

ENGN 1750: Advanced Mechanics of Solids

Introduction to Abaqus: Trusses

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 Providence 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. It involves specifying geometry, material behavior, boundary conditions, and loading, as well as discretizing the analysis domain 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 simulations 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 towards 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 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: Break the part into elements. These elements are the fundamental units of the numerical technique. We will get into 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 this exercise, 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 \Rightarrow 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 \Rightarrow Search & Browse Manuals.”

3 An illustrative example

The goal of the first exercise is to give an introduction to building models in Abaqus in the context of a simple mechanics problem – a truss. Consider the following simple truss:

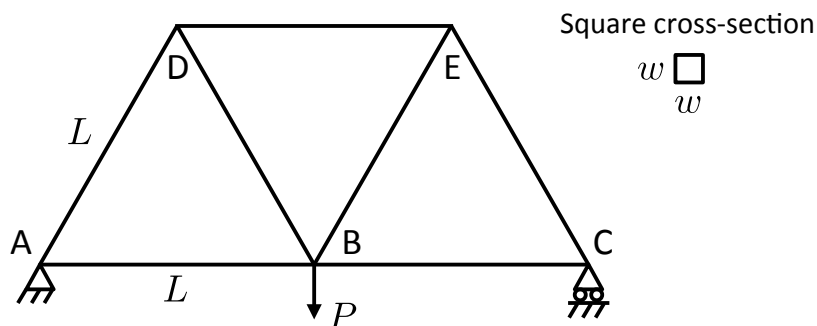


Figure 1: A simple statically-determinant truss.

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 $w = 0.05$ 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:

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

where a negative force indicates that the member is in compression. Using Castigliano's theorem (which we will learn about later in the course), we may calculate the downward vertical displacement of point B as

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

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.

3.1 Model definition and analysis

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:

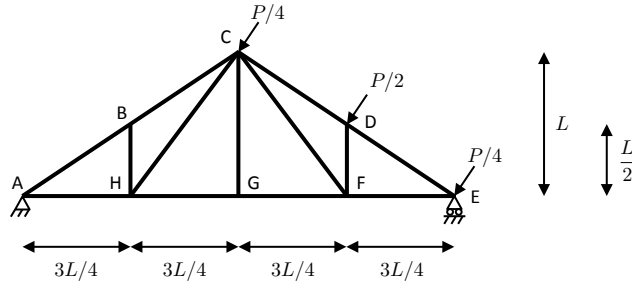
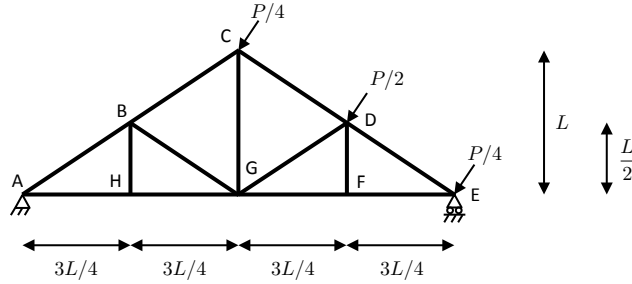
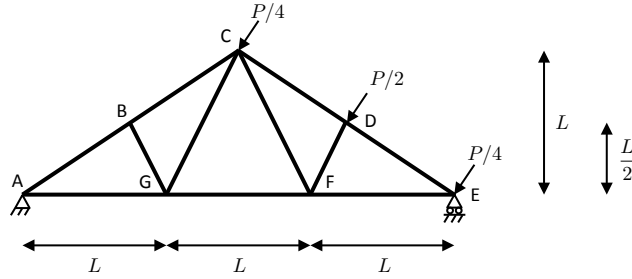
- Examine contour plots of displacement ($U1$ and $U2$) and stress ($S11$).
- Probe quantitative results at specific points of the model:
Tools \Rightarrow Query \Rightarrow Probe values
- Verify that the calculated results match their analytical counterparts. Repeat the calculation for $P = 50$ kN and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

ENGN 1750: Advanced Mechanics of Solids Abaqus Assignment 1

Due: Monday, September 29, 2014 OR Thursday, October 2, 2014 (in class)

Consider the following three trusses:



As in the class exercise, all members have a square cross-section with $w = 0.05\text{ m}$ and are made of steel. All joints are frictionless pins. Take $L = 1\text{ m}$. The truss is subjected to forces as shown in the schematics. The forces are applied *perpendicularly* to the side CDE of each truss structure. The truss is considered to fail when the magnitude of stress in any of its members reaches a value of 250 MPa (compressive or tensile), and we denote the value of P , at which the first truss member fails as P_{\max} .

1. Using Abaqus, determine the value of P_{\max} and the member that fails first for each truss.
2. Calculate the mass of each truss. Based on the maximum allowable load and mass of each truss structure, which design would you recommend? Justify your choice.