Introduction to FEM analysis with ANSYS Mechanical APDL

1. Program overview

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

Director: Professor Juan José Benito Muñoz.

Department: Construction & Manufacturing Engineering (UNED University).

2. Eligibility and requirements

A degree is required, although university students in the 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 APDL

The content of each subject is detailed below:

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

The introduction to ANSYS training material and related workbook exercises (step-

by-step solved exercises) are provided. The list of chapters and exercises is shown below:

1. Introduction
    1.1. No workshop
2. FEA and ANSYS
    2.1. Introductory workshop
3. Getting Started
    3.1. Getting started workshop
4. ANSYS basics
    4.1. Basic workshop
5. General analysis procedure
    5.1. No workshop
6. Creating the solid model
    6.1. IGES
    6.2. SAT
    6.3. SAT assembly
    6.4. Parasolid
    6.5. Parasolid assembly
7. Creating the Finite Element model
    7.1. Silo
    7.2. Pillow block
    7.3. Connecting rod
    7.4. Cotter pin
    7.5. Impeller extrusion
    7.6. Wheel
8. Defining the material
    8.1. User input material
    8.2. Material library input
9. Loading
    9.1. No workshop
10. Solution

10.1. 3-D bracket
10.2. Connecting rod
10.3. Wheel
11. Structural analysis
11.1. A. Lathe cutter
11.2. B. 2-D corner bracket tutorial
12. Thermal analysis
12.1. Axisymmetric pipe with fins
13. Post-processing
13.1. Connecting rod
13.2. Spherical shell
13.3. Axisymmetric fin with multiple load steps
13.4. Results viewer
13.5. Report generator
14. Short topics
14.1. *ABBR - Plate with hole at center
14.2. 2-D bracket using parameters
15. Appendix A – Creating the solid model
15.1. Pillow block
15.2. Connecting rod - bottom-up
15.3. Connecting rod - import/clean-up
16. Appendix B

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

The introduction to ANSYS training material and related workbook exercises (step-by-step solved exercises) are provided. The list of chapters and exercises is shown below:

1. Introduction
1.1. Welcome
2. Array parameters
2.1. Axisymmetric wheel

3. Coupling & constraint equations
   3.1. Impeller blade (coupling)
   3.2. Turbine blade (constraint equations)
   3.3. Swaybar (rigid regions)
4. Working with Elements
   4.1. Bolt torque (surface effect elements)
5. Beam modeling
   5.1. Building frame
6. Coupled field
   6.1. Thermal pipe
7. Sub-modeling
   7.1. Crank shaft
8. Modal analysis
   8.1. U-bracket
9. Introduction to nonlinear analysis
   9.1. Arched beam
10. Bonded contact
    10.1. Swaybar
11. Macro basics
    11.1. Verifying pressures
12. Appendix A

- **Practical Application Exercises with ANSYS**

The exercises represent a review of the concepts introduced in the subjects taken until now, as well as the orderly use of the ANSYS APDL.

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 optimization
- 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 material for the course. Furthermore, in order to complete the practical application exercises and training, the educational version of ANSYS 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:

- Training materials are the original ANSYS basic texts for an introductory course of ANSYS APDL
- Additional training material for the course developed by ICAEEC
- Software: ANSYS Mechanical APDL

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).

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.