

## **FEMAP&Nastran Introductory Course** **FEA Training (3 days)**

### **Course Agenda**

#### **Day 1** **Time: 08:30-16:30**

- Software Installation and Preferences
- Modeling Procedures in Femap  
*Exercise 1 - Analyzing a Solid Part*
- Femap User interface and the On-Line Help
- View and Display options  
*Exercise 2 - User Interface*
- Defining Materials and modifying materials
- Defining Properties  
*Exercise 3 – Analyzing a Beam Model*
- Mesh Sizing and Meshing  
*Exercise 4 - Beam/Plate Meshing*
- Mesh Toolbox and Geometry Subdivision Tools  
*Exercise 5 - Interactive Meshing and Geometry Manipulation*

#### **Day 2** **Time: 08:30-16:30**

- Building Geometry in Femap, Import and Modifications  
*Exercise 6 - Creating Solid Meshing*  
*Exercise 7 - Tetrahedral vs. Hexahedral Meshing*
- Midsurfacing of Geometry  
*Exercise 8 - Midsurface Modeling*  
*Exercise 9 - Interactive Midsurface Meshing*
- Definitions of Loads and Constraints  
*Exercise 10 - Bearing Load*
- FE Assembly Connections  
*Exercise 11 - Solid Assembly with Linear Contact*
- Analysis Set Manager and Nastran Solution Sequences
- Visualizing and Documenting Results  
*Exercise 12a - PostProcessing with Deformed Contour Plots*  
*Exercise 12b - PostProcessing with XY Plots*

---

#### **FEMComp Engineering AB**

Nygårdsgatan 12  
SE-722 19 VÄSTERÅS  
Sweden

Tel: +46-21-35 00 45  
GSM:+46-70-32 62 518

E-mail: [info@femcomp.com](mailto:info@femcomp.com)  
Web: [www.femcomp.com](http://www.femcomp.com)

**Day 3**

**Time: 08:30-16:30**

- Nastran for FEMAP Basic Analysis
- Nastran Overview, Analysis Types and Features
- FEA Model Input and Output
- Nastran Input Data and Element Library
- Nastran for FEMAP Analysis Set-Up
- Nastran for FEMAP Modeling Practice
  - Common Types of Errors*
  - Minimum Recommended Model Checks*
- Nastran for FEMAP Workshop
- Nastran Automated Report Generation

The agenda may be changed without notice