Dynamic Analysis with ANSYS Mechanical

1. Program overview

Title: Dynamic 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 dynamic 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. Dynamic 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.

- **Dynamic analysis with ANSYS Mechanical**

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

- Lecture 1: Introduction
- Lecture 2: Mechanical basics
- Lecture 3: Damping
- Lecture 4: Modal analysis
- Lecture 5: Harmonic analysis
- Lecture 6: Linear perturbation analysis
- Lecture 7: Response spectrum analysis
- Lecture 8: Random vibration
- Lecture 9: Transient analysis

Various exercises are also proposed, involving the following topics: Modal Analysis, Transient Analysis, Harmonic Analysis and Spectral Analysis:

- Workshop 1: Intro. (Fly Wheel)
- Workshop 2: Mechanical basics
- Workshop 3: Damping in mechanical workbench
- Workshop 4A: Modal analysis (plate with a hole)
- Workshop 4B: Modal analysis (model airplane wing)
- Workshop 5: Harmonic response (fixed-fixed beam)
- Workshop 7: Response spectrum (suspension bridge)
- Workshop 8: Random vibration (girder assembly)
- Workshop 9A: Transient analysis (caster wheel test)
- Workshop 9B: Transient analysis of a gantry crane
- Workshop 9C: Applying initial rotational velocity

- **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:

- *Harmonic* / *Frequency* analysis of a bedplate
- Transitory analysis of a bedplate
- Pre-stressed modal analysis of a guitar string
- Modal, harmonic and resonance analyses of an impulse system
- Spectral modal analysis of an instrument panel
- Mechanical analysis of a wheel rim
- Nonlinear transitory analysis of a cylinder system

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:

- Dynamic 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 application exercises.

11. Teaching staff

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

Mr. Ronald Siat (tutor & coordinator). 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 Dynamic 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*.