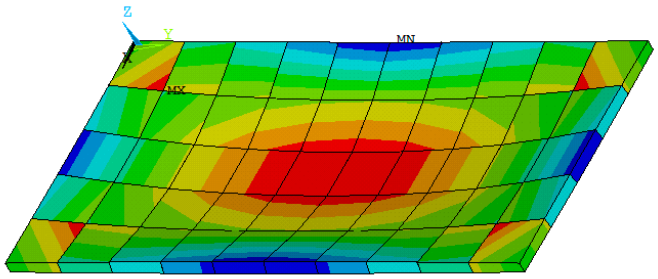
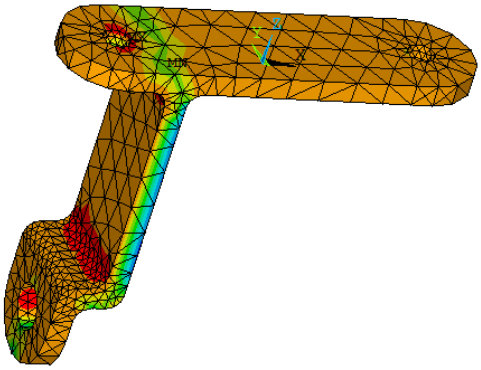
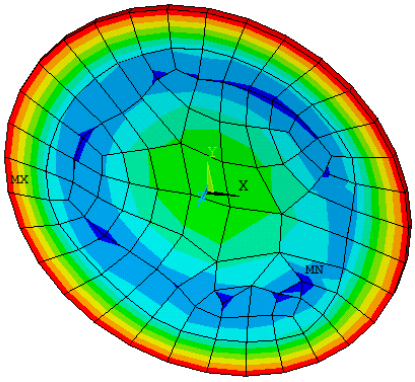
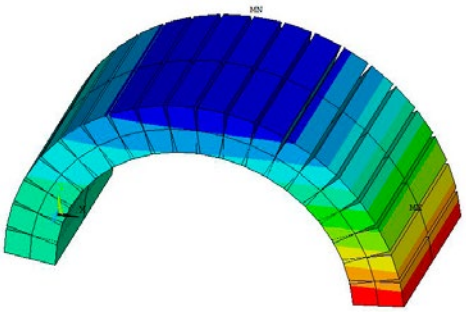




INTRODUCTION TO

FINITE ELEMENT ANALYSIS





UNIVERSITEIT GENT

POST-ACADEMIC COURSE

17, 18 AND 19 MAY 2016

INTRODUCTION

Finite Element Analysis (FEA) has become one of the most popular numerical techniques due to its applicability to a broad field of continuum mechanics and many other disciplines, its diversity, flexibility, and ability to provide approximated numerical solutions to a wide range of engineering applications. This is why FEA is receiving much attention in academia and industries. With the advances of computer technology, FEA is widely used by many scientists and becomes very common in engineering communities. Mechanical and Civil engineers are continuously facing computational problems related to their field of applications.

The FEA knowledge makes a good engineer better than others, while just user without the knowledge of FEA may produce more dangerous results. To use a FEA package properly, the user must pay attention to several points: Which elements are to be used for solving the problem in hand? How to discretise to get good results? How to introduce boundary conditions properly? How are properties of an element developed and what are their limitations? How are the plots developed in pre- and post-processor and how to understand their limitations? How to understand the difficulties involved in the development of FEA programs and hence the need for checking the commercially available packages with the results of standard cases?

Unless the user has a sound background of FEA, he may produce wrong results and may go with overconfidence. Thus, it is necessary that users of FEA package should have good knowledge of FEA theoretical background.

This three-day course covers the theoretical background of FEA and provides the opportunity of a first experience with two very well-known finite element commercial packages, namely ANSYS and ABAQUS.

Who should attend?

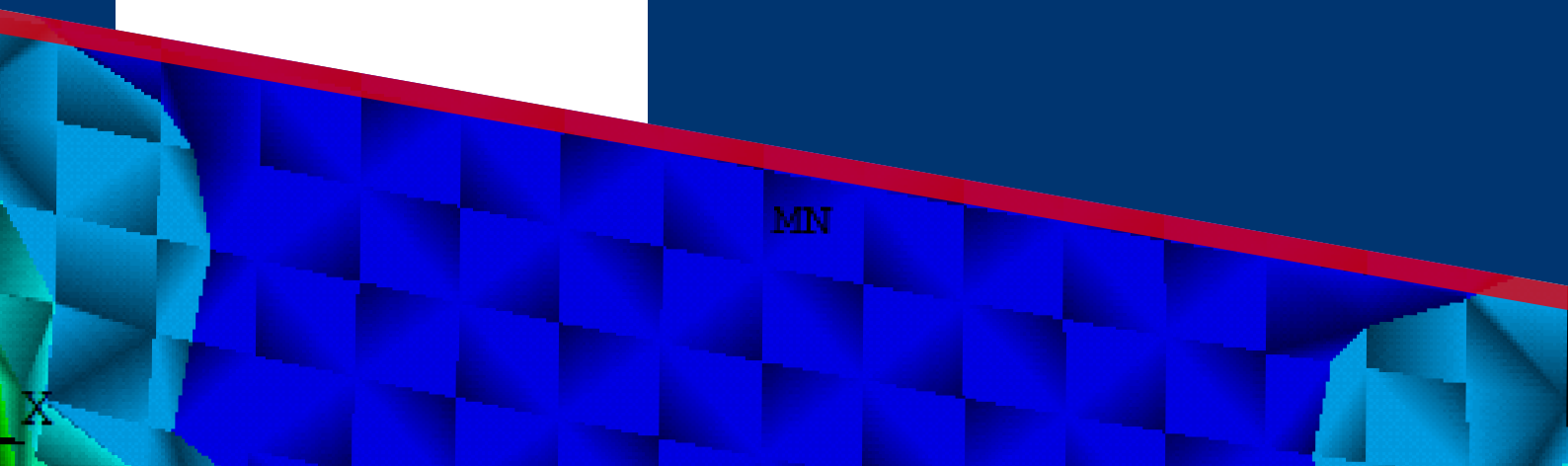
This short course is open to all professional engineers, who are interested in acquiring background knowledge of FEA and learning how to simulate mechanics problems in commercial software programmes. It is also open for postgraduate students in the framework of doctoral training programme at Ghent University.

Scientific Coordination & Teacher

Prof. Magd Abdel Wahab
Department of Electrical energy, systems and automation, Soete Laboratory, Ghent University

Post-academic course Certificate granted by Ghent University

To receive a certificate, one should attend all three days and submit a report on ANSYS and ABAQUS exercises.



PROGRAMME

17 May 2016
Theoretical background

The lectures provide theoretical background on the basis of FEA. After a brief introduction on how the finite element method works, types of problems in two-dimensional elasticity, such as plane stress, plane strain and axisymmetric problems are presented.

Furthermore, details on the different types of nodes and elements that can be used in FEA are explained. In order to derive stiffness matrix, displacement formulation using direct method, as well as, other techniques, is introduced for bar, truss and beam elements. By knowing the features of stiffness matrix, a solution technique to solve the matrix displacement equations is proposed using Gauss elimination method. Different coordinate systems, such as global, local and natural coordinates are considered. Element shape functions are derived using different techniques, such as polynomials, Lagrange and Hermite methods. Strain displacement matrix is formulated for one-dimensional and two-dimensional elements. Assembling stiffness equation is presented using direct method, Galerkin's method and Variational principles.

The FEA formulation is explained in details for 1D analysis including bar, truss, classical beam, frame and Timoshenko beam problems and for 2D analysis including plane stress and plane strain problems using constant strain triangular elements and higher order elements. The isoparametric formulation is presented and numerical integration of stiffness matrix is briefly summarised. Some hand calculation exercises are provided in order to gain insight into the theory and formulations of FEA. Finally, FEA theories of plates and shells are briefly presented, including thin plate theory, displacement models for plate analysis, Mindlin's plate element, stress smoothening, and shell element.

18 and 19 May 2016
Exercises using ANSYS & ABAQUS

These exercises start with a brief introduction to the finite element commercial packages ANSYS and ABAQUS. The steps that are required to define a problem in each package are presented. These steps includes pre-processing (defining element type, material properties, geometry, meshing), solution (defining analysis type, time step, applying load and boundary conditions, solve) and post-processing (reviewing results, plotting deformed shape, plotting contours of displacements and stresses).

For each FEA package, four exercises are planned, which cover all theoretical formulations presented in the lectures. In the first exercise, beam models are considered, including a semi-circular curved beam and a thin beam structure.

The second exercise is concerned with plane stress plate structure using 2D elements using different element types and mesh refinements.

In the third exercise, circular and rectangular plates are modelled using shell elements.

The last exercise is concerned with stress analysis of a 3D CAD model, in which 3D elements are used. A complete guidelines will be given for each exercise. Results obtained from ANSYS and ABAQUS are compared to simple analytical solutions in order to assess their accuracy and the efficiency of FEA.

MORE INFORMATION & SUBSCRIPTION
www.ivpv.ugent.be/FEA

Fee

Payment occurs after reception of the invoice.
All invoices are due in thirty days.
All fees are exempt from VAT.

All lessons	€ 900

Reduction

- When a participant of a company subscribes for the complete course, a reduction of 20% is given to all additional subscriptions from the same company. Invoicing is then done by one company invoice.
- AIG and VBIG members receive a reduction of 10% on the price mentioned in the table.
- Special prices for Ghent University staff and members of Ghent University Association.
- Reductions can't be combined.

Cancellation policy

When cancelling up to 10 days before the start of the course 25% of the participation fee will be charged.
When cancelling less than 10 days before the start of the course, the full fee is due.

Training cheques

Ghent University accepts payments by KMO-portefeuille (www.kmo-portefeuille.be; authorisation ID: DV.O103194).

Time and location

- Lessons are given from 9h until 16h (inclusive coffee breaks and sandwich break) at IVPV classroom and PC room, building **904, Technologiepark, Zwijnaarde**.
- Dates may change due to unforeseen reasons.

Language

English is used in all presentations, exercises and documentation, so a good knowledge of this language is necessary.

MORE INFORMATION & SUBSCRIPTION

www.ivpv.ugent.be/FEA

ORGANISATION
Ghent University
Institute for Continuing Education / Instituut voor Permanente Vorming (IVPV)
Technologiepark 904, 9052 Zwijnaarde
Tel: +32 9 264 55 82, Fax: +32 9 264 56 05
E-mail: ivpv@UGent.be