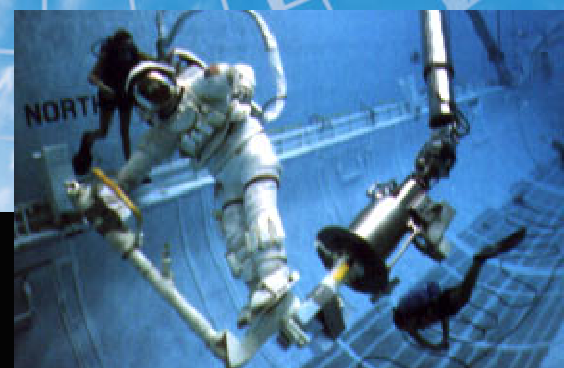
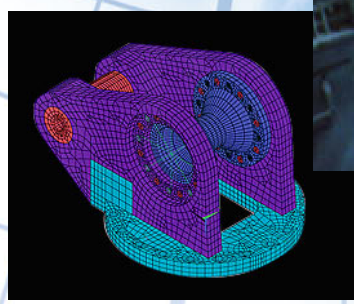
DesignSpace Redefines "Computer-Aided Design"





See related article on page 17.

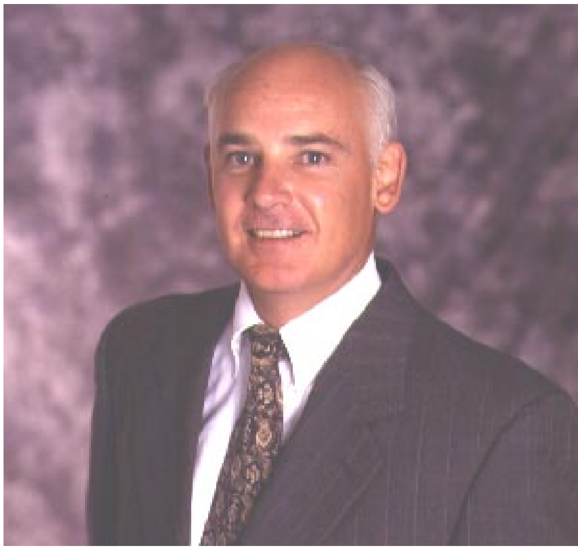
ANSYS News

Third Issue 1996

INSIDE THIS ISSUE

Completing the IPO:

ANSYS, Inc. Stock Trades on the Nasdaq National Market



Peter J. Smith
Chairman and CEO

On June 20, 1996, ANSYS, Inc. common stock began trading on the Nasdaq National Market under the symbol ANSS. This Initial Public Offering (IPO) marked an important evolutionary step for our company. For the past 26 years, ANSYS, Inc. has been privately held. Operationally, the company has always been profitable; with increasing sales, income, and investment in research and development. In the past couple of years, ANSYS, Inc. has restructured; building a strong, reinvigorated management team to lead our company into the future. Our growth rate has rapidly accelerated, and we have significantly expanded our activities to include increased product functionality, strong support of the Windows environment, an intuitive graphical user interface, and an accelerated product release schedule.

Going public means that, for the first time, our company's securities are sold to the public through the Nasdaq National Market. The ANSYS, Inc. IPO was underwritten by four notable investment banking firms including Alex. Brown & Sons, Inc.; Cowen & Company; Wessels, Arnold & Henderson; and Parker/Hunter. It generated more than $41 million for the company.

The IPO is a significant milestone for ANSYS, Inc., its employees, and its customers. It represents the completion of a very long and detailed process that involved a lot of work from many different groups of people; most important the engineers, developers, and support people responsible for the development of ANSYS® technology. We are very proud of our products and services; and, as a company, we continually strive to achieve higher standards for ourselves and our users. The successful IPO is a reflection of these achievements and a recognition of our position as a worldwide industry leader.

I would like to take this opportunity to briefly summarize the key benefits of the ANSYS, Inc. IPO to you, our users. The funds generated by the IPO have strengthened our balance sheet and improved cash flow. This allows the company to invest aggressively in research and development to maintain technological leadership in the ANSYS product set and fund strategic initiatives, such as our market-leading CAD-integration program and our DesignSpace™ development environment. We are continually expanding sales, marketing, and the support of ANSYS distributors to ensure revenue and market share growth. These factors make ANSYS, Inc. even stronger and better able to remain the stable, technology innovator you have come to rely upon.

As a customer, you should experience very little change in the ANSYS, Inc. you have trusted for years. You will, however, have a greater assurance that the company has the backing required to meet your long-term technology and support needs. Now that we're a publicly traded company, you also have the option to monitor our progress as frequently as you wish through quarterly financial updates as well as public documentation provided under the rules of the Securities Exchange Commission (SEC). This is an exciting new start for us at ANSYS, Inc., and we look forward to our continued working relationship with you.

Sincerely,

Peter J. Smith
Chairman and CEO

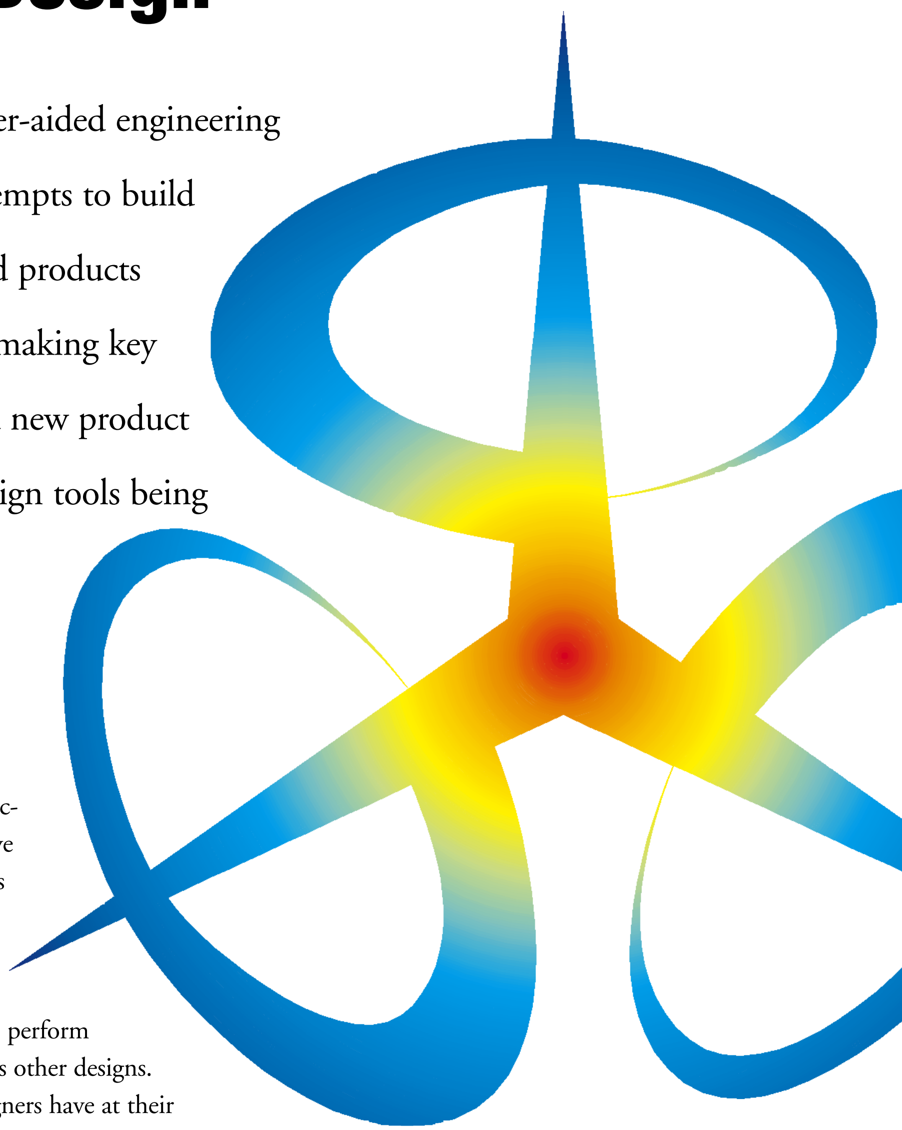
ANSYS® News®

Third Issue 1996

***ANSYS News* is now available on the ANSYS WEB: 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.

DesignSpace Redefines "Computer-Aided Design"



Computer-aided design (CAD) and computer-aided engineering (CAE) companies alike continue in their attempts to build successful finite element analysis (FEA)-based products that target design engineers. The benefits of making key design decisions early in the design cycle of a new product are clear. Yet we do not see these types of design tools being utilized on most design engineer's desktops.

Why not? Prices are too high, CPU performance too low, and the industry is confused by the products that are available. The biggest issue is that software developers are trying to put analysis tools on the designer's desktop. Products based on ANSYS DesignSpace Technology distinguish themselves from other "design analysis" tools by truly considering who the designer is, and addressing the problems a designer seeks to solve.

Industry software vendors state that the criteria for an "Analysis for Design" product are:

- Have a user interface that looks familiar to the CAD user;
- Work off of the CAD geometry database;
- Support the same computer platforms as the CAD design system;
- Simplify the analysis system for use by a designer.

These criteria certainly form a good starting point, but they fail to address the problems the designer needs to solve.

Consider the designer, who spends a work-day designing parts. The objective is to achieve aesthetic shapes that fit into a predetermined environment and promise to perform favorably versus other designs. The tools designers have at their disposal are computer-aided modeling and imaging systems, books defining design policy, an engineering education (complete with a shelf of reference books and textbooks), the experience gained in past practices, and a calculator. Designers use textbook approximations and manual calculations to "guesstimate" an optimum design for a particular part.

But, what if the modeling system could embody all of the engineering principals and knowledge contained in the designer's office, and could easily give answers specific to the physics of the model the designer is building? Herein lies the objective of DesignSpace.

DesignSpace is not an FEA program. DesignSpace is a design engineering tool that replaces the traditional stress, deflection, and frequency hand calculations with reliable and proven ANSYS FEA. In DesignSpace, engineering problems are worked in the same way as they're worked on paper, only the software handles the details of managing the complex internal mathematical models.

By fully automating the FEA systems inside the software, DesignSpace provides meaningful design information in the most natural and intuitive way, as quickly as possible, up-front in the design process where the information is most useful in decision making. DesignSpace does not seek to replace prototyping and testing. In fact, prototyping and testing will always be needed in the development of quality products. Also, DesignSpace is not an appropriate alternative to advanced analyses performed in an FEA program by an experienced analyst. DesignSpace, as with any other design tool, requires care and engineering judgment on the part of the designer to be used properly, and should be viewed as an important component in an overall strategy for quality design engineering.

In contrast, ANSYS/Multiphysics™ (also available from ANSYS, Inc.) is a complete FEA program. The ANSYS/Multiphysics user has enormous control over the internal mathematical model, how a solution is achieved, and how results are presented and interpreted. ANSYS/Multiphysics provides a vast array of analysis options, and allows the user to assess and minimize the amount of potential error in the solution. While FEA programs provide a wealth of information and control, they require a significant amount of time and expertise on the part of the user, and are not ideally suited for providing "up-front" design information.

Stated another way, the designer asks questions concerning a product's:
Form — What shapes provide the best tradeoff between performance and aesthetic value?
Fit — How does the part perform within its assembly and within its environment?
Function — Does the part satisfy criteria for weight, strength, frequencies, etc.?

Whereas, the analyst is concerned with:
Failure — Will the design function properly under real-world operating conditions?

The designer's decisions are usually of a relative nature: "Which of these two designs performs better?" or "Can I achieve relatively similar performance with a lighter design or a cheaper material?" Whereas the analyst thinks in more absolute terms: "Will this part perform in the real world?" One of the great pitfalls companies encounter in trying to bring more robust tools to the design process is to allow the analyst to be involved in choosing the designer's software.

To better understand how DesignSpace is unique, consider the general process of using the DesignSpace Technology as it is implemented in the ANSYS/AutoFEA™ 3D–Validation product.

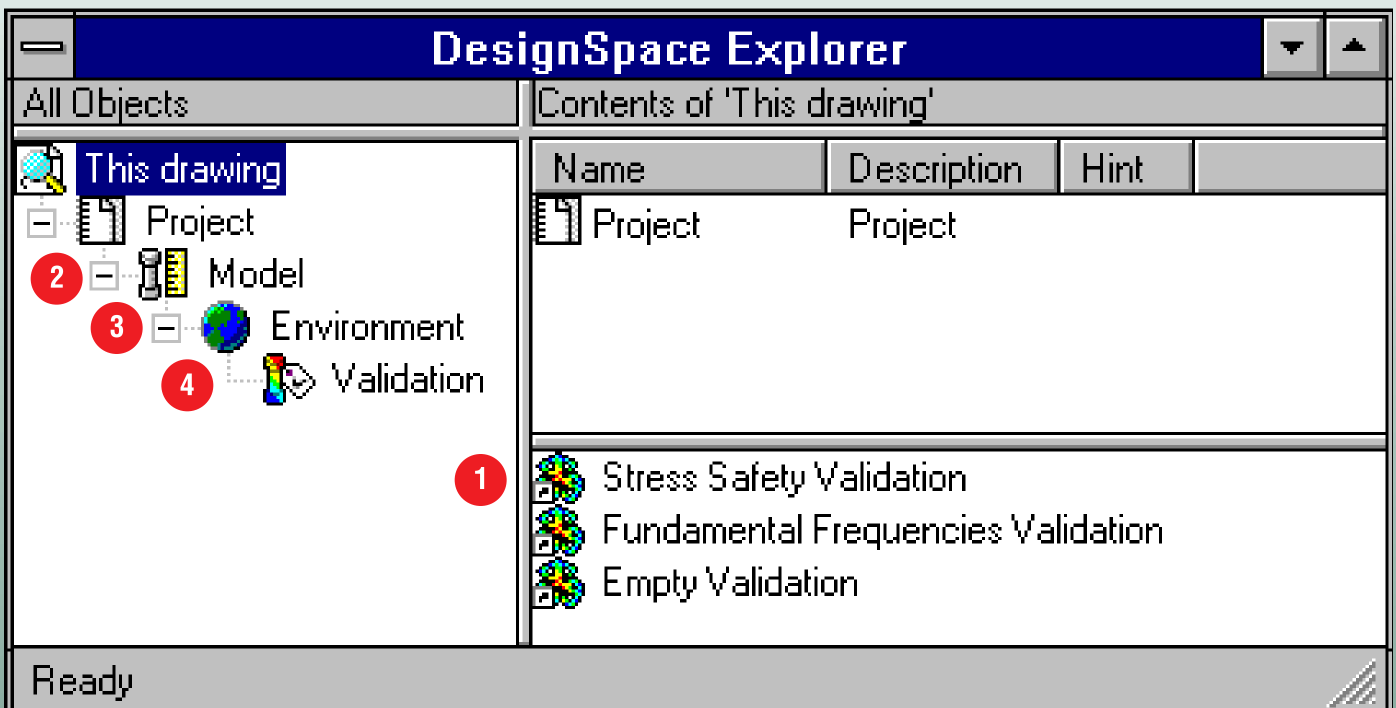


Figure 1

The DesignSpace Explorer displaying the different steps to the validation process.

DesignSpace™ TECHNOLOGY

Step 1:

Start a design project.

Drag-and-drop a "shortcut" to establish a basic framework of design information. The shortcut includes some common problem information that you can expand on to suit your needs.

Step 2:

Define a "model."

Pick a solid from the CAD drawing, then choose a material from a customizable library. Choose between speed and accuracy for your answers via a slider bar, if you so desire. The far left of the slider (speed) is appropriate for demonstration, training, and determining the natural frequencies of a part. The far right position (accuracy) is for achieving design answers that are relatively close (+/- 20 percent) to real-world performance of the part. The default is a compromise between the two that should be chosen if the purpose of the tool is to compare the relative performance of different part designs.

Step 3:

Build the "environment."

The properties of the environment focus on the global environment the part lives in. Here, you may set environmental factors such as gravity and ambient temperature. In addition, you may characterize how the part interacts in its local environment. Holes and other geometric features form the basis of the loads placed upon the part. The designer communicates with the DesignSpace tool in a language that is native to the design process. Loads are associative to the designer's solid part, meaning that parametric geometry changes are automatically accounted for in subsequent validations.

Step 4:

Find the answers.

Once a model and its environment are defined, select "Find Answers" and let DesignSpace take over.

Step 5:

Communicate the results.

The types of answers DesignSpace generates are unique as well. DesignSpace interprets results from the ANSYS engine

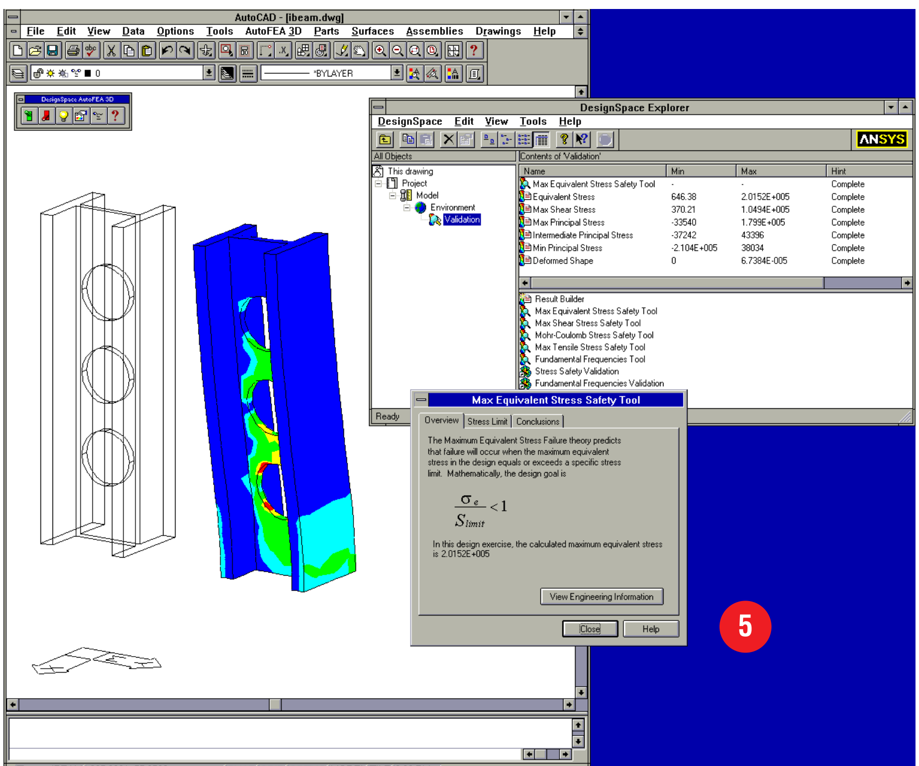


Figure 2

ANSYS/AutoFEA 3D is an implementation of DesignSpace Technology. The above image shows the results of a validation.

into useful information the design engineer is looking for. For example, stress and strain, though accessible, may not be the most useful information for the designer. Instead, the designer may seek answers to questions such as “What is my factor of safety?” or “Where is this part most likely to fail?” and “Under peak expected conditions, does the part deflect to violate other parts in the assembly?”

DesignSpace communicates results in many different ways. For example, the designer can obtain factors of safety for the design and plot the factor of safety to see where the design is relatively weak or strong. DesignSpace also allows calculation and viewing of natural frequencies and their “shapes.” Stress values, interpreted using a variety of design theories, can be computed and displayed. DesignSpace also supports results output in a virtual reality modeling language (VRML) format, allowing communication and collaboration across networks. Future releases will see the addition of fatigue tools, thermal design tools, fully automated report generation, etc.

DesignSpace redefines the meaning of “Computer-Aided Design” to include unique tools that provide:

- Abstraction from the underlying technology;
- Geometry and feature-based loading;
- An intuitive user interface created for the design engineer;
- Incorporation of the technology as an extension to the designer’s CAD system;
- Internet ready applications for concurrent engineering;
- Results interpretation for design engineering principles;
- Self-teaching tutorials and interface complemented by a comprehensive, accessible, easy-to-understand, online manual system (Figure 3); and

> **DesignSpace Validation uses a simple and intuitive user interface that focuses on design problems to create an unprecedented level of productivity for design engineers. The introduction of DesignSpace Technology marks the start of a new era in the art of mechanical design.**

- Upward compatibility with full analysis systems, complementing the enterprise-wide design solution.

DesignSpace Validation uses a simple and intuitive user interface that focuses on design problems to create an unprecedented level of productivity for design engineers. The introduction of DesignSpace Technology marks the start of a new era in the art of mechanical design.

by Peter T. Svenneby, Director OEM Marketing
ANSYS, Inc.

Tom Shadle, Development Engineer
ANSYS, Inc.

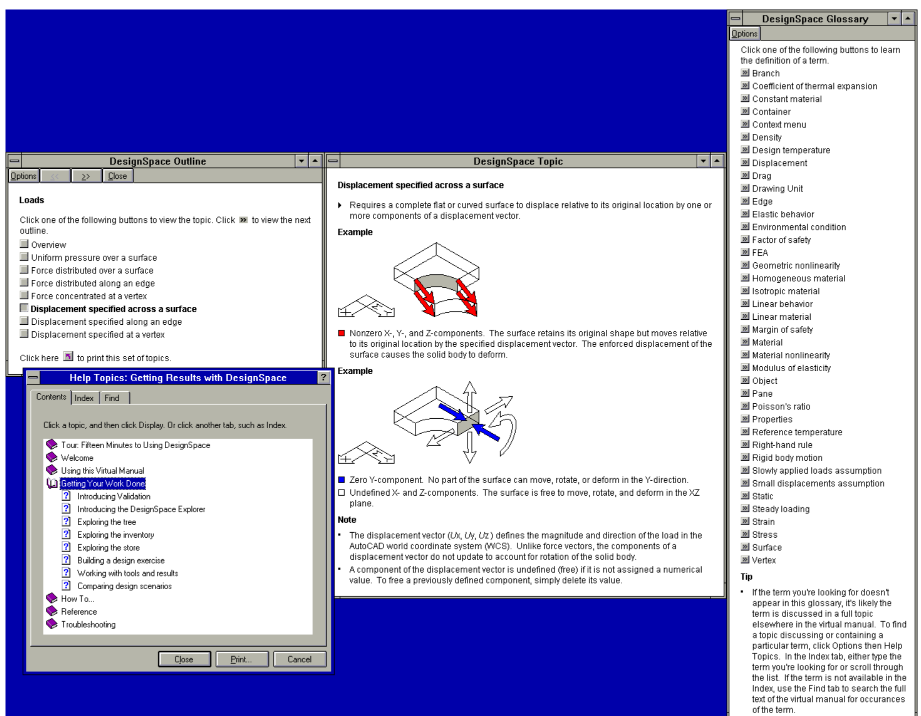


Figure 3

The DesignSpace online manual system.

Revolutionary Product Creation:

Using ANSYS to Meet Market Demands

Companies require advanced software that breaks traditional boundaries to create a revolutionary product. With the ANSYS family of products, companies receive sophisticated, multiphysics design analysis programs that help implement a process-centric engineering system and bring products to market faster.

Process-centric engineering enables companies to move beyond the traditional linear product development cycle to a more efficient, continuous, and collaborative design analysis and optimization process. This approach requires fast and flexible programs, such as those offered by ANSYS, Inc., that provide a wide range of analyses, including coupled-field capabilities where the program simultaneously determines the effects of multiple physical forces.

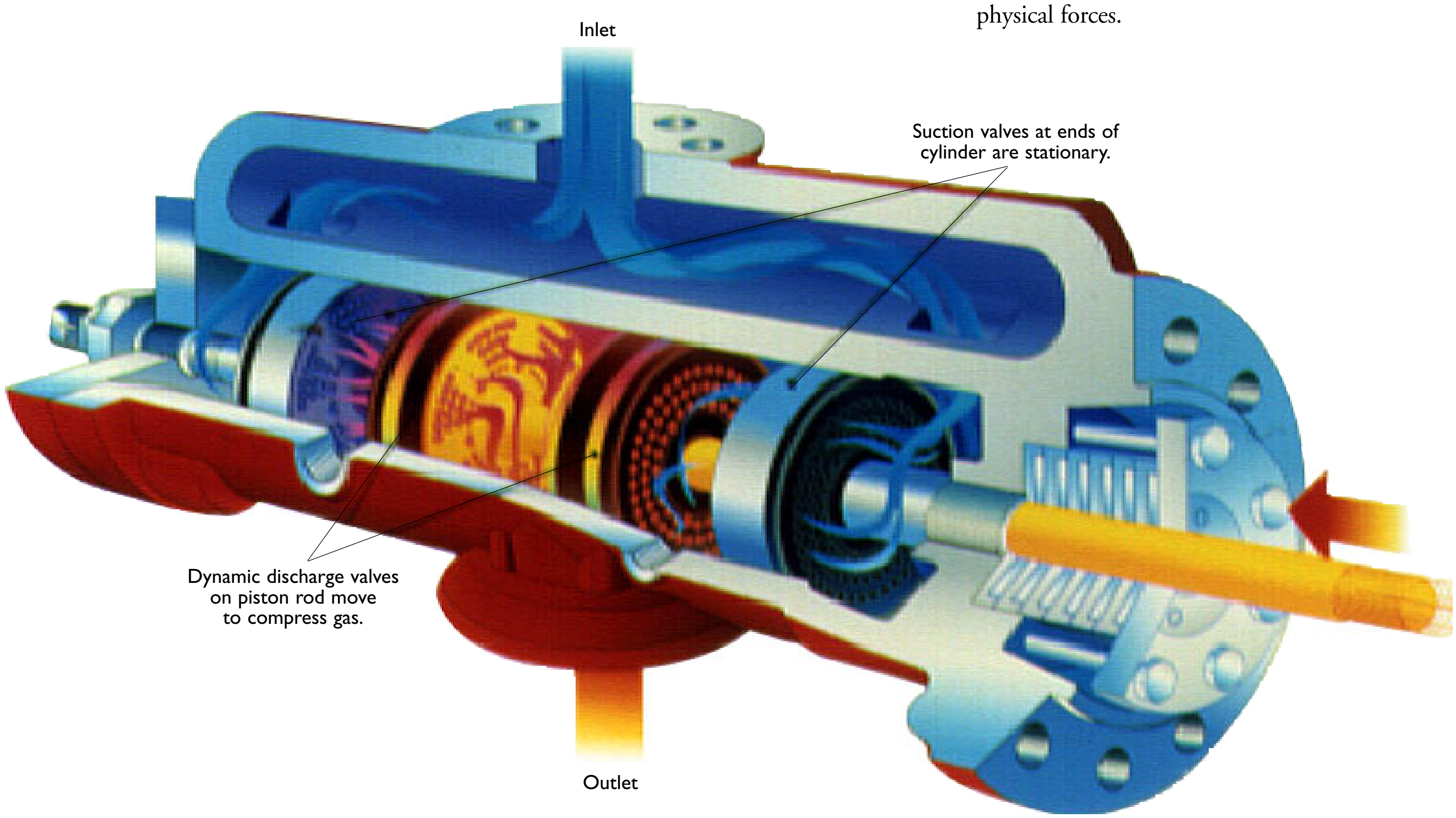


Figure 1

Simplified illustration of Dresser-Rand's valve-in-piston gas compressor.

Dresser-Rand Customers Ask for More

World-class companies, such as Dresser-Rand, use ANSYS to produce leading-edge products that keep their customers asking for more. For example, Dresser-Rand used ANSYS finite element analysis (FEA) software to respond quickly to requests for a new gas compressor. When their new valve-in-piston (VIP) gas compressor hit the market, it was an immediate success. The VIP design was a radical departure from the traditional compressor design (Figures 1 and 2). Rather than having suction and discharge valves built into the cylinder head, the VIP compressor incorporates valves directly into the piston. This simpler design, with fewer moving parts, created a gas compressor that was less expensive to operate and easier to maintain.

Dresser-Rand initially introduced a single model of the revolutionary VIP compressor but, since it was so popular, the company wanted to release ten additional models (covering a range of pressure ratings) as quickly as possible. Normally, this process is lengthy because Dresser-Rand physically tests design changes before releasing new products to market. Building and testing the original VIP prototype took six months. Testing ten additional prototypes, even if much of it was done concurrently, would have meant missing the window of opportunity for this product.

To reduce the time needed to get new compressor models to market, Dresser-Rand engineers evaluated design changes using ANSYS technology and performed physical tests on only two of the ten models. Ajay Garg, a senior mechanical engineer in Dresser-Rand Company's Engine Process Compressor Division (Painted Post, NY), estimates that this approach made it possible to evaluate the ten new models in six months, compared to two to three years of physical testing.

"Because FEA was used during the design of the VIP compressor and the results correlated with physical tests, we felt confident about evaluating variations on the original design with FEA alone," said Garg. "Within one day, we would change the model, run an analysis, and determine whether the revised design met our criteria for fatigue life." The key to the one-day turnaround was the use of 2D structural analysis after Dresser-Rand engineers performed an initial 3D analysis.

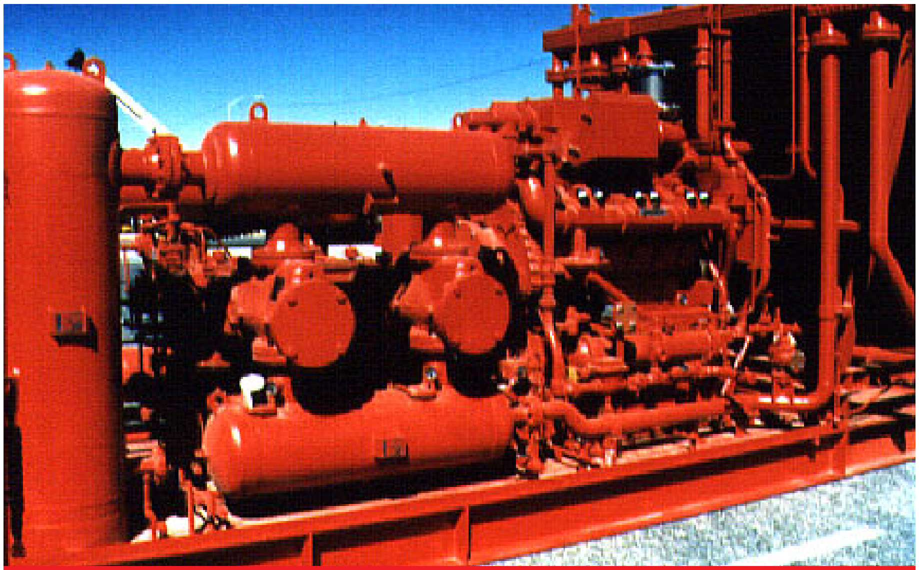


Figure 2

Photo of an assembled compression unit, including the VIP compressor.

FEA from Start to Finish

Dresser-Rand's Engine Process Compressor Division, the division that designed the VIP compressor, makes large, slow-speed reciprocating gas compressors, slow-speed integral engines, and high speed reciprocating gas compressors for companies involved in gas distribution and oil refining.

ANSYS has been part of the division's new product development process for several years. Dresser-Rand chose the ANSYS program because of ANSYS, Inc.'s proven record of excellent quality control and well-supported revisions. Dresser-Rand also feels that ANSYS offers some of the best solver technology on the market, providing comprehensive solution capabilities and robust elements. They use the software for structural, thermal, and vibration analysis. The company applies analysis results early in the design cycle to evaluate new compressor and engine designs, then again later when a design is changed to study the effects of the change.

Garg summarizes the use of FEA this way: "We use it at first to make sure a machine works for the designated application. Analysis results then become the basis for future development of that machine. Marketing might ask for an increase in operating conditions, from 1,000 psi to 1,500 psi; or manufacturing might ask us to replace a 1/8-inch fillet with a 1/16-inch fillet. Since test results for the original design are correlated with FEA, we don't have to run tests for every subsequent design change. This shortens our development cycle and reduces costs as well."

Engineers perform early analyses long before a design has been defined as a CAD model. "These analyses are done to ensure that a concept is valid for a particular market, or that it can withstand the specified operating conditions," said Garg. When users perform analyses at this stage of the design cycle, ANSYS' parametric modeling capability defines analytical models. This feature makes it easy to alter the model for what-if studies, or to evaluate design changes later in the production cycle. For example, the process of sensitivity checking is expedited with parametrics because users don't need to generate a whole new input file when they modify a model, such as when changing wall thickness.

Dresser-Rand normally performs a full series of physical tests for every new product. Cost and time limit the use of physical testing in an evaluation of the final design. It can be difficult or impossible to check stress levels in certain areas of a compressor where it is not possible to place strain gauges in physical testing. Dresser-Rand's solution has been to instrument the prototype as thoroughly as possible and use FEA to identify stresses in areas that they cannot test experimentally. "This ensures the engineers that they don't miss any hot spots or potential problems," Garg said.

Determining Cylinder Size

Initial cylinder sizes ranged from 3.5-inches in diameter to 15-inches in diameter. Dresser-Rand's goal was to create a VIP compressor lineup capable of varying pressure and flow conditions. To do this, they needed to produce a range of cylinder sizes. Changing cylinder size meant changing components of the VIP mechanism. When a size change is made, the results don't necessarily scale by the same amount; so Dresser-Rand, using ANSYS FEA, made sure that the compressor would meet operating requirements with all cylinder sizes.

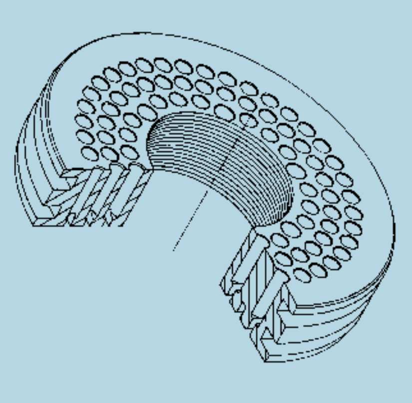


Figure 3

Cross section of a drilled and milled plate.

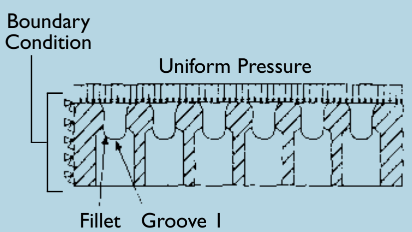


Figure 4

Fillet at the innermost groove.

The company's engineers evaluated the circular steel plate that acted as the piston, specifically its response to stress, at all the different sizes. The piston was a uniformly loaded plate containing drilled holes and milled grooves (Figure 3). It deformed as a cantilevered plate. The engineers expected to see maximum stresses at the 0.03-inch fillet of the innermost milled groove (Figure 4).

The plate operates under cyclic loads. Garg and his colleagues first performed a structural analysis of a 3D finite element model of the plate that included the holes and grooves. This 46,176 degrees of freedom model took about three weeks to create with parametrized geometry and meshing with brick elements (Figure 5, see page 10).

Results of the analysis showed that bending the plate as a cantilever created compressive stresses at the holes and tensile stresses at the milled grooves (Figure 6, see page 10). These stresses at the milled grooves were chosen as the governing factor for the design but, since the finite element model did not contain the radii of the grooves, there was a singularity at the point of interest. Engineers optimized the design (increased plate thickness) to reduce tensile stresses by minimizing compressive stresses at the surface of the drilled holes. They assumed that the percentage reduction in compressive stresses at the holes presented a similar stress reduction at the milled grooves.

From 3D to 2D Analysis

In the interest of analyzing the nine other plate sizes as quickly as possible, the engineers decided to investigate the possibility of replacing the 3D structural analysis with a 2D analysis. "In a static as well as a fatigue environment, tensile stresses are more harmful than compressive stresses," explained Garg. "Therefore, the drilled holes could be ignored and the geometry could be represented by a 2D axis-symmetric finite element model."

Parametric modeling of the 2D plate went very quickly. The model, which included the milled grooves, took about one hour to prepare for analysis. The analysis required less than two minutes of CPU time to run. The engineers correlated results of the 2D analyses for several plate sizes with testing. They redesigned some plates to improve reliability, and repeated the 2D analyses.

Prior to the development of the 2D model, the reliability factors were unknown and, consequently, some tests were unsuccessful [1]. (In Figure 7, the size of the plates are in ascending order from 3.5 inches to 20.5 inches.) To determine the reliability factor of the innermost groove, the engineers had to know the real

	Before 2D Modeling		After 2D Modeling	
Plate Diameter	Reliability Factor	Laboratory Results	Reliability Factor	Laboratory Results
A	>2	Successful		
B	2.0	Successful		
C	<1	Failed	2.0 (redesign)	Successful
D	1	Failed	>2 (redesign)	Successful
E			>2	Successful

Figure 7

Reliability factors and laboratory test results for the tested steel plates.

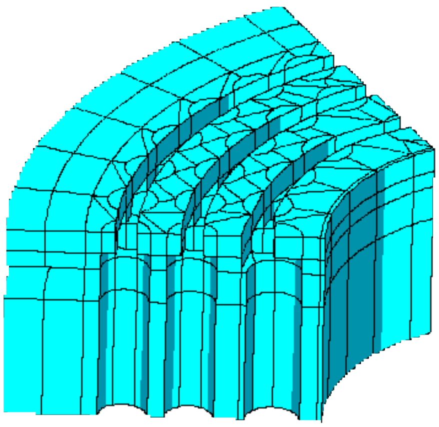


Figure 5

Parametric solid section of VIP.

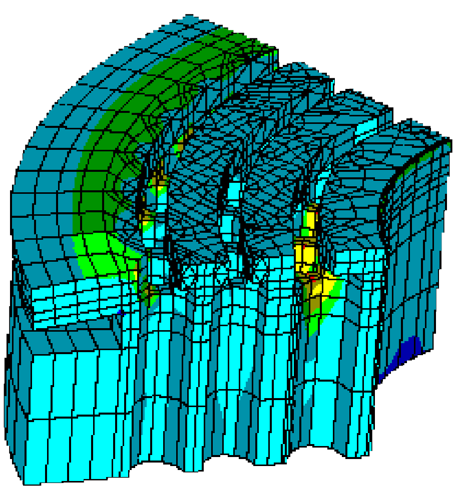


Figure 6

Stress profile under design loads.

stresses. After developing the 2D method, they discovered that some of the sizes had a reliability factor less then one. (It is desirable to have a reliability factor greater than one, possibly two.) They redesigned the failed plates to a reliability factor of two or more, and the plates tested successfully. The engineers used 2D FEA to analyze all additional plate sizes.

Garg cautions that corrections and compensations in 2D were critical to simulating 3D behavior and validating the analysis. The engineers also verified the 2D FEA method through analytical and experimental methods. Their efforts paid off. The success and failure of laboratory tests agreed with reliability factors. This correlation helped to verify the geometry, loads, and boundary condition assumptions in the finite element models. “Once we established the 2D analysis approach, it reduced the design and optimization cycles for the VIP compressor by an order of magnitude in cost and time,” said Garg. Dresser-Rand can now offer customers the most advanced and reliable way to compress gas, with revolutionary VIP compressors available in cylinder sizes to meet all needs.

Conclusion

Global competition in the marketplace is causing companies to rethink their traditional engineering process, and implement process-centric engineering to optimize productivity and bring their product to market quicker. A particularly effective way to address increasing market demands, as Dresser-Rand discovered, is to use ANSYS products up-front in the design cycle, as well as throughout the production process.

by Ajay Garg, Senior Mechanical Engineer
Dresser-Rand Company
Painted Post, NY

Jen Valachovic, Marketing Specialist
ANSYS, Inc.

References:

[1] Garg, Ajay, *“2D Finite Element Analysis of Engineering Components,”* American Society of Mechanical Engineers (ASME), 1995.

ANSYS Training Focuses on the User

ANSYS users have an impact on the development of training materials for ANSYS products. The Training Group, a division of Customer Services at ANSYS, Inc., has launched a curriculum of ANSYS training courses using the proven, industry-standard process of Instructional Systems Design (ISD), in which users play a central role in the design and development of the training. The basic principles of the ISD method are job-based training, task-oriented objectives, learner-centered training, and evaluation.

Training is job-based. Training focuses on necessary job skills. Courses are organized into lessons (small units of learning) that trace directly to users' job tasks.

Provide clear, task-oriented objectives. Trainees know where they are going and why. Students accomplish the stated lesson objectives, so they will never have to ask themselves "How well am I doing?"

Design training to be learner-centered. Participants learn through extensive practice and exercise sessions. Practice makes perfect.

Evaluate. ANSYS users are revisited months after job training and evaluated for improved job performance. This

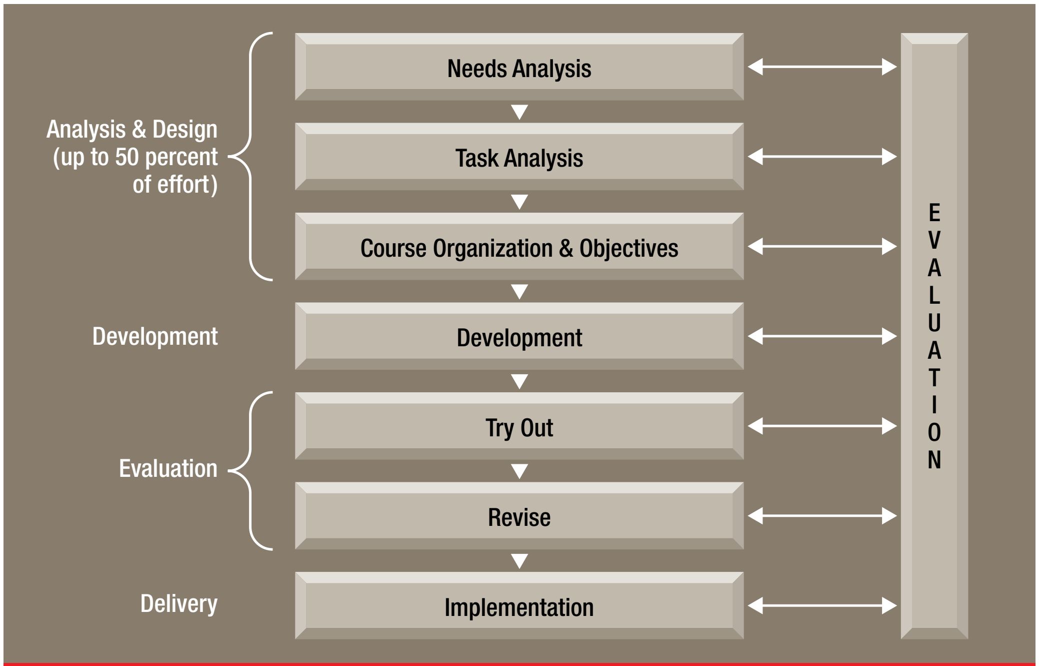


Figure 1

The ISD training method.

evaluation is most important to ANSYS, Inc. The success of ANSYS training is ultimately measured by job performance after the training.

Armed with these principles, ANSYS, Inc. instructional designers and course developers are designing instructional programs aimed not at teaching software product features, but at teaching skills that allow designers, engineers, and analysts to more effectively perform their jobs, using ANSYS as their tool. By learning more about what ANSYS users do "back at the office," course developers write training materials that reflect the "bigger picture." For example, in the *Introduction to ANSYS* course, participants learn not only *how* to perform an axisymmetric analysis in ANSYS [graphical user interface (GUI) operations and commands], but also *when* to use axisymmetry and potential cost-saving benefits of the technique. In the *Introduction to ANSYS Part II* course, users learn how to import a solid model into ANSYS, and what to do if the transfer is not perfect; in other words, common real-world problems and helpful diagnostic tips.

The ISD process used by ANSYS, Inc. provides a road map for designing and developing training materials (Figure 1, see page 11). The end-user is considered in every step; from the needs analysis, where needs are identified and training is determined to be the solution, to the final implementation of the training program. Evaluation takes place throughout the process. "Will this training course solve the original training need?" The answer to this question is always obtained by matching the course objectives to the skills required for end-users on the job. Training materials developed under the ISD method force consistency in the topics that instructors deliver in the training course, and consistency in learners' achievement of objectives. If every ANSYS instructor worldwide teaches with the same objectives, and every trainee can accomplish the objectives at the end of the training course, there will be consistent and correct usage of ANSYS programs.

The ANSYS Training Group places considerable attention on the selection of instructional media. The decision to design an instructor-led course versus a self-paced or video training course, for example, is based on the effectiveness and appropriateness of the various media to the learning tasks and the learners. The ANSYS Workbook is an excellent training tool designed for beginner or intermediate ANSYS users who want to run sample analyses, step-by-step at their own pace, on their own computer. In the future, ANSYS training will be available on interactive, computer-based training (CBT) programs.

The entire curriculum of ANSYS courses, along with the schedule of training classes at ANSYS, Inc. and all ASD sites, can be found on the ANSYS, Inc. Home Page (www.ansys.com) under Services.

by Julia O'Hara, Customer Services
ANSYS, Inc.

New Courses Added to the ANSYS Training Curriculum this Year Include:

Introduction to ANSYS Part II
A two-day follow-up course to *Introduction to ANSYS* for experienced ANSYS users that want training on advanced modeling, meshing, submodeling, modal analysis, and other advanced general topics.

ANSYS 5.3 Update
A one-day seminar detailing the new features offered in the ANSYS 5.3 family of products. Learn about Block Lanczos mode extraction, Fast Linear Solver technology, PowerDynamics, modeling and meshing enhancements, random vibration enhancements, explicit dynamics, non-Newtonian fluids, velocity effects in electromagnetics, and much more.

Explicit Dynamics with ANSYS/LS-DYNA™
A two-day course that is a necessity for any ANSYS/LS-DYNA user. Learn how to analyze highly nonlinear, dynamic phenomena including impact, crash, and drop-test analyses.

ANSYS for NASTRAN Users
This four-day course is designed to help MSC/PATRAN and NASTRAN users make the transition to ANSYS. Participants learn a wide range of ANSYS capabilities. Experienced MSC operators become efficient ANSYS users.

ANSYS WEB:

The ANSYS, Inc. Home Page, Growing to Meet Your Needs

The Internet is rapidly becoming one of the most effective vehicles for distributing and receiving information. Companies with Web sites reach a vast audience, creating marketing and sales opportunities. Potential clients find it easy and less bothersome to learn about a company and order their products directly through the Web. Web surfers can find just about everything they need via the Internet.

Currently, there are well over 11.7 million Web addresses available to visit. By the year 1999, it is estimated that the Internet will grow to more than 125 million users. The potential of this form of mass communication is why ANSYS, Inc. has made Internet development a major initiative, increasing not only the resources available but the personnel and hardware involved in creating a new site.

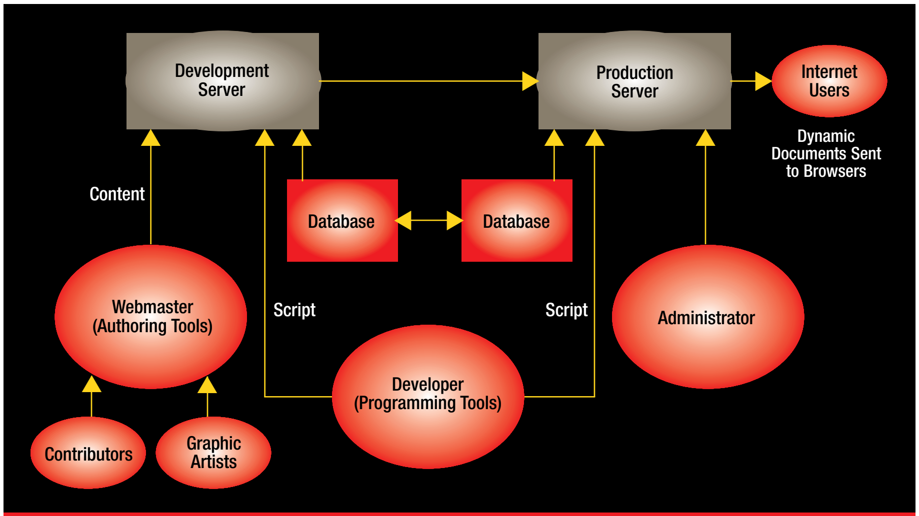


Figure 1
The new ANSYS WEB system.

This initiative involves building a restructured, state-of-the-art Web site providing a more organized and intuitive visit to the ANSYS WEB. The ANSYS, Inc. Home Page has always been viewed as an “elaborate and well-working site,” and continues to improve upon this reputation. A new ANSYS WEB offers visitors the easiest system possible for finding what they want quickly, through the use of technology including a specialized search engine, database storage of pages, Java-enabled Applets, and Shockwave enhancements.

The majority of Web sites available today contain only static pages, or already created pages that are not updated regularly. These pages reside on the browser, and users must click through pages to find what they need. With a large Web site, this could involve hundreds of pages and a lot of time. The ANSYS WEB contains few static pages, and uses mostly dynamic pages which are updated constantly, creating an interactive experience for the user. Now, thanks to the technological enhancements to the ANSYS WEB, users can create a virtual page. A virtual page is created by running a search on a topic. The search engine explores the storage database and finds all related information. When the search is complete, the user has several page options to choose from relating to a specific topic. This tailoring of the ANSYS WEB is called a virtual page environment. This is the foundation of the new ANSYS Home Page.

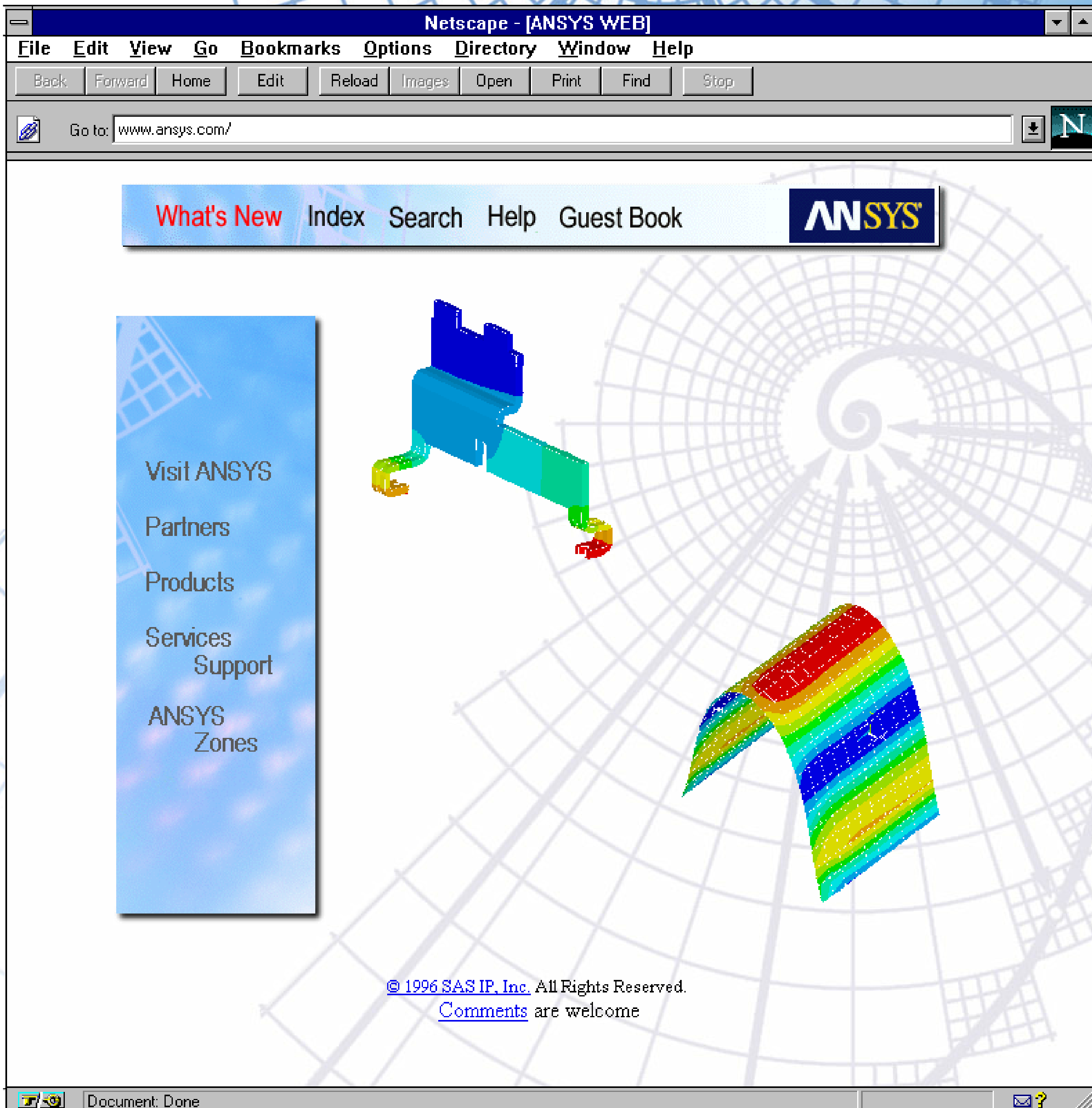


Figure 2

Here is how users will be greeted by the ANSYS WEB.

Restructuring

A set of goals was established to create the new ANSYS WEB. These goals included offering better organization of the information, more readily available information (to reach a broader audience), increased user- and browser-friendly functionality, easier system maintenance (quickly updated), and more interactivity. In developing the new ANSYS WEB to meet these goals, ANSYS, Inc. had to restructure the system that was currently in place and create an entirely new system that would provide a more intuitive Web page experience for everyone.

The old site utilized one server for development and production of Web materials. The new site uses two servers (Figure 1, see page 13); one for production or storage of finalized materials, and one for development or the creating and updating of materials. This allows the development server to act as a central storage unit inside of the firewall security system, so that files can be updated and replaced quickly and easily before they are sent to the ANSYS Home Page (production server). The addition of a second database brings with it the addition of personnel to update and maintain that database. Where the old site had one person responsible for maintenance, the new site has several.

What's New

Java Applets are the hottest addition to the ANSYS Home Page. This technology creates the interactive experience. Java Applets download to the user's computer, so that activities take place on the user's system in a more effective and timely manner; and not on the server, where it would take more time and could be restricted by the limitations of the server.

The ANSYS WEB is complete with a new look (Figure 2) in line with the company's open focus. The site contains the imagery used with the ANSYS family of products. Upon arriving at the home page, users see different ANSYS graphics appearing on the screen. By choosing one of them, you can find out about the application associated with the image.

In addition to this new look, areas known as zones provide specific levels of access to certain users. One of these zones is focused on customers and contains information concerning the ANSYS family of products, support issues, and various other programs and surveys. Two other zones are geared to the ANSYS sales channel. Access to the different zones is possible through passwords.

Along with the ability to create a virtual page using information stored in a database representing different functional areas of the company, is a system that is browser-friendly. The ANSYS WEB presents information in a familiar manner, which means that the site is translated to look the same no matter what Web browser users run on. This new Web site contains a database full of information, as well as links to other important sites that users should access. In addition, the home page can be read in text-only format for those who have a slow connection to the Internet, or who want the information fast.

As a personal touch, when users come to the ANSYS Home Page they will be given information regarding seminars and training in their specific area, thanks to the system's ability to know where the user is located. And when users register with the home page, they register to receive all information on ANSYS special events, contests, and services.

Future Plans

Future technology for the ANSYS WEB includes Macromedia Shockwave, which enables users to view interactive presentations in real time. It allows the connection between multimedia and the Internet. Active-X technology from Microsoft provides the easiest downloading of Applets to the end-user's system, providing the plug-ins necessary to conduct the activity.

In addition, plans include a system that allows customers to leave analysis problems on the home page for qualified ANSYS, Inc. analysts to solve. As for the future look and feel of the ANSYS WEB, work is already underway on making the Web site more personal. After users enter the site and register, a personalized welcome will appear complete with their name and time of last visit. In the near future, there will also be an interactive and challenging game, making visits to the ANSYS Home Page even more fun and exciting. The game is a collaborative effort by ANSYS, Inc. and Sun Microsystems, and that is all we can say about it right now. Visit the ANSYS WEB to find out more.

ANSYS, Inc. realizes the important role that the Internet will soon play in everyone's lives as well as the future of business, and is working hard to deliver the system of the future today.

by Sergey Sirotinin, Internet Marketing Specialist
ANSYS, Inc.

Daniel Parrish, Marketing Specialist
ANSYS, Inc.

ANSYS "Connects" with ACIS-Based CAD Tools

ANSYS users now have the ability to import .SAT files generated by ACIS-based design tools, with the release of the ANSYS Connection for SAT product offering.

The Power of Choice

The ANSYS, Inc. approach for providing open, flexible solutions is based upon the ability to work with a spectrum of complementary solutions, including computer-aided design (CAD) tools running on a variety of computing platforms. This philosophy supports best-of-breed technologies, encourages integration, and provides flexibility for future advances. ANSYS Inc. customers have the freedom to select the CAD tools and platforms that best suit their specific needs. Each day, ANSYS, Inc. makes new strides in extending this capability.

Continuing Down the Path of Success

ANSYS, Inc. recently released ANSYS Connection for SAT. This new product offering is the latest in a series of "ANSYS Connection" products and services delivered during the past year. These new capabilities are a result of the ANSYS, Inc. open philosophy, and are the fundamental building blocks for CAD integration solutions.

ANSYS Addresses ACIS-Based CAD Tools

ANSYS Connection for SAT is a direct geometry transfer interface that enables access to geometries generated by ACIS-based CAD tools. No IGES is involved in the transfer. Users can seamlessly import files saved in the .SAT format into ANSYS for analysis (Figure 1). In addition, customers can leverage this functionality by working with ANSYS, Inc. Program Customization Services to provide highly tailored solutions based on a unique set of customer requirements. ANSYS Connection for SAT works with any CAD tool that can generate a .SAT file. This includes products such as Hewlett Packard SolidDesigner, Intergraph SolidEdge™, the Autodesk Mechanical Desktop™, and Bentley MicroStation®, to name a few.

Unparalleled Solutions

Employing ANSYS Connection for SAT will allow customers to streamline the design process and focus their energies on performing design analysis—not product interfacing. ANSYS, Inc. is committed to customer success. That means providing high-quality products that are easy to use, technologically advanced, multipurpose, multiplatform, fully compatible, integrated with major CAD systems, and open to customization. ANSYS Connection for SAT is yet another step in that direction.

by Joanne Esposto, Product Marketing
ANSYS, Inc.

Figure 1

Models generated in several different ACIS-based CAD tools. All of these were seamlessly transferred into the ANSYS family of products using ANSYS Connection for SAT.

ANSYS Optimizes Designs:

Building an Underwater Robot Arm

Johnson Engineering Corporation (Webster, TX) used ANSYS technology to create a new training simulator for the Space Shuttle and International Space Station programs. This device, a Weightless Environment Training Facility Remote Manipulator System (WRMS), takes weightless training to new depths; offering enhanced mechanical, structural, and control capabilities.

The automatic design optimization capabilities of ANSYS helped to increase the lifting capacity of this new arm by 400 percent and trim more than 300 pounds from the previous astronaut training device. By combining the design optimization capabilities of the ANSYS simulation program with a proprietary loads analysis routine, Johnson Engineering Corporation created a better robot arm for astronaut training. The new arm can lift 200 pounds,

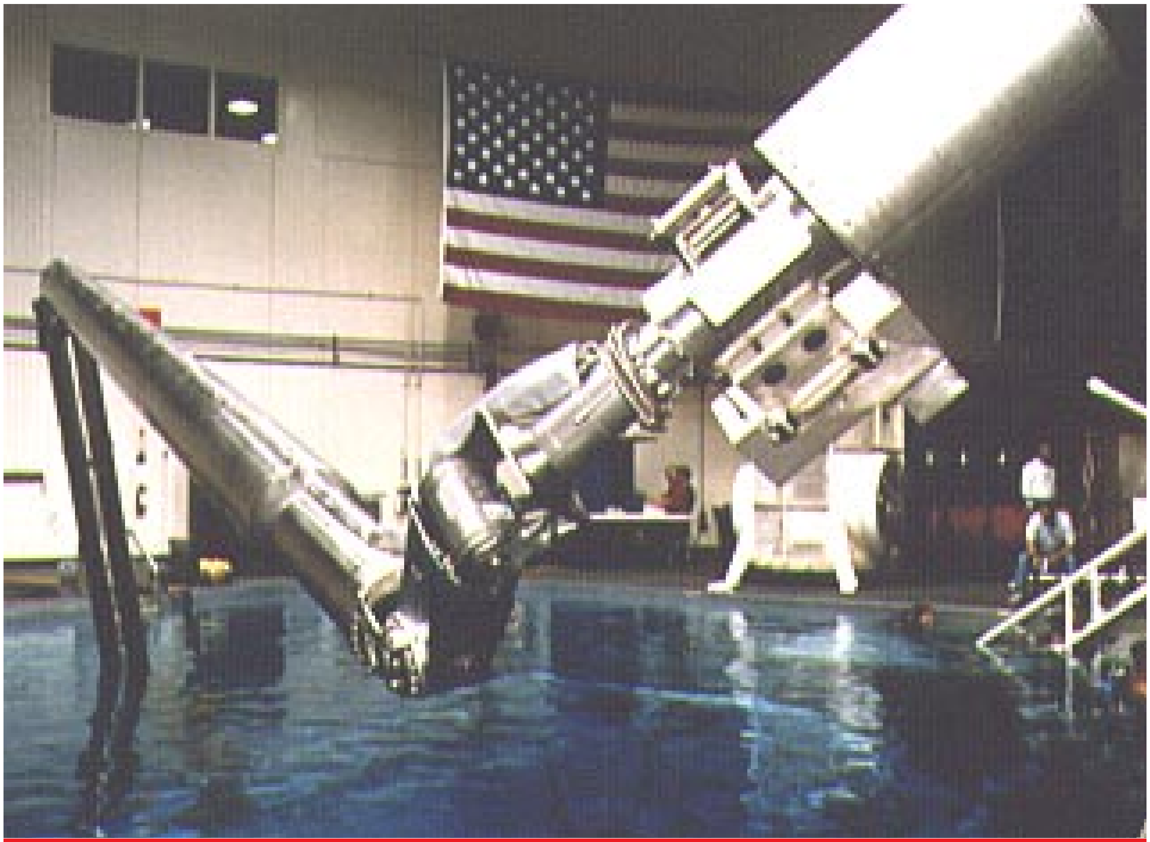


Figure 1

The WRMS demonstrates that it has the ability to operate out of the water, increasing the ease of maintenance. This arm can be serviced in approximately one-fourth the time of the old arm.

compared to the 50-pound limit of the earlier device.

Johnson Engineering, working with engineers at Engineering Cybernetics Inc., the San Antonio, Texas-based ANSYS Support Distributor (ASD), used a variety of simulation methods in the redesign of the robot arm. Some methods were traditional in nature, such as the nonlinear buckling and contact analyses performed with ANSYS design analysis software from ANSYS, Inc. Other methods, which included combining an in-house loads program with the design optimization capability of ANSYS, involved a bit of creative engineering that went a long way toward reducing weight and boosting the lifting capacity of the arm.

Figure 2

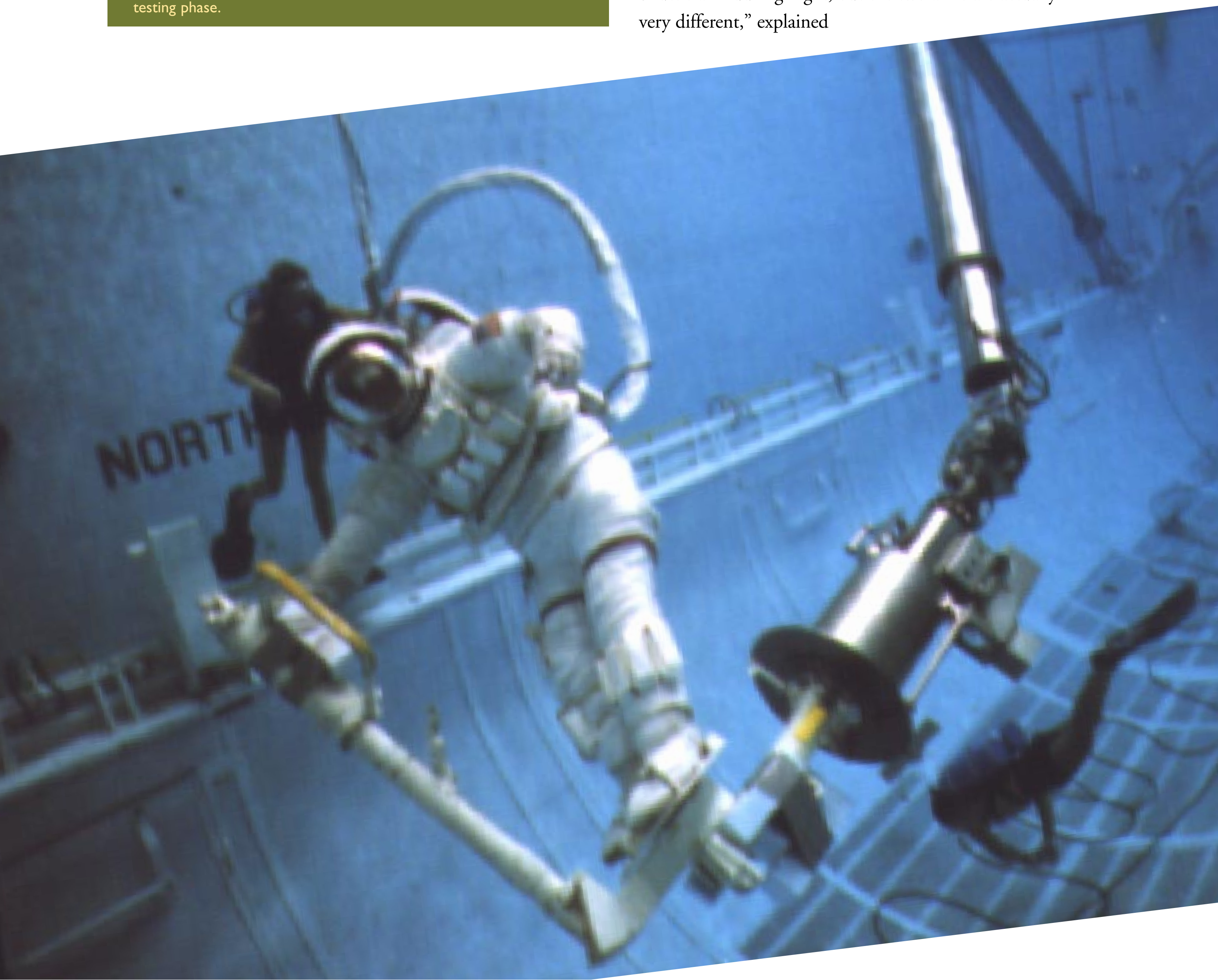
A mock-up of the payload bay of the space shuttle at the WETF. A suited subject is shown on foot restraints during the operational testing phase.

Replica of the Real Thing

The WRMS is a complicated, computer-controlled, seven-jointed robot. It is 50 feet long and built almost entirely of titanium (Figure 1, see page 17). The robot operates underwater in a 25-foot-deep tank at the Weightless Environment Training Facility (WETF) at the Johnson Space Center in Houston, TX. The facility is used to simulate weightlessness for astronaut training (Figure 2).

Although similar in operation to the Space Shuttle arm, the design of the WRMS, in most respects, is completely different. It would be impossible to duplicate the design of the actual Space Shuttle arm on earth, since it was designed to operate in zero gravity. In fact, the Space Shuttle arm cannot even pick itself up in 1G.

"Our challenge was to simulate the operation of the Space Shuttle arm during flight, but the two arms are actually very different," explained



David McMahon, a Johnson Engineering project engineer who works at the Johnson Space Center. McMahon was in charge of the structural design of the arm. “The WRMS has the same volume and controls as the real thing, but it is run with hydraulic power instead of electric power,” he added.

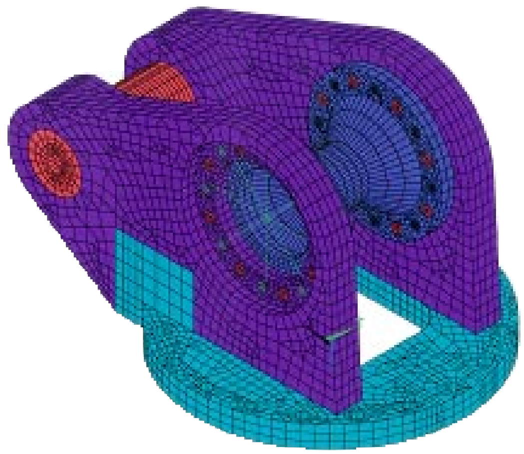
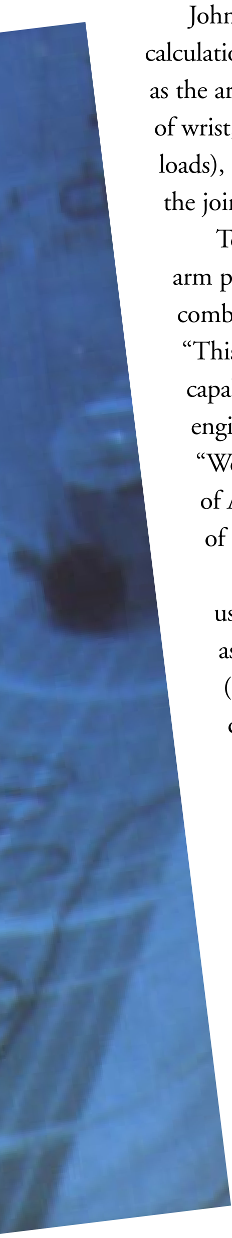


Figure 3

Finite element model of the shoulder pitch yoke including the hydraulic actuator mounting pin and stiffening spool.

Meeting Challenges

One of the complications facing Johnson Engineering in this project was the many different positions the device could assume. To evaluate lifting capacity, it was necessary to calculate torque at thousands of different positions of the wrist, elbow, and shoulder joints. To complicate matters further, in certain positions, portions of the arm rise above the surface of the water.



Johnson Engineering wrote a program to handle these calculations, including the changing buoyancy conditions as the arm position changed. For each input (any position of wrist, elbow, and shoulder joint and one of two different loads), the program computed forces and torques on the joints.

To automate the investigation of the many possible arm positions on lift capacity, engineers on the project combined the in-house loads program with ANSYS. “This was an unusual way to use the optimization capabilities of ANSYS,” admitted Doug Scheiding, engineering manager at Engineering Cybernetics. “We were using the pre- and postprocessing capabilities of ANSYS with the loads program as the solver, instead of running a finite element analysis (FEA) simulation.”

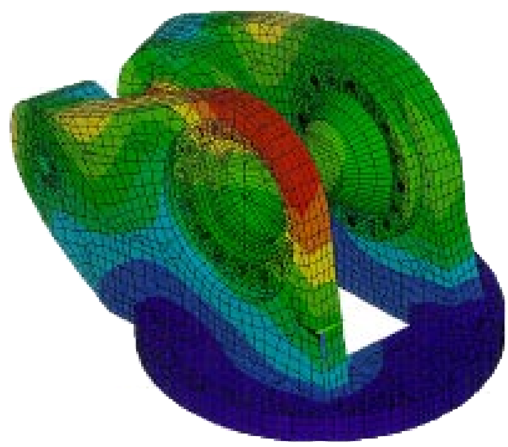


Figure 4

Deflected shape of the shoulder pitch yoke under worst-case loading as determined by the optimization routine.

Typically, with automatic design optimization, a user specifies an engineering goal or objective (such as reduced weight or a target load capacity); variables (thickness of members, material type, geometry); and design constraints such as permissible stress levels. The software runs through many of the possible combinations of variables to find the design that comes closest to the user’s requirements. During the optimization process, the software solves a series of finite element analyses to evaluate the performance of different design possibilities.

The optimization routine determined the maximum torque applied at a given joint (i.e., wrist, elbow, or shoulder). For example, the loads obtained from the worst-case arm configuration for the shoulder joint based on maximum torque were applied to the shoulder model shown in Figure 3. The resulting displaced shape is shown in Figure 4 (see page 19). Figure 5 shows the finite element model of the elbow pitch yoke.

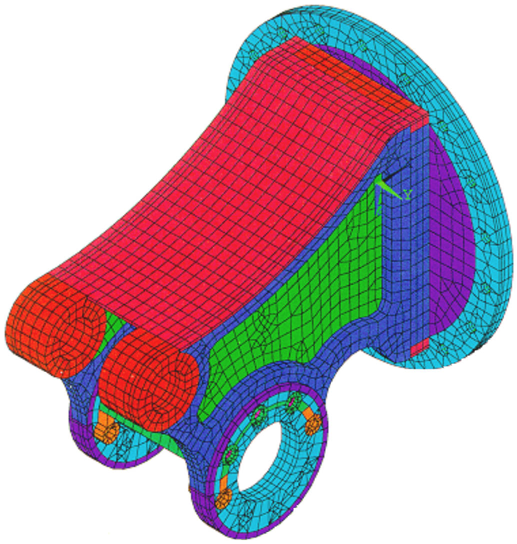


Figure 5

Finite element model of the elbow pitch yoke including the composite bearing.

Creative Engineering with ANSYS

For the robot arm optimization, however, ANSYS was not performing FEA with each design iteration. It was using the results from Johnson Engineering's loads program as though they were analysis results. "Our program gave ANSYS the forces and torques, and it used that information to determine the maximums at each joint," said McMahon. "This really helped us reduce weight because it made it possible to evaluate thousands of positioning possibilities." The high maximum loads obtained from the optimization runs for the main shoulder yoke (Figure 6) resulted in several redesigns of this part to increase the torsional rigidity to meet the stringent tip deflection criteria.

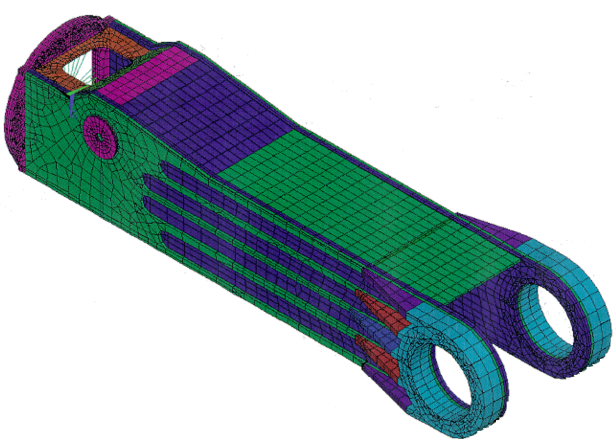
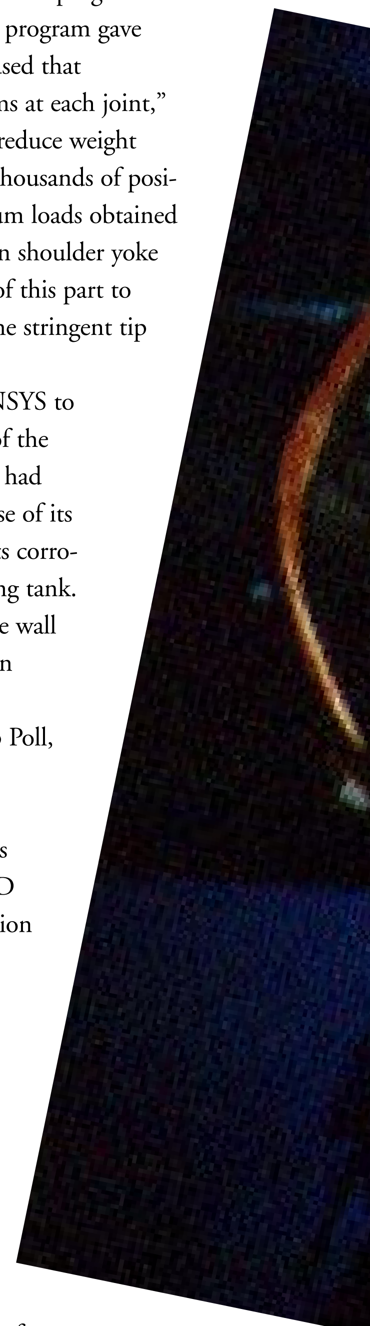


Figure 6

Finite element model of the main shoulder yoke.

In addition, the engineers used ANSYS to perform a nonlinear buckling analysis of the arm's two booms. Johnson Engineering had specified titanium for the booms because of its excellent strength-to-weight ratio and its corrosion resistance in the chlorinated training tank. The analysis was needed to optimize the wall thickness of these members for the given operating loads.

Scheiding and his colleague, Philip Poll, of Engineering Cybernetics, performed these analyses working from Johnson Engineering's Pro/ENGINEER® models of the booms. After converting the CAD data to ANSYS, they ran a large-deflection buckling analysis on the booms using the maximum load of 200 pounds. These nonlinear analyses were run on a four-processor SGI Challenge computer, taking advantage of the parallel processing version of ANSYS, and reducing solution time to less than 30 minutes for each optimization study.

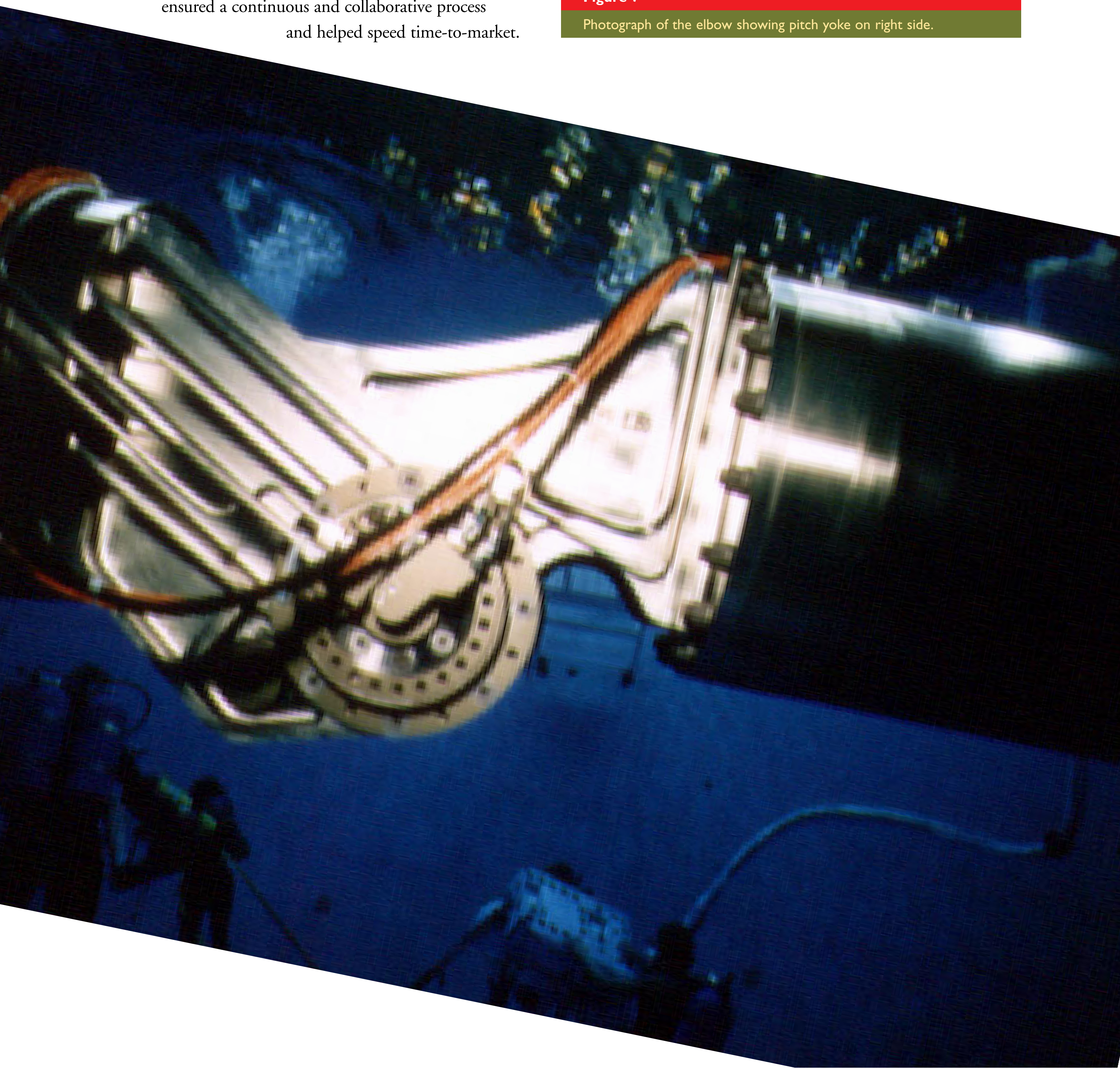


The engineers determined an optimum wall thickness of 0.080-inches from the results. Scheiding used the nonlinear buckling analysis to simulate a large dent in the boom to evaluate the possibility of the arm being damaged during use.

All analyses were done while the robot arm design was in progress. Several parts of the arm, most notably the three pieces comprising the elbow, underwent major changes when analysis showed the original design would not meet stress and deflection requirements (Figure 7). “The tight connection between ANSYS and Pro/ENGINEER, along with the ANSYS Parametric Design Language, made it easy to evaluate additional design options,” said Scheiding. Use of ANSYS throughout the production cycle ensured a continuous and collaborative process and helped speed time-to-market.

by Caren Potter, Freelance Writer
McKinleyville, CA

Figure 7
Photograph of the elbow showing pitch yoke on right side.



Expanding User Benefits through Strategic Partnerships

This new section of *ANSYS News* provides the latest information on a variety of hardware and software partner activities that provide ANSYS customers with a strategic advantage. These partnerships help ANSYS, Inc. deliver the most flexible, enterprise-wide engineering solutions, along with increased business opportunities for ANSYS customers.

ANSYS Announces Intel-Based Certification Program

ANSYS, Inc. has initiated a certification program for Intel-based hardware vendors. The program offers customers greater choice by providing them with a list of certified Intel platforms that have passed ANSYS, Inc. Quality Assurance acceptance tests.

As more engineers migrate to Intel-based PCs to perform design and analysis, quality becomes critical. ANSYS, Inc. already supports Intel-based platforms, but each manufacturer has particular characteristics that make it unique.

The certified Intel platforms list will be published on the ANSYS Home Page, along with benchmark numbers so customers can compare performance on different models. The Intergraph TDZ Series is the first platform to become certified under the program.

If you have any questions about this new program, please contact Elisabeth Mehta, emehta@ansys.com.

ANSYS Teams up with Customers to Create Quality Products

ANSYS, Inc. forms and maintains strong relationships with partners that provide customers with a strategic advantage. ANSYS, Inc. recognizes leading customers, such as Whirlpool Corporation, by featuring them in joint advertisements appearing in engineering trade publications.

The Whirlpool advertisement features how three industry leaders (ANSYS, Inc., Whirlpool Corp., and HP) worked together to create top-notch refrigerators, while reducing prototyping and material costs. Whirlpool Corporation uses ANSYS CAD-integrated software with the ANSYS/Multiphysics program (Figure 1) on HP workstations in an enterprise-wide engineering environment.

The advertisement will debut in October in *Mechanical Engineering.* Plans for the advertisement include a translated version for the Asian press.

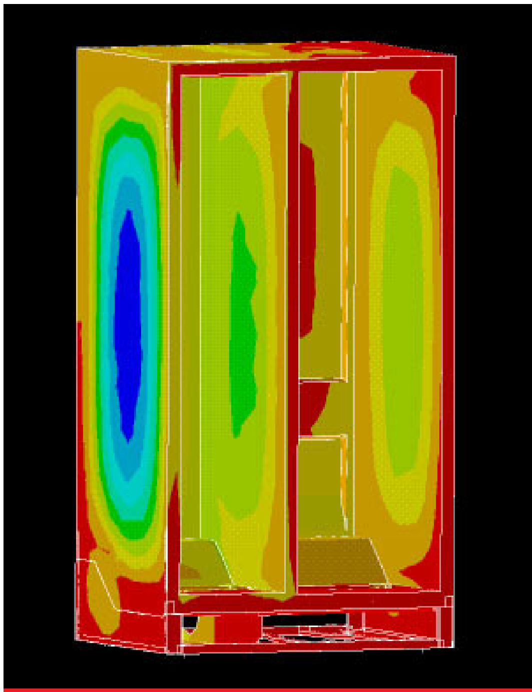


Figure 1

The ANSYS/Multiphysics analysis of a Whirlpool refrigerator.

INTERGRAPH®
SOFTWARE SOLUTIONS

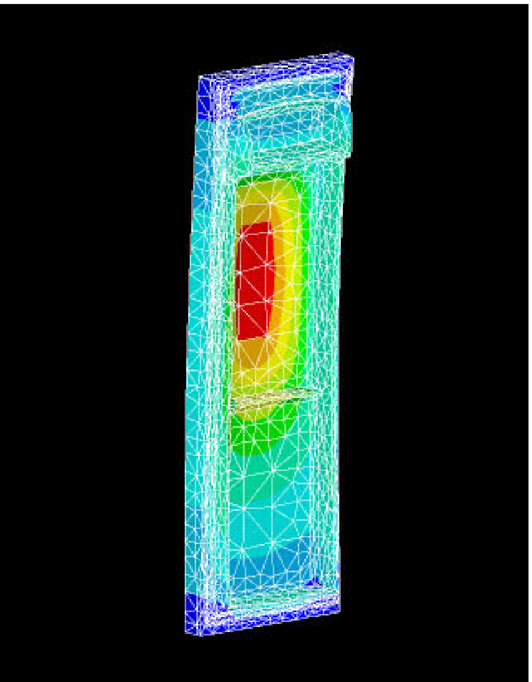


Figure 1A

Analysis of the refrigerator door.

Silicon Graphics, Inc. (SGI) Supports Concurrent Engineering Corp. (CEC) Users' Conference

SGI participated as the exclusive hardware vendor at CEC's ANSYS Users' Conference on July 9, 1996, in Minneapolis, MN. CEC is the ANSYS Support Distributor for Minnesota, northern Wisconsin, North Dakota, South Dakota, and northern Michigan.

The theme of the CEC conference was "Collaborative Engineering." Jeff Schrantz spoke on this topic at the conference, discussing how the trend is for computer-aided design, computer-aided engineering, and computer-aided manufacturing to work concurrently; so companies realize costs savings early in the design phase. John Swanson also spoke at the conference, giving an overview of the enhancements in ANSYS 5.3.

Customers and prospects such as 3M, Honeywell, Andersen Windows, and Dresser-Rand tested ANSYS 5.3 on SGI workstations in a hands-on area and in the SGI booth at the conference. Approximately 55 people attended the conference. CEC holds a users' conference with each new release of ANSYS software.

Two New Companies Join the ANSYS Enhanced Solution Partner (ESP) Program

The ANSYS ESP Program supports high-quality, vertical application developers that utilize ANSYS as a platform to create custom products. This innovative program allows developers the opportunity to transform industry-specific knowledge into usable design software. Members of the ESP Program can create their own customized solutions from within the ANSYS environment with a suite of development tools available in the ANSYS application programmer's interface (API). Two new members to the ESP Program include LMS Numerical Technologies and Materials and Engineering Research Laboratories, Ltd.

LMS Numerical Technologies, (formerly Numerical Integration Technologies), Leuven, Belgium, has been developing an interface from SYSNOISE to ANSYS for several years on an informal basis. SYSNOISE is a leading vibro-acoustic software program that predicts the radiation, diffraction, and transmission of sound waves and the structural vibrations induced by the loading effects of acoustic fluid onto a structure. The program calculates a wide variety of results such as sound pressures, acoustic sensitivities, normal codes, and structural deflections. SYSNOISE is used in the automotive, aerospace, consumer electronics, and underwater industries.

HEWLETT PACKARD

Channel Partner

SiliconGraphics

ISV

AUTHORIZED INDEPENDENT SOFTWARE VENDOR

PREMIER

Materials Engineering Research Laboratory Ltd. (MERL), Hertford, England, has undertaken a joint industry project to create hyperelastic material modeling capabilities, using the ANSYS API as the development environment. The project is concerned with calculating the rates of crack growth in elastomeric components subjected to fatigue loads. Model generation has been facilitated by interfacing a specialized solver, FLEXPAC™, with ANSYS/PrepPost™ via APDL.

by Elisabeth Mehta, Manager
Strategic Relations
ANSYS, Inc.

Top Notch Training for Users

It is imperative for companies to be knowledgeable about leading-edge technology in today's constantly changing global marketplace.

Competitive companies provide training to professionals to ensure maximum productivity and to get the most return on software and hardware investments. ANSYS, Inc. makes it easy to stay up-to-date with the latest technological inventions through a multitude of training courses offered at corporate headquarters, ANSYS Support Distributor (ASD) sites, and universities. Industry experts and active engineering professionals provide hands-on training to users to enable them to solve real-world problems with ANSYS market-leading design analysis software products.

For example, Mallett Technology, ASD for the Mid-Atlantic region, in conjunction with the University of Maryland, has sponsored an educational program for the past nine years that helps engineers of leading manufacturers get current information and training on the latest technology and methodologies. Courses provide practical information that users need to apply current technical skills to their jobs.

Mallett Technology has a training center located just outside Washington, DC that exemplifies the company's commitment to providing customers with quality training services. The training center uses the latest classroom technologies and instructional techniques to teach courses in computer-aided engineering (CAE) software, operating systems, and engineering technology. Dr. Abed Khaskia, Vice President, Mallett Technology, Washington, DC office, says his courses, which he's been teaching for years, draw attendees from manufacturing businesses. His sessions are designed to help users and management understand how technology can help companies reduce time-to-market, improve product quality, and slash production costs. While the courses are technically driven, attendees also want to learn about leading-edge technology and how they can apply it to their job environment, so they can work more efficiently and effectively.

Attendees of previous *Integration of FEA Into Design Practice* class.

Continuing Education

Courses that aid and enhance practicing engineers' skills are crucial for companies to maintain a competitive position in the marketplace. The skills, knowledge, and expertise of an engineering staff are valuable resources for today's companies. ANSYS, Inc. and the ASDs support the value of ongoing training and are committed to offering the engineering community high-quality, easily accessible courses that provide current industry knowledge.

Course Offerings:

The Integration of Finite Element Analysis Into Design Practice
This course explains data transfer from CAD systems, provides guidelines for finite element modeling, and covers advanced topics in adaptive meshing and optimization. Lecturers include Professor Mark Shepherd of Rensselaer Polytechnic Institute, Professor Sunil Saigal of Carnegie Mellon University, and instructors from Mallett Technology and ANSYS, Inc. as well.

Computational Fluid Dynamics Using Finite Element Analysis
This course is particularly valuable to ANSYS/FLOTRAN™ users. It presents basic equations and illustrates how to obtain and interpret fluid response using modern computational fluid dynamics software. Instructors of this course include Professor Steven Miner of the U.S. Naval Academy, Dr. Richard Schwirian of Westinghouse Nuclear Technology, and Mr. James Smotrel of Babcock and Wilcox Nuclear Technologies.

These three-day courses will be held at the University College Conference Center at the University of Maryland, College Park Campus. *The Integration of Finite Element Analysis Into Design Practice* is October 7-9, 1996. *The Computational Fluid Dynamics Using Finite Element Analysis* is scheduled for November 18-20, 1996.

Information regarding the content and arrangements for these courses may be obtained by contacting David Dietrich II at Mallett Technology at 301.725.0060, extension 18, or e-mail to dedii@wdc.mallett.com. Information on local ANSYS training and other seminars can be found on page 28 of this issue of *ANSYS News.*

Olympic Aspirations

Not many people have a chance to compete in one of the greatest of all sporting events, or even to participate in the preliminary trials. The ones who do will remember it for the rest of their lives.

Four years ago, "the ANSYS team," led by Michael Butz, Sales Manager for ASD Mallett Technology in Maryland, began pursuing their dream of becoming the 1996 U.S. Olympic representative for sailing. This team of three traveled all over the country to compete, and were selected as one of the top five boats for the 1996 U.S. Sailing Team.

Their commitment, which required four years of balancing priorities, came to a culmination at the end of April, 1996. The Olympic trials took place on the Wilmington River in Savannah, GA from April 30 to May 10. After several days of fleet racing, the ANSYS team, unfortunately, did not place in the top four; which meant they weren't going to the Olympics. They did, however, have the unforgettable experience of the teamwork, commitment, planning, and dedication it took to get them to the trials which, in itself, is a great accomplishment.

As the photo shows, the ANSYS logo adorns the side of the boat. Michael says, "The logo drew a lot of attention. I was asked countless times, what is ANSYS?" He used this opportunity to talk about ANSYS technology and its wide range of applications. Many of the sailors were especially interested in the computational fluid dynamics (CFD) capabilities available in the ANSYS/Multiphysics and ANSYS/FLOTRAN programs. Kudos to Michael for a most unique method of spreading the word about leading-edge ANSYS technology.

by Jen Valachovic, Marketing Specialist
ANSYS, Inc.

The ANSYS sailing team in action.

ASD Locations

North America

Advanced Engineering Technologies, Inc.
6525 The Corners Parkway, Suite 400
Norcross, GA 30092
Attn: Ted L. Kerr
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: caeama@aol.com
Telephone: (508) 791-2925
Fax: (508) 791-5477

Computer Aided Engineering Associates, Inc.
35-1 Whistler Drive
Freehold, NJ 07728
Attn: Dave Siwiec
Internet: caenj@aol.com
Telephone: (908) 303-9163
Fax: (908) 303-9174

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

CEC Corporation
Riverview Office Tower
8009 34th Avenue South
Suite 1475
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 Testing & Engineering Services, Inc.
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: Doug Scheiding
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 Loyd
Internet: info@ecihou.com
Toll Free: 1-800-707-9798
Telephone: (713) 398-7041
Fax: (713) 398-7119

Engineering Methods, Inc.
4500 Carew Tower
Cincinnati, OH 45202
Attn: William H. Hibbard
Internet: emicincy@aol.com
Telephone: (513) 381-2200
Fax: (513) 381-4118

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: Mike Smith
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: Paul J. Hamilton, Jr.
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 110
Huntington Beach, CA 92648
Attn: Dr. J.W. Jones
Internet: sscca@sprynet.com
Telephone: (714) 964-1552
Fax: (714) 964-1156

SSC, Inc.
1S443 Summit Avenue, Suite 304B
Oakbrook Terrace, IL 60181
Attn: Frank Wassmer
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: Sergio Ricardo Rodrigues
Internet: smisp@embratel.net.br
Telephone: 55-11-820-0388
Fax: 55-11-820-7361

Europe

AT&T ISTEL
2, rue du Docteur Lombard
92441 Issy-les-Moulineaux-Cedex
FRANCE
Attn: Dominique Bonnecase
Internet: 101644.32150@compuserve.com
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
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
Geschäftsstelle Hannover
Schmiedestr. 31
D-31303 Burgdorf
GERMANY
Attn: Clemens Groth
Internet: cgroth@cadfem.de
Telephone: 49-5136-85717
Fax: 49-5136-873179

CAD-FEM GmbH
Geschäftsstelle Stuttgart
Heidenheimer Str. 5
D-71229 Leonberg
GERMANY
Attn: Uli Göhner
Internet: ugoehner@cadfem.de
Telephone: 49-7152-42084
Fax: 49-7152-72984

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

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

CAD-FEM Representation Russia
Office 1703
Schelkovskoe Shosse
107497 Moscow
RUSSIA
Attn: Valery Anpilov
Telephone: 7-95-9132300
Fax: 7-95-9132300

CAD-FEM Support Associate
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

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

Structures and Computers Ltd.
Strucom House
188-196 Canterbury Road
Croydon CR0 3HF
Surrey, ENGLAND
Attn: Stewart Morrison
Internet: smorrison@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: J.Mathijssen@strucom.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: dsweeney@strucom.ie
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@bristol.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: 81-3-5978-5420
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: Shao Wanpeng
Internet: imagbj@public.bta.net.cn
Telephone: 86-1-6513-7766 Ext. 1222
Fax: 86-1-6513-9945

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: Y. K. Kwon
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@compusim.com
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@unisono.ciateq.mx
Telephone: 52-415-20547
Fax: 52-415-24279

Republic of South Africa

Wolhuter & Van Wyk
Consulting Engineering, CC.
c/o Potchefstroom University for CHE
Dept. of Mechanical Engineering
Post Point 5743
Potchefstroom 2520
REP. OF SOUTH AFRICA
Attn: Arno Wolhuter
Internet: mgiaw@puknet.puk.ac.za
Telephone: 27-148-299-1324
Fax: 27-148-299-1320

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	Dynamics	Structural Nonlinearities	Heat Transfer	Design Optimization	Solid Modeling	Introduction to FLOTRAN®	Electomagnetics	ANSYS/ProFEA®	Special Topics (See footnotes)
Sept. 29-Oct. 5	ANS ECI MTI						ATT			ANS[22] CAD[5]
Oct. 6-12	DRD EMI MCR SSC STR	DRD	DEF	SSC STR		ECI EMI	JAR	ANS	CEC	ATT[8] MTI[13]
Oct. 13-19			SSC STI	ANS AET			ECI			CAD[3] CAEA[14] CAEA[15] MTI[2] HGE[29] INAS[26] STR[24] ING[32]
Oct. 20-26	ATT DEF SSC	ANS CADIT	EMI	SSC			STR	CEC STR	DRD	ANS[2] CAD[10] CAEA[7] HGE[29] DEF[22] DRD[25] EMI[2] ING[4] JAR[4]
Oct. 27-Nov. 2		SSC	SSC							CAD[2] CAD[14] ING[28]
Nov. 3-9	CADIT CEC ECI EMI MTI STR	DEF	STR	DEF JAR		EMI	ANS			STR[6] ANS[33]
Nov. 10-16	ANS AET CAEA JAR ING	CAEA	ECI HGE			STI ING	DRD			AET[22]
Nov. 17-23	ATT DEF DRD JLR STR		ANS	EMI OCAE STR	EMI		ECI		DRD	ANS[2] CAD[16] DEF[22] ING[4] STR[24] INAS[27] MTI[9] ECI[33]
Nov. 24-30			ING STR	STR	ANS					ATT[10] INAS[20] ING[2]
Dec. 1-7	CADIT STR SSC ECI			STR						ATT[11] CAEA[2]
Dec. 8-14	ANS DRD EMI SSC STI		JAR	CADIT ECI					EMI MTI	ANS[22] CAD[23] ING[30] STR[24] MTI[31]
Dec. 15-21	ATT MTI		EMI					ANS	DRD	DRD[25]

Company Key

- ANS — ANSYS, Inc.
- AET — Advanced Engineering Technologies, Inc.
- AZE — Anker-Zemer Engineering
- ATT — AT&T ISTEL
- BER — Bercom Software Development and Consultancy, Ltd.
- CAD — CAD-FEM GmbH
- CADIT — CAD-IT Consultants Pte. Ltd.
- CAEA — Computer Aided Engineering Associates, Inc.
- CEC — Concurrent Engineering Corp.
- CEMA — CEMA International Corp.
- CSC — Cybernet Systems Co., Ltd.
- DEF — Defiance-STS/SMC
- DRD — DRD Corporation
- ECI — Engineering Cybernetics, Inc.
- EMI — Engineering Methods, Inc.
- FIG — Figes, Ltd.
- HGE — H.G.E., Inc.
- IMAG — IMAG Industries, Inc.
- INAS — Institutal Pentru Analiza Sistemelor
- ING — Ingeciber S.A.
- ITAL — ITALCAE srl
- JAR — Jordan, Apostal, Ritter, Associates, Inc.
- JLR — JLR Computer Analysis, Inc.
- JIN — Jin Young Technology
- MCR — MCR Associates, Inc.
- MTI — Mallett Technology, Inc.
- NAT — Neil Automation Technology Ltd.
- NIIT — NIIT Engineering and Geomatics Solutions Group
- OCAE — Ohio Computer Aided Engineering, Inc.
- SMI — SMI-Software Marketing International LTDA
- SSC — SSC, Inc.
- SSCM — SSC de Mexico S.A.
- STI — Stress Technology, Inc.
- STR — Structures and Computers Ltd.
- SVS — SVS FEM s.r.o.
- TADC — Taiwan Auto-Design Co.
- WORL — Worley Limited
- WVWC — Wolhuter & Van Wyk Consulting Engineering, CC.

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
8. Thermal Analysis
9. CFD Using FEA
10. Composites
11. Dynamic Analysis
12. Error Estimation
13. Integration of FEA Into Design Practice
14. Fracture Mechanics
15. Finite Element Modeling
16. Visoelasticity and Viscoplasticity
17. IGES Transfer
18. Fluid Mechanics
19. Mechanical Vibration Using ANSYS
20. Introduction to ANSYS - Revision 5.2
21. Updating FEM
22. Introduction of ANSYS Part II
23. Substructuring, Submodeling, Harmonic Elements
24. ANSYS/ED
25. Advanced ANSYS/ProFEA
26. Fatigue
27. Introduction to FEM
28. Mechanic Systems Simulation
29. Modeling and Meshing
30. Analysis of Aleatory Vibrations with ANSYS 5.3
31. Advanced ANSYS/ProEngineer Interface
32. Tutorials ANSYS 5.3
33. Advanced FLOTRAN

ANSYS

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-AN3-9/96

Printed in U.S.A.

ANSYS, INC.
UL REGISTERED FIRM
ISO 9001 A3725

ISO 9001 CERTIFICATION INCLUDES
ALL COMMERCIAL PRODUCTS