Femap with NX Nastran Structural Analysis Toolkit

Fast and efficient post-processing of Nastran results combined with 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

Features

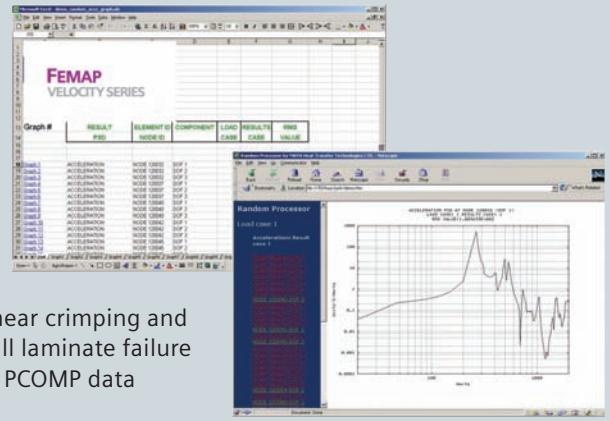
- Select nodes and/or elements, subcases, formatting options and other results processing parameters; leverage session files to store processing options and group definitions for future processing
- Interface directly with Femap and extract node and element group information for the active model

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.

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 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.

VELOCITY SERIES

www.siemens.com/velocity

SIEMENS

Femap with NX Nastran Structural Analysis Toolkit

Features *continued*

- Directly link to Microsoft Excel and eliminate cut and paste operations; also eliminate time spent creating reports by leveraging Structural Analysis Toolkit to format Nastran results with Excel
- Efficiently read Nastran binary results files generated on all Windows and Unix platforms
- Sort and identify maximum element stresses, grid point forces, element forces, energy density, and other Nastran results; calculate 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
- The toolkit reads all NX Nastran and MSC.Nastran .op2 software files created on Windows and Unix platforms for all versions since NX Nastran Version 1 and MSC.Nastran Version 70.7

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 can 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

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 computed for critical elements based on threshold
- Efficient integration methods for fast calculations
- Automatic HTML graphical result creation
- No large mass requirement
- Ability to automatically calculate an appropriate frequency resolution
- Efficient residual flexibility method of accounting for modal truncation
- Tabular output provided as ASCII text files or MS Excel spreadsheets
- Graphical XY (results vs. frequency) output 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

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

www.siemens.com/velocity

X4 11561 11/10 C