

ENGN 1750: Advanced Mechanics of Solids
Plane problems and stress concentrations

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. In this Abaqus session, we 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 strain 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:

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

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

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

Equilibrium

$$\begin{aligned}\frac{\partial \sigma_{11}}{\partial x_1} + \frac{\partial \sigma_{12}}{\partial x_2} + b_1 &= 0 \\ \frac{\partial \sigma_{12}}{\partial x_1} + \frac{\partial \sigma_{22}}{\partial x_2} + b_2 &= 0\end{aligned}$$

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

In an 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

$$\begin{aligned}\frac{\partial \sigma_{rr}}{\partial r} + \frac{\partial \sigma_{rz}}{\partial z} + \frac{\sigma_{rr} - \sigma_{\theta\theta}}{r} + b_r &= 0 \\ \frac{\partial \sigma_{rz}}{\partial r} + \frac{\partial \sigma_{zz}}{\partial z} + \frac{\sigma_{rz}}{r} + b_z &= 0\end{aligned}$$

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 session, we will consider

1. A thick-wall pressure vessel under plane strain conditions and
2. A round stepped shaft under axisymmetric conditions.

Meshing: A second objective of this session is familiarize ourselves with the art of meshing and the concept solution convergence. To this point, we have used finite elements, which 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. Our simulation results will be approximate with accuracy increasing with mesh resolution – a feature referred to as convergence. A coarse mesh can yield a poor result, and as the mesh is refined, the analytical 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 In-class 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_i , as shown

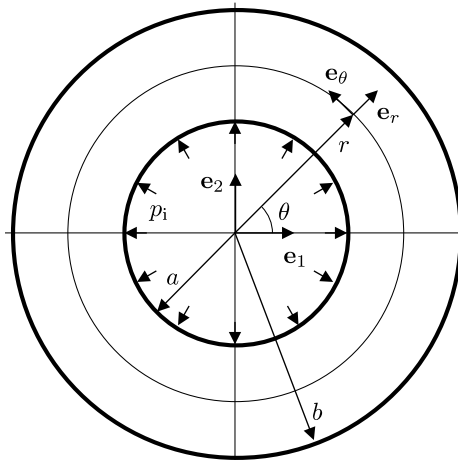


Figure 1: A thick-wall pressure vessel from a plane-strain perspective.

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

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

and the non-zero stress components are

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

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

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

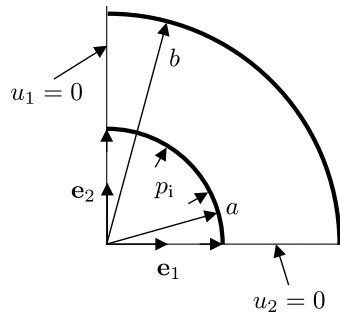


Figure 2: Schematic of the thick-wall pressure vessel showing quarter symmetry.

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 m \Rightarrow Continue
 - Sketch the part as pictured in Fig. 2 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 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.

- Examine the strain energy of the whole model.
 - Result \Rightarrow History Output \Rightarrow Strain energy: ALLSE for Whole Model \Rightarrow Plot
The final value is the total strain energy for the given conditions.
 - Select Save As/OK to save as XY data, which may be exported for external processing as above.

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:

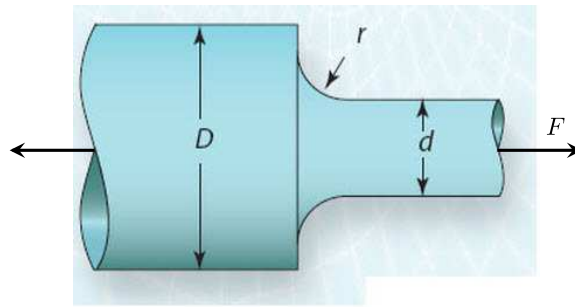


Figure 3: Schematic of a stepped shaft under tension.

Defining the nominal stress in this geometry as $\sigma_0 = F/(\pi d^2/4)$, we can define a stress concentration factor, K_t , as the ratio of the maximum principal stress $\sigma_{1,\max}$ to the nominal stress, i.e.

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

We will determine K_t for a given geometry: $D = 6$ cm, $d = 4$ cm, and $r = 0.8$ cm. Since the problem is linear and stress-controlled, the numerical values we choose for the applied load F 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. 4 as the basis of our finite element model.

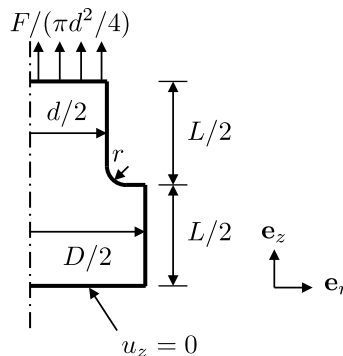


Figure 4: Schematic of a stepped shaft under tension.

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. Based on experience, we take $L = 10$ cm. (This experience is based upon something called Saint-Venant's principle.) Second, how do we apply the axial load? We want 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: $E = 70$ GPa and $\nu = 0.35$, and take $\sigma_0 = F/(\pi d^2/4) = 100$ 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 m \Rightarrow Continue
 - Sketch the part as pictured in Fig. 4 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 \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 $U_2=0$ and click OK
 - BC \Rightarrow Create \Rightarrow Mechanical \Rightarrow Displacement/Rotation \Rightarrow Continue \Rightarrow Select left face and click Done \Rightarrow Enter $U_1=0$ and click OK
 - Load \Rightarrow Create \Rightarrow Mechanical \Rightarrow Pressure \Rightarrow Continue \Rightarrow Select top face and click Done \Rightarrow 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 \Rightarrow Element Type \Rightarrow Select the entire part and click Done \Rightarrow Family: Axisymmetric Stress \Rightarrow Click OK.
 - Mesh \Rightarrow Controls \Rightarrow Element Shape: Quad \Rightarrow Technique: Free \Rightarrow Click OK
 - Seed \Rightarrow Part \Rightarrow Approximate global size: 0.002 m \Rightarrow Click OK
 - We wish to refine the mesh in the region of the fillet.
 - Seed \Rightarrow Edges \Rightarrow Select the fillet face and click Done \Rightarrow Approximate element size: 0.0005 m \Rightarrow Click OK
 - Seed \Rightarrow Edges \Rightarrow Check the “Use single-bias picking,” select the face immediately above the fillet, and click Done \Rightarrow Minimum size: 0.0005 m and Maximum size: 0.002 m \Rightarrow 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 \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, 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 $\sigma_{1,\max}$ and calculate the stress concentration factor $K_t = \sigma_{1,\max}/\sigma_0$ for this geometry. Your answer should be $K_t \approx 1.6$.

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

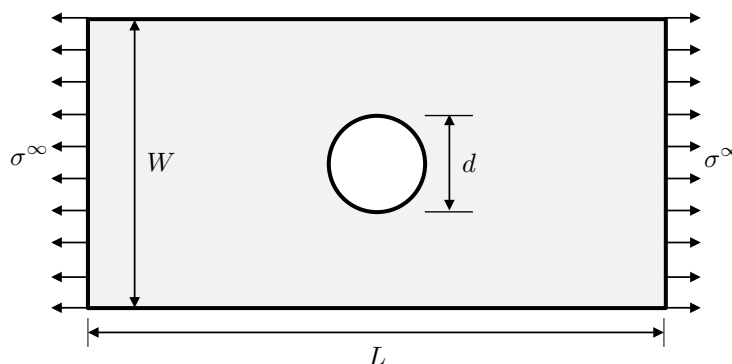
ENGN 1750: Advanced Mechanics of Solids
Abaqus Assignment 3

Due: Monday, October 27, 2014 OR Thursday, October 30, 2014 (in class)

1. Consider the axisymmetric stepped shaft from the in-class exercise. Again, we will take $D = 6$ cm and $d = 4$ cm. Repeat the analysis for $r = 0.4, 0.5, 0.6, 0.7$ cm. Submit a plot of K_t versus r/d . Be sure to normalize your simulation results in this way. Including the value you computed during class, your plot will consist of five data points for $r/d = 0.1, 0.125, 0.15, 0.175, 0.2$.

A qualitative statement of Saint-Venant's principle states that a localized geometrical feature with characteristic size r in a large solid only influences the stress in a region with size approximately $3r$ surrounding the feature. Comment in one or two sentences on the manner in which you observe Saint-Venant's principle in your simulation results.

2. Consider a plate of width W and length L with a centrally-located hole of diameter d . The plate is subject to a far-field tensile stress σ^∞ as shown below:



Defining the nominal stress in this geometry based upon the reduced area of the plate:

$$\sigma_0 = \sigma^\infty \frac{W}{W - d},$$

we again define the stress concentration factor as $K_t = \sigma_{1,\max}/\sigma_0$. Submit a plot of K_t versus d/W for this geometry, when L is sufficiently large. Consider the five specific cases of $d/W = 0.1, 0.2, 0.3, 0.4, 0.5$.

Comments:

- Model the plate as being in a state of plane stress. To do so, in element type family, select "Plane Stress."
- In your calculations, you should ensure that you have chosen L to be large enough so as not to affect the result and that your mesh is sufficiently refined.
- Take advantage of any symmetries in the problem.