NX Digital Simulation: What's new in NX 7.5

Benefits
- Redefines CAE productivity by integrating leading geometry tools with powerful new analysis technology for modeling, simulation, automation and test correlation
- Synchronizes simulation with design so analysts can keep pace with design iterations, make better engineering decisions and deliver better products faster
- Improves productivity by introducing new workflows for working with complex geometry, meshing thin-walled parts and beam and bolt modeling , as well as for performing other functions
- Eliminates data transfer errors and speeds multi-physics workflows by integrating the broadest set of solver technologies within a single environment
- Expands integrated multi-physics solutions to include durability and motion analysis with flexible bodies along with even more solutions for structural, thermal and flow analyses

Summary
NX 7.5 significantly enhances the NX™ Digital Lifecycle Simulation portfolio with increased modeling efficiency and an expanded range of integrated solutions. New product offerings in NX 7.5 include flexible body dynamics, durability analysis and model correlation. In addition, NX 7.5 offers major enhancements across the existing product portfolio in the areas of multi-body dynamics, FE modeling of thin-walled structures, beam modeling, meshing, laminate composites and support for multi-physics solvers.

Multi-body dynamics

NX Motion Simulation-RecurDyn NX Motion Simulation-RecurDyn provides an advanced yet simple-to-use solution that enables designers and engineers to understand, evaluate and optimize the complex motion behavior of assemblies and products. When used early in the design process, designers can quickly understand if their assembly will encounter package space issues or interfere with parts inside or outside of the entire product. New improvements to NX 7.5 for motion analysis include:

Modeling and simulation
- Static equilibrium result as input to dynamic solution
- 2–3 joint coupler
- Contextual menu for motion objects

Import/export
- Co-simulation launched directly from MATLAB/Simulink
- Ability to import Tecnomatix® Process Simulate Kinematics models

Post*processing*
- New bar graph option
- Assembly arrangements captured during animation

NEW Flexible body dynamics NX 7.5 introduces support for flexible body dynamics to combine both elastic
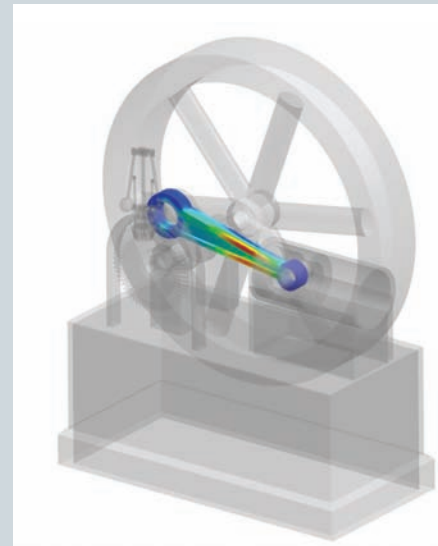
NX

SIEMENS

NX Digital Simulation: What's new in NX 7.5

Benefits *continued*
- Strengthens confidence in a model's ability to predict performance by introducing new tools to correlate digital models with measured test data
- Saves money and development effort by reducing the need for physical testing

deformation and rigid body motion. Flexible bodies are important because they change the geometry of the mechanism and can lead to serious design issues. With flexible body dynamics, you can evaluate conditions such as sharp impacts, sudden changes in motion and the effect of component flexibility on mechanism performance and durability.

Flexible body analysis requires NX Motion Simulation with the RecurDyn solver and NX Advanced Simulation with the NX Nastran solver.

Finite element modeling and analysis

NX Design Simulation NX Design Simulation delivers design-integrated structural and thermal simulation tools that help engineers compare design alternatives and optimize performance characteristics of products from the earliest stages of the design process. New improvements to NX 7.5 for Design Simulation include:

Preprocessing
- Re-usable regions for contact and glue definitions
- FEM creation from visible bodies
- Control over maximum tetrahedral mesh size for undo operations

Materials
- Plot tabular material property values using the XY graphing capability
- Units specifiable for any material property

Postprocessing
- Display of component-based deformation
- Ability to add user-derived results to existing results for the same model
- Control over the numeric format used for results values
- Node selection to define the center of rotation for post view displays

Teamcenter® software integration
- Simulation search view

NX Advanced FEM and NX Advanced Simulation NX Advanced Simulation combines the power of an integrated NX Nastran® desktop solver with NX Advanced FEM, a comprehensive suite of multi-CAD FE model creation and results visualization tools. Extensive geometry creation, idealization and abstraction capabilities enable the rapid development of complex 3D mathematical models that allow design decisions to be based on insight into real product performance. New improvements to NX 7.5 for Advanced FEM and Advanced Simulation include:

Preprocessing
General
- FEM creation from visible bodies

Display
- Shell offsets included for thickness display
- Persistent display of beam cross-sections as solid or wireframe
- Beam cross-section orientation, end releases and orientation vector displays

Geometry idealization and abstraction
- Color coding for sweepable bodies in split body command
- Mid-surface enhancements, including face pairing improvements and expanded trimming options
- Automated stitching of free edges to faces
- Replay of automatic stitching operations after geometry updates
- Ability to create circular imprints around holes
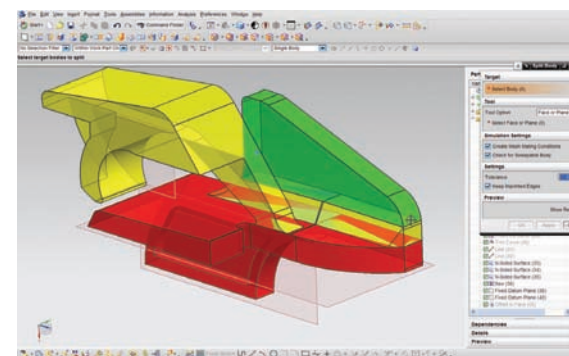
Materials and physical properties
- Material property values defined as an expression or as a field
- Tabular material property values plotted using the XY graphing capability
- Units specifiable for any material property
- Offset assigned to a shell element
- Offset defined to either or both ends of a beam or bar element's cross-section
- Orientation of a beam element cross-section defined by vector or reference node
- Orientation of a spring-damper element (CBUSH) defined by vector or node
- Orientation of a gap element (CGAP) defined by vector, coordinate system or node
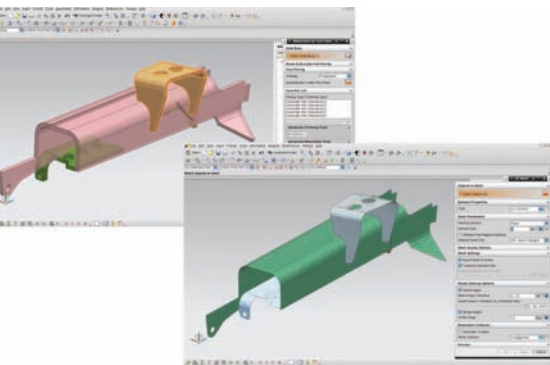
Beam meshing
- New option to ensure beam elements are correctly oriented, properly connected from start to end and numbered sequentially
- Expanded list of standard beam cross-section types
- Consistent workflow for assigning beam cross-section and orientation for all solvers
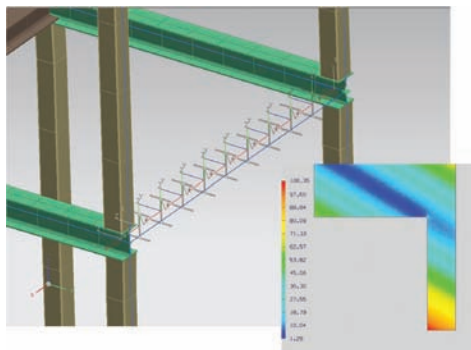
- Offset defined relative to any point on the beam section
- User-defined solid sections for defining nonstandard cross-section shapes
- Persistent display of beam cross-sections as solid or wireframe
- Beam cross-section orientation, end releases and orientation vector displays
- Better information query capability for beam mesh
- End release support for Ansys and LS-Dyna solvers

Other 1D meshing
- New bolt connection command to model bolted connections such as bolts held in place with a nut, bolts in a threaded hole and bolt-type connections represented by only a spider element at the junction plane between two mating bodies
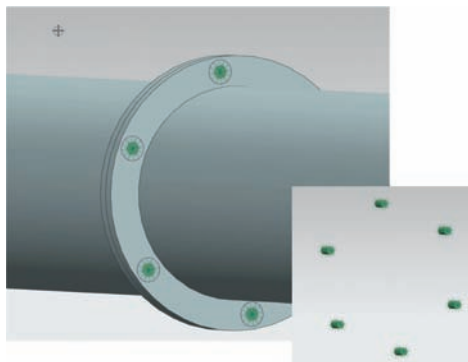
Shell (2D) meshing
- Diagonal consistency improvements for mapped meshes to ensure element diagonal points in a consistent direction across faces
- Connectivity consistency improvements to ensure consistent element connectivity throughout 2D mapped meshes
- New edge mapping option to project vertices along edges of a face to the opposite edge for 2D mapped meshing
- Ability to add or remove faces from an existing mapped mesh
- Associated elements using surface coat command to base 3D mesh to facilitate automatic updating when the 3D mesh changes
- Split elements based on their connectivity to ensure the split occurs uniformly across a large number of elements

Solid (3D) meshing
- Ability to sweep a mesh from multiple source faces
- New edge mapping option to project any vertices along boundaries of the source face or faces to the target face
- Control over the maximum size of a tetrahedral mesh allowed for 'Undo' commands
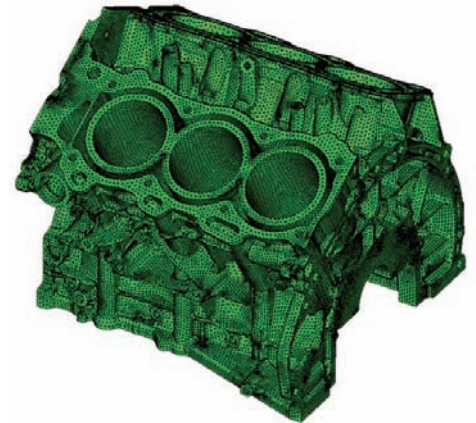
Mesh controls
- Nodal reference coordinate system assignable to nodes or polygon geometry
- Selected elements easily moved out of existing mesh into new mesh
- Deformed mesh based on displacement results from a structural analysis
- Ability to simultaneously modify mesh properties for multiple meshes
- Updates to default quality threshold values in the Nastran solver environment to better reflect values required by the Nastran solver
- New quality checks for pyramid elements, such as aspect ratio, Jacobian ratio and Jacobian zero

Assembly FEM
- Ability to merge coincident nodes between adjacent component FEMs within context of an assembly FEM
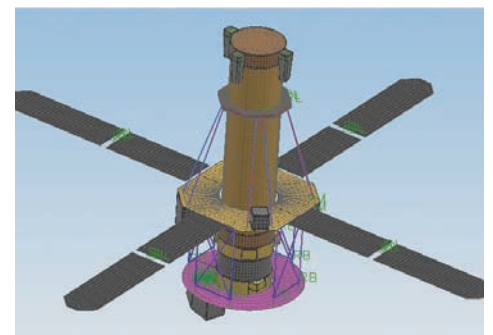
Boundary conditions
- Loads or constraints applied to groups
- Re-usable regions for contact and glue definitions
- Loads, constraints, solutions and other simulation entities importable from a source simulation into a target simulation
- New degree of freedom set to specify a list of nodes and define degrees of freedom individually for each node
- Ability to define contact parameters globally and then override select parameters locally for NX Nastran 7.0
- Expanded support for Abaqus boundary conditions such as contact interference, tie surface and surface based coupling commands
- New CONTA174 ET and CONTA174 Real Constants modeling objects for Ansys

Postprocessing
- Calculation of beam stresses and strains and display results on beam cross-sections
- Display of all stress data in a single post view for models consisting of multiple element types
- Control over numeric format used for results values
- Display of component-based deformation
- Node selection to define center of rotation for post view displays
- Support for Plotel elements
- Ability to add user-derived results to existing results for the same model
- New bar graph option
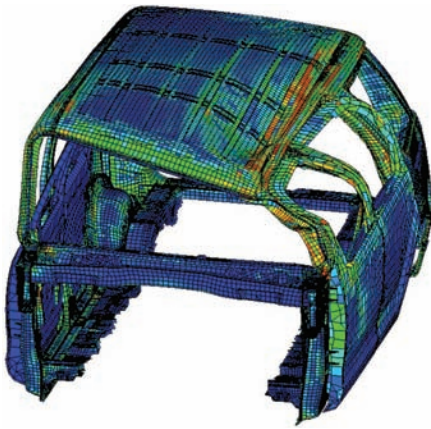
Solver support
General
- Loads or material orientation vectors importable as spatial fields

- Thermal energy units now expressed in inch-lbf units consistent with British base unit system

Nastran environment
- Support for SOL 107 direct complex eigenvalue analysis
- Support for SOL 110 modal complex eigenvalue analysis
- Support for SOL 200 optimization analysis
- Superelement reductions on FE components
- Export support for flexible bodies in motion and control systems software
- Expanded support to global iterative solver
- Support for shell thickness output in advanced nonlinear analyses
- Option to treat CGAP elements as linear contact elements
- New and expanded support for system cells, case control commands and bulk data entries

Abaqus environment
- Improved import and export support for various Abaqus keywords as well as support for Abaqus standard beam cross-sections
- New Abaqus structural output request and Abaqus thermal output request modeling objects to specify both types of results to output as well as output options
- New MSG/STA file output control options to control whether software generates Abaqus message file (.msg) and a status file (.sta) during the solve
- Broader range of Abaqus output variables generated, such as selected strain components like plastic strain (PE) or nominal strain (NE)
- Output of selected groups of element nodes or elements for a given output variable, such as reaction forces
- New user-defined text modeling object to specific text in Abaqus input file

Ansys environment
- Support for additional Ansys elements
- Support for importing and exporting Location option for Secoffset command
- Improved support for node and element groups
- Support for beam end releases

Universal file support
- Import and export of NX I-deas® universal file dataset 2477 for I-deas permanent groups containing nodes and elements
- Import and export of NX I-deas universal file dataset 776 for standard I-deas beam cross-sections
- Element associated data for linear beam elements (I-deas element type 21) imported as element associated data in NX

Teamcenter Integration
- Simulation search view

Geometry editing and synchronous technology
All NX CAE solutions include the powerful NX geometry editing capabilities, such as multi-CAD support and synchronous technology. By using NX, you can rapidly clean and prepare geometry from any CAD source for your analysis modeling. Geometry edits and the complete analysis model remain associated to the base design, which means you can easily update your analysis model each time the design changes. As a result, NX accelerates your design-analysis iterations and improves your overall productivity. New improvements to NX 7.5 for synchronous technology include the ability to:
- Change the size of a chamfered edge regardless of its feature history
- Label an angled face as a chamfer
- Optimize face to simplify surface types, merging faces, improving edge accuracy and recognizing blends
- Convert B-surface faces that appear like blends to a replace blend feature
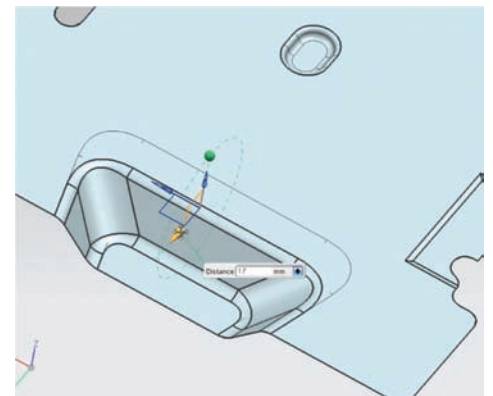
- Select faces in components other than the work part within an assembly for Move Face command
- Pattern face enhancements including propagating edits to all instances in a pattern
- Add a fixed constraint to selected faces
- Create a dimension to the virtual edge at the intersection of two faces separated by a blend
- Highlight and review relations that exist on faces
- Establish and save an offset (wall thickness) relationship with another face
- Synchronous technology face overflow options

Add-on products for Advanced FEM and Advanced Simulation
The simulation capabilities of Advanced FEM or Advanced Simulation can be expanded through additional products for durability solutions, thermal solutions, flow solutions and NX Laminate Composites.

Durability solutions
Structural fatigue analysis is used by engineers to predict design life and durability. Engineers can check whether in-service loadings are likely to cause early failure in parts subjected to heavy vibration or cyclic loads. NX 7.5 introduces two new products for evaluating durability: a durability wizard and an advanced durability solution.
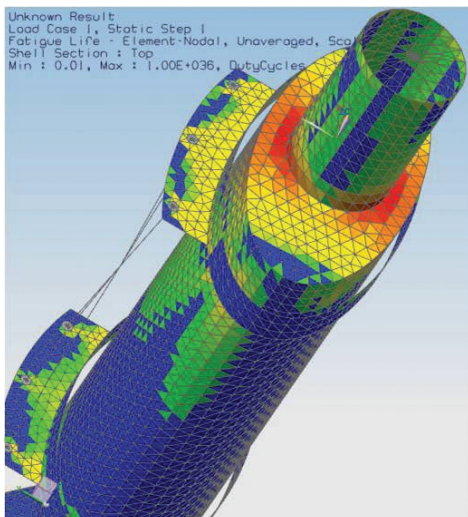
NEW Durability wizard The new durability wizard enables you to perform basic structural fatigue analysis on the static stress analysis solution. Using an intuitive, guided process for designers and less experienced users, engineers can compute strength safety factors and fatigue life.

NEW Advanced durability The new advanced durability solution assists expert users in performing more complex analyses with support for static and transient loads and the ability to handle bi-axial stresses and notch effects. The durability solution process can contain multiple static and transient events.

Static event results support
- NX Nastran or MSC Nastran: Sestatic 101 – single constraint and Sestatic 101 – multi constraint
- Ansys: linear statics
- Abaqus: static perturbation

Transient event results support
- NX Nastran: Sedtran 109 and Semtran 112
- Response Simulation solution process
- NX Nastran Semodes 103: flexible body solution with flexible body recovery option defined
- NX Nastran advanced nonlinear: Advnl 601,129 and Advnl 701 (linear stresses and strains)

Thermal solutions
NX Thermal solutions offer high fidelity simulation of radiative, conductive and convective heat transfer. Thermal solutions can also be used with NX Flow solutions for coupled thermo-fluid simulation. NX 7.5 includes improvements for NX Thermal, NX Advanced Thermal, NX Space Systems Thermal and NX Electronics Systems Cooling.

NX Thermal NX Thermal includes improved modeling, UNV import and Ansys support capabilities.

Modeling
- Vary the thickness of 2D elements
- New material orientation definition methods using tangent curve, vector, orientation angle, surface normal or spatial fields

- Group support for boundary conditions
- Ability to override specific thermal properties of a part of a model with constant, time, temperature or spatially varying values
- Negative heat loads in the thermostat and active heater controller
- Projection direction specified for overlapping elements
- Total thermal coupling conductance definable per element
- Temperature locking in transient phase change calculation
- All elements allowed to shadow for orbital and solar heating requests
- Table for specifying results sampling intervals at specified times

UNV import
- Ability to import error/warning group UNV files
- Importing function based material properties from I-deas using UNV files

Ansys support
- Mapping solver now supports the following Ansys elements: Link180, Shell281(6), Shell281(8), Solid185(4), Solid185(6), Solid185(8), Plane182(3), Plane182(4), Plane183(6), Plane183(8)

NX Advanced Thermal NX Advanced Thermal includes improved modeling and solver capabilities.

Modeling
- Both voltage and current specifiable as temperature varying electrical boundary conditions for Joule Heating simulation objects
- Both voltage and current specifiable as time or temperature varying electrical boundary conditions for Peltier Cooler simulation objects
- Between large parallel horizontal plates correlation option for across gap convection coupling
- Ablation-Charring modeling object support for axisymmetric elements
- Duct modeling
- Display duct orientation
- Multiple 1D ducts connected to same 3D flow face
- Solid area, fluid area, external perimeter or wetted perimeter specifiable when defining user-defined beam section for a duct with thick wall type of 1D mesh

Solver
- Articulation performance and accuracy improvements

NX Space Systems Thermal

Improvements in NX 7.5 include all of the NX Thermal and Advanced Thermal improvements plus enhanced modeling and import/export capabilities.

Modeling
- Ability to merge calculation points for elements of a selected region to force their temperatures to be identical
- Periodic convergence for spacecraft transient thermal analyses
- Orbit Visualizer window to display the aim, align, satellite-sun and satellite-planet vectors

Import/export
- Import and export of the STEP-TAS radiation model

NX Electronics Systems Cooling NX 7.5

includes all of the NX Thermal and Advanced Thermal improvements.
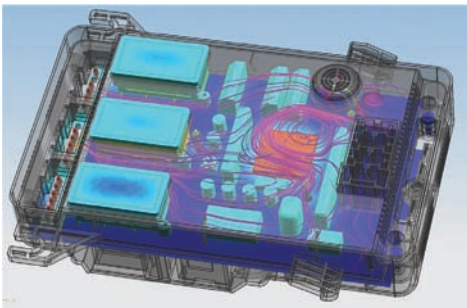
Flow solutions

NX offers complete computational fluid dynamics (CFD) and thermal solutions that are fully integrated into the native NX environment. The integrated CFD solution allows fast and accurate fluid flow simulations, limiting costly, time consuming prototype testing cycles. NX 7.5 includes improvements for NX Flow and NX Advanced Flow.
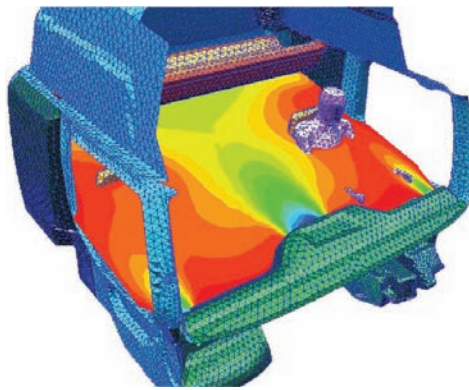
NX Flow NX Flow contains improved modeling, results and UNV import capabilities.

Modeling
- Variable thickness of 2D elements
- New material orientation definition methods using tangent curve, vector, orientation angle, surface normal or spatial fields
- Group support for boundary conditions
- Table to specify sampling intervals of desired results at specified times
- Fractional step time integration scheme
- Rotation and translation of surfaces in shear

Results
- New 3D flow results for relative pressure and absolute pressure
- Y+ results at all nodes in the flow domain

UNV import
- Import error/warning group UNV files
- Function-based material properties imported from I-deas using UNV files

NX Advanced Flow NX Advanced Flow provides improved modeling and duct modeling capabilities.
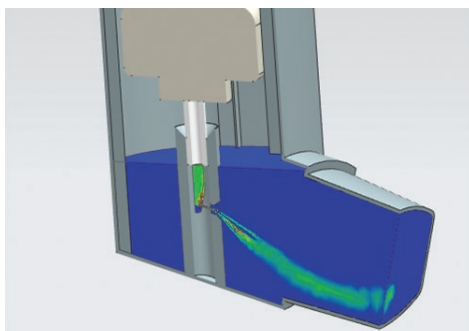

Modeling
- New mixing plane boundary condition
- Homogeneous gas mixture modeling object
- Tracer fluid modeling object
- Mixture and tracer fluid initial, ambient and boundary conditions
- Condensation and evaporation modeling
- Water content initial condition
- Water thickness results for the condensation and evaporation simulations

Duct modeling
- Display duct orientation
- Multiple 1D ducts connected to same 3D flow face

Composites modeling and analysis
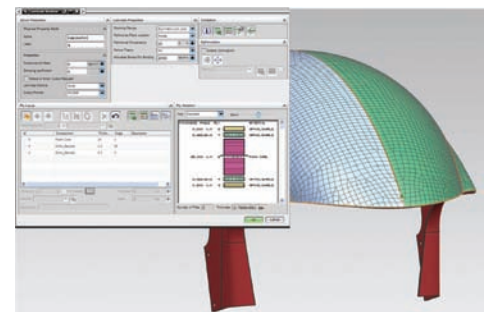
NX Laminate Composites NX Laminate Composites is a modular NX simulation toolset for laminate composite structures. Easy-to-use ply and laminate definition tools enable you to quickly create finite element models representing the laminate composite design. NX 7.5 improvements for Laminate Composites include enhanced modeling, import, export, postprocessing and journaling capabilities.
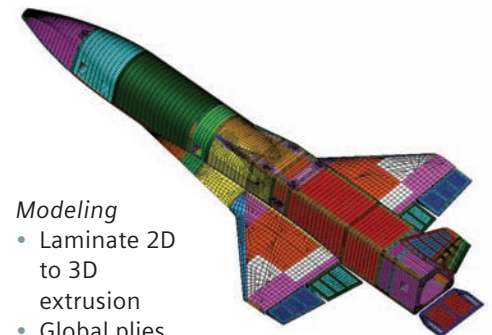
Modeling
- Laminate 2D to 3D extrusion
- Global plies assignable to element faces of a 2D mesh

- Primary draping orientation using seed curve
- Draping mesh properties set by element size or as a percentage of mesh size
- Selection of cut curves along element edges when draping global plies
- Persistent draping results

Import
- FiberSIM layups imported onto selected faces
- Laminate layup imported from CSV files

Export
- Laminate data exported to CSV files
- Results summaries exported to Excel spreadsheets or CSV files
- Fiber orientation display options as a function of the lock angle or as a percentage of 90°
- Ply orientations shown when no draping solver is selected
- New design variable: composite property modeling objects for Nastran Desopt 200 solution
- Editable ply ID numbers
- Zone creation improvements

Postprocessing
New enveloping rules for Laminate post report for max and min, min, max and absolute max

Journaling
All newly introduced Laminates commands support journaling

Analysis to physical test correlation

NX FE Model Correlation NX Finite Element (FE) Model Correlation software quantitatively and qualitatively compares simulation and modal test results, as well as two different simulations. Tools are provided to geometrically align the models, pair the modes from both solutions, view mode shapes side-by-side and calculate/display correlation metrics. NX 7.5 improvements to FE Model Correlation include enhanced correlation setup, export and postprocessing capabilities, as well as new FE model updating capability.

Correlation setup
- New pretest planning solution process
- Display of a DOF set that matches the node map between work and reference solutions
- Generation of imported reference test model including sensors and to align it with the work model prior to creating a new correlation
- Out-of-date indicators
- More flexible test data unit handling for UNV file import

Export
- Ability to export mode pairs to a spreadsheet or CSV file

Postprocessing
- Side-by-side modal results
- Cross-orthogonality correlation matrix
- MSF correlation matrix

NEW FE Model Updating New for NX 7.5, NX FE Model Updating is an advanced correlation tool designed to update finite element models to match real life test data as closely as possible. Model updating increases confidence in modeling approximations by finding the best stiffness and damping values for each FE variable to better match the test results. NX FE Model Updating is an add-on to FE Model Correlation that enables you to:
- Create a Desopt 200, a Model Update analysis solution, modal test data solution and Model Update solution process
- Create design variables that you predict are sensitive to the FE model stiffness/mass
- Select frequencies and mode shapes to be included in the optimization target function
- Select modal or physical (Guyan) model reduction
- Graphically display sensitivities for each design variable with respect to each frequency or mode shape target
- Select one of the following optimization algorithms: least squares, steepest descent or genetic
- Select shape correlation based on modal assurance criteria (MAC) or cross-orthogonality
- Export model update results to an Excel spreadsheet or CSV file

- Update initial values and bounds of the design variable modeling objects with current optimized design variable values of the active Model Update solution process
- Update the material and element properties referenced by the design variable modeling objects in the FEM and simulation file with the current optimized design variable values of the active Model Update solution process

NX Nastran 7.0
Available as a standalone enterprise solution or seamlessly integrated at the core of many NX Simulation products, NX Nastran delivers comprehensive performance simulation capabilities for a broad range of engineering disciplines and industries. NX Nastran 7.0 provides enhanced dynamics, elements and superelements, as well as advanced non-linear solution, linear contact, glue connection, optimization, process, DMP solution, numerical and platform support improvements.

Dynamics
- Modal contribution extensions to support element responses, new normalization options and SORT1/SORT2 output options
- Improved multi-body dynamic interface, including consolidated support for different MBD solvers into one interface, support for OP4 output of flexible body matrices and state-space equation format support for controls interfaces
- Support for scale factors on the case control of DMIG inputs for K2GG, B2GG, M2GG, K2PP, B2PP, M2PP and K42GG
- Direct solutions support for rotor dynamic analysis
- Replacement of the default enforced motion method (from the absolute displacement method to the constraint mode method)

Elements
- Total mass input option for non-structural mass (NSML bulk data)
- Global labels assigned to composite ply layers (PCOMPG physical property input)
- New option to sort output of composites ply results (GPSORT case control)

Superelements
- Definition of different material temperatures TEMP(MAT) for different superelements by subcases

Advanced nonlinear solution(SOL 601)
- Sussman-Bathe rubber support for hyperelastic materials to enable direct use of uniaxial stress-strain data for material properties
- Shape memory alloy (SMA) material support to simulate superelasticity behavior due to reversible phase transformation of austenite and martensite
- Support for variable node numbers for tetrahedron and pyramid elements
- Support for general nonlinear spring element that can be used for translational and rotational stiffnesses
- Moment transfer now supported under flexible RBE2 option
- Dependent DOF now allowed as independent DOF for RBE2 or RBE3 and vice versa
- Support set ID for contact case control (BCRESULTS, output request)
- Ability to output LDC load factor

Linear contact
- Local definition of contact parameters to assign different parameters to different contact pairs; BCTADD used to combine contact sets into a single contact set
- Automatic penalty factor calculation as the default (improves performance by enabling well selected penalty factors to reach convergence with less iterations)

- New, more robust contact refinement algorithm that provides more accurate contact pressure results
- Full support of the linear contact with the element iterative solver (gives better performance of contact with solid element models)
- Re-use of contact stiffness (enabling users to output final contact stiffness to DMIG punch file that can then be re-used in subsequent analyses)

Glue connections
- Local definition of glue parameters to enable users to assign different parameters to different glue pairs; BGADD used to combine contact sets into a single contact set
- New, more robust glue refinement algorithm that facilitates a more accurate load transfer for glue connections
- Glue connections now enabled for joining dissimilar acoustic meshes

Optimization
- DCONSTR bulk entry to allow frequency-dependent lower and/or upper bounds on constraints
- Power spectral density responses (PSDDISP, PSDVELO, PSDACCL) now defined for DRESP1 bulk entry
- More control for selecting constrained response IDs for printing (P2RSET parameter input DOPTPRM bulk entry)
- Alternative optimizer algorithm now can be optionally used in place of default DOT optimizer

Process Improvements
- RMAXMIN support for SOL 101 solutions to capture maximum or minimum results across multiple subcases

DMP solutions
- RDModes solution extended to SOL 111, including with acoustic meshes
- Sparse data recovery option for RD (mode that greatly improves performance when only a few response locations are specified)
- DMP support for virtual mass simulation with MFLUID
- DMP support for calculation of constrained normal modes in superelement solutions
- Serial restarts of a DMP solution supported for SOL 103, 111 and 112 (also works for superelement DMP solutions)
- Geometry partitioning option (GPARTN) default changed to 1 (a more versatile option)
- New DSTAT keyword to invoke the load domain static analysis

Numerical improvements
- Improved modal frequency response calculation for very large number or modes (uses new FRRDRU module)
- Diagonal scaling option for symmetric decomposition of sparse matrices to improve performance

Platform support
- I/O libraries added to improve I/O performance on x86-64 Linux platforms (invoked with FFIO keyword on Altix platform and MIO keyword on other x86-64 Linux platforms)