

ENGN 1750: Advanced Mechanics of Solids

Modeling beams in Abaqus

1 Introduction

In the first Abaqus assignment, we were introduced to building models in Abaqus in the context of trusses, and we saw the basic ingredients of a static finite element simulation, namely

1. Geometry,
2. Material behavior,
3. Loads and boundary conditions, and
4. Mesh.

These ingredients will continue to be integral to our subsequent exercises using Abaqus.

Trusses are the simplest of structural elements, only supporting constant axial forces (no bending or twisting), and as such, the displacements along the length of the members are linear. As we will see later in the course, finite elements assume the displacement field to have some functional form. Truss elements assume linear displacements and therefore yield exact solutions, which we observed in the first assignment.

In this assignment, we will examine another familiar structural element: Euler-Bernoulli beams. Our objective is an introduction to the modeling of this important structural element using finite elements in Abaqus. In contrast to truss members, which only carry force along their length, beams are capable of carrying shear forces and bending moments. The governing equation for the lateral displacement of a beam δ along its length x is

$$EI \frac{d^4 \delta}{dx^4} = q,$$

where q is a distributed load per unit length of the beam, E is the Young's modulus, and I is the second moment of area of the beam's cross section

When $q = 0$, the governing equation is solved exactly when δ is a cubic polynomial in x . Therefore, Abaqus Euler-Bernoulli beam elements assume that the lateral displacement field δ is interpolated by a cubic spline, and the calculated displacements and stresses are exact. Even in the case of a distributed load, Abaqus interprets the load in such a way that the calculated displacements are exact (although the calculated stresses are no longer exact).

In this exercise, we will

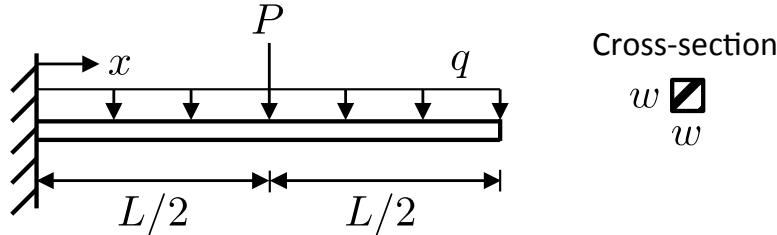
1. Learn to set up a beam analysis in Abaqus.
2. Learn to apply midpoint and distributed loads.
3. Examine the effect of mesh resolution.

Reminder on units: Before we begin, let us recall that Abaqus has no built-in set of units. Be sure that you input all numerical quantities with respect to a consistent set of units.

2 In-class exercises

2.1 Cantilever beam

Consider the following cantilever beam, subjected to a load P at its midpoint as well as a distributed load q :



The beam has a length of $L = 1$ m and a square cross-section with $w = 0.05$ m and is made of steel, so that $E = 210$ GPa and $\nu = 0.3$. A downward load of $P = 10$ kN is applied at the midpoint, and a distributed load of $q = 10$ kN/m is applied along its length. The downward displacement field due to the midpoint load is

$$\delta_{\text{mid}}(x) = \begin{cases} \frac{Px^2}{12EI}(3L - 2x) & 0 \leq x < L/2, \\ \frac{PL^2}{48EI}(6x - L) & L/2 \leq x \leq L, \end{cases}$$

and the downward displacement field due to the distributed load is

$$\delta_{\text{dist}}(x) = \frac{qx^2}{24EI}(6L^2 - 4Lx + x^2),$$

so that, by superposition, the total downward displacement field is

$$\delta(x) = \delta_{\text{mid}}(x) + \delta_{\text{dist}}(x) = \begin{cases} \frac{Px^2}{12EI}(3L - 2x) + \frac{qx^2}{24EI}(6L^2 - 4Lx + x^2) & 0 \leq x < L/2, \\ \frac{PL^2}{48EI}(6x - L) + \frac{qx^2}{24EI}(6L^2 - 4Lx + x^2) & L/2 \leq x \leq L, \end{cases}$$

and the tip displacement is

$$\delta_{\text{tip}} = \delta(x = L) = \frac{5PL^3}{48EI} + \frac{qL^4}{8EI}.$$

Below is an outline of the steps for performing the analysis in Abaqus/CAE:

- Part:
 - Part \Rightarrow Create
 - Select 2D Planar, Deformable, Wire, and Approximate size: 2 \Rightarrow Continue

- Sketch the part as pictured and click Done
- In order to later apply the load at its midpoint, we need to partition the beam. Select the Partition Edge: Select Midpoint/Datum Point button from the menu on the left.
- Select the midpoint of the beam and click Create Partition.
- Property:
 - Material \Rightarrow Create
 - Mechanical \Rightarrow Elasticity \Rightarrow Elastic
 - Enter the material properties for steel and click OK
 - Profile \Rightarrow Create
 - Rectangular \Rightarrow Continue
 - Enter the cross-sectional dimensions and click OK
 - Section \Rightarrow Create
 - Beam \Rightarrow Beam \Rightarrow Continue
 - Make sure your material and profile are selected and click OK
 - Assign \Rightarrow Section
 - Select the entire part and click Done/OK.
 - Assign \Rightarrow Beam section orientation
 - Select the entire part and click Done.
 - Accept the default orientation (in 2D, it is your only option) and click 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 left end and click Done \Rightarrow Enter $U1=U2=UR3=0$ and click OK
 - Load \Rightarrow Create \Rightarrow Mechanical \Rightarrow Concentrated Force \Rightarrow Continue \Rightarrow Select midpoint and click Done \Rightarrow Enter CF2 and click OK
 - Load \Rightarrow Create \Rightarrow Mechanical \Rightarrow Line load \Rightarrow Select the whole beam (be sure to select both sides of the partition) and click Done \Rightarrow Enter Component 2 and click OK

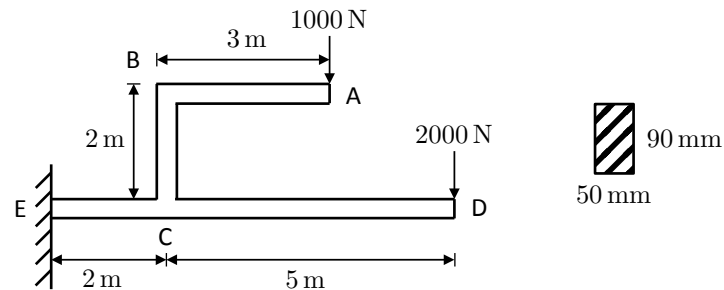
- Mesh:
 - Make sure Object is set to Part.
 - Mesh \Rightarrow Element Type \Rightarrow Select the entire part and click Done \Rightarrow Family: Beam \Rightarrow Select Beam type: Cubic formulation (This is the Euler-Bernoulli beam element.) \Rightarrow Click OK
 - Seed \Rightarrow Edges \Rightarrow Select the entire part and click Done \Rightarrow Method: By number \Rightarrow Number of elements: 1 \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
 - Verify that the calculated tip displacement matches its analytical counterpart.

To calculate displacements along the length of the beam and to better resolve the stress, we need to use more elements.

- Mesh:
 - Seed \Rightarrow Edges \Rightarrow Select entire part and click Done \Rightarrow Method: By number \Rightarrow Number of elements: 10 \Rightarrow Click OK
 - Mesh \Rightarrow Part \Rightarrow Yes.
- Job:
 - Job \Rightarrow Submit \Rightarrow Job-1
 - When the job successfully completes: Job \Rightarrow Results \Rightarrow Job-1
- Visualization:
 - Probe displacements at different nodes and verify that they correspond to the analytical expression.
 - Examine the stress field field by probing stresses in the elements.

2.2 Branched Beam

Next consider a beam with a more complex geometry, loaded as shown below:



The beam is made of steel. Below is an outline of the steps for performing the analysis in Abaqus/CAE:

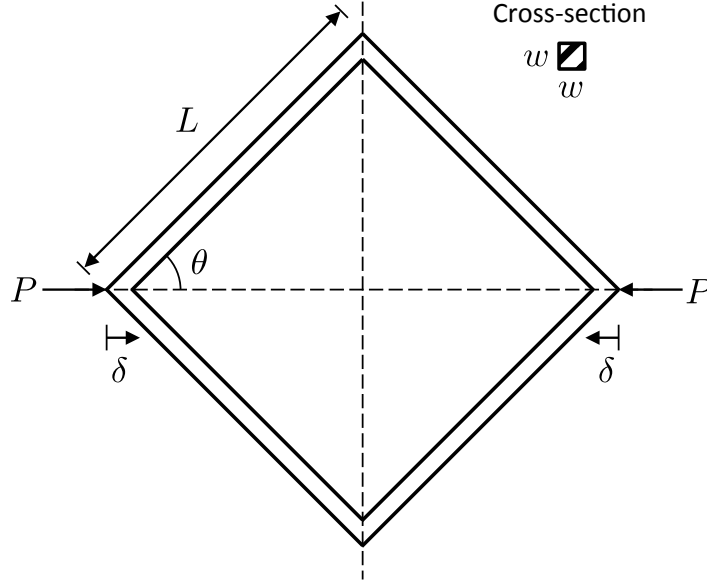
- Part:
 - Part \Rightarrow Create
 - Select 2D Planar, Deformable, Wire, and Approximate size: 20 \Rightarrow Continue
 - Sketch the part as pictured and click Done
- Property:
 - Material \Rightarrow Create
 - Mechanical \Rightarrow Elasticity \Rightarrow Elastic
 - Enter the material properties for steel and click OK
 - Profile \Rightarrow Create
 - Rectangular \Rightarrow Continue
 - Enter the cross-sectional dimensions and click OK
 - Section \Rightarrow Create
 - Beam \Rightarrow Beam \Rightarrow Continue
 - Make sure your material and profile are selected and click OK
 - Assign \Rightarrow Section
 - Select the entire part and click Done/OK.
 - Assign \Rightarrow Beam section orientation
 - Select the entire part and click Done.
 - Accept the default orientation (in 2D, it is your only option) and click 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 point E and click Done \Rightarrow Enter $U1=U2=UR3=0$ and click OK
 - Load \Rightarrow Create \Rightarrow Mechanical \Rightarrow Concentrated Force \Rightarrow Continue \Rightarrow Select point A and click Done \Rightarrow Enter CF2 and click OK
 - Load \Rightarrow Create \Rightarrow Mechanical \Rightarrow Concentrated Force \Rightarrow Continue \Rightarrow Select point D and click Done \Rightarrow Enter CF2 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: Beam \Rightarrow Select Beam type: Cubic formulation \Rightarrow Click OK
 - Seed \Rightarrow Edges \Rightarrow Select entire part and click Done \Rightarrow Method: By size \Rightarrow Approximate element size: 0.5 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

ENGN 1750: Advanced Mechanics of Solids
Abaqus Assignment 2

Due: Monday, October 13, 2014 OR Thursday, October 16, 2014 (in class)

1. Consider the plane frame consisting of four beams, pictured below.



The beams have a length of $L = 1$ m and a square cross-section with $w = 5$ cm. They are made of steel, so that $E = 210$ GPa and $\nu = 0.3$. The joints between beams are rigid, i.e., initial angles are assumed to be preserved after bending. Be sure to take advantage of the symmetry of the problem in your modeling.

- (a) Determine the stiffness of the frame loaded horizontally as shown in the figure as a function of the angle θ . The load P and displacement δ may be non-dimensionalized as PL^2/EI and δ/L , respectively. The dimensionless stiffness is then

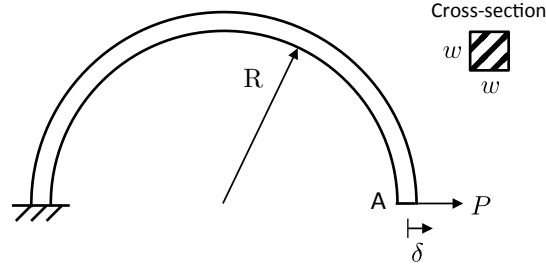
$$k = \frac{PL^2/EI}{\delta/L} = \frac{PL^3}{EI\delta}.$$

Submit a plot of k versus θ for $15^\circ \leq \theta \leq 75^\circ$, using at least seven discrete values of θ .

- (b) The maximum stress in your simulation is dependent on the number of elements. For $\theta = 45^\circ$, submit a plot of the maximum stress attained in your simulation as a function of the number of elements N per beam for $1 \leq N \leq 20$. Plot the maximum stress in dimensionless form as $\sigma_{\max} w^3/PL$.
- (c) Choose a value of N , for which you are satisfied that the simulated maximum stress is close enough to the actual value. Submit a plot of the maximum stress

in the frame as a function of θ . Prepare your plot in dimensionless form, i.e. $\sigma_{\max} w^3 / PL$ versus θ .

2. Consider the curved beam in the form of a semicircular hoop show below:



The curved beam has a radius of $R = 1$ m and a square cross-section with $w = 5$ cm. The beam is made of steel with $E = 210$ GPa and $\nu = 0.3$. We refer to the *horizontal* displacement of the point A is denoted as δ . Again, we define a dimensionless stiffness based on a non-dimensionalized load PR^2/EI and displacement δ/R :

$$k = \frac{PR^2/EI}{\delta/R} = \frac{PR^3}{EI\delta}.$$

The calculated stiffness of the curved bar depends upon the number of elements used. Submit a plot of the dimensionless stiffness k as a function of the number of elements N in the curved beam for $2 \leq N \leq 16$.