

Linear Statics, Normal Modes and Buckling using MSC.Nastran and MSC.Patran.



Course Description

This seminar introduces basic finite element analysis techniques for linear static, normal modes, and buckling analysis of structures using MSC.Nastran and MSC.Patran. MSC.Nastran data structure, the element library, modeling practices, model validation, and guidelines for efficient solutions are discussed and illustrated with examples and workshops.

MSC.Patran will be an integral part of the examples and workshops and will be used to generate and verify illustrative MSC.Nastran models, manage analysis submission requests, and visualize results. This seminar (or NAS101) provides the foundation required for intermediate and advanced MSC.Nastran applications.

By far the easiest way to get resources efficient and productive with MSC.Nastran and MSC.Patran.

Prerequisites:

- No previous finite element analysis or computer-based modeling experience is required
- A basic knowledge of statics and strength of materials is highly recommended.

For more information on this course please contact Riekert Leibbrandt 0861237837 or email r.leibbrandt@esteq.com

Topics:

- Introduction to finite element theory
- Introduction to finite element modeling
- Anatomy of the MSC.Nastran input file
 - MSC.Nastran statements
 - File management section
 - Executive control section
 - Case control section
 - Bulk data section
- Model generation and verification using MSC.Patran
- MSC.Patran overview
- Defining the analysis reference system: global versus local
- Creating geometric representation of structure
- Defining material and element properties
- Constraining the model
- Loading the model
- Meshing the model
- Model verification
- Generating and submitting a ready-to-run MSC.Nastran input file
 - Linear static analysis
 - Introduction to linear static analysis
 - Input entries required for linear static analysis
 - Normal modes analysis
 - Introduction to real eigenvalue analysis
 - Input entries required for real eigenvalue analysis
 - Buckling analysis
 - Introduction to linear buckling analysis
 - Input entries required for linear buckling analysis
- Importing an existing MSC.Nastran input file into MSC.Patran
 - Anatomy of the .f06 file: MSC.Nastran output interpretation
 - Results visualization using MSC.Patran
 - Managing results visualization using viewports
 - Animated deformation and stress contour plots
 - Using visualization tools for model validation
 - User-derived results from MSC.Nastran results
 - X-Y plots of user-selected results
 - Model debugging tools and recommendations
 - Tips for solving large problems

Receive 3 CPD Points
on this course.

