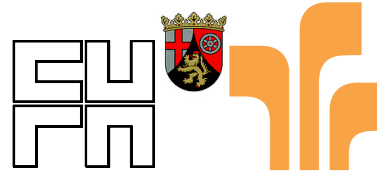




Ho Chi Minh City University of Technology
Vietnam National University - Ho Chi Minh City



Fachhochschule Trier
University of Applied Sciences

Finite Element Analysis for Structural Mechanics using ANSYS®

This course will address

- to computational and design engineers, who want to specialize in the field of Computer Added Engineering CAE
- to experimental and numerical practitioners, who want to expand their knowledge in CAE
- to managing engineers as well, who like to know the background of Applied Computational Mechanics

The participants will learn all steps of a finite element analysis from the generation of geometry to the evaluation of results using the simulation programme ANSYS.

Lecture notes: English

Lecture language: Vietnamese

- **Introduction to the FEM and to ANSYS**
 - o Bases of the FEM
 - o ANSYS data structure
 - o Handling of the program interface of ANSYS
- **Modelling**
 - o Geometry definition using ANSYS pre-processor
 - o Definition and assignment of material properties
 - o Meshing the model
- **Solution**
 - o Different element types and their use
 - o Applying boundary conditions and loads
 - o Different kinds of computation
 - o Solution commands
- **Result evaluation**
 - o Plotting the results
 - o Interpretation of the results
 - o Checking the quality of the results

Centre of Computational Mechanics (CCM)

Dr. Nguyen Tuong Long

Phone / Fax : 84-8-3651211

Hand Phone: 0919500570

Email : ntlong@hcmut.edu.vn