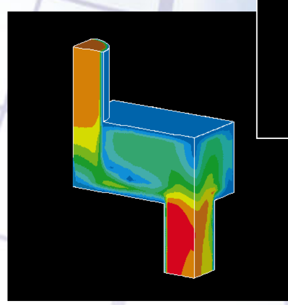
Introducing ANSYS 5.3: Technology Innovation and Leadership

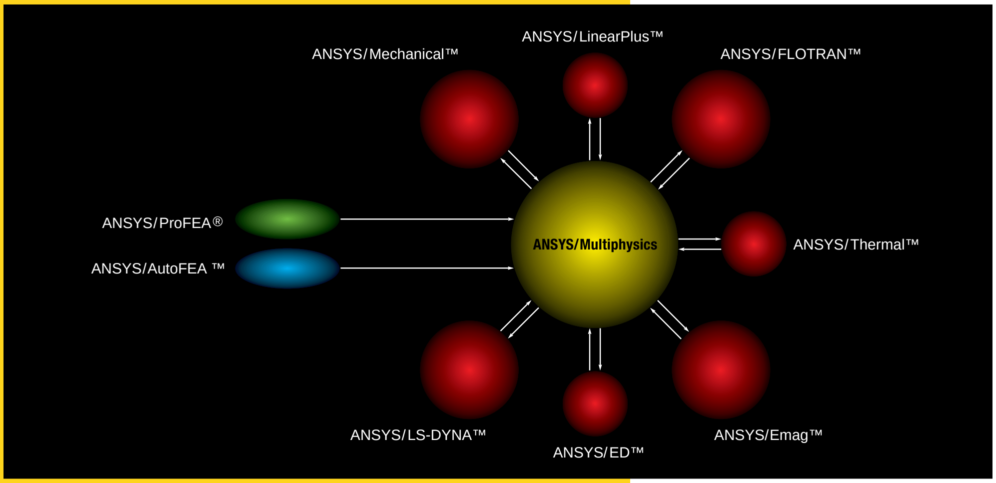


ANSYS News

Second Issue 1996

INSIDE THIS ISSUE

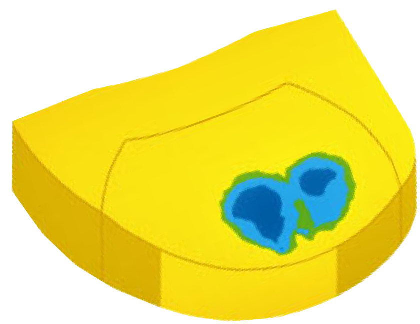
Behind it all, there's ANSYS®



ANSYS products help reduce the time and cost of development as well as improve product design and quality. The family of ANSYS products ranges from ANSYS/AutoFEA for design optimization to ANSYS/Multiphysics for advanced design verification.

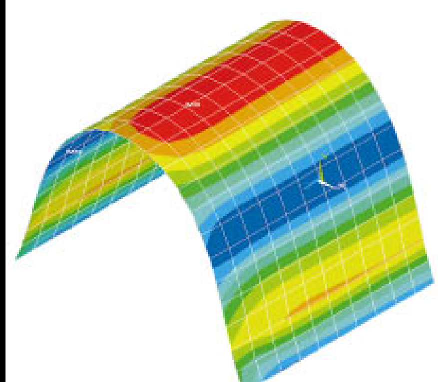
Biomedical

DePuy Inc. engineers use ANSYS technology to develop reliable and durable medical implants and the surgical instruments used to install them. The software also allows DePuy to shorten its new product development cycle by reducing the number of prototypes required for testing.



Marine Habitats

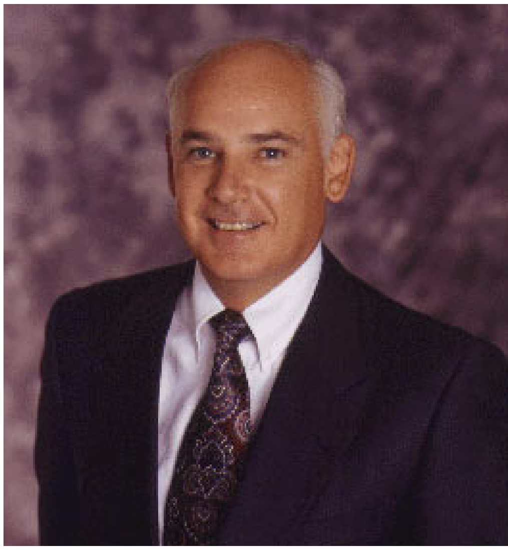
Reynolds Polymer is a global provider of aquarium systems and acrylic components for undersea habitats and marine parks. Reynolds engineers build attractive and interactive exhibits using developments in glass and acrylics, combined with the use of ANSYS analysis software to ensure product safety and cost effectiveness.



The ANSYS Approach:

Transforming the Vision into Reality

ANSYS, Inc. commences 1996 as a leading growth company in design analysis software, following strong 1995 revenues. The first quarter of 1996 shows a continuance of this successful trend.



Peter J. Smith
Chairman and CEO

We thank our customers around the world for this progress. With your guidance, we have implemented a long-term, integrated product development strategy that provides flexible engineering solutions. Our continuing strong investments in technology and support services help us meet your ever-increasing needs to solve more complex product design problems, as well as integrate analysis tools with leading computer-aided design (CAD) systems to optimize and evaluate products much earlier in your product development cycle.

Companies are using the ANSYS program to meet a wide range of requirements from accelerating time-to-market, to improving the quality of life. For example, the biomedical industry now simulates pre-operative surgery using sophisticated computer-aided engineering (CAE) technology traditionally used in the aerospace and automotive industries. (See related article on page 22.)

For Motorola, Inc., a world-class company that manufactures tens of thousands of products, the ANSYS program is a critical element of their integrated product development strategy in which everyone works together as a team, with analysis occuring throughout the product development cycle in a corporate-wide effort to reduce time-to-market. Mechanical engineers at Motorola believe ANSYS is an important tool for working collaboratively to lower costs, improve product quality, and bring products to market more quickly.

ANSYS, Inc. continues to reinforce the message that analysis and simulation tools are strategic, mission-critical applications for implementing successful, enterprise-wide engineering systems. We are releasing the most powerful version yet of the ANSYS program in June. ANSYS 5.3 will deliver breakthrough solver technology and enhanced multiphysics capabilities. (See related article on page 3.)

We are transforming our vision of enterprise-wide engineering into reality. This is evidenced by positive feedback from customers and by the company's strong growth as we continue to build momentum in the market.

It's a privilege to be part of a company that develops leading-edge design and analysis software that meets corporate-wide engineering needs through unified efforts and feedback from our valued customers and support distributors. I look forward to your continued guidance on our technical and business directions in 1996.

Sincerely,

Peter J. Smith
Chairman and CEO

ANSYS® News®

Second Issue 1996

***ANSYS News* is now available on the World Wide Web Site: http://www.ansys.com**

Published four times a year for ANSYS customers and others interested in the field of design analysis applications. There is no charge for a subscription; contact Mary Al Caldwell at 412.873.3063 to be put on the mailing list or to change your address. Although the contents of *ANSYS News* have been carefully reviewed, ANSYS, Inc. does not warrant it to be free of errors or omissions.

Managing Editor
Jen Valachovic

Design/Production Manager
Cathy L. Cimino

Copy Editor
Daniel Parrish

Editorial Advisors
Sue Batt
Ray Browell
Scott Owens
John Twerdok

Technical Advisors
Sue Batt
Ray Browell
Laura Carrabine
Mark Imgrund
Peter Kohnke
Joe Manich
Scott Owens

Contributing Writers
Sue Batt
Po-Jen Cheng
Wilson Sporting Goods
Michael Harms
3M Corporation
John Krouse
Industry Analyst
Frank Marx
Christopher G. Mollo
Automated Analysis Corporation
Daniel Parrish
Caren Potter
Freelance Writer
J. M. Smith
Structures and Computers Ltd.
Tim Trainer
W. Ben Tsoi
Automated Analysis Corp.
Jen Valachovic

Introducing ANSYS 5.3:

Technology Innovation and Leadership

ANSYS, Inc. continues the tradition of delivering innovative, leading-edge technology that addresses corporate-wide engineering needs with the introduction of ANSYS 5.3. This release features an unprecedented suite of breakthrough solver technology, new meshing and graphics capabilities, and enhancements to ANSYS/FLOTRAN and ANSYS/Emag. ANSYS, Inc., with the broadest coupled-field multiphysics analysis program available in the industry today, continues to strengthen the product line, ensuring that users have powerful and flexible design analysis solutions.

The Explicit Solver

ANSYS 5.3 contains a new explicit solution product option, ANSYS/LS-DYNA, for solving highly nonlinear structural dynamic problems. The product consists of customized ANSYS pre and postprocessing with LS-DYNA3D for solution. This 3D explicit solver, developed by Livermore Software Technology Corporation, leads the industry in simulating metal forming, crash analyses, impact involving large deformation, nonlinear material behavior, and multi-body contact typically characterized by short transient/impact behavior.

ANSYS/LS-DYNA solves highly nonlinear dynamic problems robustly and efficiently without the need to factorize a stiffness matrix. Solutions can be achieved in a fraction of the computer time normally required by nonlinear implicit solution techniques. Several examples of ANSYS/LS-DYNA capabilities are shown in Figures 1-3C.

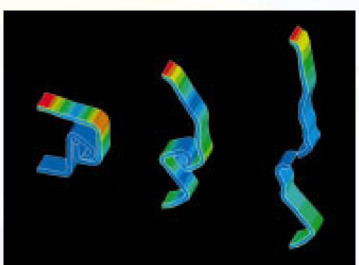


Figure 1

Simulation of an automotive seat rail pullout.

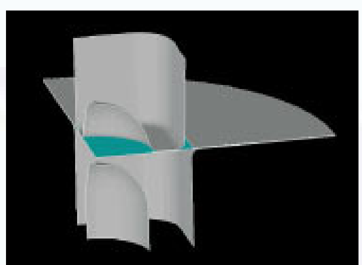


Figure 2A

Model prior to drawing operation.

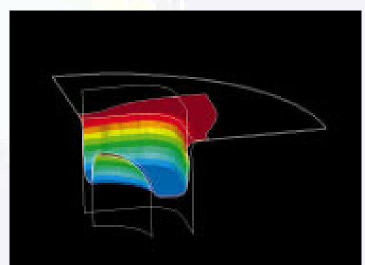


Figure 2B

Deep drawing of an aluminum sheet.

Figure 3A

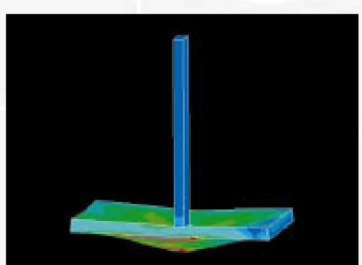


Figure 3B

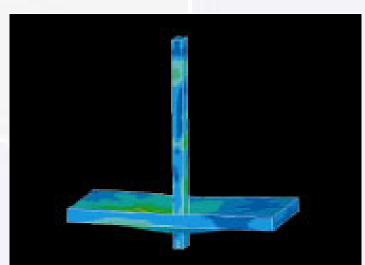


Figure 3C

Projectile penetrating a solid bar.

The Fast Linear Iterative Solver

The fast version of the PowerSolver, as described in the first issue of ANSYS News, 1996, “Pure Power: An Inside Look at the Engine Driving ANSYS”, runs approximately 50 percent faster and uses a fraction of the disk space of the original PowerSolver, a Preconditioned Conjugate Gradient (PCG) solver, at ANSYS 5.2. Problems, such as linear static, modal, transient thermal, and linear transient dynamic structural analyses, can be solved in 1/20 the time using a small fraction of the disk requirements when compared to the frontal solver (Table 3).

Comparisons to existing finite element analysis (FEA) software with iterative solvers indicate that the ANSYS 5.3 fast linear iterative solver is comparable for medium size problems and faster for larger problems. This is accomplished by closed-form integration of the element stiffness matrices, eliminating the need for several of the element disk files, and reducing the results data to basic items, such as stresses, strains, and temperatures.

The Block Lanczos Eigensolver

The new Block Lanczos Eigensolver is a welcome addition to the family of modal solvers which includes reduced, subspace, unsymmetric, and damped. Block Lanczos is a linear eigensolver that has a sturm sequence check, like subspace, to verify that all frequencies in a range are obtained. Computationally, it solves medium-size to very large problems in a fraction of the time required by the subspace method. In the comparison of an automotive radiator model supplied by a customer (Figure 4) consisting of 100,000 degrees of freedom, Block Lanczos solved the problem in 1/25 the time of the subspace solver, and required about 1/3 the disk requirements (Table 1). Block Lanczos handles all classes of elements with very competitive performance (Table 2).

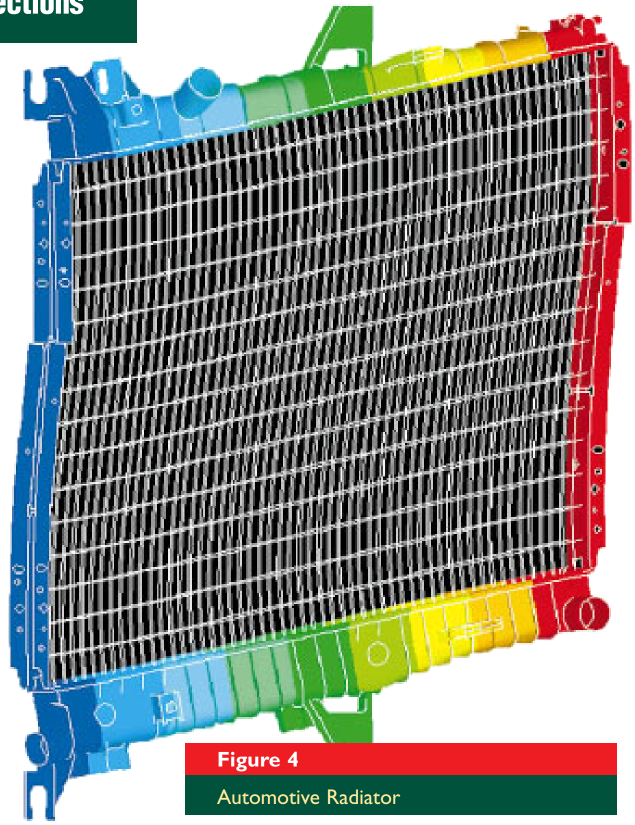


Figure 4
Automotive Radiator

Users will also see significant time and disk savings due to spectrum enhancements brought about by the new, closed-form integration for power spectral density (PSD) analysis. Users will get reliable results and can be assured that the analysis will converge.

The PowerDynamics Iterative Eigensolver

PowerDynamics is a linear eigensolver that is an extension of the PowerSolver. PowerDynamics outperforms the subspace solver by factors of 20 or more for medium to very large models and is well suited to solid element models.

The addition of these two new eigensolvers allows ANSYS users to solve very large models in less than a few hours. Users could not attempt this previously due to disk limitations or the need to obtain a solution in a reasonable amount of time. **ANSYS 5.3 solves large problems!**

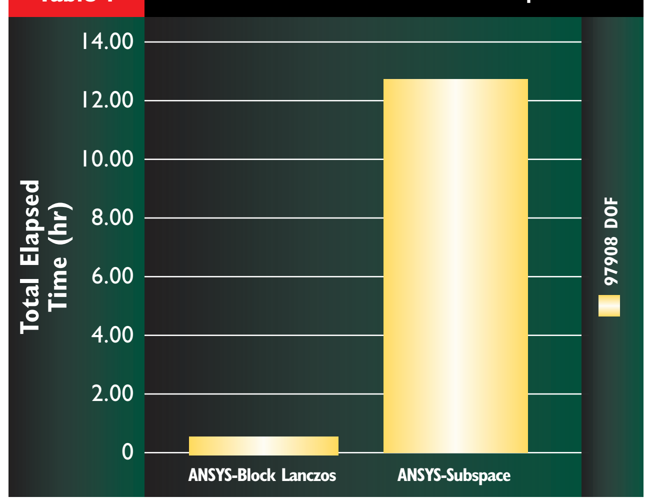
Nonlinear Improvements

Shell181, a new four-noded element, fully supports linear and nonlinear structure behaviors, including material inelastic, hyperelastic (Mooney-Rivlin), geometric large deflection, and contact mechanics. It has a finite strain formulation accounting for thickness change with very robust convergence characteristics.

Meshing Moves Forward

ANSYS 5.3 contains a multitude of meshing enhancements designed to increase usability. For example, the new default smartsize command (SMRTSIZE) contains 10 size levels from coarse to very fine. Element sizing on lines is automati-

Table 1 ANSYS—Block Lanczos vs Subspace



cally refined for proximity and curvature. Smartsizing improves the quality of quad dominant (mixed), triangle, and tetrahedral meshes.

Other additions include the new tetrahedral mesher that is 50 percent faster than in previous releases, and the simplified 2D mapped mesher that eliminates the labor of concatenation, making mapped area meshing easier. And the new status and stop bar, a graphical user interface (GUI) feature, allows the user to cleanly stop the meshing process without harming successfully meshed entities. Another convenient function of the GUI is the easy-to-use shape and size menu that allows copying of "hard" line element sizing and flipping of spacing ratios.

Geometry Transfer

Geometry transfer via the IGES standard (and future support for STEP) will include microstitching to handle small features and allow planar boolean operations on imported volumes (See related article on page 13).

New to ANSYS/FLOTRAN

With ANSYS 5.3, ANSYS/FLOTRAN will include three industry-standard non-Newtonian models, including the Power Law, the Carreau, and Bingham. Users can create their own non-Newtonian viscosity model as a function of velocity, velocity gradients, pressure, temperature, and nodal coordinates, including applications such as modeling the flow of blood in arteries and polymer flows.

For interactive computational fluid dynamic (CFD) analyses, a fully automated graphical solution tracking monitor will display the convergence information for all ANSYS/FLOTRAN degrees-of-freedom. Users will now be able to apply boundary conditions to their solid models rather than the finite element model. Radiation boundary conditions for modeling radiation heat transfer to ambient temperature is now available. Additionally, a new translator, independent of the ANSYS 5.3 release, will allow users to import models created with PATRAN.

ANSYS/Emag Expands

ANSYS made several major improvements to the area of solvers for multiphysics analysis. The capabilities of the Jacobi Conjugate Gradient (JCG) solver have been extended to include unsymmetric real, complex symmetric, and unsymmetric matrices. In addition, a new Incomplete Cholesky Conjugate Gradient (ICCG) solver has been added for symmetric and unsymmetric complex matrices.

These solvers are primarily aimed at solving time-harmonic field problems in acoustics, piezoelectrics, and electromagnetics, but can also be used for structural problems. The new solvers provide "Power" performance for large time-harmonic problems. In addition, the JCG solver can be used for solving static unsymmetric problems such as voltage-fed or circuit-fed electromagnetics, electromagnetics with velocity effects, piezoelectrics, and unsymmetric structural problems.

Electromagnetics now incorporate the effects of motion, which allow users to model moving conductors at constant linear or angular velocity. The new feature can be used for modeling eddy current braking systems, solid rotor induction machines, linear induction machines, NDT probes, MAGLEV vehicles, etc. Velocity effects can be included in static, harmonic, and transient field analyses.

Table 2 Block Lanczos Solver: ANSYS vs NASTRAN

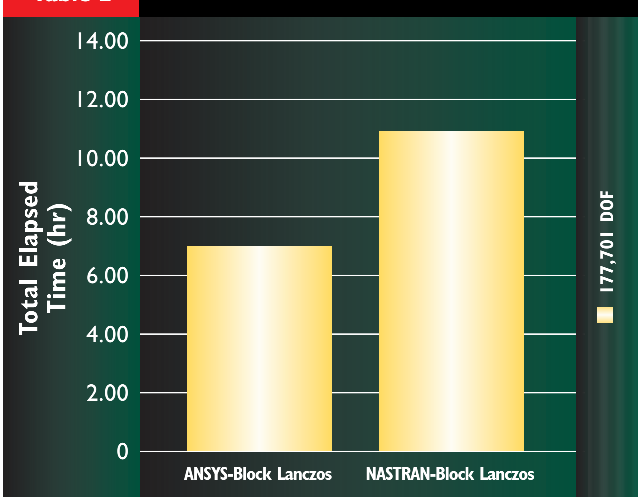
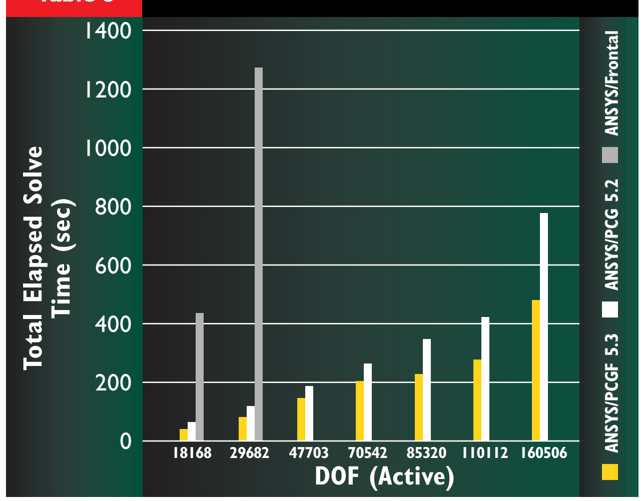


Table 3 ANSYS Static Solvers



Graphics

Improvements in the graphics area will make users' lives easier with just a few mouse clicks. New PowerGraphics capabilities allow users to look at the interior of a model. A Q-slice of results data has an increased speed of 36/1. Isosurface plot speed has also been increased by 6/1.

New features in the animation macro, such as the ability to save animation to a file, restore animation from a file, and control the rate of animation, give the user broader options when creating animations that include deformation with contours, Q-slice with contours, Q-slice with vectors, and isosurfaces. PowerGraphics capabilities accelerate improvements and reduce the time it takes to animate from six minutes to six seconds, giving users time to understand their analysis.

Other improvements based on user requests include picking within a macro, an improved contour command, and a graph display view that will be unaffected by the position of the model. Multiplotting, another new, easy point-and-click feature, allows users to display multiple types of plots on the screen (Figure 5).

Conclusion

ANSYS 5.3 promises to excite users with performance improvements that will enable them to obtain solutions in a fraction of the time previously required. Faster solutions allow users to evaluate more designs, optimize designs, and solve very large problems that previously were not possible. With ANSYS 5.3, ANSYS, Inc. delivers the most robust version of ANSYS ever and continues to maintain compatibility across files and platforms, ensuring maximum performance through an uninterrupted work cycle. ■

by Frank Marx, Manager
ANSYS Business Unit
ANSYS, Inc.

Jen Valachovic, Marketing Specialist
ANSYS, Inc.

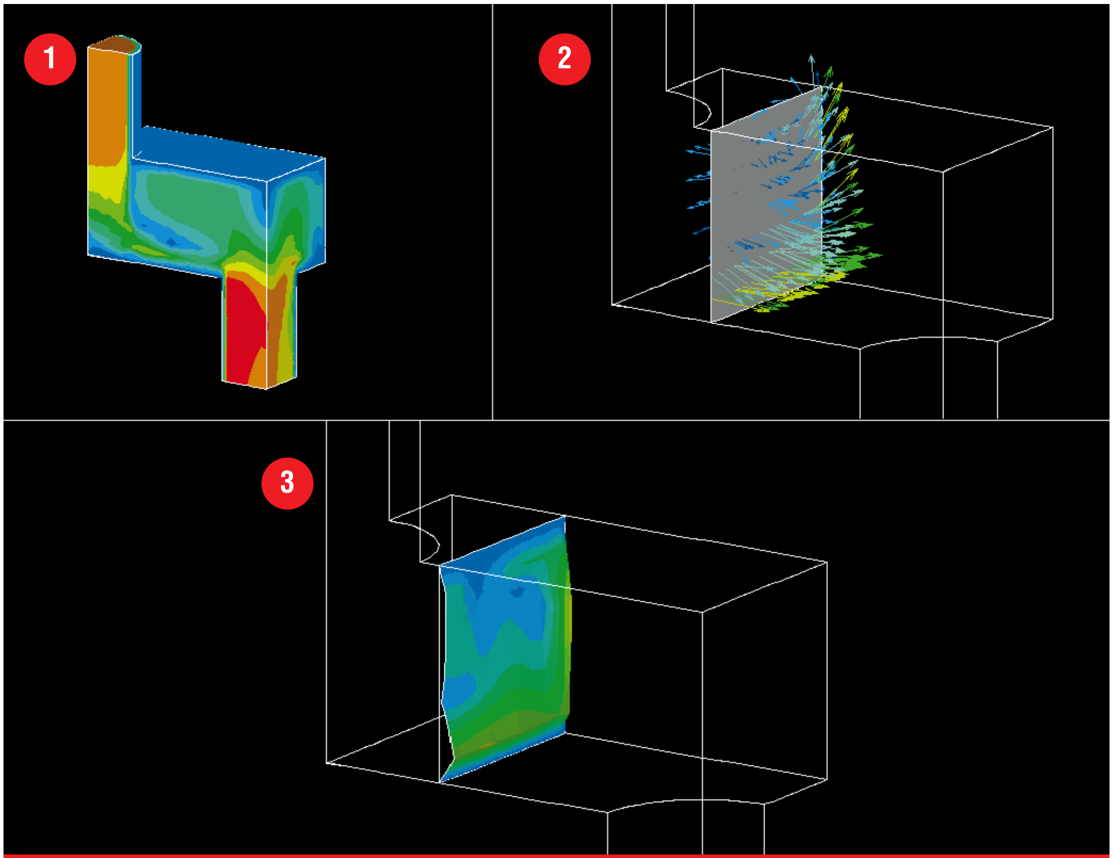


Figure 5

Example of multiplotting. Window 1 shows a velocity sum, window 2 is the Q-slice of a vector plot, and window 3 shows a nodal plot of a Vsum.

Powered by ANSYS™:

Next Generation Simulation Software

ANSYS Inc. is about to change the computer-aided engineering (CAE) landscape with the introduction of the ANSYS DesignSpace™ development environment. This exciting breakthrough technology represents a paradigm shift that redefines the industry's traditional approach to design analysis.

ANSYS developers evolved this object-oriented component analysis technology from the engine of the ANSYS program. They built a next generation system from the ground up based on the requirements of design engineers.

The DesignSpace development environment will enable computer-aided design (CAD) companies to build leading-edge, easy-to-use simulation technology into their products and offer modern analysis-based design optimization capabilities directly to their users. This is particularly important to CAD companies and their users as integrated design optimization is becoming an integral part of the product offerings of industry leaders.

One manifestation of this technology will enable users to document designs as they work through them. The CAD model, simulation engine, spreadsheet, and design report will share the same data and update automatically as users progress through the design cycle. Because of the shared data feature, users will only perform tasks once instead of multiple times.

Applications that are built using the DesignSpace development environment are compatible with the ANSYS family of products, giving users access to a full range of analysis functionality. If you need expanded functions, it is easy to upgrade to an advanced ANSYS product, such as ANSYS/Multiphysics. Users can transfer data between ANSYS programs with no data conversion or translation. This is invaluable to companies when addressing corporate-wide needs such as accelerating time-to-market and reducing product costs.

A team of expert ANSYS developers took the challenge to create the next generation of design analysis software which ensures ease-of-use through a Windows conceptual user interface. DesignSpace technology is Windows-based, using the latest in object-oriented software technology to give users unmatched flexibility. They backed this with the quality and power of ANSYS that has made it the finite element analysis (FEA) software of choice for companies around the world for the last quarter century.

DesignSpace modules enable designers to work directly within their familiar CAD package while ANSYS runs "behind the scenes" to produce accurate and reliable analysis results. CAD systems with this degree of integration are *"Powered by ANSYS"*.

ANSYS/AutoFEA 3D, the first product developed using the DesignSpace development environment, revolutionizes the design process by opening a window for assessing the integrity of designs inside AutoCAD®, while working directly with the AutoCAD model and data structures. (See related article on page 8.)

The DesignSpace initiative is central to ANSYS, Inc.'s vision of expanding access to the benefits of analysis to new audiences and integrating ANSYS with complementary technologies. Users of CAD systems that integrate DesignSpace will benefit greatly from being *"Powered by ANSYS"*. ■

by Jen Valachovic, Marketing Specialist
ANSYS, Inc.

ANSYS/AutoFEA 3D:

Good News for Design Engineers

ANSYS/AutoFEA 3D makes design engineers' jobs easier and enables them to create better designs, as well as shorten the development cycle. This product sheds a whole new light on validating the integrity of your designs.

Accessible design validation requires tight integration with a computer-aided design (CAD) system and an intuitive, goal-oriented approach. Integrated, up-front design validation shortens the product development cycle because the model is refined from the beginning of the process without requiring additional knowledge of new software and specialized functions. Time-consuming changes that involve creating new data, transferring that data, and then re-validating it, are now made directly to the original design in minutes. This intuitive approach allows design engineers to focus on producing better designs, not on running the software.

Work Inside Autodesk Mechanical Desktop or AutoCAD

Design engineers will feel like they are using one program because ANSYS/AutoFEA products provide design validation completely in Autodesk Mechanical Desktop™ and AutoCAD®. For example, an engineer designing a mountain bike can select a part for validation by picking it from the drawing (Figure 1). Since the part geometry remains in Autodesk Mechanical Desktop or AutoCAD, changes to that part are automatically made to the original mountain bike design.

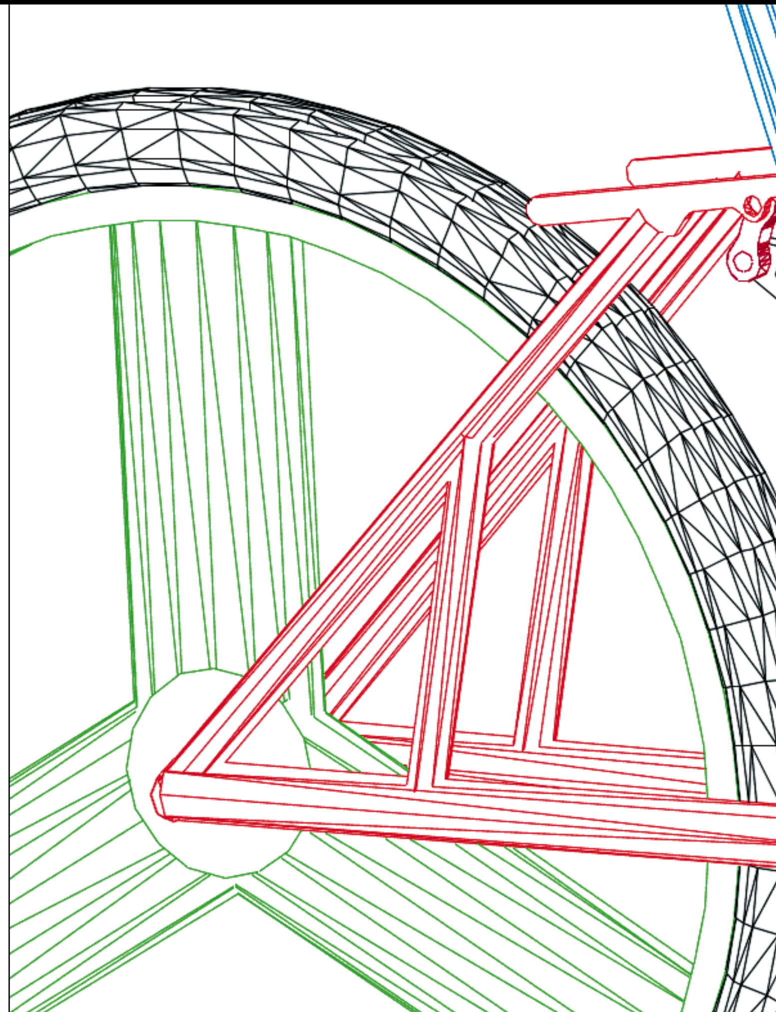


Figure 1

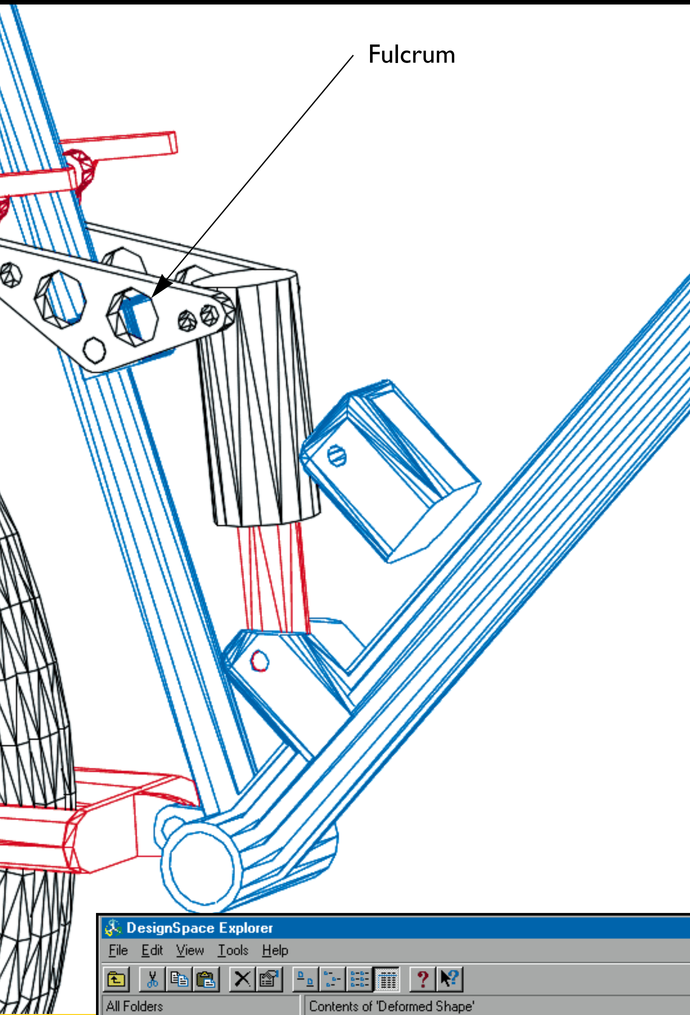
Take Control of Your Project

ANSYS/AutoFEA 3D was developed using the new DesignSpace technology from ANSYS, Inc. (See article on page 7 for further information on DesignSpace technology.) This program uses the DesignSpace Explorer, an intuitive approach to design validation that gives design engineers easy access and control to design projects through modern Windows functionality. Figure 2 shows a sample of the Explorer. Note the tree on the left. The root of the tree is the current drawing. Each project has key areas:

Model	Drawing Part, Material
Environment	Operating Conditions
Validation	Design Information and Results of the Project

The project structure is very flexible. Users can have multiple projects and/or duplicate parts of the project tree, and can examine different environments, investigate different models, and/or select other validation goals with a simple mouse click.

ANSYS/AutoFEA 3D supports right-mouse clicks on any object of the tree to get context menus. One click tells you what actions the object supports and gives users the ability to query the



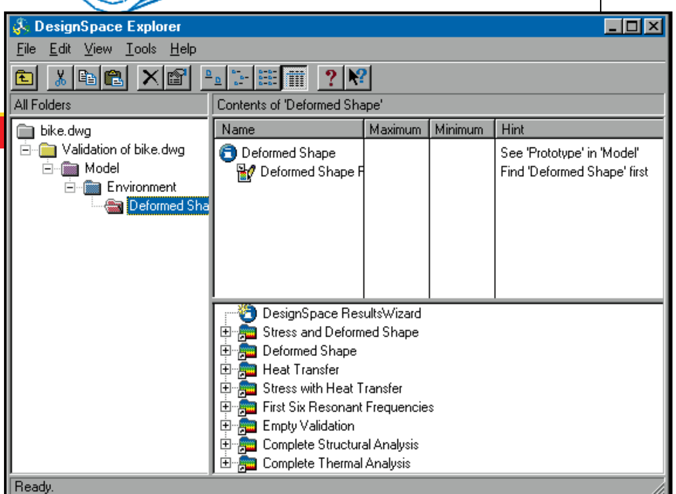


Figure 2

Design Space Explorer

object's properties. Windows tools, such as wizards, are used to add new conditions to an environment or select results information to add to a validation.

View Valuable Results

Pictures of results are displayed inside Autodesk Mechanical Desktop or AutoCAD drawings. Figure 3 shows a stress plot of the mountain bike's suspension pivot. This type of picture allows you to quickly determine regions in the parts that need further design attention.

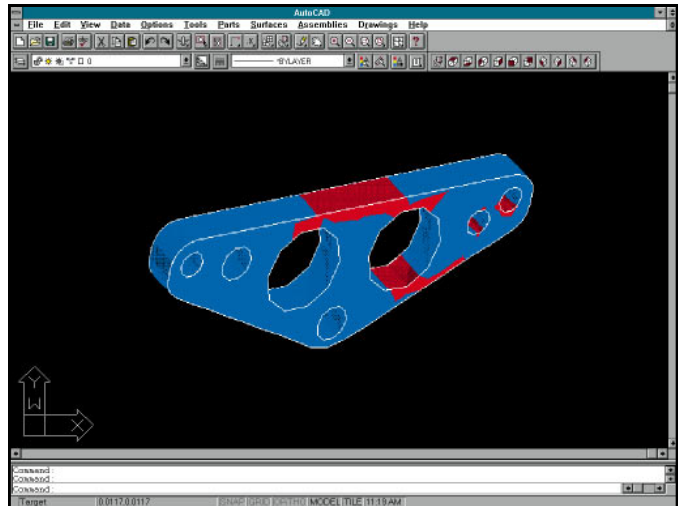


Figure 3

Stress plot of the mountain bike's suspension pivot.

Obtain Immediate Data on Design Changes

Because ANSYS/AutoFEA 3D is fully associative with Autodesk Mechanical Desktop, design engineers can quickly and easily test design changes. Environmental conditions such as pressures or forces remain attached to the part even if the geometry changes due to a parametric update. By simply choosing "find answers" from a right-button menu, a validation is done on the current geometry. No geometry transfers or re-definition of environmental conditions are required. ANSYS/AutoFEA 3D data is compatible with the entire ANSYS product line so you can easily upgrade to a program with advanced capabilities, such as ANSYS/Multiphysics.

Together, ANSYS, Inc. and Autodesk are leading the way to creating the engineer's desktop design environment and making your job, as a design engineer, easier. Produce better designs through accessible design validation using Autodesk Mechanical Desktop and ANSYS/AutoFEA 3D. ■

by Sue Batt, Vice President
Design Business Unit
ANSYS, Inc.

ANSYS Secures a Solid Swing:

Reducing the Vibration in a Tennis Racquet Frame

During a normal tennis match, a player can strike the ball over 1,000 times. When the tennis ball strikes a racquet, the impact causes vibrations that are distracting and uncomfortable to the player. Often there are forces exerted on the player's arm that can cause injuries like tendonitis or tennis elbow. To help reduce this racquet vibration, sporting goods manufacturers are including a passive damping system into the frame of the racquet. In a joint effort, Wilson Sporting Goods and 3M Company created a damping system through the use of the ANSYS finite element analysis (FEA) program. Their work has resulted in the Wilson Oversized Staff 5.7 Lite tennis racquet (Figure 1).

Figure 1

Wilson Oversized Staff 5.7 Lite tennis racquet.

"Racquet frame vibration is one aspect of the many areas of racquet performance that Wilson is constantly improving upon," said Po-Jen Cheng, Group Manager of Racquet Technology, Wilson Sporting Goods. *"String vibration is easily minimized, but from a racquet standpoint there has not been a simple way to minimize vibration,"* said Ming-Lai Lai, Research Specialist, Vibration Control at 3M. The damping system uses a viscoelastic material along with a stiff composite constraining layer that is molded on the inner surface of the tennis racquet frame. When a ball strikes the racquet, the vibration causes a shearing strain in the viscoelastic material. This strain energy is partially dissipated, thereby increasing the racquet damping. *"Using ANSYS, the effectiveness of the damping ratio to change in the key variables can be studied, minimizing the need for prototypes,"* said Michael Harms, Advanced Design Engineer, Engineering Analysis at 3M. *"This method can also be used to determine an optimum design by maximizing the damping ratio with minimal weight addition,"* he added. A Pro/ENGINEER model of the racquet was created and then exported into ANSYS for finite element modal analysis.

The Approach: Finite Element Modeling

Because of the complex geometry of the tennis racquet, it is natural to use finite element modeling to solve this problem. The technique used to determine the damping ratio is the modal strain energy method. Here is how the damping ratio in the racquet is determined. First, a finite element analysis is completed to determine the natural frequencies of the structure. In this case, using the 3M ISD damping property specification, the material loss factor (η_v) and the elastic modulus are determined for the frequency and operating temperature. Next, a modal analysis of the structure, with the damping system in place, is solved. The elastic strain energy in the viscoelastic material along the with the total strain energy in the racquet is extracted for this mode. The equation for calculating the damping ratio is shown in Figure 2.

The finite element model consists of shell and solid elements. Shell elements, used for the constraining layer and the racquet frame, were chosen over solid elements to minimize the mesh size. Since the thickness of the constraining layer and racquet shell are small compared to the cross-sectional dimensions, this assumption is justified. The viscoelastic material is represented by the solid elements because the major strain energy component in it is a result of the plane shear deformation. The plane shear deformation can only be captured by solid elements. In order to minimize the weight added to the racquet, the damping system stops about four inches from the tip. The mesh size is also reduced by using a symmetry boundary condition along the racquet mid-plane.

"It is our experience from this project and others like it that the aspect ratio for the visco elements can be very high yet yield accurate strain energy values," said H. S. Gopal, Engineering Specialist at 3M. The model used in this analysis is comprised of a mesh of roughly 4,500 nodes and 13,000 elements with aspect ratios up to 30 to one for the visco solid elements. The accuracy of the natural frequency and strain energy values were checked by running a model with over 20,000 elements. Results varied approximately two percent between models, indicating that the coarser mesh is adequate.

$$\zeta = \frac{\eta_v}{2}\left[\frac{V_v}{V_{total}}\right]$$

Where:

ζ = Damping ratio in the racquet attributed to the viscoelastic core.

η_v = Material loss factor of the viscoelastic core evaluated at the calculated resonant frequency.

V_v/V_{total} = Fraction of the elastic strain energy in the viscoelastic core relative to the total strain energy in the racquet.

Figure 2

Equation for calculating the damping ratio.

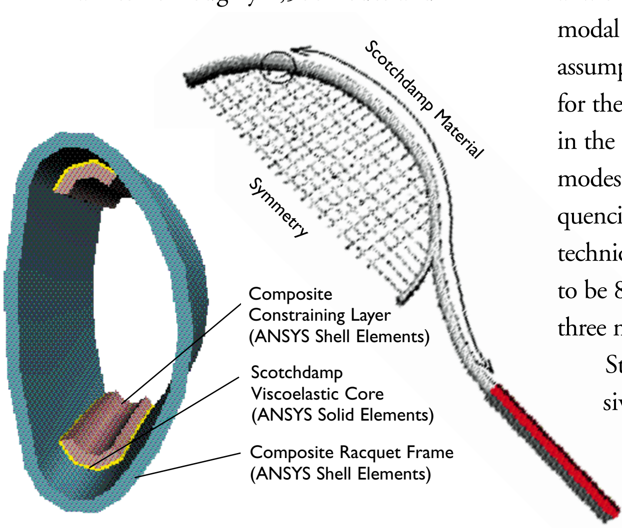


Figure 3

Racquet Geometry

The Match-Up: Analytical Versus Experimental Results

"ANSYS software allows us to isolate one variable at a time and study its effect on the damping ratio in the tennis racquet. This eliminates the manufacturing variability associated with prototypes and reduces the number needed," said Harms.

First, the bending frequency of a free, unstrung racquet was tested using accelerometers. In two measurable passes, frequency measured 150Hz and 420Hz. The racquet is made of a composite material with unknown orthotropic properties. A modal analysis was conducted with the assumption of isotropic material properties for the racquet. The elastic modulus used in the model was varied such that two modes matched the measurable natural frequencies from the test results. Using this technique, the elastic modulus was found to be 8.5 Mpsi. Figures 4-6 show the first three modes of vibration for the racquet.

Stringing a racquet induces compressive stresses in the structure, decreasing the stiffness of the structure and lowering the natural frequency. In the finite element model, the string tension is applied to the racquet frame as forces (60lbf) at each string location. The stresses due to stringing are solved for using a static stress solution. The strung natural frequency of each tennis racquet is solved using a pre-stressed modal analysis.

The damping analysis was carried out on an unstrung racquet. The damping ratio attributed to the viscoelastic core was calculated using the 150Hz first bending mode. The first mode was chosen because it had the greatest vibrational energy. The damping system was placed on the inside surface of the cross section away from the neutral axis of the first bending mode (Figure 3).

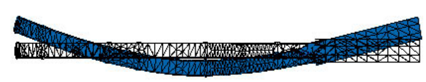


Figure 4

First bending mode (149Hz) side view.

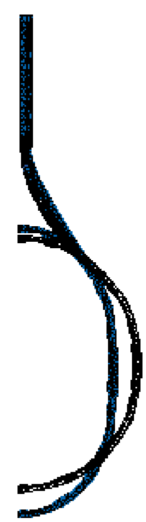


Figure 5

Second bending mode (237Hz) top view.

Figure 6

Third bending mode (403Hz) side view.

The results of the analysis found that a thin viscoelastic core with a thick constraining layer provided the highest strain energy ratio. The best case analyzed yielded a strain energy ratio of 2.79 percent. Using the information from the equation shown in Figure 2 (page 11), along with the material loss factor of 1.0 (at 150Hz and 20C), yields a damping ratio attributed to the damping system alone of 1.4 percent. Combining this with the inherent 0.7 percent damping in the racquet would give a total damping ratio of 2.1 percent.

Also of interest are the regions of the racquet where the greatest strain energy occurred in the viscoelastic core. This knowledge is valuable when determining where the damping system can be most effective. *"Using ANSYS, we were able to plot the strain energy distribution in the visco, and determine the regions with the greatest strain energy,"* said Harms.

Effecting the ANSYS Analysis: The Nylon Bladder

In the manufacturing of the tennis racquet, a thin nylon bladder (Figure 7) is used as the innermost ply of the racquet construction. This bladder, which is used to inflate the tubular structure of the racquet against the outer steel mold, remains in the racquet permanently. Because the damping system relies on the shearing deformation between the racquet and constraining layer for the dissipation of energy, it is important that this deformation is not obstructed or restrained. By using FEA, the effect of a bladder was identified as a significant factor affecting the damping ratio. The bladder was found to obstruct the shearing deformation of the damping system, thus reducing the strain energy in the viscoelastic core. With this knowledge, manufacturing techniques were changed so that the bladder would not obstruct the damping system.

Shifting Location

3M engineers considered a shifting that could occur during the manufacturing process. The shift can happen when the damping system is placed in the mold. Once again, an analysis was run shifting both the damping systems toward the neutral axis of the cross section (Figure 8). The results showed the damping ratio was insensitive to this shift in location.

Game, Set, and Match: Conclusions

FEA was used to compute the strain energy ratio in a tennis racquet utilizing a constrained layer damping system. The modal strain energy method combined with the known frequency dependent material properties of the viscoelastic material were then used to solve for the damping ratio. Using ANSYS, 3M was able to identify key variables affecting the damping ratio, without having to build additional prototypes. *"This study helped us optimize the geometry and location of the visco and constraining layers to achieve maximum damping with minimal additional weight,"* said Harms. *"This new technology built into our racquet doesn't completely eliminate racquet frame vibration, but has the potential to reduce discomfort and injury caused by racquet vibration,"* said Cheng, *"computer-aided technology such as ANSYS helps Wilson create some of the safest and most reliable sporting goods on the market today."* ■

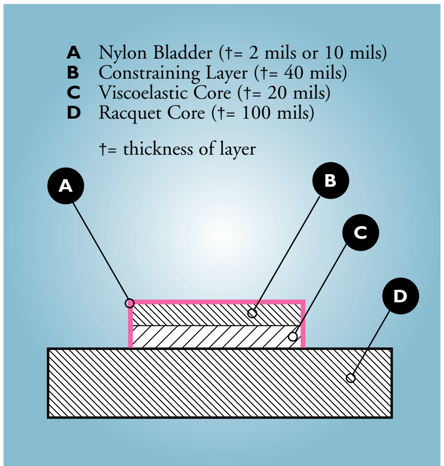


Figure 7
Bladder Model Shifting Location

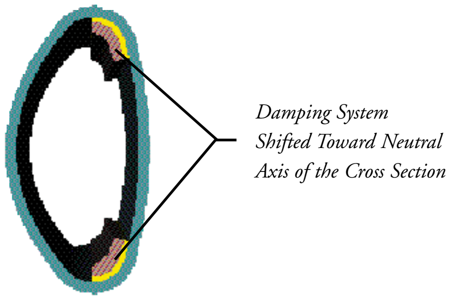


Figure 8
Shifted Damping System

by Po-Jen Cheng, Group Manager of Racquet Technology
Wilson Sporting Goods
Chicago, IL

Michael Harms, Advanced Design Engineer
3M Corporation
St. Paul, MN

Daniel Parrish, Marketing Specialist
ANSYS, Inc.

ANSYS Connection Tools Deliver Smart Solutions

The Program Customization Services group, a division of Customer Services at ANSYS, Inc., interfaces the ANSYS program with major CAD packages to create “ANSYS Connection” tools.

Figure 1
Solid model developed in Pro/ENGINEER and transferred directly into ANSYS using ANSYS Connection for Pro/ENGINEER.

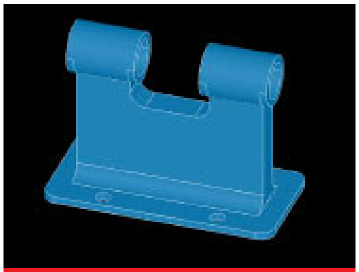


Figure 2
Solid model developed in Pro/ENGINEER and transferred directly into ANSYS using ANSYS Connection for Pro/ENGINEER.

Figure 3
Complex solid model geometry developed in Unigraphics and transferred directly into ANSYS using ANSYS Connection for Unigraphics.

They released ANSYS Connection for Pro/ENGINEER in April, which enables users to access powerful analysis tools from within the Pro/ENGINEER® environment. Users gain reliability through the seamless geometry transfer of Pro/ENGINEER solid models (Figures 1 and 2).

Comprehensive Suite of Solutions

ANSYS, Inc. has also teamed up with EDS® Unigraphics® to offer users a suite of solutions that address a broad range of integration issues consisting of:

ANSYS Connection for Unigraphics
This solution customizes ANSYS, providing direct access to Unigraphics geometry (Figure 3).

ANSYS for Unigraphics
This solution makes ANSYS directly accessible from within the Unigraphics environment. With the familiar look and feel of the design tool, users can evaluate design alternatives and optimize their concept.

Custom ANSYS for Unigraphics
This program allows users to define specific ANSYS program capabilities, customizing them so they will run inside the Unigraphics environment. This solution can also integrate other technologies as needed to meet specific requirements.

Unparalleled Power

For the first time, Unigraphics users have access to powerful and robust analysis tools. The advanced suite of ANSYS solutions coupled with the system from EDS Unigraphics gives users unbeatable engineering power. ANSYS Connection for Unigraphics improves design speed and quality, eliminates rework and data transfer delays, and improves and expands access to ANSYS technology.

In May, the Program Customization Services group delivered Phase I of the ANSYS Connection for Unigraphics system which provides a clean, one-way geometry transfer from Unigraphics to ANSYS. They are rapidly moving towards Phase II, which will provide parameter optimization by allowing bi-directional transfer of parameters between the two packages. Phase III will integrate a subset of ANSYS into Unigraphics using their menus. In all phases, an ANSYS solid model will be created. The ANSYS program will complete all finite element preprocessing such as meshing and boundary conditions, and parametric updates will be handled by the Unigraphics program.

The ANSYS Approach

The ANSYS approach is to provide an open environment that supports best-of-class technologies, encourages integration of customer-specific applications, and provides flexibility for future advances. ANSYS bridges individual islands of productivity by providing comprehensive engineering tools allowing customers to look at solutions that address every project phase, from design through manufacturing.

Users can contact the ANSYS Program Customization Services group by e-mail at cad_pcs@ansys.com. ■

by Jen Valachovic, Marketing Specialist
ANSYS, Inc.

Implementing ANSYS Early in the Design Cycle

The Enhanced Solution Partner (ESP) program supports world-class software partners whose products possess leading-edge capabilities that complement ANSYS. A variety of vertical application developers are enrolled in the ESP program to take advantage of the ANSYS program as a platform to create custom products.

As a service to our customers who are seeking a customized software program, we have published an ESP Directory on the ANSYS HomePage. This article discusses an application performed by Automated Analysis Corporation (AAC), a participant in the ESP program. AAC, located in Michigan, provides computer-aided engineering (CAE) consulting services to the "transportation" industry (for example, automotive, aerospace, heavy equipment), as well as to consumer goods manufacturers. Services include performing analyses at AAC facilities (in-house consulting); assignment of engineering personnel to client sites; development, sales, and support of engineering software; and training for advanced CAE methods.

In industries such as aerospace, automotive, heavy equipment, and consumer goods, engineers often need to predict and reduce acoustic noise problems early in the design cycle. This type of acoustic prediction is performed most effectively using a combination of structural finite element analysis (FEA) and acoustical boundary element analysis. AAC conducted the following analyses using the ANSYS program for structural FEA and COMET/Acoustics for acoustical boundary element analysis.

In order to evaluate the acoustic performance of a newly designed engine block under various operating conditions, prototypes of different designs can be built, and acoustic parameters, such as sound pressure level, acoustic intensity, and radiated sound power, can be measured at various operating conditions. This approach, however, is costly and time-consuming. To reduce the number of costly prototypes and shorten design cycles, AAC applies a procedure to evaluate acoustic radiation from an engine block by applying structural FEA together with acoustical boundary element analysis. In one such analysis, the acoustic radiation of a 2.5 liter V-6 engine was predicted in the 0 to 1kHz frequency range.

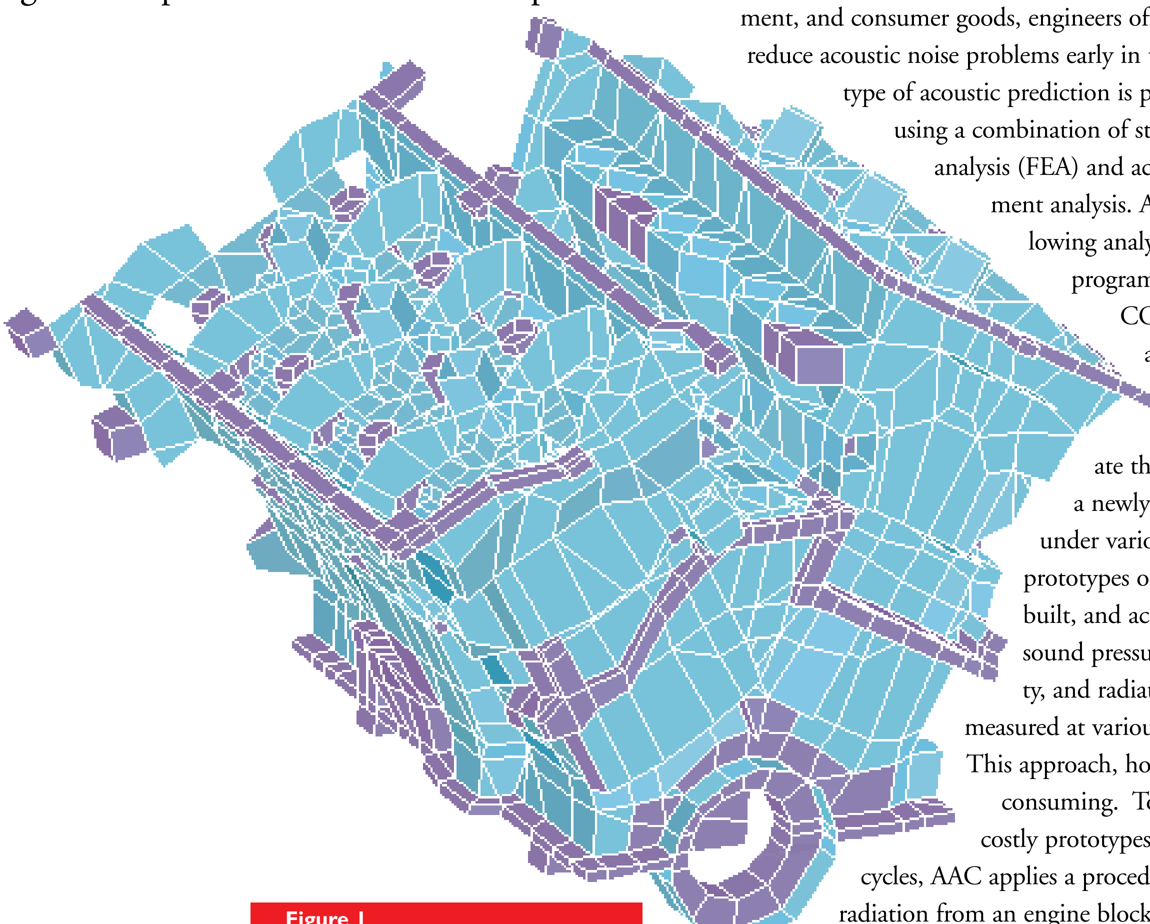


Figure 1

Structural finite element model of the engine.

Structural Finite Element Analysis

To predict acoustic radiation, the first step is to predict the structural vibrations on the surface of the engine block. To begin the analysis, engineers at AAC built a structural finite element model using the ANSYS preprocessor. This model consisted of three parts: the cylinder head, the engine block, and the crank shaft bearing caps. The cylinder head, the engine block, and the bearing caps were made of aluminum, with cast iron sleeves lining the engine block's cylinder walls. The structural finite element model is shown in Figure 1.

AAC engineers used ANSYS harmonic response capabilities with the mode superposition method to predict structural vibrations on the surface of the engine block. The first step in using the mode superposition method was to find a sufficient set of modes. Because the intent of this analysis was to predict the acoustic radiation of the engine block in the 0 to 1kHz frequency range, all modes below 2kHz were solved using the modal analysis capability in ANSYS.

Seven dynamic body modes were found below 2kHz. In the harmonic response analysis, forces for one operating condition (an engine speed of 1600RPM) were applied to the main crank bearing surfaces, the cylinder walls, and the cylinder head. The bearing forces include horizontal and vertical forces, as well as horizontal and vertical movements.

The forces applied to the cylinder walls represent piston slap that occurs at the transition at Top Dead Center (TDC) and Bottom Dead Center (BDC). These forces were applied normal to the cylinder walls. The forces applied to the cylinder head represent pressure in the direction normal to the inner surface of the cylinder head. Performing the harmonic response analysis provided the displacement amplitudes for the engine model. These results were stored in an ANSYS structural result (.rst) file for subsequent use in performing the acoustic analysis phase.

Acoustic Boundary Element Analysis

Engineers at AAC then performed an acoustic analysis using COMET/Acoustics. COMET/Acoustics is an advanced acoustic analysis software developed and marketed by AAC. COMET/Acoustics consists of a Direct Multi-Domain Boundary Element Method, an Indirect Boundary Element Method, and an Acoustic Finite Element Method. AAC chose the Direct Multi-Domain Boundary Element Method for this analysis.

First, engineers created acoustic boundary element and data recovery meshes in the ANSYS preprocessor (PREP7), which they output to two separate ANSYS "coded data base" (.cdb) files. The acoustic data recovery mesh is a representation of microphone locations where acoustic quantities are recovered (computed). ANSYS .cdb files are used to transfer model data from ANSYS to COMET/Vision, the pre and postprocessor for COMET/Acoustics. The acoustic boundary element model (Figure 2) was designed to capture acoustic radiation due to structural vibrations from exposed engine block surfaces.

To transfer the structural vibration data from the structural finite element model to the acoustic boundary element model, the ANSYS .rst file and the .cdb file containing the structural finite element model were imported into COMET/Vision. COMET/Vision was then used to generate a corresponding set of acoustic normal velocity boundary conditions for the acoustic boundary element model from the ANSYS structural vibration results. This procedure extracts the nodal displacement results from the .rst file for the nodes that are on the exterior surface of the structural finite element model and converts them to velocities. It then computes an acoustic normal velocity boundary condition, using a weighted averaging algorithm, for each element in the boundary element model. This algorithm allows for the fact that nodes of the structural and acoustic model may or may not be coincident. The

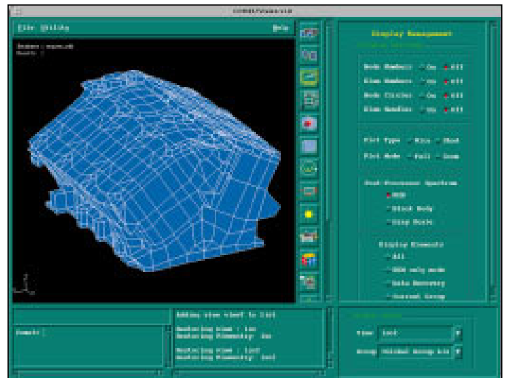


Figure 2

Acoustic boundary element model of the engine.

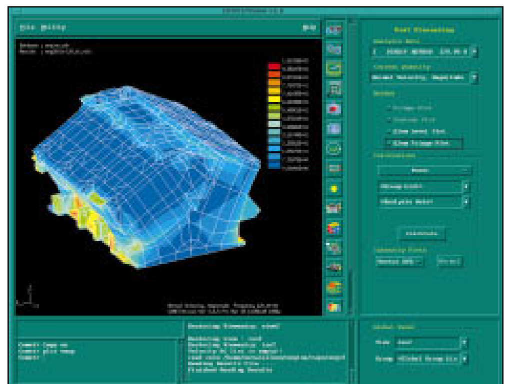


Figure 3

Magnitudes of normal velocity boundary conditions at 120Hz for the acoustic boundary element model.

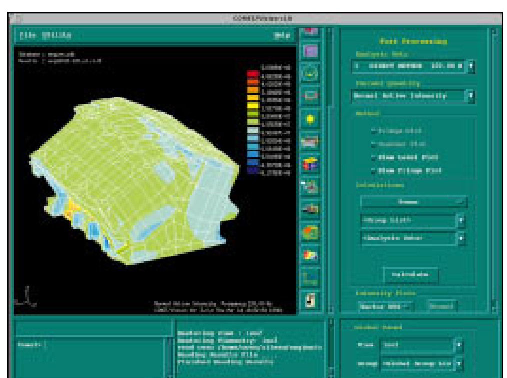


Figure 4

Acoustic normal intensity at the surface of the engine at 120Hz.

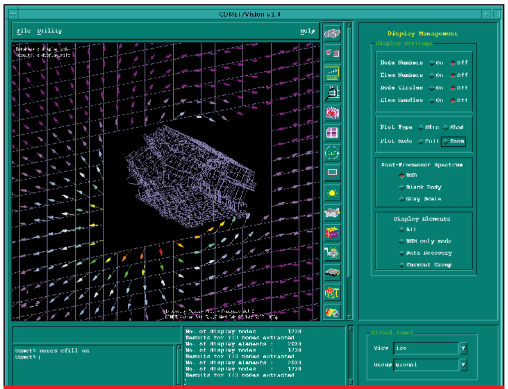


Figure 5A

Acoustic intensity at 120Hz at the acoustic data recovery surface.

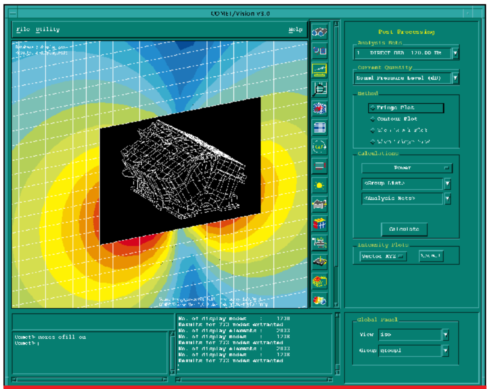


Figure 5B

Sound pressure level at 120Hz at the acoustic data recovery surface.

magnitudes of the resulting acoustic normal velocity boundary conditions for 120Hz are shown in Figure 3 (page 15).

Engineers then used COMET/Acoustics to perform a harmonic frequency response analysis for discrete frequencies in the 0 to 1kHz range. To post-process the acoustical results, normal intensity was first plotted on the acoustic boundary element model for 120Hz (Figure 4, page 15). (This information can provide useful insight for design modifications since positive intensity areas radiate noise and negative intensity areas absorb noise.) Next, intensity and sound pressure levels at 120Hz were captured on the data recovery mesh (Figures 5A and 5B). Finally, total radiated sound power was plotted against frequency in Figure 6.

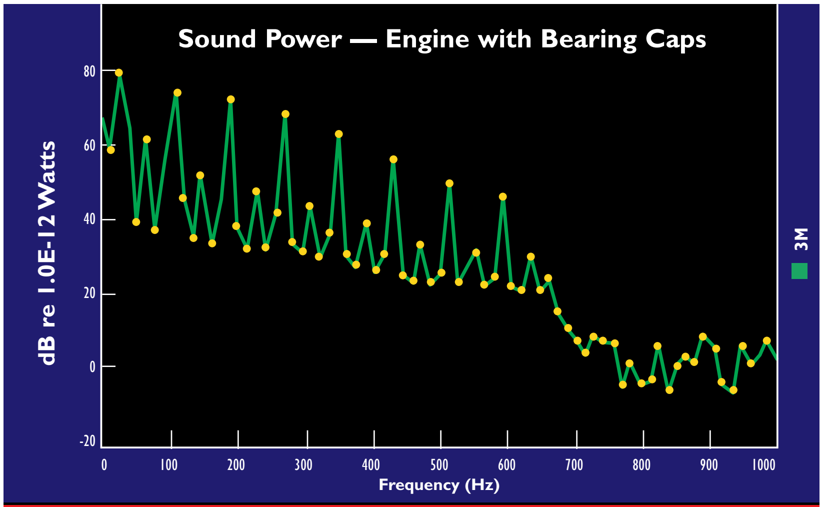


Figure 6

Sound power radiated from the engine as a function of frequency.

Conclusion

AAC engineers executed a complex acoustic performance evaluation numerically by combining structural FEA in ANSYS and acoustic boundary element analysis in COMET/Acoustics. These viable engineering tools result in important bottom-line improvements during the early stage in a design cycle: reduction in turnaround time and cost. ■

by W. Ben Tsoi, Vibro-Acoustics CAE Consultant
Automated Analysis Corporation
Ann Arbor, MI

Christopher G. Mollo, Senior Project Engineer
Automated Analysis Corporation
Ann Arbor, MI

ANSYS for Windows 95 – Advanced Multiphysics on the PC:

The Personal Computer Comes of Age

The company that pioneered design analysis on the personal computer does it again with the release of ANSYS 5.2 for Windows 95, the only coupled-field multiphysics analysis solution running under Windows 95.

The release of ANSYS on the Windows 95 operating platform represents an important stage in the evolution of analysis software. In the past, most advanced analysis capabilities, such as coupled-field multiphysics and nonlinearities, were the exclusive domain of high-end UNIX workstations and supercomputers. A majority of ANSYS customers used these systems for many years.

PC processors have become more proficient – virtually matching the computational efficiency of many UNIX workstations with the release of the Pentium chip – and are now a cost-effective analysis option for small to mid-size companies. The PC, networked through either Windows NT or Windows 95, has also become a viable alternative for design groups with large manufacturing concerns. These trends reflect a gradually changing ANSYS customer base and an elevation in respectability for the PC, with more ANSYS users solving design and engineering challenges on the desktop today than ever before.

ANSYS on Windows? It's Time!

The release of ANSYS for Windows 95 represents a significant milestone in the development of analysis software.

Power has been the trademark of computer-aided engineering (CAE) for two decades; powerful analysis software matched with powerful hardware to solve sophisticated problems on large models. Just as ANSYS has become more powerful with the addition of incredibly efficient solvers and an improved user interface, so has computer hardware. The most far-reaching developments, in terms of processing and operating environment, have been on the PC.

Cost reductions due to decreased disk space, improved price/performance ratios, faster processors, multi-tasking, and the ability to run over a network are just some of the reasons that the personal computer has climbed from a microprocessor for the home to a functioning piece of production hardware.

Ask yourself how you would have answered the following question just five years ago: What hardware should I use to solve a complex model that involves large deformation, general contact, or multiphysics? The only options available in 1991 were a supercomputer or high-end workstation. Today, some ANSYS customers solve large problems like this on the desktop.

It becomes increasingly clear that the PC has come of age in product design and engineering settings when $10,000 purchases a PC with capabilities rivaling workstations that are priced several times that amount. As PC technology continues to advance, engineers will have greater access to ANSYS capabilities than ever before.

Capability and Utility

Ask any power software user about the characteristics of a program that increase productivity, and nine times out of ten they will say it's a combination of capabilities and utility: 1) What does the software do? and 2) How easy is it to make the software do it?

The same is true of design analysis software. Just think of the changes in both features and interactivity that have occurred with ANSYS software over the past 25 years. The history of ANSYS software is marked by the continual addition of capabilities and user interface improvements.

With ANSYS for Windows 95, users enjoy the productive combination of advanced analysis capabilities and a highly utilitarian operating environment. Windows 95 is the result of three years of usability studies undertaken by Microsoft Corporation, to make what was already thought by many to be one of the world's easiest operating system (Windows), even better. It's easy for engineers to learn ANSYS in the Windows environment because they are already familiar with Windows-based business and office software. By combining the power of ANSYS with Microsoft's Windows 95 operating system, engineers can perform high-end analyses on the desktop.

As hardware and software continue to evolve, ANSYS, Inc. is committed to supporting developments in technology that help customers. ANSYS for Windows 95 is just one example of that ongoing commitment. ■

by Tim Trainer, Manager
Marketing Programs
ANSYS, Inc.

Check Out What the International ANSYS Supp

Easy ANSYS Graphics

Mr. Uli Kukulies, an ANSYS customer of CAD-FEM GmbH, the ASD located in Germany, recently developed a flexible, Windows-based program called ANS4DTP that efficiently and effectively enables worldwide users to convert ANSYS graphics to multiple formats.

Users frequently express the need to export or convert graphics from the ANSYS graphic file format (.GRPH or file33) into popular file formats such as TIFF, BMP, EPS, and AI, that can be imported into word-processing or DTP programs. Users can now create professional graphics because the ANS4DTP program enables them to vary line weight, zoom, scale, and change background colors, brightness, and shading. Vector-oriented file formats like EPS and AI can be imported into vector-oriented programs where customers can make multiple changes to an image (Figures 1-4).

CAD-FEM GmbH distributes the ANS4DTP program worldwide. For more information, contact CAD-FEM GmbH at roswald@cadfem.de.

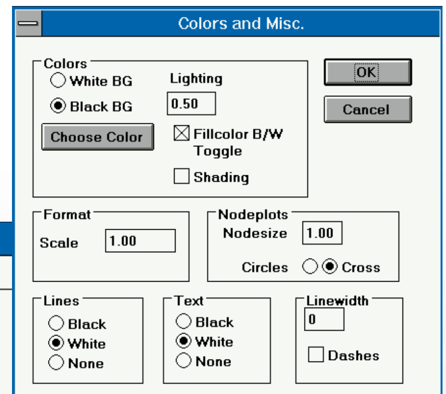


Figure 2

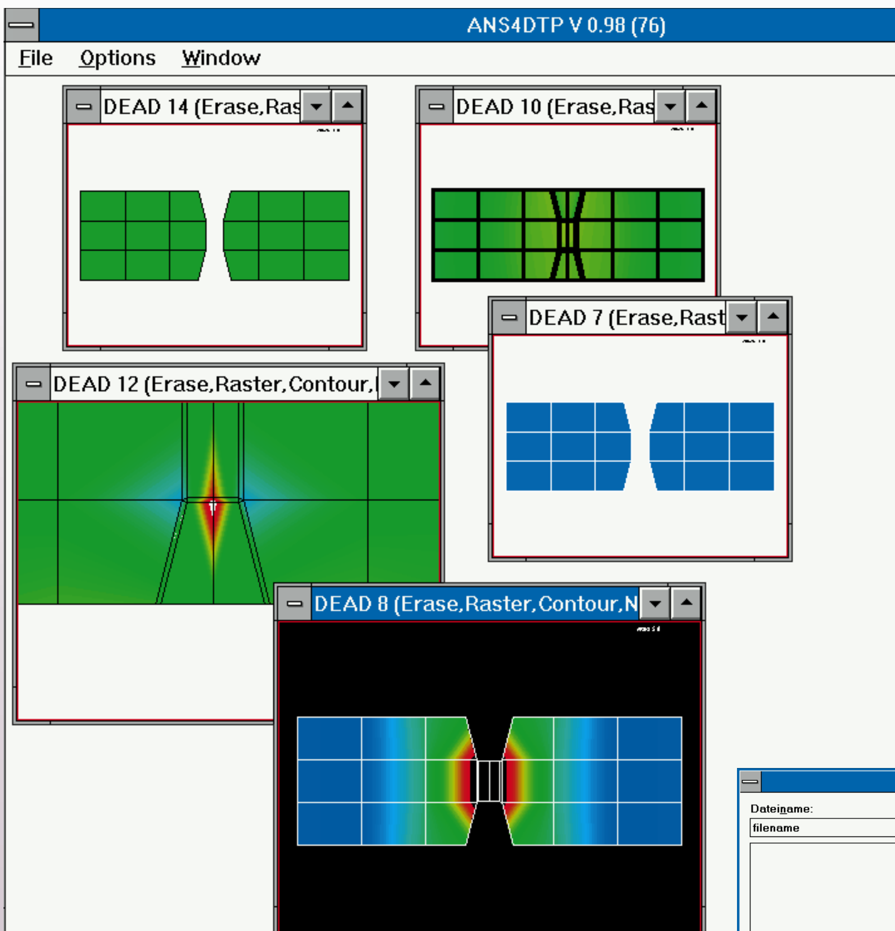


Figure 1

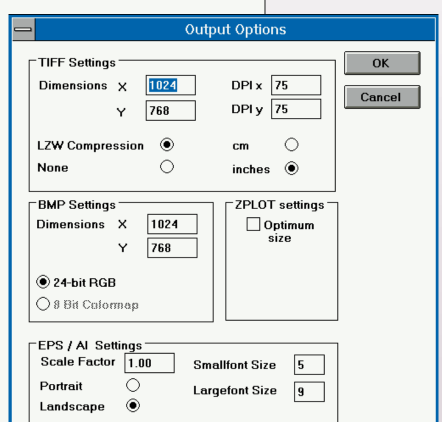


Figure 3

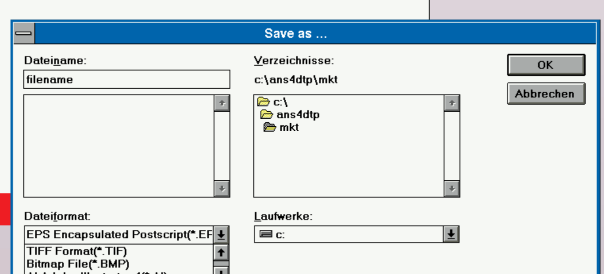


Figure 4

ort Distributors (ASDs) Have Been Doing Lately

STRUCOM's Quality System

Structures and Computers Ltd. (STRUCOM), the ASD in England, in association with NAFEMS QA LTD, is conducting a series of workshops on Quality Certification in Finite Element Analysis (FEA). Their goal is to provide FEA practitioners with the tools and information necessary to implement an in-house quality system that meets the requirements of the ISO 9001 and NAFEMS QSS quality standards. STRUCOM Consulting Engineers, a division of STRUCOM, specializes in the use of the finite element method for solving complex technical problems in the fields of mechanical, civil, structural, offshore, aeronautical, and automotive engineering. It was the first organization worldwide to achieve third-party certification by both the above-mentioned standards.

The following is a summary of a technical paper that will be published in full in the 1996 ANSYS Conference Proceedings.

The emergence of the finite element method has changed the engineering design environment radically over the last two decades. Its use within the design process gives rise to situations in which increasing reliance is placed upon results of uncorroborated analysis. It is against this background that STRUCOM Consulting Engineers has developed, documented, and implemented a quality system [1] for product qualification supported by FEA. The purpose of the quality system is to minimize the occurrence of error, thereby providing a basis for efficient, reliable, and safe product qualification. The quality system is appropriate to all situations where the finite element method is applied in the assessment of the integrity of engineering products.

The quality system meets the requirements of quality standards ISO 9001 [2] and NAFEMS Quality System Supplement (QSS) [3]. ISO 9001 is an international quality standard dealing broadly with issues of management in industrial practice. NAFEMS QSS is a quality standard dealing specifically with issues of management of FEA applications.

STRUCOM's quality system was originally certified by NAFEMS QA LTD in January 1994. It has since been in use on a daily basis for commercial product qualification where efficiency, reliability, and safety are key issues. Since certification, the implemented quality system has been subjected to several surveillance audits by NAFEMS QA lead assessors and FEA expert assessors. The system is continually under development and has undergone several revisions since the time of original certification.

The quality system is based on current best practice in the management of FEA and the requirements of the above-mentioned quality standards are supplemented by personnel competence requirements from the NAFEMS Registered Analyst Scheme [4], and product qualification concepts from SAFESA Management Guidelines [5].

Figure 1 shows Jonathan Smith, Director of STRUCOM, receiving the first NAFEMS QA certificate for Quality System ISO 9001 and NAFEMS QSS from Dr. A. Denton, previous President of the Institution of Mechanical Engineers.

Figure 1

Jonathan Smith, Director of STRUCOM, receiving the first NAFEMS QA certificate for Quality System ISO 9001 and NAFEMS QSS from Dr. A. Denton.

A Quality System for Product Qualification

Product qualification is the determination of a product's "fitness for purpose" in respect to its ability to satisfy the qualification criteria. The achievement of quality certification requires a clear definition of what is to be achieved, a description of functions and activities that need to be performed, and the control and monitoring of those functions and activities.

These goals are achieved by means of a quality system, which is a description of the way the supplier's organization conducts its business, within the scope of its quality certification. The main elements of a quality system are policy, standards, procedures, and control (Table 1, see page 20).

The organization of selected quality procedures is shown in Table 2 (See page 21).

Quality Procedures

Personnel assigned to product qualification collectively provide the required expertise appropriate to the scope and "category of

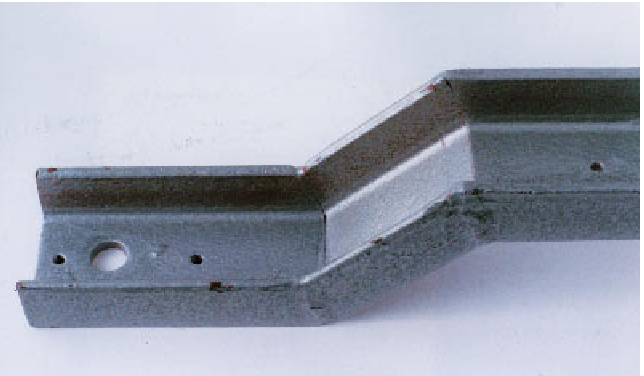


Figure 2

Potential source of idealization error in the stiffness of a welded joint.

importance" of the analysis.

Analysis software used in support of product qualification is verified before application. The analysis software supplier is also required to provide evidence of a software quality system.

The quality management system provides the analyst with a professional support environment that includes the provision of appropriate personnel and computing resources. Procedures are defined for the approval, issue, and modification of quality procedures and documents. Quality records are maintained to demonstrate achievement of the required quality and effective operation of the quality system.

Product Qualification

Product qualification determines a product's fitness for purpose in respect to its ability to satisfy the qualification criteria. Qualification criteria can be divided into three classes: empirical rules, permissible stress, and limit state.

The scope of product qualification involves defining both the criteria against which the product will be assessed and the role of FEA in the qualification.

The procedures used must reflect the category of importance of the analysis to which they may be applied because the safety of the product may be dependent upon the qualification conclusions. Contract procedures cover analysis requirements, contractual obligations, and the supplier's ability to meet them.

The assessment stage involves detailed FEA, error assessment, model validation, and results checking. Model validation is carried out by an independent assessment to ensure that the model is adequate for supporting the design qualification conclusions.

Calculated responses are compared with qualification criteria, and a conclusion, qualified or not qualified, is reached. A written report records the qualification conclusions and documents the supporting assessment, which includes all relevant items with an assessment of accuracy and engineering relevance.

Error Identification and Control

In order to minimize the occurrence of error in the qualification process, error treatment techniques are employed to identify error sources within a product qualification and reduce the effects of errors to an acceptable threshold. Four main classes of errors are recognized that relate to the stage of the qualification process in which they are introduced: idealization error, procedural error, formulation error, and system error. Figure 2 presents an example of an idealization error.

The process of decomposing the structure into well-defined components with a description of the boundary conditions will highlight the uncertainty during the idealization planning stage.

Organizations interested in obtaining a detailed description or copy of the quality system, or in attending one of the workshops, should contact the author at jons@strucom.co.uk.

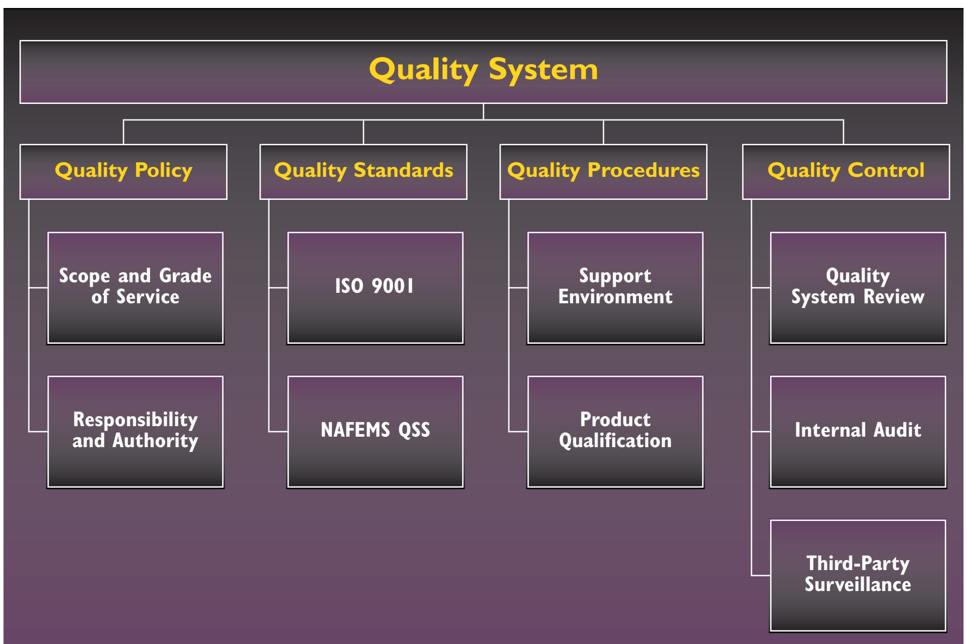


Table I

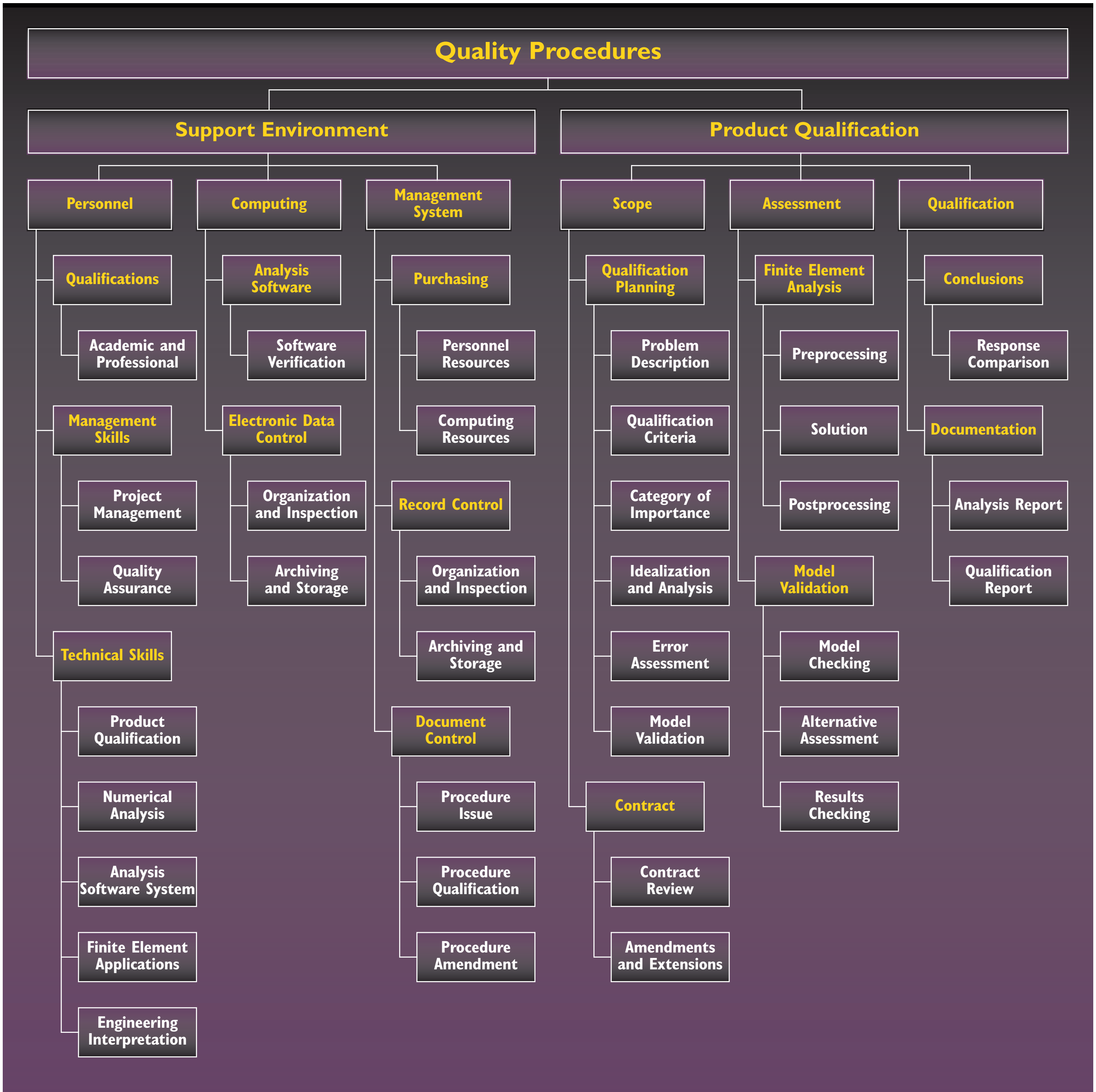


Table 2

References:

1. Smith, J.M., '*Quality Assurance Procedures for Product Qualification*', STRUCOM, 1996.

2. ISO 9001 : 1994, Quality Systems - *'Model for quality assurance in design/development, production, installation and servicing',* International Organization for Standardization, 1994.

3. Quality System Supplement to ISO 9001 *'Relating to Numerical Analysis in The Design and Validation of Engineering Products'.* Rev. 1.3, NAFEMS, 1993.

4. *NAFEMS Registered Analyst Scheme,* NAFEMS, 1996.

5. *SAFESA Management Guidelines,* NAFEMS, 1995.

by J. M. Smith, Director
Structures and Computers Ltd.
Surrey, England

Jen Valachovic, Marketing Specialist
ANSYS, Inc.

Bringing Simulation to Surgery:

Orthopaedic Surgeons Use ANSYS to Improve the Quality of Life

Surgeons and researchers at Pittsburgh's Center for Orthopaedic Research (COR) at Shadyside Medical Center and the Center for Medical Robotics and Computer-Assisted Surgery (MRCAS) at Carnegie Mellon University (CMU) are using ANSYS design analysis software to study the outcome of joint replacement operations prior to surgery. ANSYS technology makes it possible to evaluate different options and find the best surgical plan for each patient.

Led by orthopaedic surgeon Dr. Anthony M. DiGioia III, in collaboration with engineer Dr. Branislav Jaramaz from COR, the research team is developing a computerized surgical simulator that will predict the outcome of patient-specific hip replacement procedures. Their goal is to customize the joint replacement process so it works perfectly every time. Design analysis software from ANSYS, Inc., which enables surgeons to simulate physical behavior using a computer, forms the foundation of the system.

Joint replacement surgery, typically performed to restore movement to an arthritic joint, is one of the most successful procedures in medicine because it relieves pain and permits patients to return to productive lifestyles. However, there are times when joint replacements need to be redone because of loosening or wear.

A method that evaluates factors of an operation prior to surgery, such as the type of implant required for a patient, correct placement, and the best way to install and secure an implant, would enhance the conventional joint replacement process immensely. Surgeons currently cannot address these issues except by trial and error, and clinical experience that can take years to develop. Yet, their decisions may determine the success or failure of these operations.

Each year, orthopaedic surgeons in the United States perform 250,000 hip replacement operations, at a cost of approximately $25,000 each. Worldwide, the number of these surgeries is close to 800,000. If the procedure has to be repeated, costs run between $50,000 and $75,000.

Computer simulation promises that by studying the outcome of a joint replacement operation through a software program prior to surgery, surgeons can evaluate different options and find the best surgical plan for each individual patient. This would greatly reduce the number of repeat joint operations and tremendously increase the patient's quality of life.

Why Hips

In a hip replacement operation, the top portion of the patient's femur (upper leg bone) is removed. This is the "ball" portion of the ball-and-socket hip joint. The cup within the pelvis into which the ball fits is also resurfaced. These diseased bones are replaced with artificial devices called implants. The rounded end of the femoral implant fits into a new socket in the pelvis, similar to the way the original, diseased bone did (Figure 1).

A hip replacement operation is one of the more difficult joint replacement operations for several reasons. Bone geometry and mechanical properties vary widely from patient to patient. Many surgical parameters may vary as well. For example, there are different types and sizes of implants and different ways to prepare the bone during surgery.

Research at Shadyside Hospital and CMU is presently focused on one particular type of hip replacement procedure – the cementless, press-fitted implant. This implant accounts for about one-third of all femoral replacements and nearly all of the socket replacements.

This technique relies on bone growing into the porous implant over time to hold it in place. However, to achieve initial sta-

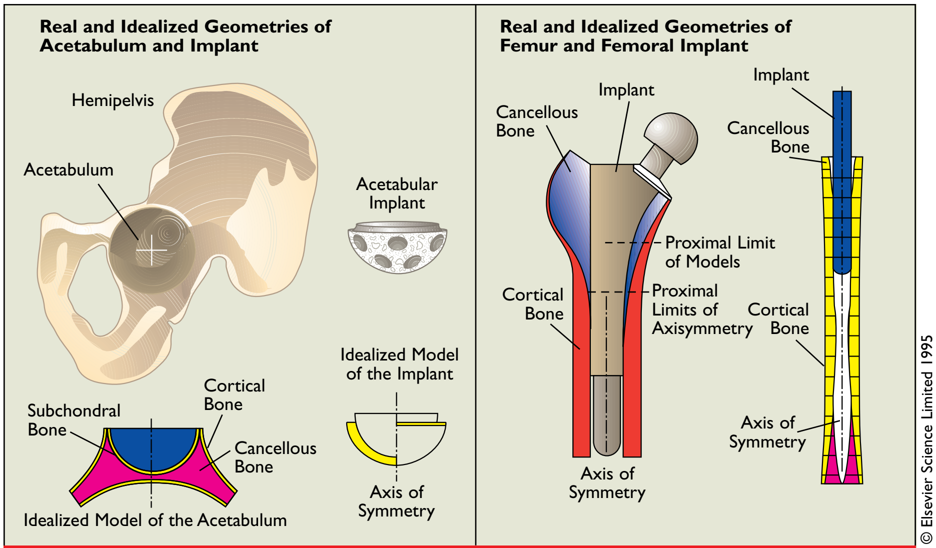


Figure 1

Images of hip replacement components.

bility, the surgeon selects an implant that is slightly oversized with respect to the cavity cut into the pelvis bone. The relative sizes of the implant and the bone cavity are critical. If the cavity is too small, the surrounding bone may deform when the implant is pushed into place, leading to fractures (Figure 2). On the other hand, if the cavity is too big, the implant will be loose and bone growth into the implant will not occur. This will cause the implant to loosen and cause pain. In fact, gaps as small as 0.25mm between the implant and bone have been shown to prevent bone ingrowth.

Installation force is another factor that complicates this procedure. To get the implant into the smaller cavity, substantial force is required. Consequently, some deformation of the bone occurs. Too much force increases the risk of cracking the bone. On the other hand, if not enough force is used, the implant may not be situated properly, resulting in residual gaps in bone ingrowth.

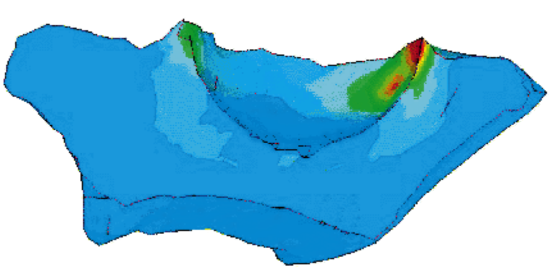


Figure 2

Cross-section cut through a 3D model of a hip socket.

Ideally, a computerized surgical simulation system will let surgeons resolve these issues in advance. The research team at Shadyside and CMU has studied the mechanics of normal, diseased, and replacement joints in its efforts to create such a system. Their goal is to give surgeons a tool that lets them determine clinically meaningful issues such as proper implant fit, the ideal amount of bone to remove when creating the cavity for the implant, and how the bone will react to the installation. Ultimately, they hope to develop a system to predict the long-term behavior of the implant, as well.

Simulation Method

The first step in simulating a hip replacement operation is acquiring geometric representations of both the patient's bone and the implant (Figure 3). The geometry of the implant is easy to obtain. Most vendors model the device in a CAD system and this data can be read into ANSYS very easily.

Obtaining a geometric representation of the patient's bone is somewhat more difficult, but critical because this data is what makes a simulation patient-specific. Currently, the patient's CT scan is converted to a solid model using published algorithms. The solid model is then converted to IGES format for input into the ANSYS program. Although this process works well, the Shadyside and CMU research team strives to improve the conversion of CT data to solid models — creating algorithms that are able to differentiate between different types of bone tissue.

Once the necessary solid models are available, the next step is preparing them for analysis. This process includes converting them to meshed finite element models, specifying material properties, and applying boundary conditions.

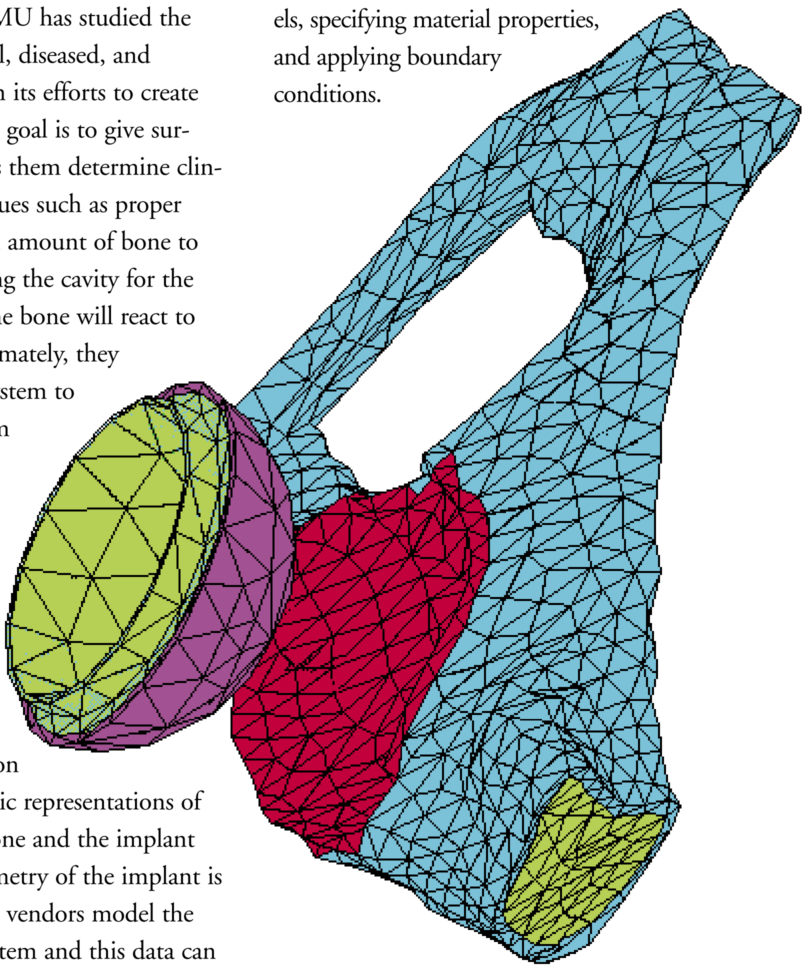


Figure 3

3D model of pelvis.

Why ANSYS

According to Dr. DiGioia, *"The research team chose the ANSYS program for our work because the finite element method can reproduce the complexity of biological structures, joints, and materials. In addition, the bone-implant system exhibits highly nonlinear behavior, both in terms of geometry and materials. The nonlinear nature of the problem is one reason surgeons' jobs are so difficult, and why there is such a need for a surgical simulator. Results of actions in the operating room, since they behave nonlinearly, are not always intuitive. Not all finite element element (FEA) packages can perform nonlinear analyses as strongly as ANSYS."*

ANSYS supports a certain type of element that is critical to understanding contact between bone and implant. This is the "point-to-surface" contact element introduced in ANSYS 5.0. This type of element is needed because the position of the implant immediately after surgery cannot be predicted. It must be obtained by a finite element analysis of the interference fit between bone and implant. Most FEA programs provide only "point-to-point" contact elements that require that the location of interference be known before the analysis begins.

As the first step towards a full-fledged patient-specific hip replacement simulator, the Shadyside and CMU researchers developed a nonlinear, axisymmetric, contact-coupled finite element model of the implant and the acetabulum (portion of pelvis that contacts the implant). Although the geometry of the acetabulum was idealized to reduce the complexity of the model, other factors were included to add realism to the analysis such as nonlinear material properties, large deformations, and general frictional contact coupling.

In the initial analysis, the implant was placed outside the bone and directed toward the prepared cavity. Incremental displacements, which in effect drove the implant into the cavity, were applied to the implant to simulate the forceful insertion that occurs during surgery.

The analysis showed radial strains and hoop strains in the bone for the four steps of the implantation procedure. This particular simulation predicted significant strains in the bone and a large gap between the implant and the bone in the polar region (Figure 4).

Researchers were also able to perform parametric studies to evaluate the influence of implant size because model geometry was described using several basic geometric parameters. The results made it clear that larger implant oversizing may result in excessive strains, especially in the periphery of the acetabulum.

These results, unavailable to surgeons in the past, give valuable information about the immediate postoperative state of the bone-implant system. The clinical consequences of these findings imply probable cracking of the bone in the areas of large strain and reduced possibility of bone ingrowth into the porous area of the implant. Researchers also learned that the assembly strains in the bone due to implant insertion can be as much as an order of magnitude larger than those caused by normal joint loads.

Putting FEA to the Test

FEA software, such as the ANSYS program, is used to meet a broad range of technical needs, typically including helping companies design better products – everything from jumbo jets to cars to electronic equipment. The combined efforts of the surgeons, engineers, and researchers at Shadyside Medical Center and Carnegie Mellon University are putting FEA to the test to bring the power of design analysis to the medical industry. This extraordinary medical application means ANSYS is being used to directly improve the quality of people's lives.

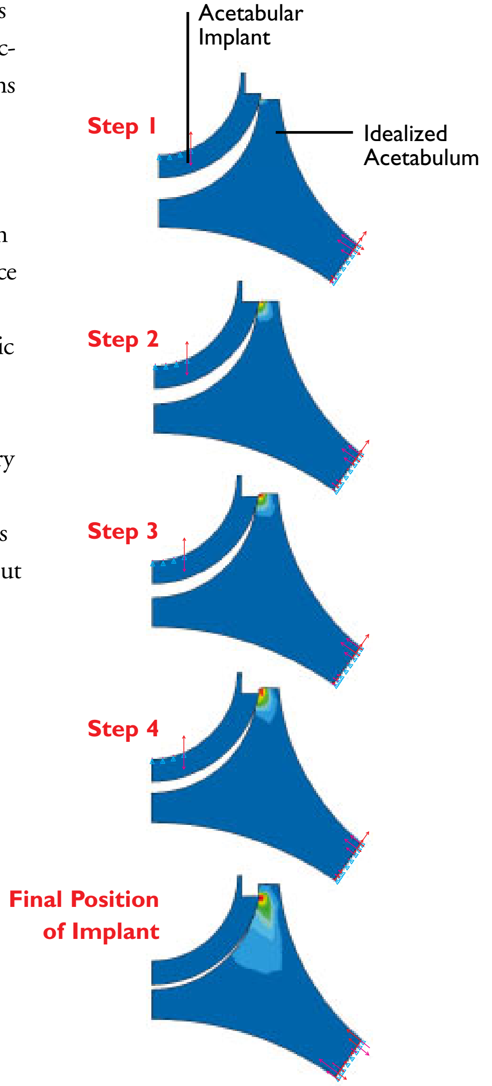


Figure 4

Step history of implant being placed in hip socket.

by Caren Potter, Freelance Writer
McKinleyville, CA

Process-Centric Engineering:

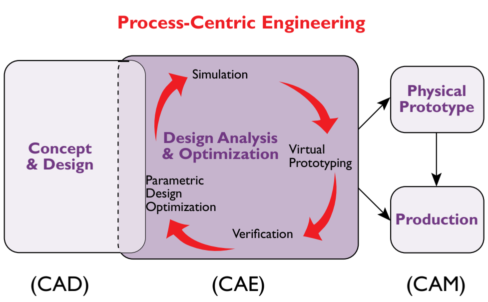
The Changing Role of Analysis

Intense competitive pressure in today's manufacturing market is driving companies to closely examine their approach to product development. To shorten time-to-market and lower costs, many firms are using analysis earlier in the product development cycle and feeding results to design much faster.

By using computer-aided engineering (CAE) in close conjunction with computer-aided design (CAD), designs can be studied and optimized in the computer before prototype testing. Not only are prototypes costly and time-consuming to build and test, but design flaws uncovered late in product development are expensive to correct because of significant up-front investments. Moreover, designs are usually not optimal because of quick fixes often done at the last minute just to get the product out the door. The hassle of prototype testing leads many manufacturers to delay critical analysis until design flaws surface on the shop floor.

CAE tools such as finite element analysis (FEA), kinematics, and dynamic simulation are used to perform "what-if" studies of various design alternatives, simulate product behavior, spot potential problems early, and optimize the design before any hardware is built. Prototype testing then becomes a way of validating the design, often with only a single series of tests.

The fast-reaction use of CAE and CAD in this manner is sometimes referred to as process-centric engineering, since product development revolves around the continuous design analysis and optimization process rather than a linear sequence of separate steps. Companies trying to implement process-centric engineering often find that programs not adapted to rapid integration and iterative processes early in the design cycle do not adequately support their new requirements. The process-centric approach requires fast and flexible programs.



Advanced software aimed more toward design analysis and optimization overcomes the limitations of point solution software. These analysis packages are powerful enough for dedicated users who need high-level functions for solving complex design verification problems. The scope of problems handled is not restricted to specific disciplines such as structural analysis. Instead, these packages have capabilities for handling coupled-field analysis where the effects of multiple physical forces, such as structural and thermal, are determined simultaneously.

Advanced analysis packages also have intuitive user interfaces so that design engineers can readily employ analysis routinely in their work. To fully integrate CAD and CAE tools for up-front design optimization, process-centric packages may utilize common data models instead of exchanging and translating data. In this approach, parametric design systems have emerged as one of the most effective ways for rapid, intuitive modification of the product model without compromising basic design concepts. Parameters facilitate the integration of analysis and CAD software for highly interactive design analysis and optimization.

Another important element in process-centric engineering is the trend toward more collaborative efforts involving interdisciplinary teams of design, analysis, and manufacturing engineers working together throughout the product design-to-manufacturing process. In this way, those ordinarily left out until late in the process can provide valuable input throughout product development and avoid costly downstream mistakes. At many companies, the culture shock of making such organizational changes is considerably more challenging than implementing the technology underlying process-centric engineering. However, the impressive results manufacturers have achieved with this approach demonstrates that these implementation efforts are unquestionably worthwhile. ■

by John Krouse, Industry Analyst
Concord, OH

ASD Locations

North America

Advanced Engineering Technologies, Inc.
6525 The Corners Parkway, Suite 400
Norcross, GA 30092
Attn: Gerry Jones
Internet: aeta@aol.com
Telephone: (770) 409-7880
Fax: (770) 409-7881

Computer Aided Engineering Associates, Inc.
398 Old Sherman Hill Road
Woodbury, CT 06798
Attn: Dr. Lawrence Durocher
Internet: caea@pipeline.com
Telephone: (203) 263-4606
Fax: (203) 266-9049

Computer Aided Engineering Associates, Inc.
7 Berkmans Street
Worcester, MA 01602
Attn: Terry Davis
Internet: tdconsult@aol.com
Telephone: (508) 791-2925
Fax: (508) 791-5477

Computer Aided Engineering Associates, Inc.
475 West End Avenue, Suite V2
North Plainfield, NJ 07060
Attn: Dave Siwiec
Internet: caenj@aol.com
Telephone: (908) 412-1622
Fax: (908) 412-1894

Computer Aided Engineering Group
5240 Poinsettia Avenue
Winter Park, FL 32792
Attn: Richard Knapp
Internet: rvfk@aol.com
Telephone: (407) 677-8005
Fax: (407) 677-8770

Concurrent Engineering Corp.
Riverview Office Tower
8009 34th Avenue South
Suite 1291
Bloomington, MN 55425
Attn: Jeff Schrantz
Internet: schrantz@concurrent.com
Telephone: (612) 854-4499
Fax: (612) 854-4491

DRD Corporation
5506 South Lewis Avenue
Tulsa, OK 74105
Attn: Chris Andersen
Internet: canderse@drd.com
Telephone: (918) 743-3013
Fax: (918) 745-9037

Defiance-STS/SMC
Troy Technology Park
1960 Ring Drive
Troy, MI 48083
Attn: Debbie Vuckovich
Internet: defiance@usa.pipeline.com
Telephone: (810) 583-7117
Fax: (810) 583-0008

Engineering Cybernetics, Inc.
1856 Lockhill Selma, Suite 105
San Antonio, TX 78213-1558
Attn: Dr. Jack Maison
Internet: info@ecisa.com
Toll Free: 1-800-707-5461
Telephone: (210) 341-5461
Fax: (210) 341-5493

Engineering Cybernetics, Inc.
16350 Park Ten Place
Suite 220
Houston, TX 77084-5148
Attn: Bill Lloyd
Internet: info@ecihou.com
Toll Free: 1-800-707-9798
Telephone: (713) 398-7041
Fax: (713) 398-7097

Engineering Methods, Inc.
Purdue Research Park
1211 Cumberland Avenue
West Lafayette, IN 47906
Attn: Jean Terry
Internet: emisup@holli.com
Telephone: (317) 497-4477
Fax: (317) 497-4776

JLR Computer Analysis, Inc.
111 S.E. Everett Mall Way
Building E, Suite 201
Everett, WA 98208
Attn: Jim Radochia
Internet: jpradochia@aol.com
Telephone: (206) 353-8089
Fax: (206) 353-593

Jordan, Apostal, Ritter Associates, Inc.
35 Belver Avenue
North Kingstown, RI 02852
Attn: Peter Kingman
Internet: kingman@jar.com
Telephone: (401) 884-3014
Fax: (401) 294-3826

Jordan, Apostal, Ritter Associates, Inc.
33 Boston Post Road West, Suite 270
Marlborough, MA 01752
Attn: Mark Kerrigan
Internet: kerrigan@jar.com
Telephone: (508) 485-8100
Fax: (508) 485-3123

MCR Associates, Inc.
111 West Evelyn Avenue, Suite 301
Sunnyvale, CA 94086
Attn: Mark Rodamaker
Internet: info@mcrfea.com
Telephone: (408) 736-1636
Fax: (408) 736-1664

Mallett Technology, Inc.
121 Hillpointe Drive, Suite 300
Canonsburg, PA 15317-9502
Attn: Brian Ball
Internet: bdb@pitt.mallett.com
Telephone: (412) 746-7000 Ext. 16
Fax: (412) 746-7001

Mallett Technology, Inc.
5 Great Valley Parkway, Suite 120
Malvern, PA 19355-1436
Attn: Sarah McCahill
Internet: slm@phila.mallett.com
Telephone: (610) 889-1340
Fax: (610) 889-1341

Mallett Technology, Inc.
Laurel Technology Center
14900 Sweitzer Lane, Suite 202
Laurel, MD 20707-2915
Attn: Mike Butz
Internet: mpb@wdc.mallett.com
Telephone: (301) 725-0060 Ext. 11
Fax: (301) 725-0061

Mallett Technology, Inc.
100 Park Drive, Suite 204
P.O. Box 14407
Research Triangle Park, NC 27709-4407
Attn: Kenn Yarina
Internet: key@rtp.mallett.com
Telephone: (919) 406-1500 Ext. 10
Fax: (919) 549-9679

Ohio Computer Aided Engineering, Inc.
1612 Georgetown Road
Hudson, OH 44236
Attn: Rufus Blackley
Internet: ocae@aol.com
Telephone: (216) 650-2057
Fax: (216) 656-4024

Stress Technology, Inc.
1800 Brighton-Henrietta
Town Line Road
Rochester, NY 14623
Attn: Mike Rieger
Internet: mrieger@stresstech.com
Telephone: (716) 424-2010
Fax: (716) 272-7201

SSC, Inc.
18700 Beach Boulevard, Suite 200-210
Huntington Beach, CA 92648
Attn: Dr. J. W. Jones
Internet: ssc@kaiwan.com
Telephone: (714) 964-1552
Fax: (714) 964-1156

SSC, Inc.
1S443 Summit Avenue, Suite 304B
Oakbrook Terrace, IL 60181
Attn: Tom Humrickhouse
Internet: ssc2@ix.netcom.com
Telephone: (708) 261-0246
Fax: (708) 261-0406

South America

SMI-Software Marketing International LTDA
Av. Brigadeiro Faria Lima,
613 - 10 Andar Cj. 101/2
São Paulo, CEP 01451-000 S.P.
BRAZIL
Attn: Jose Ricardo Nogueira
Internet: smisp@embratel.net.br
Telephone: 55-11-820-0388
Fax: 55-11-820-7361

SMI-Software Marketing
International Ltda.
Rua da Assembleia 10, 3701
Rio de Janeiro, CEP 20119-900
BRAZIL
Attn: Renaldo Pereira Nunes
Telephone: 55-21-531-2337
Fax: 55-21-531-2388

Europe

AT&T ISTEL
2, rue du Docteur Lombard
92441 Issy-les-Moulineaux-Cedex
FRANCE
Attn: Dominique Bonnecase
Telephone: 33-1-4662-2626
Fax: 33-1-4662-6578

AT&T ISTEL Rhône-Alpes
Les Bureaux Verts
22, rue Benoît Bennier
69260 Charbonnieres
FRANCE
Attn: Pierre Lovat
Internet: 100446.1577@compuserve.com
Telephone: 33-78-44-25-25
Fax: 33-78-87-62-33

Anker-Zemer Engineering A/S
P.O. Box 10 Voksenskogen
N 0708 Oslo 7
NORWAY
Attn: Jan Christian Anker
Internet: a-ze@oslonett.no
Telephone: 47-22-921-625
Fax: 47-22-142452

Anker-Zemer Engineering AB
P.O. Box 156
S-691 23 Karlskoga
SWEDEN
Attn: Eva Hyvärinen
Internet: 100140.2132@compuserve.com
Telephone: 46-586-52820
Fax: 46-586-56470

Bercom Software Development and Consultancy, Ltd.
88 Gissin St.
Kiryat-Arie
Petah Tikva 49222
ISRAEL
Attn: Amit Cohen
Internet: bercom@goldnet.net.il
Telephone: 972-3-9233522
Fax: 972-3-9233088

CAD-FEM GmbH
Marktplatz 2
D-85567 Grafing/München
GERMANY
Attn: Dr. Günter Müller
Internet: gmueller@cadfem.de
Telephone: 49-8092-70050
Fax: 49-8092-700577

CAD-FEM GmbH
Büro Hannover
Neue Torstr. 3
D-31303 Burgdorf
GERMANY
Attn: Clemens Groth
Internet: cgroth@cadfem.de
Telephone: 49-5136-85717
Fax: 49-5136-873179

CAD-FEM GmbH
Büro Stuttgart
Heidenheimer Str. 5
D-71229 Leonberg
GERMANY
Attn: Uli Göhner
Telephone: 49-7152-42084
Fax: 49-7152-72984

CAD-FEM GmbH
Büro Chemnitz
Cervantesstraße 89
D-09127 Chemnitz
GERMANY
Attn: Dr. Ulrich Stelzmann
Telephone: 49-371-267060
Fax: 49-371-742106

CAD-FEM AG
Büro Schweiz
Wittenwiler Str. 25
CH-8355 Aadorf
SWITZERLAND
Attn: Markus Dutly
Internet: 101447.1344@compuserve.com
Telephone: 41-52-611723
Fax: 41-52-612546

Figes, Ltd.
Petek-Bozkaya Is Merkezi c/304
Celal Bayar Cad. No. 6
16220 Bursa
TURKIYE
Attn: Dr. Tarik Ögüt
Internet: figes@uu20.bim.uludag.edu.tr
Telephone: 90-224-2508454
Fax: 90-224-2508457

I.N.A.S.
1, Tehnicii Street
1100 Craiova
ROMANIA
Attn: Radu Tircomnicu
Telephone: 40-051-410719
Fax: 40-51-410715

Ingeciber S.A.
Avda. Monforte de Lemos, 189
28035 Madrid
SPAIN
Attn: Honorato Girau
Internet: 100763.1224@compuserve.com
Telephone: 34-1-386-22-22
Fax: 34-1-386-45-80

ITALCAE SRL
Via Panciatichi, 40
50127 Florence
ITALY
Attn: Antonio Mancino
Internet: italcae.@fi.pisoft.it
Telephone: 39-55-432010
Fax: 39-55-4223544

SVS FEM s.r.o.
Cechynska 16
657 01 Brno
CZECH REPUBLIC
Attn: Miroslav Starek
Internet: jsch@svs-fem.anet.cz
Telephone: 42-5-45214572
Fax: 42-5-45214572

Structures and Computers Ltd.
Strucom House
188-196 Canterbury Road
Croydon CR0 3HF
Surrey, ENGLAND
Attn: Stewart Morrison
Internet: stewartm@strucom.co.uk
Telephone: 44-181-683 3999
Fax: 44-181-683 3933

Structures and Computers Ltd.
Strucom House
40 Broadgate, Beeston
Nottingham NG9 2FW
ENGLAND
Attn: Alan Perry
Internet: 100321.2052@compuserve.com
Telephone: 44-1159 677622
Fax: 44-1159 677620

Structures and Computers Ltd.
Postbus 323
3700 AH Zeist
THE NETHERLANDS
Attn: Jurgen Mathijssen
Internet: strucom@euronet.nl
Telephone: 31-30-6928800
Fax: 31-30-6924921

Structures and Computers Ltd.
18 Windsor Place
Lr. Pembroke St.
Dublin 2
REPUBLIC OF IRELAND
Attn: Derek Sweeney
Internet: strucomi@irishoff.win-uk.net
Telephone: 353 1 676 3765
Fax: 353 1 676 3766

Structures and Computers Ltd.
2 Haw Street
Wotton-under-Edge
Gloucestershire GL12 7AQ
ENGLAND
Attn: Dr. Tim Brown
Internet: tbrown@strucom.co.uk
Telephone: 44-1453-521655
Fax: 44-1453 521755

Asia Pacific

CAD-IT Consultants Pte. Ltd.
629 Alijunied Road #08-04/05
Cititech Industrial Building
SINGAPORE 1438
Attn: Florence Tan
Internet: cadit@singnet.com.sg
Telephone: 65-741-7527
Fax: 65-741-8761

CEMA International Corp.
38-C/2, Central Commercial Area
P.E.C.H.S., Karachi-75400
PAKISTAN
Attn: Amir Abbas
Internet: ideas@biruni.erum.com.pk
Telephone: 92-21-455-4794
Fax: 92-21-455-4794

Cybernet Systems Co., Ltd.
Nissay Otowa Bldg. (6-9 Floors)
15-6, Otsuka 2-chome, Bunkyo-ku
Tokyo 112
JAPAN
Attn: Shigehisa Inoue
Internet: sinoue@cybernet.co.jp
Telephone: 81-3-5978-5400
Fax: 81-3-5978-5440

Cybernet Systems Co., Ltd.
Nihon-Seimei Midosuji Bldg.
4-2-4, Minami Senba, Minamiku
Osaka, 542
JAPAN
Attn: Takao Daimyo
Internet: daimyo@cybernet.co.jp
Telephone: 81-6-241-5241
Fax: 81-6-241-1255

IMAG Industries, Inc.
IMAG Beijing Office
Peking Hotel, Room 1222
33 East Changan Avenue
Beijing 100004
PEOPLE'S REPUBLIC OF CHINA
Attn: Wang Bohua
Internet: imagbj@public.bta.net.cn
Telephone: 86-1-0513-7766 Ext. 1222
Fax: 86-1-0513-9945

IMAG Shanghai Office
Int'l. Equatorial Office Building
Room 409, 65, Yanan Road West
Shanghai 200040
PEOPLE'S REPUBLIC OF CHINA
Attn: Ning Jie
Internet:
shanghai.imag.sh@shanghai.shspt.chinamail.
sprint.com
Telephone: 86-21-248-8144
Fax: 86-21-248-8534

IMAG Industries, Inc.
Unit F/G, 18th Floor
New Times Plaza
42 WenWu Street, Xinhua Road
Chengdu, Sichuan, 610017
PEOPLE'S REPUBLIC OF CHINA
Attn: Zhao Fang-yuan
Telephone: 028-6788319
Fax: 011-86-28-6789608

IMAG Asia Pacific Limited
Room 406, Wing On Plaza
62 Mody Road, Tsimshatsui East
Kowloon
HONG KONG
Attn: Paul Chang
Internet: sylchan@imag.com.hk
Telephone: 852-2721-4083
Fax: 852-2722-5637

JIN Young Technology, Inc.
3rd Floor, Daechi Building
#965-16 Daechi-dong
Kangnam-ku Seoul
KOREA
Attn: Jin Park
Internet: ANSYS@hitel.kol.co.kr
Telephone: 82-2-565-2015
Fax: 82-2-568-2250

Neil Automation Technology Ltd.
1201/C/4, 'Shivtirth', F.C. Road
Shivajinagar
Pune 411 004
INDIA
Attn: Ketan Bakshi
Internet: shirish.neil@sml.sprintrpg.sprint.com
Telephone: 91-212-321443
Fax: 91-212-328537

NIIT
Engineering &
Geomatics Solutions Group
NIIT Ltd., NIIT House
C-125 Okhla Phase I
New Delhi 110 020
INDIA
Attn: B. Sridhar
Internet: bsridhar%niit@iris.ernet.in
Telephone: 91-11-681-7341
Fax: 91-11-681-7344

Taiwan Auto-Design Co.
10F, 635-2 Chung Hsin Rd.
Sec. 5, San Chung City
Taipei Hsien, Taiwan
REPUBLIC OF CHINA
Attn: Nerow Yang
Internet: tadc1378@ms1.hinet.net
Telephone: 886-2-9958036
Fax: 886-2-9957792

Australia & New Zealand

Worley Limited
Level 17, 300 Flinders Street
Melbourne, Victoria 3000
AUSTRALIA
Attn: Dr. Joseph Wong
Internet: jwong@worley.com.au
Telephone: 61-3-9205-0500
Fax: 61-3-9205-0505

Canada

H.G.E. Inc.
400 Carlingview Drive
Toronto, Ontario
CANADA M9W 5X9
Attn: Tony Firmin
Internet: tonyf@hge.com
Telephone: (416) 674-8505
Fax: (416) 674-8520

H.G.E. Support Associate
Compusim, Inc.
1003-D 55th Avenue N.E.
Calgary, Alberta
CANADA T2E 6W1
Attn: Imad Tabsh
Internet: tabshi@cuug.ab.ca
Telephone: (403) 295-2912
Fax: (403) 274-6908

H.G.E. Support Associate
Farnell-Thompson Applied Technologies
2222, René-Lévesque O., Suite 1
Montréal, Québec
CANADA H3H 1R6
Attn: Philippe Vidori
Internet: farthom@cam.org
Telephone: (514) 937-2644
Fax: (514) 937-4966

Mexico

SSC de Mexico S.A.
Mesones #1, Int. 10
37700 San Miguel de Allende
Guanajuato, 37710
MEXICO
Attn: Fernando Balderas Lopez
Internet: sscmex@mail.mindvox.ciateq.mx
Telephone: 52-415-24279
Fax: 52-415-24279

Republic of South Africa

Wolhuter & Van Wyk
Consulting Engineering, CC.
Potchefstroom University for CHE
Dept. Of Mechanical Engineering
2520, Potchefstroom
REP. OF SOUTH AFRICA
Attn: Arno Wolhuter
Internet: mgiaw@puknet.puk.ac.za
Telephone: 27-148-299-1324
Fax: 27-148-299-1320

Seminar Schedule

The following information represents a partial listing of ANSYS seminars and the dates the seminars will be presented. For complete details on the seminars listed below, contact the ANSYS Support Distributor (ASD) shown for that particular seminar. Contact your ASD if you are interested in a seminar not listed here. For seminars held at ANSYS, Inc., contact the Training Registrar at 412.873.2882. Reservations are recommended at least two weeks in advance.

DATE (Week of)	Introduction to ANSYS	Design Optimization	Dynamics	ANSYS/ProFEA	Heat Transfer	Introduction to FLOTRAN	Structural Nonlinearities	Solid Modeling	Electomagnetics	Special Topics (See footnotes)
June 30-July 6							CAD			SMI[8]
July 7-13	CEC JLR EMI CSC ECI JIN	DEF			CSC DEF			EMI CAD	CSC	ANS[3]
July 14-20	ANS CSC SMI					CSC		ECI MTI		ANS[22] MTI23 CSC[22]
July 21-27	DEF DRD		MTI	CSC DRD		ANS	EMI			CSC[22] EMI[2]
July 28-Aug. 3	CSC MTI	SMI				SMI	CSC			
Aug. 4-10	ANS ECI MCR DRD EMI SSC		CEC		CSC MTI SMI SSC		DEF	DRD EMI		ANS[22] MTI[2] CAEA[3]
Aug. 11-17	CSC	ANS	SSC		ANS	CSC STI	ECI EMI SSC			ANS[3] CSC[22] EMI[2]
Aug. 18-24	SSC JIN		ANS DEF	CSC DRD	EMI SSC	ECI	CEC	CSC SMI		ANS[2] DRD[25]
Aug. 25-31	CSC DEF	EMI					SSC JIN		ANS	SMI[17]
Sept. 1-7	CSC		CSC			CSC				
Sept. 8-14	ANS CSC EMI MTI CEC ECI JLR STI	DEF	MTI JIN	EMI MTI				CSC		ANS[22] SMI[8]
Sept. 15-21	CAEA DRD JIN SMI CSC SSC STR			DRD	ECI	ANS	MTI			CAEA[6] CSC[22] MTI[24] SMI[26]
Sept. 22-28	CSC SMI DEF			CSC		CSC	ANS		EMI	CAEA[2] DEF[22] ANS[2] CSC[22] SMI[18]

Company Key

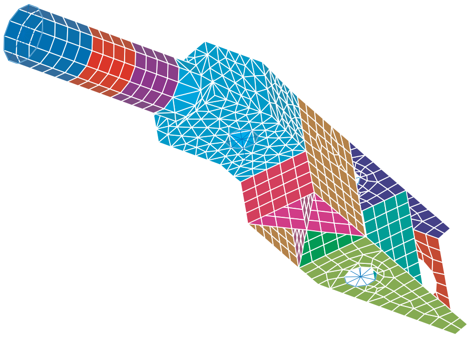
ANS ANSYS, Inc.
CAD CAD-FEM GmbH
CAEA Computer Aided Engineering Assoc. Inc.
CEC Concurrent Engineering Corp.
CSC Cybernet Systems Co., Ltd.
DRD DRD Corporation
DEF Defiance-STS/SMC
ECI Engineering Cybernetics, Inc.
EMI Engineering Methods, Inc.
JIN JIN Young Technology, Inc.
JLR JLR Computer Analysis, Inc.
MTI Mallett Technology, Inc.
SMI SMI-Software Marketing International LTDA
SSC SSC, Inc.
STI Stress Technology, Inc.
STR Structures and Computers Ltd.

Special Topics Key

1 Creating the Finite Element Model
2 Explicit Dynamics with ANSYS/LS-DYNA
3 ANSYS 5.3 Update
4 Acoustics
5 Hyperelasticity
6 Substructures
7 Random Vibrations (PSD)
8 Finite Element Methods
9 p-Method Analysis
10 Contact Surfaces
11 Basics of Numerical Analysis
12 Error Estimation
13 FEA Solvers
14 Fracture Mechanics
15 Shell Elements
16 Buckling Analysis
17 IGES Transfer
18 Electric Motors
19 Mechanical Vibration Using ANSYS
20 ANSYS Parametric Design Language (APDL)
21 Updating FEM
22 Introduction of ANSYS-Part II (Intermediate)
23 ANSYS for CADDS
24 ANSYS/ED
25 Advanced ANSYS/ProFEA
26 Fatigue

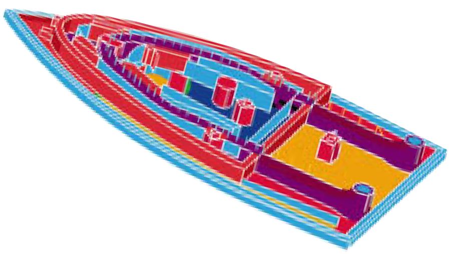
Behind it all, there's ANSYS®

Motion Picture Industry



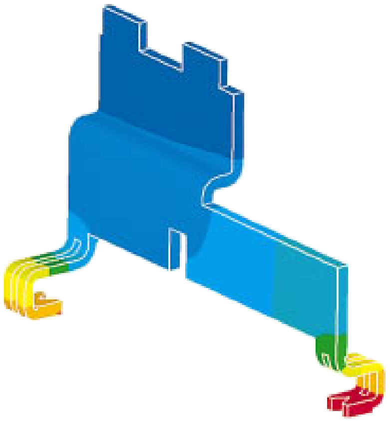
Imax Corporation is a leader in giant-screen film technology. Imax uses advanced equipment to precisely control the film, a critical factor in the clarity of its images. Imax engineers use ANSYS software to optimize and improve the design of their equipment, such as projector components.

Consumer Appliances

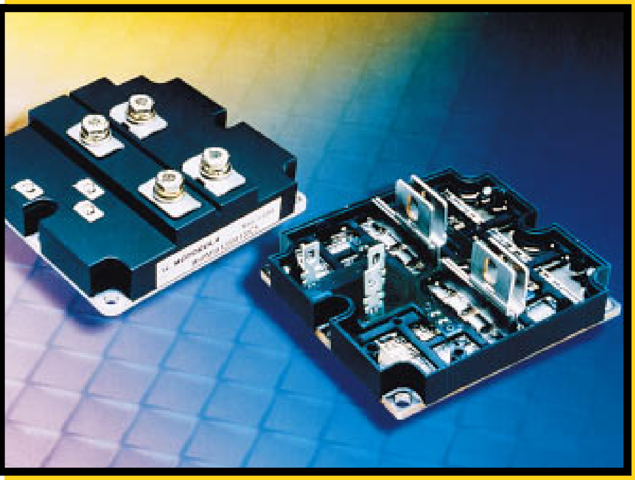


Black & Decker uses advanced technology in the development of its new commercial and household appliances. For instance, the company's engineers used ANSYS software to reduce the weight of the SurgeXpress iron, while increasing its steam output.

Electronics Packaging



Motorola's Semiconductor Product Sector manufactures components that are used in tens of thousands of products. ANSYS is one of Motorola's simulation tools used by many engineers for thermal, coupled-field thermal/electric, viscoplastic, static, and dynamic analyses to ensure products resist damaging heat build-up while withstanding vibration, shock, and temperature changes.



ANSYS, Inc.
201 Johnson Road
Houston, PA 15342-1300

ansysinfo@ansys.com
T 412.746.3304
F 412.746.9494
Toll Free USA and Canada:
1.800.WE.R.FEA.1
Toll Free Mexico:
95.800.9373321

Regional Offices:
North America
lzera@ansys.com
T 810.585.5020
F 810.585.5730
International
jtung@ansys.com
T 412.873.3086
F 412.746.9699
Europe
bbutcher@ansys.com
T 44.1.734.880.229
F 44.1.734.880.360

http://www.ansys.com

MAN-AN2-5/96

Printed in U.S.A.