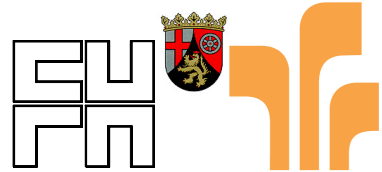**Ho Chi Minh City University of Technology**
Vietnam National University - Ho Chi Minh City

FachhochschuleTrier
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