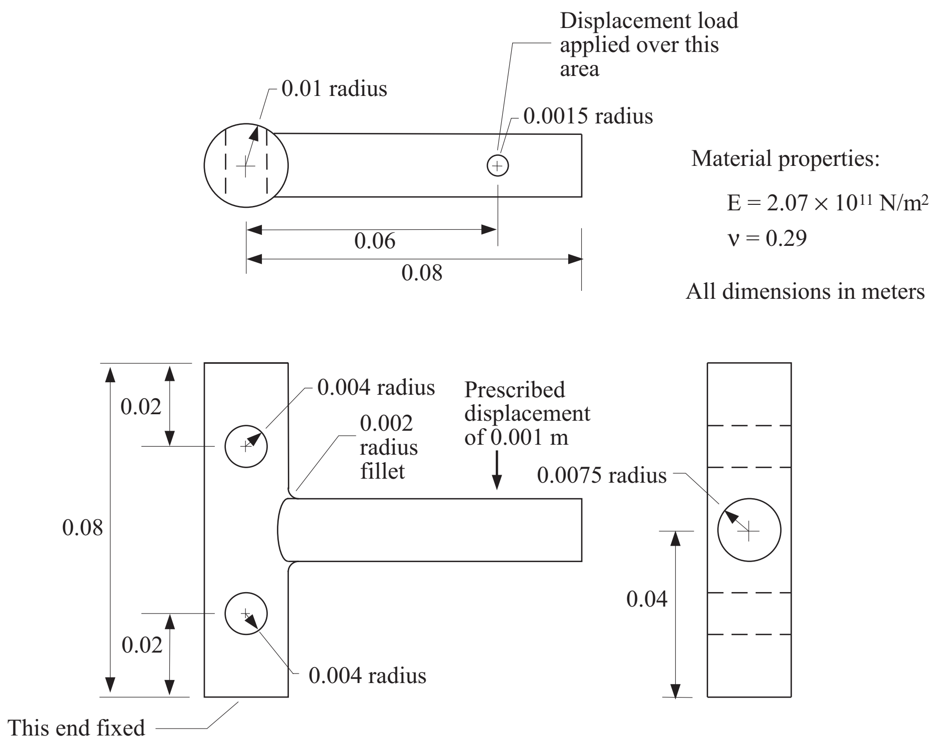
Problem description

It is desired to analyze the shaft-shaft intersection shown using a 3D finite element mesh:



Shaft-shaft intersection dimensions
and loading

The purpose of this analysis is to demonstrate the usage of ADINA-M/PS (the ADINA Modeler based on the Parasolid geometry kernel).

In this problem solution, we will demonstrate the following topics that have not been presented in previous problems:

- Construction of solid geometry using ADINA-M/PS
- Using the geometry discretization controls during meshing
- Definition of a zone using the mouse
- Plotting color shaded images
- Defining an isosurface during post-processing
- Modifying the band table range of a band plot

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI icons. Please refer to the Hints chapter of the Primer for useful hints.

Note that you must have an ADINA-M/PS license to do this problem. In addition you need to allocate at least 40 MB of memory to the AUI.

This problem cannot be solved with the 900 nodes version of the ADINA System because there are too many nodes in the model.

Invoking the AUI and choosing the finite element program

Invoke the AUI and set the Program Module drop-down list to ADINA Structures. Choose Edit→Memory Usage and make sure that the ADINA/AUI memory is at least 40 MB.

Defining model control data

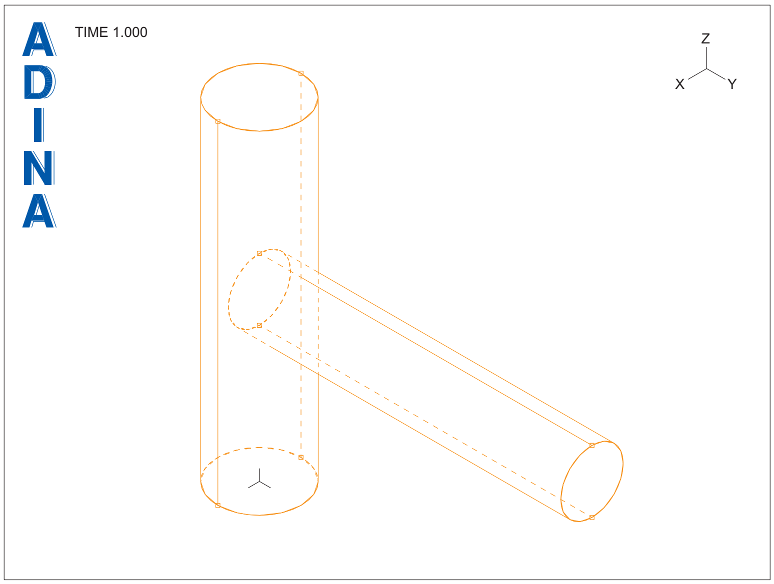
Problem heading: Choose Control→Heading, enter the heading "Problem 16: Analysis of a shaft-shaft intersection with ADINA-M/PS" and click OK.

Defining model geometry

Vertical shaft: Click the Define Bodies icon , add body 1, set the Type to Cylinder, the Radius to 0.01, the Length to 0.08, the Center Position to (0.0, 0.0, 0.04), the Axis to Z and click Save.

Horizontal shaft: Add body 2, set the Type to Cylinder, the Radius to 0.0075, the Length to 0.08, the Center Position to (0.0, 0.04, 0.04), the Axis to Y and click OK.

When you click the Wire Frame icon , the graphics window should look something like the figure on the next page.

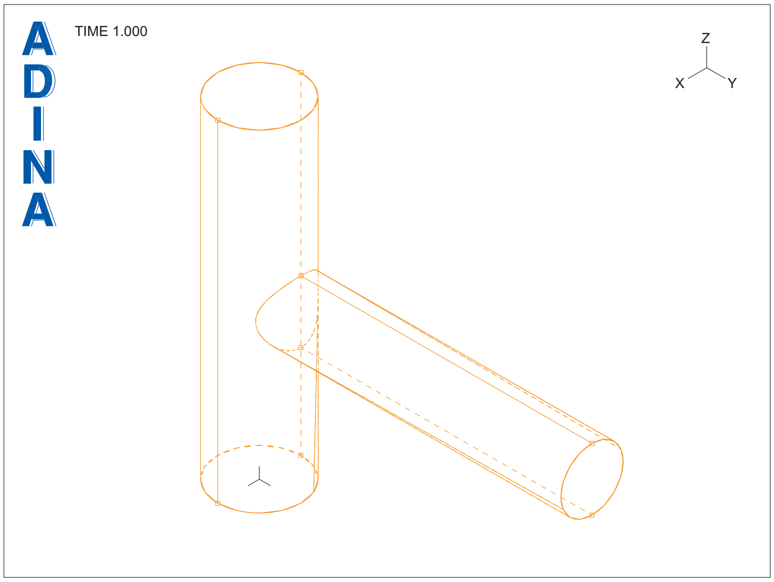


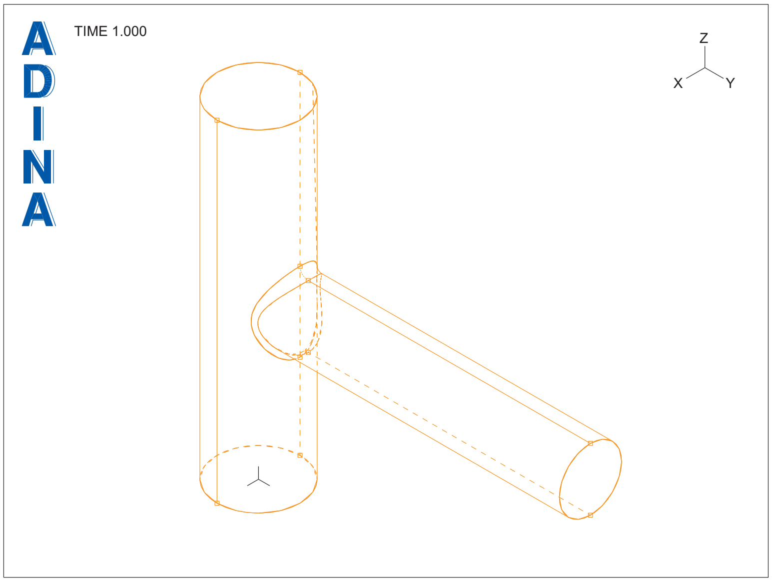
Merging the shafts: Click the Boolean Operator icon, make sure that the Operator Type is Merge and that the Target Body is 1, enter 2 in the first row of the table and click OK.

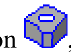
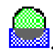
The graphics window should look something like the top figure on the next page.

Fillet: Click the Body Modifier icon, make sure that the Modifier Type is Blend, make sure that the Target Body is 1 and set the First Radius to 0.002. Then enter edges 7 and 8 in the first two rows of the table and click OK.

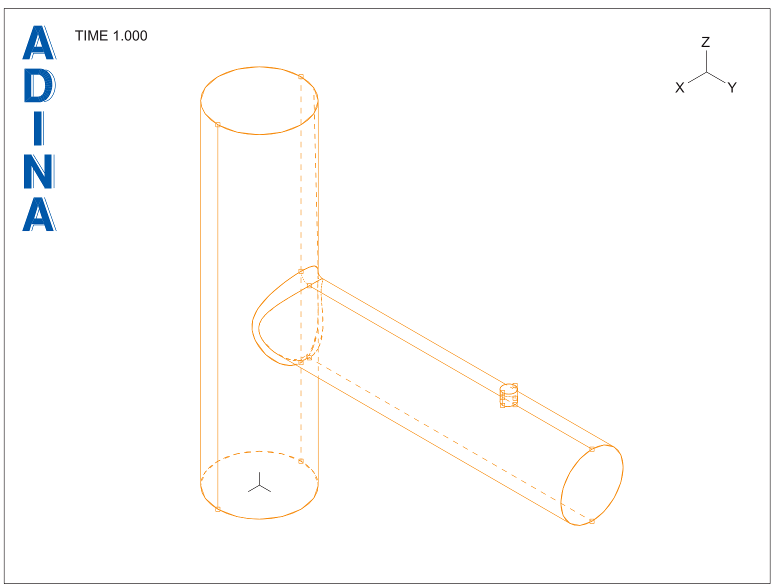
The graphics window should look something like the bottom figure on the next page.

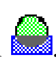

ADINA
TIME 1.000
Z
X
Y


ADINA
TIME 1.000
Z
X
Y

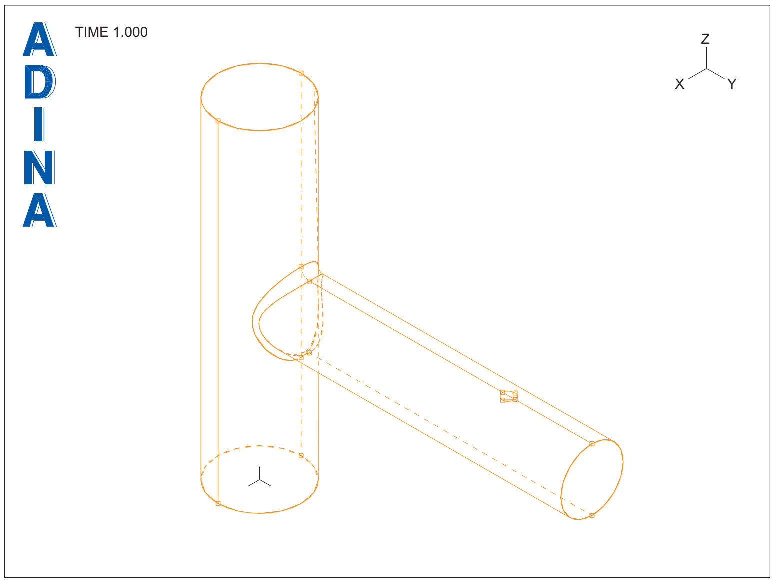
Area on which displacement is applied: To define the area on which displacements are applied, we create a cylinder and use the cylinder to create an imprinted face on the shaft body. Click the Define Bodies icon , add body 2, set the Type to Cylinder, the Radius to 0.0015, the Length to 0.01, the Center Position to (0.0, 0.06, 0.045), the Axis to Z and click OK. Now click the Boolean Operator icon , set the Operator Type to Subtract, set the Target Body to 2, check the "Keep the Subtracting Bodies" button, enter 1 in the first row of the table and click OK.

The graphics window should look something like this:



Now we make the imprint. Click the Boolean Operator icon , set the Operator Type to Subtract, make sure that the Target Body is 1, check the "Keep the Imprinted Edges Created by the Subtraction" button, enter 2 in the first row of the table and click OK.

The graphics window should look something like the figure on the next page.

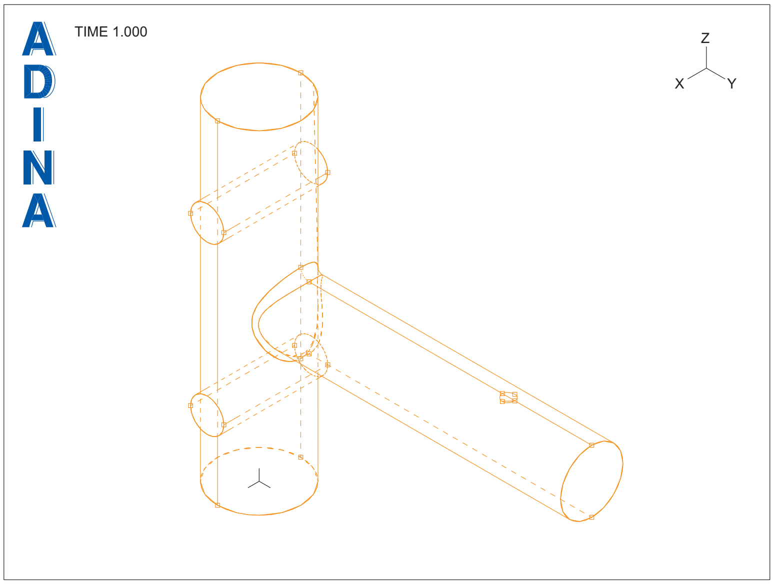


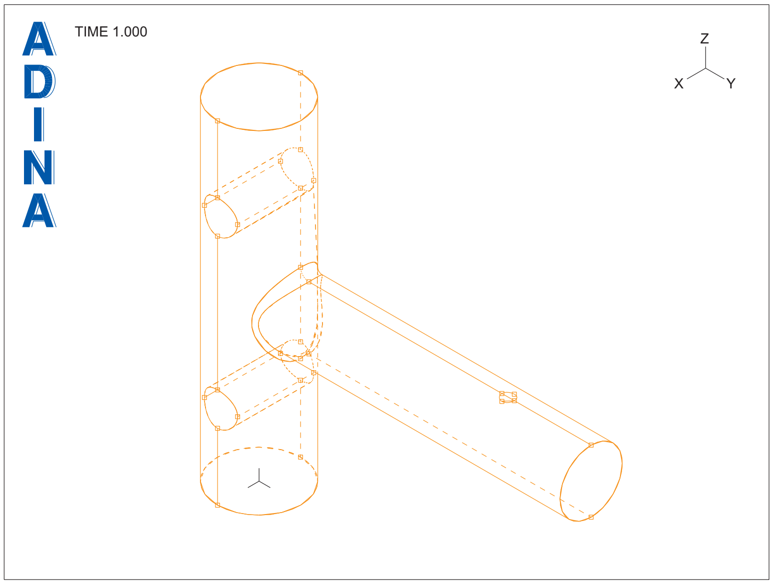
Holes: Click the Define Bodies icon, add body 2, set the Type to Cylinder, the Radius to 0.004, the Length to 0.025, the Center Position to (0.0, 0.0, 0.02), make sure that the Axis is X and click Save. Now add body 3, make sure that the Type is Cylinder, set the Radius to 0.004, the Length to 0.025, the Center Position to (0.0, 0.0, 0.06), make sure that the Axis is X and click OK.

The graphics window should look something like the top figure on the next page.

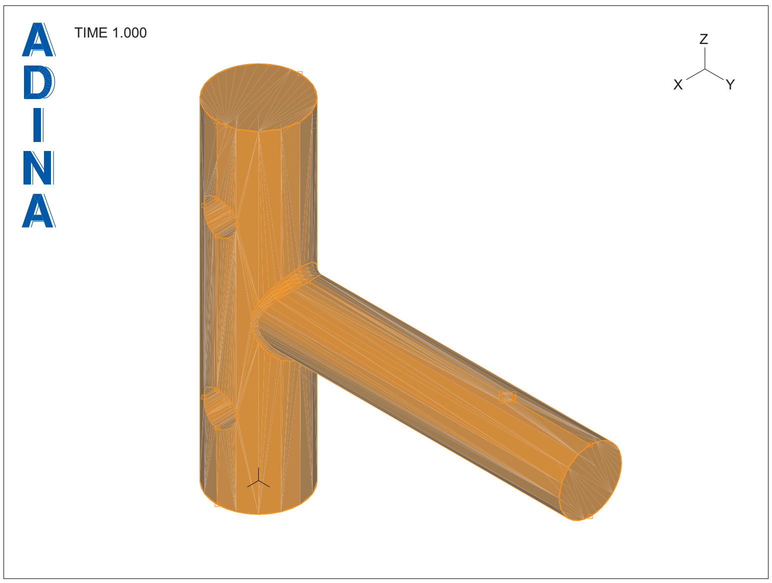
Now click the Boolean Operator icon, set the Operator Type to Subtract, make sure that the Target Body is 1, enter 2, 3 in the first two rows of the table and click OK.

The graphics window should look something like the bottom figure on the next page.


ADINA
TIME 1.000
Z
X
Y


ADINA
TIME 1.000
Z
X
Y

This completes the geometry definition. To make a color-shaded image of the geometry, click the Shading icon . The graphics window should look something like this:



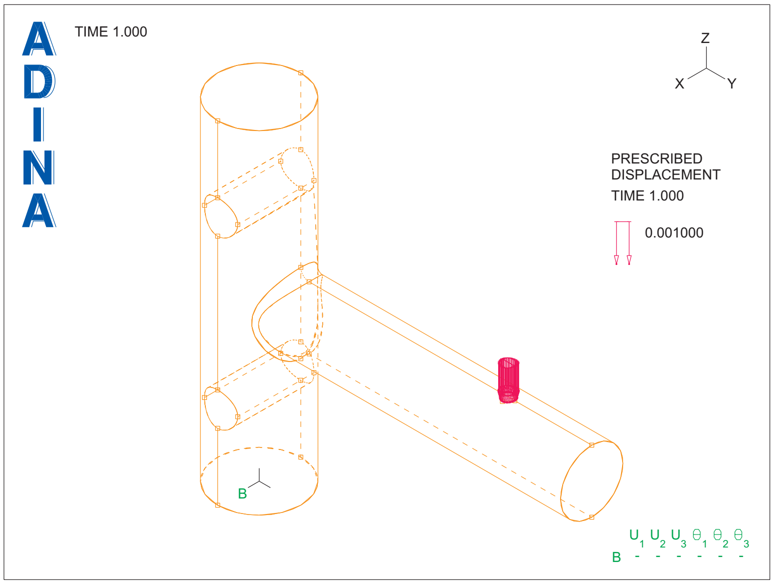
Click the Wire Frame icon to display the geometry with dashed hidden lines.

Specifying boundary conditions, loads and the material

Fixities: Click the Apply Fixity icon , set the "Apply to" field to Face/Surface, enter 5, 1 in the first row of the table and click OK.

Loads: Click the Apply Load icon , set the Load Type to Displacement and click the Define... button to the right of the Load Number field. In the Define Displacement dialog box, add displacement 1, set the Z Translation to -0.001 and click OK. In the Apply Load dialog box, set the "Apply To" field to Face, then, in the first row of the table, set the Face # to 1 and the Body # to 1, and, in the second row of the table, set the Face # to 2 and the Body # to 1. Click OK to close the Apply Load dialog box.

When you click the Boundary Plot icon and the Load Plot icon , the graphics window should look something like the figure on the next page.



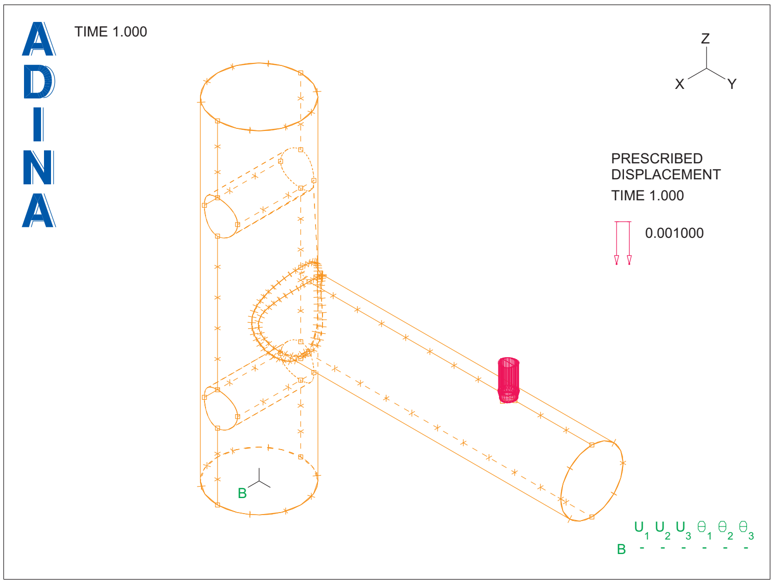
Material: Click the Manage Materials icon and click the Elastic Isotropic button. In the Define Isotropic Linear Elastic Material dialog box, add material 1, set the Young's Modulus to 2.07E11, the Poisson's ratio to 0.29 and click OK. Click Close to close the Manage Material Definitions dialog box.

Meshing

Subdivision data: We will specify a uniform element size throughout most of the ADINA-M geometry and a finer mesh near the fillet. The fillet element size will be controlled by the element size on geometry faces 10 and 11.

Choose Meshing→Mesh Density→Complete Model, set the “Subdivision Mode” to “Use Length”, set the Element Edge Length to 0.006 and click OK. Now click the Subdivide Faces icon, select face 10, set the Element Edge Length to 0.0012, enter 11 in the first row of the table and click OK.

The graphics window should look something like the figure on the next page.

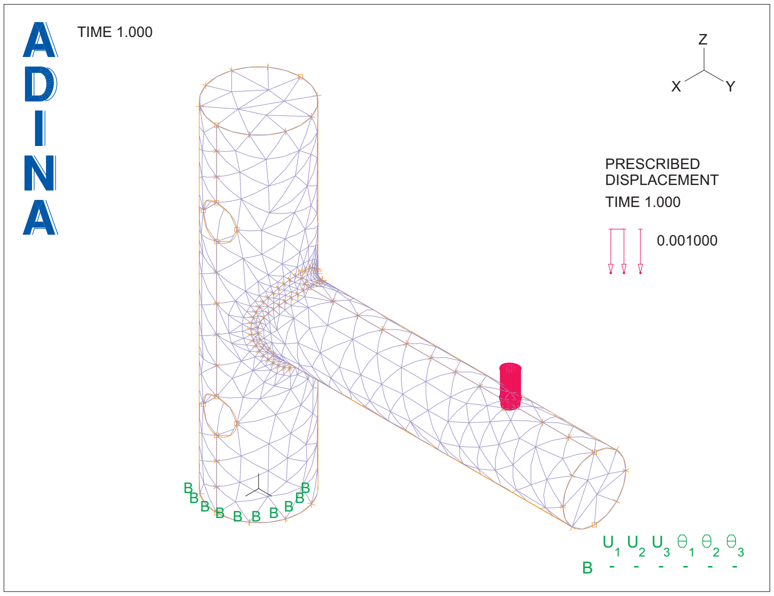


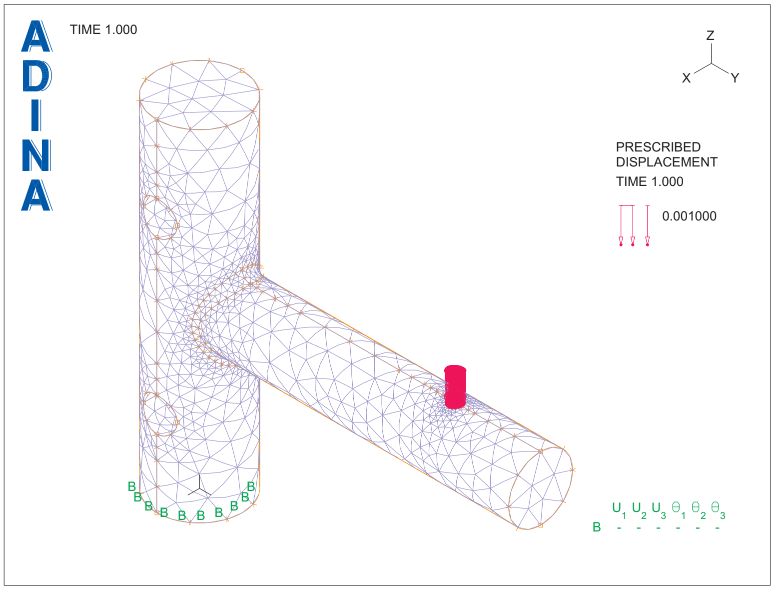
Meshing: First, click the Hidden Surfaces Removed icon (we do not want to see dashed hidden lines in the elements that we generate). Now click the Mesh Bodies icon , click the + button to the right of the Element Group text, set the Body # to 1 in the first row of the table, and click OK. The graphics window should look something like the top figure on the next page.

This mesh is acceptable, but we would prefer a mesh with a smoother transition between the fillet and the rest of the mesh.

Deleting the mesh and remeshing: Click the Delete Mesh/Elements icon , set the "Delete Elements" field to "On Bodies" if necessary, enter 1 in the first row of the table and click OK. Now choose Meshing→Mesh Density→Complete Model, set the "Subdivision Mode" to "Use Length", set the Element Edge Length to 0.006 and click OK.

This time, when we remesh, we use the geometry discretization controls. Click the Mesh Bodies icon , click the Tetrahedral tab, set the "Geometry Discretization Error" to 0.08, and the "Minimum Size of Element Allowed" to 0.0001. Now click the Advanced tab, set the "Boundary Meth" to Delaunay, the Body # to 1 in the first row of the table, and click OK. The graphics window should look something like the bottom figure on the next page.





Notice that all curved boundaries are refined. The fillet has smaller elements because its curvature is greater.

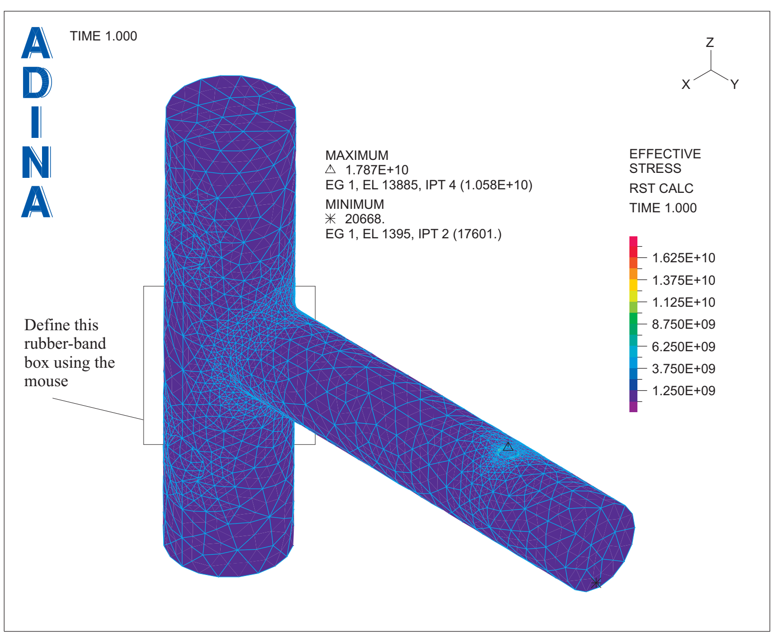
Generating the data file, running ADINA Structures, loading the porthole file

Click the Save icon and save the database to file `prob16`. Click the Data File/Solution icon, set the file name to `prob16`, make sure that the Run Solution button is checked and click Save. When ADINA Structures is finished, close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob16`.

Plotting the effective stresses

Click the Quick Band Plot icon. Note that the largest effective stress occurs at the point of load application. As we are only interested in the stresses near the fillet, we now plot only the region near the fillet. We do this by defining a zone as those elements in a box that contains the fillet.

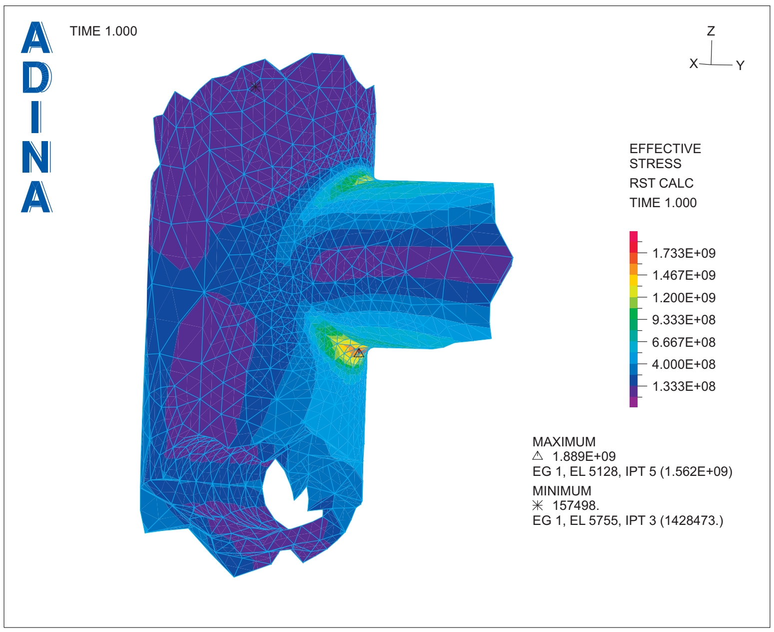
Click the Change Zone icon and click the ... button to the right of the Zone Name field. Add zone BOX and double-click in one of the rows of the table. If the "Change Zone of Mesh Plot" dialog box is covering up the mesh plot, move the dialog box out of the way. Use the mouse to create a rubber-band box that includes the shaft-shaft intersection, as shown:



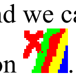
The elements within the rubber-band box become highlighted.

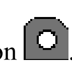
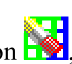
Press the Esc key to return to the Define Zone dialog box. Notice that the table is now filled in. Click OK to close the Define Zone dialog box. In the Change Zone dialog box, set the Zone Name to BOX if necessary and click OK.

Use the mouse to rotate the mesh until the graphics window looks something like this:

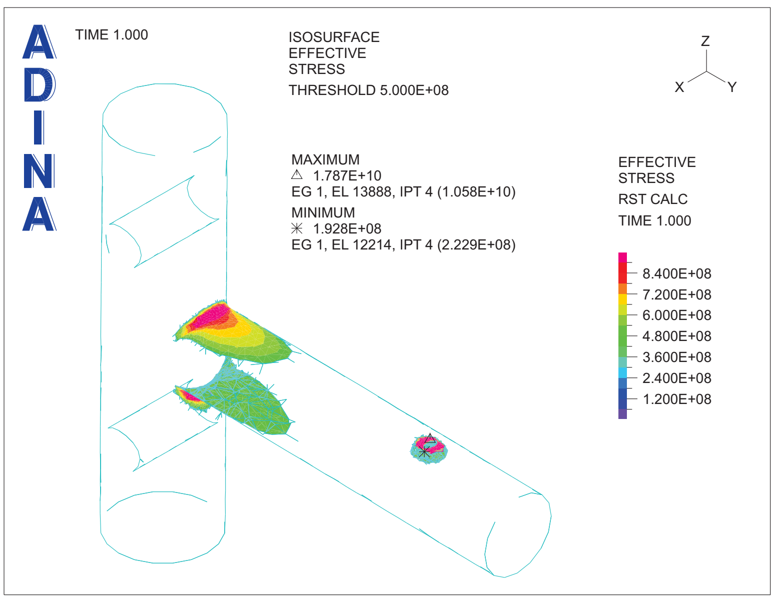


Your results may be slightly different than ours because free-form meshing produces slightly different meshes on different computers.

We are interested in effective stresses above a certain threshold, and we can view these effective stresses with an isosurface. Click the Clear Band Plot icon . In the Model Tree, expand the Zone entry, right click WHOLE_MODEL and choose Display. Click the Cut Surface icon  and set the Type to Isosurface. In the Isosurface Variable field, set the Variable to (Stress: EFFECTIVE_STRESS) and set the Threshold Value to 5E+08. In the Mesh Display box, set "Above the Isosurface" to "Display as Usual". Click OK to exit the Define Cutsurface Depiction dialog box.

Click the Model Outline icon . Click the Modify Mesh Plot icon , click the Rendering... button and set the Element Face Angle to 50. Click OK twice to exit both dialog boxes.

Click the Quick Band Plot icon to plot the effective stresses, and then click the Modify Band Plot icon and click the Band Table... button. In the Value Range field, set the Maximum to 9E+08 and the Minimum to 1E+08. Click OK twice to exit both dialog boxes. The graphics window should look something like this:



Exiting the AUI: Choose File→Exit (you can discard all changes).