Non-linear Analysis with ANSYS Mechanical

1. Program overview

Title: Non-linear 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 the last year of their course may be admitted with proof of their academic status.

Basic knowledge of linear static structural analysis with ANSYS Mechanical is required, which may have been acquired through:

- Completion of the *Introductory course to FEM with ANSYS Mechanical*, also available in ICAEEC.
- Completion of the Expert module of the Mechanical branch of the *International Master's in Theoretical & Practical Application of the Finite Element Method and CAE Simulation* of UNED-Ingeciber.

3. Presentation and objectives

The objective of this course is to introduce attendees to non-linear analysis using *Finite Elements*, allowing them to obtain the necessary skills to be able to use this method professionally.

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 two subjects:

a. Non-linear analysis with ANSYS Mechanical
b. Practical Application Exercises with ANSYS Mechanical

The documentation for both subjects is in English.

The content of each subject is detailed below:

- **Non-linear analysis with ANSYS Mechanical**

The ANSYS Mechanical training notes are structured into the following chapters:

- Lecture 0: Local introduction
- Lecture 1: Overview and procedure
- Lecture 2: Restarts and non-linear controls
- Lecture 3: Introduction to contact
- Lecture 4: Rate independent plasticity
- Lecture 5: Buckling analysis and linear perturbation
- Lecture 6: Non-linear diagnostics
- Lecture 7: Mesh non-linear adaptivity

Various exercises are also proposed:

- Workshop 1.1: Shell-disk
- Workshop 1.2: Large-deflection
- Workshop 2.1: Restart-controls
- Workshop 2.2: Line-search
- Workshop 3.1: Contact-stiffness-study
- Workshop 3.2: Symmetric-vs-asymmetric
- Workshop 4.1: Metal-plasticity
- Workshop 5.1: Linear-eigenvalue-buckling
- Workshop 5.2: Post-non-linear-buckling
- Workshop 5.3: Linear-perturbation
- Workshop 6.1: Contact-diagnostics

- **Practical Application Exercises with ANSYS Mechanical**

This subject completes the previous one with practical application exercises of professional scope in order to study the software in depth.

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

These exercises are as follows:

- Non-linear analysis of a flat square membrane structure
- Linear and non-linear buckling analysis of a thin walled cylindrical structure
- Material non-linearity: a simply supported circular plate subjected to a cyclic load
- Elastic-plastic non-linear analysis of a sphere
- Contact analysis

5. Schedule

35 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 complementary documentation.

The teaching material for this subject consists of:

- Non-linear analysis with ANSYS Mechanical training material 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 evaluation 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).

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 non-linear analysis specialized module using ANSYS Mechanical from the academic board of UNED *Master's in Theoretical and Practical Application of the Finite Element Method and CAE Simulation*.