

NX I-deas MasterFEM

Complete standalone capabilities for creating FE models and evaluating simulation results

Benefits

- Edit models and refine data prior to analysis to build an effective model
- Share and archive data with ease via PDM-supported FEMs in libraries
- Develop the model using a full range of tools, including grouping and extensive loading and boundary conditions
- Verify models before processing with a complete set of graphical and mathematical tools that help check model suitability
- View analysis results quickly and easily with a dynamic visualization tool
- Use extensive post-processing tools to continue the iterative phases of analysis or to export/import information

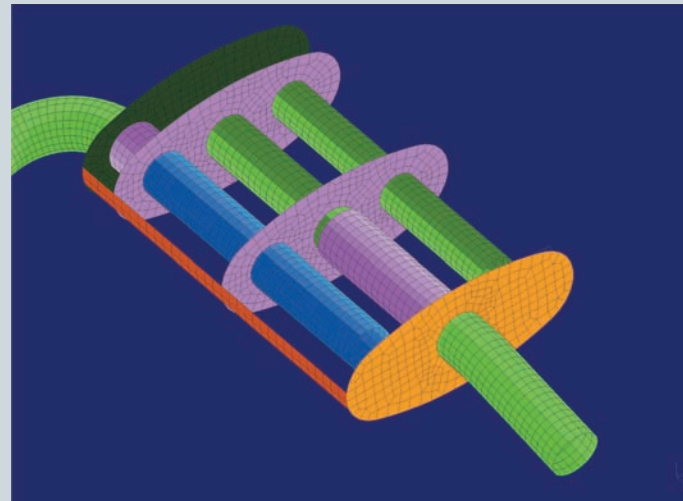
Summary

NX™ I-deas™ MasterFEM software is the standalone CAE application within NX I-deas that serves as the Siemens core simulation package for building finite element models and visualizing the analysis results. Multiple add-on solution modules such as NX Nastran are available, but this product contains the core functionality for pre- and post-processing of finite element models needed to simulate product behavior. MasterFEM's comprehensive capabilities include advanced geometry creation and abstraction, FE model preparation, loads and boundary condition development, meshing and model verification. And flexible post-processing tools make the iterative nature of finite element based simulation a user-friendly, reliable and faster process. Interfaces are included for standard geometry exchange formats and add-on interfaces for a complete range of third-party CAE and CAD systems are also available.

A comprehensive FEM system

NX I-deas MasterFEM provides comprehensive capabilities for building finite element models and reviewing and exporting analysis results. MasterFEM uses geometry directly from NX I-deas modeling and assembly packages. Robust CAD translators enable non-native NX I-deas geometry to be imported for use in MasterFEM. A variety of add-on packages furthers the finite element analysis possibilities within MasterFEM.

MasterFEM includes the fundamental modeling functions of automatic and manual mesh generation, application of loads and boundary conditions and model development and checking. A robust set of



NX

www.siemens.com/nx

SIEMENS

NX I-deas MasterFEM

Features

- Interfaces to more than 20 major analysis programs and a wide variety of CAD systems
- Accesses and manipulates materials data through a relational database
- Employs a wide range of automatic, semi-automatic, or manual techniques, including free, mapped or bottom-up meshing that can be performed in combination with one another
- Abstracts CAD topology into a suitable meshing domain using section meshing techniques
- Enters physical and material properties interactively
- Stores materials information in a relational database for future use

visualization tools quickly generates displays, enables multiple results to be viewed simultaneously and easily prints the display. In addition, extensive post-processing functions enable review and export of analysis results to spreadsheets and provide extensive graphing tools for gaining an understanding of results.

Getting and using geometry

Wireframe, surface and solid geometry from other CAD systems can be accessed through embedded, standards-based (IGES, VDA, SET, STEP) or optional direct

CAD interfaces. MasterFEM directly uses the wireframe, surface and solid geometry capabilities of NX I-deas Master Modeler and Master Assembly for construction of finite element models.

Often design geometry must be modified to build an effective model. Details may need to be suppressed or eliminated, additional geometry may be required to control mesh density, or surfaces not present in the geometry may be needed for meshing.

MasterFEM provides full-blown model editing capabilities, including:

- Interactively suppress or eliminate features defined in Master Modeler
- Add, modify, or delete entities (points, curves, surfaces, volumes) in more than 20 ways
- Extract mid-surfaces automatically in Master Modeler for accurate results using shell finite elements. Shell element thicknesses are then obtained directly from the mid-surfaces
- Analysis data is associated with original part geometry through NX I-deas Team Data Manager. (If the original part is updated, the simulation user is notified of changes so that appropriate updates can be made)
- Unique volume partitioning capabilities allow for the quick segmentation of geometry for mesh control or boundary conditions definition

State-of-the-art mesh generation

Building a mesh that describes the stiffness properties of a structure is typically the most time consuming and labor-intensive part of the FEM process. In addition, the selection of element type and proper mesh density to ensure efficiency and accurate results is a necessary part of the "art" of finite element analysis.

Free meshing

MasterFEM free meshing generates a user-controlled distributed mesh on surface and solid geometry and can produce millions of elements. Surfaces can be defined by an unlimited number of curves; solids, bounded by an unlimited number of faces. MasterFEM can directly free mesh Master Modeler geometry in a single step.

Free meshing capabilities include:

- Automatic meshing of surfaces and of volumes, with no topology restrictions
- Surface meshing with linear or parabolic quadrilaterals, triangles or quadrilateral dominant meshes that insert triangles in a quadrilateral mesh to automatically reduce element distortions
- Solid meshing with linear, parabolic or p-element tetrahedra
- Fits p-elements to underlying geometry (up to fifth-order representation)
- Can define allowable distortion for tetrahedral elements before meshing
- Recognizes anchor nodes used to control positioning of resulting nodes
- Recognizes reference geometry, points and series for mesh definition
- Enables geometry-based definition and generation of lumped masses, rigid bars, spring, gap and damper elements
- Enables automatic transition of mesh density
- Automatically defines local density
- User-specified global density with a single entry
- Develops surface mesh transitions with all quadrilaterals
- Provides automatic optimization-based smoothing for minimum element distortion
- Associates mesh generation settings with geometric features (updates occur with design geometry changes)

CAD interoperability

- Siemens offers a unique ability to integrate FEM analysis seamlessly within the CAD environment. Within NX CAD, the ability exists to bring CAD part and assembly data directly into MasterFEM and vice versa. The transfer of data is just another tool to enhance the product development cycle.
- When an NX part is brought into MasterFEM via interoperability, a node is added to the part's history tree. This node will go out of date when the NX part changes. Then the MasterFEM model can be updated so that it remains in synch with the CAD part. However, if the CAD changes have no impact on the FEM, then the analyst can choose to not update the FEM geometry to the new CAD geometry. The decision to update is left to the analyst, not the software.
- This direct interoperability offers superior geometry translation from one package to another. Alternative methods (i.e., using a translator) simply transfer a snapshot of the part geometry and don't allow for tracking of the CAD part. The seamless interoperability between MasterFEM and the CAD environment is much more effective.

Adaptive meshing

To reduce mesh-induced analysis errors, MasterFEM provides unique capabilities for automatic modification of mesh density, element order and element shape. An analyst can begin with a coarse mesh and enable adaptive meshing to refine the model only in areas where more elements are needed for accuracy.

- Adaptation is based on any scalar analysis result quantity used as error measure
- Element distortion may be considered as part of the adaptation process
- Selective node movement, element subdivision (with triangles or quadrilaterals) and re-meshing provided for shell models
- Node movement and selective re-meshing provided for solid elements
- P-element adaptivity provided for linear structural analysis using tetrahedral elements
- Selective p-order update provided for efficient solution
- Ability to freeze movement of selected nodes
- Automatic and adaptive meshing, using a combination of r, h and p adaptivity, provides fast, accurate analysis with a minimum of user interaction

Mapped meshing

Mapped meshing generates uniformly shaped elements on simple surface regions and in simple volumes. Detailed capabilities include:

- Linear, parabolic, quadrilateral or triangular elements on 3- or 4-edged surface regions
- Linear, parabolic, hexahedral (brick) or wedge elements in 5- or 6-faced volume regions
- Each edge, section or surface for mesh definition can be composed of multiple curves to simplify geometry construction and segmentation
- Degenerated regions are automatically handled with triangles or wedges
- Generate brick meshes on extrudable/revolved type volumes (The swept mesh follows the geometry through the extruded/revolved direction, and also works for unstitched geometry when section meshing is used)
- User-defined biasing to enable mesh skewing

- Transitions can be defined for surface meshes merely by specifying a different number of elements on the opposite sides of geometric regions
- Automatic transition of a number of elements across surface regions
- Mapped mesh regions developed with partitioning are associative to geometry, so mapped meshes automatically update with geometry changes
- Mesh density specification is automated through user prompting

Abstracting CAD for meshing purposes

Often, CAD topology contains details that are of no use to the analyst. Sliver surfaces, detailed embosses (for example "Made in USA"), small fillet radii and small holes are examples of details the analyst may not wish to mesh. Section meshing tools provide a solution to these problem areas without affecting the CAD in any way.

Section meshing lets the user define and generate a mesh on a collection of surfaces rather than on each individual surface. It provides the means to let the analyst mesh the CAD topology at a level of detail that sufficiently captures the design intent relevant to a particular FE analysis. CAD features may be effectively removed without modifying the CAD. In addition, it enables the analyst to mesh (as free or mapped surfaces, volumes) unstitched geometry imported from external CAD.

Meshing geometry without CAD

Sections-on-mesh technology lets the user re-mesh existing meshes or STL data without the presence of CAD geometry. The finite element mesh takes the place of CAD geometry. Free and mapped meshing of shells and solids are available with sections-on-mesh techniques.

Bottom-up meshing

While free and mapped mesh generation are useful for most tasks, bottom-up creation of nodes and elements can often be the easiest way to build or edit certain types of models. Meshes, including transitions, can be created directly between wireframe curves without defining surfaces. Surfaces are created as required and resulting models can be stitched into valid solids.

Material repository: MDS

Within NX I-deas, the Material Data System (MDS) offers you a very efficient and flexible means of specifying, modifying and displaying the mechanical, thermal and rheological properties required for mechanical product design, simulation and manufacturing applications. MDS is a robust repository for all necessary material characterizations that provide the ability to search for, display, archive, update and delete material properties and related information.

MasterFEM provides a wide range of tools for building/editing finite element models directly (bottom-up), independent of geometry, including:

- Extrude and revolve capabilities enable creation of quadrilateral shell elements from an arbitrary set of beams or hexahedral elements from quadrilateral shells (Path options include general curve local coordinate systems and twist)
- Copy and reflect capabilities enable sets of nodes and elements to be replicated
- A surface-coating command generates beams on the edges of shell elements or shells on the faces of solids
- Nodes and elements can be created and modified individually:
 - Splitting and combining elements
 - Modifying element connectivity
 - "Stitching" unconnected meshes
 - Rotating and moving nodes and elements
 - Elements may be projected onto surfaces (Optionally, element projection may project to a location that is a percentage of the distance between the elements and the surface)

Editing operations are interactive (including element split and combine; drag, move and replace nodes) and can maintain geometry associativity when desired. Nodes can be directly created on curves or projected onto a plane.

Combined methods

Automatic or manual mesh generation methods can be combined on a particular model. This allows the use of the most effective approach for each situation.

Complete model development tools

After mesh generation, a number of tasks are needed to prepare a model for analysis. MasterFEM enables a user to fully develop the model and verify its adequacy for purposes using mathematical and interactive graphic tools.

MasterFEM tools let the user define individual component FEMs. Components can then be combined into a system level FEM that relies on a CAD assembly for positioning data. Each instance in the CAD assembly defines an instance in the FEM. This "instancing" of FEMs enables the seamless transfer of assembly data back and forth between CAD and MasterFEM.

Element library

A complete library of finite elements enables the user to perform many types of analysis and modeling quickly and efficiently. More than 50 elements are provided, including linear and parabolic forms of shells and solids, axisymmetric shells and solids, beams, rods, springs, dampers, masses and gaps. Scalars and other special elements have unique graphic symbols. P-elements (solid tetrahedra) are supported for linear structural analysis.

Dynamic navigation

Dynamic navigation functions are used for easy interaction, such as pre-highlighting of entities and filtering for control of active entities that are highlighted and selected. A user may also use bounding box and logical (e.g., entity relationship) selection. Dynamic navigation makes the selection process extremely user friendly.

Grouping

Grouping enables definition and manipulation of a model's logical subsets for display, property assignment and result calculation. Groups can be defined by interactive screen selection of any and all entities; or defined by any entity attribute such as model geometry, color, model checking results and entity relationship (e.g., nodes associated with elements, physical and material property identification). Set operations can be used to join, intersect or show the difference between groups.

Dynamic groups help define a simple "recipe" for element groupings based on physical and/or material property and/or color. Optionally, related nodes, geometry and boundary conditions may be included in a dynamic group.

Coordinate systems

Cartesian, cylindrical or spherical coordinate systems can be used to define desired motions globally and locally or look at specific results such as radial and tangential stress. Material coordinate systems are defined for orthotropic and anisotropic materials. Material orientation tools, which are especially useful for composites, are provided for associating element coordinate systems to geometry.

Physical and material properties

Physical properties such as shell element thickness and spring stiffness values can be entered interactively and are stored in the model file database. Properties extracted from and associated with geometry can be applied to elements at the user's request. Property masking is available for entry of data specific to particular analysis programs.

Beam section properties

Beam section properties may be defined from a catalog of standard sections. User-defined Master Modeler wireframe sections may also be used to define a beam section for use in FEM.

Solver interfaces

- A wide variety of interfaces are available for more unusual types of analysis. Siemens provides direct interfaces and more than 20 vendors have prepared interfaces to MasterFEM.
- Direct interfaces are available to Abaqus, Ansys and Nastran. Toolkit interfaces to PAM-Crash, LS-DYNA and RADIOSS that transfer preprocessing data to/from MasterFEM are also available.
- Vendor-written and supplied interfaces to programs for nonlinear structural, acoustic, thermal, electromagnetic, manufacturing process and flow analyses are provided through the NX I-deas Solutions Network.
- User interfaces are available through ASCII NX I-deas universal files or through Open NX I-deas.

Loads and boundary conditions

MasterFEM provides extensive capabilities to define loading and boundary conditions to correctly simulate operating environments:

- Loads can be defined on and associated with geometry point-, edge-, curve-, surface- or section-based
 - Independent of mesh but applied to nodes and elements for analysis at user request
 - Functional variation using mathematical expressions
 - Surface fit through defined points
- Restraints defined on and associated with geometry
 - Point-, edge-, curve-, surface- or section-based
 - Standard restraints such as pin, slider and ball directly available for application
- Automatic definition of geometry-based contact

- Constraints and restraints, including nodal displacement and master DOF
- Structural loads
 - Nodal forces and temperatures
 - Element face (normal and traction) and edge pressures
 - Acceleration (gravity, translation, rotation)
 - Ambient and reference temperatures
- Heat transfer loads
 - Nodal and distributed heat sources
 - Face and edge fluxes, convection and radiation
- Multiple load and restraint sets defined and stored for use in analysis
- All loads and restraints displayed with unique graphical symbols
- Associativity of geometry-based loads and restraints is maintained through design geometry changes
- Define time-varying loading and boundary conditions to correctly simulate nonlinear loading conditions
- Nonlinear and direct frequency response loads and restraints defined as discrete tables of values or functions
- Scaled graphical displays provide verification of loads and boundary conditions

Complete model checking tools

Analyzing a model with errors can be time-consuming and expensive, and errors are often not detected even after analysis.

MasterFEM provides a full set of graphical and mathematical tools to help verify that a model is complete and correct before submitting it for solution:

- Coincident node and element checks eliminate duplications
- Free-edge and face checks avoid unwanted cracks in a model
- Shrink element display verifies that elements are located properly
- Coordinate system and element connectivity displays assure correct orientation of beams and shell normals. (Element connectivity can be made consistent automatically by specifying seed elements for control)
- Mass and inertia property calculations check physical and material properties
- Rigid and constraint element checks for loops and double dependencies
- Dangling checks for elements such as rigids, gaps, springs, etc., that are not connected to a mesh on both sides of the element

- Interference check of shell and solid meshes
- Interference fix for shell meshes
- Element shape checks (distortion, warping, etc.) verify that elements do not violate limits and can produce accurate results

In addition, display capabilities help the user perform model checks graphically, verifying the completeness of the model's physical/material properties, shell element thicknesses and mesh quality. There is also a graphical check for pressure and temperature loads, as well as a summation tool that sums up the total loads applied to the FEM.

Getting the best results from analysis

For the mixed science and art of analysis to impact design decisions, results must be presented in an understandable form. MasterFEM provides extensive graphics and manipulation capabilities that focus on critical data and present it for review and action. A comprehensive and flexible methodology has been adopted to enable the user to act before, during and after FEM solutions are sought. Post-processing capabilities include an advanced visualization tool, Visualizer, plus a wide range of additional tools.

Visualizer

MasterFEM's Visualizer helps an analyst make expert decisions about the results of a finite element analysis. Visualizer generates displays quickly, allows viewing of multiple results simultaneously and enables the user to easily print the display. The user can also query results with a real-time feedback odometer. Visualizer can directly interface to Microsoft Excel for reading and writing results.

Using Visualizer, the user can:

Create:

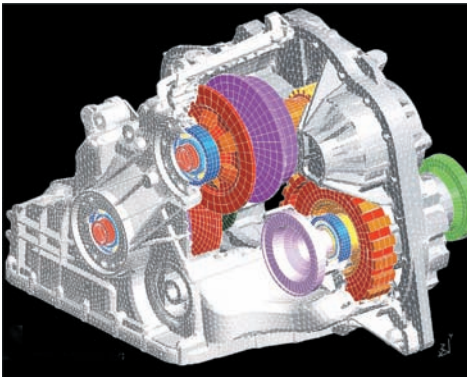
- Animated, stepped or smooth-shaded displays
- Movie files in SGI, avi and mpeg format
- Cutting plane, contour, element and arrow displays
- Templates of display options for repeated use

System requirements

NX I-deas MasterFEM shares
NX I-deas system requirements

Recommended system configuration

For information on particular operating systems or graphics cards, please visit
http://support.ugs.com/online_library/certification/



Control:

- How data is displayed (for example, data components and coordinate system to use or averaged and unaveraged data)
- Text, headers and colors
- Moving, copying and deleting results

View:

- Multiple results simultaneously
- Results in multiple viewports
- Areas with specific result values
- Deformed geometry

Insert:

- 3D probe results annotations
- 3D and 2D text annotations

Export:

- Displays for report-ready printing/plotting as CGM, PostScript, jpg, PNG or VRML files
- Single or multiple result sets to spreadsheets or directly to Excel (Windows only) for further manipulation

Import:

- Modified result sets back from Excel (Windows only) or a spreadsheet text file.

Additional post-processing tools

After analysis, MasterFEM offers a wide range of post-processing tools to further the job of simulation. Results can be reported or exported to other applications. New inputs can be fed back into the analysis database for new solutions. And a variety of output can be generated for future reference.

Specific capabilities include:

- Combine and scale results for load case combination and “what if” studies
- User-defined result computations via mathematical functions for user-defined, failure criteria evaluation or other types of evaluations
- Laminate post-processing
 - Generate laminate ply results from shell stress resultants or shell strain results
 - Ply stress/strain results
 - Ply failure criterion results
- Beam post-processing
 - Graph beam diagrams (shear, moment, etc.) on the finite element models
 - Generate contour on beam cross section displays
 - Generate component beam stress displays (axial, shear, torsion, etc.) from beam stress results or element force results
 - Generate computed beam stress contour displays from beam element force or beam stress results
- Graphing
 - Produce graphs of results
 - Along an edge
 - Based on a selection of nodes or elements
 - At a single node or element over multiple results
 - Overlay multiple graphs for comparison

Contact
Siemens PLM Software
Americas 800 498 5351
Europe 44 (0) 1276 702000
Asia-Pacific 852 2230 3333

www.siemens.com/nx

© 2011 Siemens Product Lifecycle Management Software Inc. All rights reserved. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Geolus, GO PLM, I-deas, Insight, JT, NX, Parasolid, Solid Edge, Teamcenter, Tecnomatix and Velocity Series are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. All other logos, trademarks, registered trademarks or service marks used herein are the property of their respective holders.
X1 2565 2/11 B