

ANSYS Mechanical—A Powerful Nonlinear Simulation Tool

Grama R. Bhashyam¹
Corporate Fellow, Development Manager
Mechanics & Simulation Support Group

ANSYS, Inc.
275 Technology Drive
Canonsburg, PA 15317

September 2002

¹ With grateful acknowledgment to Dr. Guoyu Lin, Dr. Jin Wang and Dr. Yongyi Zhu for their input.

Executive Summary

Numerical simulation plays an indispensable role in the manufacturing process, speeding product design time while improving quality and performance. Recently, analysts and designers have begun to use numerical simulation alone as an acceptable means of validation. In many disciplines, *virtual prototyping*—employing numerical simulation tools based on finite element methods—has replaced traditional build-and-break prototyping. Successful designs leading to better prosthetic implants, passenger safety in automotive crashes, packaging of modern electronic chips, and other advances are partly a result of accurate and detailed analysis.

Can one reliably simulate the collapse of a shell, interaction of multiple parts, behavior of a rubber seal, post-yield strength of metals, manufacturing process and so on using linear approximation? The answer is not really. With the trend toward ever-improving simulation accuracy, approximations of linear behavior have become less acceptable; even so, costs associated with a nonlinear analysis prohibited its wider use in the past. Today, rapid increases in computing power and concurrent advances in analysis methods have made it possible to perform nonlinear analysis and design more often while minimizing approximations. Analysts and designers now expect nonlinear analysis capabilities in general-purpose programs such as ANSYS Mechanical.

ANSYS, Inc. is a pioneer in the discipline of nonlinear analysis. The ANSYS Mechanical program's nonlinear capabilities have evolved according to emerging analysis needs, maturity of analysis methods and increased computing power. The program's nonlinear analysis technology has developed at such a rapid pace that some may be largely unaware of recent enhancements.

All of the following components are necessary for a reliable nonlinear analysis tool: (a) element technologies for consistent large-deformation treatment, (b) constitutive models for a variety of metals and nonmetals, (c) contact interaction and assembly analysis, (d) solution of large-scale problems (where multiple nonlinearities interact in a complex manner), and (e) infrastructure.

This paper presents a summary of the ANSYS Mechanical program's nonlinear technology. It is impossible given the scope of the paper to address every available analysis feature; rather, the paper highlights key features of interest to most analysts and designers and unique to the ANSYS Mechanical program. ANSYS, Inc. invites current and potential ANSYS Mechanical users to explore the program's capabilities further.

ANSYS Elements: Building Blocks of Simulation

The element library of Release 5.3 (circa 1994) was diverse and comprehensive in its capabilities. A clear need existed, however, for a new generation of elements to address the growing needs of multiplicity in material models and application complexities, and to bring about a higher level of consistency. ANSYS, Inc.'s Mechanics Group set out to develop a small set of elements (the 180 series) having these characteristics:

- Rich functionality
- Consistency with theoretical foundations employing the most advanced algorithms
- Architectural flexibility.

The application of conventional isoparametric fully integrated elements is limited. In linear or nonlinear analyses, serious locking may occur. As a general analysis tool, ANSYS Mechanical uses elements in wide range of applications. The following factors influence the selection of elements:

- Structural behavior (bulk or bending deformation)
- Material behavior (nearly incompressible to fully incompressible).

The indicated factors are not necessarily limited to nonlinear analysis; however, nonlinear analysis adds to the complexity. For example, an elasto-plastic material shows distinctly different patterns of behavior in its post-yield state. While it is feasible given today's state of the art to provide a most general element technology that performs accurately in virtually every circumstance, it will likely be the most expensive solution as well. Analysts and designers make such engineering decisions routinely according to their domain expertise, and their decisions often result in noticeable savings in computational costs.

With that in mind, ANSYS, Inc. views its element library as a toolkit of appropriate technologies. ANSYS, Inc. continues to develop and refine its element technologies to make ANSYS Mechanical an increasingly powerful tool for finite deformation analysis. Descriptions of existing element technologies follow.

Selective Reduced Integration Method

Also known as the Mean Dilation Method, B-Bar Method and Constant Volume Method, the Selective Reduced Integration Method was developed for some lower order solid elements to prevent volumetric locking in nearly incompressible cases. This formulation replaces volumetric strain at the Gauss integration points with an average volumetric strain of the elements.

Enhanced Strain Methods

A closer inspection of the ANSYS Mechanical program's element library (even at Release 5.2, circa 1993) provides evidence of ANSYS, Inc.'s early analysis leadership. For example, incompatible mode formulation was adapted in all first-order solid elements by default to avoid spurious stiffening in bending-dominated problems. The elements are said to have "extra shapes" formulation in ANSYS documentation.

A more general form of enhanced strain formulation was introduced at Release 6.0. The formulation modifies deformation gradient tensor and strains in lower order elements to prevent shear and volumetric locking (in nearly incompressible applications). The formulation is robust and is perhaps the best option when the deformation pattern may not be judged *a priori* as bulk or bending dominated.

Uniform Reduced Integration Method

The uniform reduced integration method prevents volumetric locking in nearly incompressible cases and is usually more efficient. In lower-order elements, this method can also overcome shear locking in bending-dominated problems. Hourglass control is incorporated, as necessary, to prevent the propagation of spurious modes. Such hourgassing is a non-issue in higher-order elements, provided that the mesh contains more than one element in each direction. This formulation also serves well as a compatible offering with our explicit offering, ANSYS LS-DYNA.

Displacement and Mixed u-P formulations

The ANSYS Mechanical program has both pure displacement and mixed u-P formulations. Pure displacement formulation has only displacements as primary unknowns and is more widely used because of its efficiency. In mixed u-P formulation, both displacements and hydrostatic pressure are taken as primary unknowns.

In the newly developed 180-series elements, the different element technologies can be used in combination (for example, B-bar with mixed u-P, enhanced strain with mixed u-P, etc.). Table 1 provides a summary of the available technologies.

ANSYS Mechanical has both penalty-based and Lagrangian multiplier-based mixed u-P formulations. Penalty-based formulation is meant only for nearly incompressible hyperelastic materials. On the other hand, the Lagrange multiplier-based formulation is available in the 180-series solid elements, and is meant for nearly incompressible elasto-plastic, nearly incompressible hyperelastic and fully incompressible hyperelastic materials.

The ANSYS Mechanical mixed u-P formulation (180-series) is user-friendly. It switches automatically among different volumetric constraints according to different material types. When used with enhanced strain methods, it excludes the enhanced terms for preventing volumetric locking to get higher efficiency because the terms are redundant in such a case. Provided that the material is fully incompressible hyperelastic, ANSYS Mechanical activates the mixed u-P formulation of 180-series solid elements even if the user does not specify it. The future promises even more automated, application-specific selection of appropriate element technology.

Table 1. Solid Element Technology Summary																	
18x Solid Elements	Numbers of Nodes	Dimensions	Element Shapes	Element Order	Interpolation	Stress States				Element Technologies			Formulation Options				
						Plane Stress	Plane Strain	Axisymmetric	Generalized	Plane Strain	3D	Selective	Reduced Integration/B-Bar	Uniform Reduced Integration	Enhanced Strain	Displacement	Mixed u/P (nearly)
PLANE182	4	2D	Quad.	Low/Linear	Bilinear	•	•	•	•	•	3D	•	•	•	•	•	•
PLANE183	8	2D	Quad.	High/Quad	Seren.	•	•	•	•	•		•	•	•	•	•	•
SOLID185	8	3D	Brick	Low/Linear	Trilinear					•		•	•	•	•	•	•
SOLID186	20	3D	Brick	High/Quad	Seren.					•		•	•	•	•	•	•
SOLID187	10	3D	Tet.	High/Quad	Tet					•		•	•	•	•	•	•

Structural Elements

The ANSYS Mechanical program supports a large library of beam and shell elements with wide applicability: composites, buckling and collapse analysis, dynamics analysis and nonlinear applications.

Most commercial FEA packages have a discrete-Kirchhoff Theory-based shell element employing an in-plane, constant-stress assumption. ANSYS Mechanical is unique, however, offering this capability with Allman rotational DOF, and enhancement of membrane behavior when used as a quadrilateral. The result is significantly higher stress-prediction accuracy. The element supports small-strain, large-rotation analysis with linear material behavior.

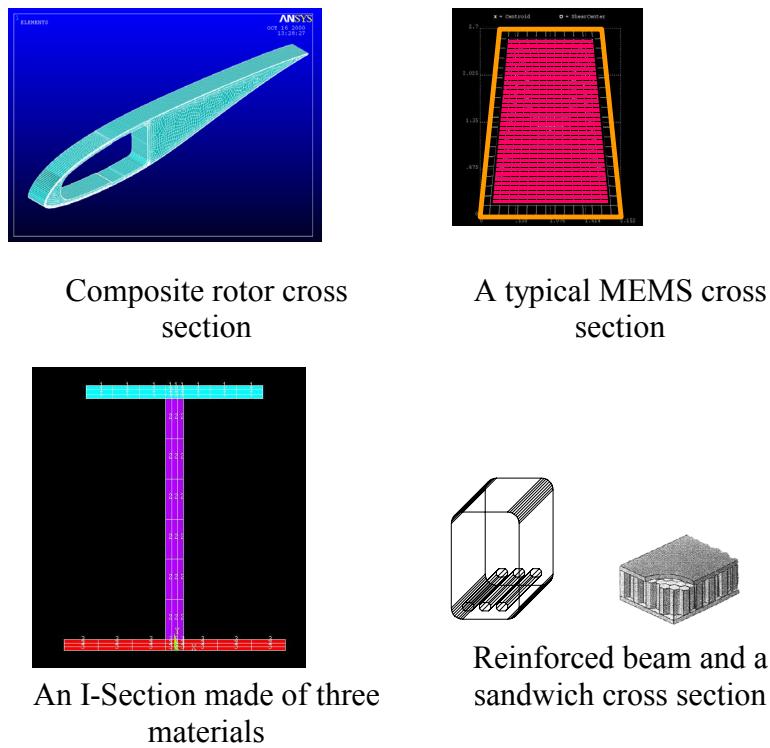
Some recent enhancements in the 180-series elements for structural applications advance the state of the art. One can now expect both robust performance and ease of use.

The beam elements (BEAM188 and BEAM189) represent a significant move towards true “reduction in dimensionality” of the problem (as opposed to simple beams). Whether one employs a simple circular cross section or a complex arbitrary cross section, a finite element cross-section analyzer calculates inertias, shear centers, shear flow, warping rigidity, torsion constant, and shear correction factors. ANSYS Mechanical 2-D modeling can sketch the arbitrary profiles. The section solver relies on the industrial

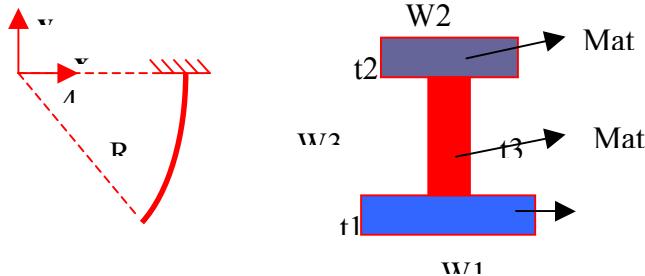
strength sparse solver, and hence the ability for solving large user-specified cross sections.

It is possible to specify the mesh quality for the section solution. The cross sections can also be comprised of a number of orthotropic materials, allowing for analysis of sandwich and built-up cross sections. ANSYS, Inc. is aware of an extreme application where a user modeled an entire rotor cross section using thousands of cells with tens of materials. The beam elements complement the finite deformation shell elements very well. The formulation employed allows for conventional unrestrained warping, and restrained warping analysis as well. The generality of formulation is such that the user is spared from details (such as selecting element types based on open or closed cross sections) and limitations found elsewhere (for example, multiple cells, a circular tube with fins). The robust solution kernel is complemented by the easy-to-use Beam Section Tool and full 3-D results visualization. All elastoplastic, hypo-viscoelastic material models may be used. It is an ideal tool for aerospace, MEMS, ship building, civil applications, as illustrated:

Flexibility in cross section modeling



It is important to understand that no significant performance compromise exists for linear analysis despite the overwhelming generality. This is valid for all 180-series elements. Figure 1 shows the typical accuracy that one can achieve while enjoying the benefits of a reduced dimensionality model.



	BEAM189 (NDOF=96)		BEAM189 (NDOF=192)		SOLID186 (NDOF=18900)
Max. displacement	Value	% diff.	Value	% diff.	Reference value
Ux	19.664	0.2	19.666	0.2	19.625
Uy	24.819	1.9	24.822	1.9	25.310
Uz	54.486	0.5	54.490	0.5	54.769
CPU Time	82.610		115.460		4587.850

Figure 1. Nonlinear analysis of a curved beam with multiple materials in cross section: a comparison of solid elements

In an upcoming release, beam section capability will allow a geometrically exact representation of tapered beams (rather than an approximate variation of gross section properties).

Similarly, the 180-series shell element SHELL181 offers state-of-the-art element technology, be it linear or nonlinear analysis with strong emphasis on ease of use. The four-node shell element is based on Bathe-Dvorkin assumed transverse shear treatment, coupled with uniform reduced integration or full integration with enhancement of membrane behavior using incompatible modes. Several elasto-plastic, hyperelastic, viscoelastic material models can be employed. The element supports laminated composite structural analysis, with recovery of interlaminar shear stresses. With this and other shell elements, ANSYS also empowers users with a detailed submodel analysis using solid elements for delamination and failure studies.

Figure 2 shows a model of a circular plate having thickness which varies with a known formula; one can create such a model interactively via the ANSYS Mechanical Function Builder. The shell element definition is therefore completely independent of meshing and enhances accuracy by directly sampling thicknesses at element Gauss points.

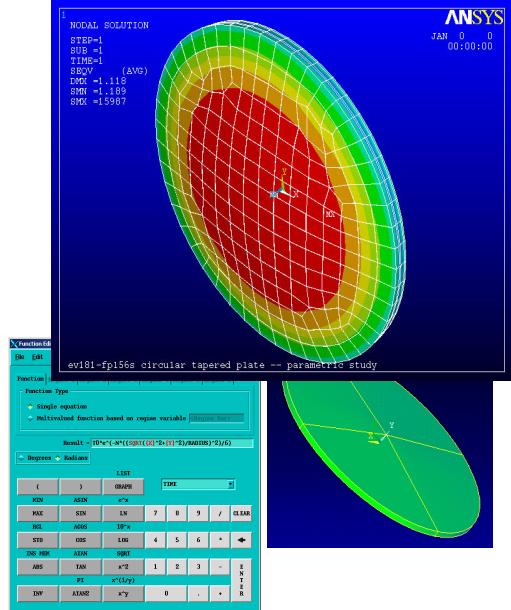


Figure 2. Circular tapered plate using Function Builder

SHELL181 applicability encompasses frequency studies, finite strain/finite rotation, nonlinear collapse, and springback analysis following an explicit forming operation. The ANSYS Mechanical contact elements work with SHELL181 to allow straightforward inclusion of current shell thickness in a contact analysis.

Figure 3 shows a beverage can in nonlinear collapse study, and Figure 4 shows a stamped part which was analyzed for springback effects using the shell element.

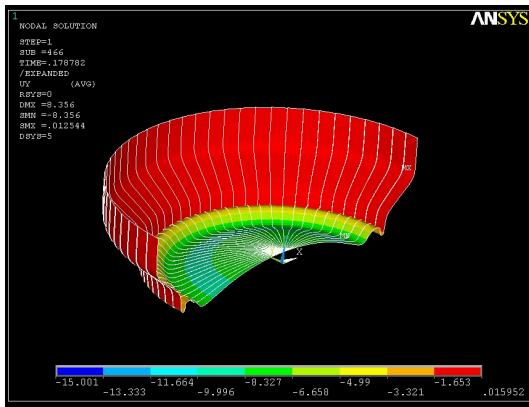


Figure 3. Nonlinear collapse study of a beverage can

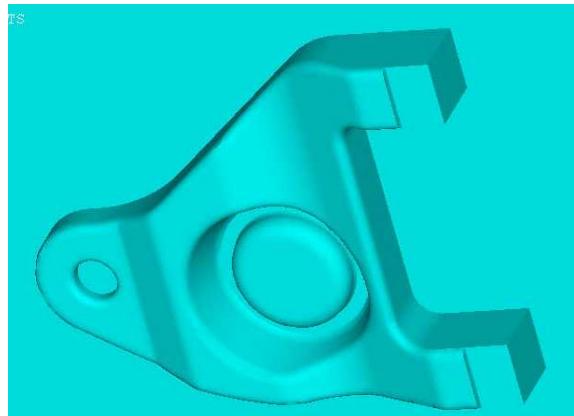


Figure 4. Stamping (ANSYS LS-DYNA) and springback analysis (ANSYS Mechanical)

Common Features

ANSYS Mechanical data input can be parametric, allowing for parametric study and optimization of structures. In the near future, the ANSYS Mechanical element library will incorporate the power of CADOE variational analysis. The resulting combination

offers promising opportunities including “what-if” studies, design sensitivity analysis, and discrete and continuous optimization.

ANSYS Mechanical was foremost in offering submodeling, layered solid elements advancing the state of art in composites analysis. In addition, rigid spars, rigid beam, shell-to-solid interfaces, slider constraints will be available in the near future. Interface elements simulate gasket joints or interfaces in structural assembly. Surface elements apply various loading. Superelement and infinite elements are also available in the ANSYS Mechanical element library.

Consistent and complete derivation of tangent stiffnesses is crucial for acceptable convergence rates. For example, the effect of pressure loads to the stiffness matrix is included by default in the 180-series elements.

The stress states supported in solid elements include: 3-D, plane stress, plane strain, generalized plane strain, axisymmetric and axisymmetric with asymmetric loading. The 180-series elements are applicable to all material models.

The ANSYS Mechanical program automatically selects appropriate shape functions and integration rules when elements are degenerated. If necessary, it may update the element technology specification. For example, when a quadrilateral element degenerates into a triangle or a hexahedron element into a prism, pyramid or tetrahedral forms, ANSYS Mechanical employs appropriate shape functions for displacement interpolation and hydrostatic pressures instead of the generic shape functions for the native element. The capability of the program to compensate for element degeneration makes element formation less sensitive to mesh distortion and more robust in geometric nonlinear analysis. The mid-side nodes at higher element can be omitted so that they can be used as the transition elements. Degenerated shapes make modeling an irregular area or volume easier².

The 180-series family of elements offers superior performance and functionality. They have provided an architecture for future advancements in material modeling, including shape memory alloys, bio-medical, microelectronics assemblies, and electronics packaging industrial needs. ANSYS, Inc. intends to support remeshing/rezoning, fracture mechanics, variational analysis, and coupled fields in future ANSYS Mechanical releases. ANSYS, Inc. development is also committed to making further infrastructural improvements in the 180-series elements to accommodate distributed processing needs. ANSYS, Inc. believes that such developments can simplify and even automate element selection in future releases.

² Degenerated element support for the 180-series of elements is a prerelease feature in Release 7.0.

Material Nonlinearity in ANSYS Mechanical

For engineering design and application, it is essential to understand and accurately characterize material behavior. It is a challenging, complex science. Lemaitre and Chaboche³ express the complexity in a dramatic manner, as follows:

“A given piece of steel at room temperature can be considered to be:

Linear elastic for structural analysis,
Viscoelastic for problems of vibration damping,
Rigid,
Perfectly plastic for calculation of the limit loads,
Hardening elastoplastic for an accurate calculation of the permanent deformation,
Elastoviscoplastic for problems of stress relaxation,
Damageable by ductility for calculation of the forming limits,
Damageable by fatigue for calculation of the life-time.”

Validity of the different models can be judged only on phenomenon of interest for a given application. (See Table 2 for plasticity models.)

Table 2. Validity of Plasticity Models*

Models	Monotonic Hardening	Bauschinger Effect	Cyclic Hardening or Softening	Ratchetting Effect	Memory Effect
Prandtl-Reuss	•		•		
Linear Kinematic	•	•			
Mroz	•	•	•		
Nonlinear Kinematic	•	•		•	
Kinematic+Isotropic	•	•	•	•	
Kinematic+Isotropic+memory	•	•	•	•	•

*Lemaitre and Chaboche, 1990

The scope and intent of this paper make it necessary to omit the details of material models. ANSYS, Inc. encourages the reader to research standard references^{4,5} and the *ANSYS Theory Guide*.

³ Lemaitre and Chaboche, Mechanics of solid materials, Cambridge University Press, 1990

⁴ Simo, J.C. and Hughes, T.J.R., Computational inelasticity, Springer-Verlag, 1997

⁵ Ogden, R.W., Non-linear elastic deformations, Dover Publications, Inc., 1984.

ANSYS provides constitutive models for metals, rubber, foam and concrete. The response may be nonlinear, elastic, elastic-plastic, elasto-viscoplastic and viscoelastic.

Plasticity and Creep

The suite of plasticity models is comprehensive and covers anisotropic behavior. All elastic-plastic models are in rate form and employ fully implicit integration algorithm for unconditional stability with respect to strain increments. ANSYS, Inc. has also made every effort to obtain consistent material Jacobian contributions in order to obtain efficient, acceptable convergence rates in a nonlinear analysis. Table 3 provides a pictorial view of ANSYS elastic-plastic models (both rate-dependent and rate-independent forms), and non-metallic inelastic models.

Table 3. Plasticity Models in ANSYS

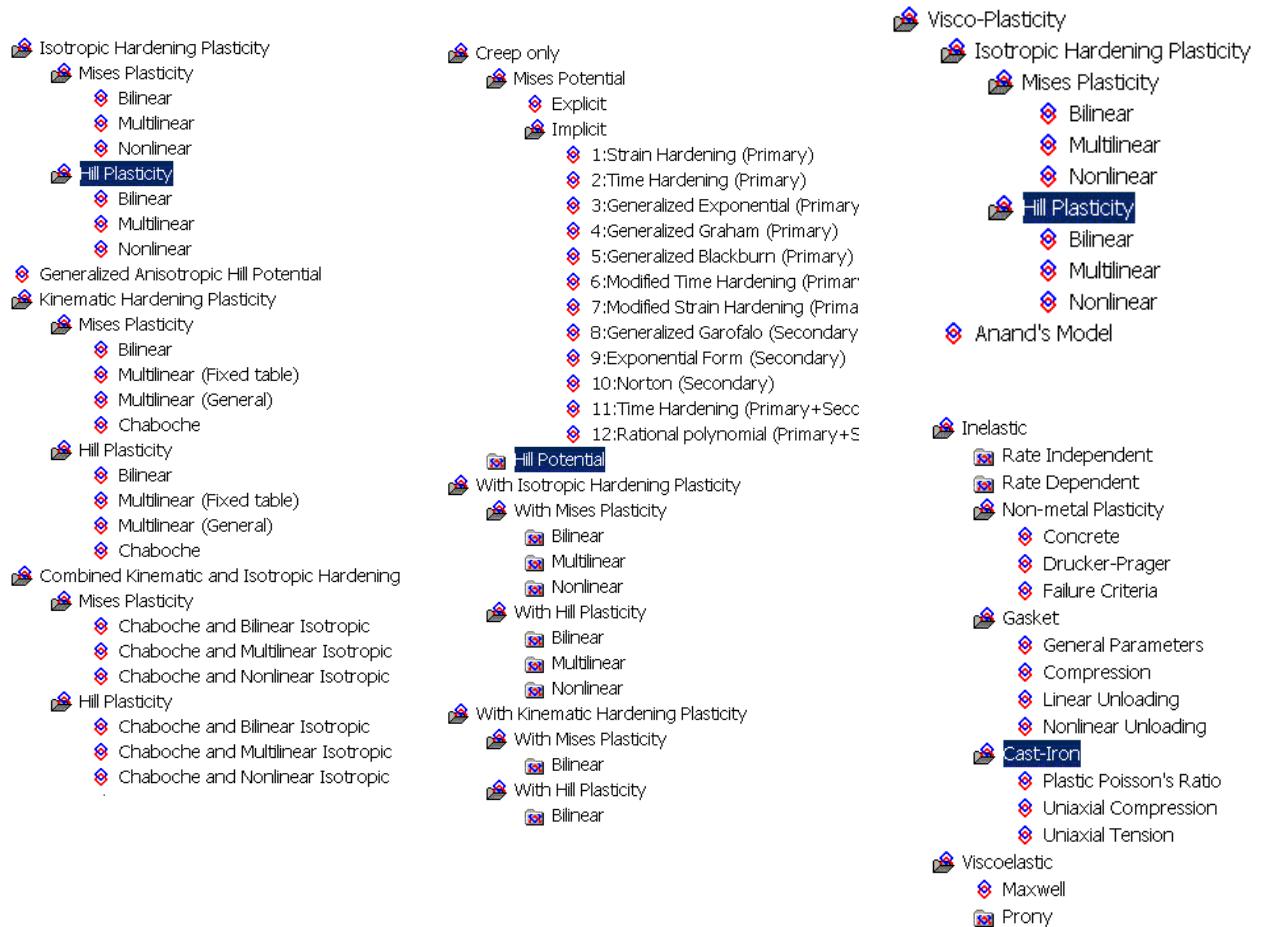


Table 3, a direct screen capture of ANSYS Mechanical Material Model Definition user interface, provides an idea of the breadth of material models supported. It conveys ANSYS, Inc.'s emphasis towards a logical, consistent tree structure that guides users along (specifically with valid combinations of material options). ANSYS, Inc.'s

development efforts for materials have closely followed customer needs. One can specify nearly every material parameter as temperature-dependent. To meet ever expanding demands for material modeling, the ANSYS Mechanical program also supports a flexible user interface to its constitutive library.

ANSYS offers several unique options; a multilinear kinematic hardening model that is a sublayer model allowing for input of experimental data directly, and the Chachoché model that offers ability of superimposing several nonlinear kinematic hardening options to accommodate the complex of cyclic behavior of materials (such as ratcheting, shakedown, cyclic hardening and hardening).

Cast Iron Plasticity

The Cast Iron (CAST, UNIAXIAL) option assumes a modified Mises yield surface, consisting of the Mises cylinder in compression and a Rankine cube in tension. It has different yield strengths, flows, and hardenings in tension and compression. Elastic behavior is isotropic, and the same in tension and compression. Applying cast iron plasticity to model gray cast iron behavior assumes the following:

- Elastic behavior (MP) is isotropic and is therefore the same in tension and compression.
- The flow potential and evolution of the yield surfaces are different for tension and compression.

Currently, the isotropic hardening rule applies to the cast iron model.

Viscoelasticity

Viscoelasticity is a nonlinear material behavior having both an elastic (recoverable) part of the deformation as well as a viscous (non-recoverable) part. Viscoelasticity model implemented in ANSYS is a generalized integration form of Maxwell model, in which the relaxation function is represented by a Prony series. The model is more comprehensive and contains, the Maxwell, Kevin, and standard linear solid as special cases. ANSYS supports both hypo-viscoelastic and large-strain hyper-viscoelasticity.

The large-strain viscoelasticity implemented is based on the formulation proposed by Simo. The viscoelastic behavior is specified separately by the underlying hyperelasticity and relaxation behavior. All ANSYS hyperelasticity material models can be used with the viscoelastic option (PRONY).

Viscoplasticity and creep

ANSYS program has several options for modeling rate-dependent behavior of materials, including creep. Creep options include a variety of creep laws that are suitable for conventional creep analyses. Rate-dependent plasticity option is an overstress model

and is recommended for analyzing impact loading problems. Anand's⁶ model, which was originally developed for high-temperature metal forming processes such as rolling and deep drawing is also made available. Anand's model uses an internal scalar variable called the deformation resistance to represent the isotropic resistance to the inelastic flow of the material, and is thus able to model both hardening and softening behavior of materials. This constitutive model has been widely used for other applications, such as analyses of solder joints in electronics packaging^{7,8}.

Hyperelasticity

Elastomers have a variety of applications. A common application is the use of an O-ring as a seal to prevent fluid transfer (liquid or gas) between solid regions. Modeling involves the hyperelastic O-ring and the contact surfaces. The rubber material relies on a compressive force which seals the region between surfaces. The application requires a robust nonlinear analysis because of these factors:

- A large (several hundred percent) strain level
- The stress-strain response of the material is highly nonlinear
- Nearly or fully incompressible behavior
- Temperature dependency
- Complex interaction of elastomeric material with adjoining regions of metal.

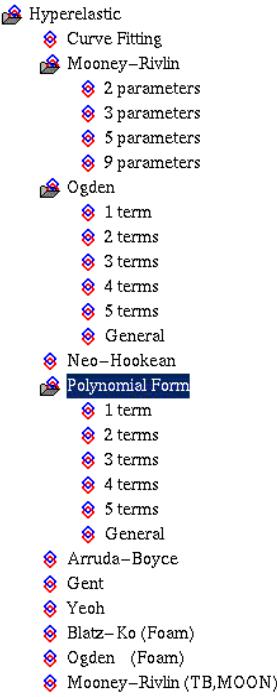
Nonlinear FEA allows approximate numerical solutions to a boundary value problem by solving simultaneous sets of equations with displacements, pressures, and rotations as unknowns. The experimental characterization of the material assumes a critical role. One must judiciously select a particular constitutive model among available options. Table 4 provides a list of options available in the ANSYS Mechanical program. Solver support, element technologies and global solution heuristics have been fine-tuned for efficient and effective hyperelastic applications.

⁶ Anand, L., "Constitutive Equations for Hot-Working of Metals", *International Journal of Plasticity*, Vol. 1, pp. 213-231 (1985).

⁷ Darveaux, R., "Solder Joint Fatigue Life Model," *Proceedings of TMS Annual Meeting*, pp. 213-218 (1997).

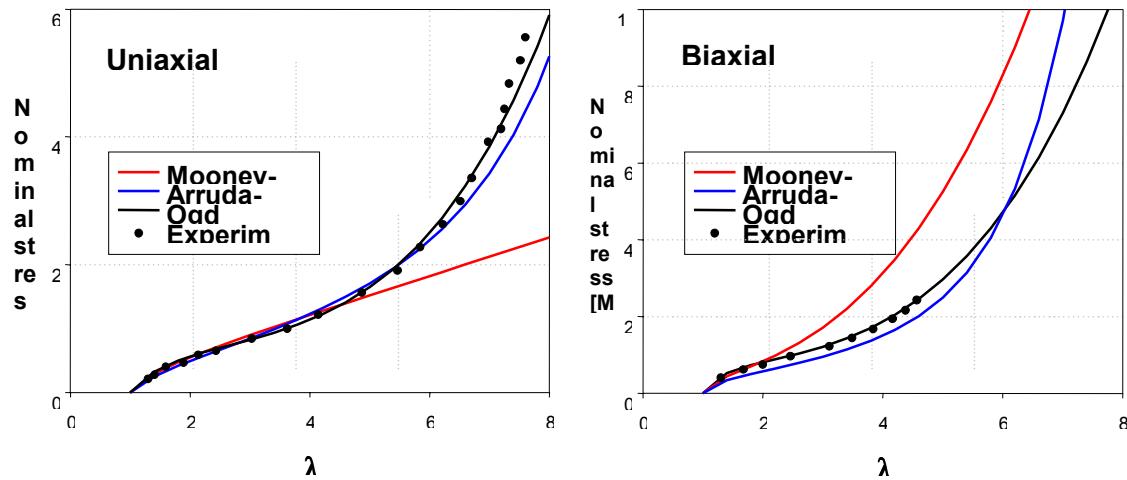
⁸ Darveaux, R., "Effect of Simulation Methodology on Solder Joint Crack Growth Correlations," *Proceedings of 50th Electronic Components & Technology Conference*, pp. 1048-1058 (2000).

Table 4. Hyperelastic Models in ANSYS



Validity and suitability of the hyperelastic models depend upon application specifics and the availability of experimental data. Figure 5 provides a glimpse at comparison of Mooney-Rivlin, Arruda-Boyce and Ogden models with experimental data for a particular test. Based on such studies, suggestions for selecting a hyperelastic model appear in Table 5.

Figure 5 Comparison of hyperelastic models



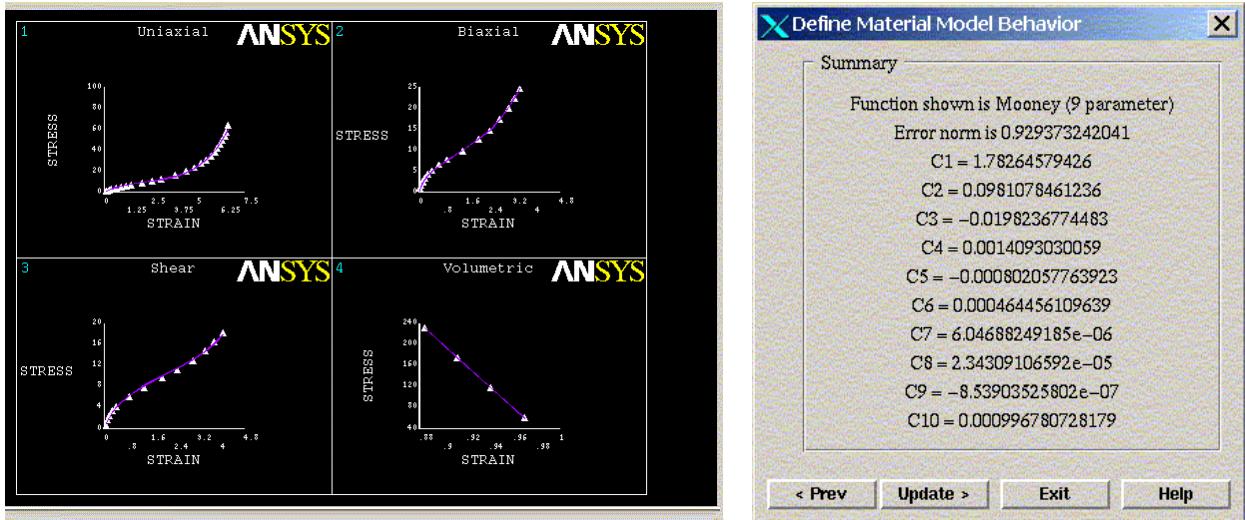
Experimental data are from Treloar, L.R.G., Stress strain data for vulcanized rubber under various types of deformation, Transactions of the Faraday Society, vol. 40, pp.59-70 (1944)

Table 5. Applicability of Hyperelastic Models

<i>Material Model</i>	<i>Applicable Strain Range</i>
<i>Neo-Hookean</i>	<30%
<i>Mooney Rivlin</i>	30-200%
<i>Polynomial</i>	Function of order N ; feasible to model up to 300%
<i>Arruda Boyce</i>	< 300%
<i>Ogden</i>	< 700%

At Release 7.0, the ANSYS Mechanical program allows one to input experimental data and obtain hyperelastic coefficients via linear and nonlinear regression analysis. The new capability is valid for all supported hyperelastic models, and future releases may extend support to viscoelasticity and creep analysis. When the experimental data is available in a text file, one can attempt the curve fit for several hyperelastic models. ANSYS Mechanical provides an error norm and compares experimental data to calculated coefficients graphically. Figure 6 illustrates the new feature.

Figure 6. Experimental Input and Curve Fit



Gasket Joint Modeling

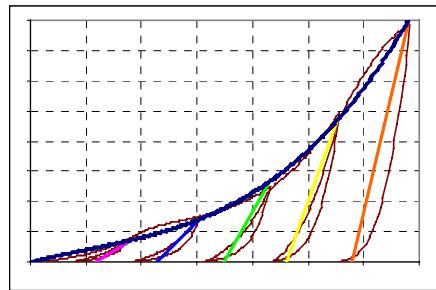
Gaskets are sealing components between structural components. They are usually thin and made of a variety materials, such as steel, rubber and composites.

The primary deformation of a gasket is usually confined to the normal direction. The stiffness contribution from membrane (in-plane) and transverse shear are much smaller (and generally negligible). The gasket material is typically under compression, exhibiting high nonlinearity. The material exhibits complicated unloading behavior when compression is released.

The GASKET table option allows one to directly input the experimentally measured complex pressure-closure curve (compression curve) for the material model, in addition to several unloading pressure-disclosure curves. When no unloading curves are defined, the material behavior follows the compression curve while it is unloaded. Other features have also been implemented with the GASKET material option for the advanced gasket joints analysis (for example, allowing initial gap, tension stress cap and stable stiffness).

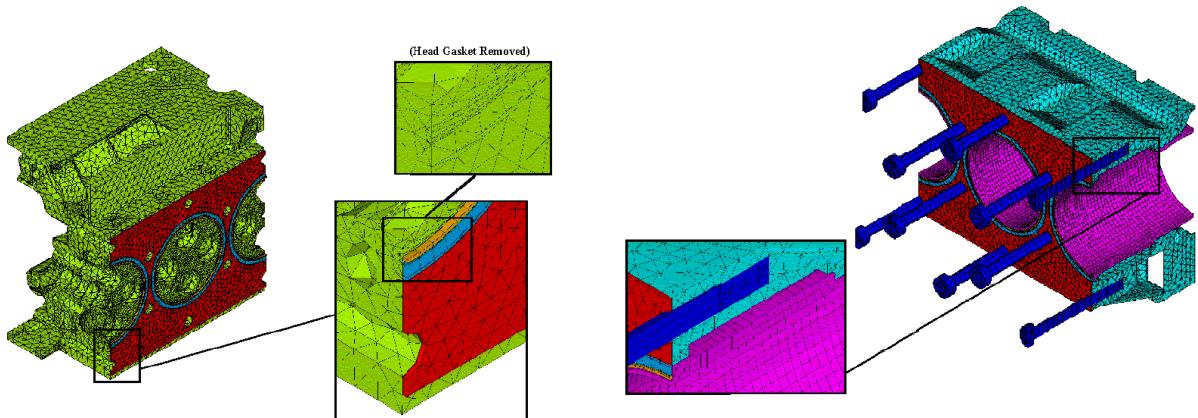
Figure 7 shows the experimental pressure vs. disclosure (relative displacement of top and bottom gasket surfaces) data for the graphite composite gasket material. The sample was unloaded and reloaded five times along the loading path and then unloaded at the end of the test to determine the material's unloading stiffness.

Figure 7. Gasket Material Behavior



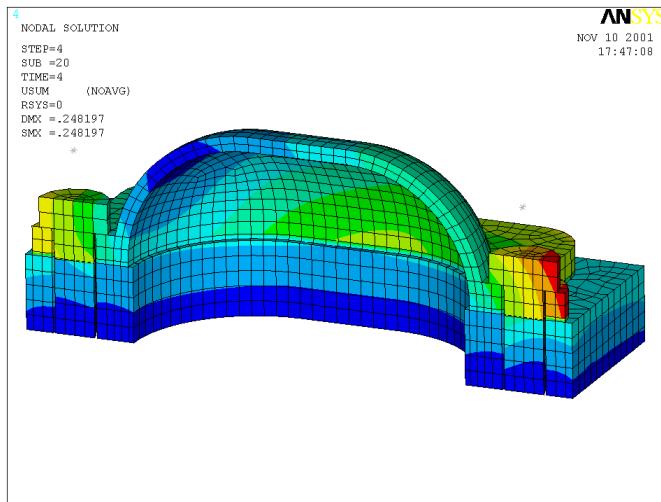
Figures 8 depicts a typical gasket application. This picture is a reproduction from a paper presented by an ANSYS Mechanical user.⁹ Figure 9 shows a manifold assembly. In such applications, many challenges can exist, such as gasket and model size, the presence of bolts, contact between parts and complex loading history. Figure 9 shows the use of pre-tension section elements (bolted joints).

Figure 8. Engine assembly and gasket Use⁵



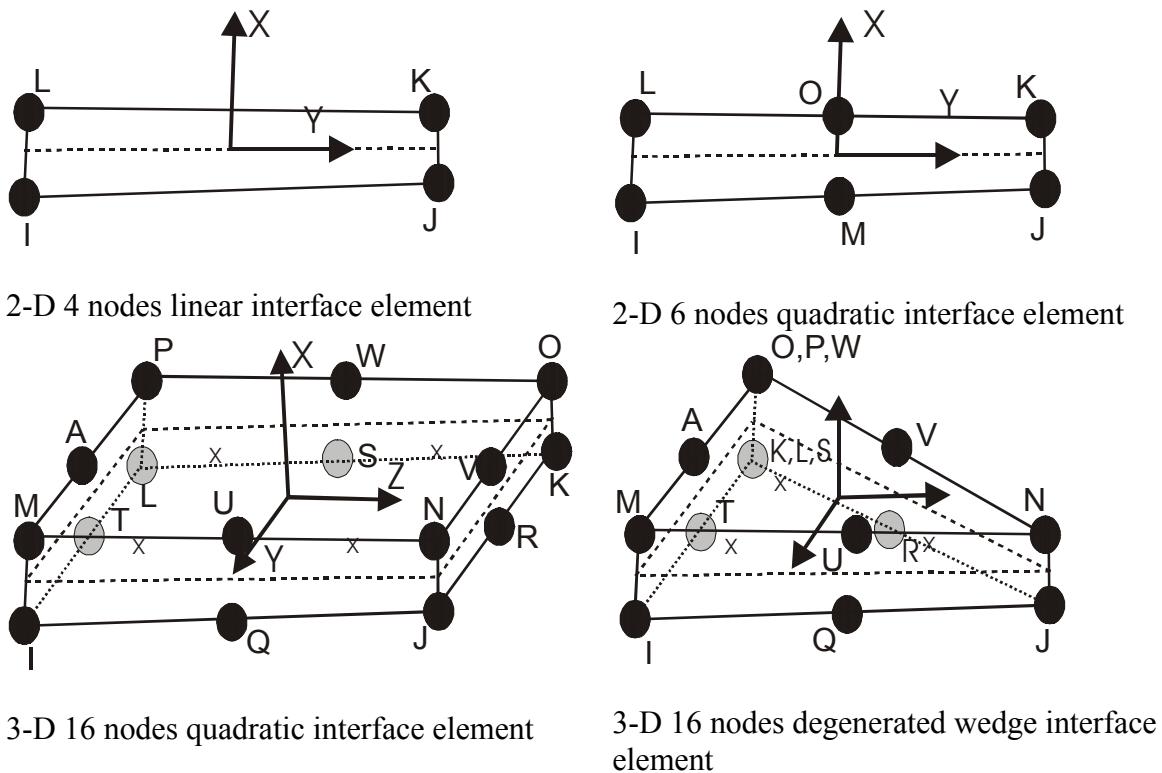
⁹ Jonathan Raub, Modeling Diesel Engine Cylinder Head Gaskets using the Gasket Material Option of the SOLID185 Element, ANSYS Conf. 2002, Pittsburgh, PA.

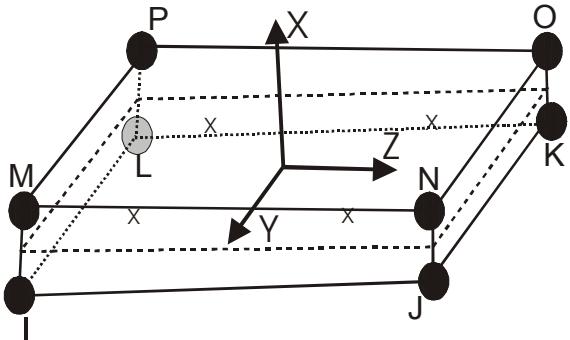
Figure 9. Manifold Assembly



Jonathon⁵ describes the use of a material option, made available by ANSYS Inc., using a general 3-D element. The ANSYS Mechanics program has since offered a series of interface elements which can model the gasket. (At present, the membrane and transverse shear are ignored for the gasket simulation.) ANSYS offers many types of interface elements which include two-dimensional and three-dimensional stress states, and linear and quadratic orders (as shown in Figure 10).

Figure 10. Gasket elements in ANSYS Mechanical





3-D 8 nodes linear interface element

In problems of this type, an iterative solver such as the AMG (Algebraic Multi Grid) equation is a particular strength of the ANSYS Mechanical program. Moreover, the calculation can take advantage of parallel processing in a shared memory environment with multiple CPUs. ANSYS, Inc. has adapted its iterative solvers for a subclass of nonlinear problems.

In addition to the material models supported in the ANSYS Mechanical program, many ANSYS, Inc. consultants and distributors offer constitutive models (for powder compaction, geomechanics, and other applications) using a host of user-programmable features. Also, ANSYS, Inc. has collaborative relationships with material specialists who offer experimental characterization and input parameters in the proper format.

Constitutive modeling analysis needs are constantly expanding. ANSYS, Inc. has taken a number of initiatives to address the needs emerging in the microelectronics, bio-engineering, composite, polymer, and manufacturing sectors. As is the case with element technology, the ANSYS Mechanical program provides a comprehensive toolkit in material models.

Contact Capabilities of ANSYS Mechanical

Applications such as seals, metal forming, drop tests, turbine blade with base shroud, elastomeric bellows of a automotive joint, gears, assembly of multiple parts, and numerous others have one common characteristic: *contact*.

The ability to model interaction between two solid regions (often accounting for friction, thermal, electric or other forms of exchange) is critical for a general purpose analysis tool such as ANSYS Mechanical. Indeed, the success of a nonlinear analysis tool is frequently judged by its contact analysis capabilities.

Robustness and performance are important, but the ability to define the model easily and manage the attributes of contact pairs is equally important, as are effective troubleshooting tools.

ANSYS offered contact analysis features as early as Release 2.0, and then evolved the state of the art according to advancing analysis needs. Table 6 provides a summary of that evolution.

Element	12/52	178	48/49	175	171-174
Type	Node-Node	Node-Node	Node-Surface	Node-Surface	Surf-Surf
Sliding	Small	Small	Large	Large	Large
High order					Yes
Augmented Lagrange		Yes	Yes	Yes	Yes
Pure Lagrange		Yes		Yes	Yes
Contact stiffness	User defined	Semi-automatic	User defined	Semi-automatic	Semi-automatic
Thermal Electric			Yes	Yes Yes	Yes Yes
Mesh tool	EINTF	EINTF	GCGEN	ESURF	ESURF

Table 6. Evolution of ANSYS Contact

The evolution of the ANSYS Mechanical program over the last two decades is approximately reflected by the element numbers.

Elements CONTAC12 and CONTAC52 simulated node-to-node contact in two and three dimensions, respectively. Initially, the elements were based upon a penalty function approach and elastic Coulomb friction model. Simplest among the class of elements, they were substantially rewritten when ANSYS introduced nonlinear capabilities at Release 5.0.

ANSYS, Inc. later developed the CONTAC48 and CONTAC49 node-to-surface contact elements for general contact problems. The underlying technology is penalty based, but with Lagrange augmentation to enforce compatibility. The elements allow for large sliding, either frictionless or with friction. In addition to solid mechanics, the elements support thermal analysis as well. The elements allowed one to solve highly nonlinear contact problems (for example, metal forming and rolling contact); however, the node-to-node and node-to-surface contact elements were perceived as difficult to use because of the penalty stiffness. When the elements were used for large surfaces, the visualization also suffered due to the numerous line elements generated; a fundamental drawback of this approach was evident in models of curved surfaces in conjunction with higher order solid elements.

This paper mentions contact elements 12, 52, 26, 48 and 49 to provide a historical perspective. Today, the ANSYS Mechanical program incorporates better, more advanced contact element technologies.

Surface-to-Surface Contact

At Release 5.4 (circa 1997), ANSYS Mechanical introduced a radical improvement in contact analysis capabilities. At first, a series of surface-to-surface contact elements (169-174) allowed one to model rigid-to-flexible surface interaction. The augmented Lagrange method with penalty is the basis for the elements, but with a significant difference. The penalty stiffness, selected by default, is a function of many factors (including the size of adjoining elements and the properties of underlying materials). It is not necessary to provide an absolute value of stiffness, but one may override default values via a scaling non-dimensional factor. This option is necessary in bending-dominated situations. A recent update of the algorithm modifies the penalty stiffness based upon the stresses in underlying elements.

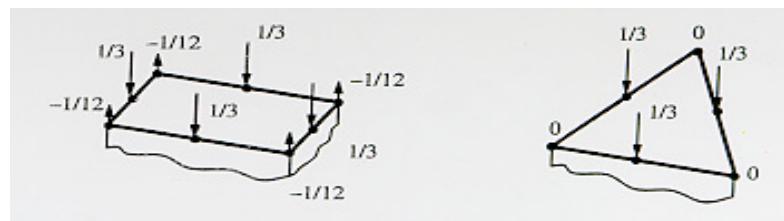
The 169-174 contact elements are easier to visualize and interpret. The output is in the form of stress rather than force. The numerical algorithms employed are efficient, even in large problems. The friction is treated in a rigorous manner as a normal constitutive law, helping convergence without the use of heuristics such as adaptive descent methods.

Furthermore, the 171-174 contact elements were extended to general flexible-to-flexible surfaces. Some of the advantages of the unique Gauss-point-based contact algorithms began to manifest themselves. The contact technology works flawlessly with higher order elements such as a 20-node brick, 10-node tetrahedron, and 8-node surface.

The topologies mentioned produce equivalent nodal contributions inconsistent with a constant pressure (as shown in Figure 11). To account for this, nodal-based contact algorithms must be complemented by alternative element technologies (for example, composite tetrahedrons or the Lagrangian family of bricks). The net effects are often less accuracy and/or increased costs.

Whereas users of other FEA software products are encouraged to use first-order elements in contact problems, we believe that ANSYS Mechanical users should take advantage of the higher accuracy-to-cost ratio offered by second-order elements and their unique Gauss-point-based surface-to-surface contact technology.

Figure 11. Equivalent nodal forces in higher order elements

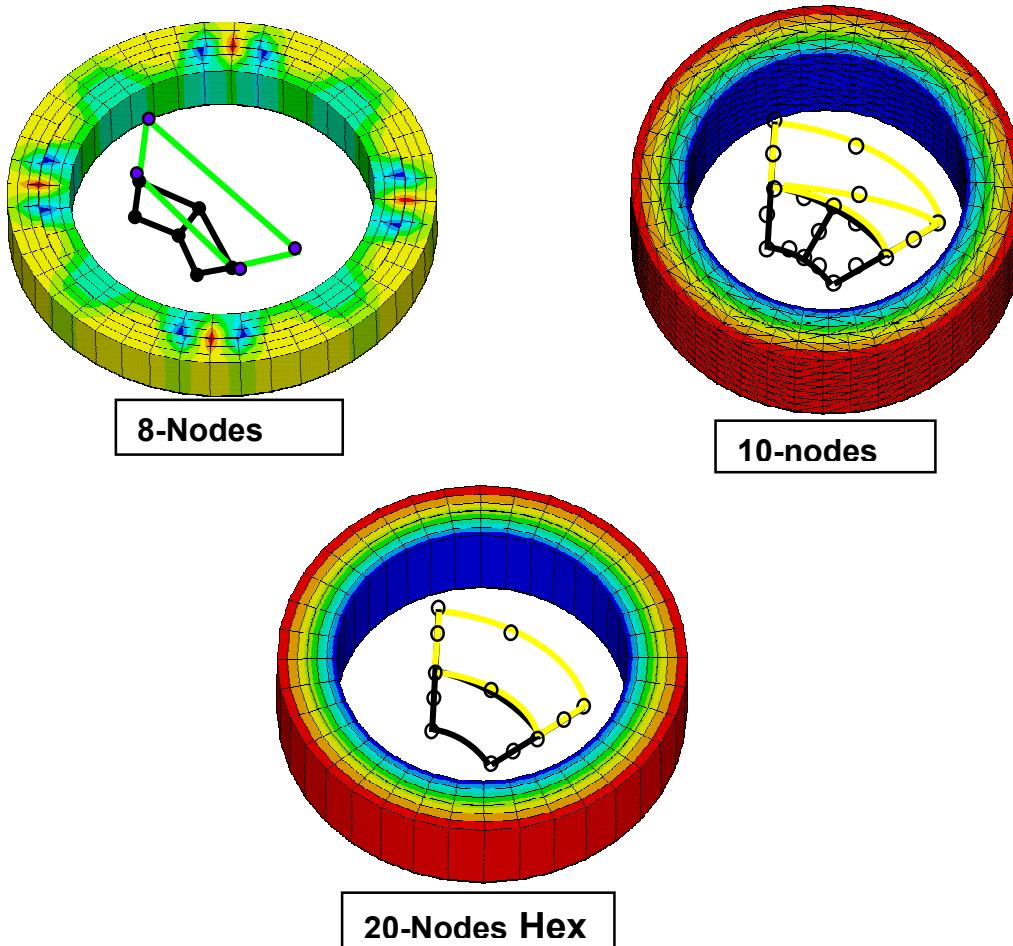


Better Geometry Representation

When using second-order elements, the ANSYS Mechanical program allows for quadratic representation in both “contact” and “target” surfaces (also referred to as the “slave” and “master” surfaces) rather than contact surface approximation by facets (a common practice in other software products). This difference alone accounts for the higher degree of accuracy that ANSYS Mechanical can achieve in many applications.

Figure 12 shows the stress contours of a circular prismatic solid. The anticipated results is a state of constant stress around the circumference. A solution based on facet approximation surfaces would yield grossly inadequate results. (Note the results for the eight-node element exhibiting spurious concentration spots.)

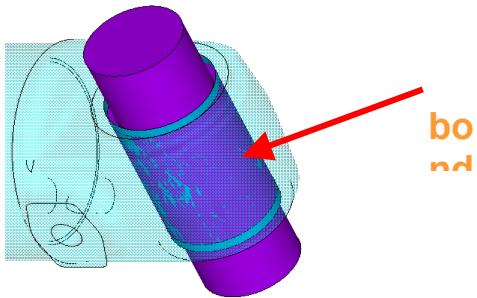
Figure 12. Need for second-order representation in modeling contact between curved surfaces



The Gauss-point-based contact algorithm avoids the ambiguity associated with direction of contact at sharp intersections. As a result, relatively coarse modeling of target surfaces is adequate. The algorithm also circumvents the common difficulty with nodal contact algorithms of “slipping beyond the edge.”

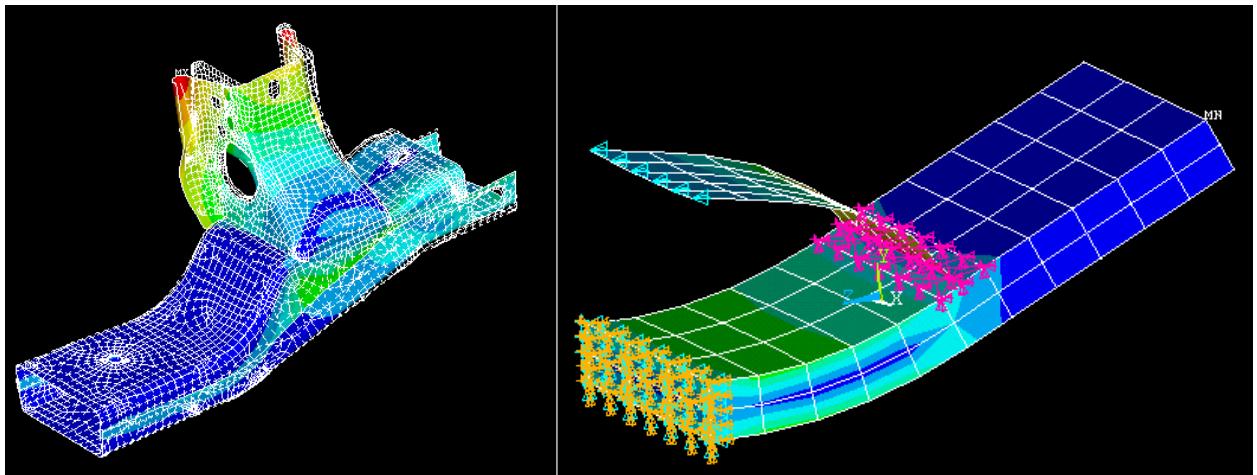
The ANSYS Mechanical program’s contact elements provide a rich set of initial adjustment and interaction models. Besides the standard unilateral contact, it offers the options of bonded, no-separation, and rough sliding contact. The bonded contact option is especially useful, as the application shown in Figure 13 illustrates.

Figure 13. Assembly contact



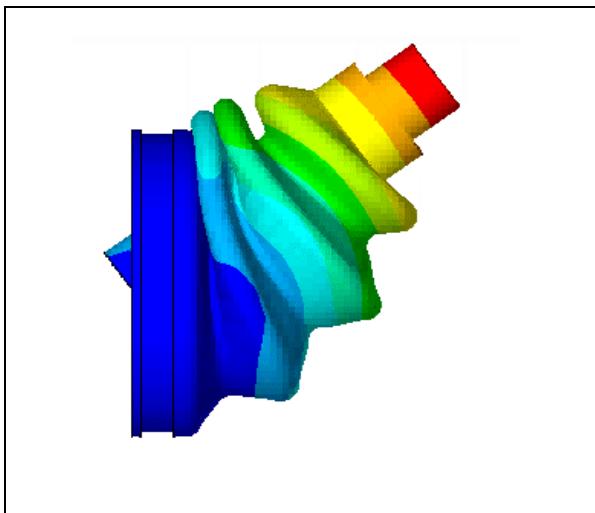
In an upcoming release, contact definition will allow one to define shell-to-shell and shell-to-solid assemblies (as shown in Figure 14). ANSYS Mechanical will employ standard multi-point constraints (MPCs) to enforce compatibilities.

Figure 14. Shell-to-shell and solid-to-shell assemblies



The ANSYS Mechanical program's surface-to-surface contact technology is especially effective in modeling self-contact (that is, a surface coming into contact with itself). Figure 15 illustrates the use of ANSYS hyperelastic elements and contact for a rubber boot analysis, with significant self-contact status. The example also highlights ANSYS Mechanical's ability to simulate complex interaction among different types of nonlinearities.

Figure 15. Rubber boot exhibiting self-contact



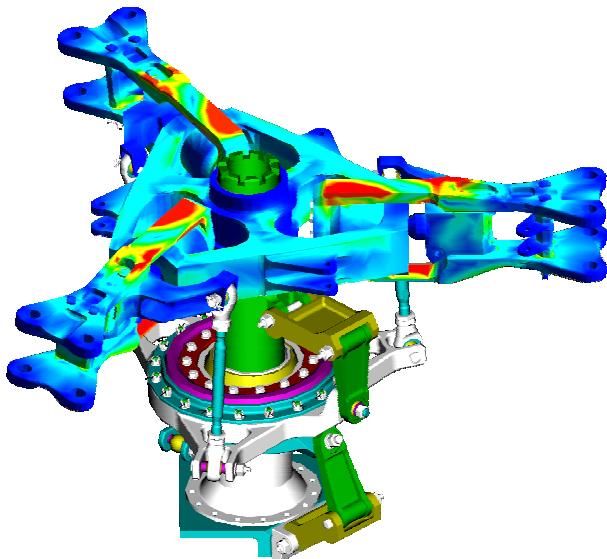
Modeling Considerations

The ANSYS Mechanical program employs the methodology of contact elements overlaid on faces of solid elements. Using the Contact Manager to define interface attributes is easy and simple. The Contact Wizard (part of Contact Manager) guides allows one to pick a pair of target and contact surfaces, then define the applicable interface properties. Tools are available to visualize initial contact status and contact directions, and even to take corrective action (for example, reversing of normals) as necessary. Similarly, ANSYS Mechanical post-processing offers many enhancements to view and interpret the results.

Simulating complex assemblies accurately is a key strength of the ANSYS Mechanical program. The Contact Manager, post-processing functionality, and the core analysis capabilities provide the tools to meet the challenge. To ensure ease of use and provide advanced analysis capabilities to a broad spectrum of specialists and non-specialists, ANSYS, Inc. has embarked upon a new initiative called ANSYS WorkBench.

The ANSYS Workbench program determines contact between parts of an assembly automatically. Figure 16 shows a helicopter rotor assembly with a large number of parts. The solution kernel is ANSYS .

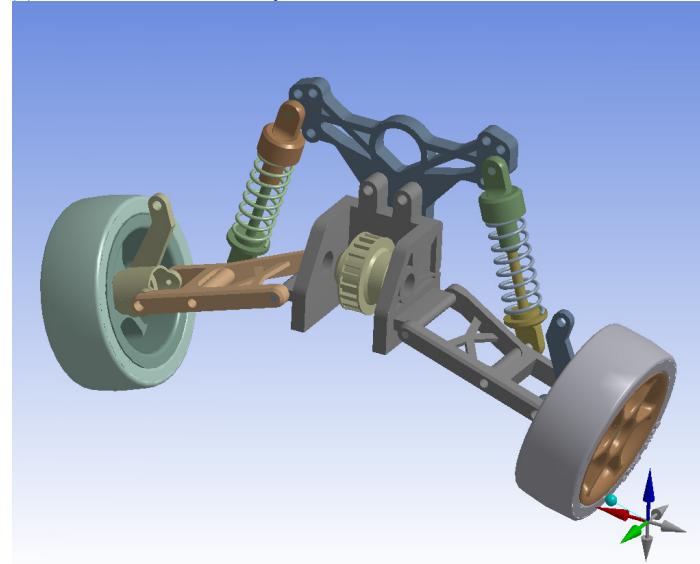
Figure 16. A helicopter rotor assembly



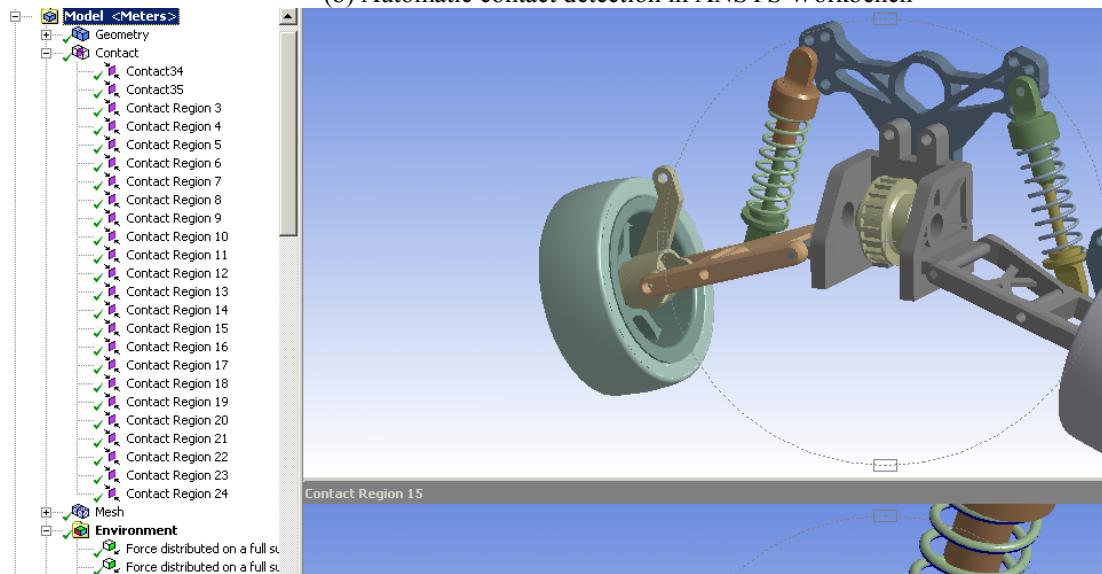
The power of ANSYS WorkBench—robust geometry, meshing, and automatic contact definition—will become available to analysts and designers at Release 7.0. It is an unparalleled combination of analysis power and ease of use. Figure 17 is self-explanatory, illustrating the salient phases of model creation, automatic contact detection, export to analysis, and the ability to use Contact Manager for more refined solution settings. In subsequent releases, ANSYS, Inc. intends to further integrate the ANSYS WorkBench and the ANSYS Mechanical solver kernel.

Figure 17. Steps towards automated contact in ANSYS Mechanical

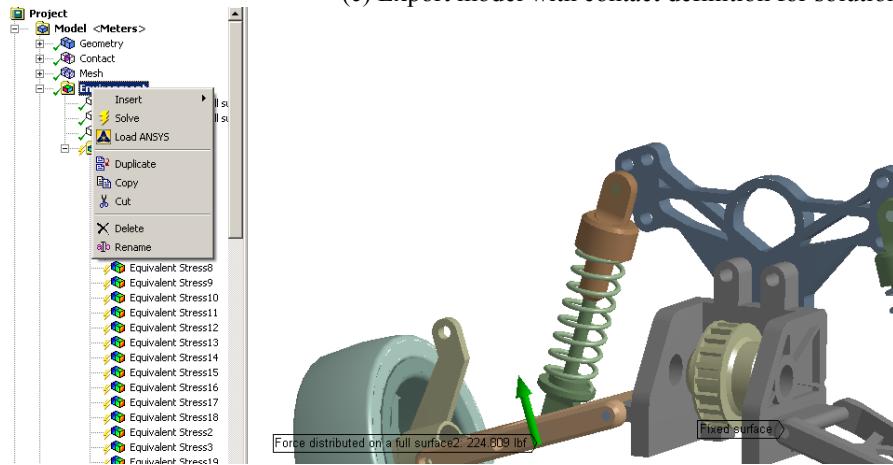
(a) Automotive assembly model



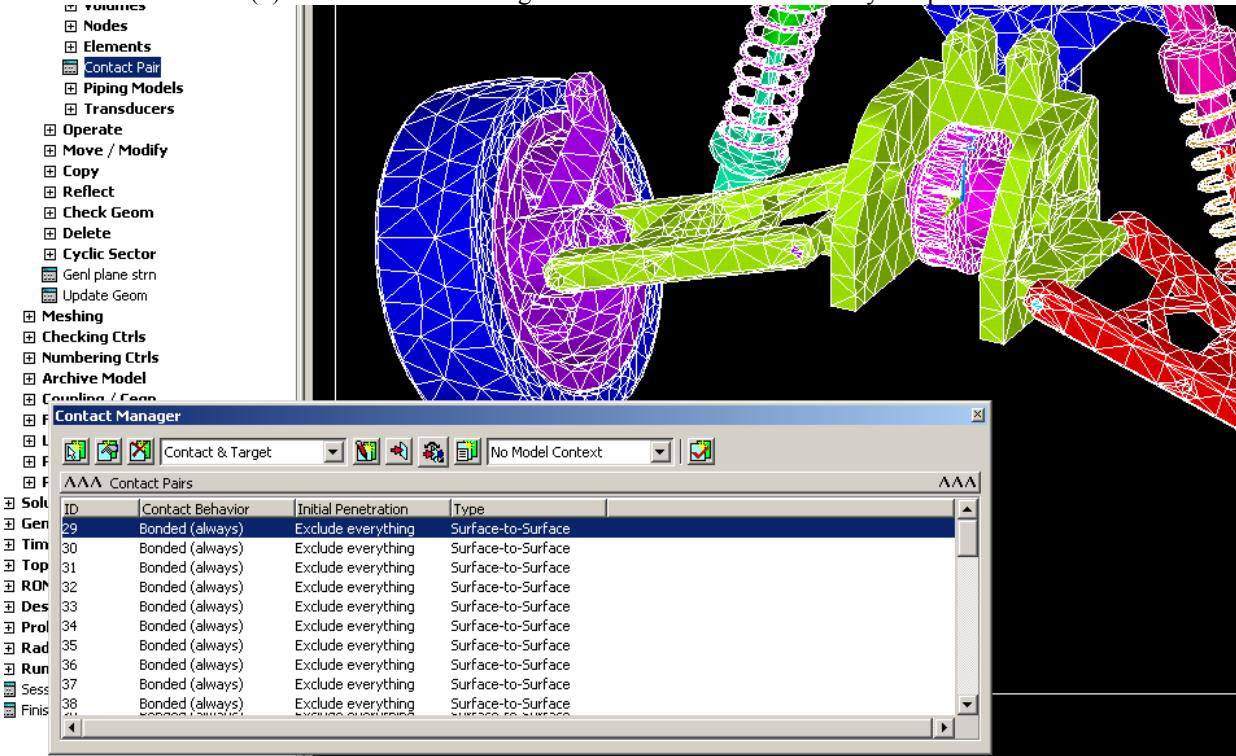
(b) Automatic contact detection in ANSYS Workbench

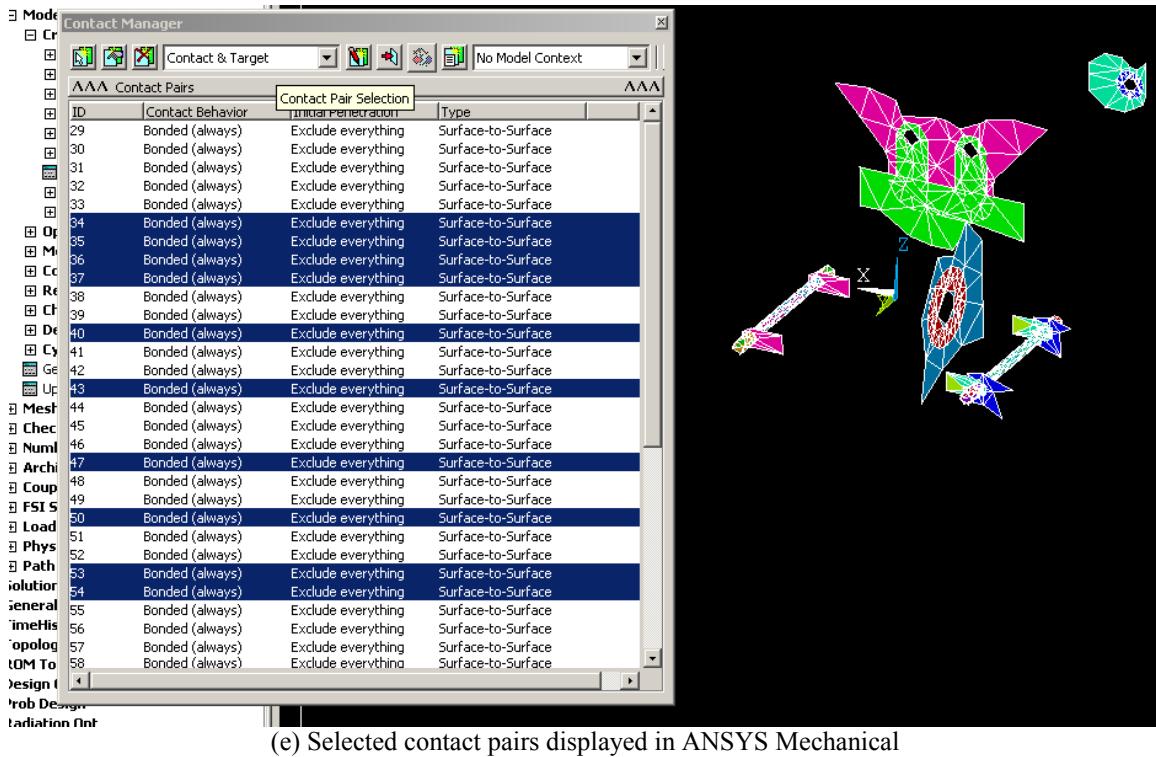


(c) Export model with contact definition for solution



(d) Invoke Contact Manager to have full control over analysis options

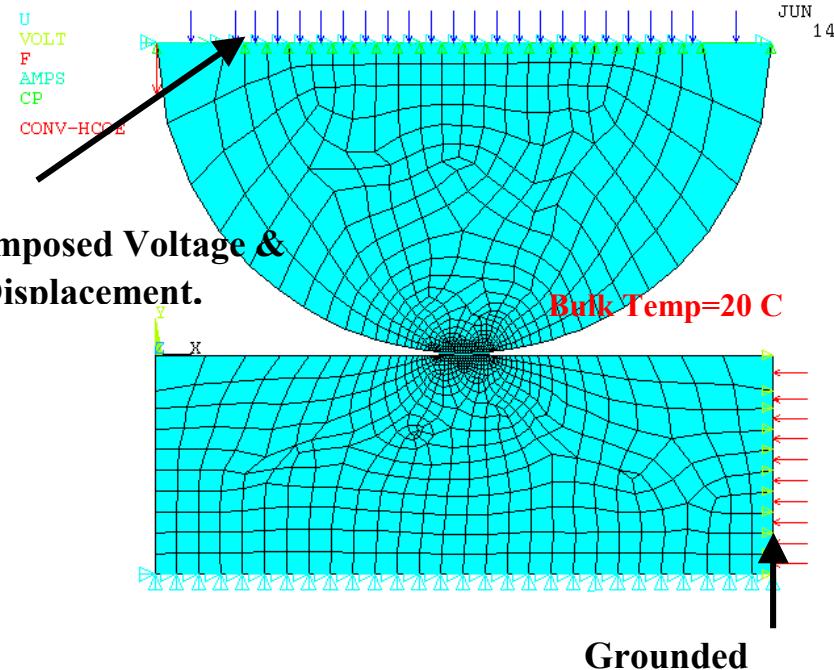




(e) Selected contact pairs displayed in ANSYS Mechanical

The surface-to-surface contact capabilities can also apply to thermal analyses. It is possible to define the thermal contact conductivity as a function of contact pressure or temperature. Similarly, electrical contact can be modeled. Figure 18 illustrates an often desired thermal-electrical-structural coupled application.

Figure 18. Thermal-structural-electric contact (multiphysics contact)



One can use contact elements for performing modal and buckling analyses with the assumptions of frozen contact status. The rigid surfaces are associated with a single pilot node acting as a handle at which displacements/forces may be prescribed. Analysis features such as material nonlinearity, bolted joints, constraint equations and procedures work in harmony.

Lagrange Multiplier Approach

The requirements of contact analysis are wide ranging, and there is no single approach that meets the needs of all. The concept of offering a toolkit is even more applicable here than in elements or materials.

The augmented Lagrangian approach, while able to solve complex contact problems, produces a certain level of penetration (generally so small that the effect is negligible); however, options are available to make it satisfactory. Still, some ANSYS customers require nearly perfect compatibility (that is, zero penetration). For those customers, Lagrange multipliers are available to enforce penetration constraints (while completely avoiding the penalty stiffness input).

The ANSYS Mechanical node-to-node contact element CONTACT178 offers the Lagrange multiplier option in addition to augmented Lagrangian treatment.

The Lagrange multiplier approach of enforcing compatibility has two disadvantages:

- An increase in the number of degrees of freedom (DOFs)
- The inability to use iterative solvers.

The Lagrange multiplier approach is also susceptible to convergence difficulties (constant chattering, necessitating artificial smoothing or viscous-based solution heuristics). ANSYS, Inc. believes that the disadvantages hamper the ability to solve large assembly models. The next section will provide more commentary on this issue.

ANSYS, Inc. is currently evaluating the Lagrange multiplier approach in general surface-to-surface contact cases, specifically for:

- Small to medium contact applications relying on direct solvers
- Contact with predominant material nonlinearity where a good guess on penalty stiffness is difficult
- An analysis environment where trial runs or the process of estimating a penalty stiffness must be avoided.

ANSYS, Inc. believes that the augmented Lagrange multiplier approach will remain its mainstream methodology for solving very large assemblies.

Recent Developments

As a result of ongoing research and development, ANSYS, Inc. has introduced a new node-to-surface element (CONTACT175). The element offers many advantages of surface-surface contact elements but without the disadvantages of the 48 and 49 elements. The primary application for the new element is edge-to-surface, corner-contact analysis. In addition, it forms a basis for upcoming Lagrange multiplier-based contact capabilities.

Equation Solvers for Nonlinear Analysis

ANSYS offers a library of equation solvers (yet another toolkit of sorts). For maximum performance, the solver selection is a function of specific problem characteristics. Issues such as predominantly bulk or bending deformation, or material behavior being compressible vs. incompressible, translate into conditionality or an eigenvalue spectrum of the system matrices influencing particular choices.

Other factors influence solver selection, such as the presence of a large number of constraint equations, whether or not multiple CPUs are available, hardware configuration details, and other factors of a similarly global nature.

Direct Solvers

By default, the ANSYS Mechanical program issues a sparse direct solver for all nonlinear problems. The sparse solver can address negative indefinite systems (common in nonlinear analysis due to stress stiffness and constitutive behavior) and Lagrange multipliers from a variety of sources (such as multipoint constraints, mixed u-P elements and contact elements). The sparse solver is applicable to real, complex, symmetric and non-symmetric systems. Non-symmetric systems are critically important for contact models with significant friction. The sparse solver is a robust choice for all forms of nonlinear analysis.

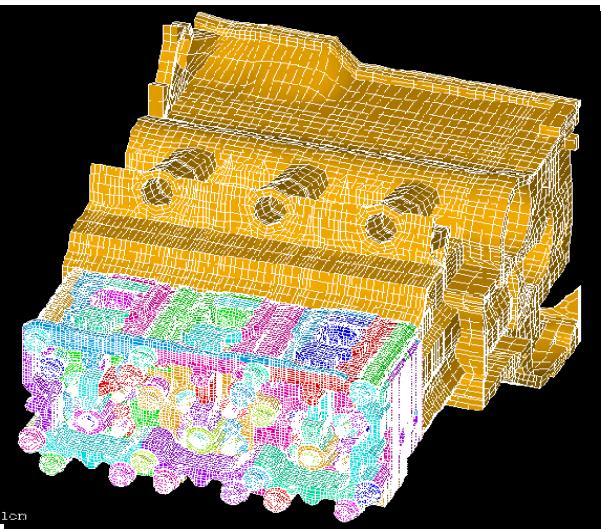
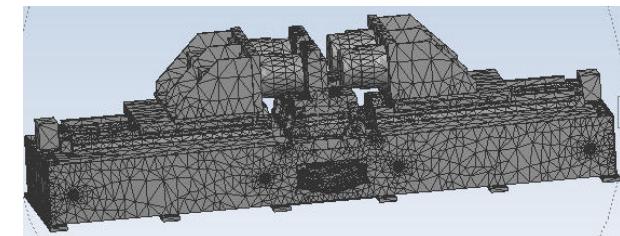
The sparse solver supports parallel processing on all supported platforms. As a general rule, one can expect a solution speed-up factor of 2 to 3.5 using 4-8 CPUs. The speed-up factor on high-end servers ranges from 3 to 6. The sparse solver performs efficiently for a wide range of problems, including multiphysics applications.

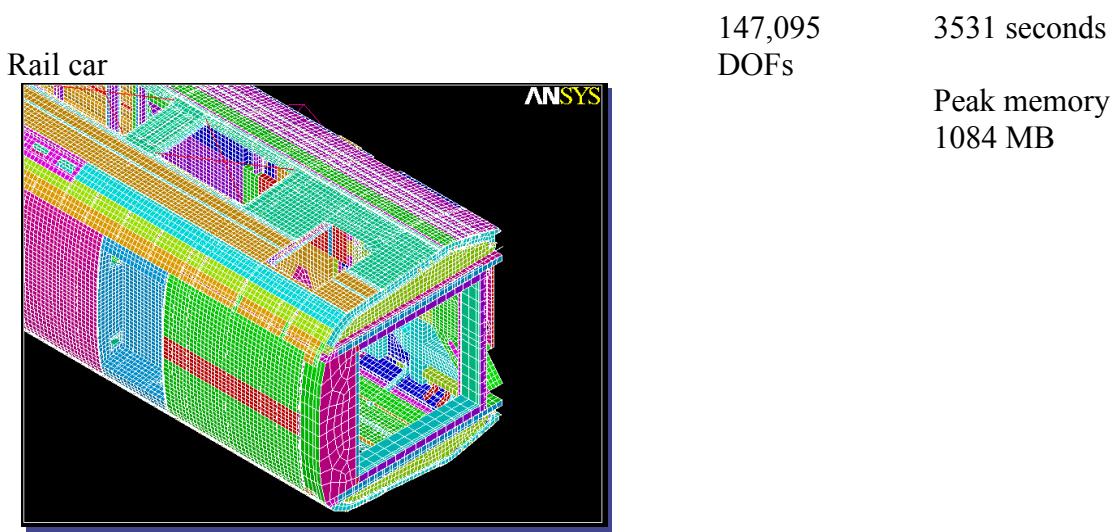
Another key strength of the sparse solver is that it provides fully out-of-core, partially out-of-core or fully in-core support; therefore, the solver can handle even large industrial problems with limited computer resources (such as low memory or disk space).

Table 7 provides some examples of sparse solver applications.

Table 7. Sparse solver examples

	Size information	Solution information
Assembly Contact	119,000 elements 590K DOFs	Maximum memory 290 MB Elapsed time 881 seconds
Engine Block – Linear analysis	410,977 hex elements 1,698,525 DOFs 20,289 constraint equations (CEs)	Solution time 7,967 seconds Peak memory 1,466 MB





Although supported by ANSYS Mechanical, this paper does not address the frontal solver because the sparse solver is by far the preferred solution option.

The sparse solver is the only reliable option for these types of applications:

- Mixed u-P elements
- Elasto-plastic analysis
- Nonlinear collapse studies using arc length
- Slender structures
- Contact with friction.

Typically, the sparse solver requires 1 GB of memory per million degrees of freedom (DOFs), and 10 GB of disk space. Computer hardware advancements have enabled the sparse solver to apply to a wide range of small- to large-scale models. For very large models (for example, those with 5-10 million DOFs), the resource requirements of a direct solver are very high.

Higher fidelity solutions and large assembly modeling often require 10 million DOFs. ANSYS, Inc. is addressing the new resource challenges via parallel processing, and iterative and domain-based solvers.

Iterative Solvers

ANSYS, Inc. was the first CAE company to introduce an iterative solver. In a nonlinear structural analysis, two of several solver options available in the ANSYS Mechanical program are relevant:

- **The PCG Solver**

The PCG solver is a preconditioned conjugate gradient solver. The solver employs a proprietary preconditioner. Initially, this iterative solver applied primarily to very large linear applications. Auto-meshing with second-order tetrahedrons combined with the PCG linear equation solver was a significant milestone in ANSYS, Inc.'s history. The PCG solver is highly efficient for bulky structures (such as engines). Its disk resource requirements are significantly lighter than those for direct solvers, while its memory requirements are similar to those of the direct sparse solver. It also offers an element-by-element option, reducing the memory requirements drastically for a given set of elements.

The PCG solver enjoys wide popularity within the ANSYS Mechanical user community. Since its debut, ANSYS, Inc. has enhanced the PCG solver for indefinite equation systems, contributing to its success in solving large, nonlinear problems.

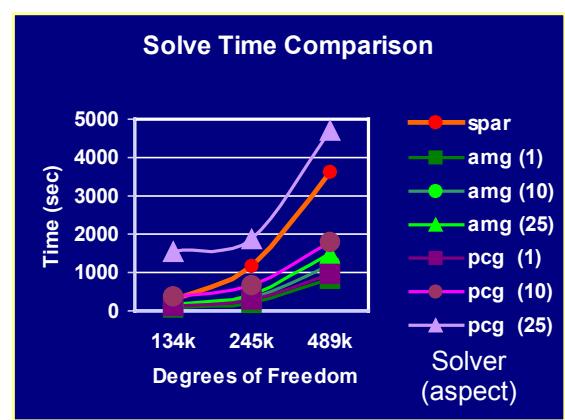
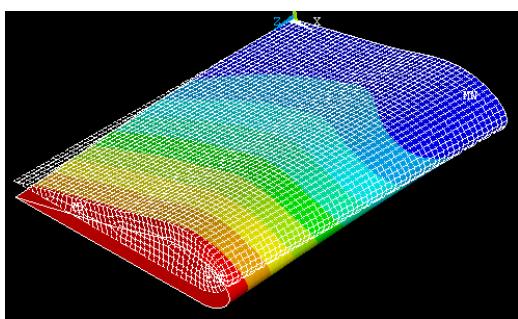
- **The AMG Solver**

The AMG solver is an algebraic multigrid iterative solver. It is more robust than the PCG solver for ill-conditioned problems (for example, problems involving a high degree of slenderness or element distortion). The solver supports shared-memory parallel processing, scaling best with about eight processors.

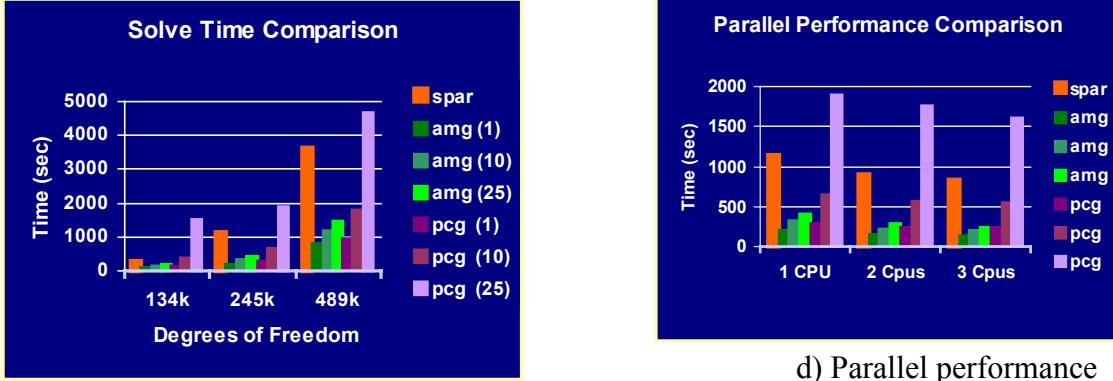
Figure 19 summarizes the results of a typical application and illustrates the factors influencing solver selection.

Figure 19. Solver comparison

A wing model illustrating solver behavior:



b) Solvers compared



d) Parallel performance

c) Increasing the aspect ratio makes matrices ill-conditioned

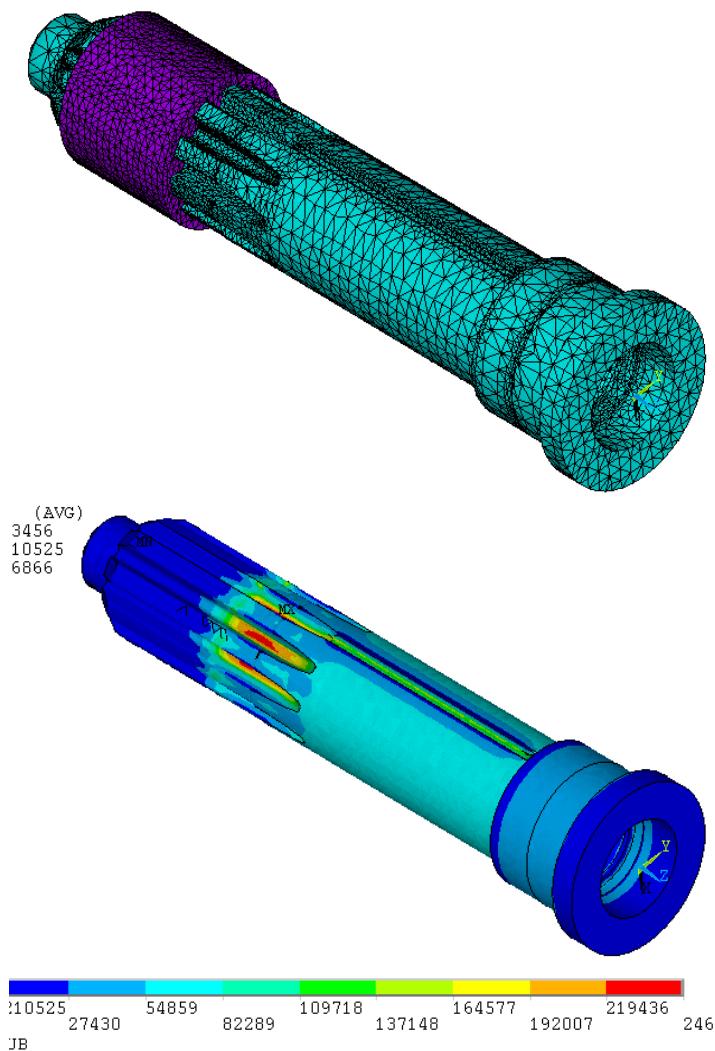
Often, iterative solvers assume positive definiteness of the system matrix, and so are inapplicable to most nonlinear problems. The PCG and AMG solvers are different, however, because they also support a subset of nonlinear applications. The subset refers to assembly analysis of multiple parts, where the nonlinearity is from contact predominantly (although certain elasto-plastic models may be used under monotonic loading). The algorithms extend the iterative solution to indefinite systems, although the solution efficiency in such cases may not be optimal.

While the PCG or AMG iterative solvers can apply to shell and beam structures, the sparse solver is more efficient. The sparse solver is also more efficient and robust for nonlinear buckling analysis.

Figure 20 shows a nonlinear contact application that employed the PCG solver. The nonlinear analysis solution time involved minutes (instead of the hours that a direct solver would have required).

A more recent study involved an engine assembly analysis. The model consisted of brick and tetrahedron elements, contact, gasket, and pre-tension sections. The model size was approximately 2.5 million DOFs. The nonlinearities involved contact and gasket elements. The direct sparse solver, although capable of solving the problem, would have required about 10 CPU hours per iteration. The PCG solver completed the solution in only 4.9 CPU hours. The AMG solver with eight CPUs completed the solution in two hours.

Figure 20. Splined shaft contact analysis



Number of 10-node tetrahedrons =
105K

Number of DOFs = 497380

Applicability of iterative solvers in nonlinear contact analysis is a significant step forward. ANSYS, Inc. has ambitious plans in the larger field of distributed processing, and currently offers a distributed domain solver (DDS). More information about the DDS and a glimpse at the future of ANSYS Mechanical solvers is available in an article by Dave Conover¹⁰.

¹⁰ Dave Conover, Towards minimizing solution time: A white paper on the trends in solving large problems, ANSYS Inc., 2001.

ANSYS Mechanical Nonlinear Analysis Support

The ANSYS Mechanical program's solution infrastructure is the common thread between the components of elements, materials, contact, and equation solvers. Together with the latest technologies implemented in kinematics, constitutive, and constraint treatment, the tools enable the efficient and accurate solution of complex problems.

ANSYS Mechanical provides automatic time stepping, requiring minimal manual intervention. The time-step size increases, decreases or holds constant based upon various convergence parameters. ANSYS Mechanical provides a status report and graphical convergence tracking. Although ANSYS, Inc. intended for the time-stepping schemes to contribute to robust analyses, they often provide the most efficient solution. The program also supports convergence enhancers such as a predictor and line search.

ANSYS has arc length method to simulate nonlinear buckling, and trace complex load-displacement response when structural is not stable. Since the displacement vector and the scalar load factor are solved simultaneously, the arc-length method itself includes automatic step algorithm.

With ANSYS Mechanical, it is possible to restart an analysis at any converged incremental step where restart files exist, allowing one to modify solution-control parameters and continue the analysis after encountering convergence difficulties. One can also modify loads to generate result data for a solved incremental step.

Solution-control heuristics are tuned to problem-specific details and reflect years of accumulated experience, hence the sometimes conservative choices. Nevertheless, one can specify solution-control parameters manually for unrivaled flexibility.

What if an analysis yields unexpected results despite ANSYS Mechanical's advanced solution technologies? In such a case, diagnostic tools are available to plot contours of element quality (in a deformed state) and visualize force residuals for a prescribed number of Newton-Raphson iterations. ANSYS Mechanical will address the need for remeshing and adaptive nonlinear analysis in subsequent releases.

Conclusion

The ANSYS Mechanical program offers comprehensive, easy-to-use nonlinear analysis capabilities and enables solutions of large-scale, complex models. An integrated infrastructure, APDL customization, programmable features, and the new paradigm of ANSYS Workbench, work together to provide tremendous simulation capabilities.

For future releases, ANSYS, Inc. intends to augment the ANSYS Mechanical program's core strengths: distributed processing, variational analysis and adaptive nonlinear analysis. ANSYS, Inc. will continue to develop advanced analysis features, with an ongoing commitment to robust capabilities, speed and ease of use.