EXERCISE 4: MESH CONTROL, ELEMENT SHAPES AND CONVERSION ANALYSIS

CONTENTS

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.

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 3D model with the following properties:
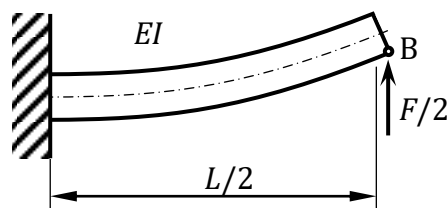

Figure 1: Cantilever beam (half of beam under 3-point bending).

$F/2$	=	2,500 N	Force at point B
$L/2$	=	1,000 mm	Half length of the full 3PB beam
h	=	60 mm	Height of the beam's cross section
t	=	20 mm	Thickness of the beam's cross section
E	=	210,000 N/mm²	Young's modulus
v	=	0.3	Poisson's ratio

Table 1: Geometric and material data

A.   Introduction to 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):

A.1   Linear vs. Quadratic ("Higher-Order") Elements

To switch between linear and higher-order elements, have **Mesh** selected, and, in the **Detail View → Advanced → Element Midside Nodes** switch between **Dropped** (linear basis functions) and **Kept** (quadratic basis functions).

A.2   Selecting Element Shapes

To create a mesh of hexahedrons, add the **Mapped Face Meshing** 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.

A.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.

B.   Setting up the Model

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.   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 *x* direction, the equivalent stress and the shear stress in the *x-y*-plane. Remember to set the **Integration Point Results → 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 B). 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 × 2 × 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?

D.   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. This procedure can be largely automated by utilizing Workbench's parameterization function.

From C 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. We can now investigate and compare the convergence rates of the different element types.

D.1   Parameterizing in ANSYS

Our goal is to 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).

First, we want 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). Delete your three X, Y,Z-sizings, and instead select **Mesh → Insert → Sizing.** As Geometry select **Body,** and as Type choose **Element Size.** Compared to the strict edge sizing option (like in C) 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

To parameterize our model, we must first select an input parameter. To do this, click on the box to the left of **Element Size**; you should see a blue 'P' in the box, and notice that a new Parameter option has shown up in the **Project View**. This will also lock you out of manually changing Element Size.
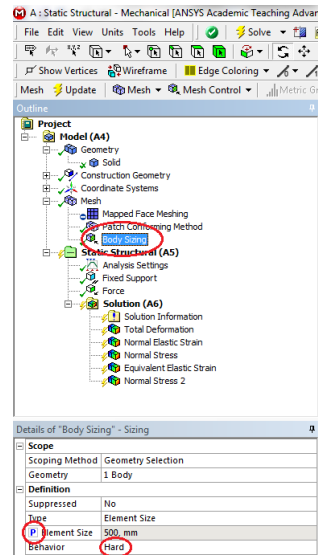

FIGURE 2 SETTING INPUT PARAMETER

To express our objective, we also need an output parameter. Retrieving the results like displacements or stresses at some *fixed point*, however, requires **Probes**. First, define the location of the probe by setting up a new coordinate system by right-clicking **Coordinate Systems** in the Outline and **Insert → Coordinate System**. Under **Origin**, enter the desired coordinates of the probe (consider *where* placement for the probes would make sense). 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**.

Create a Probe for Stress and Deformation. Set as output parameter the **Equivalent (von Mises Stress)** and **Total Deformation**. Further set as output parameter the number of elements (**Mesh -> Statistics -> Elements**) and number of nodes.
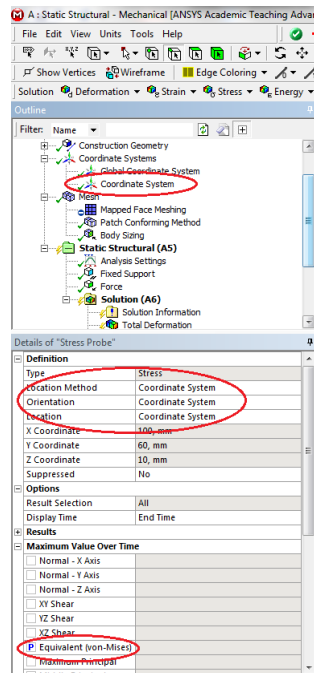
FIGURE 3 SETTING OUTPUT PARAMETER

For this first test, make sure that your Mesh **Method** is set to **Automatic** (i.e. ANSYS will use hexahedrons). Now switch to the **Design View**, and double click on **Parameter**.

Here we can create a set of **Design Points**. Enter a reasonable progression of Element Sizings in the Design Points Table. Unfortunately, the element shape or the choice of basis function cannot be parameterized. Click **Update All Design Points** in the toolbar at the top, and Workbench will run a simulation for each of your design points.

FIGURE 4 CREATING A TABLE OF DESIGN POINTS

D.2   Convergence Analyses

Finally you can create a Chart with your results. Double click on **Parameters Chart**, and create an appropriate plot (i.e. Deformation and Stress over Number of Nodes). Notice how the overall mesh quality affects your solution.

D.3   Further Tasks

1. Perform a convergence analysis for quadratic hexahedrals.
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 terahedron 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?

Hint: You can save a created chart by right clicking on the picture and choosing **Save Image As...**

D.3   Further Tasks