Femap with NX Nastran Structural Analysis Toolkit

Fast and efficient post-processing of Nastran results and Nastran modal-based response dynamics simulation combined into one ideal Nastran best-practice toolkit

fact sheet

Siemens PLM Software

www.siemens.com/plm/femap

▶ **Summary**

Femap® with NX™ Nastran® Structural Analysis Toolkit software features an advanced post-processing capability for general purpose Nastran analyses. It also includes efficient, state-of-the-art analytical tools for conducting Nastran modal-based response dynamics simulation.

Benefits

Significantly speeds up the analysis and synthesis tasks of Nastran results

Efficiently processes Nastran results for large models over many subcases

Efficiently processes static, transient, normal modes and frequency response analysis data

Eliminates tedious pre- and post processing of Nastran dynamics solution data

Increases the accuracy of results for sine and random vibration simulations

Saves valuable analysis time by organizing data from numerous large datablocks into meaningful summaries

## Best-practice Nastran results processors

Stress and margin of safety. The stress processor reads element stresses and calculates margins of safety based on user-defined element groups, material allowable, safety factors and failure theories. The minimum margins of safety for complex structures made of different materials subjected to various loadings can be assessed more efficiently than ever before. Margins are exported to MS Excel worksheets and ranked by increasing value, by element label or both and can be sorted easily with the sort-buttons created by the Structural Analysis Toolkit. Supported failure theories include von Mises stress, honeycomb sandwich panel (intra-cell buckling, shear crimping and wrinkling) as well as all the laminate failure theories found on the PCOMP data entity directly.

Element force. The element force processor tabulates Nastran element forces according to user-defined subcases and element groups. It allows for efficient evaluation of the maximum forces occurring in the entire FE model or in selected regions. Group summaries identify the maximum force components along with the associated element, subcase and consistent forces. Since the forces are written directly to MS Excel files, ranking can easily be performed with the sort-buttons created by the Structural Analysis Toolkit.

Grid point force. The grid point force processor tabulates Nastran grid point forces according to user-defined subcases as well as element and node groups. It includes an option to define structural joints (for composite material joints, laminate joints, weld calculations, etc.) and associated allowable load vectors, so that joint margins of safety can be computed. Nodal and overall joint margins are calculated and provided. This permits efficient assessment of bolted, welded, and bonded joint integrity in large models, over many subcases. The grid point forces can be calculated for all dynamic and transient Nastran solutions.

Modal summary. Understanding normal modes of vibration may be easy for simple structures, but for complex structures this knowledge is difficult to assimilate. The following criteria can be used to assess the importance of global and/or local modes of a structure:

• Effective mass
• Response of the structure to a base excitation

VELOCITY SERIES   **SIEMENS**
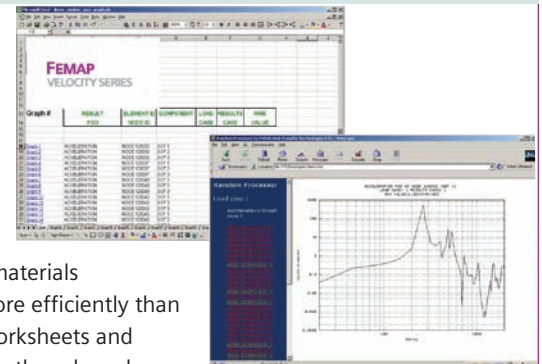
Features

The interfaces help you select nodes and/or elements, subcases, formatting options, and other results processing parameters. Session files store processing options and group definitions for future processing

Interfaces directly with Femap and can extract node and element group information for the active model

Direct link to Microsoft Excel. Cut and paste operations are no longer required. The Structural Analysis toolkit formats the Nastran results within MS Excel automatically, so you don't have to spend time creating reports

Efficiently reads Nastran binary results files generated on all Windows and Unix platforms

Sorts and identifies maximum element stresses, grid point forces, element forces, energy density, and other Nastran results. Calculates element and margins of safety automatically from the chosen failure criteria and parameters

Hardware platforms and file formats

The Structural Analysis Toolkit is available on Windows 2000 and XP platforms

Reads all NX Nastran and MSC.Nastran .op2 software files that were created on Windows and Unix platforms for all versions since NX Nastran Version 1 and MSC.Nastran Version 70.7

The Nastran modal summary tool processes the modal information from a normal modes analysis. Effective masses are tabulated and graphed in MS Excel worksheets directly. Acceleration responses for selected groups of nodes are also tabulated. Critical modes, in which effective masses and/or dynamic responses exceed user-defined thresholds, are automatically flagged by the processor and highlighted within the MS Excel worksheets.

Mass summary. The mass processor computes the mass properties of a Nastran finite element model, allowing for efficient comparison with the detailed mass budget. The mass processor will scan the Nastran results file, identify all the physical property tables and calculate the structural and non-structural mass of all the elements associated to each table. Optionally, it will calculate the mass properties of selected element groups as well. MS Excel graphs and pie charts will show the FE model's mass distribution.

Random vibration simulation. The random processor reads the results of a Nastran normal modes analysis and evaluates the responses of a structure subjected to a random base acceleration. It efficiently replaces all the steps following the initial Nastran normal modes solution. Features of the random processor include:

- Computation of stress margins of safety, including the consideration of multiple failure criteria (yield, ultimate) in a single run
- True von Mises stresses are computed for critical elements based on threshold
- Efficient integration methods for fast calculations
- Automatic HTML graphical result creation
- The large mass is not required
- Automatically calculates an appropriate frequency resolution
- Efficient residual flexibility method of accounting for modal truncation
- Tabular output is provided as ASCII text files or MS Excel spreadsheets
- Graphical XY (results vs. frequency) output is available as either plots linked to an HTML page or a MS Excel spreadsheet.

Sine vibration simulation. The sine processor is similar to the random processor except that it sets up a harmonic base acceleration analysis. It features phase-consistent calculation of maximum von Mises stresses. MS Excel, HTML and ASCII reports are generated automatically.

Direct data interfaces
- Writes Microsoft Excel files
- Writes HTML reports
- Writes ASCII text files
- Interfaces directly with Femap Version 8 and later

▶ **For more information, contact your local Velocity Series™ portfolio representative:**

▶ **Contact**
Siemens PLM Software – www.siemens.com/plm/femap

Americas 800 807 2200
Europe 44 (0) 1202 243455
Asia-Pacific 852 2230 3308

SIEMENS