

ENGN 1750: Advanced Mechanics of Solids
Introduction to Abaqus: Trusses and beams

1 Introduction

A major goal of this course is to gain a proficiency with the use of the finite-element method (FEM) in solid mechanics. Broadly-speaking, the finite-element method is simply a numerical technique for solving partial differential equations. The governing equations of solid mechanics lend themselves particularly well to this technique, and by the end of the 1970s, enabled by the advent of digital computing, the application of FEM to solid mechanics had revolutionized the field. Prior to this development, the field was primarily the domain of advanced specialists and mathematicians, since, as we shall see later in the course, by-hand, analytical solutions quickly become quite complex in all but the simplest of geometries. The use of FEM has placed the full power of solid mechanics in the hands of a broad community of engineers for use in everyday design.

It will take the majority of the semester to cover the mathematical foundations of solid mechanics and the theory behind the finite-element method in lecture. Therefore, for the sake of efficiency, we will undertake in parallel a series of exercises to develop a familiarity with using a commercial FEM program. By the end of the semester, you should have a fluency with the program as well as a knowledge of what is going on “under the hood.” In this course, we will be using the FEM program Abaqus. Abaqus is one of the major FEM codes used in industry and research (others you might encounter are ANSYS, LS-DYNA, Nastran, COMSOL, or ADINA) and was developed by Brown PhD graduates David Hibbett and Paul Sorensen along with Bengt Karlsson in the 1970s. It has since been acquired by the French company Dassault Systemès, but Abaqus headquarters remains in Rhode Island to this day.

2 Using the finite-element method

Performing a finite-element analysis consists of three stages: (1) Pre-processing, (2) Processing, and (3) Post-processing, each discussed below:

- (1) Pre-processing: In this stage, one sets up a model prior to running the calculation. The basic ingredients of a finite-element simulation are
 - (i) Geometry: What is the shape of the part under consideration? If there are multiple parts, how are they oriented relative to one another, and how do they interact?
 - (ii) Material behavior: What is the part made of, and what are the material properties?
 - (iii) Loads and boundary conditions: How is the part held, constrained, and/or loaded?
 - (iv) Mesh: The finite-element method relies on discretizing the part(s) into many smaller elements, called the mesh.

Setting up a model and troubleshooting errors is a skill, requiring practice. We will utilize the Abaqus graphical environment, called CAE, or “Complete Abaqus Environment,” for preprocessing.

- (2) Processing: The calculation is actually performed in this stage. For the user of the program, it is a “black box” operation. When a job is submitted, Abaqus will run it and inform the user whether the analysis completed successfully or not. We will learn the theory behind this step later in the course.
- (3) Post-processing: In this stage, the calculation results are displayed. Typical simulation results for the types of analyses we will be running are displacement, strain, and stress fields. It is crucial to always cast a skeptical eye toward your simulation results and ask yourself whether they make sense or not. Never blindly trust the simulation output.

When progressing through these stages, one navigates Abaqus/CAE through the use of the module menu, which is (typically) followed sequentially. The modules, in order, are (1) Part, (2) Property, (3) Assembly, (4) Step, (5) Interaction, (6) Load, (7) Mesh, (8) Optimization, (9) Job, and (10) Visualization. (The sketch module is supporting, and we won’t be making use of it in our exercises.) The basic use of each module is explained below:

- (1) Part: The Part module allows for the drawing of parts, be they 1-D, 2-D, or 3-D. It operates much like any CAD program. You may create many parts for use in your analysis, but for the time being, we will only create one part per model.
- (2) Property: This module allows for the definition of material behavior. For the bulk of the course, we will focus on the simplest material behavior: isotropic, linear elasticity, but Abaqus has an extensive library of material models. This module also allows for the specification of cross-sectional areas and profiles for analyses involving trusses or beams.
- (3) Assembly: Here, one would assemble the parts constructed in the part model into a single assembly. Since we will only be considering a single part, this step is quite simple for our present purposes.
- (4) Step: Specify the analysis type. We will mainly be performing “Static” analyses, meaning there is no contribution from inertia. This is the “bread-and-butter” analysis type in Abaqus, but the Abaqus analysis library is vast, allowing for the study of wave propagation, stress-temperature effects, instabilities, and much more physics. This module is also where one specifies the desired outputs.
- (5) Interaction: For problems involving contact, the details of the contact problem are specified here. We will come back to this module in the last assignment.
- (6) Load: Specify boundary conditions – both displacement boundary conditions as well as force/pressure boundary conditions. An important module, and the first place to look for errors if an analysis unexpectedly fails.

- (7) Mesh: Divide the part into elements. These elements are the fundamental units of the numerical technique. We will discuss the precise details of what these elements mean and do later in the course. For truss problems, each member should be a single element.
- (8) Optimization: For performing a topology optimization analysis. We will not be using the optimization module in this course.
- (9) Job: This step is where pre-processing ends and processing begins. Simply to create the job and tell the computer to calculate the solution!
- (10) Visualization: Provided the job completes successfully, you may view the simulation results in this module. This module encompasses all of post-processing. Here, we can view quantities, such as displacement, strain, and stress.

2.1 Units in Abaqus

It is important to note that Abaqus has no built-in set of units. *It is up to the user to choose a consistent set of units for dimensional quantities and stick with it.* In these exercises, we will use standard SI units.

2.2 Abaqus documentation

The Abaqus documentation is helpful for when you get stuck or want to utilize a feature with which you are unfamiliar. It is quite comprehensive – a hard copy of the documentation fills an entire shelf. The easiest way to find the information you need is to go to “Help ⇒ On context” and then click on any piece of the Abaqus/CAE environment. The corresponding page of documentation explaining that feature will then open. To simply search the entire documentation go to “Help ⇒ Search & Browse Manuals.”

3 Trusses

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 will, therefore, yield exact solutions.

The goal of the first in-class exercise is to give an introduction to building models in Abaqus in the context of trusses. Consider the simple plane truss, shown schematically on the following page. All joints are frictionless pins, and all members have the same length, which we take to be $L = 1$ m. Likewise, all members have a square cross-section with $b = 0.05$ m and are made of steel, so that $E = 210$ GPa and $\nu = 0.3$. A downward load of $P = 10$ kN is applied at pin B. Since the truss is statically-determinant, it is straightforward to calculate the forces in each of the members:

$$\begin{aligned} P_{AD} &= P_{CE} = -P/\sqrt{3}, & P_{AB} &= P_{BC} = P/(2\sqrt{3}), \\ P_{BD} &= P_{BE} = P/\sqrt{3}, & P_{DE} &= -P/\sqrt{3}, \end{aligned}$$

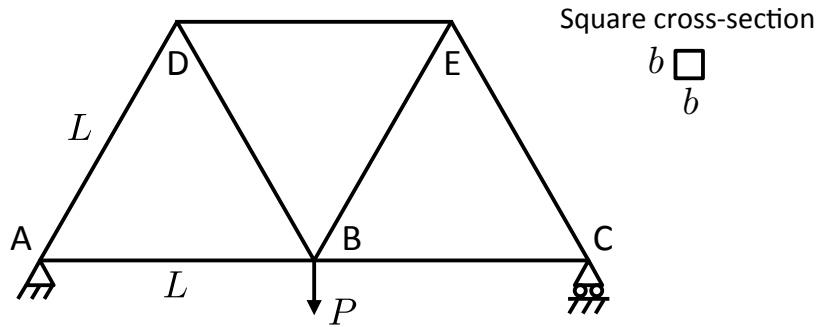


Figure 1: A simple statically-determinant truss.

where a negative force indicates that the member is in compression. The downward displacement of point B may be straightforwardly calculated (e.g., using Castigliano's theorem) as

$$\delta_B = \frac{11PL}{6EA},$$

where $A = b^2$ is the cross-sectional area. In this exercise, we will perform a numerical analysis of this truss and verify against the analytical results to convince ourselves of the effectiveness of the method.

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

(1) Part:

- Part \Rightarrow Create
- Select 2D Planar, Deformable, Wire, and Approximate size: 2
- Sketch the part as pictured in Fig. 1 and click Done

(2) Property:

- Material \Rightarrow Create
- Mechanical \Rightarrow Elasticity \Rightarrow Elastic
- Enter the material properties for steel and click OK
- Section \Rightarrow Create
- Beam \Rightarrow Truss \Rightarrow Continue
- Enter the cross-sectional area and click OK
- Assign \Rightarrow Section
- Select the entire part and click Done/OK

(3) Assembly:

- Instance \Rightarrow Create \Rightarrow OK

(4) Step:

- Step \Rightarrow Create \Rightarrow Static/General \Rightarrow Continue \Rightarrow OK

(5) Interaction: None

(6) Load:

- BC \Rightarrow Create \Rightarrow Mechanical \Rightarrow Displacement/Rotation \Rightarrow Continue \Rightarrow Select point A and click Done \Rightarrow Enter $U1=U2=0$ and click OK
- BC \Rightarrow Create \Rightarrow Mechanical \Rightarrow Displacement/Rotation \Rightarrow Continue \Rightarrow Select point C and click Done \Rightarrow Enter $U2=0$ and click OK
- Load \Rightarrow Create \Rightarrow Mechanical \Rightarrow Concentrated Force \Rightarrow Continue \Rightarrow Select point B and click Done \Rightarrow Enter CF2 and click OK

(7) Mesh:

- Make sure Object is set to Part.
- Mesh \Rightarrow Element Type \Rightarrow Select the entire part and click Done \Rightarrow Family: Truss \Rightarrow Click OK
- Seed \Rightarrow Edges \Rightarrow Select entire part and click Done \Rightarrow Method: By number \Rightarrow Number of elements: 1 \Rightarrow Click OK
- Mesh \Rightarrow Part \Rightarrow Yes

(8) Optimization: None

(9) Job:

- Job \Rightarrow Create \Rightarrow Continue/OK
- Job \Rightarrow Submit \Rightarrow Job-1
- When the job successfully completes: Job \Rightarrow Results \Rightarrow Job-1

(10) Visualization:

- Examine contour plots of displacement ($U1$ and $U2$) and stress ($S11$).
- Probe quantitative results at specific points of the model:
Tools \Rightarrow Query \Rightarrow Probe values
- Verify that the calculated results match their analytical counterparts. Repeat the calculation for $P = 50 \text{ kN}$ and verify.

Note on managers: Most of the menus encountered above have a “manager” associated with them, which you may find useful. These managers allow you to create, edit, rename, and delete the feature of interest. How you choose to navigate CAE is, in the end, a matter of taste.

4 Beams

In this set of exercises, 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 are only subjected to axial loading, beams are subjected to transverse loading and are capable of carrying shear forces and bending moments. The governing equation for the deflected shape of a beam v along its length x is

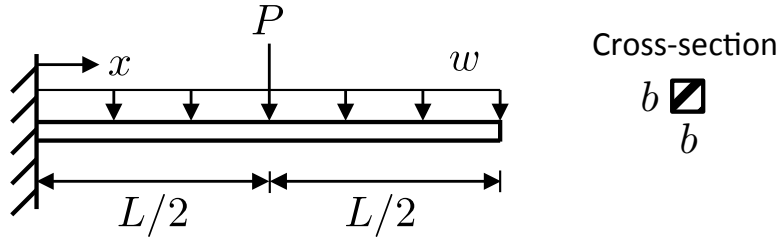
$$EI \frac{d^4 v}{dx^4} = -w,$$

where w 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 $w = 0$, the governing equation is solved exactly when $v(x)$ is a cubic polynomial in x . Therefore, Abaqus Euler-Bernoulli beam elements assume that the lateral deflection field $v(x)$ is interpolated by a cubic spline, and the calculated displacements and stresses are exact.

4.1 Cantilever beam

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



The beam has a length of $L = 1$ m and a square cross-section with $b = 0.05$ 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 $w = 10$ kN/m is applied along its length. The deflection field due to the midpoint load is

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

and the deflection field due to the distributed load is

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

so that, by superposition, the total deflection field is

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

and the magnitude of the tip deflection is

$$\delta = |v(x = L)| = \frac{5PL^3}{48EI} + \frac{wL^4}{8EI}.$$

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 mid-point 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.

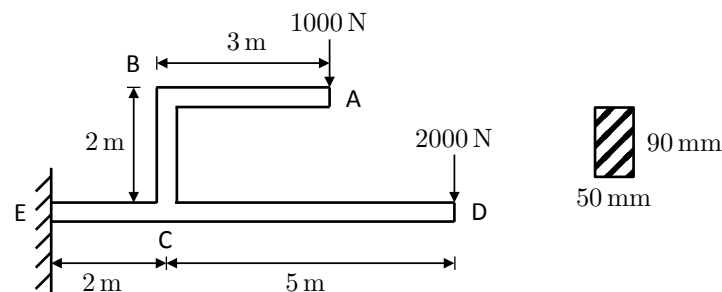
To calculate displacements along the length of the beam and to better resolve the stress, we need to use more elements.

- 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.

4.2 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