ANSYS SOLUTIONS

THE PREMIER E-ZINE FOR DESIGN INNOVATION

FEATURES

Business Benefits of FEA

Predicting Product Performance in the Real World

ANSYS Mechanical: A Powerful Tool

Mechanical Reliability Modeling

DEPARTMENTS

Editorial

Industry News

CAE Community

Simulation@Work

Product Development Strategies

Tech File

Tips and Techniques

Managing the Enterprise

Industry Spotlight: Biomedical

Business Benefits of FEA

The value of simulation in product development becomes clear when companies consider the high cost of not using the technology.

By Vince Adams
Director of Analysis Services
IMPACT Engineering Solutions Inc.

An evaluation of the economic benefits of finite element analysis is an important part of successfully implementing the technology. Such benefits can be quantified to provide the business benefits of integrating a simulation program in the product design process. These same measures should be used throughout the implementation to continually evaluate the on-going value of using FEA.

It is tempting, at times, to utilize FEA for the sake of using the technology without stopping to evaluate the effect on product cost and profitability. Yet without a business justification, this technology will not gain real acceptance within the company and may come to be viewed as a product development team's "computer game."

It is not hard to understand that a lack of economic benefit may mean that either the simulation program may not be implemented correctly or, possibly, that it is simply not the best approach to reducing costs and improving quality. On the other hand, if savings attributed to simulation are substantial and well documented, then improved hardware, additional training and expanded software capabilities should be easy to justify.

The High Price of Delays

An evaluation of the economic justification for FEA should begin with a review of all possible business benefits of simulation. Ask the following questions. *Is the goal to reduce development time or cost? Will the greatest impact come from cutting material costs? Will market share increase if we develop safer products or improve quality?* Identification of such business benefits will keep all other decisions in focus.

Focus your evaluation on areas of the product design process where you can achieve the greatest impact most quickly. A highly visible product line or component family should be chosen and the business benefit expected should be clarified.

To properly assess the business benefits of FEA, a cross-functional team should thoroughly explore the cost to the organization of delays to projects if FEA isn't used. The team should include representation from engineering, purchasing, manufacturing, marketing and even accounting. A true understanding of all aspects of an economic justification can rarely be found in a single group within the company.

The true cost of a delayed project has several facets: product development costs, missed sales, and lost market share. Each component of the cost of delay can be used to justify integrating simulation into the product development process. They are also effective measures from which to gauge the value of performing FEA after the implementation.

Justifying FEA

The use of FEA can be justified on the basis of reducing or minimizing these delays. It is more straightforward, although less tangible, to consider the value of costs avoided due to reduced prototype iterations or improved quality as opposed to trying to quantify savings. If a prototype iteration typically takes one week and has a predictable cost, the costs avoided by catching one design problem using simulation are the sum of the aforementioned prototype costs and the profit from seven days of additional market presence.

FEA can also be justified from the perspective of long-term savings from an optimized design. It is conservative to assume that an aggressive optimization program can reduce 10 percent of the cost. Savings have been realized in the excess of 40 to 60 percent.Product Development Savings

The most tangible costs of project delay are the additional direct costs of engineering a product. These include the actual development costs and expenses such as the actual payroll and benefit cost of the engineers involved in the project.

Manufacturing downtime, prototyping and testing costs, and expenses for unplanned iterations must be taken into account in such a cost evaluation. The general costs for the multitude of salaried personnel who worry, react and many times over-react to missed schedules are usually omitted but have a direct impact on the costs associated with schedule delays.

Over the course of the entire product development cycle, these costs can add up to a significant expense that can have a huge impact on company profitability. Properly implemented, FEA benefits the company by shortening the product development cycles, thus saving on these expenses and improving bottom-line revenue.

Avoiding Missed Sales

Another facet of the cost of delay that FEA can help alleviate is lost profit from missed sales. Most marketing and sales groups will be able to provide these numbers. If not, a simple approach is to take their unit sales estimates for a year divide that by 365 and multiply by the projected unit profit. Most engineers will benefit from going through this exercise, regardless of any underlying reason for asking, because it helps keep costs and schedule commitments in a tangible perspective.

Interestingly, companies seeking to justify simulation often may be able to use this one single parameter to make their case. You would be surprised how effective a member of the sales group can be as the champion of simulation when attempting to push through an engineering acquisition. In fact, the marketing value of FEA has been taken advantage of by surprisingly few companies.

One packaging company executive once remarked that “real” drop-test answers were not nearly as important as answers that looked real when customers came through engineering. Most engineers would not knowingly offer fictitious analysis results but anyone trying to justify the technology should be aware of their potential allies in other groups within the company.

Gaining Market Share

The final aspect of the cost of delay is related to lost profits but are much harder to quantify. This is the lost market-share due to a late product introduction. Improved time to market often tops the list of benefits companies want to achieve in implementing FEA, which can be a significant factor in determining if a manufacturer can launch an innovative product before its competitors. To better understand the effects of product timing on market share, a discussion of two different product development strategies is warranted.

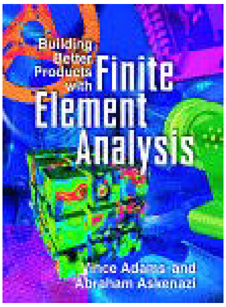
There are two types of product development scenarios in successful companies: “Market Makers” and “Fast Followers”. Market Makers are looking to introduce new technologies. They base their profitability calculations, in part, on seizing the market attention and must have a product that meets claims, at a minimum, to gain sufficient momentum so as to ward off the other type of product development organization; the Fast Follower. A Fast Follower is geared towards recognizing cutting edge technologies and in a sense riding on the coat tails of a competitors successful launch. Fast Followers must have processes in place to evaluate new technologies and isolate the strengths and weaknesses of the initial offerings. By capitalizing on the competitor's weaknesses, the Fast Follower can steal market-share.

The marketing value of FEA has been taken advantage of by surprisingly few companies.

The Sony Walkman and 3M's Post-It Notes are examples of effective Market Makers because they introduced products that had no rival at the time and sustained their leadership position with quality and marketing. Microsoft Windows and the Japanese auto industry in the early 1980s are classic examples of successful Fast Followers by either improving on products that were slow to respond to market needs, (America's Big Three auto makers), or used the best features of an existing product and bundled it differently to overwhelm the originator of the technology, (Microsoft Windows vs. Apple OS).

Market Makers tend to experience little fear of lost market-share from a delayed project standpoint. This is due to the fact that there is little initial competition. Yet, if a Market Maker comes out with an inferior product, they are fair game for a Fast Follower; then the opposite is true. A Fast Follower has much to lose by a delayed introduction. The longer the initial offerings are on the market, unchallenged, the harder it will be to displace them. Additionally, it is in the Fast Follower's best interest to be the first Fast Follower because there may be several others waiting in the wings. It is easier to grab market-share when you are only competing with the known and evaluated Market Maker. Other Fast Followers mean uncertainty and possible product failure.

All companies should understand the cost associated with project schedule over-runs and lost sales. These are very tangible figures that can be used to justify the introduction of simulation. Market Makers have less concern with lost market-share than Fast Followers, who must base their product development process on getting the design, quality and improving on the Market Maker's weaknesses right quickly. Few companies are uniquely either Market Makers or Fast Followers. Most company's product lines consist of a combination of the two. Focusing rapid product development technologies on projects as the parts within a project which have the most to gain is a key part of the evaluation process.

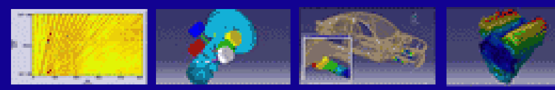


This article was excerpted from Chapter 20 of the book *Building Better Products with Finite Element Analysis* co-authored by Vince Adams, who is Director of Analysis Services at IMPACT Engineering Solutions Inc. IMPACT Engineering is an engineering-based professional services organization serving the outsourcing needs of clients in product design and engineering software consulting. The firm provides project support and high-level training in ANSYS as well as other analysis tools in its efforts to improve the efficiency and quality of the use of simulation of its clients. For more information on IMPACT Engineering Solutions, visit www.impactengsol.com.

Adams also serves as Chairman of the NAFEMS North American Steering Committee. NAFEMS is the leading international association for the engineering analysis community, dedicated to the safe and reliable use of FEA and related technologies. The NAFEMS World Congress 2003 will be held May 27-31, 2003 in Orlando, FL. This ninth international conference organized by NAFEMS brings together industrial practitioners, leading consultants, academic researchers and software developers from more than 30 countries. For more information, visit www.nafems.org.

Predicting Product Performance in the Real World

Associative link between ANSYS and LMS Virtual.Lab removes barriers to functional performance engineering.



Byline: By Dr. Urbain Vandeurzen
Chairman and CEO
LMS International

The recently announced ANSYS interface to LMS Virtual.Lab is a significant step in the engineering software industry toward integrating structural analysis for components and assemblies with system-level functional performance engineering applications for motion, noise, vibration and durability. The benefit for users is a streamlined process that enables automatic updates, eliminates time-consuming and error-prone data transfers, and significantly increases engineering productivity so companies can deliver innovative products to the market faster than ever.

Driving the development of LMS Virtual.Lab was the goal of removing barriers to accurately predicting a product's functional performance: that is, how the entire mechanical system will actually perform in the real world once all the individual parts and assemblies are put together. Massive investments have been made in CAD and digital mock-up solutions to address the form and fit of how products will look and how components will be assembled, all in virtual space.

Process Bottlenecks

Yet a major process bottleneck in designing complex products remains in accurately predicting the overall function of complex products. In the development of automobiles, for example, system-level functional performance attributes such as noise, vibration, ride, handling, comfort, and driving satisfaction often aren't apparent until prototype vehicle testing is conducted. Some problems even may not be uncovered until production begins or, worse yet, after cars are on the showroom floor. The cost to the manufacturer then becomes astronomical, not only in terms of fixing problems late in the cycle but in possibly jeopardizing market share because of customer dissatisfaction.

A major problem in handling multiple functional attributes in virtual prototypes is that data and models are not readily shared between the various CAD, CAE, and testing tools. A unified model and an integrated information base with data from each of these types of systems is essential in accurately representing the complete product. Yet differences in data formats, incompatibilities between systems, and lack of standardization usually mean that valuable engineering time is spent in non value-added activities such as reworking models, duplicating mesh representations, and copying information from one system to another.

Linking Separate Systems

To overcome this barriers to functional performance engineering, we developed LMS Virtual.Lab to provide users a unified, graphical engineering desktop and automated features for conveniently combining and synthesizing system models and loads from design, test and simulation.

Virtual.Lab is fully associative across multiple applications and disciplines, with links to leading CAD systems, finite-element codes, test data sources, multibody dynamics, acoustic radiation prediction, fatigue-life prediction, and many other CAE systems that otherwise would run separately and produce isolated results. This streamlines the process of building models, assigning loads, running the specific applications, and visualizing the results in application specific formats.

In executing the analyses, full associativity is maintained across the different applications. This means that once virtual loads are applied to calculate the vibration response of the assembly, for example, the structural responses can be used immediately to predict fatigue-life, vibration, noise, and other functional behavior. Any subsequent design change to the geometry can then be propagated in an automated manner through the entire analysis sequence. In this way, alternative design variations can be automatically evaluated and multi-attribute optimization becomes possible.

Virtual.Lab's engineering process support also facilitates enterprise deployment of functional performance engineering. The user environment captures the process flow and formalizes everyone's contribution to the product development process. In this way, team members can easily share information on different attributes and standardize on their analysis tasks across various departments and throughout the supply chain and extended enterprise.

In this way, such an approach dramatically increases value-added engineering in the simulation stage of the digital development process by reducing CAD-FEA barriers, maximizing re-use of FEA structural models for different applications, re-using real-life and virtual loads for different applications wherever possible, utilizing attribute-specific pre- and post-processing, transferring data smoothly between applications, and supporting distributed engineering in the extended enterprise.

Hybrid Simulation

Many of these functional performance capabilities of this technology are being applied in an emerging approach called hybrid simulation, where empirical data are combined with virtual simulation models to make the best use of information garnered from both analysis and test-derived models and loads. The approach is both efficient and pragmatic, especially for complex products such as automobiles, for example, where in some cases almost 80 percent of a new design is actually just a modification of an existing platform.

This approach to integrating, design, analysis and testing has the potential to greatly impact the product development process

Hybrid simulation provides a way to combine test data from these existing components with predictive data from mechanical simulation of new parts and assemblies. This makes use of the most reliable model and load data throughout the design process, with test results from previous products and existing components replaced by simulation results, as they become available. Feedback from physical prototype tests is used to update virtual models and calibrate early assumptions, which in turn provides a strong foundation for design optimization. Moreover, control elements and mechatronics can be incorporated that are often too complex to digitally model with the required accuracy.

This approach to integrating, design, analysis and testing has the potential to greatly impact the product development process. Using this technology, engineers can develop and refine product target specifications at the concept stage using data gathered from the testing of preceding designs and competitive products. Analytical models can be calibrated with test data, exercised based on these known loads, and modified in successive simulations that come closer to reality with every run.

In this way, virtual prototyping that includes capabilities for effective functional performance engineering leads to fewer, but better, hardware prototypes that serve to verify a refined design at greater levels of sophistication rather than to hurriedly troubleshoot last-minute problems. This can result in significant time reductions, cost savings, quality improvement, and design innovation for the product at hand.

Dr. Urbain Vandeurzen is Chairman and CEO of Belgium-based LMS International, a provider of CAE software, testing systems, decision-support software, and consulting services for functional performance engineering in mechanical product development. For more information on LMS International and the LMS Virtual.Lab software, visit www.lmsintl.com.

ANSYS Mechanical: A Powerful Nonlinear Simulation Tool

Software handles the growing range of problems where linear approximations are not acceptable.

By Grama R. Bhashyam
grama.bhashyam@ansys.com
Corporate Fellow, Development Manager
Mechanics & Simulation Support Group
ANSYS Inc.

and

Erke Wang
ewang@cadfem.de
Head of ANSYS Services
CAD-FEM GmbH
Germany

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. With the trend toward ever-improving simulation accuracy and the decreasing cost of computing resources, approximations of linear behavior have become less acceptable.

At the forefront of this trend, 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 a rapid pace and the robustness has been improved such that most of the advanced nonlinear applications are not restricted for experts alone.

This article summarizes the salient nonlinear features of ANSYS Mechanical in meeting user needs in each of these areas: contact interaction and assembly analysis, constitutive models for a variety of metals and non-metals, element technologies for consistent large deformation treatment, solution of large scale problems (where multiple nonlinearities interact in a complex manner), and infrastructure.

The 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 simulation capabilities for a broad range of complex problems.

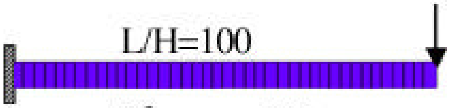
ANSYS Elements: Building Blocks of Simulation

A new generation of elements for nonlinear simulation, which address the growing needs of multiplicity in material models and application complexities, has been under development since 1995. The new generation of the elements (the 180 series) have these robustness and accuracy orientated characteristics: rich functionality, consistency with theoretical foundations employing advanced algorithms, and architectural flexibility.

As a general analysis tool, ANSYS Mechanical uses elements in wide range of applications and views its element library as a toolkit of appropriate technologies. 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. ANSYS Inc. continues to develop and refine its element technologies to make ANSYS Mechanical an increasingly powerful tool for finite deformation analysis. Brief descriptions of existing element technologies follow.

Poisson's ratio	Analytical	Plane182	conventional isoparametric fully integrated elements
0	3.75	3.75	3.7236
0.3	4.5825	4.5825	4.5227
0.49	5.0399	5.0399	4.1971
0.4999	5.0623	5.0623	0.2441

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, as shown below for a thick wall cylinder under inner pressure. **Enhanced Strain Methods**. To avoid spurious stiffening in bending-dominated problems, A 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, as shown below.



$$\delta = \frac{PL^3}{3EI}(1.0 + \frac{3EI}{KGAL^2})$$

Number of Elements		Normalized tip deflection	
Depth	Length	With enhanced strain	Without enhanced strain
1	10	0.9972	0.02
4	10	0.9973	0.02
10	10	0.9974	0.02
10	20	0.9993	0.07

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. This class of elements offers compatibility with explicit solvers.

Table 1. Solid Element Technology Summary

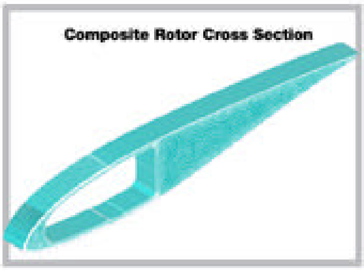
18x Solid Elements	Numbers of Nodes	Dimensions	Element Shapes	Element Order	Interpolation	Stress States					Element Technologies			Formulation Options		
						Plane Stress	Plane Strain	Axisymmetric	Generalized Plane Strain	3D	Selective Resuced Integration/B-Bar	Uniform Reduced Integration	Enhanced Strain	Displacement	Mixed u/P (nearly)	Mixed u/P (fully)
PLANE182	4	2D	Quad.	Low/Linear	Bilinear	•	•	•	•		•	•	•	•	•	•
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					•		•		•	•	•

Displacement and Mixed u-P formulations. The ANSYS Mechanical program has both pure displacement and mixed u-P formulations. In mixed u-P formulation, both displacements and hydrostatic pressure are taken as primary unknowns. Both penalty-based and Lagrangian multiplier-based mixed u-P formulations are available as follows.

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.

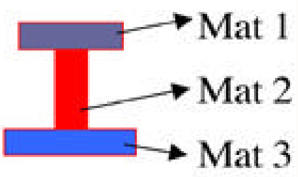
Some recent enhancements in the 180-series elements for structural applications advance the state of the art. One can now expect both robust performance and ease of use.

The beam elements (BEAM188 and BEAM189) represent a significant move towards true “reduction in dimensionality” of the problem (as opposed to simple beams). The new beam elements support complex arbitrary cross section, and calculate the inertias, shear centers, shear flow, warping rigidity, torsion constant, and shear correction factors for cross sections.


Composite Rotor Cross Section

The cross sections can also be comprised of a number of orthotropic materials, allowing for analysis of sandwich and built-up cross sections. 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 easy-to-use Beam Section Tool and full 3-D results visualization complement the robust solution kernel. All elastoplastic, hypo-viscoelastic material models may be used. The following example of a composite rotor shows this 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. The figure below shows the typical accuracy that one can achieve while enjoying the benefits of a reduced dimensionality model.

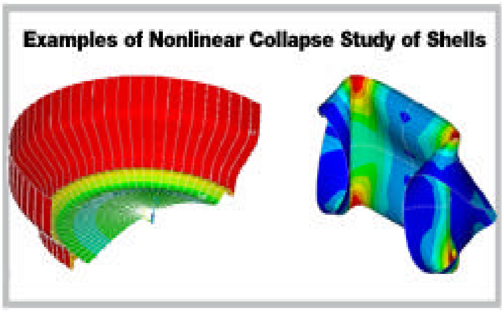


	BEAM189 (NDOF=96)		BEAM189 (NDOF=192)		SOLID186 (NDOF=18900)
Max. displacement	Value	% diff.	Value	% diff.	Reference value
Ux	19.664	0.2	19.666	0.2	19.625
Uy	24.819	1.9	24.822	1.9	25.310
Uz	54.486	0.5	54.490	0.5	54.769
CPU Time	82.610		115.460		4587.850

Nonlinear Analysis of a Curved Beam with Multiple Materials in Cross Section: a Comparison of Solid Elements

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.

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.

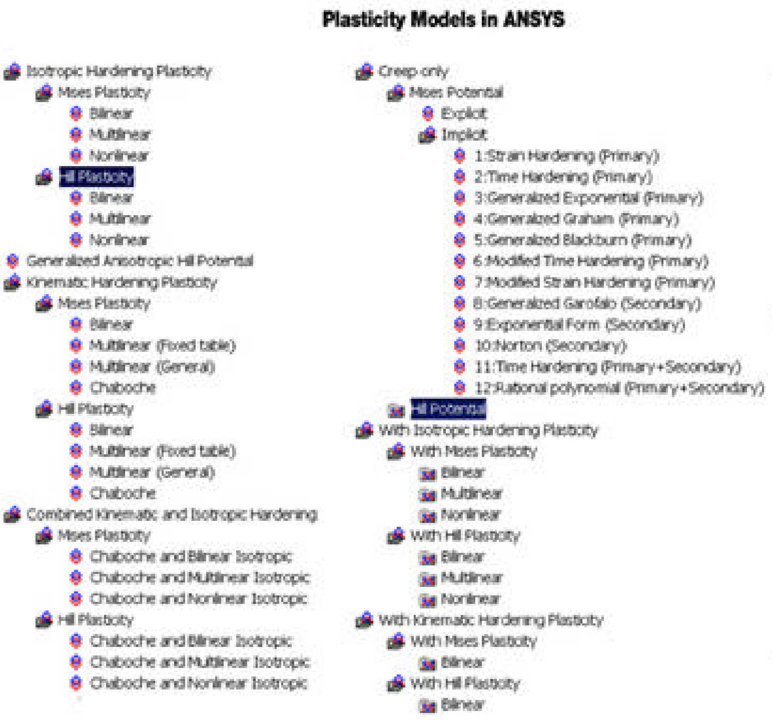

Examples of Nonlinear Collapse Study of Shells

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. The shell element thickness definition is independent of meshing and enhances accuracy by directly sampling thicknesses at element Gauss points (using ANSYS Function Builder).Examples of Nonlinear Collapse Study of Shells

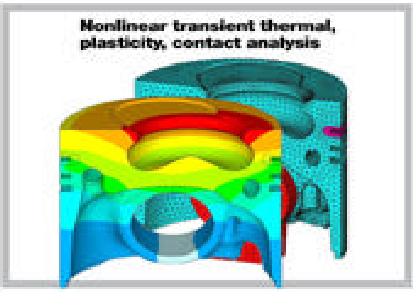
Material Nonlinearity in ANSYS Mechanical

For engineering design and application, it is essential to understand and accurately characterize material behavior. It is a challenging, complex science. Validity of the different models can be judged only on phenomenon of interest for a given application. 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. also has made every effort to obtain consistent material Jacobian contributions in order to obtain efficient, acceptable convergence rates in a nonlinear analysis. The following table provides a pictorial view of ANSYS elastic-plastic models (both rate-dependent and rate-independent forms), and non-metallic inelastic models.
Plasticity Models in ANSYS



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; the Chachoche 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).

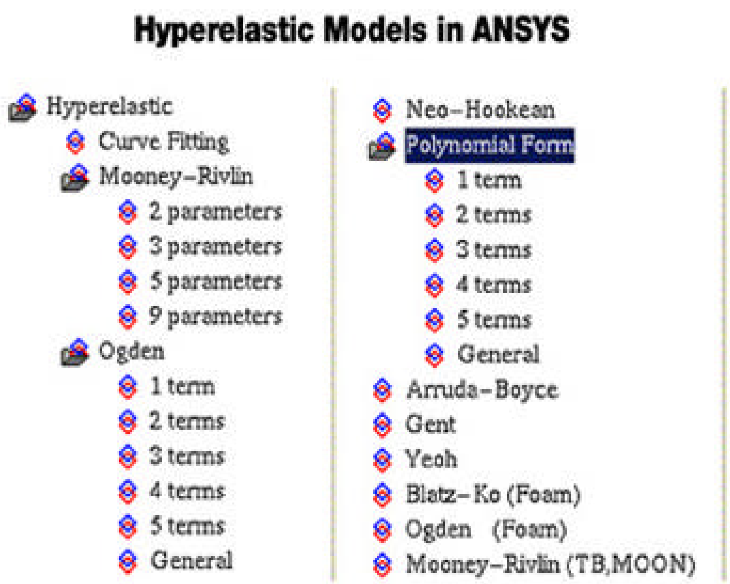
Cast Iron Plasticity. The Cast Iron (CAST, UNIAXIAL) option assumes a modified Mises yield surface, consisting of the Mises cylinder in compression and a Rankine cube in tension. It has different yield strengths, flows, and hardenings in tension and compression. Elastic behavior is isotropic, and the same in tension and compression.

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. ANSYS supports both hypo-viscoelastic and large-strain hyper-viscoelasticity.

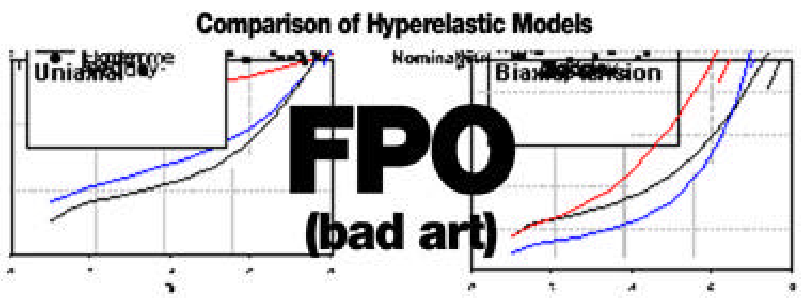
Viscoplasticity and Creep. ANSYS program has several options for modeling rate-dependent behavior of materials, including creep. Creep options include a variety of creep laws that are suitable for convention creep analyses. Anand's model (Anand, L., "Constitutive Equations for Hot-Working of Metals," International Journal of Plasticity, Vol. 1, pp. 213-231, 1985), 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.

Hyperelasticity. Elastomers have a variety of applications. 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, and complex interaction of elastomeric material with adjoining regions of metal

The table below 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.

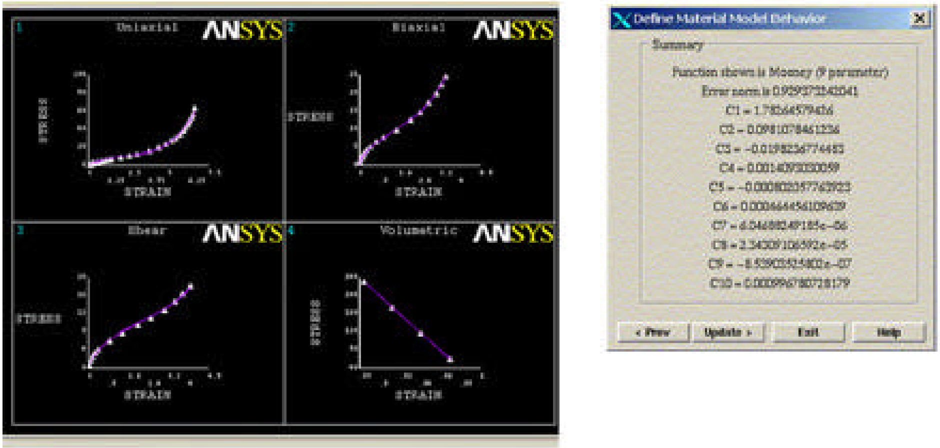


Validity and suitability of the hyperelastic models depend upon application specifics and the availability of experimental data. The figure below provides a glimpse at comparison of Mooney-Rivilin, Arruda-Boyce and Ogden models with experimental data for a particular test.



ANSYS Mechanical software allows one to input experimental data and obtain hyperelastic coefficients via linear and nonlinear regression analysis. The capability is valid for all supported hyperelastic models, and future releases will extend support to viscoelasticity and creep analysis. The following figure illustrates the new feature.

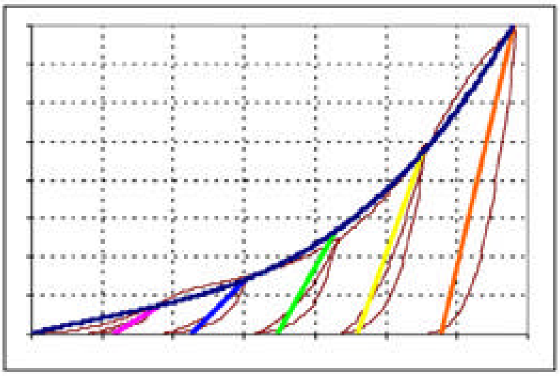
Experimental Input and Curve Fit



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

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. The figure below shows the experimental pressure vs. disclosure (relative displacement of top and bottom gasket surfaces) data for the graphite composite gasket material.

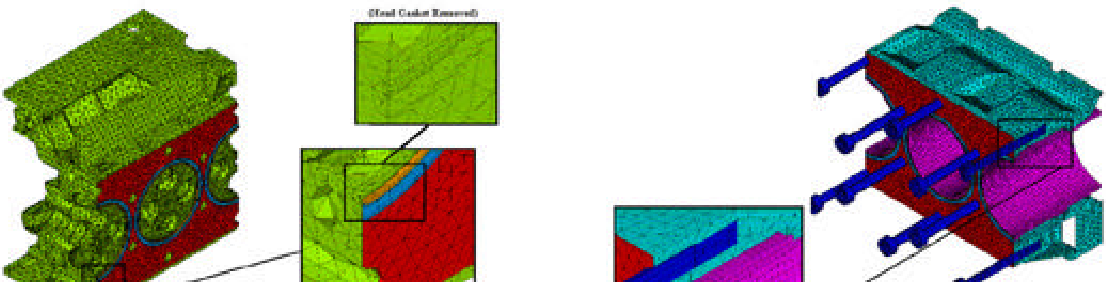
Gasket Material Behavior

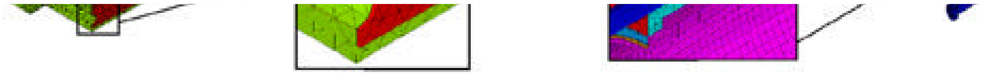


The following figures depict a typical gasket application. These images are from a paper entitled *Modeling Diesel Engine Cylinder Head Gaskets Using the Gasket Material Option of the SOLID185 Element* presented by an ANSYS Mechanical user Jonathan Raub at the ANSYS User Conference 2002 in Pittsburgh, Pa.

The following figures depict a typical gasket application. *These images are from a paper Modeling Diesel Engine Cylinder Head Gaskets Using the Gasket Material Option of the SOLID185* Element presented by an ANSYS Mechanical user Jonathan Raub at the ANSYS User Conference 2002 in Pittsburgh, Pa.

Engine Assembly and Gasket Use





Constitutive modeling analysis needs are constantly expanding. ANSYS, Inc. has taken a number of initiatives to address the needs emerging in the microelectronics, bioengineering, 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.

ANSYS offered contact analysis features as early as Release 2.0. The following table provides a summary of that evolution.

	Point-to-Point			Point-to-Surface			Surface-to-Surface	
	CONTAC 12	CONTAC 52	CONTAC 178	CONTAC 26	CONTAC 48	CONTAC 49	CONTAC 171,172 TARGET 169	CONTAC 173,174 TARGET 170
Point-to-Point	Y	Y	Y					
Point-to-Surface				Y	Y	Y		
Surface-to-Surface					Y	Y	Y	Y
2-D	Y		Y	Y	Y		Y	
3-D		Y	Y			Y		Y
Sliding	small	small	small	Large	large	large	large	large
Curved Surfaces							Y	Y
Cylinderical Gap	Y		Y					
Pure Lagrange Multiplier			Y					
Augmented Lagrange Multiplier			Y		Y	Y	Y	Y
Contact Stiffness	user-defined	user-defined	semi-automatic	user-defined	user-defined	user-defined	semi- automatic	semi- automatic
Auto-meshing Tools	EINTF	EINTF	EINTF	None	GCGEN	GCGEN	ESURF	ESURF
Lower-Order	Y	Y	Y	Y	Y	Y	Y	Y
Higher- Order				Y			Y	Y
Rigid-Flexible	Y	Y	Y	Y	Y	Y	Y	Y
Flexible- Flexible		Y	Y	Y	Y	Y	Y	Y
Thermal Contact					Y	Y	Y	Y

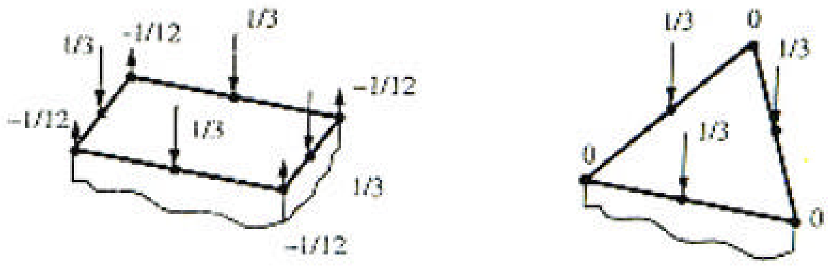
Surface-to-Surface Contact. At Release 5.4 (circa 1997), ANSYS Mechanical introduced a radical impovement in contact analysis capabilities. The augmented Lagrange method with penalty is the basis for the elements, but the penalty stiffness, selected by default, is a function of many factors (including the size of adjoining elements and the properties of underlying materials) and can be updated based upon the stresses in underlying elements.

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 the figure below). 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.

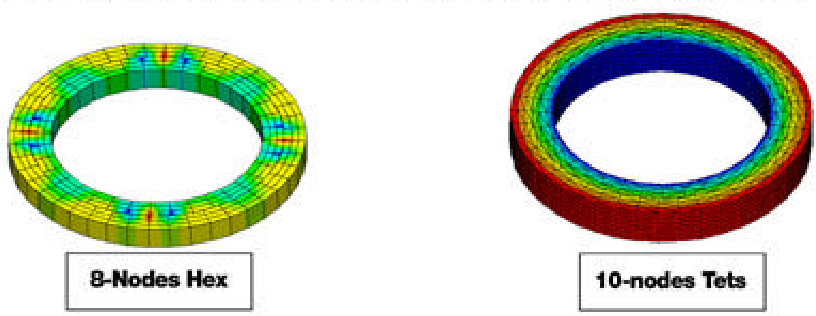
Equivalent Nodal Forces in Higher Order Elements



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).

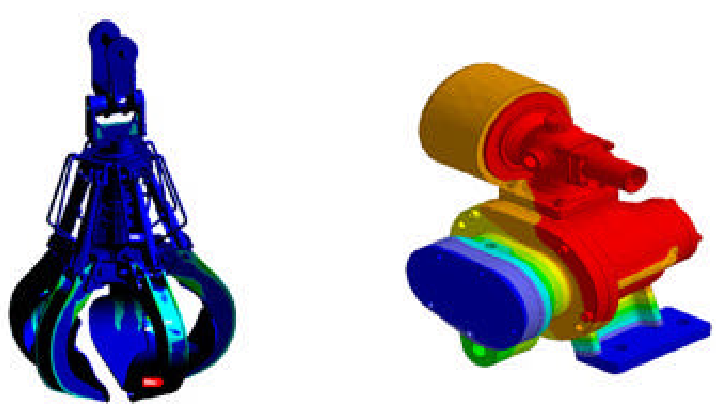
The figure below shows the stress contours of a circular prismatic solid. The anticipated results is a state of constant stress around the circumference. A solution based on facet approximation surfaces would yield grossly inadequate results. (Note the results for the eight-node element exhibiting spurious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

Need for Second-Order Representation in Modeling Contact between Curved Surfaces

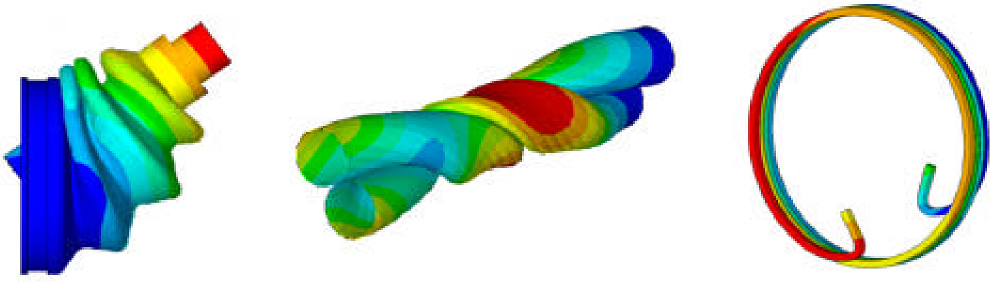


The ANSYS Mechnical program’s contact elements provide a rich set of initial adjustment and interaction models. Besides the standard unilateral contact, it offers the optons of bonded, no-separation, and rough sliding contact. The bonded contact option is especially useful, as the application shown in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

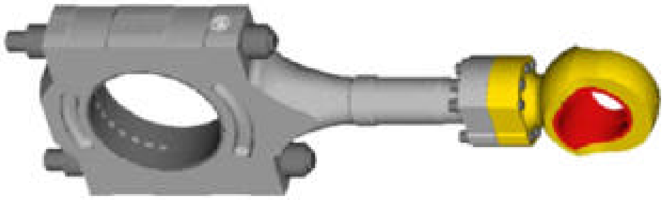
Assembly Contact



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



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.



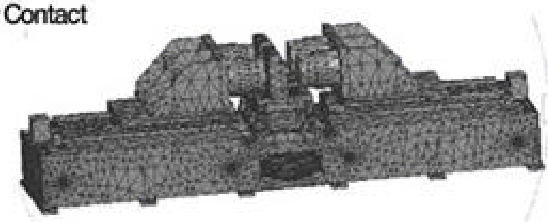
Equation Solvers for Nonlinear Analysis

ANSYS offers a library of equation solvers. 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.

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.

Sparse Solver Examples

Assembly Contact		119,000 elements 590K DOFs	Maximum memory 290 MB Elapsed time 881 seconds

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.

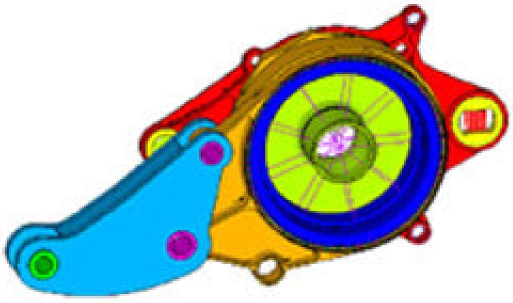
Iterative Solvers. ANSYS Inc. was the first CAE company to introduce an iterative solver. In a nonlinear structural analysis, two of several solver options available in the ANSYS Mechanical program are relevant: the PCG solver and the AMG solverThe PCG solver is a preconditioned conjugate gradient solver.

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. 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 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.

The figure below summarizes the results of a typical application and illustrates the factors influencing solver selection.

Solver Comparison



1.16 Mio. DOFs
nonlinear contact analysis:
AMG with 4 CPUs: 30 Min.
PCG with 1 CPU: 2 H

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.

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.

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 Towards Minimizing Solution Time: A White Paper on the Trends in Solving Large Problems, ANSYS Inc. 2001 by Dave Conover.

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.

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.

ANSYS Mechanical will address the need for remeshing and adaptive nonlinear analysis in subsequent releases.

The authors wish to gratefully acknowledge the input of Dr. Guoyu Lin, Dr. Jin Wang, and Dr. Yongyi Zhu in the preparation of this article.

Please visit the ANSYS Nonlinear site for more information

Analysis of IC Packaging

FEA helps electronics manufacturers save time, cost and resources in developing packaging for semiconductor-based products.

By Tong Yan Tee
Lead R&D Engineer and CAE Team Leader
STMicroelectronics

Mechanical simulation, in particular finite element analysis, is a critical part of analyzing the reliability of integrated circuitry (IC). The traditional design-build-test-redesign development cycle is no longer efficient for the competitive semiconductor industry. With standard package and board-level reliability tests typically taking up to three to four months to complete, computer-aided engineering (CAE) in conjunction with design of experiments (DOE) studies of package geometry, material properties, and test conditions become more efficient in terms of time, cost and manpower. Figure 1 describes the role of CAE to evaluate new ideas and develop optimized design with less reliance on physical prototype testing.

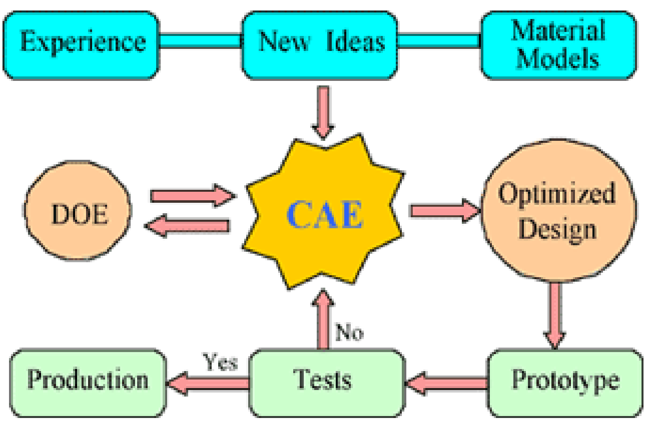


Figure 1. Role of CAE in New Package Design

The mechanical modeling capabilities used by the Corporate Package Development Group of STMicroelectronics are shown in Figure 2. Efforts cover both package and board-level modeling using structural analysis, mass and heat transfer, thermodynamics, dynamic simulation, and other types of multiphysics modeling. ANSYS Mechanical and ANSYS LS-DYNA with Drop Test Module are utilized for detailed static and dynamic parametric modeling. Selected modeling scenarios and case studies are presented below.

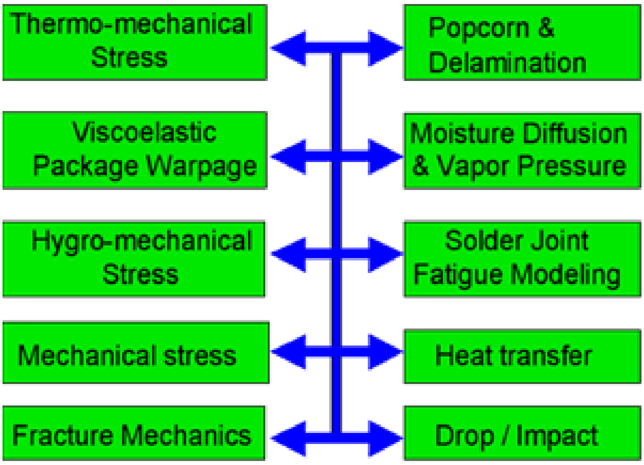


Figure 2. CAE capabilities in STMicroelectronics Thermo-Mechanical Stress Modeling

Linear-elastic stress modeling is applied frequently to analyze the structural reliability of packages such as BGA, PLCC, TSOP, touch chip, etc. This type of analysis is fairly easy to model and useful when there is limited time given by the customer. Sometimes, a more detailed elastic-plastic stress model may be used if the material properties required are available. The models are usually correlated with warpage measurements from thermoiré, and the failure mode is compared with failure analysis (i.e. cross section). The validated model can be used to identify areas with high stress concentration, which are susceptible to failures. The analysis can be extended with fracture mechanics to analyze die and package cracking, and interfacial delamination.

Viscoelastic Warpage

Viscoelastic modeling is usually applied to provide an accurate prediction of package warpage. The analysis is more sophisticated than linear-elastic modeling and requires time-dependent viscoelastic material properties (i.e. for mold compound). For linear viscoelastic thermo-rheologically simple material, the constitutive relation may be expressed as

$$\sigma(t) = E_{\infty}\varepsilon_{o}(t) + \int_{0}^{t} \Delta E[\xi(t) - \xi(\tau)]\frac{d\varepsilon_{o}}{dt}d\tau \qquad (1)$$

A strip of TFBGA (Thin-profile Fine-pitch BGA) with four blocks (5x6 matrix) is studied. The warpage induced after the post mold cure process is critical for block TFBGA because it affects the solder ball attach process, and also the subsequent package singulation process. Comprehensive warpage analysis has been performed on block BGA through the experimental warpage measurement, the material characterization of mold compound viscoelastic properties, the real-time shadow moiré measurements, and the viscoelastic finite element modeling with parametric studies. The 3-D viscoelastic model correlates well with the experimental warpage measurement, as shown in Figure 3. The validated 3-D viscoelastic finite element model with consideration of chemical shrinkage can be used as a reliable tool for BGA block warpage prediction.

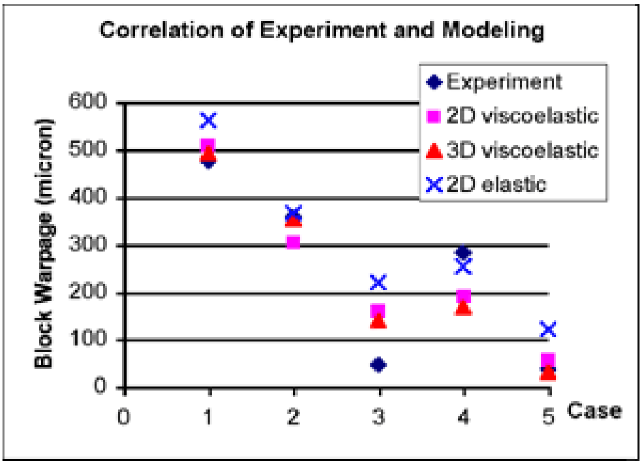


Figure 3. Correlation of Modeling and ExperimentMoisture Diffusion

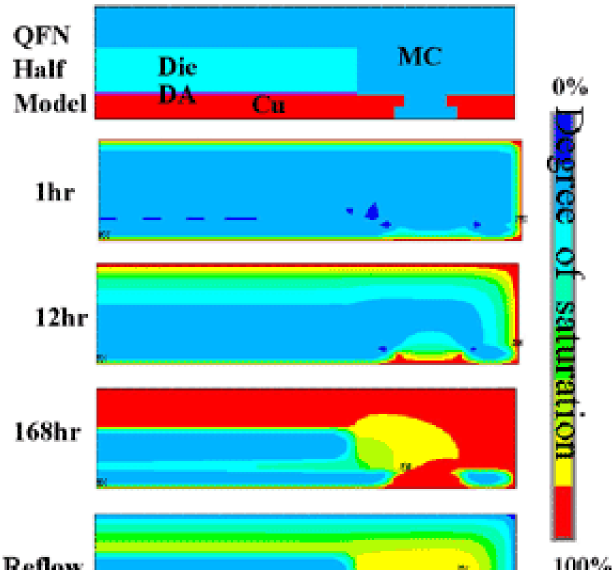
The moisture-induced failures (i.e., popcorn and delamination) of IC packages are common phenomena during solder reflow. The failures are due to sudden vaporization of moisture absorbed by the package at high temperature condition. It is well known that the package cracking is not controlled by the absolute water weight gain, rather it is due to the local moisture concentration at the critical interface. Therefore, the knowledge of moisture distribution in the package is critical to minimize the moisture-induced failures. The transient moisture diffusion equation is analogous to heat conduction, and it can be described by Fick's Law as

$$\frac{\partial C}{\partial t} = D\left(\frac{\partial^2 C}{\partial x^2} + \frac{\partial^2 C}{\partial y^2} + \frac{\partial^2 C}{\partial z^2}\right) \qquad (2)$$

where C is the local moisture concentration, x, y, z are the spatial coordinates, D is the diffusivity, which measures the rate of diffusion, and t is the time.

Figure 4 shows the transient moisture distribution of QFN (Quad Flat No-lead) package during moisture preconditioning and reflow. The moisture diffuses into the package through mold compound, and gradually spreads into die attach layer. At the end of 168 hours of moisture preconditioning under 85°C/85%RH, the package is almost fully saturated with moisture.

Results of moisture distribution can be used as input for further hygroswelling stress modeling and vapor pressure modeling (see later sections). Moisture diffusion modeling can also be applied to calculate an equivalent moisture preconditioning time for certain JEDEC level under a different moisture condition. For example, for CLCC (ceramic leadless chip carrier) package which should pass JEDEC Level 3 (30°C/60%RH, 192hr), the required testing time is too long for sample screening. An accelerated test condition (85°C/85%RH, 50min) is approximated by moisture diffusion modeling to screen the test samples in much shorter time.



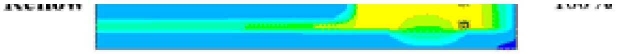


Figure 4. Transient Moisture Distribution in QFN Package Hygroswelling

Thermo-mechanical stress is induced by CTE mismatch of multi-material system. Similarly, hygroswelling or hygro-mechanical stress is due to CME (coefficient of moisture expansion) mismatch of packaging materials. Polymeric materials expand or swell when moisture is absorbed. Different material has unique moisture and hygroswelling material properties. Special material characterization technique is required to measure the CME. The hygro-strain can be defined as

$$\varepsilon_h = \beta C \qquad (3)$$

where β is the CME, and C is the moisture concentration. The equation is analogous to thermal strain

$$\varepsilon_T = \alpha \Delta T \qquad (4)$$

where α is the CTE, and ΔT is the temperature loading. Therefore, The hygro-mechanical problem can be solved using the same procedure as a typical thermo-mechanical solution.

Figure 5 shows an example of UBM opening failure of FCBGA after the pressure cooker test. The tensile hygroswelling stress along solder bump/UBM interface is responsible for the failure. The magnitude of hygroswelling stress is of the same order as thermo-mechanical stress.

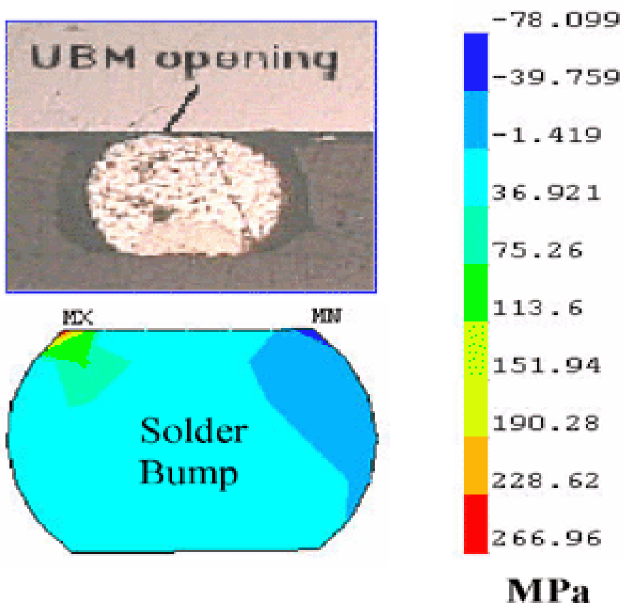


Figure 5. UBM Failure Induced by Hygroswelling Stress

Vapor Pressure

The package vapor pressure distribution during reflow is the key factor in understanding the popcorn failure mechanism. Moisture diffusion model is applied to predict the local moisture concentration at the critical interfaces, which can be used for the subsequent vapor pressure calculations. The vapor pressure modeling applies the micro-mechanics approach, the Representative Volume Element (RVE), with consideration of the micro-void effect.

There are three distinct cases at which vapor pressure can be computed. High moisture concentration weakens the critical interfacial adhesion, generates vapor pressure during reflow, and induces hygro-mechanical stress in the package. The vapor pressure saturates much faster than the moisture diffusion. At reflow temperature, the vapor pressure generated can never go beyond the saturated pressure, i.e. pressure of 2.32 MPa at 220°C. Figure 6 shows an example of QFN vapor pressure distribution during the reflow. The vapor pressure induces additional mismatch to the package, which is of the same order as the CTE and CME mismatch.

Figure 6. QFN Vapor Pressure Distribution during Reflow

Integrated Stress Modeling

The actual package failure mechanism during reflow is complex, contributed by combined effects of process defects, interfacial adhesion strength, moisture, vapor pressure, thermo-mechanical stress, and hygro-mechanical stress. Therefore, there is a need for comprehensive modeling studies on moisture diffusion, thermal, hygro-mechanical stress, thermo-mechanical stress, and vapor pressure models during reflow.

An integrated stress model with interfacial fracture mechanics (see Figure 7) is a useful tool to analyze and enhance the reliability of package during reflow, minimizing the delamination and popcorn failures. The vapor pressure induced stress is in the same order of magnitude as thermo-mechanical stress and hygro-mechanical stress.

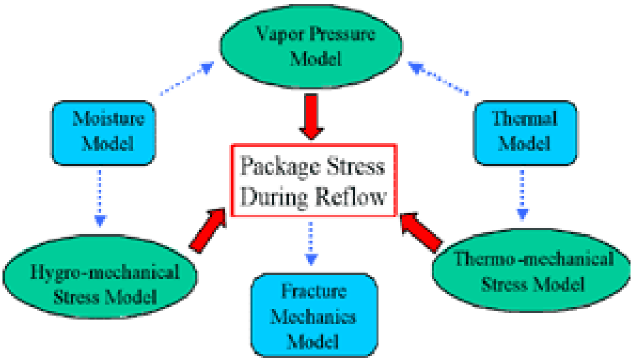


Figure 7. Integrated Package Stress Model during Reflow

Board-Level Solder Joint Reliability

Board-level solder joint reliability (SJR) is a critical issue for CSP such as TFBGA and QFN packages during the thermal cycling test. A typical board level thermal cycling test requires about 2-3 months, and the experiment matrix is limited. A validated SJR model can save time, cost, and manpower in performing the thermal cycling test. It also helps end customers to design more reliable products and shorten the time-to-market. The SJR modeling also can be integrated with electrical simulation and thermal analysis for a complete board design solution.

For typical solder materials (e.g. 63Sn37Pb), creep is the dominant process during thermal cycling condition. Steady-state creep of solder can be expressed by a constitutive equation in the form of

$$\frac{d\epsilon_s}{dt} = C_{ss}[\sinh(\alpha\sigma)]^n \exp\left(\frac{-Q_a}{kT}\right) \qquad (5)$$

where $\frac{d\epsilon_s}{dt}$ is the steady state strain rate, k is the Boltzmann's constant, T is the absolute temperature, σ is the applied stress, Q_a is the apparent activation energy, n is the stress exponent, α is the stress level at which the power law dependence breaks down, and C_{ss} is a constant.

Figure 8 shows an example of SJR analysis for TFBGA package. The fatigue model applied is based on modified Darveaux's approach with nonlinear viscoplastic analysis of solder balls. The model includes detailed pad geometry and the realistic shape of solder balls. Temperature-dependent material properties are considered for the BGA packaging materials used. The critical solder ball is observed to be at the diagonal corner. The failure is observed to be along top solder ball/pad interface, correlates well with the region with the maximum strain energy density (SED), calculated by SJR model.

The model is correlated with six sets of BGA thermal cycling data to establish a connection between the SED obtained from the FEA model and the actual characteristic life during the thermal cycling test. Higher SED leads to shorter fatigue life. The FEA-thermal cycling correlation is within ±13% error limit. This excellent fatigue life prediction capability gives confidence to customers to rely on modeling for evaluation of board level SJR for new TFBGA packages, and minimize on the time-consuming and expensive thermal cycling tests.

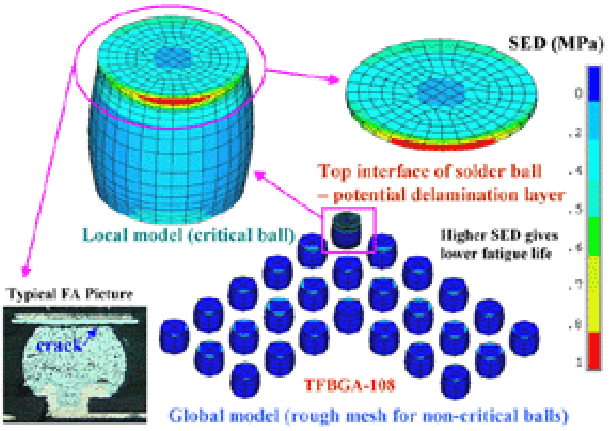


Figure 8. Strain Energy Density Distribution in Solder Balls

QFN packages are getting popular as a low-cost solution for applications with low pin-count requirements. From the failure analysis (see Figure 9), it is clear that the peripheral solder joint cracks and propagates through the top interface. The failure mode agrees well with high SED region predicted by SJR modeling of QFN.

The solder joint fatigue life calculated by modeling is correlated with 8 sets of thermal cycling data of QFN and OQFN (optical QFN) packages. The correlation is good, within ±34% difference. The parametric modeling studies on package and board geometry, material properties, and thermal cycling test condition, are useful to package designer in the package development stage.

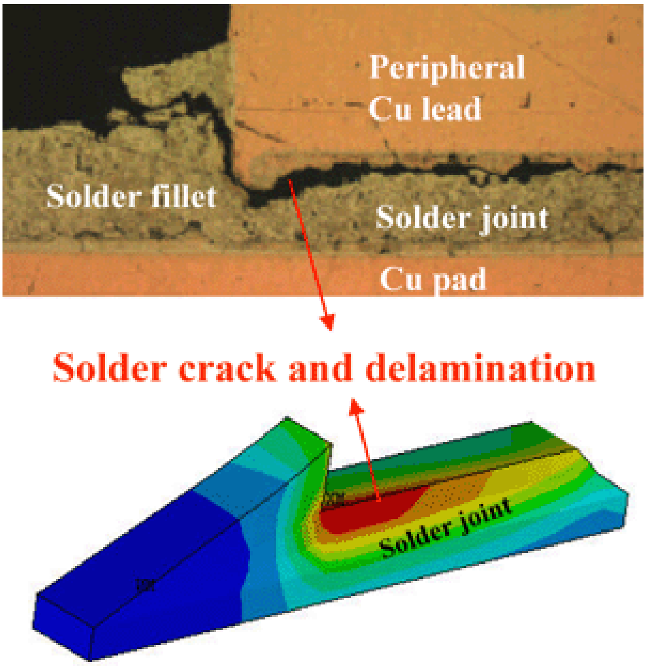


Figure 9. Correlation of QFN Failure Mode with Modeling

Drop Test Simulation

Board level solder joint reliability during drop impact is a great concern to semiconductor and electronic product manufacturers, especially for portable telecommunication devices such as mobile phone and PDA. The mechanical shock resulted from mishandling during transportation or customer usage, may cause solder joint failure, which leads to malfunction of product.

Figure 10 shows an example of TFBGA108 under board level drop test. Both experiment and modeling show that the critical solder ball is at the outermost corner, and failure occurs along the solder/PCB pad interface. The solder joint failure is induced by the combined stress of mechanical shock and PCB bending.

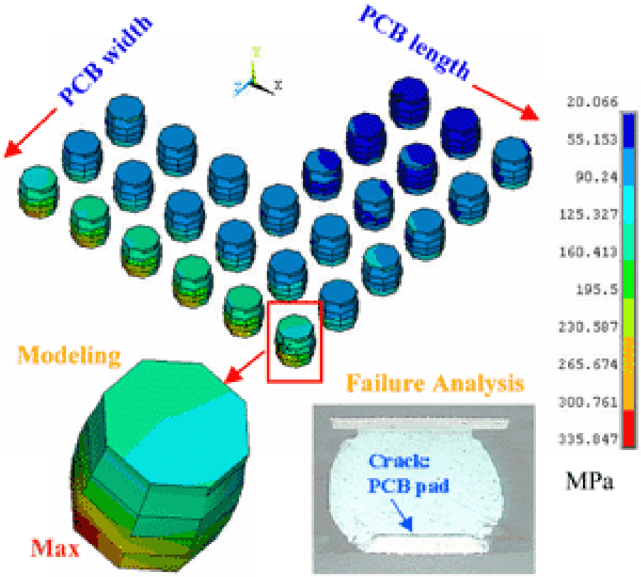
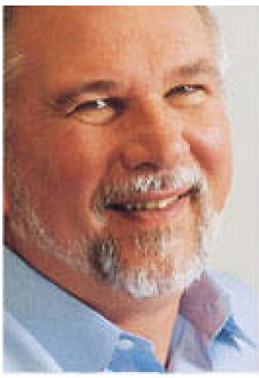


Figure 10. Von Mises Stress Distribution of Solder Ball

Innovation Requires Strategic Investments

Companies need creative people with the right simulation tools and product development processes.



John Krouse
Editorial Director

Until the last few years, manufacturers who launched products faster, made them better, or had the lowestprices were the most successful. Time-to-market, quality, and cost still are critical to maintaining business success.

But in today's world, they just aren't enough. Instead, the competitive advantage now goes to companies able to satisfy these market demands plus deliver true product innovation. Not just new colors or different sizes or jazzed-up slogans, but designs never seen before, ones no other company can deliver, something that catches competitors by surprise and knocks the socks off buyers. Product innovation is what differentiates companies and identifies the brand value they bring to the market. Innovation is what drives success in today's highly competitive markets.

Innovation doesn't just happen by itself, of course. And it won't come about because managers order workers to be innovative or because executives wish it to be. Innovation can't be mandated, nor does it emerge overnight. Rather, it must be fostered and facilitated over time.

A company needs creative people with the right tools and processes conducive to developing fresh ideas. With the specialized expertise and knowledge they possess, people are a company's greatest asset to be leveraged in the product development process. Tools such as simulation software can be applied in optimizing designs, evaluating alternatives, doing tradeoff studies, and analyzing product performance not practical or otherwise possible. Streamlined processes that embrace change, invite new thinking, and give workers the freedom to make mistakes foster a creative atmosphere where innovative concepts can emerge and take shape.

All this requires some level of investment in salaries to hire and retain talented and skilled workers, money to license the latest software, and resources to drive collaborative processes for getting work done efficiently.

Unfortunately, some companies have made poor decisions in this regard by cutting back on, delaying, or entirely neglecting to make such investments. One company axed some of their best people in the name of "rightsizing," a term they thought would make their cost-cutting attempts sound better than "downsizing." No innovation going on there now. Core talent is gone, along with their imaginative ideas. Vanished also is any improvement in the bottom line, with revenues down considerably due to lower product quality and a damaged industry image.

To save money in the short-term, companies may forego giving engineers badly needed solutions such as simulation technology because it's not in the budget. Executives see projects getting done without the benefit of such tools, so (as the reasoning goes) maybe people should just do their jobs faster or put in longer hours to improve output. Many organizations try to accelerate processes merely by telling people to work harder rather than making substantive changes in project workflow, operating procedures, or company culture.

While these companies are skimping on important expenditures to make quarterly bottom-line numbers look good, other forward-looking firms with a long-range view are demonstrating decisive leadership positions in their respective markets through strategic investments in the people, technology, and processes required for product innovation. Astute executives at these companies know that when the economy kicks back in again, their innovative products will quickly overtake those of competitors limping along selling their same old wares.

Industry News

Recent Announcements and Upcoming Events

ANSYS Announces Expanded File Transfer Support for Electronics Industry (ECAD) Users

ANSYS has formed an alliance with Bay Technology, developer of LinkCAD/ANSYS software. LinkCAD/ANSYS is a robust and powerful utility that translates file formats from GDS-II, DXF, CIF, Gerber RS-274D/X or PostScript to ANSYS file formats: ANSYS ANF, the ANSYS Workbench-compatible Parasolid XT and the ANSYS Emax-compatible TETIN. Additionally, LinkCAD/ANSYS provides ANSYS users greater flexibility in layer management. This translator automatically imports 2-D geometry and all necessary changes to the geometries required to meet the new file format. An additional benefit to users is the View feature, which displays and allows review of individual layers. This addresses a common occurrence where AutoCAD DXF files have broken polylines that appear correct when output to pen plotters but are unusable for modelers. The View feature gives users a visual display of the reported broken areas and allows them to determine if they should "merge" broken polylines or add thickness.

Samsung Electronics Selects ITI TranscenData's DEXcenter to Integrate Engineering Supply Chain

Samsung Electronics Co., Ltd. has selected ITI TranscenData's DEXcenter (Data Exchange Center), an Intranet/Internet Web application, for integrating the company's network of suppliers and departments. Following a successful pilot at its Visual Display and Printer divisions, Samsung will extend additional DEXcenters throughout all divisions. Complete deployment, scheduled for early 2003, will provide more than 5,000 users access to DEXcenter services.

ITI TranscenData has worked with Samsung Data Systems to deliver a fully integrated, Korean version of DEXcenter. SEC and SDS have made further customizations for their development environment including the ability to certify User Authentication, receive data directly from PDM systems, and integrate Enterprise Portal systems.

DEXcenter provides a single, easy to use, browser interface that enables any Samsung user to send CAD models to suppliers or other internal users even though they may be using dissimilar CAD systems. At Samsung the DEXcenter is accessible through the SEC engineering information portal and is tightly integrated with the company's corporate PLM environments. By deploying ITI TranscenData's CADIQ Model Quality assurance tools, engineering CAD models exchanged with suppliers are checked for errors that may impact downstream applications.

ANSYS Completes Acquisition of CFX

In February, ANSYS Inc. announced the completion of its acquisition of CFX, a leading supplier of computational fluid dynamics (CFD) software and services, from AEA Technology PLC for approximately $21 million in cash.

CFX brings to ANSYS state-of-the-art CFD technology with modern software architecture that is easy to use, fast and accurate and that is currently employed by more than 4,000 active users worldwide. In addition, CFX's global sales, support and consulting offices extend ANSYS' and CFX's reach to enterprises worldwide.

ANSYS Announces CFX-TurboGrid 2.1

In March, ANSYS Inc. announced the release of CFX-TurboGrid version 2.1 featuring new technology developed by recently acquired CFX, a leader in computational fluid dynamics (CFD) software and services for turbomachinery. CFX-TurboGrid 2.1 provides analysts of rotating machinery with the mesh creation tool required to increase productivity and optimize design. CFX-TurboGrid uses newly developed technology to create a high-quality hexahedral mesh while preserving the underlying geometry, allowing for accurate and fast CFD analysis. New automated functions further simplify the grid-generation step in the analysis process.

As part of the CFX system of integrated software for turbomachinery applications which includes CFX-5, CFX-TASCflow and CFX-BladeGen, earlier versions of CFX-TurboGrid serves more than 250 users worldwide in more than 150 companies. Grid generation is a key element in CFD analysis, particularly in the exacting turbomachinery field. High-quality grids are required to generate the accurate results needed to compare performance simulation of multiple designs. In industries where meeting rigorous specifications is paramount and small increases in efficiency can lead to millions of dollars profit, CFX-TurboGrid is a software tool that provides fast turnaround of the grid-generation process to ensure maximum utility from the CFD analysis process.

The latest version of CFX-TurboGrid can automatically create a customized mesh topology for the blade geometry being studied - pre-defined templates are not required. This topology is propagated to the hub and shroud allowing CFX-TurboGrid to generate a consistent high-quality hexahedral element mesh. As in previous versions, geometry can be imported using formats familiar to blade designers, and meshes can be exported for use with other products in the CFX software system, including CFX-5.

LMS International and ANSYS Inc. Expand Alliance

ANSYS Inc. announced the signing of reciprocal agreements for the cross-licensing integration and distribution of certain engineering technologies that expand the breadth and depth of the solutions they both provide for virtual product development. In the initial phase, ANSYS Inc. will develop and distribute a state-of-the-art motion simulation product built upon LMS DADS multiple-body dynamics technology, and LMS will develop and distribute a nonlinear flexible body co-simulation capability built upon ANSYS structural analysis capability. This announcement expands on an earlier arrangement to interface their respective flagship products, LMS Virtual.Lab and ANSYS Workbench, and will make it easier for customers to access complementary tools to produce more innovative products faster.

The first products of this collaboration will include LMS motion capability in ANSYS Workbench products and ANSYS structural analysis capability in LMS Virtual.Lab Motion. First products are targeted to be generally available in the second half of this year.

ANSYS AND SATYAM Forge Global Strategic Alliance, Offer Process Integrated Engineering Simulation Solutions

ANSYS Inc. and Satyam Computer Services, Ltd., a leading global provider of product design and engineering and IT services, have formed a strategic alliance that will help customers dramatically accelerate product innovation, time-to-market and quality.

This alliance will allow the two companies to jointly deliver product design solutions that combine ANSYS simulation software products—including ANSYS Workbench, an engineering simulation platform—with Satyam's engineering consulting, configuration, customization and on-site deployment services worldwide. Industries that use engineering simulation as a product development strategy, including aerospace, automotive, power generation, heavy machinery, consumer products and the electronics industries will benefit from these solutions.

Coupled with Satyam's customization and implementation strength, the ANSYS products and simulation platform will enable customers to deploy engineering simulation solutions early in the development process to make timely design decisions where they have maximum impact on product quality, cost and time to market.

ANSYS Announces Support for HP's EV7-Based AlphaServer Family
ANSYS now supports the full suite of HP's new family of AlphaServer systems. The availability of ANSYS' advanced engineering simulation solutions on HP's new high-performance servers enables users to simulate and analyze designs faster and more efficiently.

ANSYS and HP are long-standing partners, with a mutual dedication to meeting the evolving engineering simulation needs of ANSYS customers. This new generation of HP servers provides the increased performance, reliability and scalability that complements the ANSYS 7.0 release. This latest suite of engineering simulation tools from ANSYS allows companies to apply simulation and virtual prototyping throughout the product development process, from design concept to final-stage testing and validation.

Upcoming Events

PLM World 2003
April 28 - May 2
Anaheim, California

Offshore Technology Conference
May 5 - 8, 2003
Houston, Texas

All Together Now featuring ANSYS Multiphysics
May 8, 2003
Belcan Corporation
Downers Grove, IL

14th Annual Technology Trends in Automotive Engineering Symposium
May 13, 2003
Plymouth, MI

Electronic Components & Technology Conference
May 27 - 30
New Orleans, Louisiana

NAFEMS World Congress 2003
May 27 - 31
Orlando, Florida

Daratech iDPS Automotive Event
June 9 - 10
Detroit, Michigan

21st CAD-FEM Users´ Meeting
International Congress on FEM-Technology
November 12 - 14
Potsdam/Berlin - Germany

Announcements

ANSYS Welcomes New Support Distributor

ROI Engineering Inc., located in Toronto and Montreal, is the newest addition to ANSYS Inc.'s growing support distributor family. Serving Canada, ROI has developed a unique coupled analysis approach based on cycling between two databases, which includes the effect of thermal strains on cracking and the effect of cracking on heat transfer. To learn more about ROI and their services, visit www.roieng.com.

New MDM Solution Offers Benefits

Introducing Matereality, the definitive solution for material data management (MDM). Matereality represents a paradigm shift in MDM by permitting:

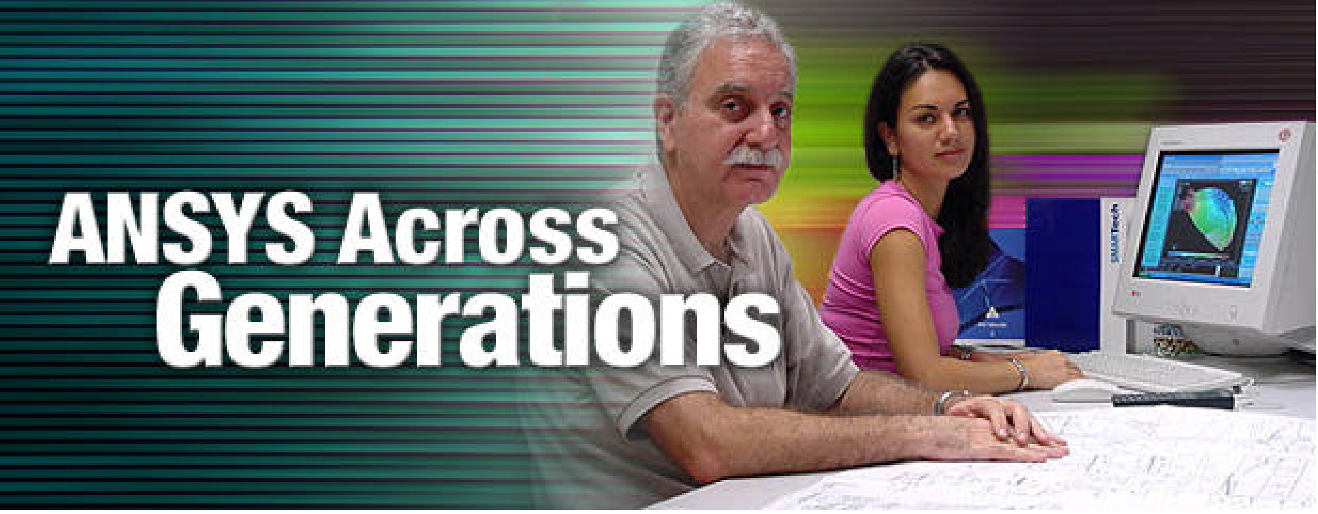
- Organization of public and private material data within a single system
- Global, internet-based secure access to data
- Owner-controlled access for sharing of private data
- Fidelity, to ensure consistent data usage across developmental platforms
- Ability to handle highly diverse material data, both simple and complex
- Traceability, the means to assess reliability and quality of data
- Diverse output formats, including CAE material model parameters
- Elimination of data mining efforts and necessity to maintain multiple MDM
- Highly scaleable and extensible, from single user access to enterprise solutions

MDM is brought to you by the familiar names of Founder, CTO, President of DatapointLabs Hubert Lobo, a pioneer in the field of material characterization for CAE. DatapointLabs is the material properties experts team supporting virtual product design engineers in their materials related issues ever since material characterization and quantification of material behavior became significant. Matereality reflects the needs of clients of DatapointLabs, packaged into a single and complete MDM solution for PLM worldwide.

Who will benefit?

Companies that work with a variety of materials and seek to securely store and share their material data across a distributed development platform of internal users and external collaborators. Material suppliers can publish or selectively distribute their data instantly across the globe. Submission of materials for certification by large OEMs becomes automatic and effortless. CAE companies can leverage Matereality to pipe-in application-appropriate data models seamlessly, enhancing ease of use and design confidence. Material testing companies such as DatapointLabs have begun using Matereality for data delivery. Matereality is thus pre-populated with design quality material properties, providing immediate high value content.

Beta Testing for Matereality is now in progress:
info@matereality.com, or register at www.matereality.com.
Click here to view the Matereality Web cast.



Father teaches daugther the advantages of ANSYS

Though launched only 33 years ago in the United States, ANSYS already has a second generation of users. In Brazil, this occurs within a family. The mechanical engineer André Beim, 51 years old, one of the first software users in the country, shares today the advantages and the success provided by ANSYS technology with his daughter Kira, a recently graduated 25-year-old naval engineer. Kira learned with her father very early on how to make analyses with ANSYS.

“She learned quickly, and errors, which took me years to find out, were noticed immediately by her,” said André, a present partner of Kira at Tresca Engenharia, Consultoria e Anlise de Tensíes, located in São Paulo.

The history of the user family started in 1977, when Kira was only a baby, and André worked for Confab, a manufacturer of equipment for oil, chemical and nuclear industries, in São Caetano do Sul, São Paulo. At that time, the calculations were made manually by using several formulas, and there was the risk of the method adopted being wrong. Seeking capabilities to manufacture more complex equipment, the company invested in ANSYS. “The computers were slow and the software did not provide product visualization or interactivity. The data input was made by punched cards,” said André.

In 1981, André Beim went to work for Ultratec, also an ANSYS user in the structural analysis area. Thirteen years later, André decided to open his own company, Tresca, thus inspiring his daughter to study Naval Engineering in 1997. Kira went to work as a trainee at her father's company, in her second year at college. “I learned to use one of the most important technologies in engineering, which made a difference in my resumé,” said Kira, a 2001 graduate of the Polytechnic School, University of São Paulo.

According to father and daughter, ANSYS had a great influence on the business success. The calculations are accurate and better evaluate the structure and the components of parts.

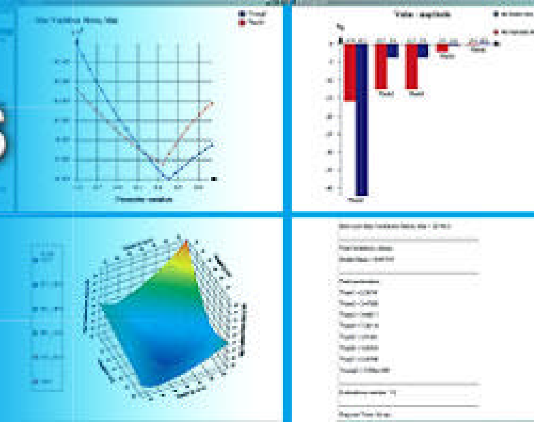
“Kira is a young entrepreneur who talks about ANSYS as if she were a long time expert,” said Andre. “The product provides the necessary solution and ensures the client that the product will work safely.”

Tresca, specializing in the analysis of equipment for the chemical, petrochemical, metallurgical and nuclear industry, has been expanding the number of clients, at present more than 30. According to father and daughter, ANSYS had a great influence on the business success. The calculations are accurate and better evaluate the structure and the components of parts. “We were able to optimize, increase the durability and even reduce the product weight,” said Kira, an ANSYS Professional user.

“Every father wishes that his child follows in his steps professionally, mainly when he is aware that the market requires professionals who provide solutions and increase competitiveness of its products,” said André, who purchased ANSYS from SMARTtech Sistemas e Serviáos, exclusive ANSYS distributor in the south region and in the State of São Paulo. Tresca continuously has been renewing the license with

SMARTtech in order to be able to profit from the constant innovations and improvements brought by the software.

A Return to Basics



Getting back to engineering fundamentals while taking advantage of the latest technologies.

By Dr. Howard Crabb
President and CEO
Interactive Computer Engineering
hccrabb@aol.com

Just as the Industrial Age completely changed the world's economy and social structure, the Information Age is doing the same. Customer requirements are driving the need to develop the highest-quality products quickly and at a reasonable cost. Product development must undergo fundamental changes to achieve leapfrog quality gains while simultaneously reducing time-to-market and cost-to produce. Training and education will play major roles in transforming companies to information-based design engineering.

We are now seeing major breakthroughs for computer-aided technologies. Personal computers have become personal workstations, as the continued blurring of the microprocessor price/performance ratios make PCs and engineering workstations one in the same. Only a few years ago, high-end engineering workstations cost about $55,000. Today, a good high-end PC goes for about $6,000. Continued breakthroughs in microprocessor and scalable parallel processing technology are moving super-computer processing to the desktop. Engineers can now perform complex, accurate calculations and simulations at their desktops without worrying about computer issues such as cost. They can thus direct their full attention to solving the problem rather than spend time selling it.

High quality at a reasonable cost is the primary expectation of all customers of engineered products. This requires a fundamental change in the product development process to exploit new and emerging enablers for competitive differentiation. Simultaneous engineering must become the standard workplace environment, and continuous improvement must be inherent to the overall process. This is not now the case in most companies.

Technology enablers improving product development are solid modeling, predictive engineering, rapid prototyping, 3-D printing, and visual engineering. Taken as a spectrum of solutions, these technology enablers fundamentally change product development. Concepts requiring months or years to develop are now done in hours or days. Engineering tools requiring large investments per design engineer ($75,000 to $100,000) are now under $20,000. With order-of-magnitude improvements in speed and performance, Moore's law that doubles the number of transistors per IC every couple of years will continue for a least another decade. The business model for capital acquisition is no longer valid.

A generation ago, engineers solved problems with drafting boards, slide rules and Mark's Engineering Handbook. Knowledge delivery consisted of reference books piled next to large drawing tables. Most new products were derived from previous designs. Upon completion, a new drawing defined the concept.

Production part suppliers were consulted early and often. Key manufacturing equipment suppliers participated early in product design. Engineering and knowledge reuse were not buzzwords but rather an actual way of doing business. Purchasing was an active partner serving the engineer and supporting the process.

CAD/CAM changed product development from engineering a concept to stressing packaging and building prototypes to validate the design. Emphasis was placed on designers and their ability to develop alternatives that met the CAD requirements. Emphasis shifted from engineering the product for manufacturing to using existing drawings to modify designs to fit in the design space. If the design did not meet objectives after physical prototypes were built and tested, the engineers were brought back to analyze the design. Such analysis required long lead times and expensive calculations using high-powered super-computers. The total process required months or years and was extremely costly. The designer and test engineer drove the process, and product/manufacturing engineers were moved to the latter design stages. Today, computer equipment costs have dropped by orders of magnitude, allowing the process to fundamentally change while allowing for a return to basics.

Product and manufacturing engineers must have the ability at the concept stage of design to simulate the product's intended environment, both as it is being manufactured and as it is operating in service. This ability is called "predictive engineering" and allows simulation to drive the process. With exact data representation via solid modeling, the product/manufacturing engineer can present the concept and its results in the design/manufacturing environment within hours or days rather than months or years.

Where computer-based technologies accelerate the pace of change, companies must return to the basics of people, process, and technology, and apply them systematically to improve engineered products. Many companies have good people, workable processes, and excellent technologies. The key is to step back and take a systems engineering approach to improve the business using these building blocks.

The best way to do this is through a total systems approach where products are designed the way they used to be: by leveraging an engineer's know-how and accounting for a broad range of requirements from throughout the organization including manufacturing, marketing and customer support. The new twist now is that best-in-class companies can implement state-of-the art CAE technologies to develop and verify the highest quality product concepts the first time — before physical prototypes are ever built.

Dr. Howard Crabb spent more than 30 years at Ford Motor Company, where he led initiatives to implement solid modeling and predictive engineering performed at the concept level of design. He is author of the book The Virtual Engineer, from which this column is excerpted and abstracted, and is currently president and CEO of the consulting firm Interactive Computer Engineering which assists clients across a range of manufacturing industries in improving product development concepts and processes to strengthen their competitive positions.

Working with Multiple Toolbars

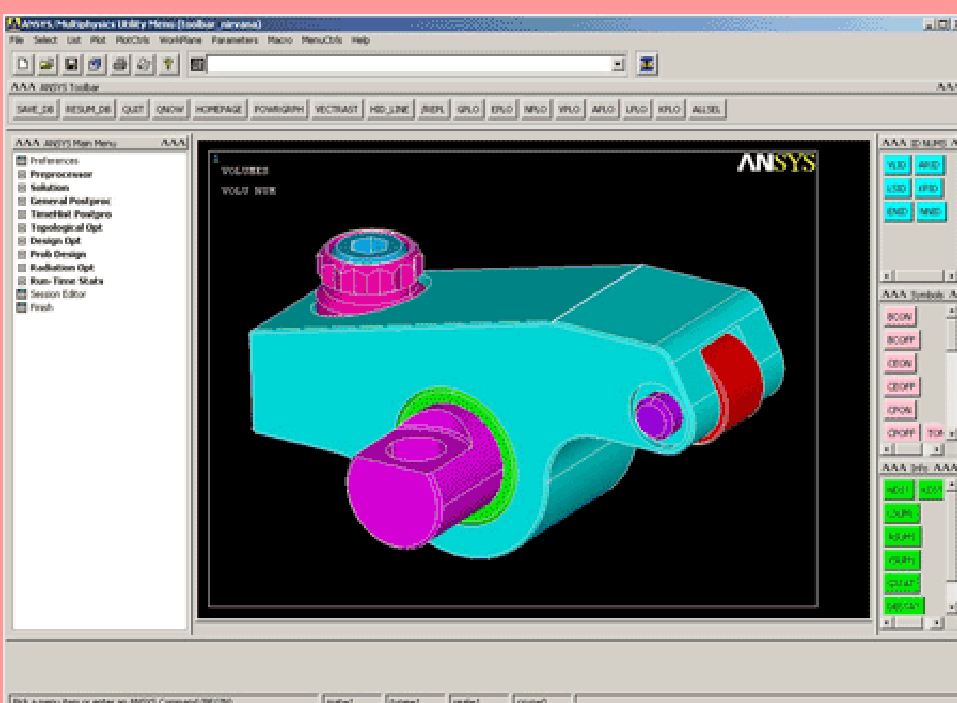
ANSYS 7.1 will let you create multiple toolbars and place them almost anywhere.

By John Crawford
Consulting Analyst

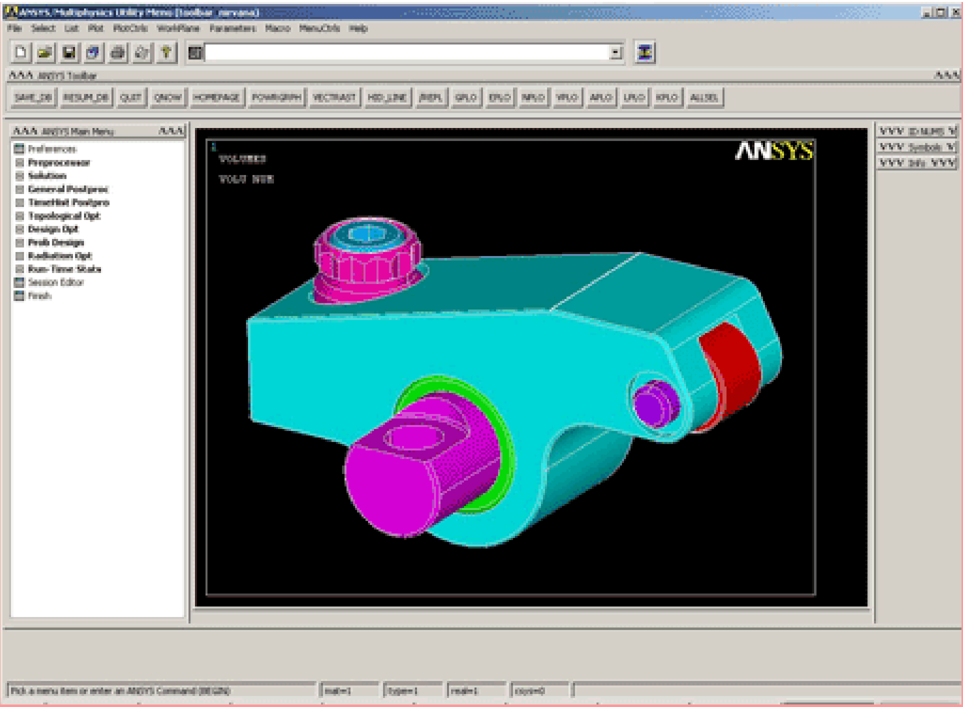
ANSYS 7.1 will have an interface that allows you place a toolbar just about anywhere you want, to the left of the graphics window or to the right, above it or below it, or any or all of these. You can even put more than one toolbar in any of these locations. The possibilities are endless.

These new features have their roots in a graphical user interface introduced in ANSYS 6.1, the first change in how ANSYS looked since version 5.1 hit the streets in 1994. Not only does it feature a single window for all the graphics and menu functions, it also enables the user to have much more control over the appearance and functionality of the interface itself.

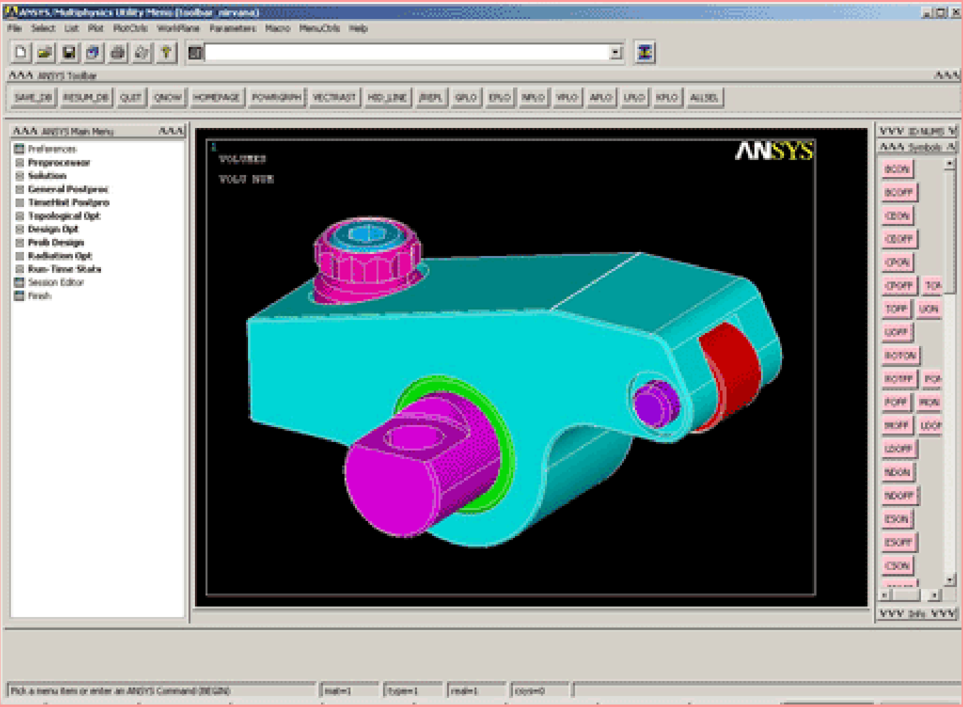
One of the benefits of the new interface is the ability to employ multiple toolbars. To illustrate this, let's take a look at how we would replace the standard ANSYS toolbar with our own, and also place three toolbars on the right side of the graphics window.



Our customized default toolbar is shown above the graphics window, and our additional three toolbars are on the right side of the graphics window. The buttons in each toolbar have been given a common color for ease of identification.



Each of the custom toolbars on the right side of the graphics window can be minimized in the same manner that the default toolbar can be minimized.



If we have a specific toolbar we want to use we simply expand it. The other toolbars remain minimized.

For this example we will be working on a Windows operating system, but the situation is nearly the same for UNIX. We need to download 8 files from the *ANSYS Solutions* website, and make two directories.

Directories

- toolbars – this directory will contain the toolbar files and will be made in your home directory
- macs – this directory will contain a macro and will be made in your home directory

Files

- start71.ans - ANSYS start file that is placed this in your home directory
- tlbrlist71.ans - this is a list of toolbars and is placed in your home directory
- MYABBR.TLB - place this in the toolbars directory
- SYMBOLS.ABBR - contains the abbreviations for the SYMBOLS toolbar
- NUMIDS.ABBR - contains the abbreviations for entity ID numbers
- INFO.ABBR - contains the abbreviations for information items
- toolbar - this is placed in the toolbars directory
- numtog.mac - a macro that toggles entity ID numbers, and is placed in your macro directory.

All files can be found at *****place web address for files here*****

1. Begin by making two directories in your home directory called toolbars and macs.
2. Next, copy the files start71.ans and tlbrlist71.ans to your home directory. Each file contains commands that ANSYS uses when starting. start71.ans has a /PSEARCH command that tells ANSYS to look for unknown commands in your macs directory. It also has an ABBRES command that loads the toolbar abbreviations stored in toolbar. tlbrlist71.ans has a line added to it that tells ANSYS that we want to add the toolbars described in MYABBR.TLB to the graphical user interface.
3. Copy numtog.mac to the macs directory. We will use this to toggle ID numbers and other symbols off and on.
4. Copy toolbar to the toolbars directory. This is the toolbar that we'll use to replace the standard ANSYS toolbar.
5. Copy MYABBR.TLB to the toolbars directory. This is the file that tells ANSYS where to place our extra toolbars, and where it can locate the files that contain the abbreviations used in each toolbar.
6. Copy INFO.ABBR, SYMBOLS.ABBR, and NUMIDS.ABBR to the toolbar directory. These files contain the abbreviations used in each toolbar.

Now we can start ANSYS and see how our new toolbars look. They appear on the right side of the graphics window. By editing their definition in MYABBR.TLB we could have them appear in any of the four toolbar/menu zones that surround the graphics window.

Each toolbar can be minimized and expanded by picking its title. This is handy if you have several toolbars. It also allows you to have a number of toolbars available for use when you need them, and hidden when you don't need them.

The power in the toolbar capability of ANSYS 7.1 is significant. The user can create toolbars for preprocessing, postprocessing, geometry operations, activating symbols, and lots more. In the extreme, it would be possible to construct your own menu system using toolbars. After being limited to one toolbar for such a long time, it is a pleasant surprise to have the ability to generate any number of toolbars. I can see that I'll be busy for a while creating toolbars for the commands I use most frequently.

Thanks to Eric Clevenger, Senior Member Technical Staff of the ANSYS GUI Development Department, for information and insight used in preparing this column.

Accounting for Damping in ANSYS

Various forms of damping are available for different types of analysis

By ANSYS Inc. Technical Supports
ANSYS Inc.

Q: How is damping specified in ANSYS?

A: It is first important to note how damping is defined in the *full method* compared with the *mode superposition method*. Because the former deals with nodal coordinates and the latter with generalized coordinates, accounting for damping is different for the two.

In the *full method* of modal(1), harmonic, or transient analyses, the equation of motion is:

$$\{F\} = [M]\{\ddot{x}\} + [C]\{\dot{x}\} + [K]\{x\}$$

The damping matrix [C] is formed from the following components(2):

$$[C] = \alpha[M] + \beta[K] + \left(\frac{\xi}{\pi f}\right)[K] + \sum_{j=1}^{M}\beta_j[K_j] + \sum_{k=1}^{N}[C_k]$$

• a is the constant mass matrix multiplier for alpha damping (ALPHAD command)
• b is the constant stiffness matrix multiplier for beta damping (BETAD command)
• x is the constant damping ratio, and f is the current frequency (DMPRAT command)(3)
• bj is the constant stiffness matrix multiplier for material j (MP,DAMP command)
• [Ck] is the element damping matrix for supported element types (ET and TYPE commands)

On the other hand, in the *mode superposition method* for harmonic, transient, or spectrum analyses, the equation solved for is:

$$\{\phi_i\}^T\{F\} = \{\ddot{y}_i\} + 2\omega_i\xi_i\{\dot{y}_i\} + \omega_i^2\{y_i\}$$

Instead of creating a damping matrix [C], an effective damping ratio xi d is created *for each mode i* (4):

$$\xi_i^d = \left(\frac{\alpha}{2\omega_i}\right) + \left(\frac{\beta\omega_i}{2}\right) + \xi + \xi_{mi} + \frac{\sum_{j=1}^{M} \xi_j E_j^s}{\sum_{j=1}^{M} E_j^s}$$

• a is the inversely-related damping parameter for alpha damping (ALPHAD command)(5)
• b is the linearly increasing damping parameter for beta damping (BETAD command)
• x is the constant damping ratio (DMPRAT command)
• xmi is the damping ratio specified for mode i (MDAMP command)
• xj is the damping ratio specified for material j (MP,DAMP command)(6)
• Ejs is the strain energy for material j, calculated by ANSYS as _{fj}T[Kj]{fj}

For spectrum analyses, damping is included not in the calculation of mode coefficients but in mode combination only. Also, in the case of mode-superposition method, material-dependent damping is added in the expansion of modes, so the user *must* include material-dependent damping (MP,DAMP) *and* request element stress calculations (MXPAND) before running the modal analysis.

The accompanying Table "Damping for Different Analysis Types" (same as Table 5.5 of the *ANSYS 7.0 Structural Analysis Guide*) provides a summary of when different forms are damping are available for different analysis types.

Mode superposition methods support the use of QRDAMP, but the user should know that although it is a mode-superposition method, damping is included in the modal analysis phase (QRDAMP), so the *full method damping equation [C]* above should be used to determine how damping is accounted for. As mentioned in footnote 6 of the Table, if you use the QR damped mode extraction method [MODOPT,QRDAMP], and you specify any kind of damping during preprocessing or in the modal analysis, ANSYS ignores damping specified during the mode superposition analysis.

It is extremely important to note that MP,DAMP means different things, depending on the analysis method used.

In the *full method*, material-dependent damping values represent a *stiffness matrix multiplier* for that material, similar to *viscous damping* (linearly-dependent on frequency) but per material. Hence, in this case, the value for MP,DAMP will be equal to x/p·f or to c/k for a single DOF system. When multiple materials are present, the damping matrix [C] simply applies each value of bj to the portion of the stiffness matrix associated with that given material j:

$$[C]_{MP,DAMP} = \sum_{j=1}^{M} \beta_j [K_j]$$

In the mode-superposition method, however, material-dependent damping values indicate the damping ratio for that material, similar to structural damping (independent of frequency). This means that the value supplied via MP,DAMP will be equal to x for a single DOF system. When multiple materials are present, the Modal Strain Energy Method (MSE) is used to calculate an 'effective' damping ratio for the system, as shown below:

$$\xi_i^{MP,DAMP} = \frac{\sum_{j=1}^{M} \xi_j E_j^s}{\sum_{j=1}^{M} E_j^s}$$

This means that an ‘effective’ constant material-dependent damping ratio is calculated for all modes.

Footnotes
1 This is the DAMP or QRDAMP eigenvalue extraction methods.
2 This is similar to Equation 15.19 of the ANSYS 7.0 Theory Manual.
3 This term is not available in damped modal or full transient analyses.
4 This is similar to Equation 15.94 and 17.91 of the ANSYS 7.0 Theory Manual.
5 No alpha damping permitted for certain types of spectrum analysis (SPRS, MPRS, DDAM)
6 No material-dependent damping ratio is permitted for certain types of spectrum analysis (PSD)

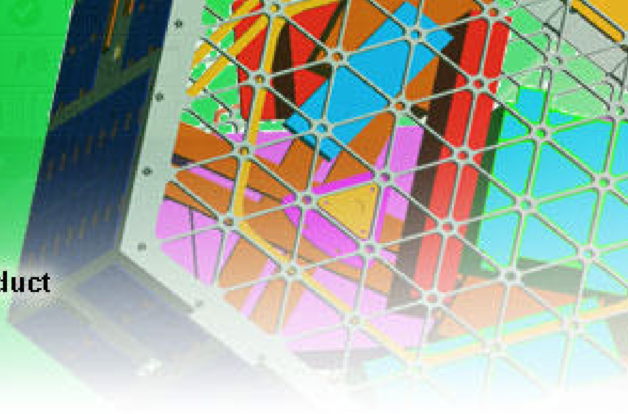
Table: Damping for Different Analysis Types

Analysis Type	Alpha, Beta Damping [ALPHAD, BETAD]	Material-Dependent Damping [MP, DAMP]	Constant Damping Ration [DMPRAT]	Modal Damping [MDAMP]	Element Damping [3] (COMBIN7, and so on)
Static	N/A	N/A	N/A	N/A	N/A
Modal					
Undamped	No[5]	No[5]	No[5]	No	No
Damped	Yes	Yes	Yes	No	Yes
Harmonic					
Full	Yes	Yes	Yes	No	Yes
Reduced	Yes	Yes	Yes	No	Yes
Mode Sup	Yes[6]	Yes[4,6]	Yes[7]	Yes[7]	Yes[6]
Transient					
Full	Yes	Yes	No	No	Yes
Reduced	Yes	Yes	No	No	Yes
Mode Sup	Yes[6]	Yes[4,6]	Yes[7]	Yes[7]	Yes[6]
Spectrum					
SPRS, MPRS[2]	Yes[1]	Yes	Yes	Yes	No
DDAM[2]	Yes[1]	Yes	Yes	Yes	No
PSD[2]	Yes	No	Yes	Yes	No
Buckling	N/A	N/A	N/A	N/A	N/A
Substructure	Yes	Yes	No	No	Yes

Table Notations
N/A Not applicable
1. _ damping only, no _ damping
2. Damping is used only for mode combination and not for computation of mode coefficients
3. Includes super element damping matrix
4. If converted to modal damping by expansion of modes
5. If specified, an effective damping ratio is calculated for subsequent spectrum analyses
6. If you use the QR damped mode extraction method [**MODOPT**,QRDAMP], and you specify any kind of damping during pre-processing or in the modal analysis, ANSYS ignores damping specified during the mode superposition analysis

7. If you use the QR damped mode extraction method [**MODOPT**,QRDAMP], **DMPRAT** and **MDAMP** are not supported

The New CAD



The integration of simulation into product development will forever change computer-aided design.

By Joel Orr
President and Principal Consultant
Orr Associates International

Design is mystical. Something does not exist...then it does. Design is not deterministic. Given a set of requirements, there is no formula into which they can be pumped to yield an optimum design, or even a good one. The process of design is invariably one of trial and error.

Fascinatingly, and counterintuitively, the process offers no promise of convergence. That is, there is no guarantee that if we perform more trials, we will somehow get closer to an optimum design. So great designs are all the more impressive for that.

For centuries, the only improvements in the design process were in documentation: stable media, better pencils and pens, mechanical drafting aids, electric erasers, eradicating fluid, pin registration, and copy machines. In his 1950 science fiction story, *The Door Into Summer*, Robert Heinlein described “Drafting Dan,” a keyboard-controlled drawing robot whose essentials began to be realized in the early CAD systems of the 1960s.

Then came 3-D, photorealism, kinetic motion — and design documentation acquired more and more verisimilitude at the pace of Moore’s Law, by which computing power increases ten-fold every three to four years.

Design can be characterized as dialectic, a term used by the philosopher Hegel in his attempt to explain the spasmodic rhythms of the unfolding of history. While it may not work well in history, it does work for design. In the design dialectic, the “thesis” is the concept in the mind of the designer. The “antithesis” is the external manifestation of the design — drawing, model, or prototype. Then the designer brings the thesis to bear on the antithesis to yield the third stage of the dialectic: *synthesis*. This is usually a refined model or prototype, which becomes the “thesis” for the next cycle.

And although there is no guarantee of convergence, designers need to iterate — to go through multiple cycles of the dialectic — to refine the design.

That’s what makes the computer a great design environment. It is a place where detailed and accurate models can be built quickly and inexpensively, and quickly tested to reveal their inadequacies, thus providing the basis for another iteration.

But for the design environment to be truly wonderful, it must be capable of as complete and precise a model as can be achieved — and as complete an environment as possible in which the model can be embedded. For example, the behavior of a product can best be modeled in a “world” in which “gravity” and “friction” are provided by the computer.

Simulation models can be produced either by analysis or by synthesis. Analysis is the hard way, where you determine precise mathematical equations to represent the functioning of all parts of the model. Since many behaviors are difficult to represent in this way, building such a model can prove impractical or impossible.

The synthetic approach makes no assumptions about underlying mechanisms. Instead, it simply seeks to produce a “converter” — an equation or a mechanism — that imitates reality. Given the same set of inputs, it produces outputs similar to those of the actual system being modeled.

This behavioristic approach is at the root of the powerful therapy called “neurolinguistic programming,” or NLP. Ideally, practitioners are taught to observe behaviors, and not to attempt explanations. The fact that a person makes certain behavioral choices does not “mean” anything, except perhaps that they are “stuck,” and need to know about other choices available to them. No assumptions are to be made about what people think or feel.

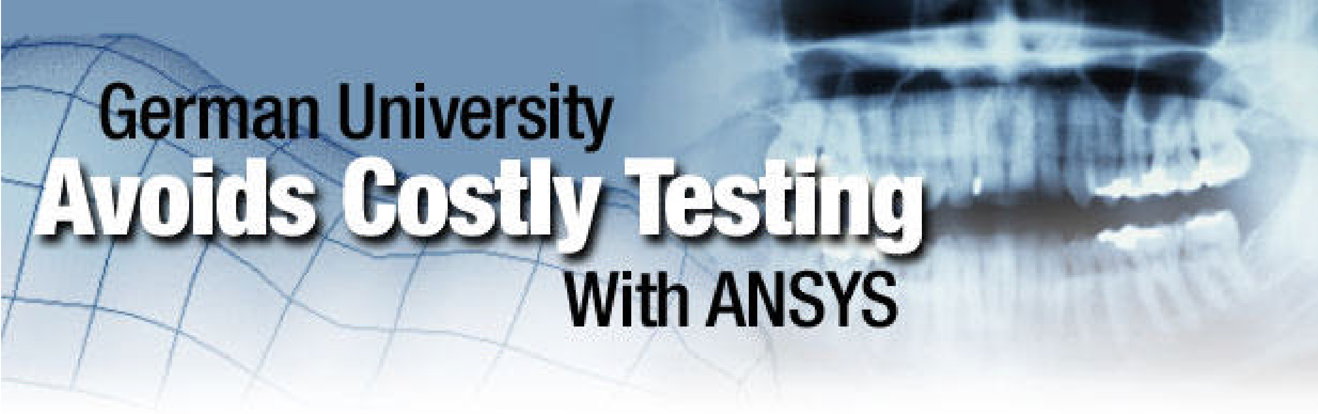
“End” has multiple meanings.
It can mean the *terminus*, the final point; it can also mean the *goal.*

Similarly, we can create simulation models that behave like the things we are modeling, without knowing (or caring) whether the mechanisms that convert the inputs to the outputs are in any way the same. For example, the famous “flocking” algorithm published by Craig W. Reynolds in 1987 models the behavior of flocks of birds, schools of fish, and other groups in motion, by applying simple rules to the behavior of each member of the simulated group. The visual effect is astonishingly realistic, and we do not need to know or care if birds or fish use a similar set of rules to govern their behavior when they move in groups.

“End” has multiple meanings. It can mean the *terminus*, the final point; it can also mean the *goal*. In the title of this commentary, I used it in both senses. I believe that the terminus of all CAD will ultimately be the creation of simulation models, and also that the *goal* of computer-aided design is in fact simulation — that simulation is the highest form of computer-aided design.

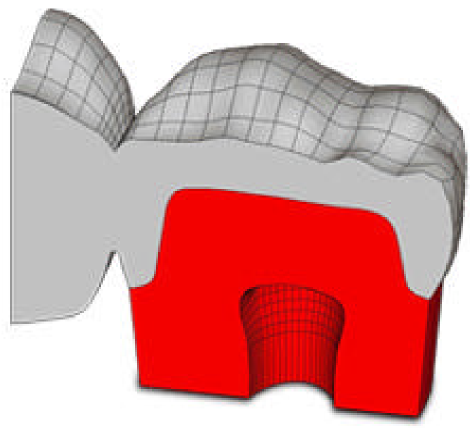
Simulation is still a specialty for the most part, but it ought not to be. Anyone involved in thinking about products should be able to “see them work” on the screen, where mistakes are easy and inexpensive to find and fix. We are not quite there yet. But with the rapid pace of evolution in engineering software, we soon will be, and simulation then will be truly integrated into the design process.

Dr. Joel Orr is President and Principal Consultant of Orr Associates International and also serves as Vice President and Chief Visionary of the publishing and consulting firm Cyon Research. He is a recognized authority in the engineering software industry and many Fortune *500 companies use his services and perspective in the areas of business restructuring, engineering automation, and the future of technology. Orr speaks frequently at conferences and has authored several books; the most recent of which is* The Victorious Engineer*.*



German University Avoids Costly Testing With ANSYS

Introduction



The role of computer-aided engineering in dental applications allows biomedical engineers to improve design structures so that closely mimic real teeth and still stand up to the daily stresses of the mouth more efficiently.

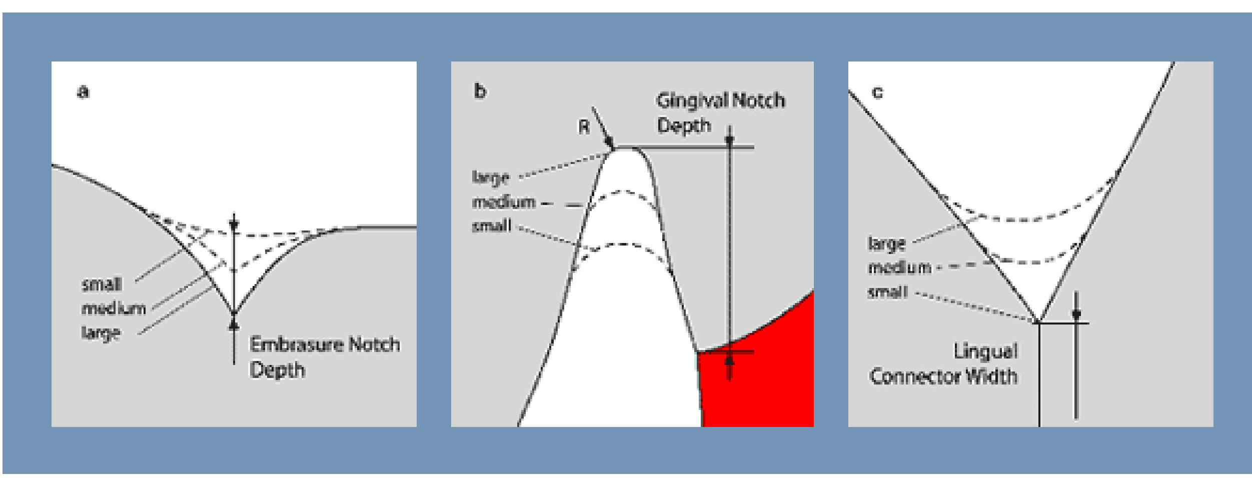
Nils Götzen, an engineer working in the Biomechanics Section at the Technical University of Hamburg-Harburg, Germany, has recently made a breakthrough in the development of better all-ceramic bridges using the combination of ANSYS software, along with a special ceramic program called CARES/Life developed at the NASA Glenn Research Center (Cleveland, Ohio, U.S.A.).

“I chose ANSYS software for my work because of its compact form, ease-of-use, ability to interact with other software, the variety of solvers, and most importantly, its linear elastic analyses capabilities which are most needed for dental ceramic studies,” said Götzen.

Challenge

Traditionally, dental ceramics have been used for aesthetic purposes only. Bridges have long been constructed from metallic core structures with a thin layer of ceramic fused to the metal. While these structures are very strong, the manufacturing process can be problematic, and due to its metal core, residual thermal stresses exist.

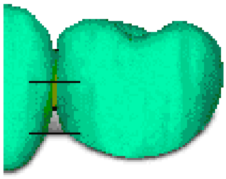
An all-ceramic bridge would provide an easier manufacturing process, a more aesthetically superior finished product, and no residual thermal stress. However, ceramic bridges lack the high strength of their metallic core structure counterparts—an obstacle that must be overcome before they can replace metallic bridges. “All-ceramic bridges represent a new generation of dental ceramics,” said Götzen. “The goal of my work is to extend the lifetime and lower the failure probability of these bridges.”



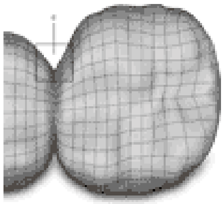
Solution

To create an all-ceramic, three-unit bridge that works both in the anterior and posterior regions of the mouth, Götzen needed to focus on why failures occur and where they most often occur. The bridges are constructed of two retainers, which are connected to ground down teeth on each side of the missing tooth, and the pontic, which replaces the missing tooth. The assumption was that the connector area between each of the retainers and the pontic is where the majority of failures occur because they have the smallest cross-section and act like stress raisers due to their notch shape.

A three-dimensional, three-unit geometrical model was created in ANSYS. The bridge connector was meshed particularly fine to ensure adequate stress resolution. Ceramic and dentin were modeled as linear elastic materials. For his work, Götzen concentrated on specific geometric attributes of the bridge connector and developed three “high impact” design features. The first is the “embrasure notch depth” or the distance from a virtually smooth line, connecting the marginal ridges of two adjacent teeth, to the lowest point of the upper connector edge. The second is the “gingival notch depth” or the distance from the marginal height (dentin-ceramic interface) to the highest point of the lower connector edge. The third design feature is the “connector width” or the distance from the buccal (side of the cheek) to the lingual (side of the tongue) surface of the connector. Varying these three design features resulted in 27 different finite element models of the connector areas to analyze.



Linear elastic stress analyses were conducted on all of Götzen's bridge designs in ANSYS to determine stress distribution. Each model was submitted to a simulated biting force of 100 N, applied directly at a single vertical point in the valley of the biting surface of the pontic (middle tooth). All analyses were conducted on a Silicon Graphics 02 workstation.



Once the linear elastic stress analyses were completed, Götzen turned to the NASA developed program known as CARES/Life or Ceramic Analysis and Reliability Evaluation of Structures. CARES/Life determines fracture statistics and predicts the fast fracture reliability and the lifetime of isotropic ceramic components. Götzen fed input data from his ANSYS stress analyses into CARES/Life to develop reliability estimations. CARES/Life uses material parameters and load factors to determine the failure/survival probability for each material component separately and also determines their combined values.

The combined ANSYS and CARES/Life analyses confirmed that the peak stresses are always located directly at the bridge connector. Götzen's work went even further, proving that the gingival notch depth, considered one of his “high impact” design features, has the most crucial impact on the structural reliability of all-ceramic bridges. In some cases a small shift in the size of the notch can increase the bridge's failure probability by more than 1,000 percent. No other area of the bridge showed this much of an impact on failure.

Benefit

Using the combination of ANSYS and CARES/Life, Götzen has unveiled a new and powerful tool in the dental research industry. “The combination of traditional finite element analysis with ceramic-specific post-processing has enabled us to identify critical design features for all-ceramic bridges,” said Götzen. “I am very pleased with the results that I have obtained and believe that the data developed from these analyses will lead to breakthroughs in the development of all-ceramic bridges and eventually the optimization of their reliability.”

Götzen, now working on his Ph.D. in Bone Mechanics in Hamburg, has studied and worked in the field of aerospace and biomedical engineering in both Germany and the United States for the past eight years.

Visit the ANSYS Biomedical Spotlight.