

FEMAP & Nastran Introduction Seminar

The purpose of the seminar is to show for participants different FEM modelling and solutions techniques using FEMAP & Nastran packages. The seminar will cover the following issues:

- How to use CAD-FEM interfaces? (for example CAD created geometry),
- How to define FEA model material properties?
- How to apply loads and constraints?
- How to develop a Finite Element Analysis (FEA)?

During the seminar we are going to use FEMAP & Nastran FE programs.

Date: to be announced

Place: FEMComp Engineering Head Office, Nygårdsgatan 12, 722 19 Västerås

Time: 09.30 - 15.00

Price: to be announced

Registration: Please contact us or use Register On-Line

AGENDA

(09.30 - 09.45) Registration

(09.45 - 10.00) Introduction and presentation of FEMComp Engineering

(10.00 - 11.30) FEMAP General Presentation followed by:

- Geometry creation and import of CAD geometry.
- Automatic and manual meshing
- Material and property definition
- Load and constraint application
- Nastran Analysis Solution
- Results evaluation and communication

(11.30 - 12.00) Introduction to FEMAP & Nastran exercises

(12.00 - 13.00) Lunch Break

(13.00 - 14.30) FEMAP & Nastran exercises

(14.30 - 15.00) Summary and discussion