## **EXERCISE 3: MESH CONTROL, ELEMENT SHAPES AND CONVERSION ANALYSIS**

# CONTENTS

Goals		2		
Input Da	1ta	2		
A. Inti	roduction to Meshing Options	3		
A.1	Linear vs. Quadratic ("Higher-Order") Elements	3		
A.2	Selecting Element Shapes	3		
A.3	Controlling the Mesh Resolution			
B. Set	ting up the Model	3		
C. Me	shing Experiments	4		
D. Cor	vergence Analyses	4		
D.1	Parameterizing in ANSYS	4		
D.2	Convergence Analyses	6		
D.3	Further Tasks	7		

#### 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
$E = 210,000 \text{ N/mm}^2$	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**  $\rightarrow$  **Advanced**  $\rightarrow$  **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**  $\rightarrow$  **Method**  $\rightarrow$  **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**  $\rightarrow$  **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 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  $25 \times 2 \times 2$  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?

#### **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**  $\rightarrow$  **Insert**  $\rightarrow$  **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.

File Edit View	Units Tools Help 🛛 🧭 💈 Solve 👻 📷
• · · • •	. «
F Show Vertices	Wireframe   III Edge Coloring • 76 •
Mesh 🔰 Update	Metric 🐨 Mesh 🔻 🔍 Mesh Control 👻 🔐 Metric
Outline	
Project	
🗄 💮 🔯 Model (A	4)
🗄 – 🖓 Geon	ietry
¥®	Solid
E Cons	truction Geometry
E Coor	dnate Systems
e ven mes	Manned Face Meshing
	Patch Configuring Method
6	Body Sizing
ં վ 🔁	ic Structural (A5)
T-√∆	Analysis Settings
<b>P</b>	Fixed Support
~~~ <u>~</u> ~	Force
e	Solution (A6)
	Solution Information
	V I I I I I I I I I I I I I I I I I I I
	Normal Stress
	Equivalent Elastic Strain
	Normal Stress 2
Details of "Rody Sizi	na" - Siting
Scone	ing stang
Scope Coming Mathed	Commeter Selection
Scoping Method	1. De de
Definition	1 DOUY
Cennuon	M-
the second second second	NU
Suppressed	Class and Class
Suppressed	Element Size
Type Plement Size	Element Size 500, mm

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**  $\rightarrow$  **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**  $\rightarrow$  **Probe**  $\rightarrow$  **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.

File Edit View Uni	ts Tools Help	
🚏 🕂 🦌 🐨	💱 🖪 💽 🖪 🦃 -	S R
🖵 Show Vertices  🕌	🖓 Wireframe 🛛 📕 Edge Coloring 👻	6-
olution 🧐 Deforma	tion 👻 🔍 Strain 👻 🍕 Stress 👻 🔍	Energy
utline		
sume		
ilter: Name 🝷	🖉 < 🗄	
E Construc	tion Geometry	
E Coordina	te Systems hal Ceerfinate System	
Con Con	ordinate System	
Ast Mesn		
🛛 🔂 Maj	pped Face Meshing	
🖓 Pat	ch Conforming Method	
Bot	ly Sizing	
E generations	tructural (AS)	
Eix	ed Sunnort	
S. For	ce	
🗄 🖉 🙆 So	lution (A6)	
· · · · · · · · · · · · · · · · · · ·	Solution Information	
	Total Deformation	
tails of "Stress Probe	Total Deformation	
tails of "Stress Probe Definition	Total Deformation	
tails of "Stress Probe Definition Type	Stress	
tails of "Stress Probe" Definition Type tocation Method	Stress     Coordinate System	
tails of "Stress Probe" Definition Type Totation Method Orientation	Total Deformation     Stress     Coordinate System     Coordinate System	
tails of "Stress Probe" Definition Type Cocation Method Orientation	Total Deformation Stress Coordinate System Coordinate System Coordinate System Coordinate System	
tails of "Stress Probe Definition Type tocation Method Orientation Tocation X Coordinate	Stress Coordinate System Coordinate System Coordinate System 100, mm	
tails of "Stress Probe Definition Type Vocation Method Orientation Techtion X Coordinate Y Coordinate	Total Deformation Stress Coordinate System Coordinate System 100, mm 60, mm	
Coordinate Z Coordinate	7 Total Deformation 5 5 5 5 5 5 5 5 5 5 5 5 5	
Coordinate Suppressed	Stress Stress Coordinate System Coordinate System Coordinate System 100, mm 10, mm 10, mm	
Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate	Stress Stress Coordinate System Coordinate Syst	
tails of "Stress Probe Definition Type Cotation Method Orientation Protion X Coordinate Z Coordinate Z Coordinate Z Coordinate Suppressed Options Result Selection	Total Deformation  Terristic Coordinate System Coordinate System Coordinate System Goldmane System Goldmane Ino All	
Hand Stress Probe Definition Type totation Method Orientation Deation X Coordinate Z Coordinate Suppressed Options Result Selection Display Time	Stress Stress Stress Stress Coordinate System C	
Hand Stress Probe Definition Type Sociation Method Orientation V Coordinate X Coordinate X Coordinate X Coordinate X Coordinate X Coordinate Suppressed Options Result Selection Display Time Results	Stress       Coordinate System       Coordinate System       Coordinate System       Coordinate System       100, mm       100, mm       10, mm	
Latalis of "Stress Probe" Definition Type Costation Method Orientation Deation X Coordinate X Coordinate X Coordinate Suppressed Options Result Selection Display Time Results Maximum Value Over	Stress Coordinate System Coor	
Lease of "Stress Probe" Definition Type Kraction Method Orientation Creation X Coordinate X Coordinate X Coordinate X Coordinate X Coordinate S Coordinate Options Result Selection Dipplay Time Results Maximum Value Ove Naximum Value Ove Naxi	Stress Stress Coordinate System Coordinate Syst	
Latalis of "Stress Probe Definition Type Stocation Method Orientation N Coordinate X Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Suppressed Options Result Selection Dipløy Time Results Maximum Value Ove Normal - X Axis	Stress Coordinate System Coor	
Latals of Stress Probe Oriention Type Coation Method Orientation Decesion X Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Suppressed Options Result Selection Display Time Results Maximum Value Ove Normal - X Axis Normal - X Axis	Trail Deformation  Stress  Stress Stress  Stress  Stress Stress  Stress  Stress  Stress  Stress  Stress  Stress  Stress  Stress  Stress  Stress  Stress  Stress  Stres	
Latals of Stress Probe Definition Type Costion Method Orientation X Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Maximum Value Ove Result Selection Display Time Result Maximum Value Ove Normal - X Axis Normal - X Axis Normal - X Axis Normal - X Axis	Stress Coordinate System Coor	
Latals of Stress of Definition     Type     Coordinate     Normal - X Axis     Normal - X X Axis     Normal - X X Axis     Normal - X X X Shear     Normal - X X Shear     Norm	Total Deformation  Stress Stre	
Latals of Stress Probe Definition Type Moniton Method Orientation Design of Control V Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Z Coordinate Maximum Value Ove Normal - X Axis Normal - X Axis Normal - X Axis X Shear Z Shear	Stress Coordinate System Coor	

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.

Outline: No data 🔷 🗸 🦊					▼ ₽ Х	Table of Design Points				<b>▼</b> ₽
	А	В	с	D			А	В	c	
1	ID	Parameter Name	Value	Unit		1	Name 💌	P1 - Body Sizing Element Size 💌	P2 - Stress Probe Maximum Equivalent (von-Mises) 💌	P3 - Deformat
2	<ul> <li>Input Parameters</li> </ul>					2	Units	mm 💌	MPa	
3	🖃 🚾 Static Structural (A1)					3	Current	500	27,328	0,40632
4	ι <mark>ρ</mark> Ρ1	Body Sizing Element Size	500	mm 💌		4	DP 1	400	38,198	0,8972
•	New input parameter	New name	New expression			•				
6	<ul> <li>Output Parameters</li> </ul>									
7	🖃 🚧 Static Structural (A1)									
8 pJ P2		Stress Probe Maximum Equivalent (von-Mises)	27,328	MPa						
9	p⊋ P3	Deformation Probe Maximum Y Axis	0,40632	mm						
10	P4	Mesh Elements	2							
11	P5	Mesh Nodes	12							
*	New output parameter		New expression							

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. Performa 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...