AE4ASM003 Linear Modelling Assignment 3: beam element

Due 6 October 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 5 of this assignment.

As a guideline, the report should be 5-10 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. For part 1 (especially derivations), you can make pictures of your hand written solution (on the condition that it is readable) rather than type it out to save some time making the report.

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.

Part 1: quadratic bar elements

The blades of the Ingenuity helicopter are modelled as bar elements. To simplify modelling, the bar has a length L, a constant cross section A and elastic modulus E. The bar is clamped in on one side and a linearly varying distributed in-plane load, occurring due to the spinning of the blades, is applied as shown in Figure 1. The load is given by $q_z(z) = a+b*z$. Due to the loading condition, it is opted to use quadratic bar elements rather than the linear elements we discussed in class.

Please solve the first three sub-questions in a symbolic way, and only use the values provided in Table 1 after obtaining and reporting the symbolic values.

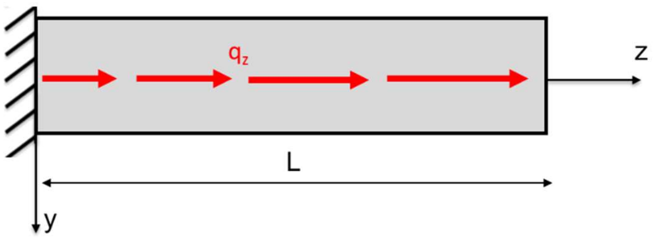

Figure 1: clamped beam, with in-plane loading shown.

Table 1: numerical values of the variables

Symbol	Value
A	120 mm^2
E	70 GPa
L	500 mm
a	13 N/mm
b	0.13 N/mm^2

Questions:

a) Derive the analytical expression for the elongation u and stress in the bar as a function of x.

b) Derive the expression for the stiffness matrix of a quadratic bar element. The shape functions are given in the lecture notes (page 68, eq 7.4) and can be directly used.

c) Derive the expression for the force vector of quadratic bar elements with a linearly varying distributed in-plane load, as in this case, starting from the external work done.

d) Write your own Python/Matlab implementation of this problem, where the number of elements can be varied. No input is provided, the code can be specifically written for the current problem. You have to upload the code (as .py or .m file), which upon running creates the plot(s) or the data used to solve question e. (lay-out of the plots may be different)
If you struggle with coding, then please solve the problem using 1 and 3 elements (either hard-coded or by hand), this leads to a maximum of 36/40 for this assignment

e) Perform a convergence study using your own code on this example. Evaluate and report the outcome. Also state when and why you consider your solution to have converged.
If your code cannot vary the number of elements, please indicate how you would perform the convergence study

f) If the cross section would vary (linear variation of the height, constant thickness), would using the cross section in the middle of each element be a good approximation? Explain why (not).

Part 2: model beam/truss in Abaqus

The ingenuity has 2 rotor blades, that are rotating in different direction (for an interactive 3D model, see https://www.jpl.nasa.gov/missions/ingenuity). At the bottom there are some avionics systems that can exert a moment. On top there is the solar panel and antenna. In the following, we are looking at the worst case of the avionics system being stuck exerting a moment while the rotors are engaged. This means on the supporting beam we have a moment M of 500 Nmm from the avionics, and 2 forces F of 850 N from the blades acting. In addition, there is the weight of the solar panels that is leading to a distributed load q of 25 N/mm. For convenience, the beam is rotated 90 degrees. You can assume the beam is fully clamped at point A. Beam ABCDE and EF are made of steel (E = 210 GPa, , v=0.3) with a square cross section with a side of 12 mm, as shown in Figure 3 on the left.

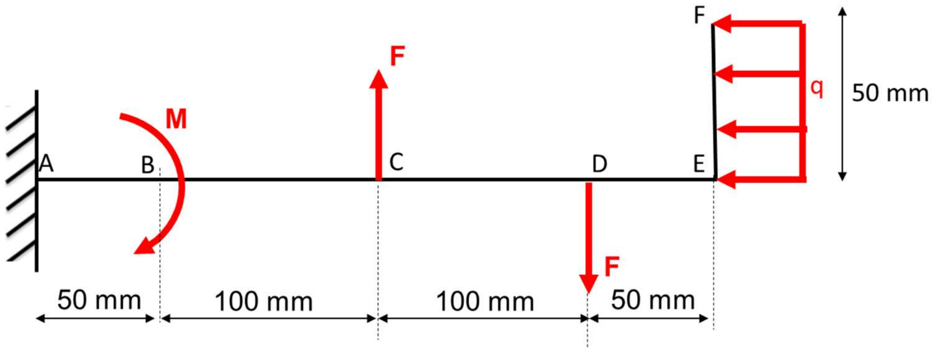

Figure 2: idealization of the beams in Ingenuity (drawing not to scale)

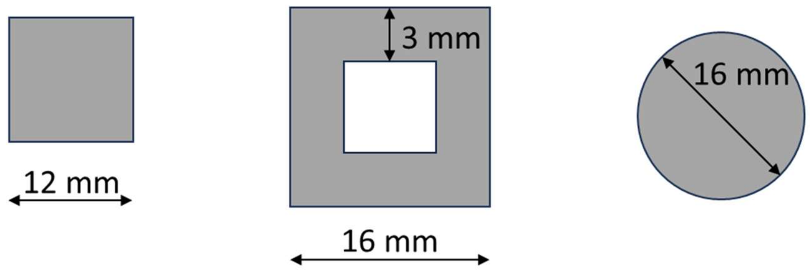

Figure 3: 3 possible cross sections (not to scale): original solid square (left), hollow square (middle) and solid circular (right)

Questions:

a) Model the part, shown in figure 2, in Abaqus using B23 elements.

b) If you were free to choose the element type, would you use B23 elements, or another type of beam element? Justify your answer!
Hint: for an overview of the types of beam elements in Abaqus, you can consult the following link:
https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/usb/default.htm?startat=pt06ch23s03alm07.html

c) How many elements (use B23 elements regardless of your answer in question b) do you need to use in Abaqus for a converged solution? Report the outcome and why you consider this number of elements to be sufficient. Contrary to good FEM practise, in this case you can fully rely on FEA, no need to perform an analytical calculation to check your solution.

d) A re-design is in order, where you want to find the lightest option possible. You have two materials available to you: aluminium (E = 70 GPa, v=0.31, density = 2700 kg/m^3, yield strength 270 MPa) or steel (E = 210 GPa, , v=0.3, density = 8000 kg/m^3, yield strength 550 MPa) and three types of cross sections: the solid rectangular section currently used, and two profiles with dimensions shown in Figure 3 (all have the same thickness in all parts): a solid circular profile with diameter of 16 mm or hollow square profile, with outer dimension of 16 mm and thickness of 3 mm. For manufacturing reasons, the horizontal part (ABCDE) has to be the same material and cross section all throughout the part ABCDE (but it can be different from the original design); the vertical part (EF) can be a different material and cross section than part ABCDE. The following constraints have to be satisfied:
   - Max angle at point F is 5 degrees
   - Max displacement in y direction is 17.5 mm
   - No yielding in the structure

e) Explain the result you found during the optimisation. Start by clearly stating the final solution you pick, and then comment on whether this combination makes sense (i.e., is it what you expected before checking multiple options).

Submission and naming:

- submit your own Python/Matlab code, the report in pdf format and the Abaqus model using the following naming convention:
    - LMex3_StudentNumber.pdf
    - LMex3_StudentNumber.py or LMex3_StudentNumber.m
    - Your Abaqus model of the original helicopter beams (no need to show the redesign): LMex3_StudentNumber.cae
    - 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).
- Your submission should look like this right before submitting (check all files have been uploaded and are visible). Do not forget to click ‘submit’: