

Simcenter Nastran Multistep Nonlinear

Solution 401/402

Predict behavior of parts or assemblies with nonlinear contact, nonlinear material or large deformations

Benefits

- Reduce design risk by using simulation to save time and cost compared to physical build-and-break test cycles
- Accelerate your innovation through rapid iteration and numerous “what-if” studies
- Use the same models already built and analyzed linearly with Simcenter Nastran – Basic
- Improve confidence in final designs by virtually investigating your product’s performance under all possible operating conditions
- Obtain more accurate solution results than linear analysis when standard linear assumptions are not valid

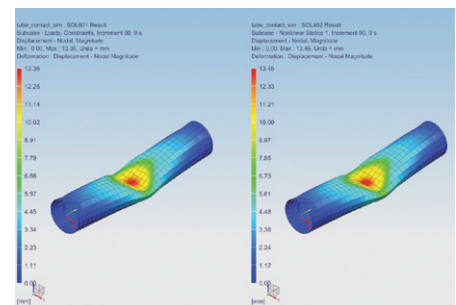
Summary

The Simcenter™ Nastran® Multistep Nonlinear software solution enables you to analyze models with nonlinearity from contacting parts, material property nonlinearities and/or geometric nonlinearities (that is, large deformations and large strains). This solution is an add-on module of Simcenter Nastran – Basic in either enterprise or desktop versions. The Multistep Nonlinear license provides access to both the SOL 401 Multistep Nonlinear solver and the SOL 402 Multistep Nonlinear Kinematic solver. Both use nonlinear implicit methods based on similar formulations. SOL 401 is implemented in the standard Simcenter Nastran architecture and leverages the same code base as other Simcenter Nastran solutions. SOL 402 is an integration of the well-known and highly regarded Simcenter Samtech Samcef™ solver into Simcenter Nastran. Either solver can be used to solve a large class of nonlinear solutions and give very similar results. Other classes of nonlinear solutions may be better solved using one or the other. Users can

refer to the Simcenter Nastran documentation to see which solver may best support their applications or consult with the Siemens support organization.

The multistep structure of the SOL 401/402 solutions provides powerful and flexible solution capabilities. Multistep solutions are important in nonlinear simulation because unlike linear simulations, the sequence of loading affects the results. With multistep capability, users can step through different types of solutions all in one solution run. For example, you can apply bolt preloads with contact definitions in the first step, follow with a set of service loads in the next step, and then compute modes about the deformed loaded condition in still another step. Any number and type of steps are allowed. This gives users the ability to simulate assembly sequences or modal frequency effects from loading and contact, and to apply pre-stress before starting a dynamic solution.

There are many reasons for users to go beyond linear solution methods and look to nonlinear. For example, geometric nonlinear effects need to be



Plasticity

Simcenter Nastran Multistep Nonlinear

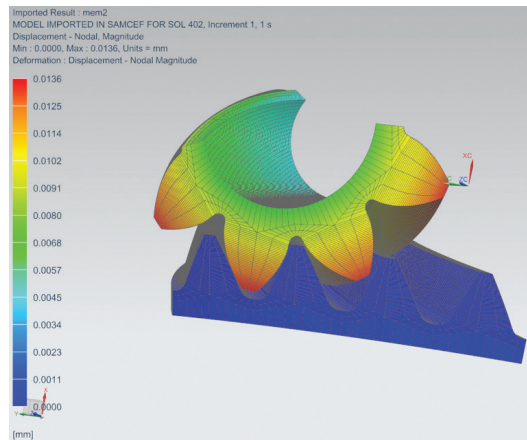
simulated when stiffness properties or loads change significantly as the result of deformation. Analysis of snap-through buckling is an example in which geometric nonlinearity effects are important. Material nonlinear effects should be modeled when the material properties cannot be considered linear for the loading conditions considered. Example usages are for analyzing hyper-elastic (rubber) materials or analyzing metals that exhibit plastic behavior because they are stressed beyond yield limits. Advanced contact capabilities allow you to simulate surface contact using either shell or solid elements. Many mechanical simulations involve parts coming into contact under load. With the advanced Solution 401/402 surface contact capabilities, the solver determines the extent of surface contact and load transfer across the contacting surfaces as part of the solution.

Both Simcenter Nastran Multistep Nonlinear solutions also have very robust solution algorithms and efficiently obtain converged solutions for some of the most difficult and intractable nonlinear models. The input and output formats for SOL 401 and 402 are also very similar and it is very easy to set up a Nastran model to run for either solution. These solutions offer the following capabilities.

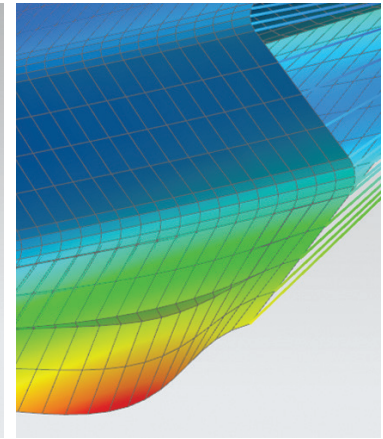
Major capabilities

Solution step types

- Static
- Dynamic
- Preload
- Modal
- Buckling
- Cyclic symmetry modes
- Fourier harmonic modes



Contact conditions



Composite delamination

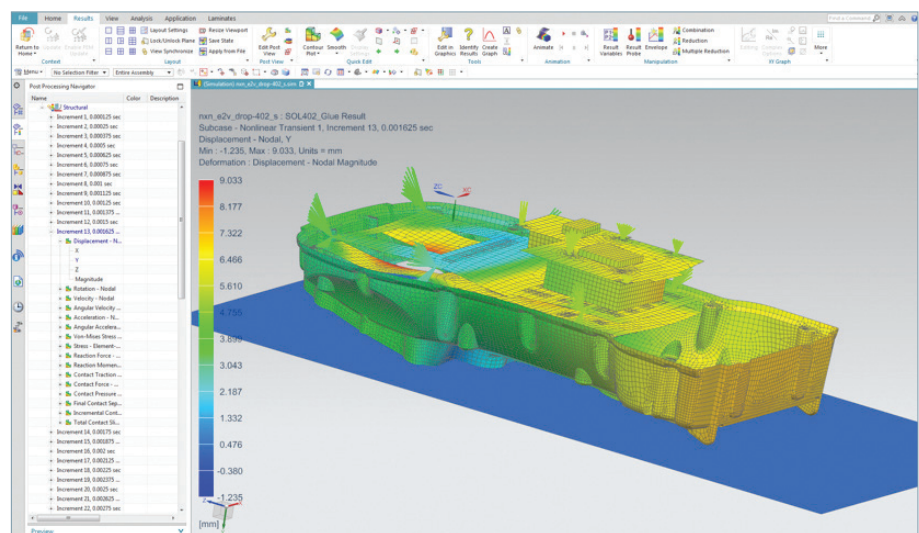
Contact

- Shell and solid element face contact
- Edge contact for axisymmetric modeling
- Single- and double-sided contact
- Self-contact
- Multiple friction models
- Tied contact
- Contact surface offsets
- Gap elements

- Contact activation/deactivation per subcase
- Contact pressures and force results
- Contact separation and sliding results

Geometric nonlinear

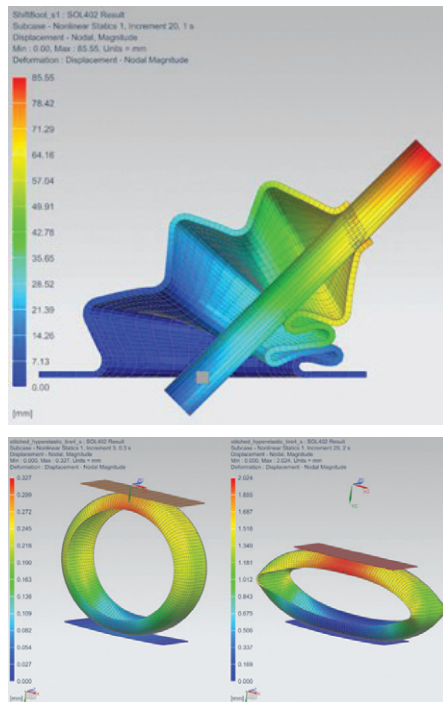
- Large deformations
- Large strain
- Snap-through analysis (post-buckling)
- Follower forces



Drop test

Material nonlinear

- Hyperelasticity models
 - Mooney-Rivlin
 - Ogden
 - Hyperfoam
 - Mullins effect
 - Damping with Prony series
- Elasto-plastic
 - Von Mises yield criterion
 - Isotropic hardening
 - Kinematic hardening
 - Mixed hardening
- Thermal elasto-plastic
- Creep
- Plasticity
- Strain measures: engineering, logarithmic
- Combined creep and elasto-plastic



Hyperelastic

Composite modeling

- Ply layers for solid or shell elements
- Failure indices
- Strength ratios
- Cohesive delamination
- Progressive damage

Element types

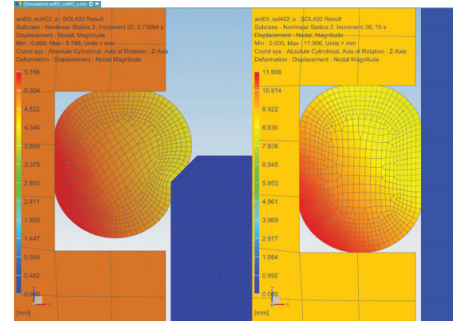
- 2D and 3D solids
- Shell
- Beams
- Springs and bushings
- Rigid
- Mean elements (RBE3)
- Plane stress
- Plane strain and generalized plane strain
- Chocking

Other modeling capabilities

- Fixed and sliding glue connections
- Cyclic symmetry boundary conditions
- Mesh imperfections
- J-integral for crack simulation
- Co-simulation with thermal solver

Robust solution methods

- Full Newton iterations, with or without line searches
- Arc length solution method
- Automatic time stepping (ATS) method
- Energy, force and deformation convergence criteria
- Dynamic solution with Newmark method for direct implicit integration
- Sparse solver and iterative solver
- Stiffness stabilization for static solutions
- High-performance computing (HPC) on multiprocessors



Hyperelastic and axisymmetric computation

Load conditions

- Bolt preload
- Concentrated loads
- Distributed loads
- Follower loads
- Pressure loads
- Inertia loads
- Enforced motion
- Applied temperatures
- Initial conditions for displacement, velocity and temperature
- Initial stress/strain conditions
- Edge load support for plain stress and plain strain elements

Siemens Digital Industries Software
[siemens.com/plm](https://www.siemens.com/plm)

Americas +1 314 264 8499
 Europe +44 (0) 1276 413200
 Asia-Pacific +852 2230 3333

Restricted © 2019 Siemens. Siemens and the Siemens logo are registered trademarks of Siemens AG. Femap, HEEDS, Simcenter, Simcenter 3D, Simcenter Amesim, Simcenter FLOEFD, Simcenter Flomaster, Simcenter Flotherm, Simcenter MAGNET, Simcenter Motorsolve, Simcenter Nastran, Simcenter Samcef, Simcenter SCADAS, Simcenter STAR-CCM+, Simcenter Soundbrush, Simcenter Sound Camera, Simcenter Testlab, Simcenter Testxpress and STAR-CD are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries or affiliates in the United States and in other countries. All other trademarks, registered trademarks or service marks belong to their respective holders. 67117-C7 6/19 H