

MECH0059: Advanced Computer Applications in Engineering Finite Element Analysis - Assignment

THE PROBLEM

Use finite element method to carry out a strain/stress analysis of the 2D structure shown in Figure 1A (under plane stress condition). Use following parameters: $E=40$ GPa, $\nu=0.3$ and plate thickness of 2 mm.

Part 1: Write a MATLAB program to determine the displacement and strain in the thin plate shown in Figure 1A, consisted of 4 noded quadrilateral elements (see Figure 1B), under uniform tension. Please use any arrangement and number of elements that you would like to (You may consider using four elements). Note that you must consider the Θ based on the first letter of your family name (see Figure 1C).

Part 2: Re-run your program in Part 1 to determine the displacement and strain, (i) for three additional elastic modulus (ii) for three additional loading conditions i.e. changing the loading direction from $\alpha=90^\circ$. Keep other variables constant. Here compare the results of the additional simulations performed.

Part 3: use ANSYS Mechanical APDL to: (i) validate the MATLAB program, (ii) carry out mesh convergence, for a specific configuration of your choice (in terms of elastic modulus and loading condition).

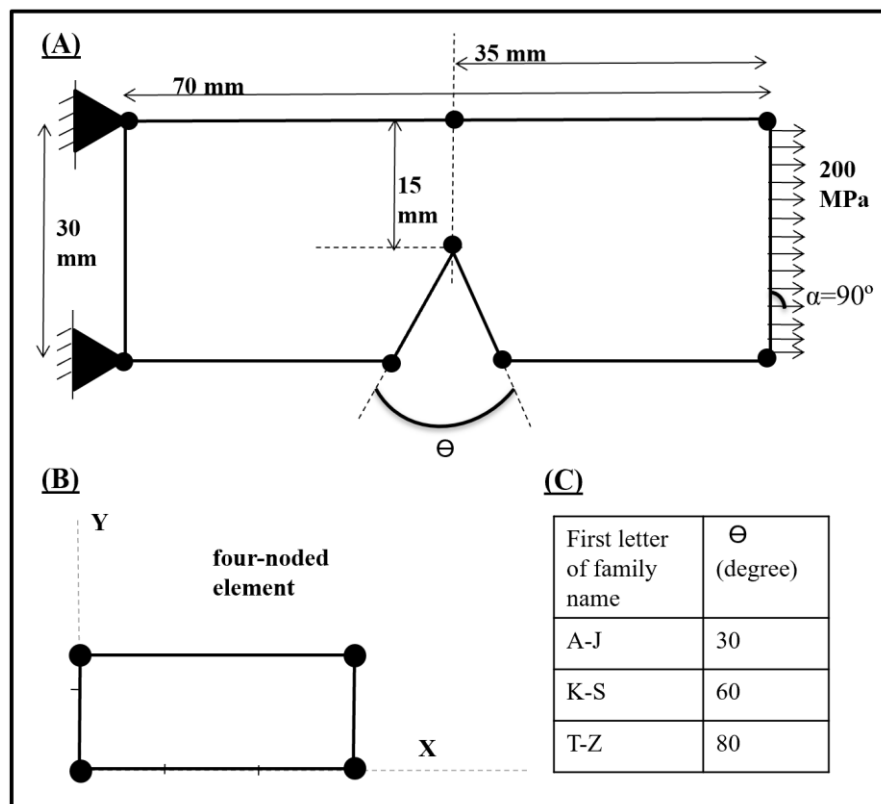


Figure 1: (A) thin plate under tension; (B) four-noded element [1]; (C) family name and corresponding Θ angle.

REPORT OUTLINE AND MARKING SCHEME

Prepare your report in the format of a journal article based on the following guidelines:

Title page

- title of report, your name, affiliation, email address and word count

Abstract -----

Introduction -----

- describe the problem; review previous studies/literature; describe the aim of the study...

Materials and Methods-----

- model description; material properties; boundary conditions and loads; element type and mesh convergence; simulations, and measurements (computer models & theory)...

Results -----

- results of the simulations in the same order that mentioned in the method...

Discussion -----

- discuss the results of each simulation in the same order as in the Results section; comparison between different models, with theory and/or experiments; comment on the validity and limitations of the model; summary/conclusion...

References

Figures and Tables

Appendix -----

- **MATLAB code must be included here**

See marking rubric.

REPORT REQUIREMENTS AND SUBMISSION DATELINE

- Your report must be submitted online on the relevant moodle page **1400 on 23rd January 2023.**
- The main text (excluding title page, abstract, references, equations, tables, figure captions and appendices) must not exceed 2500 words. Word count must be included on the first page.
- Total number of Figures plus number of Tables must be less than 10.
- Do not include the Figures and Tables in the main text. Include all the Figures and Tables at the end of the text after References. Figures and Tables must be cross-referenced in the text.
- Include page and continuous line number.
- Your report including all sections (i.e. including signed cover-sheet) must not exceed 15 pages.
- Use font Arial 11, line spacing 1.15, margins 25.4 mm on all sides, continuous line and page number. The formatting does not apply to the Appendix.
- International System of Units (or SI units) must be adopted throughout the report.
- Note up to 20 marks penalty will apply if you do not follow the guidelines.
- All students are reminded of the UCL plagiarism policies given in the following webpage, viz. <http://www.ucl.ac.uk/current-students/guidelines/plagiarism>

REFERENCE

- [1] Seshu P. Textbook of finite element analysis. PHI Learning Private Limited 2003.

Prof M Moazen
21st November 2022