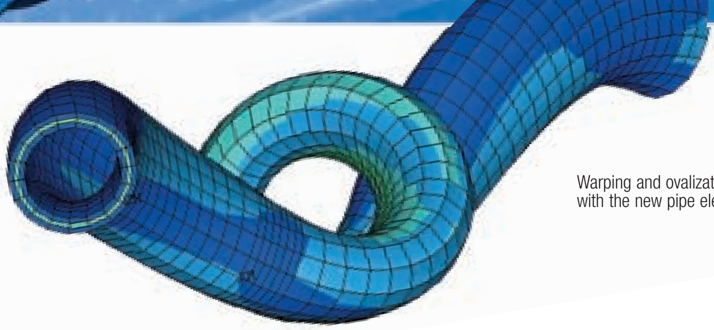


Warping and ovalization of pipe structures with the new pipe elements

# Designing with **Structure**

## Advancements in structural mechanics allow more efficient and higher-fidelity modeling of complex structural phenomena.

The ability to drive the engineering design process in structural applications has taken a significant step forward with the improvements in release 12.0. New features and tools, many integrated into the ANSYS Workbench platform, help reduce overall solution time. Specific improvements focus on elements, materials and contact and solver performance, along with linear, rigid and flexible dynamics.

### Elements

The most notable new element in release 12.0 is the four-noded tetrahedron for modeling complex geometries in hyperelastic or forming applications. The element provides a convenient way to automate the meshing of complex structures, avoiding the need for pure hexahedral meshes. This reduces the time it takes to develop a case from geometry through solution, while maintaining the accuracy of the solution. See the table below for a summary of new and enhanced elements.

When simulating a nonlinear process, large deformation can introduce too much distortion of the elements. Resolving this requires local remeshing during the simulation process. The 2-D rezoning introduced with release 11.0 extends further in ANSYS 12.0, increasing the flexibility of the remeshing process: The user can now define transition regions within the refined zones and use meshes created in external meshing tools.

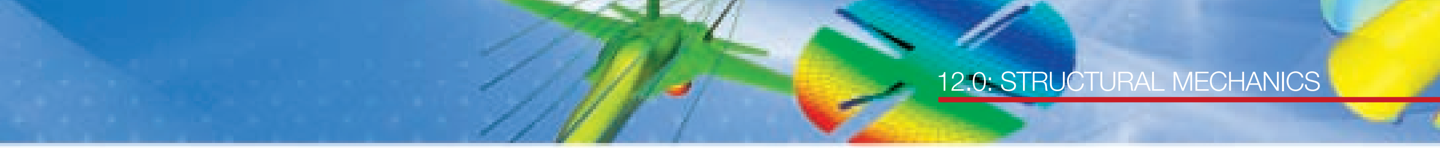
### Materials

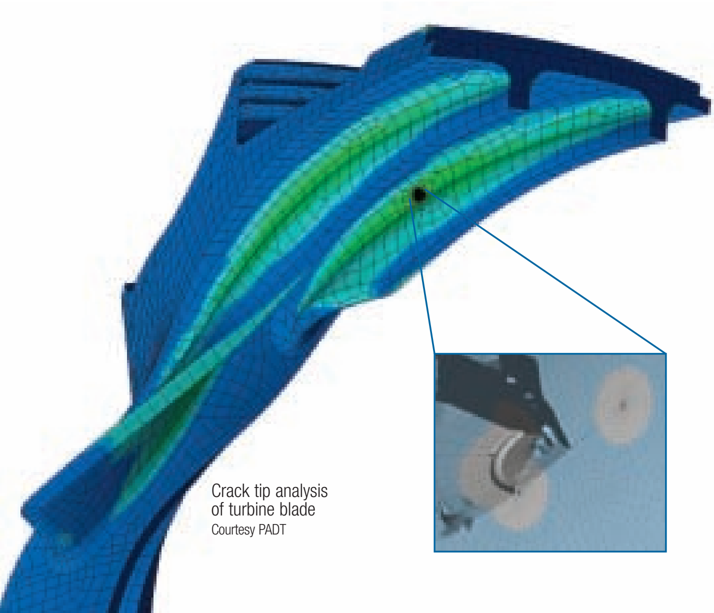
Accounting for proper cyclic softening or hardening or damage of materials is a key factor for elastomer applications and, more generally speaking, any structure whose material variation depends on the strain rate. Release 12.0 introduces several additions to the wide choice of materials already available. Other feature improvements include:

- Rate-dependent Chaboche plasticity, which can benefit turbine and engine design
- Bergström–Boyce model to enhance elastomer modeling capabilities
- New damage model based on the Ogden–Roxburgh formulation

| Element | New | Improved | Capability | Applications |
|---|---|---|---|---|
| Four-noded tetrahedron | X | | Provides a convenient way to automate meshing of complex structures, avoiding need for pure hexahedral meshes | Modeling complex geometries for forming or hyperelastic applications |
| General axisymmetric element | X | | Supports contact | Compatible with 3-D non-axisymmetric loading and can use arbitrary axis of rotation |
| Various pipe model elements | X | | Increased accuracy | To provide refined behavior of structures in case of ovalization, warping or similar deformations of cross section for thin or moderately thick pipes and nonlinear material behavior support |
| Shell: linear, quadratic, axisymmetric | | X | Improved shell thickness updating scheme and improved convergence | Provides greater accuracy in the behavior of shell models as well as a faster solution for nonlinear problems |
| Beam | | X | Supports cubic shape function | Provides additional accuracy to coarse meshes and greater support of complex load patterns |
| Reinforcement elements | | X | Allows modeling of discrete fibers with a variety of nonlinear material behavior | Stresses in reinforcements can be analyzed separately from host elements |

Summary of new and enhanced element features in ANSYS 12.0 structural analysis products





Crack tip analysis of turbine blade
Courtesy PADT

- Anand's viscoplasticity model, useful for metal forming applications such as solder joints
- Improvements in the calculation of J-integrals to account for mixed-mode stress intensity factors, which benefit improvements in fracture mechanics
- Initial strain and initial plastic stress import capabilities that allow for state transfer from a 2-D model to a 3-D model

## Contact

As assemblies have become a de facto standard in simulation, the need for advanced contact features has grown accordingly. ANSYS 12.0 developments include a number of additional contact modeling features as well as significant improvements in solving contact problems.

While Coulomb's law for friction is widely used, there are circumstances in which more elaborate modeling is required, such as wear modeling or pipelines resting on sea beds. Release 12.0 supports a friction coefficient definition that depends upon the contact state itself and accounts for complex frictional behavior. Specifically, the user is able to define the dependency of the friction on contact parameters, such as sliding distance or contact pressure.

A typical contact application involves seals that are subject to fluid pressure. Release 12.0 provides support of fluid pressure penetration, to model scenarios in which pressure rises higher than the contact pressure around the seal. Pressures in such cases can be applied only on the free faces of the structure and evolve with the contact state.

Contact simulation is usually a time-consuming process. The latest release introduces contact modeling improvements that significantly reduce computation time and results file size. These enhancements include new contact search algorithms, contact trimming logic and smart over-constraint elimination for multipoint constraint (MPC) contact.

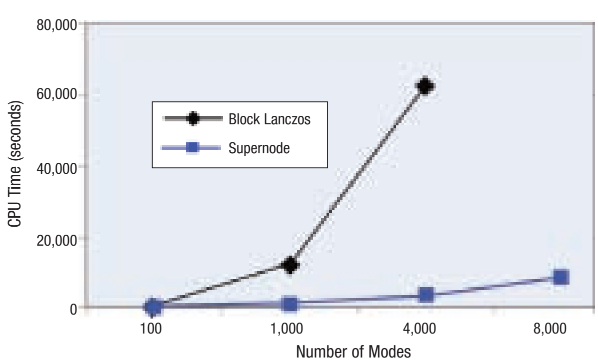
## Solver Performance

Solver performance has improved in many different areas. ANSYS 12.0 introduces a new modal solver, called SNODE, that increases the speed of computation for problems with a large number of modes — in the realm of several hundred — on large structures that typically have over a million degrees of freedom. This solver is well suited for automotive or aerospace applications and for large beams and shell assemblies. Beyond its ability to compute a larger number of modes in a reduced amount of time, SNODE also significantly reduces the amount of I/O required to compute the solution. (See the Supernode Eigensolver article.)

Many enhancements have been made to the distributed solver to improve the scalability of the solution. (See the article on High Performance Computing.) More solver techniques are supported, including:

- Partial solve capability that computes only a portion of the solution
- Prestressed analysis
- Models that employ the use of unsymmetric matrices, which are useful for scenarios that involve high-friction coefficients, for example

These new features can be combined for applications such as brake squeal, which might combine the partial solve and unsymmetric matrix capabilities.

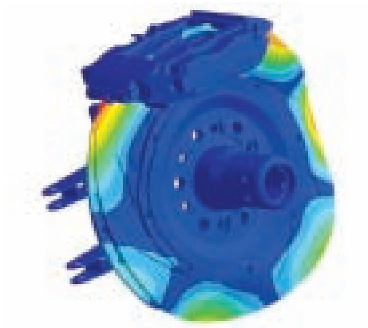



Performance of new modal solver

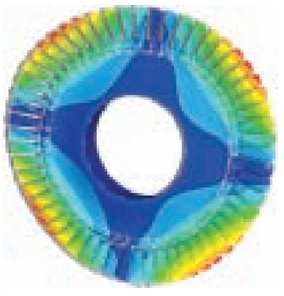
## Linear Dynamics

Some of these element, material, contact and solver improvements benefit the field of linear dynamics as well. They are complemented by enhancements specific to this simulation area, especially for mode superposition analysis. For harmonic or transient loadings, the mode superposition methods exhibit better performance, especially during the so-called expansion pass that computes results at each frequency or time step on the full model. For very large structures, the total computation effort can be reduced by up to 75 percent. The mode combination for spectral analysis benefits from similar advancements. Instability predictions, such as the case of brake squeal, can be computed faster due to several enhancements to the damped eigensolver.

The introduction of ANSYS Variational Technology provides faster mode computation for cyclic symmetric structures, such as those found in many turbine applications. Using this technique can typically improve solution speed by a factor of three or four — the greater the number of sectors, the better the performance.



Instability analysis for brake squeal



Modal analysis of a cyclic–symmetric geometry
Courtesy PADT, Inc.

Rotating machinery applications profit from an extended set of capabilities for rotordynamics analysis. These include the extension of the gyroscopic effect to shell and 2-D elements and inclusion of rotating damping that takes hysteretic behavior into account.

Random vibration and spectral analysis users gain new tools as well as a greater flexibility in modeling structures, including support of spectrum analysis in the ANSYS Workbench platform. New tools include the United States Nuclear Regulatory Commission–compliant computation of missing masses and support of rigid modes, along with the ability to use residual vectors to account for higher energy modes. The global number of spectra applied simultaneously to the structure has been increased up to 50 as has the number of modes used in a combination — now up to 10,000.

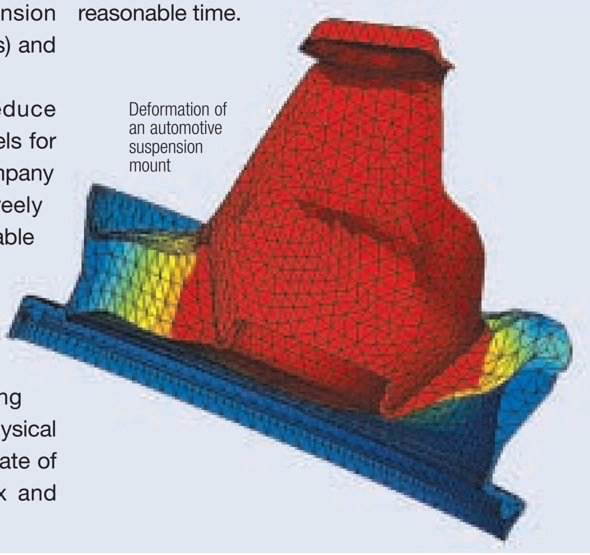
When analyzing design variations, comparing data from different simulation cases, or correlating simulation and test data, comparison between modal content of the models is required. The modal assurance criterion (MAC) in release 12.0 provides a convenient tool to compare the results of two modal analyses. Typical use cases for the criteria include tuning of misaligned turbine blades or validation of new component designs, each with respect to their vibration behavior.

## New Element Reduces Meshing Time

ZF Boge Elastmetall GmbH develops, manufactures and supplies vibration control components and parts for the automotive industry. These components include plastic parts, energy-absorbing elements for vehicle safety, and rubber–metal components such as chassis suspension mounts, control arm bushes (also known as bushings) and engine mounts.

The German company uses simulation to reduce development time and costs. When developing models for components with hyperelastic material properties, company engineers require an element type that can be freely meshed; can accommodate extreme deformation, stable contact and short computing time; and can provide reliable results.

By using the new SOLID285 four-noded tetrahedron element available in ANSYS 12.0, ZF Boge Elastmetall engineers considerably reduced meshing time. Close correlation between the simulation and physical measurement allowed them to determine the spring rate of strongly deformed structures without the complex and time-consuming meshing that was previously required when using hexahedral elements. Boge's work proved that by employing this new element, users can determine the stresses and strains for a durability calculation in a reasonable time.



Deformation of an automotive suspension mount

### ANSYS Workbench Integration

The integration of the structural applications within the ANSYS Workbench platform provides additional productivity to users, including:

- New meshing techniques to improve mesh quality
- Support of additional elements, such as gasket elements as well as quadratic shells and beams that include offset definitions
- Boundary condition definitions that provide a spatial dependency for loads
- Coupling conditions
- Remote points
- Ability to associate contact to the top or bottom of shell face

Post-processing capabilities have drastically improved with release 12.0. The user can now plot any structural simulation data stored in the results files. Mathematical operations involving elementary results can be introduced to create additional user-defined criteria. Complex mode shapes, plotting on linear paths, stress linearization (which depends upon path plotting), and the ability to display unaveraged results at element nodes complement the list of the features that increase productivity at ANSYS 12.0. ■

Pierre Thieffry and Siddharth Shah of ANSYS, Inc. contributed to this article.

## Multibody Dynamics

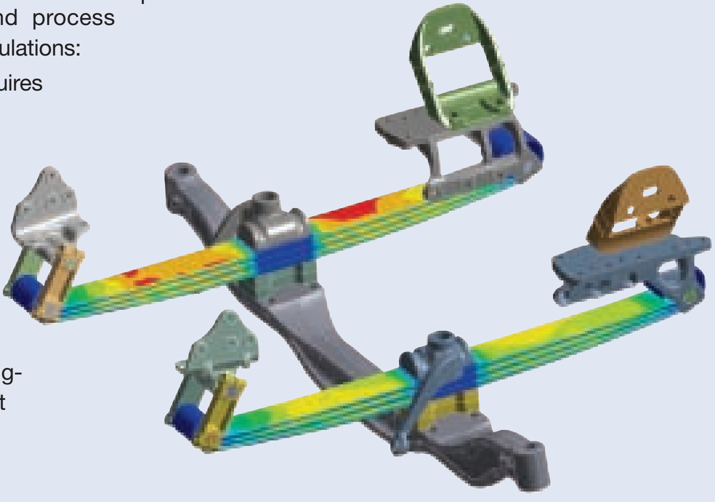
At release 12.0, a number of improvements in the general area of multibody dynamics enable the rapid design and analysis of complete mechanical systems undergoing large overall motion. ANSYS Rigid Dynamics software has a new Runge–Kutta 5 integrator, the preferred solution for long transient simulations. A new bushing joint, a “stops and locks” option for most other joint types, and the ability to specify preload for springs give new flexibility when simulating complex multiple-part assemblies and component interactions.

For complex assemblies, conducting an initial simulation with the ANSYS Rigid Dynamics product is the key to achieving robust flexible dynamics results. Creating overconstrained assemblies is an inconvenient reality; release 12.0 adds a redundancy analysis and repair tool to identify overconstrained assemblies, points out which joints or degrees of freedom are redundant, and allows selective unconstraining to create a properly constrained mechanism.

A number of improvements to data and process handling increase ease of use for multibody simulations:

- Enhanced load data fitting (no longer requires curve fitting)
- Ability to read in complex load input, such as simulated or measured multi-channel road surface or seismic data, and apply as load data to parts or joints
- Ability to use remote solution manager (RSM) to offload the solving effort to a server or other capable CPU (benefits long-duration and multi-channel input transient simulations)
- Ability to export forces and moments at any time within a transient simulation

For durability studies, exported loads can be used in a static structural analysis as an efficient first-pass failure analysis. Although it won’t provide the complete picture obtained from comprehensive flexible dynamics simulation, a static structural simulation is typically much less computationally expensive. Flexible dynamics simulations benefit at release 12.0 from robust component modal synthesis, or CMS. This method uses an internal substructuring approach and requires that the CMS parts of an assembly are constructed with linear materials. The procedure simplifies a problem by accounting only for a few degrees of freedom, which results in solution times that are often a fraction of those found using the standard full computation method. Time-to-solution reductions of several hundred percent are not uncommon.



Multibody dynamics capabilities were used to simulate this leaf spring suspension.