

Siemens PLM Software

What's new in NX 10 for simulation

Solving the most complex simulation problems faster

Benefits

- Streamline coupled thermal-structural and other multiphysics analyses
- Achieve better solution convergence and accuracy with adaptive meshing
- Simulate delamination of laminate composite structures
- Increase productivity for automotive NVH analyses
- Simulate open volume enclosures
- Gain greater control over boundary layer meshing
- Generate fluid body models from mesh data without geometry
- Leverage data relationships for greater understanding of results

Summary

Solve the most complex simulation problems even faster with the latest release of NX™ software. NX 10 greatly expands the types of solutions that can be solved using NX CAE by introducing new environments for multiphysics analyses and the LMS Samtech Samcef™ Solver Suite software. New preprocessing capabilities enable complete acoustics analysis workflows from within NX CAE and also give you full control over your computational fluid dynamic (CFD) boundary layer meshes. New capabilities in NX Nastran® software increase solution speed and extend structural solutions.

NX CAE 10

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. The NX CAE 10 release provides enhancements that extend these capabilities, improve efficiency and reduce simulation process time.

Simulation modeling and results visualization

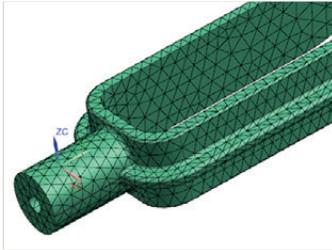
NX Advanced Simulation and NX Advanced FEM

NX Advanced FEM is a comprehensive multi-computer-aided design (CAD) finite element model (FEM) 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 10 release includes many new simulation modeling and results visualization improvements for NX Advanced FEM and NX Advanced Simulation, such as:

Geometry idealization and abstraction

- Control the location where the software computes the thickness when you select mid-surface as the source of the thickness for a 2D mesh
- Designate faces that should be excluded from all automatic or manual geometry abstraction processes

What's new in NX 10 for simulation



Adaptive meshing automates the lengthy process of running multiple analyses with different mesh densities to achieve better solution convergence and accuracy for structural, thermal and multiphysics applications.

General meshing

- Use adaptive meshing to refine meshes for a number of different analysis types in several different solver environments. This automates the lengthy and repetitive process of running multiple finite element analyses with different mesh densities
- Manually associate nodes to selected geometry during a mesh morphing process to gain more granular control over which nodes are associated to specific geometry
- Search for both circular and noncircular holes that meet the criteria you have specified when creating a mapped-hole mesh
- Control whether manually created meshes are written out to the solver input file when you export or solve your model
- Edit multiple meshes and retain the existing element type of each mesh

Shell (2D) meshing

- Improve quadrilateral meshes on largely orthogonal surfaces (surfaces with a number of 90-degree corners)
- Improve quadrilateral meshes on relatively thin-walled parts with curvature, such as automotive body panels

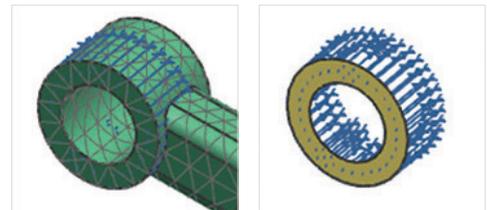
Solid (3D) meshing

- Automatically or manually create hexahedral meshes between selected faces that can update as the geometry updates

Boundary conditions

- Use the boundary condition identification (ID) to define interdependencies between boundary conditions, such as when you want to model flow recirculation of thermal streams

- Create a known time-varying displacement to geometry or finite element (FE) entities
- Define pressure loads of the type 'On axisymmetric elements (normal to edge) ZX'
- Specify the total load to apply for edge loads of the type 'Edge load on plane strain and stress elements (normal to edge) ZX'
- Create time delays and phase leads for dynamic loads
- Create a single load that combines both force and moment
- Define the locations of acoustic sources
- Define the bolt geometry from polygon bodies or 3D elements for bolt preload for NX Nastran solutions 106, 129 and 601, as well as NX Multiphysics
- Create bolt preloads on 2D elements in NX Multiphysics
- Control output results selection by region
- Control the display of boundary conditions and geometry associated with the boundary conditions for easier model checking



Show geometry associated with a specific boundary condition.

Fields

- Create spatial fields with axisymmetric independent domains in order to apply the axisymmetric spatial field, which is two-dimensional across a three-dimensional model
- Create 4D fields with a single function to define the magnitude and distribution of the enforced displacement and temperature load boundary conditions

- Define tabular data that has a complex dependent domain when creating a table field
- Specify an extrapolation method if any of the Delaunay methods are selected as the interpolation method in a table field
- Obtain the coordinates for the tabular data directly from the graphical interface when creating a spatial field to save time

Expressions

- Use 30 new NX CAE-specific functions to expand the power available within expressions and simplify how they are defined
- Create user-defined, dynamically evaluated functions for use within expressions that update throughout an analysis
- Enter certain variables directly into expressions without the need for "UG_VAR" syntax
- Use the new "fd()" function instead of "ug_fieldVar" and "ug_fieldVarAt" syntax when entering fields in expressions

Materials

- Add material properties to existing material library definitions
- Specify stress-strain properties for orthotropic and anisotropic materials
- Define stress-strain data for nonlinear materials as either engineering stress-strain or true stress-strain

Solution management

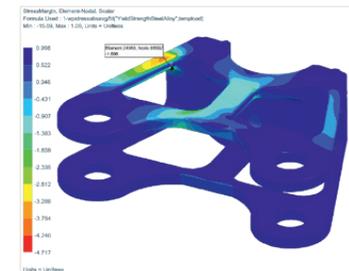
- Import load data from an external source and use it to generate a solution, subcases and loads. These new load recipes are typically used to apply loads to an acoustics simulation model
- Modify a condition sequence (such as removing time points), and then update the solution that was created by that condition sequence
- Scale static loads directly in the subcase/step association manager for NX

Multiphysics, Nastran and Abaqus environments

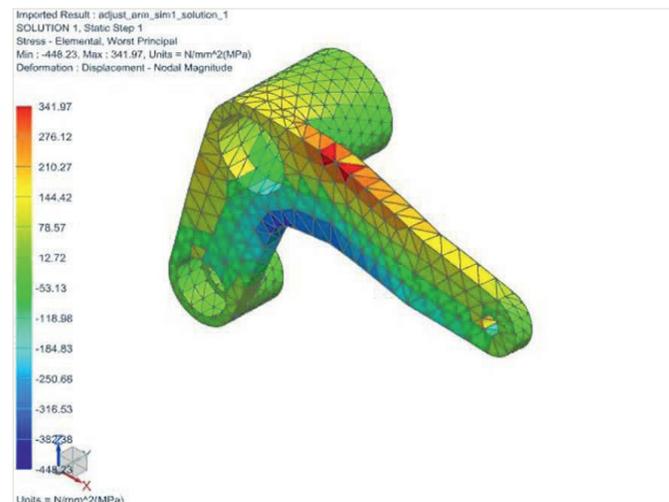
- Make copies of subcases or steps to streamline solution setup
- Export boundary condition step associations to a spreadsheet, make changes in the spreadsheet and then import the associations back into NX

Postprocessing and graphing

- Postprocess NX Multiphysics results
- Generate post outputs, such as contour plots, animations, graphs, or table fields by specifying the geometry context, iterations (for example, value combined across a range of iterations or at a specific iteration) and a formula. This gives you greater control over the type of results you can visualize
- Create a user-defined result expression defined by result type (displacement, stress, strain, rotation or temperature), component and location (nodal, elemental, element nodal). You can also specify beam- and shell-specific result options, complex options and transformation coordinate system
- Specify global text formatting preferences for all post views



Derive the worst principal stress tensor component for stress results at nodes on elements.



Leverage relationships in results data for postprocessing using extended expressions capabilities.

- Display and manipulate results in Kelvin when one or more results files are loaded
- Create a smooth graph of the changes in data values between points
- Define the type of weighted averaging used (for example, area, volume or length) based on the selection type.
- Derive the worst principal stress tensor component for stress results at nodes on elements
- Plot multiple functions simultaneously to separate subgraphs of a new graph
- Plot multiple 2D functions to a 3D graph
- Capture an image from a graph
- Save graphs to a csv file
- Plot unlimited number of functions to a single 2D graph
- Edit multiple functions simultaneously

Name	Color
wishbone_sim 1.sim	
wishbone_fem 1.fem	
wishbone_fem 1_i.prt	
Mesh Controls	
Polygon Geometry	
Polygon Body (1)	Yellow
Override Container	
0D Collectors	
3D Collectors	
Steel	
3d_mesh(1)	Green
Fields	
CSYS	
Groups	
Fields	
Modeling Objects (Filtered)	
Regions	
Simulation Object Container	
Constraint Container	
Pin(1)	Blue
UserDefined(1)	Dark Blue
Load Container	
Bearing(1)	Red

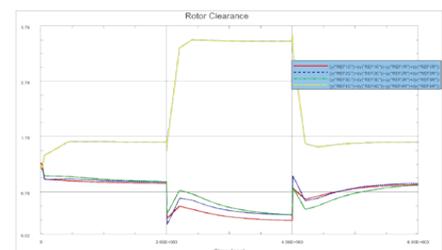
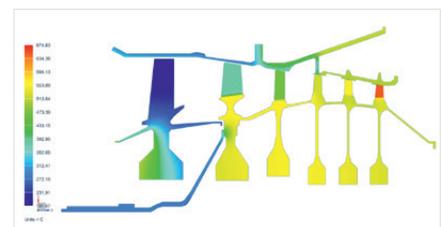
Identify objects, such as meshes, boundary conditions and more, by color in the simulation navigator to more easily recognize different areas of your model.

New NX Multiphysics environment

The new multiphysics environment introduced in NX CAE 10 takes simulation integration to a new level to help you connect two or more solvers to streamline the process of performing complex multiphysics simulation. This environment delivers a consistent look and feel for performing multiphysics simulations so you can easily build coupled solutions on the same mesh using common element types, properties and boundary conditions as well as solver controls and options.

This initial release of the multiphysics environment provides the ability to solve thermomechanical problems in loosely (one-way) or tightly-coupled (two-way) modes. Coupled thermal-structural analysis enables you to leverage the new NX Nastran SOL 401 multi-step nonlinear solver and a thermal solution from the NX Thermal solver.

Using NX CAE 10, you can more easily tackle complex simulations, such as blade clearance analysis in aircraft engine systems, or structural analysis of automotive powertrain components in high-temperature environments. Other applications for electronic components and metalworking processes are also well suited for the multiphysics environment. The following are details of capabilities available in the NX CAE 10 multiphysics environment:



The new multiphysics environment can be used to streamline bidirectional thermal-structural coupling to study blade clearance in an aircraft engine.

Coupled thermal-structural analysis

- Perform a coupled thermal-structural analysis. This analysis type couples a structural solution from the NX Nastran SOL 401 Multi-Step Nonlinear solver and a thermal solution from the NX Thermal solver
- Define contact and glue conditions at the solution level
- Control the time steps in a solution step for when the structural and thermal solvers perform their respective solves
- Specify which solver starts the coupled solution
- Control the time step increment at which NX exchanges data between the structural and thermal solvers
- Specify either sequential or concurrent coupling modes. In sequential mode, the structural and thermal solvers pass information to each other once per coupling time and do not attempt to converge. NX solves the first solution until convergence is obtained, and then solves the other solution using the values from the first solution. In iterative mode, the structural and thermal solvers pass information to each other for several iterations until the variables that are being passed converge
- Select the types of data to exchange between the thermal and structural solvers, such as thermal temperatures and thermal boundary conditions, and structural contact and displacements
- Define heat transfer through contact and gluing simulation objects
- Use the solution monitor to simultaneously track solution progress for coupled, structural and thermal solutions
- Postprocess structural and thermal results together

Import and export

- Import additional time points into an NX Multiphysics structural solution to match a reference solution

Solution support

- Use the step association manager to manage boundary condition assignment details for large models
- Control plasticity and creep effects
- Request contact results and additional strain results for an NX Multiphysics structural and coupled analysis
- Use a preload step type for structural or coupled solutions
- Handle sharp changes in temperature at boundary conditions better with a new adaptive time stepping scheme
- Request specific thermal results on selected geometry, nodes, elements or groups
- Request output of the results at elements, nodes, or elements and nodes for additional results sets
- Model ablation and charring

Boundary conditions

- Create temperature loads that are time dependent
- Define a thermal stream with a radial coordinate system. This load provides an automated way to include the effects of convection and advection in a thermal model
- Define a time-varying or temperature-varying capacitance. For example, this is useful for modeling the capacitance of fuel in aircraft wings. The fuel mass and thus the capacitance varies with time during a flight
- Recover local temperature error estimate results generated by the thermal solver. The adaptive meshing solution process uses the

maximum local temperature error estimates to refine the mesh for conduction thermal analysis

- Define the resistance magnitude at an edge interface by specifying the heat transfer coefficient since this value is more often quoted in handbooks

Properties and materials

- Model thin geometries, such as panels and layers, using multilayer shell elements. This helps you to obtain a detailed picture of the conduction through the geometry without using thin solid (3D) elements
- Use orthotropic materials for thin shell elements
- Define a primary fluid material and up to five additional fluid materials

Elements

- Use plane strain elements

Postprocessing

- Recover and plot convective heat flux results on 2D shell elements
- Recover and plot swirl velocity results on 2D shell elements or on 1D axisymmetric elements

NX Advanced FEM environment for Nastran solver

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

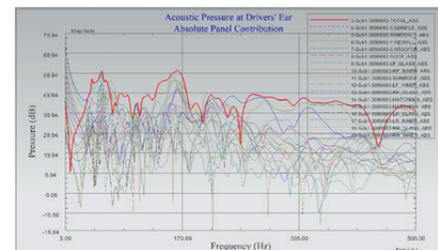
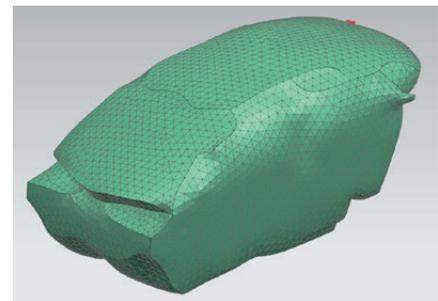
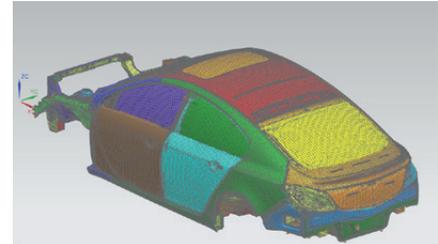
Import and export

- Improve Nastran bulk data import performance for models containing 3D elements with material orientation vectors. NX can be used to automatically create spatial fields to represent the individual material orientation vectors, which then speeds the import process

- Control the degree to which NX evaluates the validity of the file you are importing, which can impact import performance speed
- Have NX automatically create groups that contain the nodes and/or elements contained in the file that you are appending to the current simulation file
- Improve the import performance for models that contain a large number (greater than 10,000) multipoint constraints. This option offers faster import time for a large model at the expense of a more detailed representation of your input file in NX
- Control the units used for temperatures in your model during export. This gives you the flexibility to define temperatures in a different unit system, such as Kelvin, than the rest of your model, which may be in metric or British units
- Control whether you want to use the THRU format when exporting a Nastran data file
- Control how Nastran data is organized in the exported file during a round trip process after importing a Nastran model. For example, you can specify the order in which you want NX to export entities, the unit system to use, or how included files are used during the export
- Various import and export support improvements for Nastran bulk data, case control, parameter and system cell cards (see documentation for details)

Acoustics analysis support

- Create two new analysis types: acoustic, in which only the acoustic features of solid elements and physical properties are available; or vibro-acoustic, in which both the acoustic and the structural features are available
- Import load data from an external source and use it to generate a solution, subcases and loads
- Create an absorbent panel simulation object to describe specific acoustic surface impedance
- Create a panel simulation object that contains 2D structural elements
- Define a fluid-structure interface
- Define damping separately for the structure and fluid parts for vibro-acoustic solutions
- Define the modal contribution parameters for structural, acoustic or vibro-acoustic solutions
- Define the panel contribution parameters in an acoustic or vibro-acoustic solution
- Analyze contribution data using NX CAE postprocessing capabilities, including XY plotting to show contribution of panels and modes to the output, and XYZ plotting to show the contribution of nodal points to the output



Perform an integrated, end-to-end vibro-acoustic workflow to increase productivity for automotive noise, vibration and harshness (NVH) analysis.

Rotor dynamics solution support

- Define axial stiffness and damping for CBEAR elements

Solution support

- Request the calculation of residual vectors

Elements and properties

- Utilize the new NX Nastran generalized plane strain element introduced in NX Nastran 10 for both NX Multiphysics and general linear statics analysis
- The Jacobian element quality check in the NX Nastran and MSC Nastran environments now calculates the Jacobian values for tetrahedral elements only at their Gauss points

Loads and boundary condition support

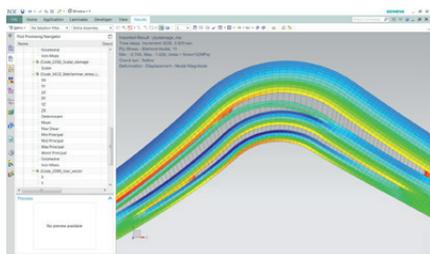
- Organize loads in NX Nastran, MSC Nastran and NX Multiphysics simulation files with load sets

Postprocessing

- Import both the legacy and the new XDB file formats supported by MSC Nastran and MSC Patran

New NX Advanced FEM Environment for the LMS Samcef Solver Suite

NX CAE 10 introduces a new environment for LMS Samcef Solver Suite, which offers a nonlinear finite element solver for structural analysis covering a wide range of applications in the



Simulate complex, nonlinear phenomena in composites in addition to linear static, modal and buckling behaviors using the LMS Samcef Solver Suite.

aerospace, defense and automotive industries. With the LMS Samcef Solver Suite, you can run linear and nonlinear static analyses, as well as modal and buckling analyses. The solver includes unique capabilities for the prediction of complex, nonlinear phenomena such as progressive damage in the unidirectional and woven fabric plies of a laminated composite structure, as well as delamination with coupling to the damage inside the plies. Below are details of what the environment supports for the LMS Samcef Solver Suite:

Import and export

- Import a structural LMS Samcef model into NX. You can import the nodes, elements, physical properties and materials. The import supports unit conversion, and can create groups or regions

- Elements are grouped in mesh collectors and meshes according to their types, physical properties and materials

Elements (see documentation for details)

- 0D lumped mass, grounded bushing, spring and damper elements
- 1D beam and rod elements as well as 1D connection elements
- 2D Mindlin thin shell, Heterosis shell, 2D membrane or shell membrane and metallic or composite elements
- 3D solid and solid shell, metallic or composite elements
- 3D cohesive elements

Materials

- Use LMS Samcef damage material models for laminate composites analysis to predict fiber breakage, matrix cracking and de-cohesion between the two. Delamination between two plies is also taken into account

Loads, constraints and contact

- Nodal force, pressure, speed and temperature loads
- Fixation, enforced displacement and plane of symmetry constraints
- Nodes to surface, solid surface to shell edge and shell edge to shell edge glue conditions
- Nodes to surface, surface to surface and flexible-rigid contact conditions

Solution setup

- Perform only structural analysis for linear static, nonlinear static, modal and buckling structural solutions
- Add specific text to include solver commands or options that are not yet supported by NX CAE

Laminate composites

- Use the NX Laminate Composites toolset along with the LMS Samcef Solver Suite environment to define laminates, global layups, and add LMS Samcef cohesive elements and create 3D meshes from layups
- Postprocess laminate results created by LMS Samcef

Postprocessing

- Review results from a LMS Samcef analysis directly in NX CAE using *.des / *.fac files
- Create postprocessing displays of LMS Samcef results in NX CAE, such as contour plots and animations

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

- Import Abaqus load cases that are associated with static perturbation steps into NX CAE
- Import load, constraint and solution step data associated with part instances in an Abaqus assembly model
- Control the units used for temperatures in your model during import and export. This gives you the flexibility to define temperatures in a different unit system, such as Kelvin, than the rest of your model, which may be in metric or British units
- Various import and export support improvements for Abaqus keywords (see documentation for details)

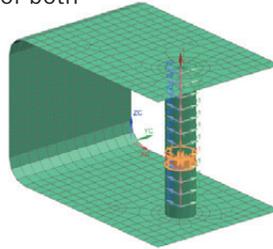
Elements and properties

- Use a number of additional Abaqus element types, such as distributing coupling, connector, 2D diffusive heat transfer, surface, dashpot, axisymmetric shell and membrane, rigid and cohesive elements
- Select a non-default hourglass control approach for shell and membrane elements (S4R and M3D4R)
- Define the material orientation vector of Abaqus 3D elements using a spatial field. This means that you can now use a field to define the spatially varying distribution of orientation angles on the layers of Abaqus composite solid continuum elements and composite shell elements from within NX CAE

- Use Abaqus cohesive elements to model the behavior of adhesive joints and interfaces in composites

Loads and boundary condition support

- Use a field to define a force or tightening adjustment that varies with time for both 1D and 3D bolts



Use a field to define a force or tightening adjustment that varies with time for both 1D and 3D bolts.

Solutions and output requests

- Create Visco type steps in an Abaqus general analysis solution to calculate a transient static response in a solution that includes time-dependent material behavior
- Analyze a transient linear dynamic problem using modal superposition
- Perform Abaqus implicit dynamic stress and displacement analyses
- Perform a direct cyclic procedure for nonlinear, non-isothermal, quasi-static analyses that can be used to predict progressive damage and failure for ductile bulk materials, and delamination or de-bonding growth at the interfaces in laminate composites in low-cycle fatigue analyses
- Create both steady-state and transient heat transfer steps in axisymmetric thermal analyses
- Monitor nodes to determine solution progress
- Restart Abaqus solutions from NX CAE. For example, you can solve a solution that uses saved data from a previous analysis of the same finite element model

- Request that Abaqus report nodal temperature data in a structural solution

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

- Improve the import performance for models that contain a large number (greater than 10,000) multipoint constraints
- Control the locations where the ANSYS input file is created during export, where scratch files are created during the solve and where the result file is created when the solve is completed
- Specify the unit system in which you want to export the ANSYS input file
- Control whether NX exports the nodal coordinate data to the ANSYS input file in single precision or double precision format
- Specify whether to export the entire model, or just selected portions of it
- Control the units used for temperatures in your model during import and export. This gives you the flexibility to define temperatures in a different unit system, such as Kelvin, than the rest of your model, which may be in metric or British units

Solutions and output requests

- Perform coupled thermal-structural analyses in the ANSYS structural environment
- Perform transient thermal analysis to analyze temperatures and other thermal quantities over time

- Perform transient dynamic analysis (also called time-history analysis) to determine the dynamic response of a structure when time-dependent loads are applied

Elements and properties

- Define ANSYS rigid bodies used for multi-body dynamics analyses
- Utilize the latest ANSYS element types (see documentation for details)

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 in this release include:

Import and export

- Merge parts of an LS-Dyna input file into an existing NX CAE model
- Control the name and location of the LS-Dyna keyword file when it is exported
- Specify whether all data with an absolute value smaller than the specified filter is written to the LS-Dyna keyword file as 0.0
- Control whether all data or just selected types of data is exported to the LS-DYNA keyword file
- Selectively import and export LS-Dyna data

Modeling

- Define mass-weighted nodal damping
- Use updated controls and database modeling objects to support a number of additional keyword options (see documentation for details)

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 10 release for NX Laminate Composites include:

Fibersim interface

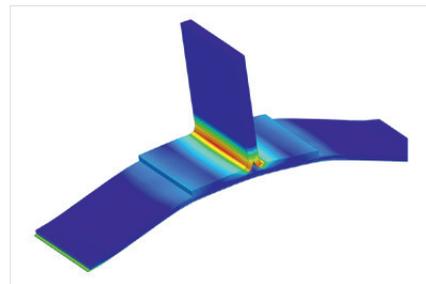
- Import ply layups directly from an NX part containing Fibersim data without passing through an HDF5 file
- Import the zones from the Fibersim™ portfolio of software for composites engineering HDF5 file onto selected polygon faces or 2D elements
- Create a group of unmapped elements for those elements that have no mapped ply data when importing plies of a Fibersim ply-based laminate

Import and export

- Include ply description when exporting or importing to or from a spreadsheet application

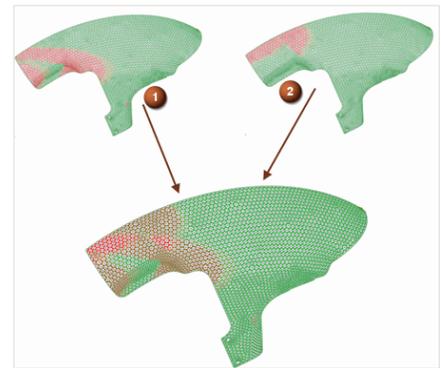
Modeling

- Specify cohesive layers using the new 3D cohesive element between plies of your global layup to model delamination



Create cohesive layers between extruded plies to model and simulate delamination using the LMS Samcef Solver Suite.

- Use the extrude laminate and fill laminate commands in conjunction with the LS-Dyna environment
- Easily reverse the stacking sequence of a layup
- Display fiber orientations for multiple plies at the same time to compare the orientations of one or more plies



Display fiber orientations for multiple plies at the same time to compare the orientations of one or more plies.

Properties

- Edit multiple laminate physical properties or solid laminate physical properties at the same time
- When you use an expression to define the value of the ply thickness, ply angle, or layup offset, NX CAE now updates this value if you change the expression
- Change the global ply identifications (ID) for all plies of selected layups to make postprocessing results easier to understand

Postprocessing

- Output laminate post reporting results as element-nodal results to display them in the same way as you display the structural results
- Generate results from multiple graphical reports simultaneously

- Review all six failure index values for solid laminate plies that have a specified maximum stress or maximum strain criterion

Draping

- Enable multithreading, using shared-memory parallelization, to improve the draping performance for the woven and unidirectional solvers
- Specify a coordinate system instead of a vector for more workflow flexibility when aligning fibers for draping

NX Response Simulation

NX Response Simulation software is used to predict 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 in this release include:

- Composite elements in a response simulation analysis
- Generate root mean square (RMS) and level crossing rate (LCR) stress contour results for random type events when the dynamic solution results file contains ply stress and/or homogeneous stress modes

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 10 release for NX Advanced Durability include:

- The ability to generate the solver results in BUN files, which contain the geometry

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 CFD solution for coupled thermofluid simulation. Enhancements in the NX 10 release for NX Thermal include:

Boundary conditions

- Use solver-evaluated expressions when you define thermal boundary conditions and interdependencies between boundary conditions. For example, you can define the outlet temperature of one thermal stream load as the input temperature of another
- Use 4D fields to define boundary conditions that vary both in space and time simultaneously. For example, at each node, your magnitude can vary in time independently of other nodes
- Define the resistance magnitude at an edge interface by specifying the heat transfer coefficient since this value is more often quoted in handbooks

Materials

- Define a primary fluid material and up to five additional fluid materials using the new multiple fluid material type

Solutions

- Monitor solution progress for coupled, structural and thermal solutions simultaneously through the solution monitor
- Handle sharp changes in temperature at boundary conditions better with a new adaptive time stepping scheme
- Recover local temperature error estimate results generated by the

thermal solver. The adaptive meshing solution process uses the maximum local temperature error estimates to refine the mesh for conduction thermal analysis

- Request output of the results at elements, nodes, or at elements and nodes for additional results sets

Postprocessing

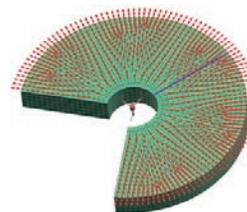
- Recover and plot convective heat flux results on 2D shell elements

NX Advanced Thermal

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

Boundary conditions

- Define a time-varying or temperature-varying capacitance. For example, this is useful for modeling the capacitance of fuel in aircraft wings. The fuel mass and thus the capacitance varies with time during a flight
- Use the boundary condition ID to define interdependencies between boundary conditions, such as when you want to model flow recirculation of thermal streams
- Define a thermal stream with a radial coordinate system. This load provides an automated way to include the effects of convection and advection in a thermal model



Define a thermal stream with a radial coordinate system to include the effects of convection and advection in a thermal model.

Simulation objects

- Model the thermal effects of a spinning model, such as a spinning spacecraft that spins rapidly around a specified axis

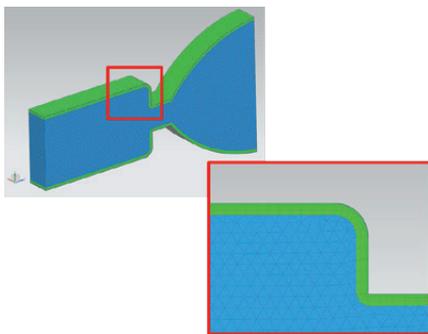
Postprocessing

- Recover and plot swirl velocity results on 2D shell elements or on 1D axisymmetric elements

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. New capabilities in this release of NX Advanced Fluid Modeling include the ability to:

- Create boundary layer meshing directly in the FEM
- Generate a fluid body by selecting imported meshes that are not associated to a polygon body when using the Create surface wrap fluid domain command. This is useful when you only have legacy mesh data available



Create and control the boundary layer mesh directly in the FEM file for CFD and acoustics applications.

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:

Results output

- Output results to ParaView format to import your results into third party software for postprocessing of injected particles

Parallel flow solver

- Use parallel flow solver to speed solve time for large modes for even more analysis types, such as homogeneous gas mixture and tracer fluids, high speed flows (fully coupled pressure-velocity parallel flow solver scheme), shear stress transport (SST) and K-omega turbulence models and non-Newtonian fluids
- Restart from a previous solution
- Run the parallel flow solver using just one process

Note: Without the NX Thermal/Flow distributed memory parallel (DMP) license, you can only perform an analysis in parallel on a single workstation with access for up to eight 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

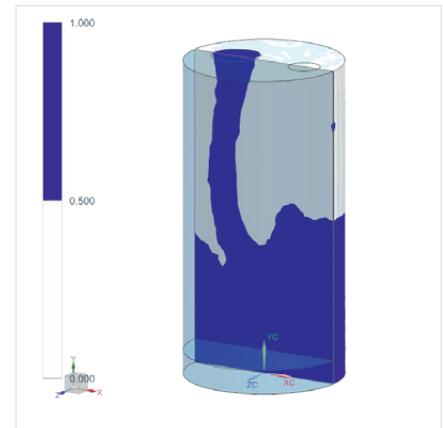
NX Advanced Flow

NX Advanced Flow is an add-on module for 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 10 release for NX Advanced Flow include:

Two-phase, immiscible fluid simulation

- Simulate fluid flow problems for a mixture of any two immiscible fluid constituents
- Simulate open volume enclosures, such as emptying or filling of a tank
- Specify the tracked fluid area fraction of the mixture at ambient condition



Simulate open volume enclosures, such as the filling or emptying of a tank.

Particle tracking

- Solve steady state particle tracking simulation using the flow field from the solved steady state solution
- Simulate particles that stick to the walls or that rebound
- Request particle impact and pass-through reports
- Use new options to control particle tracking

Solutions

- Recover shear rate and dynamic viscosity results on 3D elements when you use non-Newtonian fluids in your model

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 10 include all of the NX Advanced Thermal improvements in addition to these new capabilities:

- Define the thermal resistor models for printed circuit boards (PCB) and electronic components in spacecraft applications
- Distinguish among different components of the spacecraft model by color

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 10 release includes all of the capabilities introduced in NX Thermal and NX Advanced Flow.

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 Teamcenter integration with NX CAE in this release include the ability to:

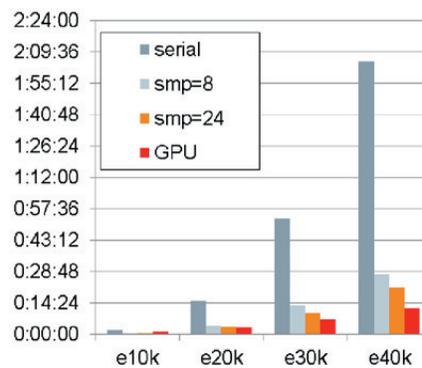
- Leverage new solver types that are supported as named references when managed mode is used with Teamcenter 11.1 or Teamcenter 10.1.3 and all later releases
- Control the item types for CAE parts on import
- Add revised CAE item models to the assembly when exporting an NX assembly from Teamcenter

NX Nastran 10

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 is a list of all of the new capabilities and enhancements that are available in NX Nastran 10:

New graphics processing unit computing

- Solve jobs using graphics processing unit (GPU) computing to divide computations across a large number of relatively small, inexpensive cores to speed solution time of large problems
- Supports Windows and Linux operating systems



Solve using graphics processing unit computing to speed solution time of large problems using a sizable number of relatively small, inexpensive cores.

Dynamics

- Output ply-layer stresses and strains for frequency response (SOL 108 and SOL 111), random response (SOL 108 and SOL 111) and transient response (SOL 109 and SOL 112) analyses for laminate composites modeled with solid elements or shell elements
- Compute Von Mises stress and strain by default for a deterministic frequency response analysis in SOL 108 or SOL 111 when stress and strain results are requested
- Write the output data blocks that contain modal and panel contribution results to the .op2 file for postprocessing acoustics analyses

Multibody dynamics

- Write the full modal damping matrix to standard and state-space MATLAB files, standard and state-space OP4 files and ADAMS MNF files by default

Rotor dynamics

- Specify the reference rotor speed that the software uses to compute the reduced modal basis for a SOL 107 rotor dynamic solve with complex modal reduction
- Define new stiffness and damping terms for CBEAR elements, such as the translational stiffness and viscous damping in the axial direction; the coupling terms for translational stiffness and viscous damping between the axial direction and axes in the plane normal to the axial direction of the rotor; the rotational stiffness and viscous damping about axes in the plane normal to the axial direction; and the coupling term for rotational stiffness and viscous damping between the axes in the plane normal to the axial direction of the rotor

- Model CBEAR element stiffness and viscous damping as functions of speed and displacement, or speed and force for direct frequency response (SOL 108), direct transient response (SOL 109), modal frequency response (SOL 111) and modal transient response (SOL 112)
- Specify that the software uses composite relative displacements or composite relative forces when it looks up speed and displacement-dependent, or speed and force-dependent bearing stiffness and bearing viscous damping
- Optionally include the time-dependent coupling terms in the equation of motion for SOL 107, 108 and 109 to perform rotor dynamic analysis without any symmetry restriction on the rotors in these solutions
- Model interconnected coaxial rotors in a rotor dynamic analysis. Interconnected coaxial rotors are concentric rotors in which one rotor is supported by the other
- Improve the computational efficiency of a SOL 107 rotor dynamic solve by applying a superelement-style reduction to the rotors

New multistep nonlinear solution 401

- SOL 401 is the structural solution used by the new NX multiphysics environment within the NX Advanced Simulation product. The NX CAE 10 multiphysics environment supports all combinations of structural-to-thermal and thermal-to-structural coupling with the NX Thermal solution. SOL 401 is also supported as a standalone NX Nastran solution

Element and grid point enhancements

- Stiffness, mass and loads for the CTRAX3, CQUADX4, CTRAX6, CQUADX8, CTRIAx, and CQUADX elements are now based on a 2*PI section basis instead of on a per radian section basis

Advanced nonlinear

- Use a new potential-based fluid element for SOL 601,106 that can be used in a static analysis in which the pressure distribution in the fluid and the displacement and stress distribution in the structure is of interest
- Define strain-rate dependent plastic material in SOLs 601 and 701 to increase the yield stress with an increase in strain rate
- Use the new 3D shell element in SOLs 601 and 701 to account for the change in thickness and the shift of the shell mid-surface from the halfway position for a large strain analysis
- Define shell composites with the PCOMPG entry, which has global ply IDs for SOL 601
- Optionally define axisymmetric, plane stress and plane strain elements on the XY plane for SOL 601 and SOL 701
- Output results for the bolt loading iterations in SOL 601
- Use the MATCRP bulk entry to alternately define the power creep law in SOL 601.

Optimization

- Handle large optimization jobs for SOL 200 using the enhanced Siemens Analytic Design System (SADS) optimizer that the Design Optimization Tools (DOT) optimizer may not be able to handle in terms of the number of constraints and design variables

Contact

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

www.siemens.com/plm

© 2014 Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Fibersim, Geolus, GO PLM, I-deas, JT, NX, Parasolid, Solid Edge, Syncrofit, Teamcenter and Tecnomatix 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.

43376-Y5 11/14 H