NX for Simulation: What's New in NX 8

Benefits
- Perform new analysis solutions for optimization and multi-physics
- Speed simulation of large models through advances in computational performance
- Link simulation results directly with product requirements
- Reduce simulation modeling and solution time up to 70 percent

Summary

NX 8 for Simulation can drastically reduce the time you spend preparing and solving analysis models by up to 70 percent. Siemens has strengthened and expanded upon the technologies from its long simulation legacy and has brought them together to form NX CAE, a modern and integrated high-end analysis environment, and NX Nastran, a premium finite element solver. NX 8 introduces over 200 new capabilities to both NX CAE and NX Nastran that enable you to rapidly build, update and simulate analysis models, make smarter engineering decisions and deliver better products faster.

NX 8 CAE

Optimization solutions

***NEW!* NX Topology Optimization** Bring simulation to the earliest stage of conceptual design. Topology optimization helps analysts reduce component weight by computing a conceptual optimal design based on a finite element model of the design space and loading conditions. Integration in the NX environment allows easy collaboration between analysts and designers because the conceptual design can seamlessly be handed to designers for further refinement.

NX Topology Optimization is an add-on to the NX Advanced Simulation product, and so setting up of the topology optimization model is very simple:

- Create a FE model of the design space, loads, and boundary conditions within NX Advanced Simulation
- Define which elements in the model the optimization can modify
- Define a design objective, such as maximizing stiffness
- Define design constraints, such as volume, and also manufacturing constraints to ensure that the component can be produced
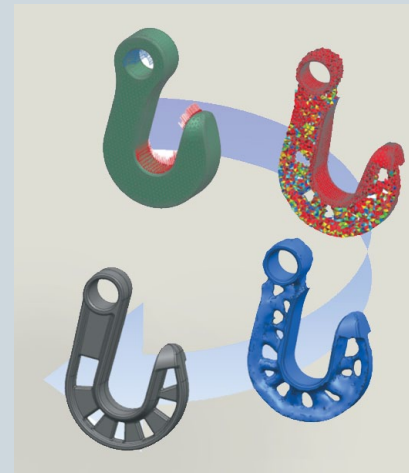

NX

SIEMENS

NX for Simulation: What's New in NX 8

Finite element modeling and analysis solutions

NX Advanced FEM and NX Advanced Simulation NX™ Advanced Simulation software combines the power of an integrated NX Nastran® desktop solver with NX Advanced FEM, a comprehensive multi-CAD FE model creation and results visualization tool. New improvements to NX 8 for NX Advanced FEM and NX Advanced Simulation include:

Geometry idealization and abstraction
- Create a face from an existing mesh
- Suppressing holes in sheet bodies
- Ability to create a node at the center of a hole
- Advanced control for polygon body tessellation
- Measure bodies improvements for polygon bodies
- Tools for identifying and repairing polygon body problems
- New mid-surface display options
- Additional trimming control for replacement mid-sheets
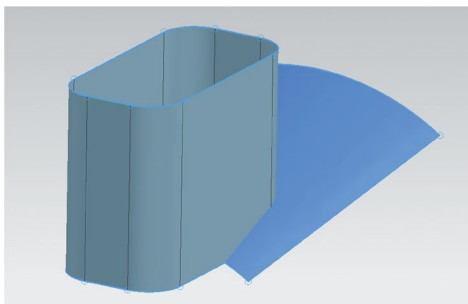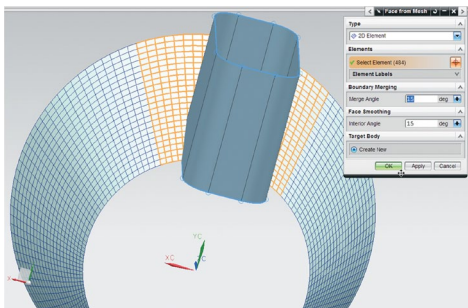
*Create face from existing mesh.*

Material properties
- Preview material orientation vector graphics
- Quickly determine which bodies in your part have no assigned material

- New workflow for editing attributes of multiple meshes
- Specify a directory of material library XML files
- Synchronize loaded materials with library definitions
- Library location reference for exported materials
- Support for MatML databases from Matereality
- Define 3D element material orientation using element faces in Nastran and Abaqus environments
- Kelvin-Maxwell (Rheological) Constants – Supports Nastran creep laws 111, 112, 121, 122, 211, 212, 221 and 222 as well as Abaqus creep law HYPERB.
- Time Hardening (Norton-Bailey) Power Law – Supports Nastran creep law 300, Abaqus creep law TIME, and Ansys creep law Time Hardening.
- Strain Hardening Power Law – Supports Abaqus creep law STRAIN and Ansys creep law Modified Strain Hardening.
- Uniaxial (Rheological) Tabular Input – Supports input of creep as table data or definition of linear viscoelastic behavior for Nastran.
- Laminate orthotropic strength properties
- Temperature-dependent stress-strain curves for isotropic materials

0D, Beam and 1D element meshing
- Create a series of Nastran CFAST or CWELD connections along an edge or curve
- Define a standard, General Geometry or Face of Solid type of beam cross section and save it as a template in the NX Reuse Library while in managed mode
- Default stress recovery points for beam cross sections
- Ability to create multiple 0D elements
- Element Modify Connectivity enhancements

Shell (2D) meshing
- Local remeshing for 2D meshes to refine elements in a specific region without regenerating the entire mesh
- Project nodes directly onto the under-lying CAD geometry rather than onto the CAE polygon geometry
- Shell element thickness contour displays to more easily verify your mesh

- Improved handling of sheet bodies for 2D meshing which stores information about the associated CAD sheet body with your mesh definition
- Interactively define the location of the split line for splitting shells
- New method for dividing triangular elements when splitting shells
- Option for splitting elements based on their warp value
- Option to control the merging of new nodes
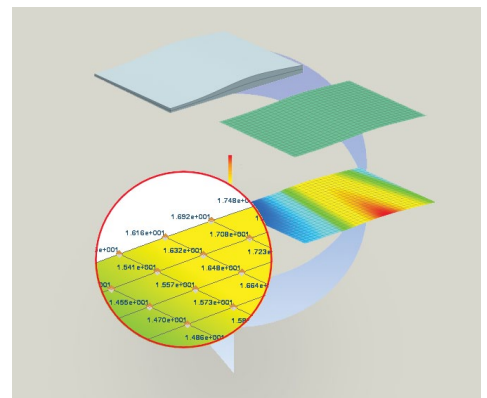
Element thickness viewed as a contour.

Solid (3D) meshing
- Sweep a 3D mesh between bodies
- Target face smoothing option for 3D swept meshes
- Manual element creation for linear and parabolic tetrahedrons, hexahedrons and pentahedrons
- Default available for pyramid element transitions
- Default available to allow minor modifications to 2D seed meshes during tetrahedral meshing
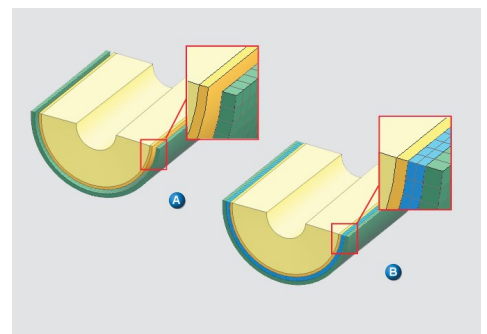
Sweep 3D mesh between bodies.

Assembly FEM
- Replace individual components in your assembly FEM with external superelements to define a system model
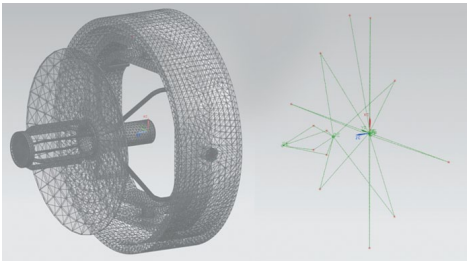
Superelements supported in Assembly FEM.

General meshing
- Element extract enhancements so the mesh retains the mesh associated data and display properties of the source mesh
- Merge mesh enhancements so that the first mesh you select becomes the merged mesh, and the software adds the remaining selected meshes to the first mesh
- More comprehensive information for meshes such as a new Mesh Recipe section, which lists all the options you selected or defined when you created the mesh

Boundary conditions
- Pressure and temperature load contour displays
- Axial deformation loads for 1D elements in Nastran environment
- Varying nodal pressure loads for the Nastran and Ansys environments
- Material temperatures to evaluate temperature-dependent material properties in Nastran
- Specify default initial, material and load temperatures for all other nodes in your model for which you have not defined a temperature load for Nastran
- Time-dependent default temperature loads in advanced nonlinear analysis
- Checks for constraint conflicts at the DOF level rather than at the nodal level
- More quickly add loads, constraints or simulation objects to the active solution or solution step for large models
- Ansys thermal boundary condition enhancements

Optimization
- Support for NX Topology Optimization
- Wizard-like user interface that guides you through shape optimization
- Improved shape optimization algorithm to provide a greater degree of accuracy

Topology optimization of camera gimbal.

General FE Model Import, Export and Solve
- Import and append a solver input file into an existing FEM or Simulation file
- New customer defaults to control the default unit system the software uses to import and export solver input files for all supported solvers
- Control whether a solve aborts when the Model Setup Check detects errors
- Import loads as field data for Nastran and Abaqus environments

Nastran environment

Import
- Improved round-trip bulk data capabilities to indicate that you want to preserve the format of your data for future export
- Selective import options to control which bulk data entries NX imports
- When importing an ASCII input file, import an additional second ASCII format file, such as a parameter file
- Ability to specify the unit system used in the input file that you are importing
- Automatic detection of units on import
- Ability to import temperatures in a different unit system
- Control over the import of unsupported syntax

Export
- Specify the unit system in which to solve the model
- Control the specific data that NX includes in the Nastran input file
- Specify offset values for the different IDs in the input file, such a grids or physical properties
- Export the model in a different coordinate system than the one used in the Simulation file
- Control whether Nastran prints the bulk data section of your input file in the results file
- Descriptions now exported as comments
- Selectively exclude certain bulk data entries from export
- Ability to specify solution parameters on export from a FEM file
- Customer defaults support for parameters
- Option to export DOF Sets in the alternate format
- Change the orientation of a model on export
- Coordinate system names retained on export and import
- Export nodes as fluid grid points

Solution support
- Ability to specify subcase labels for solution steps
- Perform Nastran transient heat transfer analyses in NX Advanced Simulation
- NX now automatically creates a Subcase-Eigenvalue Method type of solution step for SEMODES 103 type solution
- The NLSTATIC 106 solution type has been divided into two different solution types: NLSTATIC 106–Single Constraint for subcases with unique loads but with identical constraints, and NLSTATIC 106–Multi Constraint for subcases with unique loads and unique constraints
- Support for static subcases to define differential stiffness in SEMODES 103, SEMFREQ 111, and SEMTRAN 112 solutions

Material support
- Sussman-Bathe hyperelastic material for NX Nastran ADVNL 601,106, ADVNL 601,129, and ADVNL 701 solutions
- Shape Memory Alloy is a new material for NX Nastran ADVNL 601,106 solutions

- Apply orthotropic materials to the CHEXA, CPENTA, CPYRAM, and CTETRA solid elements for all solutions except ADVNL 601 and 701
- Apply orthotropic materials to the CTRAX3, CTRAX6, CQUADX4, CQUADX8, and CTRIAX6 axisymmetric elements

Connection support
- Surface-To-Surface Contact is available in NX Nastran solution SEBUCKL 105
- Edge-to-Surface Gluing simulation object to connect an edge to a surface to prevent relative motion in all directions
- Create edge regions and surface regions when working with glue commands
- NX Nastran subcase manager to easily manage the loads used in each subcase
- New checks for unintentional grounding and mass reduction
- Request the output of NX Nastran glue results directly from NX
- Post-process NX Nastran glue results for forces and surface tractions

Loads and boundary condition support
- Axial deformation loads for 1D elements in Nastran environment
- Varying nodal pressure loads
- Material temperatures to evaluate temperature-dependent material properties in Nastran
- Time-dependent default temperature loads in advanced nonlinear analysis
- Specify default initial, material, and load temperatures for all other nodes in your model for which you have not defined a temperature load for Nastran

Abaqus environment
- Assign boundary conditions to individual steps
- Remove elements from an analysis step
- Change the friction properties of contact pair interactions and change the friction definition for each solution step
- New solution control parameters to control convergence control in a nonlinear solution and to control the time incrementation scheme
- Define sink temperatures and film convection coefficients for a heat transfer analysis
- Extended support of temperature-dependent stress-strain data for isotropic plastic materials

- Reference nodes for bolts now imported
- Define frequency-dependent or time-dependent properties for representing the dynamic behavior of materials such as glasses, rubbers, and high polymers under stress
- Improved import support for all the Abaqus output variables supported in NX and any group references (defined with the Abaqus *ELSET and *NSET keywords) that are associated with those output variables
- Specify a user-defined orientation for a laminate stacking sequence
- Support for a number of new Abaqus keywords as well as enhancements to previously supported keywords

Ansys environment
- Support for additional thermal element types (FLUID116, SURF151, SURF152, PLANE55, PLANE77)
- Ansys thermal boundary condition enhancements
- Varying nodal pressure loads
- Import support for nodal thickness values on 2D elements
- Importing material orientation vectors as spatial fields
- Include comments or files in an Ansys input file
- Contact now supported in modal and buckling solutions

LS-Dyna environment
- Import LS-Dyna models
- Import d3plot result files from the LS-Dyna solver for postprocessing
- Insert specific comments, commands or include files when exporting an LS-Dyna input file
- Support for solution options and output requests
- Create several types of pre-processing displays to help validate your model for material thickness and orientation
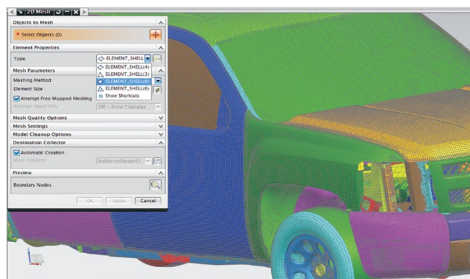
LS-Dyna dataset courtesy of the National Crash Analysis Center at George Washington University.

Postprocessing
- Export a result quantity to an expression to validate against product requirements
- Render axisymmetric results as a virtual solid model
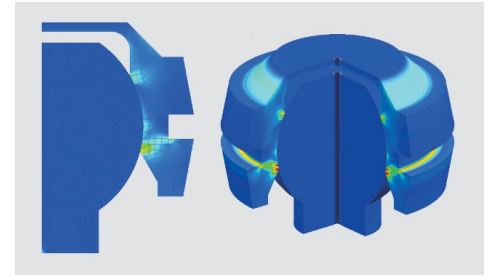
Axisymmetric results as a 270-degree 3D cutaway display.

- Apply post view settings to multiple post views simultaneously
- New options for results overflow/ underflow
- Scalable fonts for headers, color bars, and markers
- Customizable header and color bar
- Transform and scale deformations with respect to a selected node on the model
- Create new results by performing operations on one or more existing results sets
- Enhanced coordinate system support for native, material, selected rectangular, selected cylindrical, and selected spherical coordinate systems
- Free body results display
- Display force or moment results summed at the nodes of all elements or a subset of the elements
- Box selection methods for elements and nodes in post-processing
- Display the stress or strain results on both the top and bottom of 2D elements simultaneously
- Bending stress and strain results for 2D elements

General capabilities:
- Browse command to open the directory containing the files generated by the solver, including the input and results files.
- New default file name uses the master part file name as a prefix

- Update Agent command to identify particular solutions within a simulation to re-solve automatically when the solid geometry or the points, curves, or coordinate systems in the master CAD part are modified. The mesh, boundary conditions, and solutions can also be updated.
- Persistent element quality displays
- Units conversion for scalar values
- Select multiple edges or faces on an element
- Limit the number of objects that display in the simulation navigator
- Sort, filter and search for an entity by name in simulation navigator folders
- Display geometry free edges on the model
- Solid Properties Check now reports total surface area of selected 2D elements and total length of selected 1D elements in model
- Save an XY table function in an AFU file to a table field
- Create groups of meshes automatically from common attributes
- Define mesh color basis by Mesh Collector and by Mesh
- Tangent-continuous edge selection method to more easily select polygon edges
- Control over default names of polygon bodies
- Merge manually created, extracted or imported meshes in the Simulation Navigator

Add-on products for NX Advanced FEM and NX Advanced Simulation
The simulation capabilities of Advanced FEM or Advanced Simulation can be expanded through additional products for thermal solutions, flow solutions, laminate composites, durability solutions, frequency response, analysis to physical test correlation.

Thermal and Flow solutions

NEW! **NX Thermal and Flow DMP** NX Thermal and Flow DMP is a new add-on product to the NX Thermal and Flow, NX Electronic Systems Cooling, and NX Space Systems Thermal products. NX Thermal and Flow DMP removes any software limitations on the number of solver processes per run for parallel processing, and provides access to more computing power for thermal and flow analyses. It also enables parallel solutions over networks and clusters.
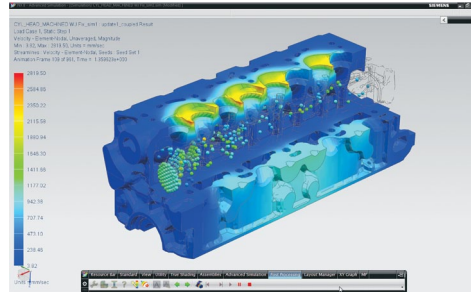
Without the new DMP license, access to up to 8 processes per run, on a single workstation, is possible if you have NX Advanced Thermal or NX Advanced Flow licenses.

The following modules can be run in parallel:
- Flow solver
- View factor calculations

NX Flow NX Flow is a computational fluid dynamics (CFD) solution that is fully integrated into the NX Advanced FEM or NX Advanced Simulation environment. It delivers sophisticated tools to model and simulate fluid flow for complex parts and assemblies. For NX 8, NX Flow contains enhancements for file storage, modeling, and solution set up.

File storage:
- Store catalogs in any folder

Modeling:
- Better control the size, shape metric, and rate of transition between the boundary layer mesh and internal mesh structures when meshing the fluid domain using the Fluid Domain simulation object
- Mesh and boundary conditions of the two models need not be identical when the initial conditions for the current model solution are taken from another model's solution in a different directory
- Preview the material orientation vector

Solution set up:
- Define customer defaults for options in the Solution and Solver Parameters dialog boxes
- Global flow imbalance fraction to ensure that the mass and momentum imbalances are within a specified fraction
- Freeze and reactivate flow field calculation at specified times
- Select initial condition time step from a previous solution
- Specify duct flow solver options

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. For NX 8, NX Advanced Flow delivers additional modeling and solution enhancements.
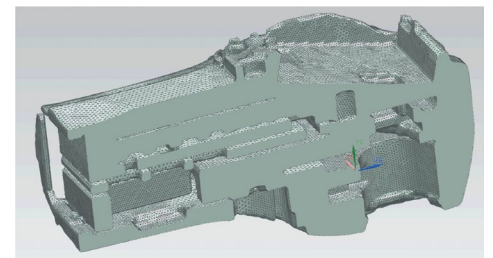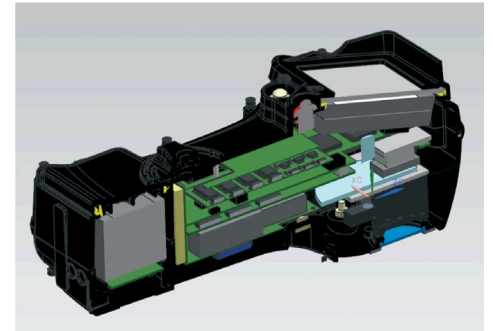
Fluid domain mesh of a camera interior.

Modeling:
- Generate polygon fluid body geometry using a surface wrapping algorithm
- Duct with mass flow network to define fluid networks with fixed mass flows

Solutions:
- Run flow analyses and coupled thermal and flow analyses in parallel on a single workstation using up to 8 processes with NX Advanced Flow
- Model two-phase, immiscible fluid simulation problems such as water sloshing in a bottle or fuel levels in a gas tank

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 thermo-fluid simulation. In NX 8, NX Thermal includes enhancements to file storage, modeling, solver and solution set up, multi-physics, and postprocessing.

File storage:
- Store catalogs in any folder

Modeling:
- Support of axisymmetric thermal elements in non-axisymmetric solutions

Temperature distribution on an axisymmetric light bulb and a non-axisymmetric filament.

- Use new independent variables when defining the magnitude of select boundary conditions
- New thermal coupling options in Surface-to-Surface Contact
- Relaxation of mapped temperature bounds
- Mesh and boundary conditions of the two models need not be identical when the initial conditions for the current model solution are taken from another model's solution in a different directory
- Preview the material orientation vector
- Spatial vector field for orthotropic thermal conductivity override

Solver and solution set up:
- Improvements to calculating total conductance and total resistance thermal coupling
- View factor calculation based on view factor residual
- Define customer defaults for options in the Solution and Solver Parameters dialog boxes
- Exponential advection method for duct thermal discretization
- Element properties in thermal solver parameters
- Track the convergence of the thermal solver during the solve
- Track the results from the full model during the solve
- Track and report minimum, maximum or average temperatures during a solve
- Select initial condition time step from a previous solution
- Specify duct flow solver options

Multi-physics:
- Transverse temperature gradient mapping to structural solutions for Nastran, Abaqus, and Ansys input files

Postprocessing:
- Recover transverse temperature gradient results on multi-layer shells

NX Advanced Thermal NX Advanced Thermal extends the modeling and simulation capabilities of NX Thermal or NX Electronic Systems Cooling. 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®. NX Advanced Thermal includes these additional modeling enhancements:
- Duct with mass flow network to define fluid networks with fixed mass flows
- New Film Cooling constraint to implicitly model the cooling of a surface by a fluid film
- Thermal cyclic symmetry using perfect contact thermal coupling
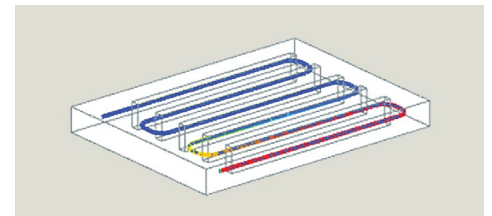
Duct with mass flow network.

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 8 include all of the NX Advanced Thermal improvements plus enhanced video export capabilities.
- Export orbit animations to videos (ASF, AVI, FLV, MOV, MPG, SWF, WMV)

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
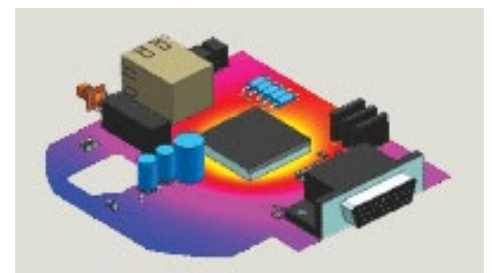
systems. NX 8 includes capabilities introduced in NX Advanced Thermal and NX Advanced Flow along with these new capabilities:

- More easily define the conductive and convective properties of a printed circuit board (PCB), and the thermal models of the electronic components
- Create a board mesh with orthotropic thermal conductivity in a new Simulation file

Structural analysis solutions

NX Laminate Composites NX Laminate Composites is a toolset for modeling laminate composite structures for analysis. Easy-to-use ply and laminate definition tools enable you to quickly create finite element models representing the laminate composite design. NX 8 improvements for laminate composites include the FiberSIM interface, modeling, postprocessing and draping capabilities:

FiberSIM Interface
- Export laminate physical properties to the FiberSIM PDI format to either Excel or a tab separated text file
- Export global plies in a global layup to FiberSIM using the HDF5 format
- FiberSIM ply name in Ply Description field

Modeling
- Solid Laminate physical property for NX Nastran environment
- Nastran support for Extrude Laminate and Fill Laminate commands
- Material orientation vectors for extruded elements in both Nastran and Ansys environments
- Quickly define a layup using the lamination shorthand format
- Fill laminate overview to create a 3D laminate between a set of Ansys 2D dependent meshes
- Output laminate physical property table to Nastran's PCOMPG
- Improved performance for importing plies

- Automatic grouping of layups and global plies
- View the laminate physical property for a selected element
- Element selection for layup offset and material orientation
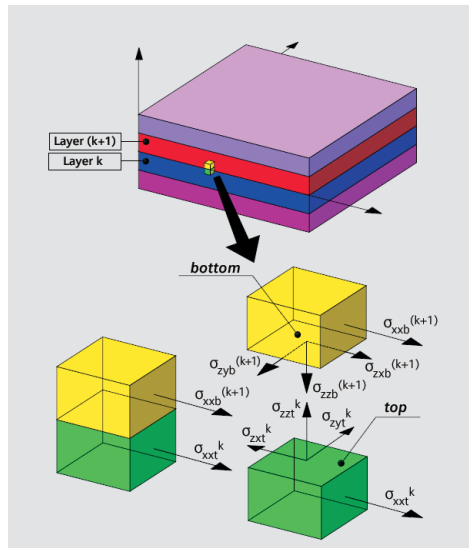
Postprocessing
- Laminate post processing enhancements
- Support for PSHELL in Advanced Post Report

Draping
- Select faces or 2D elements that are not contiguous when draping global plies using the projection draping solver
- Display draped fibers that shear beyond the lock angle for all plies of the selected global layup

NX Response Simulation NX Response Simulation 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.
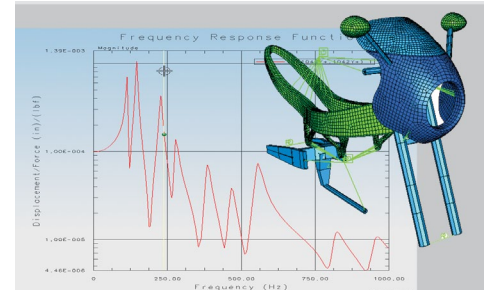
- Specify a DDAM excitation with explicit coefficients for calculating the reference acceleration and reference velocity.
- Change the minimum acceleration (minimum g) value used in the equation that calculates the shock design value of acceleration

NX Design Simulation NX Design Simulation delivers design-integrated structural and thermal simulation tools that help engineers compare design alternatives and optimize performance characteristics of products from the earliest stages of the design process. NX 8 includes enhancements to NX Design Simulation in the areas of meshing, optimization and postprocessing.

Meshing
- New mesh control capabilities to more precisely control the number and distribution pattern of elements that the software creates along a selected edge or face

Optimization
- Wizard-like user interface that guides you through shape optimization
- Improved shape optimization algorithm to provide a greater degree of accuracy

Postprocessing
- Output groups for further post-processing

Durability solutions

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. In NX 8, NX Advanced Durability includes enhancements for modeling, solution set up, and postprocessing.

Modeling
- Support for stress and strain results from models with defined laminate physical property tables
- Create durability functions from strain gage rosette data
- Use materials for which durability properties are predefined
- Pyramid element support
- Assembly FEM support
- Reference a NX Response Simulation results file that does not contain geometry information
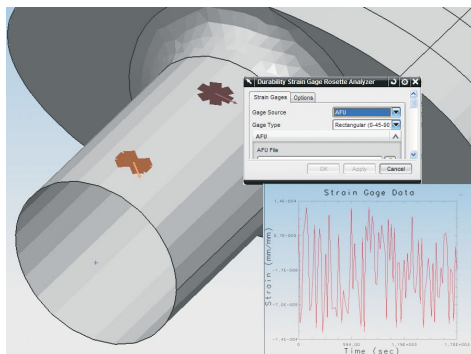
Create durability functions from strain gage rosette data.

Solution set up
- Dang Van fatigue safety factor output
- Thickness correction for BWI and Stress Life criteria
- Perform cycle counting to identify all stress or strain cycles in a duty cycle and to calculate the damage due to identified cycles

- TWI fatigue life criterion to extend the BWI fatigue life criterion into the low cycle and high cycle fatigue regimes
- BWI fatigue life criterion can now set the probability of failure value instead of the number of standard deviations
- Calculate durability results from the stress or strain results that are saved in the NX Nastran SOL109 and SOL112 SORT2 format in the OP2 file
- Show durability or event solution log
- Create diagnostic groups that show elements that do not have sufficient durability requirements
- Simulation Navigator enhancements

Postprocessing
- Export fatigue damage results from the durability solution process to an Excel or CSV file
- When a durability event is modified, its node is now flagged as obsolete in the Simulation Navigator

NX Durability Wizard NX Durability Wizard guides you in validating your product's structural integrity over its entire lifecycle. For NX 8, the NX Durability Wizard can now be extended beyond the NX CAE product suite.
- Now available from the NX Modeling application so the wizard can be used by designers for earlier, up-front evaluation of their designs

Analysis to physical test correlation

NX FE Model Correlation and NX FE Model Updating NX Finite Element (FE) 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 finite element models to match real life test data as closely as possible based on the correlated results. NX 8 improvements to NX FE Model Correlation and NX FE Model Updating include enhanced correlation setup, pre-test planning, export, postprocessing and model updating capabilities.

Correlation setup:
- U-Set DOF node supported for SEMODES 103 solution
- Correlation and model update related solution sequences supported in NX Nastran
- Automatic node map size reduction to limit the size of the node map that is created between the work solution and the reference solution
- Efficient mode shape data handling
- Manual mode pairing
- View imported complex test mode shapes
- Complex mode conversion options
- Supports UNV files that contain dataset 2414 for mode shape result data

Pre-test planning:
- Select exciters when you create a pre-test solution process
- New exciter configuration commands to create, edit or solve exciter selection configurations

Export:
- Export the active sensor selection configuration to a UNV file

Postprocessing
- Display COMAC results in postprocessing
- Correlation Metrics node in simulation navigator
- Phase adjustment for side by side mode shape display

Model updating:
- New supported design variables for NX FE Model Updating
- MSC Nastran DESOPT 200 solution support for model updating
- Sensitivity node in Simulation Navigator

Multi-body dynamics solutions

NX Motion Simulation NX Motion Simulation provides an advanced yet simple-to-use solution that enables designers and engineers to understand, evaluate and optimize the complex motion behavior of assemblies and products. When used early in the design process, designers can quickly understand if their assembly will encounter package space issues or interfere with parts inside or outside of the entire product. NX 8 includes enhancements to NX Motion Simulation for modeling and simulation, postprocessing, and flexible body dynamics:

Modeling and simulation
• Unit and expression support
• Check motion model for various problems prior to solve
• Copy and paste bushing, 2D contact or 3D contact parameters to another object of the same type
• Easier definition of joint and bushing orientation
• Curve parameterization added to Point on Curve dialog box
• User-specified names for motion simulations

Post-processing
• Exporting an animation as an AVI
• Interactive legend during animation

Flexible body dynamics
• Define packaging options, including interference, measure and trace
• Highlight the step in which the minimum or maximum stress, displacements or other result types occurred
• Plot modal degrees of freedom vs. time
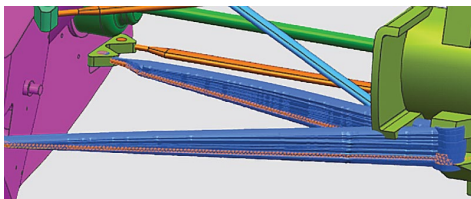
Trace packaging on a flexible link.

• Data recovery solve can be limited to only relevant steps
• Post View settings are now preserved as long as the motion simulation is open in NX

NX Nastran 8.0

Available as a standalone enterprise finite element 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. NX Nastran 8.0 delivers enhancements in dynamics, acoustics, elements and superelements, contacts and glue, non-linear analysis, optimization, DMP RDModes, numerical and output control.

Dynamics
• Request residual vectors
• Modal contributions capability has been enhanced to allow selection of panels for modal contribution output and allow examination of panel contributions to the structural dynamic response
• MODSEL case control command has been enhanced to exclude a list of modes from the response calculation, include different MODSEL case control commands or combinations of MODSEL case control commands in subcases, and use MODSEL case control commands in a restart run
• The IC case control command beta capability introduced in NX Nastran 7.1 is now fully supported and has been further enhanced to account for the effects of differential stiffness in a direct transient analysis (SOL 109) or a modal transient analysis (SOL 112), and to use the static deflection in physical space resulting from the loading at time = 0 as the initial condition for a direct transient analysis (SOL 109)
• Define the analysis set for static condensation in a SOL 101 analysis
• Multi-body dynamics and control system software interface enhancements
• SOL 111 improved accuracy with modal damping and frequency dependent stiffness

Acoustics
• New calculation method for acoustic source strength
• Fluid-structural coupling created by SFE AKUSMOD can be imported into a NX Nastran dynamic analysis
• Request the output of the fluid-structure coupling matrix AGG to the punch file

External superelements
• Request the software to include the full finite element geometry with the external superelement output
• New method for storing loads with an external superelement is available

Element enhancements
• Material nonlinear capabilities of solid elements has expanded support for support plasticity, nonlinear elasticity, and creep to parabolic CHEXA and CPENTA elements and linear and parabolic CPYRAM elements
• Total strain output in SOLs 106 and 129
• Request element property summary (ELSUM) be grouped according to property type, include only mass totals for the element type or property type grouping, and include non-structural mass in the mass totals for the element type or property type grouping
• ELSUM case control command is supported in SOLs 601 and 701
• Enhancements for plane stress and plane strain elements
• Edge loads for CPLSTN and CPLSTS elements
• Composite laminate solid elements
• Alternate material coordinate system specification method for solid elements
• Rigid element processing
• Element, property and material selection tables

Contact for linear solutions
• Contact conditions with inertia relief
• Contact with static condensation

Glue
• Glue surface tractions for solid and shell elements

Advanced Nonlinear

- New viscoelastic material models are available for SOL 601
- Mullins effect (Ogden-Roxburgh model) for hyperelastic material
- Combine creep and elasto-plastic material models within the same SOL 601 analysis
- CBUSH element support in SOL 601 and SOL 701
- Edge load support for plane stress and plane strain elements
- MAT11 and MATCID support in SOL 601 and SOL 701
- DMIG support in SOL 601
- Stress output at Gauss points for solid and axisymmetric elements
- Variable thickness support for plane stress elements
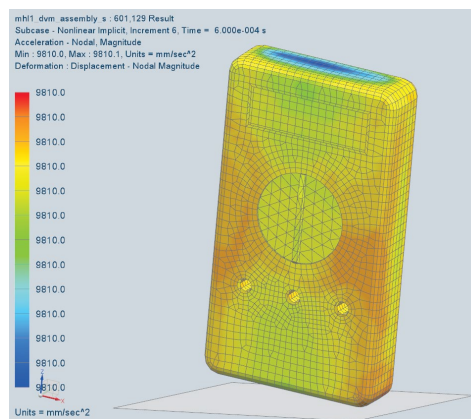
Optimization

- Designable frequency dependent properties
- Design sensitivity is supported with SOL 200 using the DSAPRT case control command
- Design cycle data block
- Improved FREQ3, FREQ4, FREQ5 handling
- Improvements in PBARL/PBEAML handling to significantly improve both computational speed and efficiency of disk use

RDModes

- Modal transient response analysis (SOL 112)
- Modal analysis with optimization (SOL 200)
- Restarts when using SOLs 103, 111 or 112 in either serial or distributed memory parallel (DMP) runs
- Logic for RDMODES is improved to minimize I/O usage
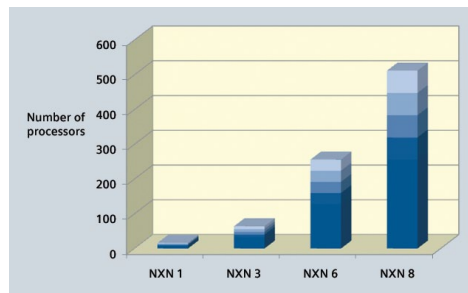
Improved scalability for RDModes.

Numerical enhancements

- FRRDRU module has been improved with additional support for structural damping
- Factor caching is introduced for the Lanczos real eigensolver

Output control for EPT and MPT data blocks

- Ability to disable the output from the EPT and MPT data blocks

Contact
Siemens Industry Software
Americas   +1 800 498 5351
Europe   +44 (0) 1276 702000
Asia-Pacific   +852 2230 3333

www.siemens.com/plm