Assignment 2: Finite element analysis of fibre-reinforced laminate

February 12, 2019

1 Introduction

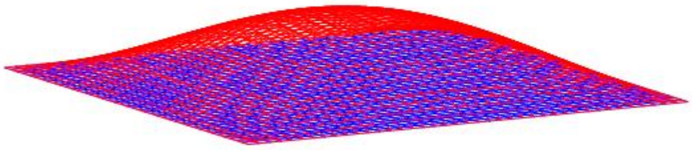

Figure 1: Undeformed and deformed laminate analysed using MATLAB

The assignment is a part of the course TME240 Composite Mechanics during the academic year 2018/2019. The assignment should be solved by use of MATLAB and ANSYS (Composite PrePost). The aim of the assignment is to gain knowledge and experience of how to: *i*) implement the finite element formulation of a laminate for an isoparametric quadrilateral four-node element and *ii*) to construct and solve a model for the same laminate using the commercial software package ANSYS using the pre- and postprocessing module ANSYS Composite PrePost.

1.1 Requirements

The computer assignment is to be carried out in groups of two students. Use the same groups as for assignment 1. The assignment tasks are in total worth at most 3 points (out of 18 for the full course examination), see course memo for details. Deadline for handing in the report is **Monday 25 February 2019**, **17:00** by uploading it to Ping-Pong. All reports will be automatically checked for plagiarism using the urkund-system (covers also the code in pdf-format which is therefore essential to include).

In order to obtain any points for the assignment work, you need to (before the deadline):

- upload a full report in pdf-format including the MATLAB source code for subtasks 1 and 2, motivations for assumptions made and choice of methods used, the main results and discussion thereof and
- upload the MATLAB source-code (subtask 1 and 2) as an archive (zip/rar)

Please note that you should not upload any ANSYS files on the course homepage. The report should be written in a way that the reader (corrector) can be convinced that the modelling of the plate is correct. Specifically, an image showing how the ply material properties are defined in ANSYS should be included in the report.

1.2 Point distribution

The rough point distribution is as follows:

- Implement the element routine for an isoparametric four-node (quadrilateral) element in which the element stiffness matrix and the element force vector due to out-of-plane loads are computed (1.5p)
- Implement the element routine for an isoparametric four-node (quadrilateral) element in which the in-plane stress components are evaluated at the centre of the element. In particular, plot the through-the-thickness stress distribution in the middle of the plate. (0.5p)

- Analyse the same laminated plate using ANSYS. (1.0p)

Full points also require a well written report. The report must contain the following parts:

- Problem description. *A brief description of the problem. What is known and what is sought after?*
- Method. *Presentation of the solution. Including assumptions and references to any equations/formulas used.*
- Results. *Presentation of the results sought after: specific output data, figures etc.*
- Discussion. *Brief discussion on the results and assumptions made. Are the results expected or not? How do they relate to the assumptions?*
- Source code. *A pdf-copy of all the MATLAB source code used to solve the problem.*

2 Assignment description

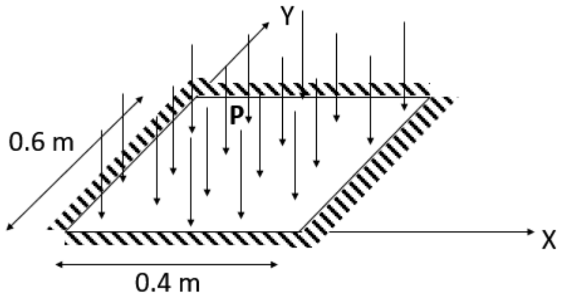

Figure 2: The plate considered in Assignment 2

A rectangular 1 mm thick carbon-fibre/epoxy laminate $[0_2/+45/90/-45]_S$ (where 0° corresponds to the x-direction) with side lengths 0.4 m (in x-direction) and 0.6 m (in y-direction), cf. Figure 2, is clamped at all four edges and loaded by a vertical uniform load p = -5 kPa.

Lamina/ply properties:

$$E_L = 181 \text{ GPa},\ E_T = 10.3 \text{ GPa},\ \nu_{LT} = 0.28$$
$$G_{LT} = 7.17 \text{ GPa},\ G_{TT'} = 3.5 \text{ GPa}$$

Main task

Determine the maximum out-of-plane displacement using MATLAB and ANSYS.

Subtasks

1. Implement, in MATLAB, the element routine 'laminateelement.m' for an isoparametric four-node (quadrilateral) element in which the element stiffness matrix and the element force vector due to out-of-plane loads are computed. Use the provided small FE-program ('assignment2_main.m') and the associated support file for generating the mesh ('rectangularmesh.m') [available for download on PingPong] to find the largest value of the out-of-plane displacement in any point of the plate. (1.5p)

   Points of discussion:

   - Is your implementation correct? Please note that you have the possibility in the main program to change boundary conditions to a simply supported plate. Think of how you can use this to verify your implementation against Kirchhoff plate theory. Do you need to modify/simplify the lay-up to verify your implementation? How much do the results for a simply supported plate differ between the FE-results and Kirchhoff predictions? How does the plate deform and why?

2. Implement, in MATLAB, the element routine 'laminatestress.m' for an isoparametric four-node (quadrilateral) element in which the in-plane through-the-thickness stress distribution (σ_L; σ_T and τ_{LT}) is calculated for the element midpoint. Please note that the main file ('assignment2_main.m') is prepared to call the function 'laminatestress.m' at the bottom of the program. A function to plot the stress over the domain is also provided for your convenience. (0.5p)
3. Analyse the same plate using ANSYS to find the maximum out-of-plane displacement. Also include a figure of the deformed shape of the plate coloured by the vertical displacement. The report should include a figure displaying how the material properties are defined in the software. (1.0p)

Points of discussion:

- How does the results correspond to the results in task 1?
- How many elements are required to obtain an acceptable solution? Motivate!

2.1 More details for each task

Subtask 1

The element subroutine

The element routine for a laminated plate element using Mindlin theory should be implemented as a MATLAB-function (in the function-file 'laminateelement.m') with the function head as:

function [ke,fe] = laminateelement(ex, ey, ep, A, B, D, A tilde, eq)

where 'ke' is the element stiffness matrix, 'fe' is the element force vector due to out-of-plane uniform loads and where the input is defined as:

- ex: vector containing the x-coordinates of nodes 1-4 in the format [x_1; x_2; x_3; x_4]
- ex: vector containing the y-coordinates of nodes 1-4 in the format [y_1; y_2; y_3; y_4]
- ep: Element parameters according to [irs, irb] where, 'irs' = the number of integration points in each direction for the shear stiffness terms and 'irb' = the number of integration points in each direction for the remaining stiffness matrix contribution and the force vector
- 'A': the [A]-matrix of the laminate (3x3)
- 'B': the [B]-matrix of the laminate (3x3)
- 'D': the [D]-matrix of the laminate (3x3)
- A_tilde: the $[\tilde{A}]$ -matrix of the laminate (2 x 2). Note! Don't forget to account for the shear correction factor!
- eq: vertical load per unit area

Please note that in order to simplify the task, a MATLAB-shell of the function-file 'laminateelement.m' is available at the course homepage under Documents → Computer assignment 2 → MATLAB files. This can be used as a starting point of your implementations. Please note that both the numerical integration scheme with possible so-called selective integration for the shear stiffness terms and the remaining terms has already been implemented as well as functions to evaluate the shape functions and their derivatives with respect to ξ and η.

The main program assignment2_main.m

You can of course use your own code to mesh and set-up the finite element solution procedure for the laminated plate in question. As an alternative, download 'assignment2_main.m' from the course homepage and use that to drive the simulation. Please note that you need to update this routine to calculate the laminate matrices [A], [B], [D] and $[\tilde{A}]$.