**Overview**

Small and medium sized businesses face the same challenges as large enterprises in lowering costs and reducing product development time.

The greatest challenge a Mechanical Engineer faces is the creation of efficient and cost effective designs, within budgetary and time constraints.

With MSC.Nastran™ for Windows®, the engineer can quickly and accurately predict the behavioral response of a conceptual design, working entirely within a familiar Windows look and feel Virtual Product Development (VPD) environment.

**What is MSC.Nastran for Windows?**

MSC.Nastran for Windows is a powerful, general purpose finite element analysis program with an integrated graphical user interface used by engineers and designers around the world to analyze the stress, vibration, dynamic, nonlinear, and heat transfer characteristics of structures and mechanical components.

**What is FEA?**

FEA (finite element analysis) is the numerical simulation of the behavior of mechanical components. Industries already successfully utilizing MSC.Nastran for Windows include:

- Aerospace
- Electronics
- Automotive
- Heavy equipment
- Bicycle component design
- Semiconductor
- Consumer products

**Capabilities**

- **Model Preprocessing**
  - Comprehensive CAD integration allowing model import from all of the major CAD packages
  - Variety of methods for applying element properties, constraints, loads, and more

- **Engineering Analysis**
  - Robust structural analysis capabilities involving several engineering disciplines including linear statics, dynamics, heat transfer, nonlinear, and optimization
  - Multi-discipline analysis - utilize the results of one kind of analysis as input to another analysis type
  - Complex problem analysis involving nonlinear materials, geometric behavior, and contact

- **Results Postprocessing**
  - Advanced visualization techniques to gain complete insight of product performance

- **Open Architecture**
  - Customizable graphical user interface and analysis process

**Benefits**

- Reduce the time required to develop high-quality engineering analysis models, increase productivity, and reduce risk with proven simulation tools
- Achieve tighter collaboration between design and analysis engineers through comprehensive CAD system data integration allowing model import from all of the major CAD packages
- Improve designs by understanding how they function in operating environments that involve nonlinear materials, geometric behavior and contact
- Gain access to the power of integrated computer-aided engineering at an affordable bundle price
MSC.Nastran™ for Windows

• **EXPERT MODELING SYSTEM**
The MSC.Nastran for Windows Expert Modeling System includes basic structural, normal modes and buckling analysis capabilities using the MSC.Nastran 2004 solver. Additional modules are available for:
  - Heat transfer
  - Dynamic response
  - Nonlinear
  - Sizing optimization
  - Nonlinear+ advanced nonlinear capabilities from MSC.Marc®

• **EXPERT MODELER CAPABILITIES**
  - Geometry creation and modeling
  - Surface modeling
  - Automatic and mapped meshing
  - Multi-surface meshing
  - Tetrahedral solid meshing
  - Hexahedral meshing with solid subdivision
  - Import and export of ACIS, DXF, IGES, Parasolids, STEP, and STL geometry formats
  - Direct import of CATIA, I-DEAS, ProEngineer, Solid Edge, and Unigraphics data
  - Solid mid-plane extraction tools
  - Import data from most FEA systems
  - Line, plate and solid element types
  - Isotropic, orthotropic and anisotropic materials
  - Multiple libraries of properties, materials and functions supplied
  - User customizable property, material and function libraries
  - Beam cross section calculator
  - API customization tools

• **BASIC ANALYSIS**
  - Linear statics
  - Normal modes
  - Buckling

• **NONLINEAR ANALYSIS**
  - Static and transient
  - Large displacement
  - Nonlinear materials
  - Nonlinear elastic, plastic, hyperelastic, and creep
  - Gap and sideline contact
  - Nonlinear buckling

• **HEAT TRANSFER**
  - Steady-state and transient conduction, convection, and radiation
  - Temperature dependent material properties and loads

• **DYNAMIC RESPONSE**
  - Transient response
  - Frequency response
  - Random vibration
  - Enforced motion
  - Response spectra generation and analysis

• **SIZING OPTIMIZATION**
  - Minimize weight
  - Static analysis - multiple load cases
  - Plate, bar, and rod property sizing
  - Displacement, stress, and strain constraints

• **NONLINEAR+ BASE MODULE**
  - Linear statics
  - Normal modes
  - Buckling
  - Deformable - deformable contact

• **NONLINEAR+ PLASTICITY AND ELASTOMERS MODULE**
  - Elastic-plastic
  - Elastomers
  - Mooney-Rivlin
  - Ogden
  - Foam

• **EXTENSIVE RESULTS VISUALIZATION**
  - Deformed, contour, iso-surface, section cuts, and vector plots
  - Dynamic query
  - Animated results
  - Shear and bending moment diagrams
  - X-Y plots customizable results
  - User-defined results calculations and load combinations
  - Intelligent results processing for discontinuities
  - Enhanced results display for shell structures
  - Free body diagram and nodal force display
  - MSC.Nastran Binary OP2 results support

• **SYSTEM REQUIREMENTS**
  - Pentium class processor or higher
  - Windows 2000, Windows XP
  - 128 Mb RAM
  - 600 Mb disk space
  - SVGA graphics card with OpenGL support

---

Corporate
MSC.Software Corporation
2 MacArthur Place
Santa Ana, California 92707
Telephone 714 540 8900
www.mscsoftware.com

Europe, Middle East, Africa
MSC.Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone 49 89 431 98 70

Asia-Pacific
MSC.Software Japan LTD.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjuku-Ku
Tokyo, Japan 160-0023
Telephone 81 3 6911 1200