UNIVERSIDADE BEIRA INTERIOR

DEPARTAMENTO DE CIÊNCIAS AEROESPACIAIS

MECÂNICA ESTRUTURAL – 10391/10411

2020/2021

Mini-Project 1

1. OBJECTIVES

The main objectives of this project are:

- To learn how to implement a computer code to solve a structural problem, using the finite element method (FEM).
- To develop a one-dimensional (1D) finite element (FE) which couples two beam elements one torsion element to create a single linear element with 10 degrees-of-freedom (DOF).
- To use the developed 10-DOF element to create a representation of the wing, fuselage, horizontal tail and vertical tail structures.
- To implement a computer code to perform static and mode analyses of a given aircraft structure with prescribed cross-section geometries, materials, loads and constraints.
- To develop critical thinking when developing mathematical and computational models.
- To develop critical thinking when analysing computer codes' results.

2. PROBLEM

It is necessary to develop a numerical tool which uses beam finite elements and thin-walled cross-section data to analyse the structure of an airplane made of wing, fuselage, horizontal tail and vertical tail. The tool should be able to calculate deflections and cross-section stresses due to static loads and mode shapes with corresponding frequencies.

3. PROJECT REQUIREMENTS

The development of the computer code to calculate the static deflections and stresses and the natural mode shapes of vibration with the corresponding natural frequencies of an airplane structure shall be divided into the following tasks:

1. Investigation of the typical structure of a FEM program.
2. Development of a flowchart indicating the problem structure and the relationship between its various modules.

3. Development of the mathematical models for the wings, fuselage, horizontal tail and vertical tail:
    a. One-dimensional 10-DOF linear beam model for static analysis.
    b. One-dimensional 10-DOF linear beam model for modal analysis.
    c. Rotated beam model.
    d. Loads and boundary conditions.
    e. Geometry and structural properties.
    f. Meshing.
    g. System equations and problem solving.
    h. Data post-processing and visualization.
4. Development of the algorithms for the mathematical models previously developed.
5. Implementation in FORTRAN and testing of the code modules of the previously developed algorithms.
6. Integration of the various program modules to create the FEM program for the complete airplane structure.
7. Preparation of a final report describing the previous tasks. The report should not have more than 70 pages. The code should be included in an appendix.

4. TASK RESPONSIBLITY

This is a group project where each student should select a different task to contribute to the final program.

Table 1: Tasks and students' names.

Taks	Name
1	all
2	all
3,4,5 wings	
3,4,5 fuselage	
3,4,5 horizontal tail	
3,4,5 vertical tail	
3e,4e,5e	
3h,4h,5h	
6	
7	all