1   Introduction and objectives

To this point, we have familiarized ourselves with Abaqus in the context of structural elements, such as trusses and beams, which involve simplifying assumptions due to their geometry. Now, we shift to solving full-field boundary-value problems in linear elastostatics. We will explore plane problems in elastostatics using the finite-element method. By plane problems, we mean those that may be reduced to two-dimensions. Plane formulations include plane stress, plane strain, and axisymmetric, each discussed below:

1. **Plane stress**: A plane-stress assumption is used to model a thin, plate-like solid, which is loaded in its plane. The solid must have uniform thickness, and the thickness must be much less than any cross-sectional dimensions. It is then assumed that $\sigma_{13} = \sigma_{23} = \sigma_{33} = 0$.

2. **Plane strain**: A plane-stain assumption is used to model solids which are very thick in one direction. The thickness must be greater than the cross-sectional dimensions, so that the geometry is approximately in the form of a long prismatic solid. It is then assumed that $\epsilon_{13} = \epsilon_{23} = \epsilon_{33} = 0$.

   In both plane stress and plane strain, there is no need to solve for the out-of-plane displacement u_3, so that a two-dimensional mesh is sufficient to calculate $u_1(x_1, x_2)$ and $u_2(x_1, x_2)$. The plane problem of elastostatics is then

Displacement:

$$u_1(x_1, x_2), \quad u_2(x_1, x_2)$$

Strain-displacement relations:

$$\epsilon_{11} = \frac{\partial u_1}{\partial x_1}, \quad \epsilon_{22} = \frac{\partial u_2}{\partial x_2}, \quad \epsilon_{12} = \frac{1}{2}\left[\frac{\partial u_1}{\partial x_2} + \frac{\partial u_2}{\partial x_1}\right]$$

Stress-strain relations:

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & 1-\nu \end{bmatrix} \begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{12} \end{bmatrix} \qquad \text{for plane stress}$$

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & 1-2\nu \end{bmatrix} \begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{12} \end{bmatrix} \qquad \text{for plane strain}$$

where E is the Young's modulus and ν the Poisson's ratio.

Equilibrium

$$\frac{\partial \sigma_{11}}{\partial x_1} + \frac{\partial \sigma_{12}}{\partial x_2} + b_1 = 0$$

$$\frac{\partial \sigma_{12}}{\partial x_1} + \frac{\partial \sigma_{22}}{\partial x_2} + b_2 = 0$$

where $\mathbf{b}$ is the external body force.

In a plane-stress or plane-strain simulation, the nodal solution variables are the displacements u_1 and u_2. Field output will include the displacement, strain, and stress components listed above.

3. **Axisymmetric**: An axisymmetric assumption is used to model solids that have rotational symmetry, which are subjected to axisymmetric loading. Under an axisymmetric simplification, i.e., $u_\theta = 0$ and $\partial(\cdot)/\partial\theta = 0$ for all quantities, the elastostatics problem becomes

Displacement

$$u_r(r, z), \quad u_z(r, z)$$

Strain-displacement relations

$$\epsilon_{rr} = \frac{\partial u_r}{\partial r}, \quad \epsilon_{\theta\theta} = \frac{u_r}{r}, \quad \epsilon_{zz} = \frac{\partial u_z}{\partial z}, \quad \epsilon_{rz} = \frac{1}{2}\left[\frac{\partial u_r}{\partial z} + \frac{\partial u_z}{\partial r}\right], \quad \epsilon_{r\theta} = \epsilon_{\theta z} = 0$$

Stress-strain relations

$$\begin{bmatrix} \sigma_{rr} \\ \sigma_{zz} \\ \sigma_{\theta\theta} \\ \sigma_{rz} \end{bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & (1-2\nu) \end{bmatrix} \begin{bmatrix} \epsilon_{rr} \\ \epsilon_{zz} \\ \epsilon_{\theta\theta} \\ \epsilon_{rz} \end{bmatrix}$$

Equilibrium

$$\frac{\partial \sigma_{rr}}{\partial r} + \frac{\partial \sigma_{rz}}{\partial z} + \frac{\sigma_{rr} - \sigma_{\theta\theta}}{r} + b_r = 0$$

$$\frac{\partial \sigma_{rz}}{\partial r} + \frac{\partial \sigma_{zz}}{\partial z} + \frac{\sigma_{rz}}{r} + b_z = 0$$

where $\mathbf{b}$ is the external body force.

In an axisymmetric analysis, the nodal solution variables are the displacements u_r and u_z. Field output will include the displacement, strain, and stress components listed above. In Abaqus notation, r is the 1-direction, z is the 2-direction, and θ is the 3-direction.

In this handout, we will consider

1. A thick-wall pressure vessel under plane-strain conditions and

2. A round stepped shaft under axisymmetric conditions.

Meshing: A key objective of these exercises is to familiarize ourselves with the art of meshing and the concept of mesh convergence. To this point, we have used finite elements that are capable of exact solutions. However, this was only due to the simplicity of the structural elements considered and is not a general feature. The finite-element method is, at its heart, an approximate technique, and in practice, an approximate displacement field is constructed as a piecewise-continuous, low-order polynomial within small (but finite-sized) "elements." In two-dimensional models (e.g., plane stress, plane strain, and axisymmetric), the elements are typically triangular or quadrilateral-shaped. A key feature of the method is that the accuracy of the approximate solution improves, approaching the "exact" solution as the size of the elements is systematically reduced by mesh refinement – a feature referred to as convergence. A coarse mesh can yield a poor result, and as the mesh is refined, the "exact" solution is asymptotically approached. However, simulation time also increases with mesh resolution, so a trade-off arises. It is important to obtain a sufficiently accurate solution without needlessly wasting computer time.

2 Exercises

2.1 Thick-wall pressure vessel

We will first analyze a thick-wall pressure vessel made from a homogeneous, isotropic linear elastic material under internal pressure and *plane-strain conditions*. Consider the following thick-wall cylinder with inner radius a and outer radius b, subjected to an internal pressure p, as shown.
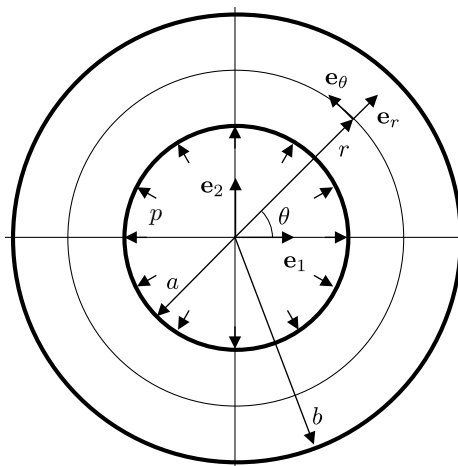
We assume that the outer surface at $r = b$ is traction-free and that no body forces are present. Under these conditions the analytical solution for the displacement field is

$$u_r(r) = \frac{(1+\nu)}{E} \frac{rp}{(b/a)^2 - 1} \left[1 - 2\nu + \left(\frac{b}{r} \right)^2 \right],$$ (1)

and the non-zero stress components are

$$\sigma_{rr}(r) = \frac{p}{(b/a)^2 - 1} \left[1 - \left(\frac{b}{r} \right)^2 \right],$$

$$\sigma_{\theta\theta}(r) = \frac{p}{(b/a)^2 - 1} \left[1 + \left(\frac{b}{r} \right)^2 \right].$$

We take the cylinder to be made of aluminum with $E = 70\,\text{GPa}$ and $\nu = 0.35$, the inner and outer radii to be $a = 1\,\text{m}$ and $b = 2\,\text{m}$, respectively, and the internal pressure to be $p = 100\,\text{MPa}$. We will take advantage of the symmetry of the problem and model one quarter of the cylinder as shown below.
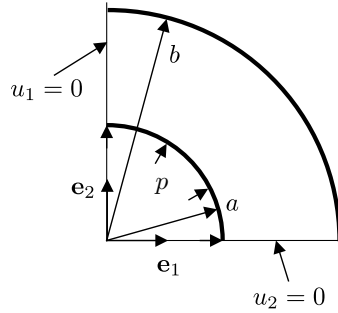

Figure 1: Schematic of the thick-wall pressure vessel showing quarter symmetry.

Below is an outline of the steps for performing the analysis in Abaqus/CAE:

- Part:

    - Part $\Rightarrow$ Create
    - Select 2D Planar, Deformable, Shell, and Approximate size: $5\,\text{m} \Rightarrow$ Continue
    - Sketch the part as pictured in Fig. 1 and click Done.

- Property:

    - Material $\Rightarrow$ Create
    - Mechanical $\Rightarrow$ Elasticity $\Rightarrow$ Elastic
    - Enter the material properties for aluminum and click OK
    - Section $\Rightarrow$ Create

- Solid $\Rightarrow$ Homogeneous $\Rightarrow$ Continue
- Make sure your material is selected and click OK
- Assign $\Rightarrow$ Section
- Select the entire part and click Done/OK

- Assembly:

  - Instance $\Rightarrow$ Create $\Rightarrow$ OK

- Step:

  - Step $\Rightarrow$ Create $\Rightarrow$ Static/General $\Rightarrow$ Continue $\Rightarrow$ OK

- Load:

  - BC $\Rightarrow$ Create $\Rightarrow$ Mechanical $\Rightarrow$ Displacement/Rotation $\Rightarrow$ Continue $\Rightarrow$ Select bottom face and click Done $\Rightarrow$ Enter U2=0 and click OK
  - BC $\Rightarrow$ Create $\Rightarrow$ Mechanical $\Rightarrow$ Displacement/Rotation $\Rightarrow$ Continue $\Rightarrow$ Select left face and click Done $\Rightarrow$ Enter U1=0 and click OK
  - Load $\Rightarrow$ Create $\Rightarrow$ Mechanical $\Rightarrow$ Pressure $\Rightarrow$ Continue $\Rightarrow$ Select inner face and click Done $\Rightarrow$ Enter Magnitude and click OK

- Mesh:

  - Make sure Object is set to Part.
  - Mesh $\Rightarrow$ Element Type $\Rightarrow$ Select the entire part and click Done $\Rightarrow$ Family: Plane Strain $\Rightarrow$ Click OK
  - Mesh $\Rightarrow$ Controls $\Rightarrow$ Element Shape: Quad $\Rightarrow$ Technique: Structured $\Rightarrow$ Click OK
  - Seed $\Rightarrow$ Part $\Rightarrow$ Approximate global size: $0.1\,\mathrm{m}$ $\Rightarrow$ Click OK
  - Mesh $\Rightarrow$ Part $\Rightarrow$ Yes

- Job:

  - Job $\Rightarrow$ Create $\Rightarrow$ Continue/OK
  - Job $\Rightarrow$ Submit $\Rightarrow$ Job-1
  - When the job successfully completes: Job $\Rightarrow$ Results $\Rightarrow$ Job-1

- Visualization:

  - Examine contour plots of displacement, stress, and strain.
  - Probe quantitative results at specific points of the model:
    Tools $\Rightarrow$ Query $\Rightarrow$ Probe values
  - Create a path and plot and export field output data along the path.

* Tools $\Rightarrow$ Path $\Rightarrow$ Create $\Rightarrow$ Type: Edge list $\Rightarrow$ Continue $\Rightarrow$ Add After... $\Rightarrow$ Select edges to be inserted into the path: by shortest distance $\Rightarrow$ Select first element face (make sure Start appears next to the node you intend, click Flip if it does not) $\Rightarrow$ Select end node for the path and click Done $\Rightarrow$ Click OK
* Tools $\Rightarrow$ XY Data $\Rightarrow$ Create $\Rightarrow$ Source: Path $\Rightarrow$ Continue $\Rightarrow$ Make sure the path/field output variable are correct $\Rightarrow$ Save As/OK
* Report $\Rightarrow$ XY $\Rightarrow$ Select the appropriate XY Data $\Rightarrow$ Setup: Provide a file name $\Rightarrow$ OK

The XY data will appear in a text file to be used in Matlab, Excel, etc.

Go back and repeat for different mesh resolutions.

- Mesh:

    - Seed $\Rightarrow$ Part $\Rightarrow$ Approximate global size: (Try finer and coarser meshes.)
    - Mesh $\Rightarrow$ Part $\Rightarrow$ Yes

- Job:

    - Job $\Rightarrow$ Submit.
    - When the job successfully completes: Job $\Rightarrow$ Results.

2.2 Stepped shaft under tension

Next, we consider an example of a stress concentration. A stress concentration is the phenomenon, in which the magnitude of the stress is raised in the region of a geometric feature. We will examine a stepped shaft under tension, as shown below.
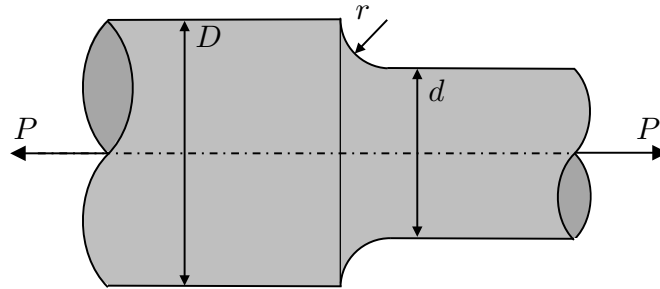
Defining the nominal stress in this geometry as $\sigma_0 = P/(\pi d^2/4)$, we can define a stress concentration factor, K_t, as the ratio of the maximum principal stress $\sigma_{1,\mathrm{max}}$ to the nominal stress, i.e.

$$K_t = \frac{\sigma_{1,\mathrm{max}}}{\sigma_0}.$$

We will determine K_t for a given geometry: $D = 6\,\mathrm{cm}$, $d = 4\,\mathrm{cm}$, and $r = 0.8\,\mathrm{cm}$. Since the problem is linear and stress-controlled, the numerical values we choose for the applied load P and the material properties E and ν are arbitrary in determining K_t.

Since the geometry and loads are symmetric about the center axis of the shaft, this configuration is an example of an axisymmetric problem, opening the door for two-dimensional modeling. We use the axisymmetric representation of the problem shown in Fig. 2 as the basis of our finite-element model.
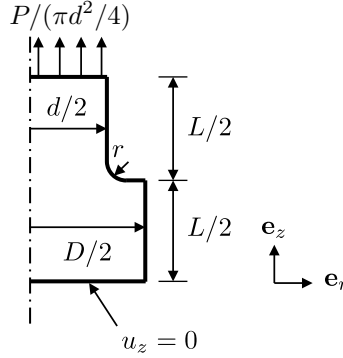

Figure 2: Schematic of a stepped shaft under tension.

Two important issues must be taken into consideration. First, the shaft must be taken to be sufficiently long so as not to affect the determination of the stress concentration factor. Based on experience, we take $L = 10\,\text{cm}$. (This experience is based upon something called Saint-Venant's principle.) Second, how do we apply the axial load? We wish to apply the load in such a way that the stress is uniform away from the shoulder. To do so, we apply a constant negative pressure ($= -\sigma_0$) on the top surface, and constrain the nodes on the bottom surface in the vertical direction.

To make our analysis concrete, we take material properties for aluminum: $E = 70\,\text{GPa}$ and $\nu = 0.35$, and take $\sigma_0 = P/(\pi d^2/4) = 100\,\text{MPa}$. Below is an outline of the steps for performing the analysis in Abaqus/CAE:

- Part:

    - Part $\Rightarrow$ Create

    - Select Axisymmetric, Deformable, Shell, and Approximate size: $0.5\,\text{m} \Rightarrow$ Continue

    - Sketch the part as pictured in Fig. 2 and click Done. (Use the Create Fillet tool when sketching the part.)

- Property:

    - Material $\Rightarrow$ Create

    - Mechanical $\Rightarrow$ Elasticity $\Rightarrow$ Elastic

    - Enter the material properties for aluminum and click OK

    - Section $\Rightarrow$ Create

    - Solid $\Rightarrow$ Homogeneous $\Rightarrow$ Continue

- Make sure your material is selected and click OK
- Assign ⇒ Section
- Select the entire part and click Done/OK.

- Assembly:

  - Instance ⇒ Create ⇒ OK

- Step:

  - Step ⇒ Create ⇒ Static/General ⇒ Continue ⇒ OK

- Load:

  - BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Continue ⇒ Select bottom face and click Done ⇒ Enter U2=0 and click OK
  - BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Continue ⇒ Select left face and click Done ⇒ Enter U1=0 and click OK
  - Load ⇒ Create ⇒ Mechanical ⇒ Pressure ⇒ Continue ⇒ Select top face and click Done ⇒ Enter Magnitude ($= -\sigma_0$) and click OK
    (Note: We could use the more general "Surface traction" type of load, but since the load under consideration is applied normal to the surface, it is simpler to consider it as a negative pressure.)

- Mesh:

  - Make sure Object is set to Part.
  - Mesh ⇒ Element Type ⇒ Select the entire part and click Done ⇒ Family: Axisymmetric Stress ⇒ Click OK.
  - Mesh ⇒ Controls ⇒ Element Shape: Quad ⇒ Technique: Free ⇒ Click OK
  - Seed ⇒ Part ⇒ Approximate global size: $0.002\,\mathrm{m}$ ⇒ Click OK
  - We wish to refine the mesh in the region of the fillet.
    Seed ⇒ Edges ⇒ Select the fillet face and click Done ⇒ Approximate element size: $0.0005\,\mathrm{m}$ ⇒ Click OK
  - Seed ⇒ Edges ⇒ Check the "Use single-bias picking," select the face immediately above the fillet, and click Done ⇒ Minimum size: $0.0005\,\mathrm{m}$ and Maximum size: $0.002\,\mathrm{m}$ ⇒ Apply (Make sure the fine seeds are on the end near the fillet. If not click Flip and OK)
    This step will lead to a mesh with a more gradual mesh size gradient.
  - Mesh ⇒ Part ⇒ Yes

- Job:

  - Job ⇒ Create ⇒ Continue/OK

- Job $\Rightarrow$ Submit $\Rightarrow$ Job-1

- When the job successfully completes: Job $\Rightarrow$ Results $\Rightarrow$ Job-1

- Visualization:

  - Examine contour plots of displacement, stress, and strain, especially in the region of the stress concentration.

  - Set the output field to Max. Principal stress and probe quantitative results at specific points of the model: Tools $\Rightarrow$ Query $\Rightarrow$ Probe values
    Specifically find the maximum value of the Max. Principal stress $\sigma_{1,\text{max}}$ and calculate the stress concentration factor $K_{\text{t}} = \sigma_{1,\text{max}}/\sigma_0$ for this geometry. Your answer should be $K_{\text{t}} \approx 1.6$.

Go back and repeat for different minimum seed sizes and recalculate the stress concentration factor K_{t}.

- Mesh:

  - Seed $\Rightarrow$ Edges $\Rightarrow$ Select the fillet face and click Done $\Rightarrow$ Try a finer or coarser element size.

  - Seed $\Rightarrow$ Edges $\Rightarrow$ Check the "Use single-bias picking," select the face immediately above the fillet, and click Done $\Rightarrow$ Adjust the minimum element size accordingly

  - Mesh $\Rightarrow$ Part $\Rightarrow$ Yes

- Job:

  - Job $\Rightarrow$ Submit $\Rightarrow$ Job-1

  - When the job successfully completes: Job $\Rightarrow$ Results $\Rightarrow$ Job-1