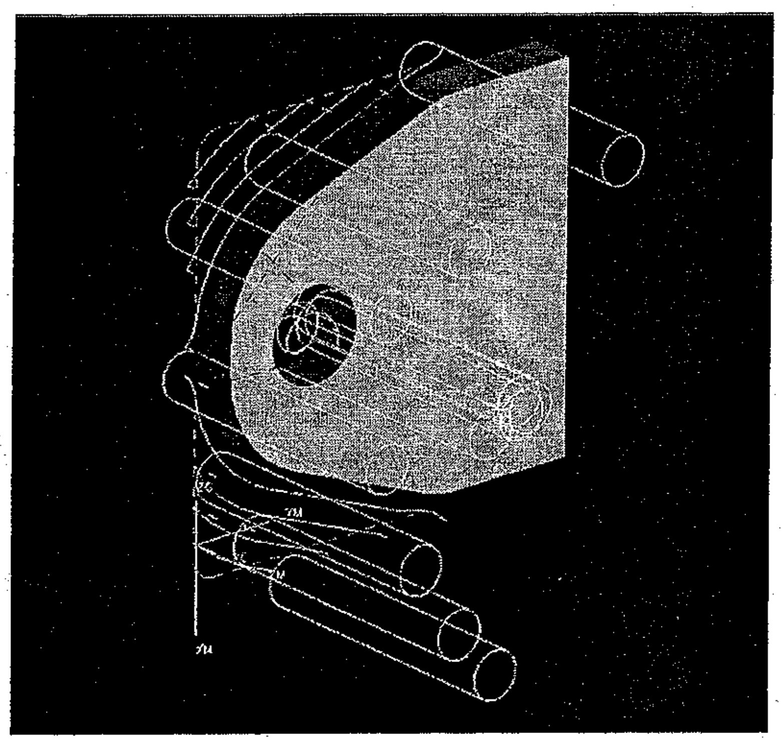
8.6 Select the blank part in the graphics window. Click "OK" to close the Part Geometry dialog.

Step 9. Generate the tool path.

9.1 On the Cavity_Mill dialog, click the "Generate" icon near the bottom of the dialog.



9.2 Dialog "Display Parameters" appears. Click "OK". The tool path is shown on the part. Continue to click "OK" until it finishes generating the tool path.



9.3 Animate the tool path for verification. Click "Verify" icon right to the "Generate" icon. Dialog "Toolpath Visualization" appears. Click tab "Dynamic" located on the top of the dialog. Click the play foward button at the bottom and observe animation of the cutting process. Click "OK" to take it back to the Cavity_Mill dialog.

9.4 Click "List" icon  right next to "Verify" icon. This lists the generic NC program list called the cutter location source file (CLSF). This can be post-processed to a specific NC machine tool. Close the list window.

A top portion of CLSF

```
TOOL PATH/CAVITY_MILL,TOOL,MILL
TLDATA/MILL,0.5000,0.0000,3.0000,0.0000,0.0000
MSYS/0.0000,0.0000,0.0000,0.0000000,0.0000000,1.0000000,0.0000000,1.000
0000,0.0000000
PAINT/PATH
PAINT/SPEED,10
PAINT/COLOR,4
RAPID
GOTO/0.3400,0.6552,0.6000,0.0000000,0.0000000,1.0000000
PAINT/COLOR,1
RAPID
GOTO/0.3400,0.6552,0.3500
PAINT/COLOR,6
FEDRAT/IPM,10.0000
GOTO/0.3400,0.6552,0.2500
GOTO/0.2400,0.4075,0.2500
PAINT/COLOR,3
CIRCLE/-
1.8465,1.2500,0.2500,0.0000000,0.0000000,1.0000000,2.2501,0.0020,0.5000,0.
5000,0.0000
GOTO/-0.1603,-0.2399,0.2500
PAINT/COLOR,7
RAPID
GOTO/-0.1603,-0.2399,0.6000
```

9.5 In dialog "Cavity Mill", click "OK" to accept this tool path. Save and close all the files.

Exercise Problems

2.1. Suppose the block size of Section 2.1 changes from .5 x 4 x 4 to .5 x 4 x 5. Redo all the steps of Sections 2.1, 2.2, and 2.3 for CAD/CAE/CAM.

2.2. In Problem 2.1, you are asked to redo all the steps from scratch in order to address the design change in the block size. Is there an easier way?