Introduction to FEM Analysis with ANSYS Mechanical

1. Program overview

Title: Introduction to FEM analysis with ANSYS Mechanical – online course.

Director: Professor Juan José Benito Muñoz.

Department: Construction & Manufacturing Engineering (UNED University).

2. Eligibility & requirements

A degree is required, although university students in their last year of their course may be admitted with proof of their academic status.

3. Presentation and objectives

The objective of this course is to introduce attendees to the use of *Finite Element* analysis software, allowing them to acquire the basic skills to enable them to work with this type of analysis in their professional practice.

This course originated as a collaboration project between UNED and Ingeciber, S.A., a company specializing in Computer-Aided Engineering (CAE).

4. Content

The course consists of three subjects:

a. Introduction to the use of the application software I
b. Introduction to the use of the application software II
c. Practical Application Exercises with ANSYS Mechanical

The content of each subject is detailed below:

- **Introduction to the use of the application software I**

ANSYS SpaceClaim, as well as ANSYS Mechanical (Meshing Introduction), will be studied during the course.

ANSYS SpaceClaim (ANSYS Geometry)

Attendees will study SpaceClaim, which is the ANSYS Geometrical Pre-processor. With SpaceClaim attendees will be able to create simple and complex geometries as well as modify geometries from standard CAD software.

The study of SpaceClaim has been structured into the following chapters:

- Lecture 1: Core skills
- Lecture 2: Creating geometry
- Lecture 3: Repairing geometry
- Lecture 4: FEA modeling
- *(Module 5 "CFD modeling" is not included, as it is not part of this subject)*
- Lecture 6: SpaceClaim to Workbench

The following solved exercises will be provided:

- Workshop 1.1: Basics
- Workshop 1.2: Creating simple bracket
- Workshop 2.1: Creating geometry
- Workshop 3.1: Repairing geometry
- Workshop 4.1: Preparing for FEA analysis
- Workshop 6.1: Parameters

ANSYS Mechanical (Meshing Introduction)

ANSYS Mechanical is a project-management tool. It can be considered as the top-level interface linking all our software tools. Workbench handles the transfer of data among ANSYS Geometry/Mesh/Solver/Post-processing tools.

This part of the ANSYS Mechanical study is about Meshing. It has been structured into the following chapters:

- Lecture 1: Core skills
- Lecture 2: Meshing methods
- Lecture 3: Global mesh controls
- Lecture 4: Local mesh controls
- Lecture 5: Mesh quality and advanced topics

The next solved exercises will be provided:

- Workshop 1.1: ANSYS Mechanical meshing basics
- Workshop 2.1: ANSYS meshing methods
- Workshop 3.1: Global mesh controls
- Workshop 4.1: Local mesh controls
- Workshop 5.1: 2D axisymmetric plate
- Workshop 5.2: Shell pressure vessel

- **Introduction to the use of the software application II**

As a continuation to the 1st subject, this subject covers ANSYS Mechanical in depth. The sections of this subject are:

- Lecture 1: Introduction
- Lecture 2: Pre-processing
- Lecture 3: Structural analysis
- Lecture 4: Post-processing
- Lecture 5: Mesh control
- Lecture 6: Connections and remote boundary conditions
- Lecture 7: Modal, thermal and multistep analyses
- Lecture 8: Eigenvalue buckling and sub-modeling analyses

The next exercises from the ANSYS Mechanical workbook of the introductory course will also be provided in order to complete the practical approach:

- Workshop 1.1: Mechanical basics
- Workshop 2.1: 2D Gear and rack
- Workshop 2.2: Named selections
- Workshop 2.3: Object generator
- Workshop 2.4: Object generator with named selections
- Workshop 3.1: Linear structural analysis
- Workshop 3.2: Beam connections
- Workshop 4.1: Mesh evaluation
- Workshop 4.2: Parameter management
- Workshop 5.1: Mesh creation
- Workshop 5.2: Mesh control
- Workshop 6.1: Contact offset control
- Workshop 6.2: Joints

- Workshop 6.3: Remote boundary conditions
- Workshop 6.4: Constraint equations
- Workshop 7.1: Modal analysis
- Workshop 7.2: Thermal analysis
- Workshop 7.3: Multistep analysis
- Workshop 8.1: Eigenvalue buckling
- Workshop 8.2: Sub-modeling

- **Practical Application Exercises with ANSYS.**

The objective of this subject is to complete the concepts explained previously in the first two subjects through a number of exercises that must be completed using SpaceClaim and ANSYS Mechanical.

The exercises represent a review of the concepts introduced in the subjects taken till now, as well as the orderly use of the ANSYS SpaceClaim and ANSYS Mechanical software.

These exercises will be delivered to the tutor in order to get feedback and recommendations.

The exercises will be similar to the following ones:

- Advanced analysis of a warehouse with temperature jump
- 3D truss bridge structural analysis
- Offshore platform design for different structural loads
- Structural analysis and validation of a space satellite
- Structural analysis of a steam condenser
- Pre-stress bolt design of a union

5. Schedule

50 hours of study. The course lasts from 1 to 6 weeks with full flexibility since no specific delivery date is indicated.

6. Methodology

Distance learning methodology, including pre-prepared study materials and bibliography, tutorials, audiovisual resources and practical application exercises.

7. Teaching materials

Attendees will receive the teaching guide and the corresponding materials for each module, which will basically consist of the subject texts.

Furthermore, in order to complete the practical exercises and training, the educational version of ANSYS Mechanical will be provided by the course.

The course uses a virtual classroom as a training facility where study tools can be found and also as the main communication channel with the attendees.

Other tools will also be used, including audiovisual resources as well as other supplementary documentation.

The teaching materials for this subject consist of:

- The introduction to ANSYS SpaceClaim training material and related workbook exercises as well as an Introduction to ANSYS Mechanical and related workbook exercises.
- Additional training material for the course developed by ICAEEC.
- Software: ANSYS SpaceClaim and ANSYS Mechanical.

8. Attendee services

The teaching staff will respond to attendee inquiries via telephone, email, or in person. Phone tutorships will be available within the following hours:

Monday to Friday during office hours and always subject to tutor's availability.

9. Evaluation and grading criteria

Attendee evaluation will be performed through the practical application exercises.

10. Certification

Certification will consist of a diploma from ICAEEC & Ingeciber indicating successful completion of the subject by the attendee as well as the grade obtained in the practical application exercises.

11. Teaching staff

Professor J. J. Benito (director). Construction & Manufacturing Engineering Department (UNED).

Mr. Ronald Siat (coordinator & tutor). Ingeciber, S.A.

Mr. Ambrosio Baños (tutor). Ingeciber, S.A.

12. Fees

Tuition fees are 450,00 €.

Current and former attendees of the UNED *Master's in Theoretical and Practical Application of the Finite Element Method and CAE Simulation* are eligible for a 33% discount.

13. Validation

Attendees who pass this course can request validation of the application and practical course subjects of the mechanical branch of the ANSYS Mechanical expert module from the academic board of UNED *Master's in Theoretical and Practical Application of the Finite Element Method and CAE Simulation*.