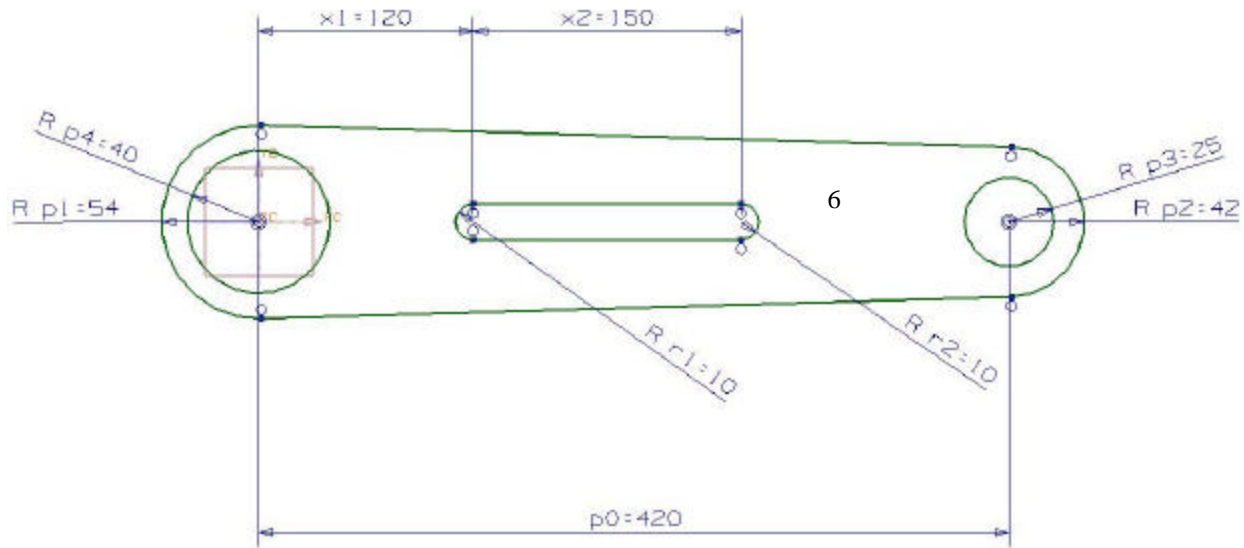# ME 475 – Analysis of a Torque Arm

Analysis Problem Statement:

The following problem will be analyzed using Abaqus.

Figure 1. Cross-sectional torque arm geometry. All units are in millimeters.

The torque arm has an extruded thickness of 3 mm, which is very small compared to the other dimensions. Therefore, this part can be analyzed using the plane stress assumption. In the middle of the torque arm is a cutout, which is created to save mass. The left-hand circle is pinned, while the right-hand circle has two loads applied: -2,789 N in the x-direction and 5,066 N in the y-direction.

Analysis Procedure:

1. Open Abaqus/CAE in a similar manner as in past labs.
   a. Create a folder called "Torque_Arm_FEA" in C:\temp
   b. Start → Run → type: "cmd", press enter
   c. Type: "C:", press enter
   d. Type "cd C:\temp\Torque_Arm_FEA"
   e. type: "abaqus cae"
2. In the Part Module, create a new part. It should be a 2D Planar, Deformable, Shell Part.

3. Sketch the geometry, as shown in Figure 1. Since this part will be optimized in the next lab, this sketch should be fully constrained (geometry that is fully constrained turns from yellow to green in the sketcher) and defined exactly as described. In addition to the dimensions shown (which should be entered in millimeters), the following sketch constraints are necessary to fully constrain the sketch.

   a. First, make a point a (0, 0) and fix it. Create the left-hand circle so its center is coincident with this point.

   b. The left-hand circle is concentric with the left-hand arc. Likewise for the right-hand side.

   c. All locations where a line meets an arc should have a tangent constraint applied.

   d. Put a horizontal construction line through the point created in step 3(a). Constrain the center of the right-hand circle to be coincident with the construction line. Likewise for the arcs in the cutout. Note: no lines other than the just created construction line can have a horizontal constraint. Remove them if they exist.

4. Finish the part creation.

5. Put a reference point in the center of the right-hand circle.

   a. Tools → Reference Point

6. Save the database.

7. In the property module, create a new material called "Steel" with the properties below. The part has units of mm, so the properties must be kept consistent.

   a. E = 207.4 GPa (enter 207.4e3)

   b. n = 0.3

   c. r = 7,810 kg/m^3 (enter 7.810e-6)

8. Create a new (solid, homogeneous) section, use the material created in the last step. Use a plane stress thickness of 3.

9. Apply the section to the torque arm geometry.

10. In the Assembly Module, instance the part as an independent part instance.

11. In the Step Module, create a Static, Linear Perturbation step after the Initial step.

12. In the field outputs, ensure that both "S" (stress) and "EVOL" (element volume) are requested.

13. In the Interaction Module, create a coupling constraint between the reference point and the right-hand circle. This couples the motion of the reference point with the DOF of the nodes on the surface the constraint is applied to (select "Surface" as the constraint region type).

    a. Use "Kinematic" coupling and only constrain displacements (U1 and U2).

14. In the Load module, create a boundary condition that constrains the displacements of the left-hand circle (U1=U2=0).

15. Apply a concentrated force to the reference point with a value of -2789 in the 1-direction and 5066 in the 2-direction.

16. In the Mesh Module, use a global mesh seed of 4.

17. Set your mesh controls to mesh with Quads (not Quad-dominated) and use the Advancing front algorithm with mapped meshing.

18. Specify an element type of CPS4, a 4-node bilinear plane stress quadrilateral element with full integration.

19. Mesh the part.

20. In the Job Module, create a job named: Torque_Arm_Analysis

21. Write the input deck of the job just created.

22. Save your CAE database and close Abaqus/CAE.

23. Run the analysis as in the previous lab.

    a. type in command prompt: abaqus interactive job=Torque_Arm_Analysis

24. The analysis is now complete. Post-process the results in Abaqus/Viewer.

As in the last lab, this part is similar to a cantilevered beam with a tip load, and in this case, an axial load as well. Display the S11 stress (in this case, S11 refers to the global 1-1 stress, for truss and beam elements local coordinates were used). Query the stress in the elements at the locations shown in Figure 2.
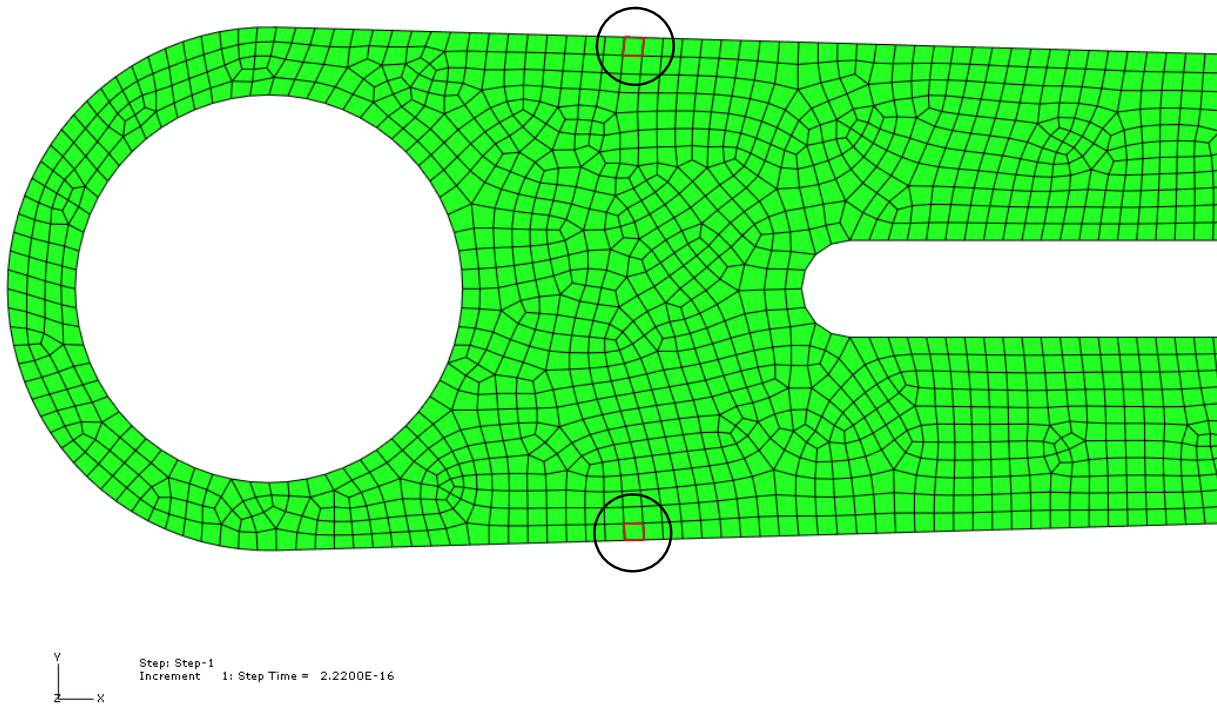

Figure 2. Elements at which to compare validation stress calculation (elements of interest highlighted in red and circled).

You will get four values for each element (one for each integration point in the element). Average these values for each element and this will be an approximate stress value for those locations.

Using a height of 103.843 mm (the height of the torque arm at the locations shown in Figure 2) and a width of 3 mm (the thickness) and assuming a rectangular cross-section calculate the stress in 1-1 direction at the top and bottom locations. The moment arm here is 345 mm. Report the

percent difference, and justify whether your hand calculations sufficiently validate the FE solution.

Hints:

(a) This is a combined loading problem. Use superposition of the loads.

(b) Since the locations of interest are at the top and bottom of the cross-section, the shear stress is zero here. The axial and bending stresses are completely in the 1-1 direction, so we can compare to the S11 stress, instead of calculating the von Mises stress.

(c) The axial and bending stresses add or subtract, depending on the location.

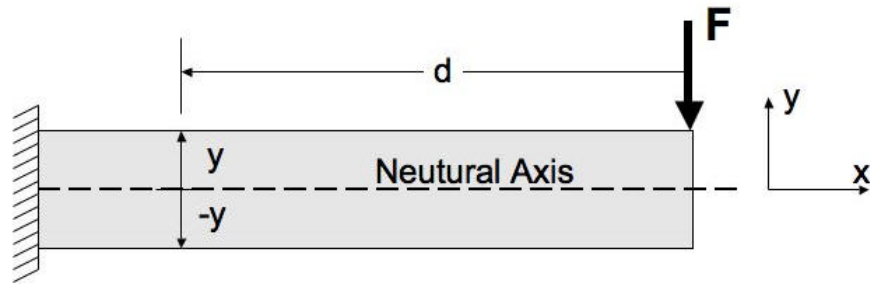(d) For bending stresses, refer to Figure 3 and Equation 1 (axial stress not shown).

Figure 3. Cantilever beam placed in bending by a load (F).

$$s = \frac{-M y}{I} \text{ or } \frac{-(F \cdot d) y}{I} \qquad (1)$$

Comparison of Kinematic and Distributed Coupling:

Rerun the analysis using Distributed coupling (do not constrain UR3, use Uniform weighting with the influence radius set "To outermost point on the region").

Worksheet for Analysis of a Torque Arm:

1. Present deformed plots of the von Mises component of stress (with the location of maximum stress indicated) and the U2 component of displacement for the results, both using Kinematic coupling.

2. Assuming a yield stress of 800 MPa and the von Mises failure criterion, what is the factor of safety for the structure?   _____

3. Model Validation: (Use S11 stress results)

h	b	I	My/I	P/A
103.843	3			

	Top	Bottom
FE Stresses (S11)		
Hand Calculations		
% Difference		

4. Also, show a deformed plot of the von Mises component of stress for the results using Distributed coupling. What difference do you notice between the two methods for constraining the right-hand circle? It may be necessary to increase the deformation scale factor greatly to see the difference (hint: it has to do with the right-hand circle).