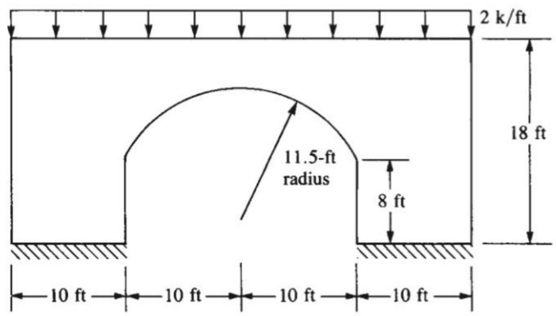
Lab #4: MSC Marc/Mentat For 2D Elasticity Analysis (Plane Strain)

For the concrete overpass structure shown, determine the maximum equivalent Von Mises and Principal stresses and their locations. Assume plane strain conditions. Mesh the structure with 36-inch elements. Redo the analysis with 12-inch elements and compare the results. The overpass is loaded by uniformly distributed load of 2000 lb/ft, $E = 3.0 \times 10^6\ psi$ and $\nu = 0.30$.



A. SETTING UP

Create folder “Lab04” in your user directory. Start Marc Mentat, set current directory to the folder “Lab04”, and save the model as “lab04”.

B. GEOMETRY AND MESH GENERATION

1. Open **Geometry & Mesh** tab in Main Menu and change units to “*in*”. Don’t forget to convert feet to inches as you construct your model!
2. To display desired entities on your screen go to **View → Plot control** and check the boxes next to the desired entities (Nodes, Points, etc.)
3. Open **Geometry & Mesh** widget and create a 2D outline of the structure. Use **Line** curves to generate straight edges and **Arc Cen/Pnt/Pnt** (center of the arc is 2.32 ft from the bottom) to generate the half-arc between two points.
4. Generate mesh:
    a) Apply curve divisions (see previous tutorial) with target length control.
    b) Automesh the outline (**Automesh → Planar → Triangles (Adv. Frnt) → Tri Mesh!**) and select all curves.

C. GEOMETRIC AND MATERIAL PROPERTIES

1. Create the geometric property (**Structural → Planar → Plane Strain**) and assign it to all elements.
2. Create the material property, enter the given material parameters and assign to all elements.

D. BOUNDARY CONDITIONS AND LOADS

1. Create and assign the built-in boundary condition for the base.
2. Create and assign the distributed loading boundary condition (**Edge Load, Force/Unit Length**).

E. JOB AND RESULT OUTPUT

1. Create a new structural job, check “Linear Elastic Analysis”, select “Plane Strain” for Analysis Dimension.
2. In **Job Results** check “Stress” tensor and “Equivalent Von Mises Stress” scalar.
3. Assign correct element type **Element Types → Planar → Solid → 6** to all elements
4. Save the model.

F. CALCULATION

1. Run the simulation. Remember, the successful error code is 3004.

G. POST-PROCESSING

1. Open the results. Take 3 screenshots of Contour Bands: “Component 11 Stress”, “Equivalent Von Mises Stress” and “Maximum Principal Value of Stress”. Include these 3 screenshots in your report.
2. Do you expect to see any stresses in Z-direction? Why or why not? (brief statement in your report)
3. Remesh the model and rerun the simulation. Before doing anything save the model as “lab04_refined” and then click “Clear Mesh” in **Geometry & Mesh** widget. Produce and include a screenshot of new Von Mises Stress distribution.
4. Plot stress distribution along the midspan depth of the structure using “Path Plot” as follows. Switch to “Sample Points” mode, then “Sample Points”, select start location (bottom node of the midspan, choose approximately in the middle of lower arc), then end location (top node), then enter the number of sample points (100). Click “Add Curves”, then “Add Curve”, then “Arc Length”, then “Maximum Principal Value of Stress”, then “Fit” to show the entire plot. Add two more curves to the same plot (“Comp 11 Stress” and “Comp 22 Stress”). Change the value of “Show IDs” from 1 to 0 to remove the data point labels. Take a screen shot of your graph, include in the report.
5. What can you say about max principal stress based on this plot? (brief statement in your report). To switch back to the model view, click “Window” (main menu) → “Model”.
6. Compare your predictions for Equivalent Von Mises Stress using 36-inch and 12-inch models, comment.