

# Exercise 2: Mesh 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 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 (you may also reuse your beam model from last time or simply load the provided template):

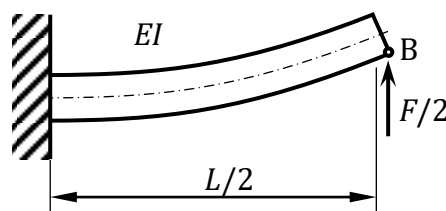


Figure 1: Cantilever beam (half of beam under 3-point bending).

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

Table 1: Geometric and material data

## B. Tasks

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

### B.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** (Figure 2).

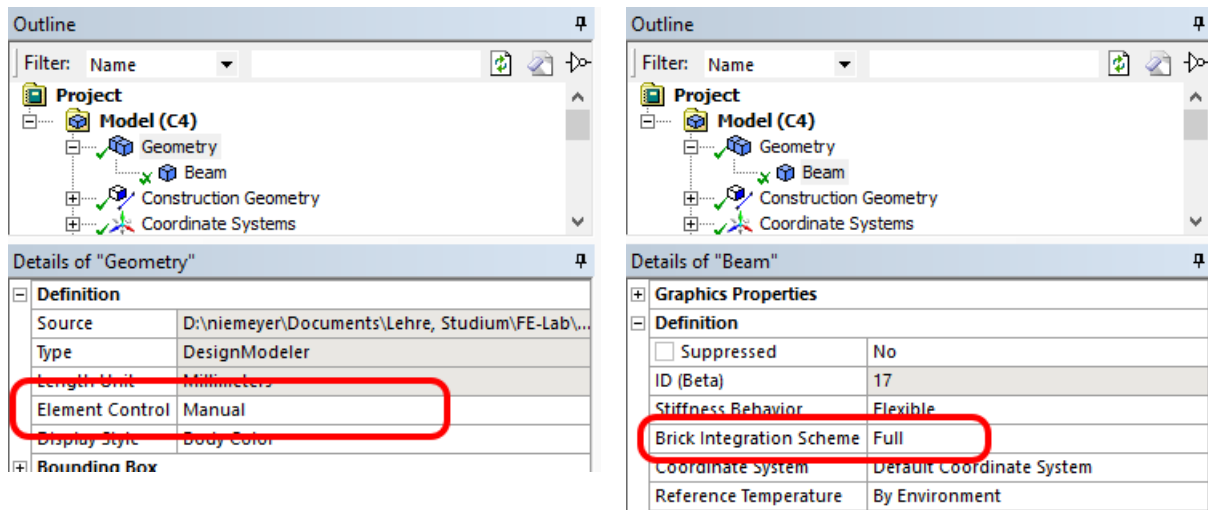
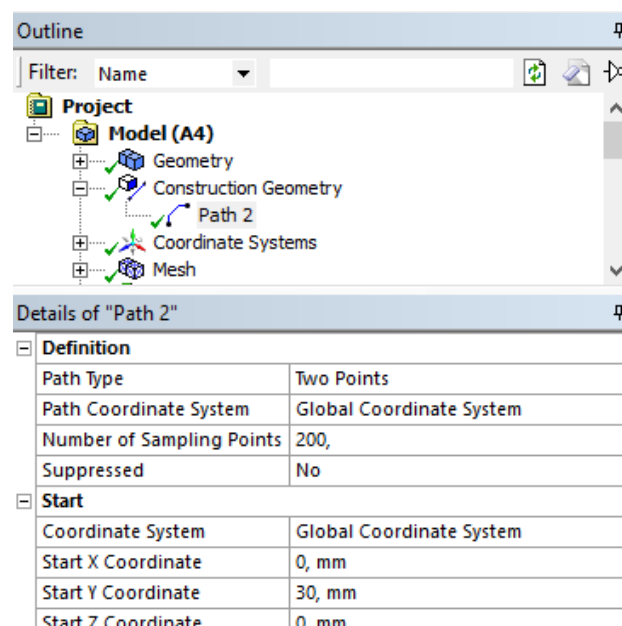


Figure 2: Choosing "full integration" for linear hexahedra

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.



## B.2 Meshing Experiments

Investigate the influence of different element types and mesh resolutions:

1. Mesh the beam geometry with a *single*, linear hexahedral element (**read the section “C. Meshing Options” first if you don’t know how to achieve this!**). 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 option **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.1). Evaluating this result item will create a graph of stress vs. beam length.
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?

## B.3 Convergence Analysis

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. Because we typically don’t know the true solution, though, we accept the mesh resolution as sufficient as soon as the results between two refinement steps don’t change by more than some arbitrarily chosen threshold (e. g. 1 %).

From B.2 we can conclude that not all element shapes and basis functions work equally well to describe beam bending. You shall now investigate and compare the convergence rates of the different element types more closely:

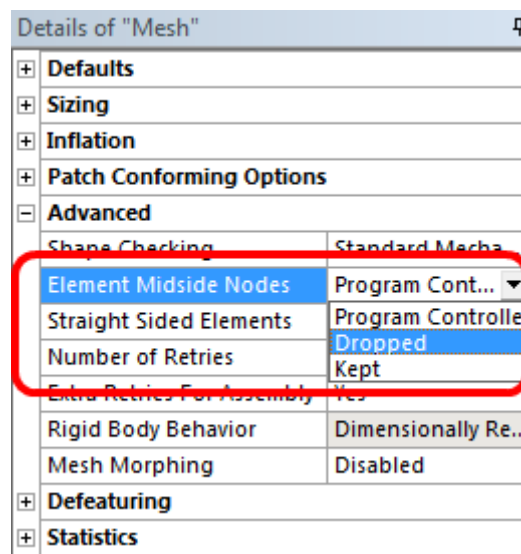
1. Draw convergence diagrams for linear and quadratic hexahedral meshes by plotting the computed maximum deformation over the number of nodes (or DOFs) for increasing mesh resolutions. **Note: Read the “C. Meshing Options” section first!**
2. Do the same for the linear and the quadratic tetrahedral element types!
3. Why isn’t it a good an idea to simply use the global maximum von Mises stress as a convergence indicator?
4. Based on your findings, which element type would you choose? Why? What are the tradeoffs of one type over the other?

## C. Meshing Options

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

### C.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).



### C.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 (cf. Exercise 1). If you want to create a mesh consisting of tetrahedrons, insert the **Method** item instead and select the **Definition** → **Method** → **Tetrahedrons** option in the details panel.

### C.3 Controlling the Mesh Resolution

You can vary the mesh resolution by controlling the resolution for the x y and z axis separately (useful for task B.2 when you only want to change the resolution along one axis). 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.

For the convergence analysis, it may be convenient to use a global **body sizing** option: 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) and you only need to control a single parameter. Compared to the strict edge sizing option 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 and the results converge more quickly to the true solution.

#### C.4 Determining the Number of Nodes

After successfully generating a mesh, you can determine the number of nodes by looking at the **Statistics** section the **Details** pane of the **Mesh** node.

Details of "Mesh"		
+	Defaults	
+	Sizing	
+	Inflation	
+	Patch Conforming Options	
+	Advanced	
+	Defeaturing	
-	Statistics	
<input type="checkbox"/>	Nodes	549
<input type="checkbox"/>	Elements	80
	Mesh Metric	None

#### C.5 Parameterization

For the convergence analysis (task B.3), instead of manually changing the mesh resolution parameterize your model and create a set of **Design Points (Parameter Set → Table of Design Points**; see also the description of the bonus task of lab 1). Unfortunately, the element shape or the choice of basis function cannot be parameterized.

Details of "Edge Sizing X" - Sizing		
-	Scope	
	Scoping Method	Geometry Selection
	Geometry	4 Edges
-	Definition	
	Suppressed	No
	Type	Number of Divisions
	<input checked="" type="checkbox"/> Number of Divisions	1
	Behavior	Hard
	Bias Type	No Bias

Click the empty box

Details of "Total Deformation"		
-	Scope	
	Scoping Method	Geometry Selection
	Geometry	All Bodies
-	Definition	
	Type	Total Deformation
	By	Time
	Display Time	Last
	Calculate Time History	Yes
	Identifier	
	Suppressed	No
-	Results	
	<input type="checkbox"/> Minimum	0, mm
	<input checked="" type="checkbox"/> Maximum	10,819 mm
+	Information	

Figure 3: Defining input (left) and output parameters (right)

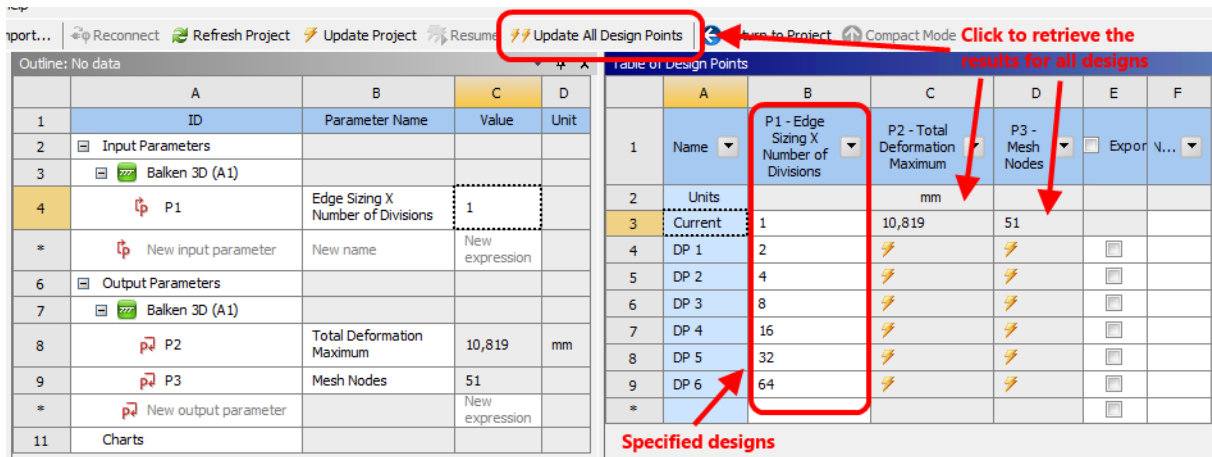


Figure 4: After defining parameters the box **Parameter Set** appears in Workbench’s project view. Double click this box to open the table of Design Points (right). Create new “designs” (= combinations of parameter values) by entering the desired parameter values in the corresponding columns. Clicking **Update All Design Points** automatically runs all required simulations.

#### D. 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.

With this special beam element type, it is possible to describe beam bending problem using a 1-dimensional model. Once again start a new Workbench project. Define material properties just as you did before and start the **DesignModeler**. Create a **Line Segment** directly from two **Points**: From the main menu, choose **Create** → **Point** and in the detail view select **Manual Input** for the **Definition** option (Figure 5). You can now enter arbitrary coordinates for the point. Clicking the **Generate** button will create the point. Place the first point at the origin and place another one at (1000, 0, 0).

Next, choose **Concept** → **Line From Points** from the main menu and select both your points (you can select multiple entities by holding down Ctrl) and press **Generate** once more. This adds a new line body in the **Tree Outline**; however, the body is missing a cross-section definition. Remedy this by choosing **Concept** → **Cross Section** → **Rectangular** from the main menu and enter the appropriate values. Assign the newly created cross section to your line body (detail view).

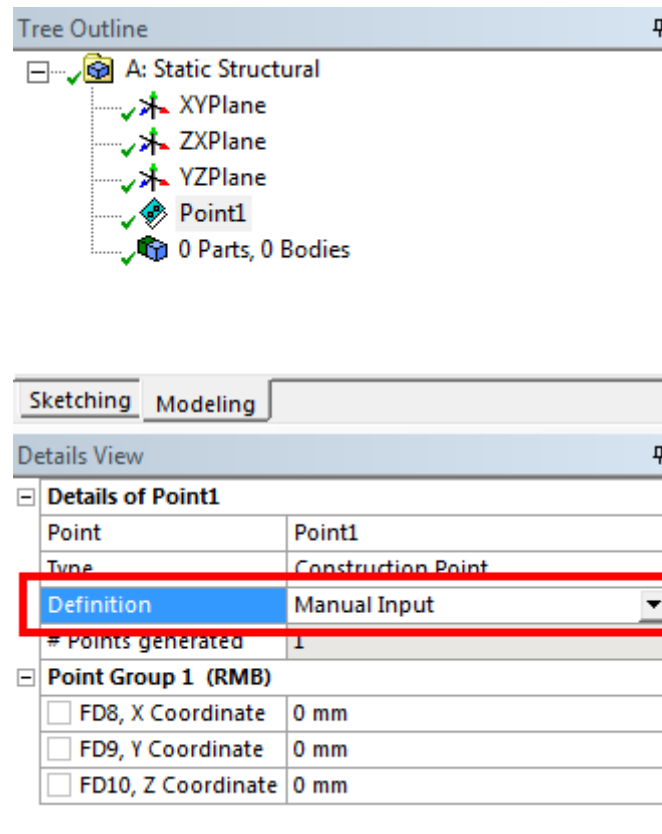


Figure 5: Creating a point at arbitrary coordinates

Complete your analysis analogous to the previous 3D FEAs and compare the simulation results.

Chances are your simulation result will be wrong; the reason is simple, though not obvious: You may have noticed that, once we generated our line body, there is a strange vector pointing out from it. This vector is used to determine the alignment of the cross-section. Open the **DesignModeler** again, right click on **Line Body** and choose **Select Unaligned Line Edges**. Choose **Vector** in the **Alignment Mode** section, and enter a vector such that the Y-Axis alignment of your cross-section corresponds with the direction of your vector. That is, if you entered a height  $H_1$  of 60mm for your cross section, you must define a vector of  $(0, 1, 0)$  in the **Alignment Mode**.

You can verify whether your alignment was correct by entering the **Mechanical** module and choosing **View** → **Cross Section Solids**. Your answer should now coincide with our previous simulations.

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?