

Simcenter Nastran 2019.2

Introduction to Finite Element Analysis with Femap for Pre/Post

Course Code **NAS110-GH**
User Level **Beginner to Intermediate**
Language **English**
Price **\$3,300.00 (USD)** (Price may not include taxes applicable to your billing region)
Training Center Duration **3 Days**

For More Information **Learning Services, USA (training.usa.plm@siemens.com)**

The **Introduction to Finite Element Analysis with Femap for pre/post** course is an introductory course for static finite element structural analysis using NX Nastran. The course is designed to teach NX Nastran independent of pre- and post- processors. However, additional material is available for demonstration of use with Femap. It is intended for analysts who want to learn the details of how to perform static structural analysis using NX Nastran.

The course covers the details of the structural analysis processes to define loads and boundary conditions, FEA model checking and solving, and interpreting the results. This course also covers NX Nastran deck format, executive section, Case control, Output control, and common element uses and limitations.

This course **does not** cover any topics on FEMAP simulations. For information on FEMAP, please refer to the prerequisites information below.

WHO SHOULD ATTEND

This course is intended for designers, engineers and finite element analysts who would like to understand how to perform linear static analysis using NX Nastran.

PREREQUISITES

Other recommended courses:

- Femap (FEMAP101)

- Basic understanding of finite element analysis principles, statics, solid mechanics, and basic dynamics.

PROVIDED COURSE MATERIAL

- Student Guide
- Activity Material

COURSE TOPICS

- Introduction to static finite element analysis
- Static solution sequences
- Case control
- Output control
- Common elements, uses and limitations
- Constraint and load types
- Model verification and checking
- Buckling analysis