

Answers for industry.

What's new in NX 9 for simulation

Improving modeling efficiency while reducing simulation process time

Benefits

- Quickly update existing meshes to conform to modified geometry
- Gain more control over mesh quality in fewer steps
- Easily manage and simulate complex transient solutions with varying boundary conditions
- Solve large thermal models up to 8 times faster
- Create more accurate fluid domain geometry by up to 50 percent faster
- Cut NVH simulation time by 50 percent

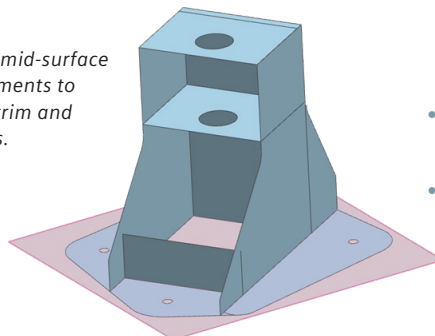
Summary

Simulation is a critical part of the product engineering process, and Siemens PLM Software continues to invest in extending the multidiscipline simulation capabilities within NX™ for structural, thermal, flow, motion, multiphysics and optimization applications. NX 9 for simulation introduces new capabilities and enhancements in NX CAE and NX™ Nastran® to help you make smarter engineering decisions.

NX CAE 9

NX CAE is a modern simulation environment for analysis modeling; structural, thermal, flow, motion and multiphysics simulation; optimization; simulation data management and simulation-driven design. NX CAE 9 releases enhancements spanning these categories to improve efficiency and reduce simulation process time.

Focused mid-surface improvements to extend, trim and file holes.



Simulation modeling and results visualization

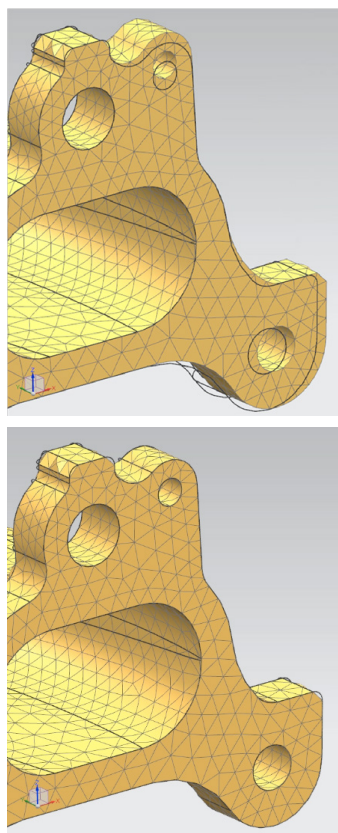
NX Advanced Simulation and NX Advanced FEM

NX Advanced FEM is a comprehensive multi-CAD finite element (FE) model creation and results visualization tool that forms the foundation for the entire NX CAE product suite. NX Advanced Simulation combines the power of an integrated NX Nastran desktop solver with NX Advanced FEM for an integrated structural analysis solution. The NX 9 release includes many new simulation modeling and results visualization improvements for NX Advanced FEM and NX Advanced Simulation, such as:

Geometry idealization and abstraction

- Create mid-surface by face pairs more easily through an improved algorithm that better controls how the software extends, trims and fills holes in the generated mid-sheets
- Merge multiple pairs into a single face pair
- Select an existing datum plane when using the replacement mid-sheet option

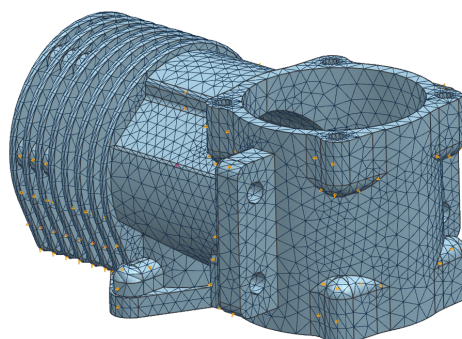
What's new in NX 9 for simulation



Existing mesh (top) will be morphed to fit new geometry dimensions, which are outlined. Morphed mesh (bottom) is the same mesh except nodes have been moved to new locations to match revised geometry.

General meshing

- New mesh morphing capabilities that can morph an existing mesh to conform to geometry modifications. When morphing a mesh, the software keeps the overall mesh topology constant, which means it computes new locations for the nodes in the existing mesh to conform to the changed geometry, and expands or shrinks the mesh to fit the modified geometry. Morphing provides an alternative to the mesh update that occurs within the FE Model Update command. With a mesh update, NX deletes and recreates meshes that are affected by change to the underlying geometry. Mesh morphing can also be used to morph legacy, orphaned meshes to fit new, slightly different geometry
- Fillet and cylinder mesh controls to create better quality 2D and 3D meshes by controlling the distribution of elements along the axis of the fillet or cylinder and about the circumference of the fillet or cylinder
- Create a mesh control definition without assigning the mesh control to any specific geometry to simplify the mesh definition process for complex parts. For example, you can use options in the new mesh control definition dialog box to create a fillet mesh control definition that specifies an element size of 5 millimeters (mm) on all fillets that have a radius between 0 and 3 mm and a maximum angle less than 150 degrees



Tetrahedral mesh utilizing mesh controls for cylinders.

Shell (2D) meshing

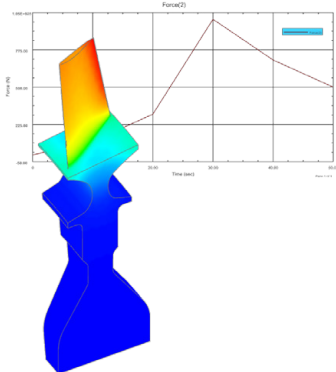
- Automatically suppress holes in sheet bodies during 2D meshing
- Target minimum element edge length option to prevent the generation of any elements smaller than a computed minimum size value
- Edit 2D mesh and element properties by selecting polygon faces to streamline 2D meshing workflows

Batch meshing

- Support for new mesh control definitions in template FEM files to create a better quality mesh
- New geometry assignment module to match the mesh control definitions in the FEM file to the geometry tags and attributes in the part file. For example, if the template FEM file contains a mapped hole mesh control definition, the software searches the computer-aided design (CAD) part file for holes that fall within the specified Filter criteria
- Updates to the parameter file that contains the variables that define the mesh. Changes include the addition of parameters, such as the 'two_element_through_thickness_toggle' parameter, that correspond to options that were added to the 2D mesh and 3D mesh dialog boxes in recent releases. These changes also include the new 'body_type' parameter that allows you to specify the type of body you are meshing, such as sheet, solid or sheet body produced by the mid-surface command
- User functions have been replaced with NX Open functions to aid in integration with automated routines

Boundary conditions

- Create folders to manage simulation objects, loads and constraints for easier organization
- Dialog memory customer default is available to retain boundary condition values, such as numeric and orientation values, to the last values used for that boundary condition type



Generate XY plot and animated contour displays for varying boundary conditions.

- Display boundary conditions attached to polygon edges or element edges as offsets from the edges for easier visualization
- Create an animated contour display of boundary conditions that vary with frequency, heat flow, mass flow, pressure, temperature, temperature difference, thermal capacitance, time, velocity or volume flow
- Generate an XY plot of loads, constraints and solver-specific simulation objects that vary with quantity, such as time or temperature, to verify loading conditions, generate high-quality visualizations for reports or presentations and interrogate and extract loading data

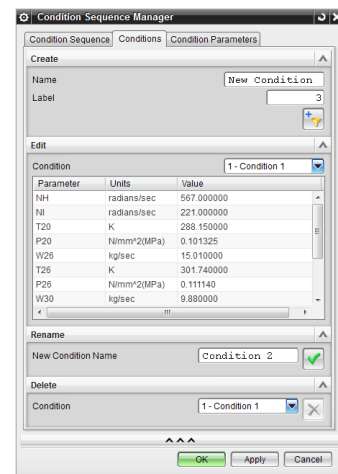
Fields

- Extrapolation options for table field data that can be used to control how NX evaluates independent variable values outside the range of tabular data
- Additional interpolation options are now available for XY tabular data to represent a boundary condition or a material property. New interpolation options available in the algorithm list include linear-linear, log-linear, linear-log and log-log
- Ability to edit table field labels
- Include skips and jump discontinuities in table field data
- Specify that the solver shift, scale or both shift and scale the tabular data entered in a table field
- Support for reference fields that create fields from postprocessed results by referencing numerical data used in a post view. For example, it is possible to create a reference field using nodal results for scalar quantities such as temperature and pressure
- Table of fields to approximate a function that has two or more independent variables, and can be created from any combination of table fields, formula fields, reference fields and linked fields

- Overlay and plot up to 50 fields together on the same graph to more easily compare data visually

Solution management

- Import and manage condition sequences or mission files, and use them to drive boundary condition values at specified time steps. Condition sequences can include any combination of structural, thermal, or flow parameters. An example of a condition sequence would be the conditions applied to an aircraft engine during takeoff, climbing to altitude and level flight



Create and manage mission profiles using the new condition sequence manager.

- Step association manager and subcase association manager are now available for all applicable solvers and solutions. These are useful to manage boundary conditions that are associated with a given solution step or subcase, change multiple boundary condition assignments across solution steps and export information about the boundary condition assignments to a hypertext markup language (HTML) page or a spreadsheet
- Solution association manager to manage boundary condition associations across

multiple solutions within a simulation for solvers and solutions that do not use steps or subcases to manage boundary conditions, such as NX Thermal and NX Flow

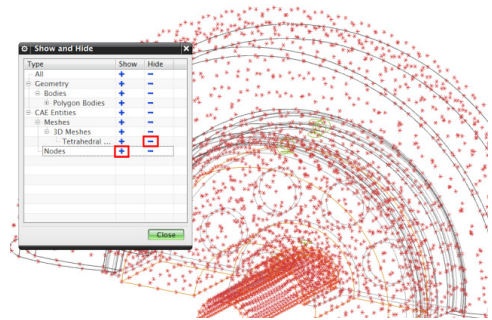
- Edit attributes of multiple subcases, such as output requests, preloads, control parameters, description text and more for all selected subcases in one action

Postprocessing and graphing

- Create fields from identified results to streamline the process of using results from one analysis to drive loads or boundary conditions in a subsequent analysis
- Compare two or more results of the same type and component, and return the minimum or maximum values at nodes
- Combine two or more results of the same type using standard NX expression syntax
- Reduce one or more result components to scalar values at nodes
- Save manipulated results directly to an NX field which can be used to plot, export and save to an auxiliary data file for a functions format (AFU) file, or use it to define field-based boundary conditions in subsequent analyses
- Plot graphs in a secondary window to display and compare multiple graphs without overwriting the model or post view display in the NX graphics window viewports
- When graphing results on a path, create a temporary path or define a group or FE entities directly from the graph dialog box
- Improved methods for selecting entities to define a path or to graph across iterations, such as selecting nodes or elements by group, on feature edges or faces, or by using a selection box
- Greater control over the X and Y axes of graphs. For example, set the X axis to node or element IDs, path length, length

along a vector or coordinates along an axis in the model coordinate system

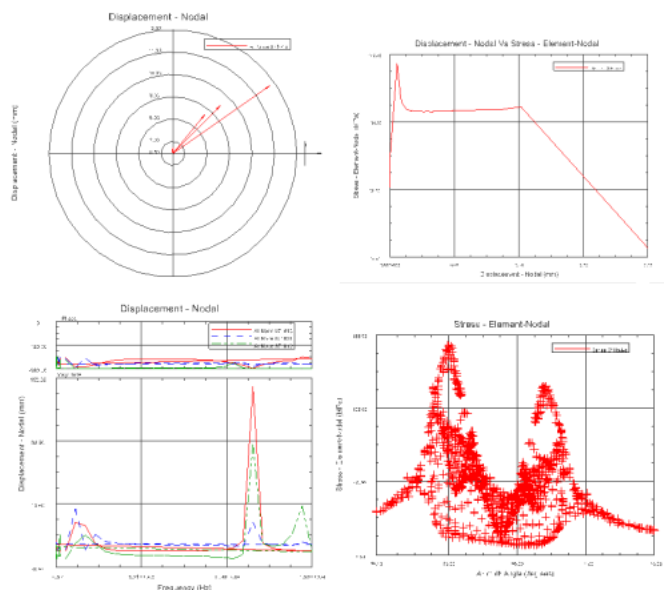
- Create a two-function plot directly from two selected post views
- Create a standard orbit plot graph of X versus Y displacements in the plane of rotation when viewing displacements from a rotor dynamics analysis
- Graph complex components to create Nichols, Argand, polar, orbit and multi-axis plots when working with results in complex format
- Immediately display results in NX that have been reduced to a scalar value without the need to save them to an external file or field
- Graph reduced values against iteration ID, time step, frequency and more (depending on the solution type)
- Plot graphs with only the point marker and no line to create a scatter plot
- Support for PERMAS native results (*.post) files



Display nodes independently of elements.

General capabilities

- Control the visibility of the nodes of a mesh independently of the elements
- New options to control the visibility of meshed and unmeshed polygon bodies, mesh points, mesh controls and mesh mating conditions
- Display only the meshes in your FEM without showing the hierarchy of mesh collectors in the simulation navigator.



Examples of new graph types. Clockwise from top left: polar coordinates, two-function plot, scatter plot and multi-axis plot.

NX Advanced FEM environment for Nastran solver

The Nastran environment for NX Advanced FEM enables engineers to build finite element models (FEM), define solution parameters and view the solution results for either the NX Nastran or MSC Nastran solvers. Enhancements in this release include:

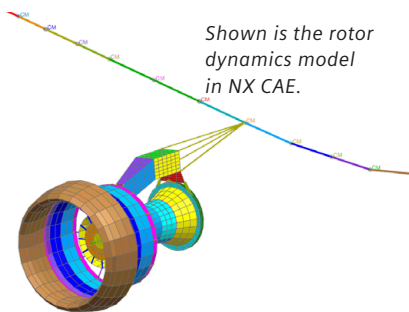
Import and export

- New options for handling data conflicts that control how NX handles conflicts between nodes, materials and physical property data in the source solver input file and the destination FEM or simulation file when appending solver data
- Ability to create a new solution when a solver input file is appended into NX
- Verbose messaging during import to control whether the software lists additional information about issues found during the import process
- Control over entity naming on import
- Preservation of load and boundary condition naming when exporting a Nastran input file from NX and subsequently re-importing the same file
- Preservation of mesh and mesh collector naming when exporting a Nastran input file from NX and subsequently re-importing the same file
- Import and export support for PARAM, OIBULK and YES to preserve entity names, such as materials, physical property tables and meshes when importing the OP2 results file

Solution support

- Create models for NX Nastran Rotor Dynamics analysis. For example, you can define bearing supports, the rotating and stationary portions of the model, and system-wide and rotor-specific solution options among others to simulate the performance of aircraft engines, drive shafts and other rotating machinery

- Maneuver load analysis support, which is a linear static structural analysis that is commonly performed in the aerospace industry, to account for the inertial and gyroscopic loads that arise from the motion of an aircraft
- Support for the new amplitude-frequency (AF) mode normalization method introduced in NX Nastran 9



Elements and properties

- Support for axisymmetric elements (CTRAX3, CTRAX6, CQUADX4, CQUADX8) in a structural or thermal analysis
- Display the material orientation vectors of axisymmetric elements in the NX Nastran and MSC Nastran environments
- Prevent rotation of your model, which is useful when working with axisymmetric or other 2D elements
- Support for bearing elements and properties using NX Nastran CBEAR elements to directly model bearings with speed-dependent properties used for rotor dynamic analyses
- Support for plane stress/plane strain elements and properties in the NX Nastran structural and axisymmetric structural environments

Connection support

- Support for previewing the location of contact and glue elements that are created during the solve
- Control the stiffness of the edge-to-surface glue to change how the edge-to-surface glue stiffness is distributed on the surface being glued
- New thermal-mechanical coupling properties for advanced nonlinear contact that enable control for the contact heat transfer coefficient, the proportion of heat going to the contactor body and the proportion of heat going to the target body

Loads and boundary condition support

- Support for follower forces and moments using selected nodes to define the direction in which the force or moment acts
- Non-structural mass support
- Support for edge loads (PLOADE1 bulk data entry) for plane stress and plane strain elements
- Edge-to-edge contact support in structural analyses for axisymmetric elements, plane strain elements and plane stress elements
- Enhancements and additions to boundary conditions for NX Nastran axisymmetric (2D solid) elements
- Extend support for frequency-dependent and time-dependent dynamic loads to also write TABLED2 and TABLED3 bulk entries
- Extend support for temperature-dependent material properties to also write TABLEM2 and TABLEM3 bulk entries

Solution files

- New F56 file in the solution monitor that is a summary version of the data in the .f06 output data file containing the solution and a list of any warning or error messages issued by NX Nastran during the solve

NX Advanced FEM environment for Abaqus solver

The Abaqus environment for NX Advanced FEM or NX Advanced Simulation enables engineers to build FE models, define solution parameters and view the solution results for the Abaqus solver. Enhancements in this release include:

Import and export

- Merge parts of an Abaqus input file into an existing NX model
- Selective import for Abaqus input files to control which keywords NX imports by either keyword name or keyword category
- Initial support for importing Abaqus assembly models into NX using the assembly-related Abaqus keywords, *PART and *END PART; *ASSEMBLY and *END ASSEMBLY; *INSTANCE and *END INSTANCE
- Import support for solid laminate composites
- Import support for analysis input file processor options CONTACT, ECHO, HISTORY, MODEL, PARSUBSTITUTION, and PARVALUES
- Combine Abaqus surfaces on import resulting in a simulation region in NX
- Ability to define physical constants written to the solver input file
- Improved support for the Abaqus *HEADING keyword
- Various import and export support improvements for Abaqus keywords (see documentation for details)

Contact and connections

- Additional options for enforcing contact constraints

Elements and properties

- Support for a number of additional Abaqus element types, such as plane stress, plane strain, membrane, truss and special purpose elements
- Certain Abaqus axisymmetric elements are now available when performing structural and thermal analyses
- Prevent rotation of your model, which is useful when working with axisymmetric or other 2D elements
- Control how Abaqus distributes the specified mass among the lumped mass elements
- Define the cross-sectional area for gap elements
- Ability to define sides for element-based surfaces

Material support

- Support for reference temperatures for the thermal expansion coefficient for isotropic, anisotropic or orthotropic materials
- New layer symmetry option for laminate physical properties to indicate whether the layers in the laminate (composite shell) are symmetric about a central core

Loads and boundary condition support

- Create additional types of Abaqus multi-point constraints (MPCs), such as beam, link, tie and pin constraints.

Solutions and output requests

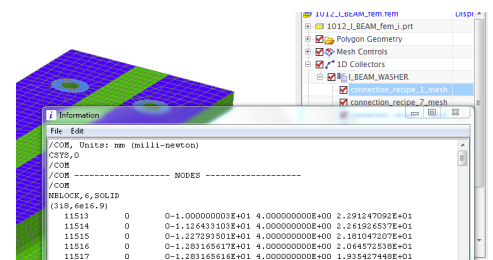
- Support for the Abaqus iterative solver
- Support for transient solution steps in thermal analyses
- Ability to store results as field and history output in the ODB file

NX Advanced FEM environment for ANSYS solver

The Ansys environment for NX Advanced FEM or NX Advanced Simulation enables engineers to build FE models, define solution parameters and view the solution results for the Ansys solver. Enhancements in this release include:

Import and export

- Import support for ANSYS RMODIF command real constants data FKN, FTOLN, PINB and FKOP
- Export models in a user-defined orientation and location
- Identify identical nodal rotations and compact them into a single displacement coordinate system during the import process
- Import support for output control commands, OUTRES and OUTPR
- Preview how a selected entity will be written out to an ANSYS input file
- Merge parts of an ANSYS input file into an existing NX model



You can preview the ANSYS solver syntax before writing input file.

Contact and connections

- Create axisymmetric structural and thermal contacts (node-to-edge or edge-to-edge) in the structural and thermal environments

Elements and properties

- Retain axisymmetric, plane strain and plane stress elements when switching to non-axisymmetric analysis environment
- Prevent rotation of your model, which is useful when working with axisymmetric or other 2D elements
- Set the material orientation for planar elements (PLANE182 and PLANE183) in both axisymmetric and non-axisymmetric environments

Loads and boundary conditions

- Temperature loads in modal analyses

NX Advanced FEM environment for LS-Dyna solver

The LS-Dyna environment for NX Advanced FEM and NX Advanced Simulation enables engineers to build finite element models and define solution parameters for the LS-Dyna solver. Enhancements to this release include: *INCLUDE keyword, which is now supported for import.

Structural analysis

NX Laminate Composites

NX Laminate Composites software is a toolset for modeling laminate composite structures for analysis. Easy-to-use ply and laminate definition tools enable you to quickly create FE models representing the laminate composite design. Improvements to the NX 9 release for NX Laminate Composites include:

Fibersim interface

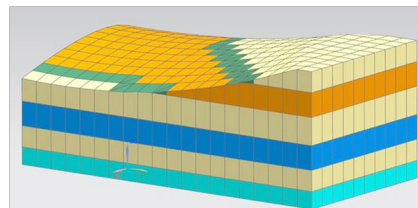
- Edit the draping domain of imported plies to help resolve inflation problems caused by gaps between closely spaced elements
- Export Fibersim ply names that are stored as descriptions in the layup modeler to the Fibersim file
- Export mesh collector names to Fibersim in portable database image (PDI) format when exporting the laminate physical properties

Failure theories

- Code user-defined failure theories in C++, C or Fortran to predict ply and interlaminar failure, and link them with NX Laminate Composites

Modeling

- Define cutting faces that limit the 3D extrusion domain or create drop-off resin elements
- Prevent the creation of degenerate solid elements when extruding thick laminates from a concave face using normal smoothing options
- Extruded element validation to avoid creating degenerate 3D elements due to underlying geometry and extrusion distance
- Copy elements with global layouts to eliminate the need to recreate the layup for the copied elements
- When automatically grouping elements by material type, NX takes into account materials that are assigned at the ply level in laminate and solid laminate physical properties



Use cutting faces to limit the 3D extrusion domain to simulate the post-cure machining of a composite part.

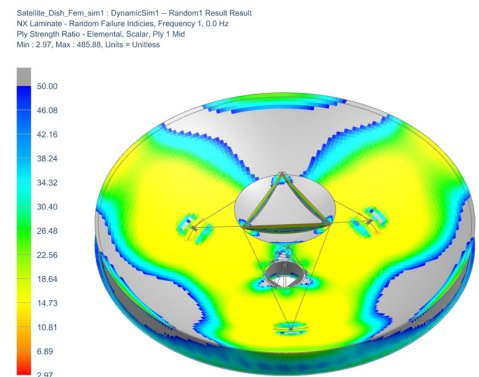
Properties

- Export a 3D laminate physical property to the NX Nastran, Abaqus or ANSYS solvers as a homogeneous physical property pointing to an equivalent orthotropic material
- For inflated 3D meshes, export unlayered, homogeneous properties with equivalent orthotropic materials to shorten solution time

- Account for weft fiber directions when computing zones and creating physical properties for woven ply materials

Solutions

- New laminate dynamic simulation solution process to generate ply results for a base-driven random vibration event
- NX automatically creates the ANSYS KEYOPTS modeling object when exporting a structural solution with a laminate physical property table to ensure that the ANSYS solver automatically computes the ply results at the top, middle and bottom of the plies



Ply strength ratio computed from a base-driven random vibration simulation for a space antenna made from a laminate composite material.

Postprocessing

- Quickly visualize a large number of plies and move quickly from one ply to the other while keeping the same result and component
- Show critical load case ID option to visually identify which load case is critical for each ply and element of the model
- Ability to select elements and plot the ply strain or stress as a function of the laminate thickness

NX Response Simulation

NX Response Simulation software predicts the dynamic responses of structural systems under various loading conditions. Augmenting the capabilities of NX Advanced Simulation, NX Response Simulation produces a broad range of XY plot results and color contour results that aid the user in determining the integrity and suitability of product designs subject to dynamic loads. Enhancements to the NX 9 release include:

- Use a general type of function in an excitation if it has the correct abscissa and ordinate units for the excitation
- Choose stress or strain as the result type when evaluating nodal function response results
- Choose reaction force as the result type when evaluating peak results in response spectrum or DDAM events.

NX Advanced Durability

NX Advanced Durability helps to validate a product's structural integrity over its entire lifecycle under either simple or complex loading conditions. Enhancements in the NX 9 release for NX Advanced Durability include:

- Orthotropic material support
- Enhanced laminate support to compute fatigue life, fatigue damage, strength safety factor and margin of safety on all plies for models with laminate physical properties defined
- Enhanced interface with NX Response Simulation to perform a durability analysis on modal response results from a response simulation solution process event
- Support for the following NX Nastran solutions for both static and transient durability events: SOL 106 Nonlinear Statics – Global

Constraints; SOL 106 Nonlinear Statics – Subcase Constraints; SOL 129 Nonlinear Transient Response; SOL 601,106 Advanced Nonlinear Statics; SOL 601,129 Advanced Nonlinear Transient; SOL 701 Explicit Advanced Nonlinear Analysis

NX FE Model Correlation and NX FE Model Updating

NX Finite Element Model Correlation quantitatively and qualitatively compares simulation and modal test results, as well as two different simulations. NX FE Model Updating is then used to update FE models to match real life test data as closely as possible based on the correlated results. The NX 9 release includes these enhancements for NX FE Model Correlation:

- Save alignment data in an XML file and then apply the previously saved alignment data to the test model to quickly align test and analysis models
- Fine tune alignment to perform small changes in the existing alignment of test and analysis models

Thermal and flow analyses

NX Thermal

NX Thermal can simulate conduction, convection and radiation phenomena for complex products and large assemblies. In addition, NX Thermal can also be used with NX Flow, an NX-integrated computational fluid dynamics (CFD) solution for coupled thermo-fluid simulation. Enhancements in the NX 9 release for NX Thermal include:

Modeling

- Thermal models now support large numbers of elements, nodes and conductances, and the upper limit for element and node labels is now 100 million

- Customer defaults to control metric and English units

Thermal structural mapping

- Improve the temperature mapping from thermal to structural solutions for rotating machinery by using rotational periodicity zone-mapping constraints
- Extend the mapping of temperatures for of regular 0D, 1D and 2D elements in the source axisymmetric zones of the model symmetrically around the appropriate axis on the target elements
- Obtain more accurate results at time values that are between the times at which the results are stored

Radiation

- Directly specify the effective emissivity to get more precision for the selected surface in the heat flux computation

Simulation objects

- Control the accuracy of the coupling in the surface-to-surface contact simulation object by specifying a coupling resolution option

Solutions

- End a thermal transient analysis based on a specified temperature or temperature change that can be useful for performing a temperature stabilization analysis on a region

Reporting

- NX creates a comma separated values (CSV) file that contains the thermostat and active heater controller status at each time step when the thermostat or active heater controller modeling objects are used in a solution

NX Advanced Thermal

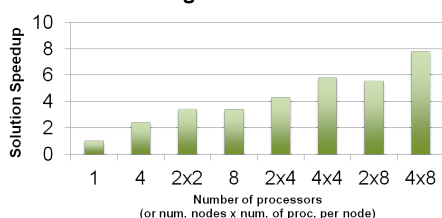
NX Advanced Thermal extends the modeling and simulation capabilities of NX Thermal. Thermo-fluid coupling is enabled with NX Flow and NX Advanced Flow, and thermo-elastic effects can be simulated by mapping temperature results to NX Nastran for structural analysis. This release of NX Advanced Thermal includes these enhancements:

High-performance computing

- The new parallel thermal solver uses domain decomposition techniques to split the thermal system of equations and distribute the computation workload across multiple processes to solve large problems faster

Note: Without the NX Thermal/Flow DMP license, you can only perform an analysis in parallel on a single workstation with access for up to 8 processes per run. The NX Thermal/Flow DMP add-on product is required to remove any software limitations on the number of processes per run for parallel processing and enables parallel solutions over networks and clusters.

Engine Block



Solution speed is increased by using the parallel thermal solver for an automotive engine block model with 1.7 million elements and 7.6 million conductances.

Boundary conditions

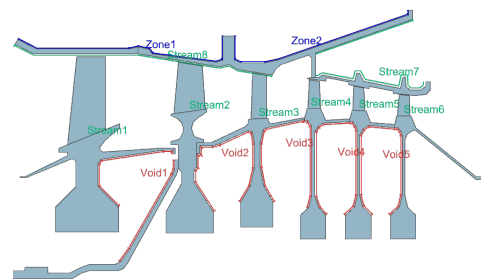
- Thermal void load to define the heat transfer for a part of the model that is convecting to a fluid at an unknown temperature
- Thermal convecting zone load to define the heat transfer for a part of the model that is convecting to a fluid at a known temperature
- Thermal stream load to define convection due to fluid flow over surfaces, or over edges of axisymmetric models
- Rotation load to specify the rotation axis and angular velocity on a complete model or a model subset

Simulation objects

- Thermal rotational periodicity simulation object to define rotational periodicity in a thermal model for conductive and radiative heat transfer
- One-sided or two-sided total temperature effects types of duct flow boundary condition simulation objects to account for total temperature effects in convective heat transfer due to high speed rotating parts

New product: NX Advanced Fluid Modeling

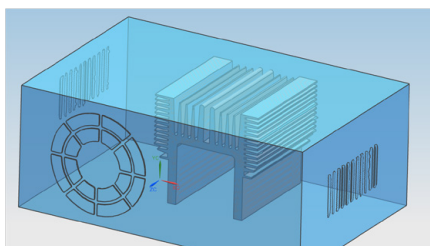
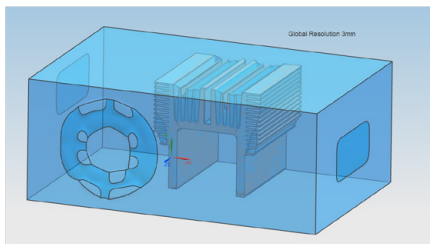
NX Advanced Fluid Modeling delivers state-of-the-art tools to facilitate modeling and meshing of fluid regions in a simulation model used for either CFD or acoustics simulation. Using market-leading CAD and computer-aided engineering (CAE) associativity in NX CAE, this product makes CFD workflows highly efficient by allowing a faster initial build of the model, as well as quicker simulation iterations. With NX Advanced



New thermal boundary conditions to reduce complex boundary condition setup time by 75 percent.

Fluid Modeling, engineers can build fluid domain regions with a powerful surface wrapping and filling toolset. In addition, specialized fluid meshing techniques are offered to refine the mesh in the fluid boundary layer to allow for good accuracy in the friction and heat transfer predictions. Export to other CFD codes is possible with NX Advanced Fluid Modeling via the industry-standard CFD General Notation System (CGNS).

These capabilities used to exist in the NX Flow and NX Electronic Systems Cooling products, but they have been moved to the new NX Advanced Fluid Modeling product in order to introduce a dedicated product for advanced fluid modeling that is solver-agnostic and complementary to NX Advanced FEM and the NX CAE open, multi-CAE philosophy. This also means that NX Flow will be a pure solver product without any modeling or meshing functionality.



Using only a global resolution to create the fluid domain geometry (top) doesn't capture the vent opening and heat sink features accurately. New surface wrap local resolution settings (bottom) used for vent openings and heat sink create a more refined and accurate fluid body.

In addition to the fluid modeling capabilities that were previously available in NX Flow and NX Electronic Systems Cooling, the NX 9 release introduces advanced surface wrapping capabilities to NX Advanced Fluid Modeling that includes:

- Contact prevention constraint to ensure that two polygon objects are not connected in the fluid body and small gaps are maintained
- Local resolution constraint to refine or coarsen fluid bodies in selected regions

NX Flow

NX Flow is a CFD solution that is fully integrated into the NX Advanced FEM or NX Advanced Simulation environment. It delivers sophisticated tools to simulate fluid flow for complex parts and assemblies. Enhancements in the NX 9 release for NX Flow include:

Boundary conditions

- Set the relative pressure as an external condition for the opening type of the flow boundary condition simulation object
- Customer defaults to control metric and English temperature units

Fluid modeling

- Surface wrapping, advanced surface wrapping, fluid domain meshing and export to CGNS functionality has been removed from NX Flow and added to the new NX Advanced Fluid Modeling product

NX Advanced Flow

NX Advanced Flow is an add-on module to both NX Flow and NX Electronic Systems Cooling that extends the flow simulation capabilities of these products to include internal or external fluid flow, including compressible and high-speed flows, non-Newtonian fluids, tracking of heavy particles and multiple rotating frames of reference. Enhancements in the NX 9 release for NX Advanced Flow include:

Parallel flow solver

- Up to 40 percent reduced memory usage for parallel flow solver for large models
- Increased performance for parallel flow solver
- Parallel flow solver support for more simulation objects: rotating frame of reference, mixing plane and disjoint fluid mesh pairing
- Parallel flow solver support for tracer fluid and non-Newtonian fluid modeling objects
- Improved convergence and accuracy when using the k-epsilon turbulence model
- Improved handling of dense porous blockages

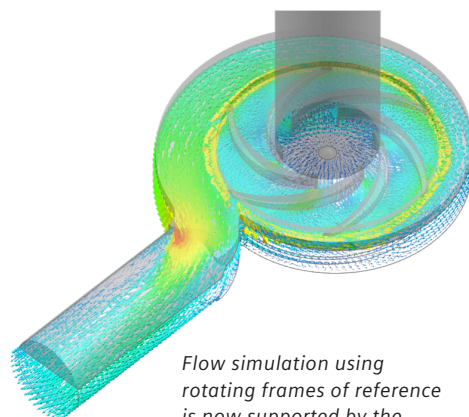
Note: Without the NX Thermal/Flow DMP license, you can only perform an analysis in parallel on a single workstation with access for up to 8 processes per run. The NX Thermal/Flow DMP add-on product is required to remove any software limitations on the number of processes per run for parallel processing, and enables parallel solutions over networks and clusters.

Boundary conditions

- Set the relative pressure as an external condition for the static pressure type of the flow boundary condition simulation object

Simulation objects

- Disjoint fluid mesh pairing simulation object to connect disjoint fluid meshes when solving models with the parallel flow solver. This allows fluid to flow between polygon bodies that have one or more coplanar face pairs where the nodes, elements or nodes and elements of the associated fluid meshes are not coincident.



Flow simulation using rotating frames of reference is now supported by the parallel flow solver.

NX Space Systems Thermal

NX Space Systems Thermal is the space industry vertical application that provides a comprehensive set of tools to simulate orbital thermal analysis within NX Advanced FEM or NX Advanced Simulation. Improvements in NX 9 include all of the NX Advanced Thermal improvements in addition to this new capability:

- Free molecular heating simulation object to account for the heat generated by the interaction between a spacecraft and the low density gas molecules in low altitude orbits

NX Electronic Systems Cooling

NX Electronic Systems Cooling is an industry-specific vertical application for simulating 3D air flow and thermo-fluid behavior in densely packed, heat-sensitive electronic systems. The NX 9 release includes all of the capabilities introduced in NX Thermal and NX Advanced Flow, but the following changes have also been made:

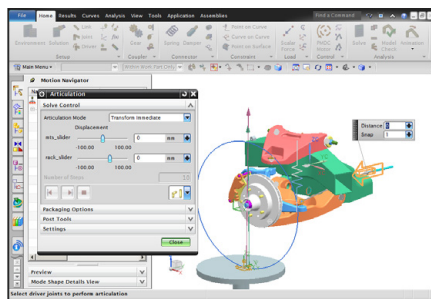
Fluid modeling

- Surface wrapping, fluid domain meshing and export to CGNS functionality have been removed from NX Electronic Systems Cooling and added to the new NX Advanced Fluid Modeling product

Motion analysis

NX Motion Simulation

NX Motion Simulation provides an advanced yet simple-to-use solution that enables engineers and designers 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. Enhancements in the NX 9 release for NX Motion Simulation include:



New interactive articulation options let you drag joint displacements directly in the graphics window.

- Interactively articulate models by dragging joints to new positions which can be useful for checking interference at critical parts of the motion envelope
- Joint limits available during articulation
- Full journaling support for solve and postprocessing commands to enable automation and customization of motion solutions

Simulation-driven design

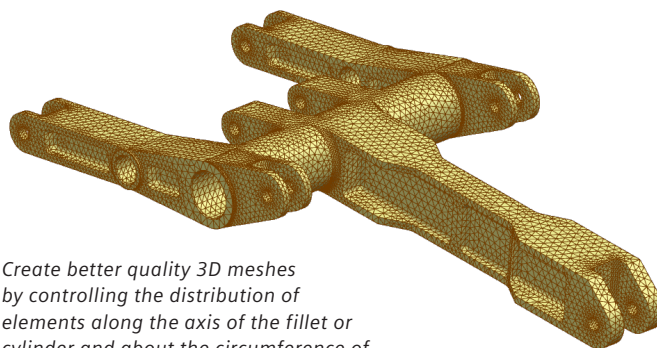
NX Design Simulation

NX Design Simulation delivers design-integrated structural and thermal simulation tools that help designers rapidly compare design alternatives and optimize performance characteristics of products from the earliest stages of the design process. Enhancements to the NX 9 release for NX Design Simulation include the ability to:

- Create folders to manage simulation objects, loads and constraints for easier organization
- Display boundary conditions attached to polygon edges or element edges as offsets from the edges for easier visualization
- Fillet and cylinder mesh controls to create better quality 3D meshes by controlling the distribution of elements along the axis of the fillet or cylinder and about the circumference of the fillet or cylinder
- Create a mesh control definition without assigning the mesh control to any specific geometry to simplify the mesh definition process for complex parts

Simulation data and process management

NX CAE seamlessly integrates with the entire Teamcenter® software data management portfolio, including the simulation process management module. Simulation data management capabilities work out-of-the-box, and companies can implement a complete environment for managing CAE data, processes and workflows as part of a wider product development environment. This reduces waste by promoting re-use of existing designs and engineering knowledge. It synchronizes data and makes it readily accessible through data mining, visualization and reporting. Enhancements for



Create better quality 3D meshes by controlling the distribution of elements along the axis of the fillet or cylinder and about the circumference of the fillet or cylinder.

Teamcenter integration with NX CAE in this release include:

- Control of the new FEM file will contain polygon bodies for any component-level CAE model revision that does not contain an existing NX FEM dataset
- Overrides the default component FEM positioning in the assembly FEM for any CAE model revision with a relation to a CAD item revision
- Related CAE parts added to assembly clone when cloning an NX assembly and there are related CAE item revisions
- Control the default names created for Teamcenter-managed simulation files when creating a new file from the simulation navigator
- New support for Teamcenter Integration for NX with 4th Generation Design (4GD). Create a new FEM and optional idealized part based on the currently loaded 4GD workset

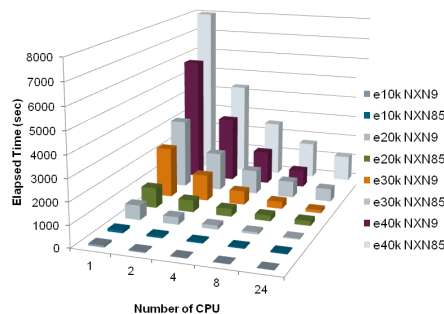
NX Nastran 9

Available as a standalone enterprise FE solver 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. The following lists all of the new capabilities and enhancements that are available in NX Nastran 9:

Dynamics

- In-core frequency response method to take advantage of large amount of random access memory (RAM) and multiple central processing units (CPUs) to improve both serial and shared memory parallel (SMP) solutions by as much as 50 percent
- Support for laminate ply results in transient dynamics solutions
- Root mean square (RMS) von Mises stress is now included in the output for those elements that support von Mises stress calculations when computing the response of a structure to random loads
- Relative motion datablock OUGV is written when either relative or absolute is requested for enforced motion dynamic analysis. This reduces the amount of output data when relative displacement, velocity

In-core frequency response method



The in-core frequency response method improves serial and SMP solutions on a 1.5 million degree of freedom (DOF) shell model.

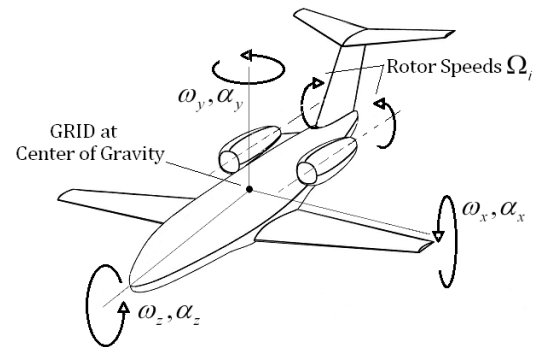
and acceleration output is requested

- EIGR and EIGRL bulk entries now include the amplitude-frequency (AF) mode normalization option

Rotor dynamics

- Rotor gyroscopic and damping forces for rotor dynamics can now be computed in a linear static solution (SOL 101)
- Bearing stiffness and viscous damping can be dependent on speed and displacement, or speed and force when using SOL 101.
- Request element forces for CBEAR elements.

Element and grid point enhancements



Compute rotor gyroscopic and damping forces in a linear static solution (SOL 101).

- CTETRA stress recovery in SOLs 106 and 129 to ensure that the values for elastic strain, plastic strain and stress that are recovered at the grid points lie on the stress-strain curve of the material

Glue

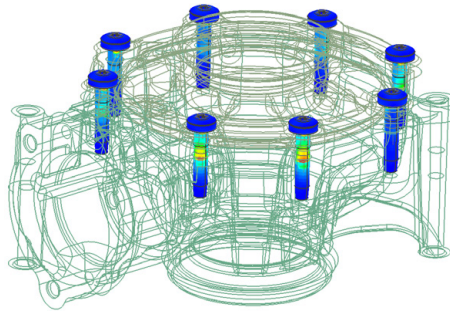
- Ability to output the glue stiffness matrix in DMIG format

Advanced nonlinear

- Define a plastic-cyclic material model for solutions 601 and 701. The new model provides unique inputs for the isotropic hardening rule, the kinematic hardening rule and the rupture criterion
- Model bolts and preload with 3D solid elements for SOL 601
- Edge-to-edge glue is now supported in solution 601 for axisymmetric elements, plane stress elements and plane strain elements
- New rotational load, RFORCE1, is available to apply an angular velocity (ω) and/or acceleration (α) inertia load to the grid points associated to specific elements

Optimization

- Equivalent radiated power (ERP) output can be defined as a design response to minimize noise radiated from a structure
- Selective modification of design variable bounds to reduce the computational effort required to solve optimization problems that contain a large number of designed properties that are linearly related to a single design variable



Bolt preload with 3D solid elements for nonlinear analysis using CHEXA, CPENTA, CTETRA or CPYRAM elements for solution 601.

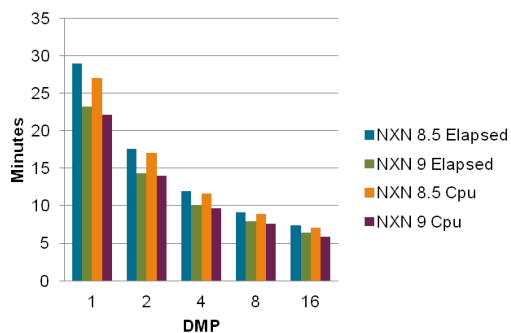
RDModes

- Performance improvements to reduce input/output operations that can reduce overall elapsed time by as much as 50 percent, as in the case of a car body model containing 384,000 elements, 391,000 grids and 2,300,000 degrees of freedom

General enhancements

- Multiple ACCEL1 entries with the same SID are now permitted, and the LOAD bulk entry is not required.
- Control the numerical precision of matrix data that NX Nastran writes to OP4 files for the EXTSEOUT and MBDEXPORT case control commands.

RDMODES Improvement - Time



Improved distributed memory parallel (DMP) performance and scalability in RDMODES.

Contact

Siemens Industry Software
 Americas +1 314-264-8499
 Europe +44 (0) 1276 413200
 Asia-Pacific +852 2230 3308

www.siemens.com/plm

© 2013 Siemens Product Lifecycle Management Software Inc. 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.

Y8 37074 9/13 A