AE4ASM003 Linear Modelling Assignment 1: bar element

Due 15 September 2024, 17:00 CET

AIAS level 3 (AI Assessment level; definition: see slides lecture 1)

Please upload on Brightspace under the 'assignments' tab (on top of the page). You can upload multiple times, but only the last version you upload before the deadline expires is retained and will be checked and graded. Grading will only start after the deadline has passed.

Please note that the deadline is enforced: solutions uploaded less than 2 hours after the deadline will get a maximum of 80%; solutions handed in more than 2 hours after the deadline will not be graded. The plagiarism check is enabled in Brightspace. Please follow the submission guidelines on page 3 of this assignment.

As a guideline, the report should be 1-3 pages (including figures from Abaqus). This is just a guideline, your final report can be longer/shorter. Make sure you have answered all the questions as stated in the assignment.

The aim is to provide you with feedback and a grade two weeks after the deadline, if grading takes longer this will be announced on Brightspace. Please refrain from asking questions about grading until two weeks after the assignment deadline or the new date announced on Brightspace.

Important note on collaboration ONLY for assignment 1: you can work in groups of up to three on the coding. Clearly state the group members you have coded together with. The report is still individual, meaning everyone has to write their own report.

In the first week we have discussed several options to define the finite element formulation of a tapered bar under an end load (shown in Figure 1). In this exercise you are asked to write a generally applicable (i.e., different values for applied loads, different boundary conditions, different number of elements) code to solve this type of problem. As an input, your code should take the number of elements, material elastic modulus, total length of the bar, magnitude of the force applied at the end, and boundary conditions. The shape of the bar is not a variable, the load is always applied on the final node, and each element in a single model has the same length. Your code should calculate the displacement of each node, and stress and strains in each element.

The input file for the code (with the variables that can be changed) is attached to this assignment. Please code in either Python or Matlab, for both an input file is provided. You do have to fill in the correct numbers for this case, but **do not change the form of the input file (i.e., keep the names as given, follow the format described for each variable and do NOT add additional variables)**. Only the functionality of the code will be graded, not whether 'good coding practises' have been used.

Hint: use as many functions as possible that you can re-use in future assignments. Example of functions that can be useful in this assignment are

- define the area of each element (this one can be specifically for the case at hand, but by doing it as a function this is the only function that needs to be changed if you want to calculate a bar with a different shape)
- define the length of each element
- define the equivalent stiffness of each element
- assemble the global stiffness matrix using the connectivity matrix
- apply the boundary conditions
- calculate the strain and stress

As a verification case for your code, please use the tapered bar (with constant thickness t), discretised with 4 elements in Abaqus. The total length of the bar is 250 mm, other relevant data is given in Table 1.

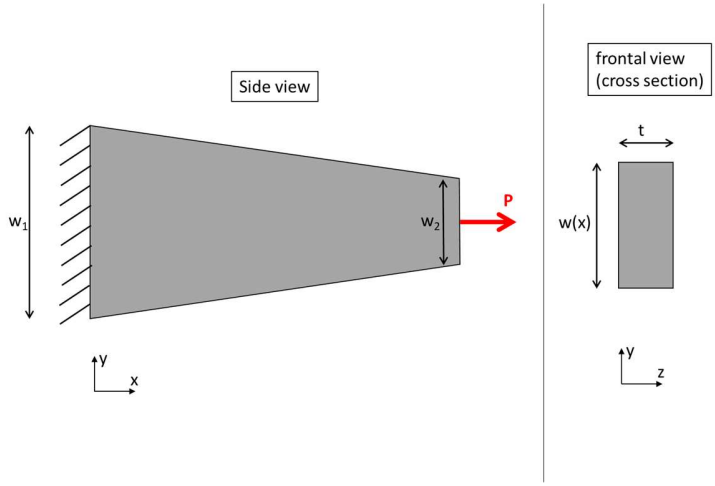

Figure 1: side and frontal view of the bar

Table 1: material and geometry properties

Property	Value
Elastic modulus	70 GPa
w_1	50 mm
w_2	25 mm
Thickness t	3.125 mm
Load P	1 kN

Tasks

1. Complete the code given in the assignment. Please make sure your code works with the input file provided! No need to describe the formulas in the report, just uploading the code is fine.
   Instead of coding such that you can vary the input, you can also hard-code the verification case (hence fixed loading, fixed boundary conditions and fixed number of elements; no need to use the input file in that case), in which case you can get a maximum of 7/10 for this assignment.
   Note: you also have to upload your input file since you have to fill in some numbers (upload the version used for verification); but **do not change the data format of the input file (also do not define additional variables)!**
   Note that upon running the code, the results should be printed, without prompting for input.
2. In the report, mention the two most important assumptions (i.e., the ones having the largest effect on the solution) that you made while solving this problem, indicating what effect they have on your solution.
3. In the report, show the verification you did with 4 elements using Abaqus.
4. In the report, answer the following question: if the area of the bar would be constant, would it make a difference whether you use one or multiple elements to model the problem (the load is still applied at the end of the bar, and the left side is clamped)? (Please answer this question using reasoning, no Abaqus or Python/Matlab output allowed in your answer)

Submission and naming:

- submit your own (or your groups) Python/Matlab code and the personal report in pdf format using the following naming convention:
    - report: LMex1_StudentNumber.pdf
    - input: HW1ForStudents_StudentNumber.py or HW1ForStudents_StudentNumber.m
    - functions: HW1functions_StudentNumber.py or HW1functions_StudentNumber.m
    - change StudentNumber by your student number in the naming convention
- Upload each individual file, no external links to the files, no zip folders.
- Make sure you add all files and click 'submit' (only last submission is visible for grading, so if you hit submit for each file only 1 file will be visible for grading; if you want to make a change to a file, you have to upload all files again).
- Only the functionality of the code will be graded, not whether 'good coding practises' have been used.
- Your submission should look like this right before submitting (check all files have been uploaded and are visible). Do not forget to click 'submit':