

ME 477: Finite Element Analysis
Fall 2007

ANSYS Assignment #2
Linear Elastic Plane Stress Analysis of a Bracket

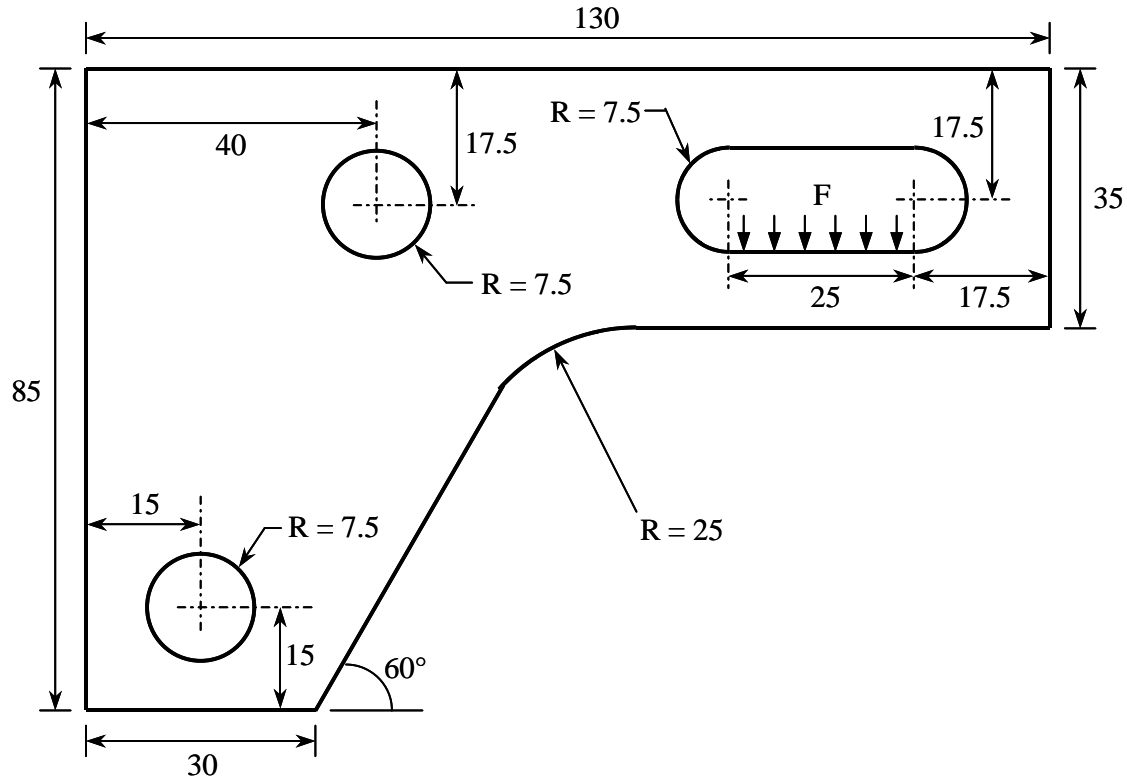
The bracket shown below is made of steel ($E = 200$ GPa, $\nu = 0.3$) with a yield strength of $S_y = 260$ MPa, and is 10 mm thick (all dimensions are in mm). The bracket is supported by two 15 mm diameter bolts through the offset holes. The bolts can be assumed to be rigid and frictionless. The bracket supports a uniform distributed load (F) that acts along the bottom edge of the slot as shown (assume the load acts along the entire 25 mm lengths of the slot).

Using ANSYS, perform a linear elastic, plane stress analysis of the bracket, using 8 node, 2D isoparametric plane stress elements (PLANE 82). The following tasks should be completed as part of this analysis:

- Determine the maximum load (F) that can be applied to the bracket without causing yielding and the location of the maximum stress (i.e., where would yielding initiate?). Note that since this is a linear elastic analysis, stresses in the component will scale linearly with loads. Also determine the maximum deflection at the right end of the bracket caused by this load. Specify the yield criterion employed in this analysis.
- Provide evidence that the discretization error (due to mesh size) is reasonably small. This will require you to run at least two analyses with different meshes to check for convergence of the solution.
- If the design specifications for the bracket require that it support a load that is 20% larger than the maximum load found in the previous analysis (without yielding), modify the bracket to satisfy this requirement (i.e., “beef up” the bracket where necessary to reduce the maximum stresses at the critical locations, so that it can support a load that is 120% of the load obtained previously without yielding). For this analysis, the location and size of the bolt holes and slot may not be altered, nor the thickness of the bracket or position and distribution of the load (only the outer shape of the bracket may be altered). The objective is to redesign the bracket to support the increased load, without unnecessarily increasing the weight (i.e., don’t “beef it up” more than necessary). Verify your redesigned bracket using ANSYS.

Submit a brief report describing the objectives, assumptions, and results of your analysis. The report should be aimed at someone with an engineering background, but who is not familiar with the details of the project. The report should include the following:

- **Introduction:** Briefly describe the component, loading and constraints, and the objectives of the analysis.
- **FE Model:** Describe the FE model; i.e., the type and size of elements, how the loads and constraints were modeled in ANSYS, pertinent assumptions, etc.
- **Results and Discussion:** Summarize the results of your analysis. Include, for each analysis, a plot of the deformed geometry, a pertinent stress contour plot, the locations of the maximum stresses, the maximum load (F) that can be applied, and the magnitude of the deflection at the right end of the bracket. Provide some interpretation of the results, including:
 - Do the results seem reasonable - qualitatively or quantitatively? Can you provide any verification via hand calculations?
 - Comment on the influence of the constraints (boundary conditions) on the stress distribution in the bracket. How realistic are the assumptions made in the FE analysis?Also provide a dimensioned drawing/sketch of the re-designed bracket.
- **Summary and Conclusions:** Summarize your findings for both the original and modified bracket.



Modeling Hints:

- The load can be applied as a pressure along the lower edge of the slot.
- To model the displacement constraints imposed by rigid, frictionless pins (bolts), a good technique is to restrict the *radial* displacements of nodes along the hole circumference. This is easily done by applying symmetry boundary conditions along the lines defining the hole circumference. Symmetry boundary conditions restrict nodal displacements perpendicular to the line (radial direction in this case), but not tangent to the line.
- There are a variety of tools and methods that can be used to create the geometry. Some modeling tools (menu paths) are shown on the following page. Basic area shapes (rectangles, circles) can be defined as in the previous example (it may be necessary to subtract, overlap or delete extraneous areas). Additional lines can be defined by first defining keypoints, or by specifying an angle with respect to another line, etc. A line fillet can be created by specifying the two lines and the radius. Once lines have been defined, areas can be defined by picking the lines that bound the keypoints. Extraneous lines may also need to be deleted. In some cases, coincident keypoints may be created (i.e., two keypoints at the same location). The extra keypoints can be eliminated by merging coincident keypoints (Numbering Ctrl → Merge Items → Keypoints).
- Since this is a linear analysis, stresses in the component are linearly proportional to loads.
- When meshing, ensure elements are small in regions of high stress gradients. Elements may be larger where stress gradients are less severe.

