

Chapter 8: Aircraft Fuselage Window

Keywords: 2D elasticity, plane stress, symmetry, superposition

Modeling Procedures: ruled surface, convert, biased mesh

8.1 Problem Statement and Objectives

An aircraft fuselage structure must be capable of withstanding many types of loads, and stress concentrations near cutouts are of particular concern. In this exercise, internal pressure in a structure similar to a Lockheed L-1011 commercial aircraft fuselage is considered. The objective of the analysis is to determine the stress state and the factor of safety in a square fuselage panel containing a window cutout. The geometrical, material, and loading specifications for the panel are given in Figure 8.1.

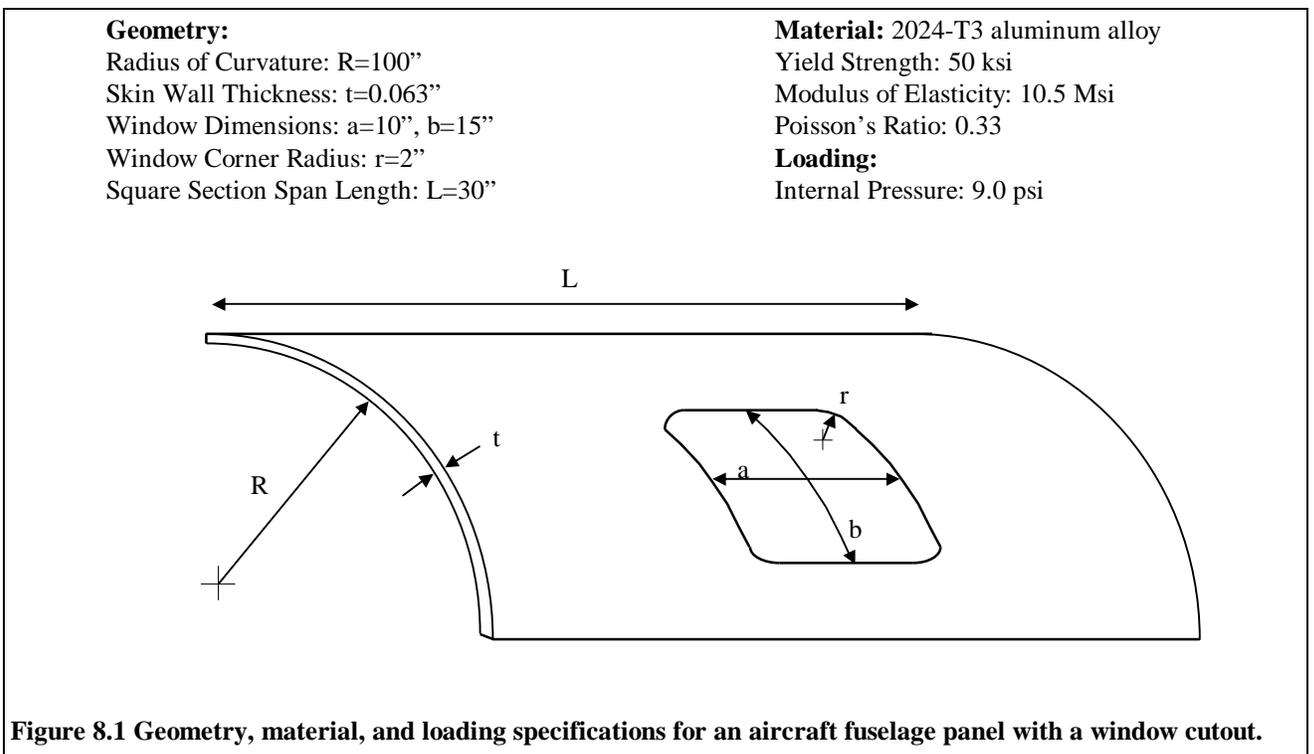


Figure 8.1 Geometry, material, and loading specifications for an aircraft fuselage panel with a window cutout.

8.2 Analysis Assumptions

1. The analysis model will consider only the fuselage structure with an open cutout. In other words, the analysis will be performed as if no window is present (yet pressurization is maintained).
2. The radius-to-thickness ratio (R/t) of the fuselage skin is approximately 1,587. Therefore, the effect of curvature can be ignored in the present analysis.
3. The span-to-thickness ratio (L/t) of the fuselage skin is approximately 476. Therefore, the structure is very thin and a state of plane stress can be assumed.
4. The panel edges are a sufficient distance from fuselage stiffeners, so very little bending will occur due to the applied internal pressure load. The effects of bending will thus be neglected.

8.3 Mathematical Idealization

Based on the assumptions above, a 2D plane stress model of the structure will be developed. The loads can be obtained from cylindrical pressure vessel theory. In other words, the internal pressure will cause an axial and a circumferential stress in the cylinder, which can be applied at the edges of the structural section being modeled. The idealized model is the same as in Figure 8.1, except that the curvature is ignored.

8.4 Finite Element Model

The finite element model of this structure will be developed using 2D plane stress four-noded quadrilateral finite elements. The present analysis can be greatly simplified by taking advantage of existing symmetries in the idealized model. Note that both horizontal and vertical mirror planes of symmetry exist, as shown in Figure 8.2a. The geometry, material properties, and loading conditions are all symmetric about these two planes. Therefore, the response of the structure (i.e., displacements, strains, and stresses) will also be symmetric about these two planes. Hence, it is necessary to model only a quarter of the panel, as shown in Figure 8.2b. The boundary conditions on the symmetry planes are those that occur naturally on these planes, as can be verified by obtaining a solution using the entire panel domain. Taking advantage of symmetry reduces the modeling effort, the amount of computer memory, and the amount of CPU time required to obtain a solution.

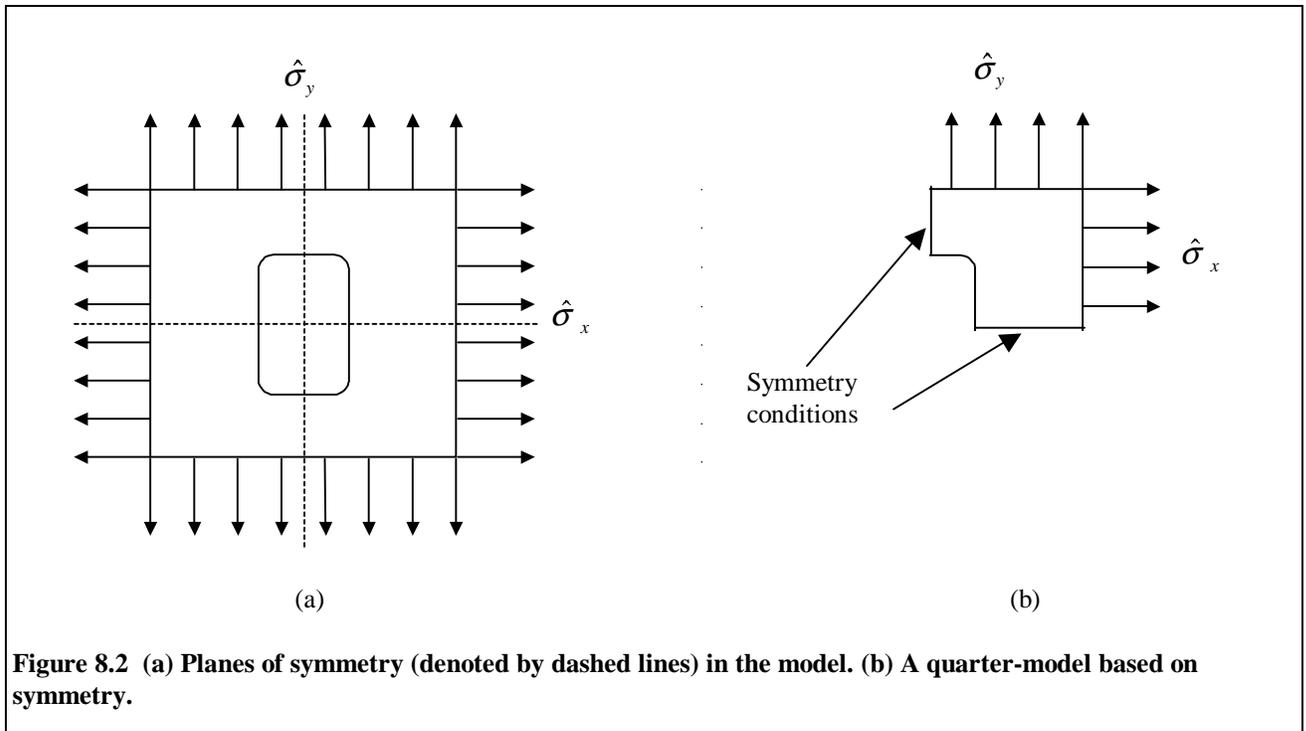


Figure 8.2 (a) Planes of symmetry (denoted by dashed lines) in the model. (b) A quarter-model based on symmetry.

8.5 Model Validation

Some simple hand calculations can be performed to estimate the stresses in the structure. An exact solution for the current problem is not known. However, solutions for similar problems exist, and these can be used as a guide to validate the model.

The Kirsch solution for an infinite plate with a hole is well known. For the geometry and loading shown in Figure 8.3, the circumferential stress at point A (which is the same as σ_x at this point) is three times the stress applied at infinity (or $3\sigma_0$), and the circumferential stress at point B (which is the same as σ_y at this point) is the negative of the stress applied at infinity (or $-\sigma_0$).

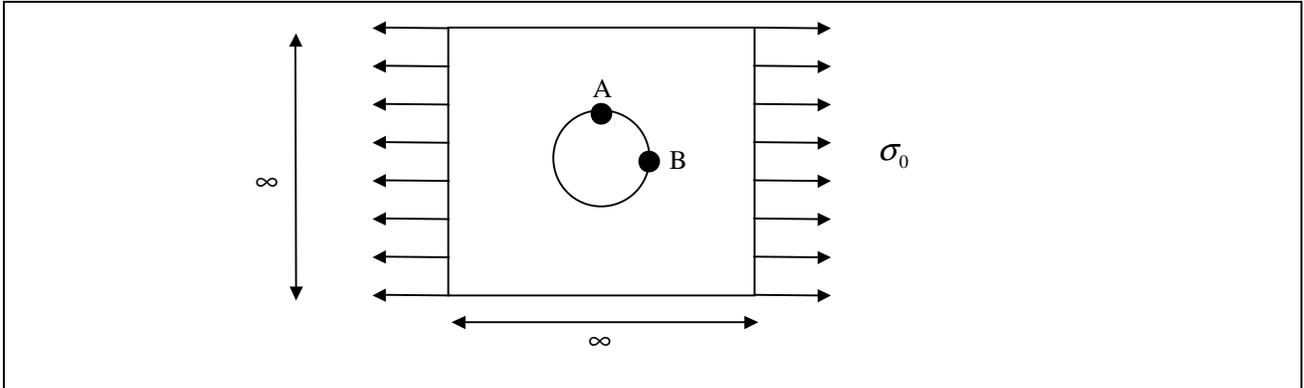


Figure 8.3 Geometry and loading for the Kirsch problem of an infinite plate with a hole.

For an elliptical hole as shown in Figure 8.4, the stresses at points A and B are:

$$\sigma_x = \left(1 + 2 \frac{a_1}{a_2}\right) \sigma_0 \quad \text{at point A}$$

$$\sigma_y = -\sigma_0 \quad \text{at point B.}$$

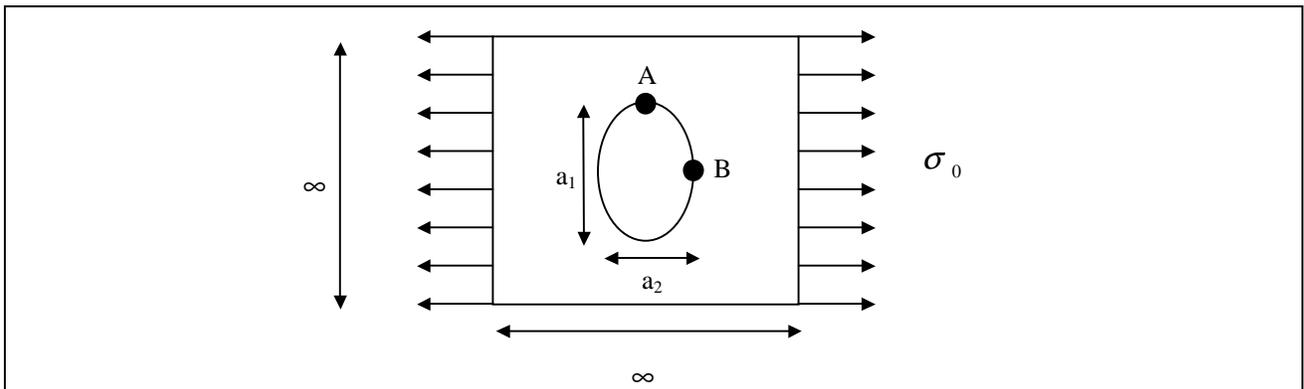


Figure 8.4 Infinite plate with an elliptical cutout.

The fuselage window cutout geometry is actually a rectangle with large radii, though it resembles an ellipse. Note also that the rounded corners of the rectangular cutout may exhibit a response similar to that of a circular cutout. It is expected, then, that the solutions for loading of a plate with a circular or elliptical cutout can be helpful to validate the present finite element solution.

The fuselage panel section under consideration is not infinite. So boundary effects may alter the stresses in comparison to those in the above analytical solutions. These geometrical and boundary effects will alter the state of stress, but not greatly

Application of the solution for an elliptical or circular cutout to the present problem requires use of the *principle of superposition*, as demonstrated in Figure 8.5. In order to validate the finite element analysis, obtain estimates of:

$$\sigma_x \approx \text{-----} \quad \text{at point A}$$

$$\sigma_y \approx \text{-----} \quad \text{at point B}$$

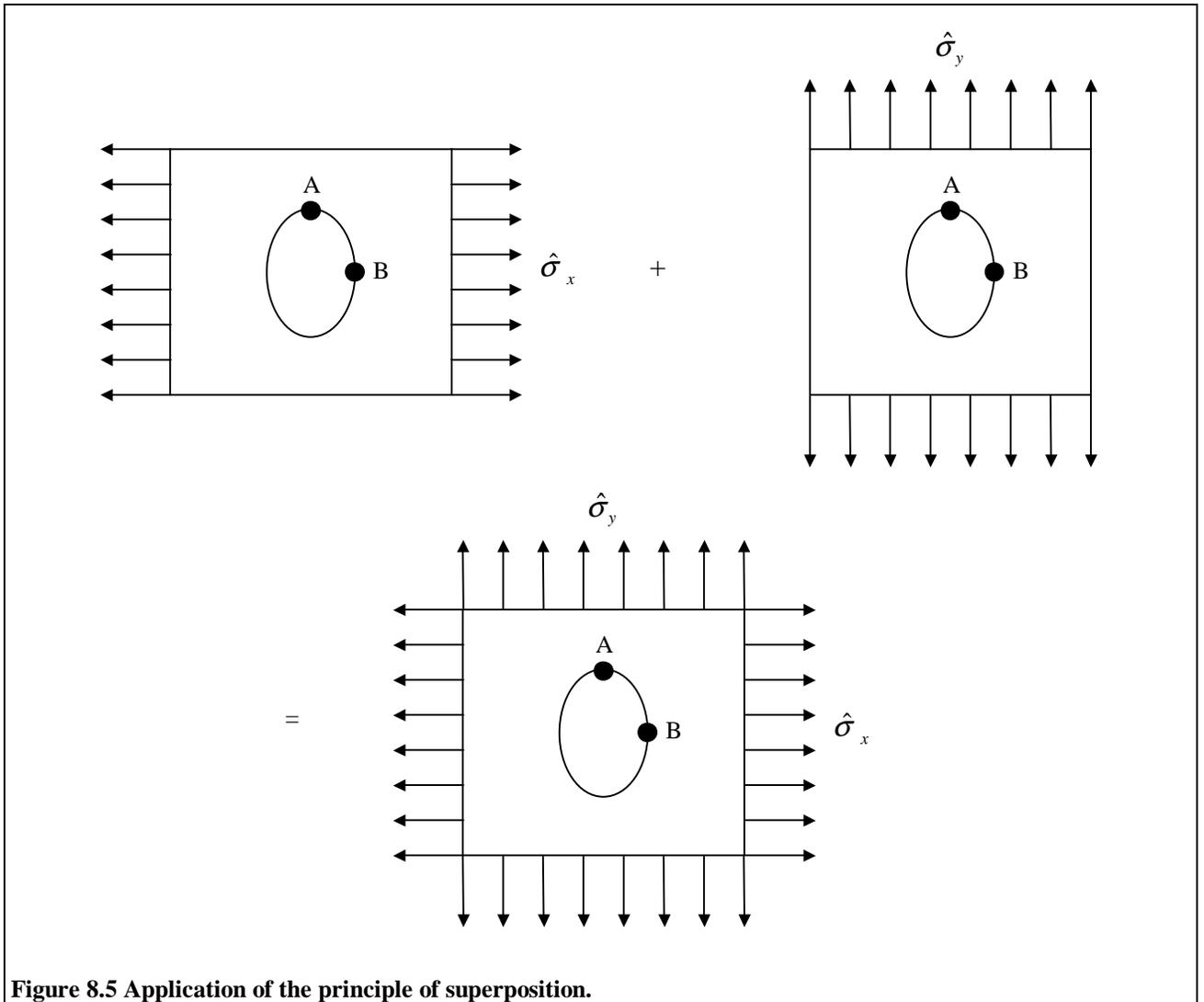


Figure 8.5 Application of the principle of superposition.

FUSELAGE PANEL WITH A CUTOUT

1. Add points to define geometry.

1a. Add points.

```
<ML> MAIN MENU / MESH GENERATION
<ML> MAIN MENU / MESH GENERATION / PTS ADD
```

Enter the coordinates at the command line, one point per line with a space separating each coordinate.

```
> 5.0 0.0 0.0
> 5.0 5.5 0.0
> 0.0 7.5 0.0
> 3.0 7.5 0.0
> 3.0 5.5 0.0
> 15.0 0.0 0.0
> 15.0 8.0 0.0
> 15.0 15.0 0.0
> 8.0 15.0 0.0
> 0.0 15.0 0.0
```

The points may not appear in the Graphics window because Mentat does not yet know the size of the model being built. When the FILL command in the static menu is executed, Mentat calculates a bounding box for the model and fits the model inside the Graphics window.

```
<ML> STATIC MENU / FILL
```

The points should now be visible in the Graphics window.

1b. Display point labels.

```
<ML> STATIC MENU / PLOT
<ML> STATIC MENU / PLOT / LABEL POINTS
<ML> STATIC MENU / PLOT / REDRAW
```

1c. Return to MESH GENERATION menu.

<MR> or <ML> RETURN

The result of this step is shown in Figure 8.1

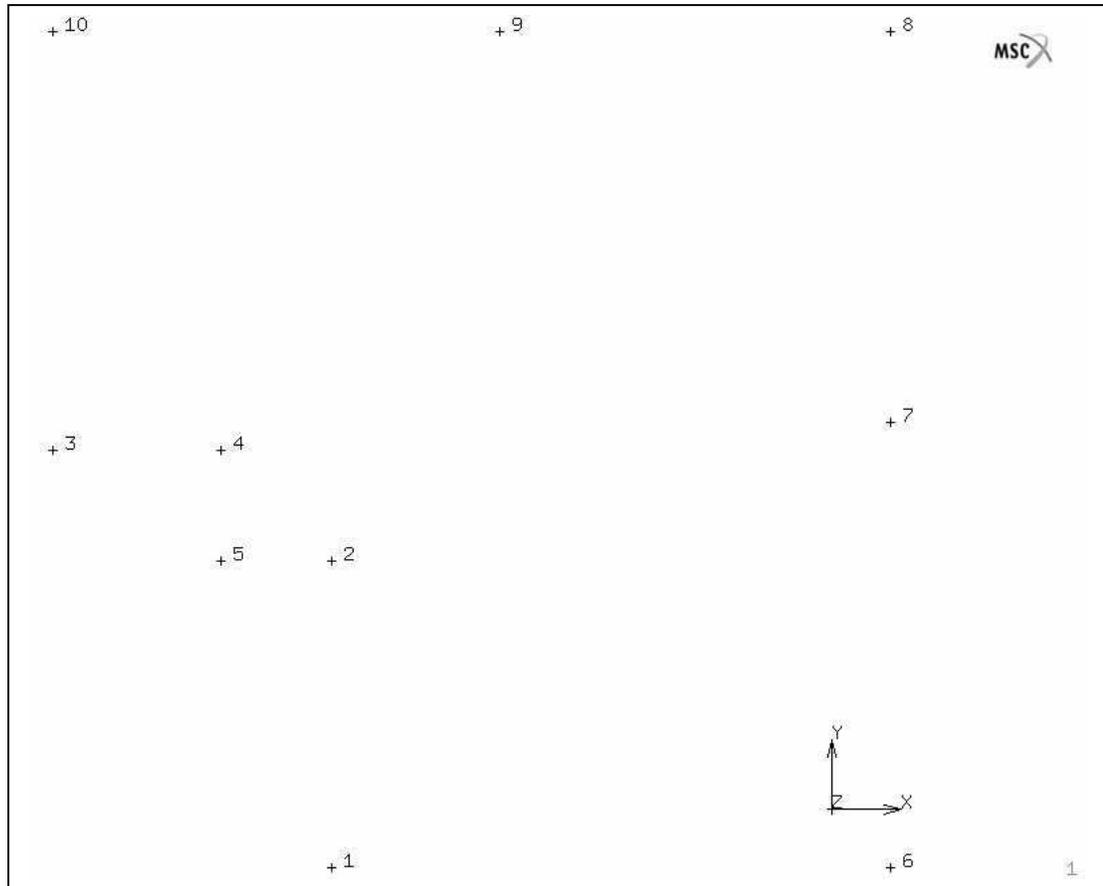


Figure 8.1

If the steps above were not followed precisely (e.g., if the points were entered in an order different than the order in which they appear in the above list), then the point labels will differ from those shown in Figure 8.1. These labels are simply used as identifiers in the following step, and do not affect the model. As long as the correct coordinates were

entered, do not worry if the labels are not exactly as shown in Figure 8.?. Just keep track of the differences between the labels so that the appropriate procedures will be followed in the steps below.

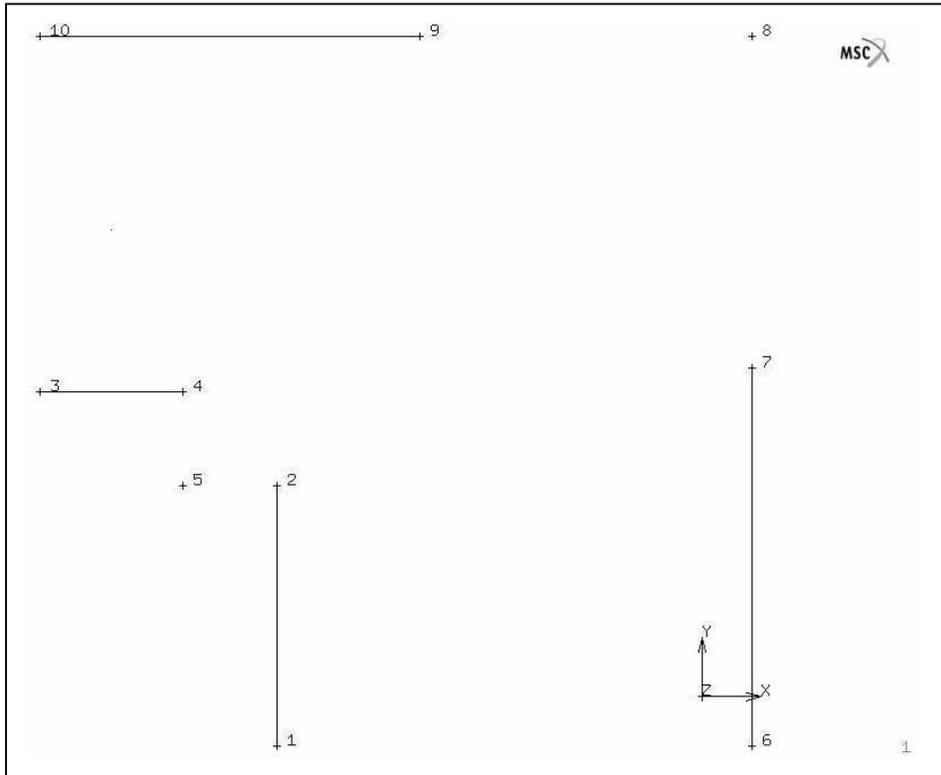
2. Add lines that will be used to generate the three ruled surfaces.
 - 2a. Select CURVE TYPE.

In the MESH GENERATION menu, the currently selected type of curve that can be generated is displayed to the immediate right of the CURVE TYPE button. Confirm that the curve type is: LINE. If true, then proceed to step 2b. If the curve type is not LINE (or if you are not sure what is the selected curve type), then change the curve type as follows:

```
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE / LINE
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE / RETURN
```

- 2b. Add four lines as shown in Figure 8.2 below


```
<ML> MAIN MENU / MESH GENERATION / CRVS ADD
```



```
<ML> to select point 1 and then point 2 to create a line from
point 1 to point 2.
<ML> to select point 6 and then point 7 to create a line from
point 6 to point 7.
<ML> to select point 3 and then point 4 to create a line from
point 3 to point 4.
<ML> to select point 10 and then point 9 to create a line from
point 10 to point 9.
```

Note: To ensure that the ruled surfaces can be properly created in step 3, the two vertical lines and the two horizontal lines should be created in a consistent manner. For example, in the case of the two vertical lines, if each line represents a vector whose direction is defined from the first point to the second point, then the two vectors should have the same direction. This is accomplished by selecting the line end points in the correct order, as indicated in the procedure above.

2c. Change the CURVE TYPE to POLYLINE:

```
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE / POLYLINE
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE / RETURN
```

2d. Add a polyline to create the upper right corner of the panel.

```
<ML> MAIN MENU / MESH GENERATION / CRVS ADD

<ML> to select point 7.
<ML> to select point 8.
<ML> to select point 9.
<MR> or END LIST
```

2e. Change the CURVE TYPE to ARC_CPP (which stands for ARC_CENTER/POINT/POINT):

```
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE / ARCS
CENTER/POINT/POINT
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE / RETURN
```

- 2f. Add an arc to create the upper right corner radius of the cutout.

```
<ML> MAIN MENU / MESH GENERATION / CRVS ADD
```

```
<ML> to select point 5, the center point of the arc (or enter  
the coordinates (3.0, 5.5, 0.0) at the command line).
```

```
<ML> to select point 2, the starting point of the arc (or  
enter the coordinates (5.0, 5.5, 0.0) at the command  
line).
```

```
<ML> to select point 4, the ending point of the arc (or enter  
the coordinates (3.0, 7.5, 0.0) at the command line).
```

Note that some additional points are added to the drawing. These are construction points for the arc.

- 2g. Turn off point labels.

```
<ML> STATIC MENU / PLOT
```

```
<ML> STATIC MENU / PLOT / LABEL POINTS
```

```
<ML> STATIC MENU / PLOT / REDRAW
```

- 2h. Turn on curve labels.

```
<ML> STATIC MENU / PLOT / LABEL CURVES
```

```
<ML> STATIC MENU / PLOT / REDRAW
```

- 2i. Return to MESH GENERATION menu.

```
<MR> or <ML> RETURN
```

The result of this step is shown in Figure 8.3

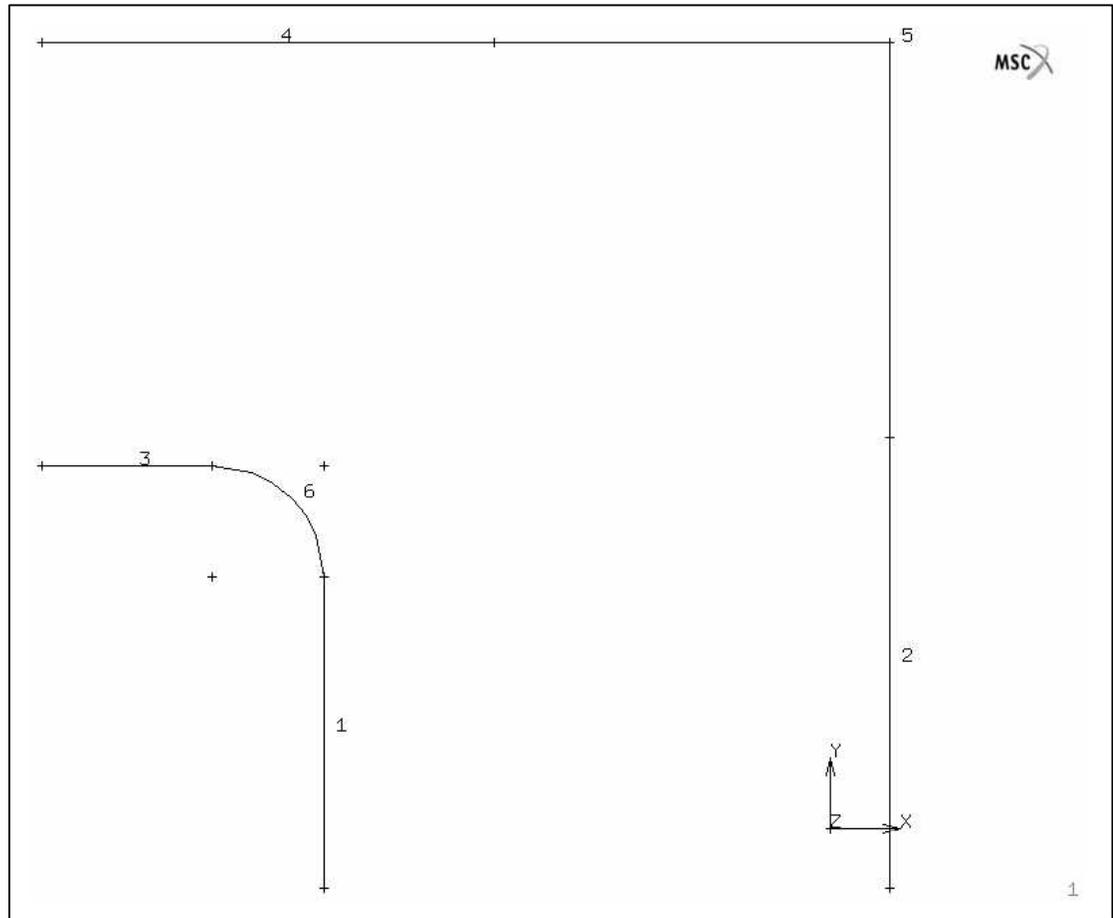


Figure 8.3

3. Create three ruled surfaces.

3a. Change the SURFACE TYPE to RULED:

```
<ML> MAIN MENU / MESH GENERATION / SURFACE TYPE  
<ML> MAIN MENU / MESH GENERATION / SURFACE TYPE / RULED  
<ML> MAIN MENU / MESH GENERATION / SURFACE TYPE / RETURN
```

3b. Create the ruled surfaces.

```
<ML> MAIN MENU / MESH GENERATION / SRFS ADD
```

```
<ML> to select curve 1 and then curve 2 to create a ruled  
surface from curve 1 to curve 2.
```

```
<ML> to select curve 3 and then curve 4 to create a ruled  
surface from curve 3 to curve 4.
```

```
<ML> to select curve 6 and then curve 5 to create a ruled  
surface from curve 6 to curve 5.
```

3c. Turn off curve labels.

```
<ML> STATIC MENU / PLOT  
<ML> STATIC MENU / PLOT / LABEL CURVES  
<ML> STATIC MENU / PLOT / REDRAW
```

3d. Turn on surface labels.

```
<ML> STATIC MENU / PLOT / LABEL SURFACES  
<ML> STATIC MENU / PLOT / REDRAW
```

3e. Return to MESH GENERATION menu.

```
<MR> or <ML> RETURN
```

The result of this step is shown in Figure 8.4

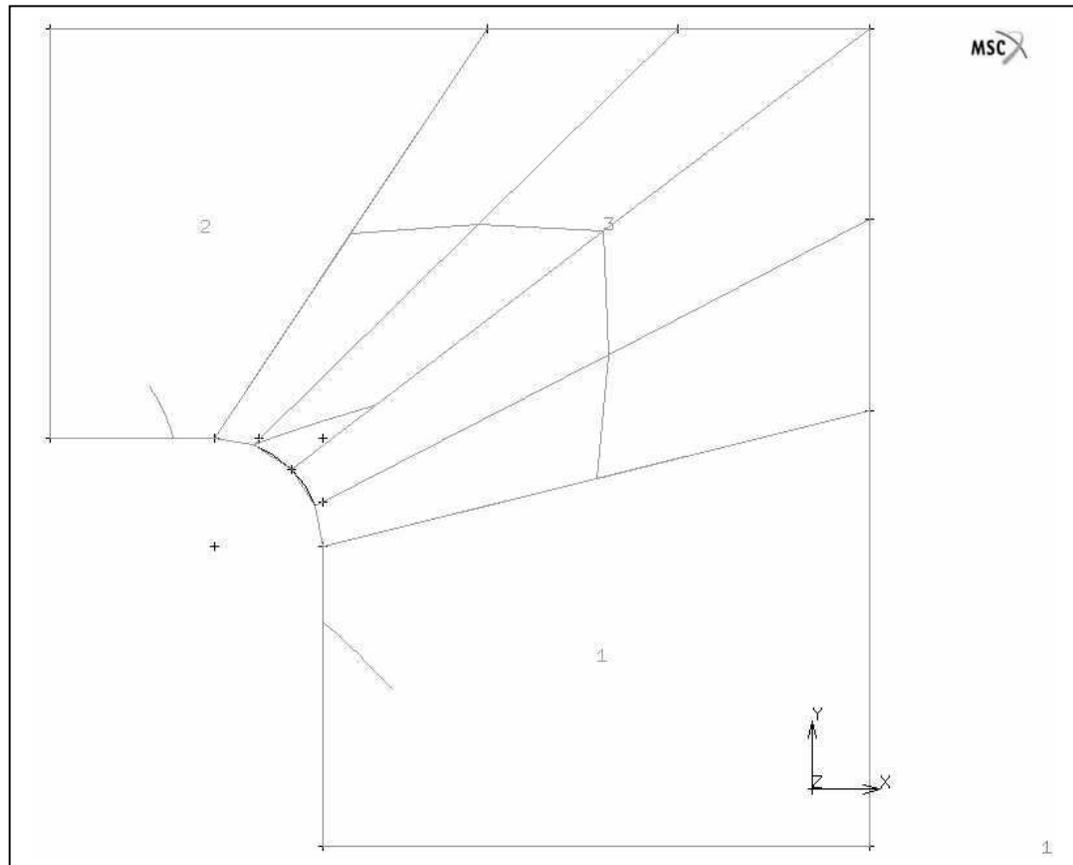


Figure 8.4

4. Mesh each ruled surface using the CONVERT option.

4a. Mesh surface 1.

<ML> MAIN MENU / MESH GENERATION / CONVERT

4a1. Select the mesh divisions.

<ML> MAIN MENU / MESH GENERATION / CONVERT / DIVISIONS

Enter the mesh divisions at the command line, with a space separating each value.

```
> 10 8
```

4a2. Select the mesh bias factors.

```
<ML> MAIN MENU / MESH GENERATION / CONVERT / BIAS FACTORS
```

Enter the mesh bias factors at the command line, with a space separating each value.

```
> 0.0 -0.35
```

4a3. Mesh the surface.

```
<ML> MAIN MENU / MESH GENERATION / CONVERT / SURFACES TO  
ELEMENTS
```

```
<ML> to select surface 1.
```

```
<MR> or END LIST
```

4b. Mesh surface 2.

```
<ML> MAIN MENU / MESH GENERATION / CONVERT
```

Note: You should already be located in this menu.

4b1. Select the mesh divisions.

```
<ML> MAIN MENU / MESH GENERATION / CONVERT / DIVISIONS
```

Enter the mesh divisions at the command line, with a space separating each value.

```
> 10 8
```

Note: The mesh divisions should already be set at these values from step 4a.

4b2. Select the mesh bias factors.

<ML> MAIN MENU / MESH GENERATION / CONVERT / BIAS FACTORS

Enter the mesh bias factors at the command line, with a space separating each value.

> 0.0 -0.35

Note: The mesh bias factors should already be set at these values from step 4a.

4b3. Mesh the surface.

<ML> MAIN MENU / MESH GENERATION / CONVERT / SURFACES TO
ELEMENTS

<ML> to select surface 2.

<MR> or END LIST

4c. Mesh surface 3.

<ML> MAIN MENU / MESH GENERATION / CONVERT

Note: You should already be located in this menu.

4c1. Select the mesh divisions.

<ML> MAIN MENU / MESH GENERATION / CONVERT / DIVISIONS

Enter the mesh divisions at the command line, with a space separating each value.

> 10 8

4c2. Select the mesh bias factors.

<ML> MAIN MENU / MESH GENERATION / CONVERT / BIAS FACTORS

Enter the mesh bias factors at the command line, with a space separating each value.

> 0.0 -0.35

4c3. Mesh the surface.

<ML> MAIN MENU / MESH GENERATION / CONVERT / SURFACES TO
ELEMENTS

<ML> to select surface 3.

<MR> or END LIST

4c. Turn off surface labels.

<ML> STATIC MENU / PLOT / LABEL SURFACES

<ML> STATIC MENU / PLOT / REDRAW

4d. Turn off surface displays.

<ML> STATIC MENU / PLOT

<ML> STATIC MENU / PLOT / DRAW SURFACES

<ML> STATIC MENU / PLOT / REDRAW

4e. Turn off point displays.

<ML> STATIC MENU / PLOT / DRAW POINTS

<ML> STATIC MENU / PLOT / REDRAW

4f. Exit the PLOT menu.

<MR> or <ML> RETURN

4g. Return to MESH GENERATION menu.

<MR> or <ML> RETURN

5. Sweep the mesh to insure that all elements are properly connected.

<ML> MAIN MENU / MESH GENERATION / SWEEP

<ML> MAIN MENU / MESH GENERATION / SWEEP / ALL

Note: Duplicate geometrical and mesh entities will be deleted so that proper mesh connectivity is achieved.

Return to MESH GENERATION menu.

<MR> or <ML> RETURN

6. Check for upside down elements.

<ML> MAIN MENU / MESH GENERATION / CHECK

<ML> MAIN MENU / MESH GENERATION / CHECK / UPSIDE DOWN

Note: All elements should be numbered locally in a counter-clockwise direction. Those elements numbered locally in a clockwise fashion are defined as upside down, and are highlighted when the above command is issued. Executing the FLIP ELEMENTS command should flip these elements.

<ML> MAIN MENU / MESH GENERATION / CHECK / FLIP ELEMENTS

Note: The upside down elements are already selected.

<ML> MAIN MENU / MESH GENERATION / CHECK / ALL: SELECT.

Note: Verify that all elements are now oriented correctly.

<ML> MAIN MENU / MESH GENERATION / CHECK / UPSIDE DOWN

Return to the MAIN menu.

<ML> MAIN MENU / MESH GENERATION / CHECK / MAIN

The result of this step is shown in Figure 8.5

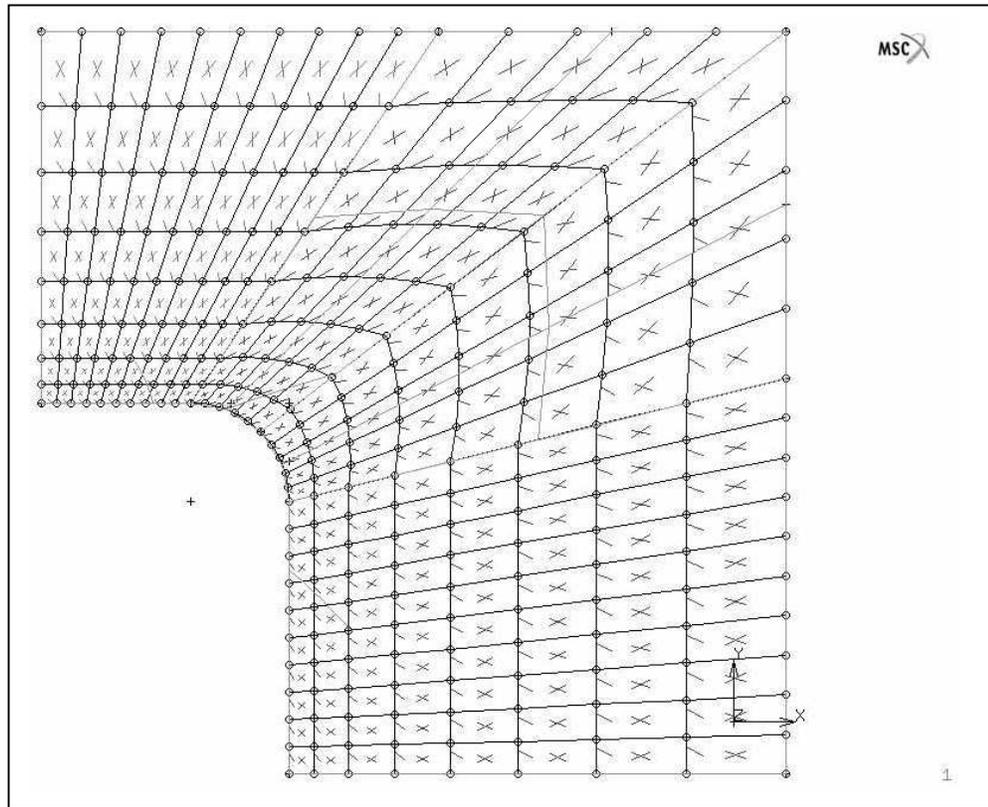


Figure 8.5

7. Add boundary conditions.
 - 7a. Specify the symmetry condition (zero horizontal displacement) on the left edge.
 - 7a1. Set up a new boundary condition set.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NEW  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NAME
```

At the command line, enter a name for this boundary condition set.

```
> SymLeft
```

7a2. Define the nature of the boundary condition.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED  
DISPLACEMENT
```

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED  
DISPLACEMENT / ON X DISPLACE
```

The small box to the immediate left of the ON button for X DISPLACE should now be highlighted.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED  
DISPLACEMENT / OK
```

7a3. Apply the condition to nodes along the left edge.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NODES ADD
```

Box pick the nodes lying on the left edge of the model, or

```
<ML> to select each node individually.
```

```
<MR> or END LIST
```

The result of this step is shown in Figure 8.6

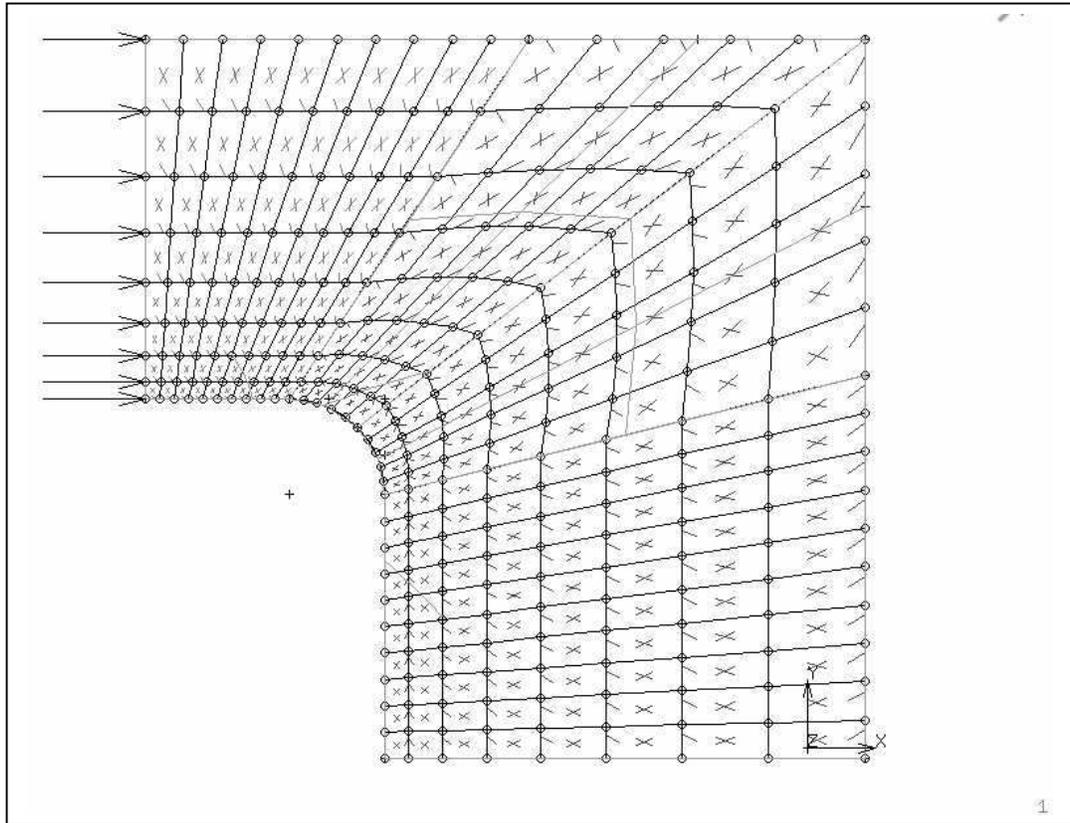


Figure 8.6

7b. Specify the symmetry condition (zero vertical displacement) on the bottom edge.

7b1. Set up a new boundary condition set.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NEW  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NAME
```

At the command line, enter a name for this boundary condition set.

```
> SymBottom
```

7b2. Define the nature of the boundary condition.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED  
DISPLACEMENT
```

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED  
DISPLACEMENT / ON Y DISPLACE
```

The small box to the immediate left of the ON button for Y DISPLACE should now be highlighted.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED  
DISPLACEMENT / OK
```

7b3. Apply the condition to nodes along the bottom edge.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NODES ADD
```

Box pick the nodes lying on the bottom edge of the model, or
<ML> to select each node individually.

```
<MR> or END LIST
```

The result of this step is shown in Figure 8.7

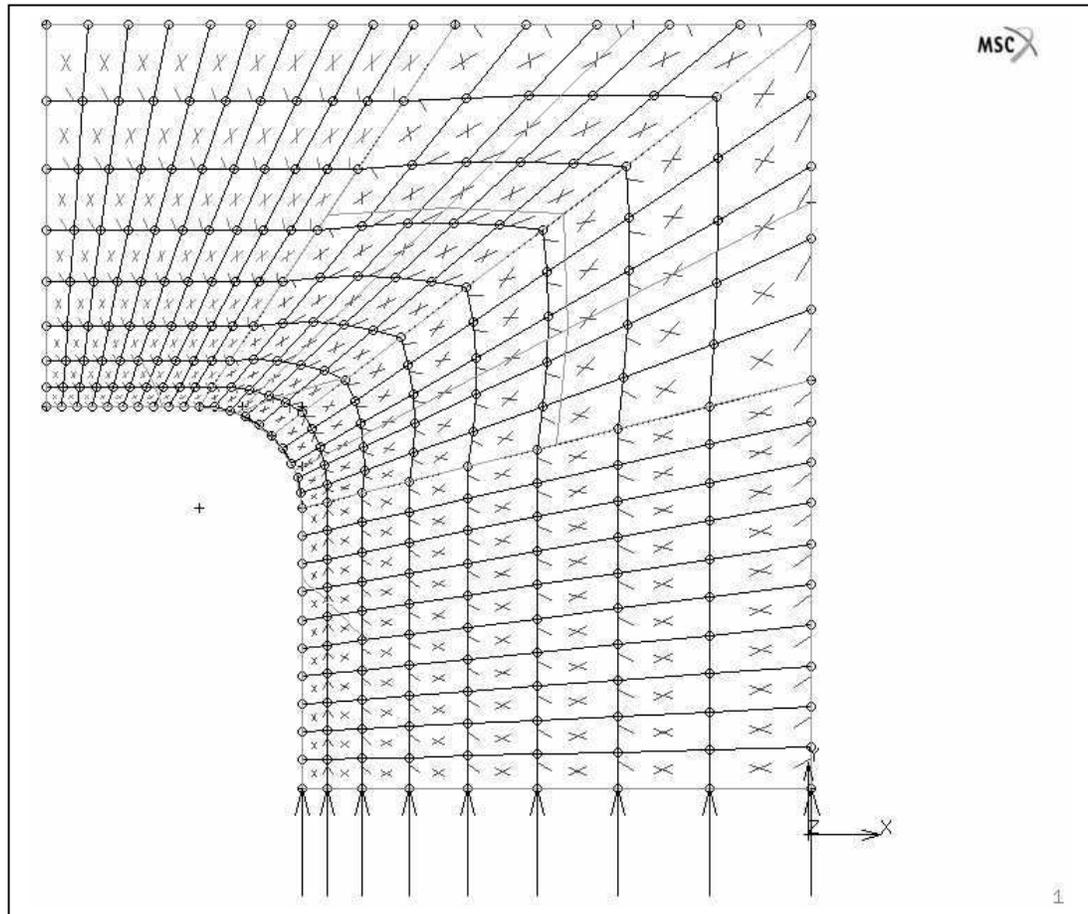


Figure8.7

7c. Specify the edge pressure load on the top edge of the model.

7c1. Set up a new boundary condition set.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NEW  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NAME
```

At the command line, enter a name for this boundary condition set.

```
> PressureTop
```

7c2. Define the nature of the boundary condition.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / EDGE LOAD  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / EDGE LOAD  
      / ON PRESSURE
```

The small box to the immediate left of the ON button for PRESSURE should now be highlighted. Next, the value of the pressure must be specified. Note that a tensile load is specified as a negative pressure.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / EDGE LOAD  
      / PRESSURE
```

```
> -14285.7
```

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / EDGE LOAD  
      / OK
```

7c3. Apply the edge load to element edges along the top edge.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / EDGES ADD
```

Box pick the element edges lying on the top edge of the model, or <ML> to select each edge individually.

```
<MR> or END LIST
```

The result of this step is shown in Figure 8.8

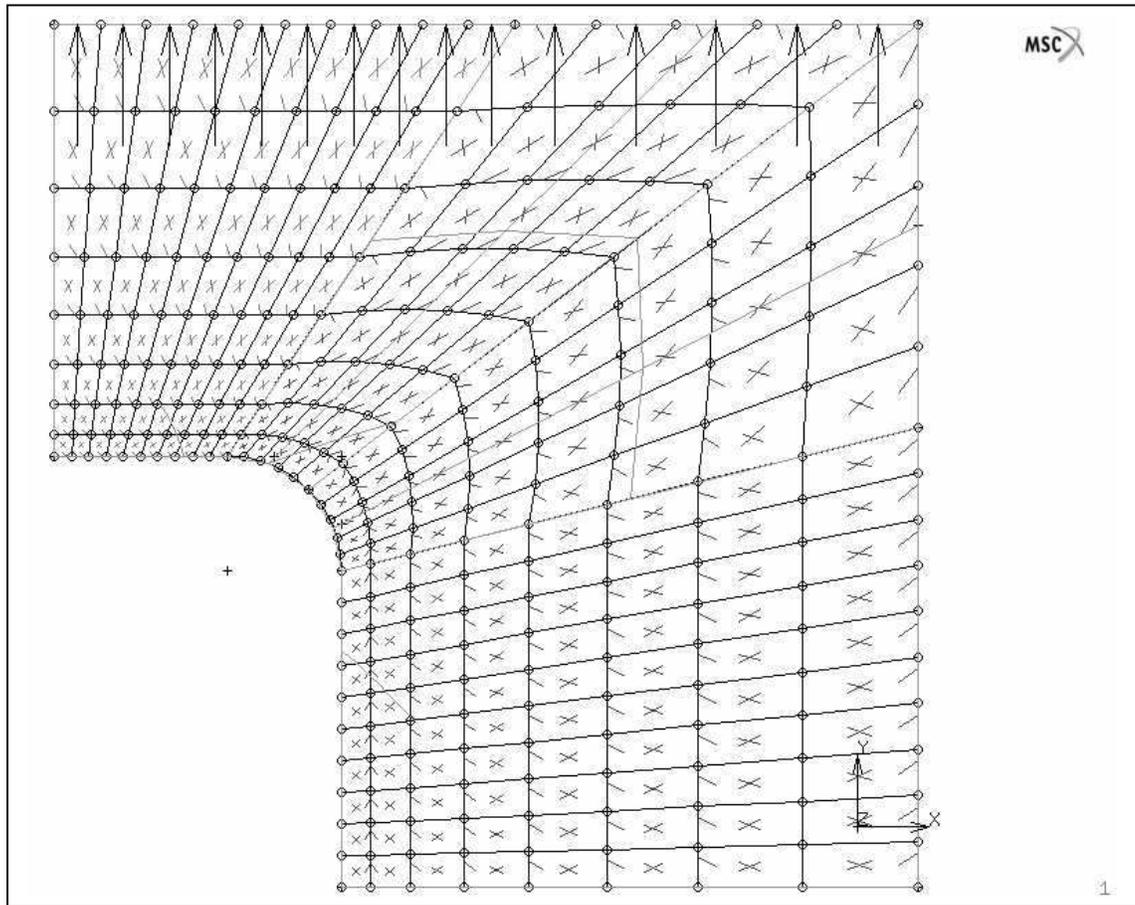


Figure 8.8

7d. Specify the edge pressure load on the right edge of the model.

7d1. Set up a new boundary condition set.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NEW  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NAME
```

At the command line, enter a name for this boundary condition set.

```
> PressureRight
```

7d2. Define the nature of the boundary condition.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / EDGE LOAD  
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / EDGE LOAD  
      / ON PRESSURE
```

The small box to the immediate left of the ON button for PRESSURE should now be highlighted. Next, the value of the pressure must be specified. Note that a tensile load is specified as a negative pressure.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / EDGE LOAD  
      / PRESSURE
```

```
> -7142.8
```

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / EDGE LOAD  
      / OK
```

7d3. Apply the edge load to element edges along the right edge.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / EDGES ADD
```

Box pick the element edges lying on the right edge of the model, or <ML> to select each edge individually.

```
<MR> or END LIST
```

The result of this step is shown in Figure 8.9

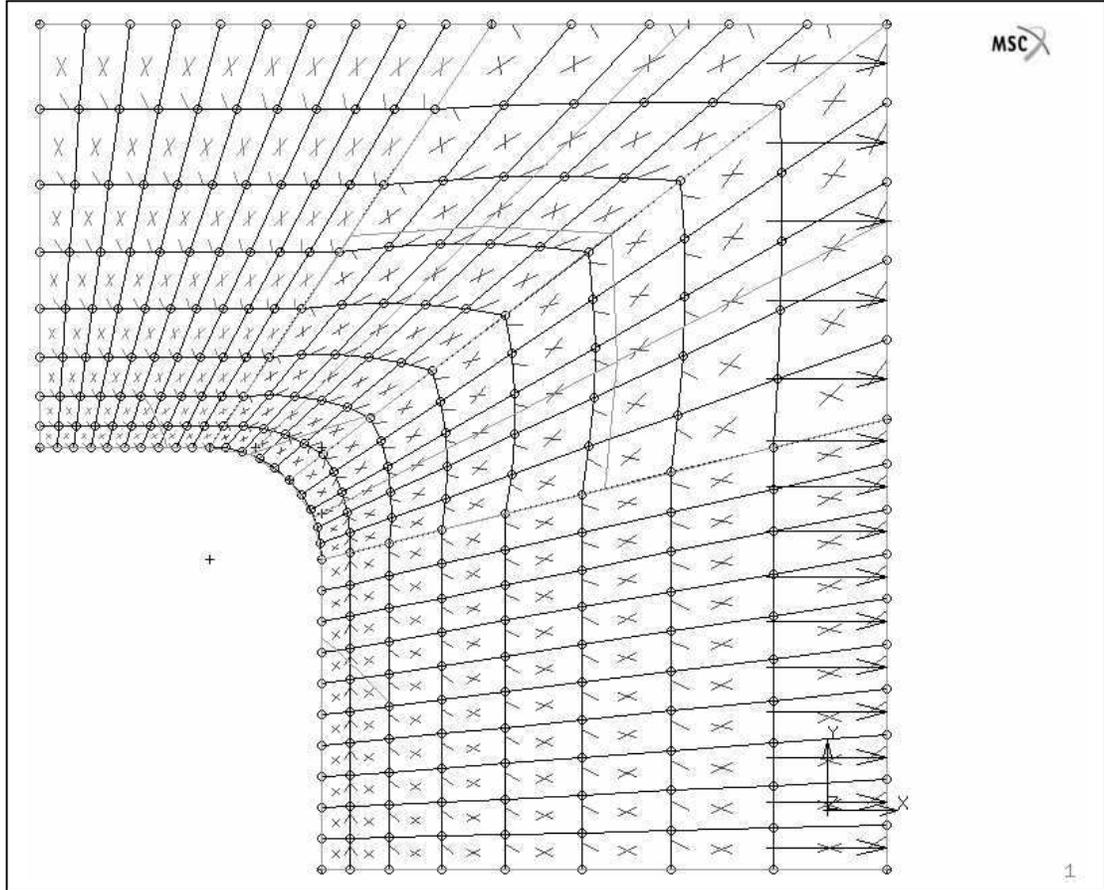


Figure 8.9

7e. Display all boundary conditions for verification.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / ID BOUNDARY CONDS
```

After verifying that boundary conditions have been applied properly, turn off the boundary condition ID's by repeating the last command.

7f. Return to the MAIN menu.

```
<ML> MAIN MENU / BOUNDARY CONDITIONS / MAIN
```

8. Specify the material properties of each element.

8a. Set up a new material property set.

```
<ML> MAIN MENU / MATERIAL PROPERTIES
<ML> MAIN MENU / MATERIAL PROPERTIES / NEW
<ML> MAIN MENU / MATERIAL PROPERTIES / NAME
```

At the command line, enter a name for this material property set.

```
> AL2024-T3
```

8b. Define the nature of the material.

```
<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC
<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC / YOUNG'S
      MODULUS
```

```
> 10.5e6
```

```
<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC / POISSON'S
      RATIO
```

```
> 0.33
```

Note: Only Young's modulus and Poisson's ratio need to be specified for this problem.

```
<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC / OK
```

8c. Apply the material properties to all elements.

```
<ML> MAIN MENU / MATERIAL PROPERTIES / ELEMENTS ADD
```

Since the properties are being applied to all elements in the model, the simplest way to select the elements is to use the ALL EXISTING option.

```
<ML> ALL: EXIST.
```

- 8d. Display all material properties for verification.

```
<ML> MAIN MENU / MATERIAL PROPERTIES / ID MATERIALS
```

After verifying that material properties have been applied properly, turn off the material property ID's by repeating the last command.

- 8e. Return to the MAIN menu.

```
<ML> MAIN MENU / MATERIAL PROPERTIES / MAIN
```

9. Specify the thickness of each element.

- 9a. Set up a new geometric property set.

```
<ML> MAIN MENU / GEOMETRIC PROPERTIES  
<ML> MAIN MENU / GEOMETRIC PROPERTIES / NEW  
<ML> MAIN MENU / GEOMETRIC PROPERTIES / NAME
```

At the command line, enter a name for this geometric property set.

```
> Thickness
```

- 9b. Define the nature of the geometric property.

```
<ML> MAIN MENU / GEOMETRIC PROPERTIES / PLANAR  
<ML> MAIN MENU / GEOMETRIC PROPERTIES / PLANAR / PLANE STRESS  
<ML> MAIN MENU / GEOMETRIC PROPERTIES / PLANAR / PLANE STRESS  
    / THICKNESS
```

```
> 0.063
```

```
<ML> MAIN MENU / GEOMETRIC PROPERTIES / PLANAR / PLANE STRESS  
    / OK
```

- 9c. Apply the geometric property to all elements.

```
<ML> MAIN MENU / GEOMETRIC PROPERTIES / PLANAR / ELEMENTS ADD
```

Since the property is being applied to all elements in the model, the simplest way to select the elements is to use the ALL EXISTING option.

```
<ML> ALL: EXIST.
```

- 9d. Display all geometric properties for verification.

```
<ML> MAIN MENU / GEOMETRIC PROPERTIES / ID GEOMETRIES
```

After verifying that geometric properties have been applied properly, turn off the geometric property ID's by repeating the last command.

- 9e. Return to the MAIN menu.

```
<ML> MAIN MENU / GEOMETRIC PROPERTIES / MAIN
```

10. Prepare the loadcase.

```
<ML> MAIN MENU / LOADCASES
```

```
<ML> MAIN MENU / LOADCASES / MECHANICAL
```

```
<ML> MAIN MENU / LOADCASES / MECHANICAL / STATIC
```

```
<ML> MAIN MENU / LOADCASES / MECHANICAL / STATIC / LOADS
```

Verify that all loads (i.e., boundary constraints and edge loads) created in step 7 are selected. The small box to the immediate left of all selected loads will be highlighted. If they are not already selected, then select them using the <ML>.

```
<ML> MAIN MENU / LOADCASES / MECHANICAL / STATIC / LOADS / OK
```

```
<ML> MAIN MENU / LOADCASES / MECHANICAL / STATIC / OK
```

```
<ML> MAIN MENU / LOADCASES / MECHANICAL / MAIN
```

11. Prepare the job for execution.

- 11a. Specify the analysis class and select loadcases.

```
<ML> MAIN MENU / JOBS
<ML> MAIN MENU / JOBS / MECHANICAL
<ML> MAIN MENU / JOBS / MECHANICAL / lcase1
```

11b. Select the analysis dimension.

```
<ML> MAIN MENU / JOBS / MECHANICAL / PLANE STRESS
```

11c. Select the desired output variables.

```
<ML> MAIN MENU / JOBS / MECHANICAL / JOB RESULTS
<ML> MAIN MENU / JOBS / MECHANICAL / JOB RESULTS / stress
<ML> MAIN MENU / JOBS / MECHANICAL / JOB RESULTS / von_mises
<ML> MAIN MENU / JOBS / MECHANICAL / JOB RESULTS / OK

<ML> MAIN MENU / JOBS / MECHANICAL / OK
```

11d. Select the element to use in the analysis.

```
<ML> MAIN MENU / JOBS / ELEMENT TYPES
<ML> MAIN MENU / JOBS / ELEMENT TYPES / MECHANICAL
<ML> MAIN MENU / JOBS / ELEMENT TYPES / MECHANICAL / PLANE
    STRESS
```

Select element number 3, a fully-integrated, four-noded quadrilateral.

```
<ML> MAIN MENU / JOBS / ELEMENT TYPES / MECHANICAL / PLANE
    STRESS / 3
<ML> MAIN MENU / JOBS / ELEMENT TYPES / MECHANICAL / PLANE
    STRESS / OK
<ML> MAIN MENU / JOBS / ELEMENT TYPES / MECHANICAL / OK
```

11e. Apply the element selection to all elements.

Since the element type is being applied to all elements in the model, the simplest way to select the elements is to use the ALL EXISTING option.

<ML> ALL: EXIST.

11f. Display all element types for verification.

<ML> MAIN MENU / JOBS / ELEMENT TYPES / ID TYPES

After verifying that element types have been applied properly, turn off the element type ID's by repeating the last command.

<ML> MAIN MENU / JOBS / ELEMENT TYPES / RETURN

11g. SAVE THE MODEL!

<ML> STATIC MENU / FILES

<ML> STATIC MENU / FILES / SAVE AS

In the box to the right side of the SELECTION heading, type in the name of the file that you want to create. The name should be of the form FILENAME.mud, where FILENAME is a name that you choose.

<ML> STATIC MENU / FILES / SAVE AS / OK

<ML> STATIC MENU / FILES / RETURN

11h. Execute the analysis.

<ML> MAIN MENU / JOBS / RUN

<ML> MAIN MENU / JOBS / RUN / SUBMIT 1

11i. Monitor the status of the job.

<ML> MAIN MENU / JOBS / RUN / MONITOR

When the job has completed, the STATUS will read: Complete. A successful run will have an EXIT NUMBER of 3004. Any other exit number indicates that an error occurred during the analysis, probably due to an error in the model.

<ML> MAIN MENU / JOBS / RUN / OK

<ML> MAIN MENU / JOBS / RETURN

12. Postprocess the results.

12a. Open the results file and display the results.

```
<ML> MAIN MENU / RESULTS
<ML> MAIN MENU / RESULTS / OPEN DEFAULT
<ML> MAIN MENU / RESULTS / NEXT INCREMENT
<ML> MAIN MENU / RESULTS / CONTOUR BANDS
```

A contour plot of the X-displacement should appear.

12b. Display a different output variable.

```
<ML> MAIN MENU / RESULTS / SCALAR
<ML> MAIN MENU / RESULTS / SCALAR / Comp 11 of Stress
<ML> MAIN MENU / RESULTS / SCALAR / OK
```

A contour plot of the stress in the X-direction should appear.

12c. Display nodal values of the output variable.

```
<ML> MAIN MENU / RESULTS / NUMERICS
```

It is difficult to read the values when the entire model is displayed. To view the nodal values, zoom in on the region of interest using the zoom box on the static menu. To view the entire model again, use the FILL command on the static menu.

TURN IN:

1. A stress contour plot for:
 - (a) x-component of stress
 - (b) y-component of stress
 - (c) von-mises stress
2. A brief discussion of the validity of the model, based on the hand calculations performed prior to the finite element analysis.

3. A prediction of whether the structure will fail under the applied loads.
4. An answer to the question "What could be done to reduce the stresses in critical regions near the cutout?"