Exercise 2

Mesh Control: Spatial Resolution, Element Shapes, Basis Functions & Convergence Analyses

Goals
In this exercise, we will explore the strengths and weaknesses of different element types (tetrahedrons vs. hexahedrons, linear vs. quadratic basis functions) and the influence of the spatial mesh resolution on the computed results (displacements, strains, stresses) of an FEA.

A. Given Input Data
For our experiments, we will use the same geometry and boundary conditions as we did last time when modeling three point bending of a beam. Therefore, first create a model with the following properties:

![Cantilever Beam Diagram](image)

**Figure 1: Cantilever beam (half of beam under 3-point bending).**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F/2$</td>
<td>2,500 N</td>
</tr>
<tr>
<td>$L/2$</td>
<td>1,000 mm</td>
</tr>
<tr>
<td>$h$</td>
<td>60 mm</td>
</tr>
<tr>
<td>$t$</td>
<td>20 mm</td>
</tr>
<tr>
<td>$E$</td>
<td>210,000 N/mm²</td>
</tr>
<tr>
<td>$ν$</td>
<td>0.3</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Force at point B</td>
</tr>
<tr>
<td>Half length of the full 3PB beam</td>
</tr>
<tr>
<td>Height of the beam’s cross section</td>
</tr>
<tr>
<td>Thickness of the beam’s cross section</td>
</tr>
<tr>
<td>Young’s modulus</td>
</tr>
<tr>
<td>Poisson’s ration</td>
</tr>
</tbody>
</table>

**Table 1: Geometric and material data**
B. Meshing Options

ANSYS Workbench offers a number of possibilities to control the meshing procedure. To successfully complete the tasks described in section C you will need to know about the following configuration options (all accessible through the Mechanical module):

B.1 Linear vs. Quadratic (“Higher-Order”) Elements

To switch between linear and higher-order elements, switch the global mesh option Mesh → Advanced → Element Midside Nodes between Dropped (linear basis functions) and Kept (quadratic basis functions).

B.2 Selecting Element Shapes

To create a mesh of hexahedrons, add the Mapped FaceMeshing item to the Mesh node and select all 6 faces of the beam for the “scope” option. If you want to create a mesh consisting of tetrahedrons only, insert the Method item instead and select the Definition → Method → Tetrahedrons option in the details panel.

B.3 Controlling the Mesh Resolution

To be able to control the resolution of the mesh in each dimension, add a total of three Sizing items to the Mesh node. Then select the first Sizing item and pick the four beam edges in x direction as its scope and set Type to Number of Divisions (Note that this name is a bit misleading; “Number of Divisions” really refers to the number of elements along the selected axis, hence it really is number of divisions + 1!). Also set Behavior to Hard to override any automatic mesh sizing. Do the same for the remaining two Sizing items to control the resolution in y in z direction.

C. Tasks

You shall now investigate the influence of different element types and mesh resolutions on the predicted deflection and stresses for our well-known three point bending case.

C.1 Prerequisites

For this exercise, you should explicitly turn off so-called “reduced integration.” In order to do so, select Geometry in Mechanical’s model outline and set Element Control to Manual. Then select the beam part/solid and set Brick Integration Scheme to Full.

To be able to plot the displacements/stresses along the beam as a graph, add a “path”: In the model outline, right-click on Model (A4) and insert Construction Geometry. Right click on the new item and insert a Path. Set the path’s start and end coordinates, to (0 mm, 30 mm, 0 mm) and (1000 mm, 30 mm, 0 mm), respectively. Set the Number of Sampling Points to 200.
C.2 Meshing Experiments

Investigate the influence of different element types and mesh resolutions:

1. Mesh the beam geometry with a single, linear hexahedral element. Create contour plots of the total deformation, the normal stress in the x direction, the equivalent stress and the shear stress in the x-y-plane. Remember to set the Integration Point Results \(\rightarrow\) Display Option to Unaveraged for all contour plots! Add an additional Normal Stress result item (again, along the x axis), but set its Scoping Method to Path and select the previously defined path (see C.1). Evaluating this result item will create a graph of stress vs. beam length. Save the plots to PNG files using Windows’ Snipping Tool.

2. Increase the number of elements along the x axis to 5. Produce the same plots as before. How have the results changed with respect to the single-element mesh?

3. Switch to quadratic hexahedrons (same mesh resolution). How does this influence the results?

4. Further increase the mesh resolution to \(25 \times 2 \times 2\) elements (x, y, z direction). Compare the results for linear and quadratic basis functions.

5. Switch to linear tetrahedrons. What do you observe with respect to the predicted stresses?

6. What happens when you switch to higher-order tetrahedrons?

C.3 Convergence Analyses

To determine the required mesh resolution one typically performs a so-called convergence analysis: Increasing the FE mesh resolution step by step makes the results (deformation, strain, stress) converge towards the true, analytical solution. As soon as the results between two refinement steps don’t change by more than some arbitrarily chosen threshold (e.g. 1\%), we accept the mesh resolution as sufficient.

From C.2 we can conclude that not all element shapes and basis functions work equally well to describe beam bending: Linear elements seem to require higher mesh resolutions than quadratic elements, in particular linear tetrahedrons. You shall now investigate and compare the convergence rates of the different element types:

1. Draw convergence diagrams for linear and quadratic hexahedral meshes by plotting the computed maximum deformation and the maximum von Mises stress over the number of nodes (or DOFs). Note: Read the “Tips” section first!

2. Do the same for the linear and the quadratic tetrahedral element types!

3. Now also directly compare the linear to the quadratic element types (i.e. linear tetrahedron vs. linear hexahedron and quadratic tetrahedron vs. quadratic hexahedron)!

4. Why isn’t it a good an idea to simply use the global maximum von Mises stress as a convergence indicator?

5. Based on your findings, which element type would you choose? Why? What are the tradeoffs of one type over the other?
Tips:
- To retrieve the results like displacements or stresses at some fixed point, use **Probes**. First, define the location of the probe by setting up a new coordinate system by right-clicking **Coordinate Systems** and **Insert → Coordinate System**. Under **Origin**, enter the desired coordinates of the probe. Now define the probe itself by right-clicking on **Solution A6 in Mechanical's Outline** and **Insert → Probe → Deformation/Stress/Strain**. Switch to **Coordinate System** as the **Location Method** and choose your previously defined coordinate system for **Location**.
- You can either manually change the mesh resolution...
- ... or parameterize your model and create a set of **Design Points** (**Parameter Set → Table of Design Points**; see the description of the bonus task of lab 1). Unfortunately, the element shape or the choice of basis function cannot be parameterized.
- You can vary the mesh resolution by choosing a fixed y and z resolution and only change the number of elements along the x axis (cf. C.2)...
- ... or use **Mesh → Insert → Sizing** to enforce a certain average element edge length (set the scope of the sizing method to the whole solid beam body); this gives the mesher some freedom to maintain certain quality metrics (like the aspect ratio of the generated elements). Compared to the strict edge sizing option (like in C.2) however, body sizing makes it harder to control how many nodes the mesher will actually generate. On the other hand, the created meshes are typically of superior quality.

### C.4 Bonus Task: Computing the Exact Deflection

Let's assume we are only interested in the maximum deflection of the beam, but we need an exact value instead of an approximation. To solve this problem, we could either look up the analytical formula in Wikipedia or again use ANSYS, but switch to a specialized element type for beam bending problems.

#### C.4.1 Defining the Material Properties

Create a new **Static Structural** modeling system and add a material model for the beam (**Engineering Data** cell).

#### C.4.2 Modeling the Geometry

Start the **DesignModeler (Geometry cell)**. Add a new **Sketch** to the **XYPlane** (Figure 2). Right-click on the newly generated **Sketch1** item in the **Tree Outline** and select **Look At**. Activate the **Sketching** tab and select the **Line** tool. Draw a horizontal line starting at the origin along the x axis (Figure 3). The length of the line does not matter for now.

Now switch to the **Dimensions** sub-toolbox and select the **Horizontal** tool. Then click first on the left and then on the right corner point of the line. Enter the length of the (half-)beam into the **Details View** of the horizontal dimension (Figure 4).
Figure 2: Adding a sketch to the XYPlane

Figure 3: Sketching a line
Select the **Modeling** tab again and insert a new line body (**Main Menu → Concept → Lines From Sketches**). Choose the line as the **Base Object** and click the **Generate** button. This should create a new part, containing a single line body. Next, we need to define the cross-section of the beam. **Choose Concept → Cross Section → Rectangular** from the main menu and set the **B** and **H** dimensions of the cross section to 60 mm and 20 mm, respectively. Assign the newly defined cross section to the line body.

Close the **DesignModeler** and return to the **Project Schematic**.

**C.4.3 Meshing**

Start the **Mechanical** module (**Model cell**). First, make sure that the **Model Type** for the line body is set to **Beam** (**Outline → Model (A4) → Geometry → Line Body → Details of “Line Body”**). In the outline, right click on the **Mesh** node and choose **Generate Mesh**. Again, you can control the resolution of the mesh by adding appropriate **Sizing** items to the **Mesh** node.

**C.4.4 Load and Boundary Conditions**

Right-click on the Static Structural node and insert a fixed support (left corner point). Apply a force of 2500 N in positive y direction to the right corner point.
C.4.5 Solving & Post-Processing
Insert the Total Deformation solution item (under Static Structural → Solution (A6)). You can also add beam-specific result items by choosing some of the Beam Results items.

C.4.6 Questions
Using the beam element type, how many elements/nodes/DOFs are necessary to compute the exact result value for the maximum beam deflection (modulo rounding errors and the limits of machine precision)? Why?

Figure 5: Computed deflection using beam elements