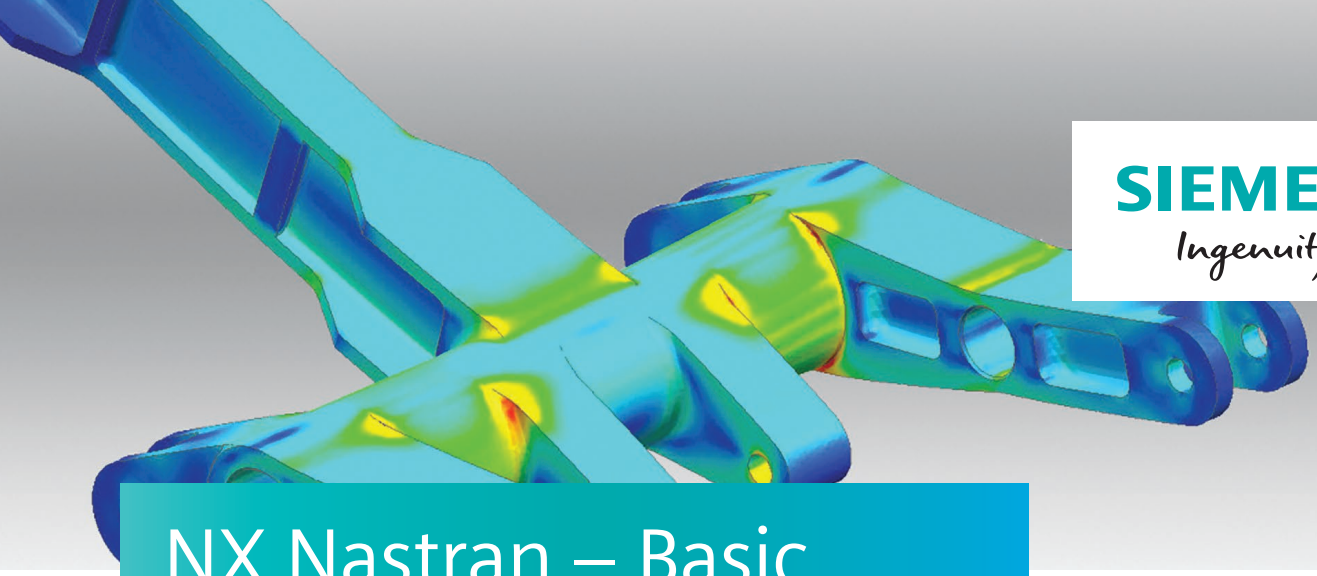


SIEMENS
Ingenuity for life

NX Nastran – Basic

The core structural analysis FEA solver used by leading product development firms for over 40 years

Benefits

- Reduce risk by using simulation to save time and cost compared to physical test cycles
- Accelerate innovation through rapid iteration and numerous "what-if" studies
- Investigate product performance virtually under all possible operating conditions, including thermally-influenced operating conditions

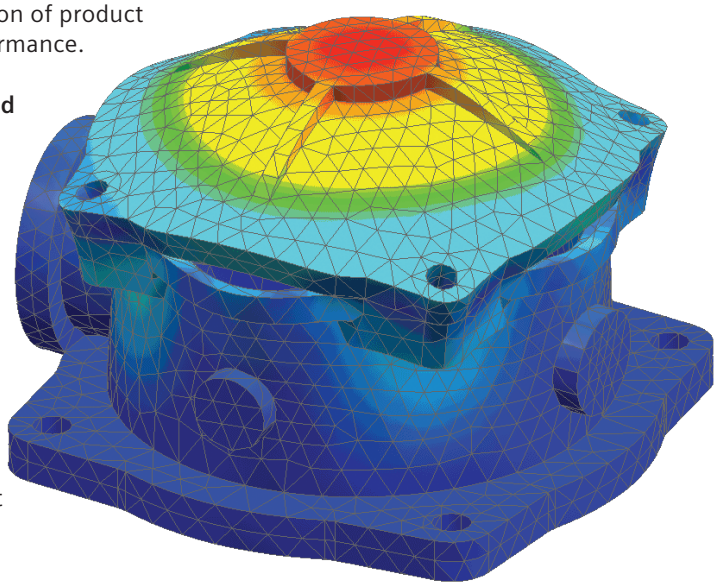
Summary

NX™ Nastran® is part of the Simcenter™ portfolio of simulation tools, and is used to solve structural, dynamics and acoustics simulation problems. NX Nastran – Basic is the core subset of NX Nastran software and includes a robust suite of linear statics, normal modes, buckling analyses, heat transfer and basic nonlinear capabilities. NX Nastran – Basic can play a key role in your virtual product development process by providing the most widely used CAE solutions for digital prototyping and simulation of product functional performance.

Start simply, add as your needs evolve

NX Nastran – Basic will allow you to initiate digital simulation into your product development process by providing access to a broad library of finite element types and material models, robust manipulation of load cases, along with several efficient solution sequences for linear statics (including buckling) and normal modes analyses on models of unlimited size. You can also perform sensitivity studies based on these analysis types. NX Nastran's powerful analysis capabilities will provide you with the tools you need for:

- Linear statics, including inertia relief
- Normal modes
- Buckling
- Design sensitivity (statics, modes, buckling)
- Model checkout
- Heat transfer
- Basic nonlinear
- Coupled structure/acoustic modes



www.siemens.com/plm/nxnastran

NX Nastran – Basic

NX Nastran – Basic includes a complete element library including 1D, 2D and 3D low- and higher-order elements; scalar and special elements including spot weld as well as p-elements (that can be combined with other elements).

Solution types supported by NX Nastran – Basic

SOL Number	SOL Name	Description
101	SESTATIC	Linear statics
103	SEMODES	Normal modes
105	SEBUCKL	Buckling
106	NLSTATIC	Nonlinear or linear statics
114	CYCSTATX	Cyclic statics
115	CYCMODE	Cyclic normal modes
116	CYCBUCKL	Cyclic buckling
129	NLTRAN	Nonlinear or linear transient response
153	NLSCSH	Static structural and/or steady state heat transfer analysis
159	NLTCSH	Transient structural and/or transient heat transfer analysis

Table 1 – Element types supported by NX Nastran – Basic

Element type	Element name	Description
Scalar	ELAS	Scalar spring (several variations)
	MASS	Scalar mass (several variations)
1D	BAR	Simple beam element
	BEAM	Complex beam element including shear center offset and variable cross section
	BEND	Curved beam, pipe or elbow
	ROD CONROD TUBE	Rod element tension-compression-torsion element
2D	QUAD4	Quadrilateral plate with membrane-bending or plane strain behavior
	QUAD8	Higher-order quadrilateral shell element
	QUADR	Quadrilateral membrane or shell
	SHEAR	Shear panel
	TRIA3	Triangular plate with membrane-bending or plain strain behavior
	TRIA6	Higher-order triangular shell element
	TRIA	Triangular membrane or shell
	CPLSTN3	Plane strain triangular element connection
	CPLSTN6	Plane strain high-order triangular element connection
	CPLSTN4	Plane strain quadrilateral element connection
	CPLSTN8	Plane strain high-order quadrilateral element connection
	CPLSTS3	Plane stress triangular element connection
	CPLSTS6	Plane stress high-order triangular element connection
	CPLSTS4	Plane stress quadrilateral element connection
	CPLSTS8	Plane stress high-order quadrilateral element connection
3D	HEXA	Six-sided solid element with 8-20 grid points
	PENTA	Five-sided solid element with 6-15 grid points
	TETRA	Four-sided solid element with 4-10 grid points
	PYRAMID	Five-sided solid element with 5-13 grid points
Rigid	RBAR	Rigid bar element
	RBE1	Rigid body connected to an arbitrary number of grid points
	RBE2	Rigid body with independent DOFs at a grid point and dependent DOFs at an arbitrary number of grid points
	RROD	Pin-ended rigid rod
	RTRPLT	Rigid triangular plate
Interpolation	RBE3	Defines motion of a reference point as the weighted average of the motions at a set of grid points
	RSPLINE	Multipoint constraints for the interpolation of displacements at grid points
Composites	BEAM	Complex beam element
	QUAD4	Quadrilateral plate
	QUAD8	Higher-order quadrilateral plate
	QUADR	Quadrilateral plate

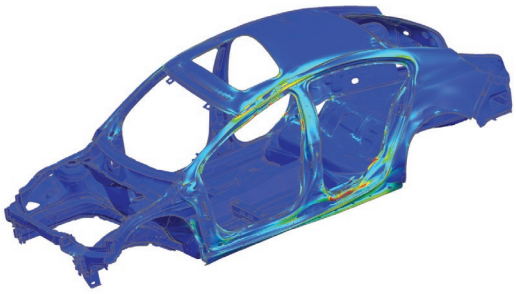


Table 1 ***continued***

Element type	Element name	Description
	TRIA3	Triangular plate
	TRIA6	Higher-order triangular plate
	TRIAR	Triangular plate
	CHEXA	Six-sided solid element with 8-20 grid points
	CPENTA	Five-sided solid element with 6-15 grid points
p-elements	HEXA	Six-sided solid element with 8-20 grid points
	PENTA	Five-sided solid element with 6-15 grid points
	TETRA	Four-sided solid element with 4-10 grid points
Axisymmetric	CONEAX	Conical shell
	TRIAX6	Triangular cross section ring
	CTRAX3	Triangle
	CTRAX6	Higher-order triangle
	CQUADX4	Quad
	CQUADX8	Higher-order quad
Crack tip	CRAC2D	Two-dimensional crack tip element
	CRAC3D	Three-dimensional crack tip element
General	CONM1	6-by-6 symmetric mass matrix
	CONM2	Concentrated mass with offsets
	DMI	Direct matrix input
	GENEL	General element
Weld	CWELD	Weld connection element
Fastener	CFAST	Fastener element for shell patch connection

NX Nastran – Basic provides a full range of material models: isotropic, orthotropic, anisotropic and temperature-dependent. It also allows for easy combination (or addition) of load cases, such as point, line and surface loads on elements; loads applied directly to geometry; thermal loads; enforced deformation; and weighted combinations of each type.

Table 2 – Static loading types in NX Nastran – Basic

Load type	Load name	Description
Point	FORCE	Concentrated force (several variations)
	MOMENT	Concentrated moment (several variations)
Curve	GMLOAD	Load distributed along a geometric curve
	PLOAD1	Concentrated, uniform or linear load applied to 1D elements
Edge	PLOADE1	Edge load on plane strain and place stress elements
Surface	GMLOAD	Load distributed along a geometric surface
	PLOAD	Pressure load applied to 2D elements or the face of 3D elements (several variations)
Volume	GRAV	Steady-state acceleration vectors
	RFORCE	Angular velocity or acceleration
	ACCEL	Spatial varying acceleration load
Bolt preload	BOLTFOR	Bolt preload applied to beam elements
Enforced	GMBC	Enforced displacements for geometry motion (curves motion and surfaces)
	GMSPC	Constraints applied to geometry
	SPC	Constraints applied to grid points (several variations)

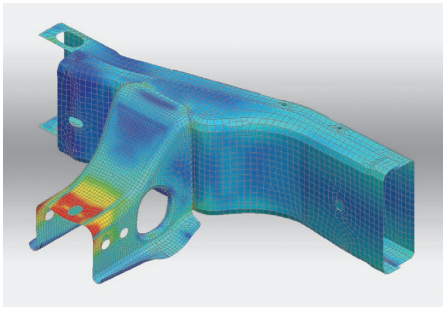


Table 2 *continued*

Load type	Load name	Description
Thermal	TEMP	Temperatures applied to grid points (several variations)
	TEMPP1	2D element temperature field
	TEMPRB	1D element temperature field
Axisymmetric	FORCEAX	Concentrated force
	MOMAX	Concentrated moment
	PLOAD1X	Surface traction
	PRESAX	Pressure loading
	SPCAX	Constraints
	TEMPAX	Applied temperatures
General	DMI	Direct matrix input
Combination	LOAD	Combine load sets
Non-structural mass	NSM	Non-structural mass sets

NX Nastran – Basic provides several non-elemental approaches for connecting meshes and transferring loads. This can greatly simplify modeling procedures.

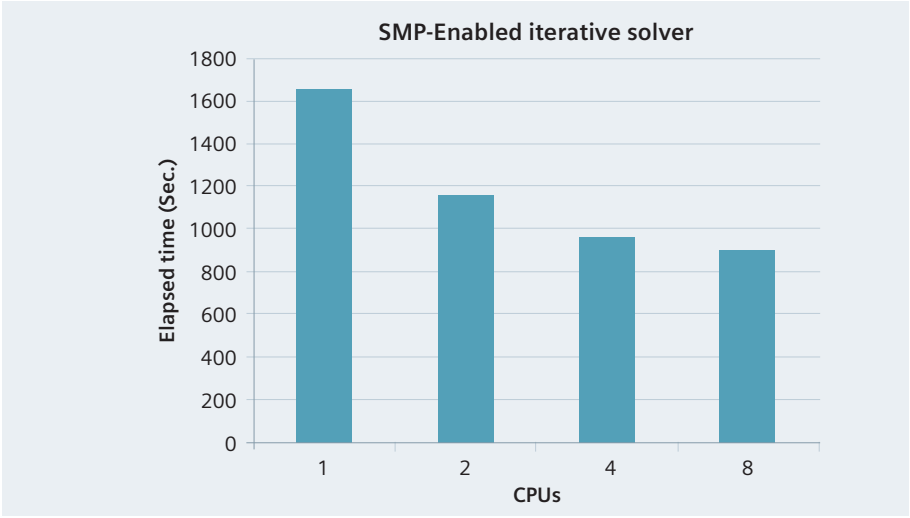
Table 3 – Non-elemental mesh connections in NX Nastran – Basic

Type	Name	Description
Constraint	MPC	Constraint equations used to connect specified degrees-of-freedom
	RSSCON	Constraint relation to connect shell to solid elements
Contact	BSURF	A set of shell element faces that define a contact surface
	BSURFS	A set of solid element faces that define a contact surface
	BCTSET	Pairs of contact surfaces that can contact in a linear static solution
Glue	BSURF	A set of shell element faces that define a glue surface
	BSURFS	A set of solid element faces that define a glue surface
	BGSET	Pairs of glue surfaces that are in connection in any solution type
	BLSEG	Defines a glue or contact edge region, or a curve for slideline contact

Additional capabilities for linear static and Eigenvalue solutions

Linear static solutions

- Edge-to-surface contact to glue the edges of shell elements to the faces of solid or shell elements
- Surface-to-surface contact for shell and solid elements
- Edge-to-edge glue between the edges of shell, axisymmetric, plane stress and plane strain elements
- Inertia relief for unrestrained models
- Shared memory parallel (SMP) processing enabled element-based iterative solver for very fast solutions of tetrahedron meshed models
- Bolt preload effects
- Thermal expansion for rigid elements



SMP-enabled iterative solver reduces linear static solution time by as much as 45 percent on 8 CPUs for higher-order solid models.

Normal mode solutions

- Lanczos
- Residual vectors for residual flexibility
- Differential stiffness effects
- Unconstrained model solutions
- Solution about a contact condition
- Export modes to ADAMS or RecurDyn

Design sensitivity analysis for assessing design changes

- Shape and sizing design variables
- Preset objective and constraints
- Weight, volume
- Element stress, strain, force
- Displacement, rotation, reaction force
- Normal modes Eigenvalue
- Buckling load factor
- Composites: lamina strain, force and failure index
- User-defined objective and constraints
- Efficient handling of hundreds of design variables, constraints and load cases buckling in a single run

Efficient solvers

- Sparse matrix solvers for faster speed and minimal disk space usage
- Automatic internal resequencing for bandwidth reduction
- Restarts to take advantage of previously computed solutions

Steady-state and transient thermal analysis

NX Nastran – Basic provides heat transfer solutions to steady-state and transient thermal analysis design problems. This capability may also be used in combination with NX Nastran structural analyses to perform thermal stress analysis.

If changes in temperature and the flow of heat within your product could affect its performance, heat transfer should play a key role in your digital simulation process. Heat transfer can span the full range from system-level analysis of global energy balances to the detailed analysis associated with temperature and thermal stress limit levels. It allows you to investigate linear or nonlinear problems, steady-state or transient effects, as well as all three types of heat transfer (conduction, convection and radiation), displaying the characteristics associated with each.

Heat conduction

- Temperature-dependent conductivity
- Temperature-dependent specific heat
- Anisotropic thermal conductivity
- Latent heat of phase change
- Temperature-dependent internal heat generation
- Weighted temperature gradient-dependent internal heat generation
- Time-dependent internal heat generation

Free convection boundaries

- Temperature-dependent heat transfer coefficient
- Weighted temperature gradient-dependent heat transfer coefficient
- Time-dependent heat transfer coefficient
- Nonlinear functional forms
- Weighted film temperatures

Forced convection
- Tube fluid flow field relationships
- Temperature-dependent fluid viscosity, conductivity and specific heat
- Time-dependent mass flow rate
- Temperature-dependent mass flow rate
- Weighted temperature gradient-dependent mass flow rate

Radiation to space
- Temperature-dependent and wavelength-dependent emissivity
- Diffuse 3D view factor calculations with self and third-body shadowing
- Adaptive view factor calculations
- Net view factors
- User-supplied exchange factors
- Radiation matrix control
- Multiple radiation enclosures

Applied heat loads
- Direction and surface normal heat flux
- Grid point nodal power
- Temperature-dependent and weighted gradient-dependent heat flux
- Time-dependent heat flux
- Temperature boundary conditions
- Temperature initial conditions

Basic nonlinear analysis
NX Nastran – Basic enables you to analyze models with geometric nonlinearities; that is, large deformations or with material nonlinearities. Point-to-point contact nonlinearity can also be simulated. This basic nonlinear capability allows users to evaluate whether the small displacement and linear material assumptions used in linear analysis are accurate.

Geometric nonlinear behavior
- Large deformations
- Large strain for hyperelastic material
- Snap-through analysis (post-buckling)

Material nonlinear behavior
- Plasticity
- Hyperelasticity
- Thermoelasticity
- Viscoelasticity (creep)

Automated solution methods – statics
- Load control method
- Displacement control method
- Adaptive load increment

Other features
- Static and transient solutions
- Restart analysis
- Identical element types in linear and nonlinear analysis
- Point-to-point contact with gap elements

Siemens PLM Software
www.siemens.com/plm

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

2661-A13 11/16 C