ANSYS

SOLUTIONS

THE PREMIER E-ZINE FOR DESIGN INNOVATION

Summer 2003

Click here to take our

User Survey

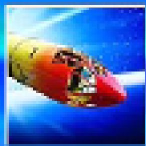
invent

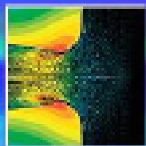
FEATURES:

- FEA Gets Da Vinci Space Project Off the Ground
- Highlights of ANSYS 7.1
- Coupled-Field Solutions Create Competitive Advantages
- Industry Spotlight: Government and Defense

DEPARTMENTS:

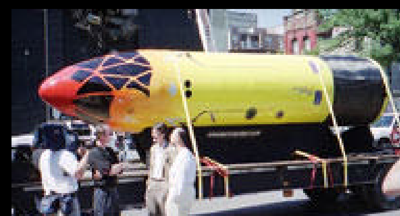
- Editorial
- Industry News
- CAE Community
- Simulation@Work
- Product Development Strategies
- Tech File
- Tips & Techniques
- Managing the Enterprise
- Software Profile





Download pdf | ANSYS Homepage | Contact Us | Archived Issues

This article is based on the work of the following members of the da Vinci Project Team:



- Vladimir Kudriavtsev, Team Leader, Engineering and R&D
- Brian Feeney, Da Vinci Project Team Leader
- Max Buneta, System Analysis and Design
- James Porcher, Ground Operations
- Asier Ania, Thermal Analyst
- Michael Trauttmansdorff, System Design and CAD
- Ta-Liang Hsu, Stress Analyst
- Marek Krzeminski, Dynamics and Control
- Kalman Rooz, Senior Consultant

The da Vinci Project is the first Canadian entry in the international X-Prize Competition to promote the development and flight of spacecraft able to provide low-cost commercial transport of humans into space The X-prize Foundation is providing a purse of US$10 million to the first competitor who can safely launch and land a manned spacecraft to an altitude of 100 kilometers, twice in a two-week period. The competition parallels the Orteig Prize won by Charles Lindbergh in 1927. The day before Charles Lindbergh flew, a 40-passenger clipper class flying boat was inconceivable. Where's the technology? You'll never be able to finance it! Less than 7 years later the flying boats were gracing the sky's of the larger Pacific let alone the Atlantic. We face a similar challenge of the mind today.

The X-Prize competition is opening up space for the average person. Not just literally providing them with a more cost effective way of getting to space, but breaking down the psychological barriers that have built up through the past decades. The crux of this goal is to inspire a new generation of forward-thinking youth to show them just as Lindbergh did in 1927 that the seemingly impossible can be done, not only to space but also to the broader challenges and opportunities we face in life.

Development Efforts

The da Vinci Project began its vehicle development shortly after the X Prize was announced in 1996, with official entry of the team into the competition in 2000. The da Vinci Project is wholly owned by ORVA Space Corp., a Canadian company. Years of engineering research, design and developmental testing has gone into the vehicle design, propulsion and flight guidance system. The da Vinci Project has teamed up with one of North America’s leading Aerospace Rocket Propulsion companies to build the final flight configuration engines.

A core of over 200 professionals volunteer their contributions and more than 35 corporate technical sponsors back the project. Aerospace engineers, experts in project management, finance, media, public relations and graphic design volunteer their time and expertise towards the realization of the next step in Human discovery.

The da Vinci Project employs an aircraft “FAR” engineering approach to design, manufacturing and system redundancy – safety. This approach basically says that the primary objective of any flight is the safety of the flight crew and of civilian populations. The approach therefore is to always be in a position to lose and or abort a mission without loss of the vehicle or it’s crew. Propulsion systems design and performance criteria as well as overall systems architecture were selected to deliberately favor this criteria. Loss of engine power can take place at

any point in the launch sequence followed by abort to re-entry, recovery mode and full recovery of the capsule and propulsion sections.
A full-scale flight-engineering prototype of the manned rocket has been constructed. Detailed engineering and fabrication of the full-scaled manned rocket named Wild Fire Mk Vl is currently underway. Flight-testing of the manned rocket and X Prize Competition flights are targeted to start in late 2003 and continue throughout 2004.

For R&D efforts on the project, we utilized a wide range of engineering software for CAD, basic engineering calculations, trajectory analysis, dynamics and mission control, supersonic external aerodynamics, and internal heat flow. ANSYS simulation software was used for structural stress analysis and thermal analysis of the thermal protection system. Our main engineering approach was to identify potential problem areas, study them in detail using available computational tools and to develop corresponding design solutions.

Vehicle Configuration

The da Vinci Project’s novel rocket design will be launched from the world’s largest reusable helium balloon at an altitude of 80,000 feet (24,400 meters). Reentry will be acomlished with a parachute and airbag configuration. Return to the earth will be accomplished by the spacecraft separating the manned capsule from the primary propulsion section, and having both sections return to the ground independently of each other. The separability of the propulsion system and the manned capsule adds to the crew safety by providing an escape option at all stages of the mission.

The spacecraft consists of two primary subsections: a capsule housing the crew, and a rocket block containing the propulsion system. (See Figure 1). The engine block will be fully recovered alongside the manned capsule, rather than being allowed to burn up on re-entry after its purpose has been served.

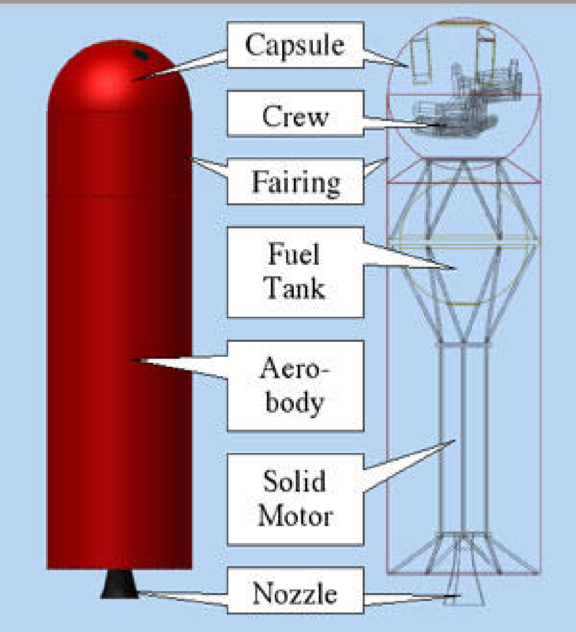


Figure 1. Assembled Wild Fire Mk Vl spacecraft

The capsule is a spherical body with an outer diameter of approximately 2 meters. It is being designed as a fully composite structure, made up primarily of Kevlar and fiberglass. The outer skin will be a semi-structural element, with the addition of ribs in the vertical direction for reinforcement. Structural components such as hatch opening ring and parachute mounts will be filament-wound onto the shell, creating an integrated structure. Ribs in the capsule structure will serve several purposes. They provide stiffness for the sphere, to prevent damage resulting from large deformation. The ribs will also be the primary mounting points for subsystems within the capsule, including air tanks and seats. Lastly, they will serve as load transferring members, to distribute the loads from parachute cables to a larger portion of the sphere.

The base of the capsule will feature a reinforced mounting ring which will be paired to a ring on the top of the rocket block. This will serve to mate the two structures during balloon ascent and the launch phase of the flight. Explosive bolts will trigger after the engine has finished firing, and the two modules will be caused to separate by the pressurized airbag housed between them.

The capsule will have a single hatch located at its top at the centerline. It will serve as the means of entry and egress, both during normal operations and emergency escape. The interior of the capsule will feature clearance to permit the rapid escape of the pilot and crew in the event that emergency egress is warranted. The capsule is required to have a capacity of three people six feet in height.

In light of our aim to minimize the replacement and repair of components between flights we plan to implement a system of cushioning airbags upon landing, to prevent such damage. A parachute will be used to slow the capsule down from terminal velocity to a safe landing speed of 6m/s. The parachute will be deployed at the maximum possible altitude, to provide the most descent time for the deployment of a backup if the primary parachute fails.

The rocket block is a cylindrical body approximately 5 meters in length and 2 meters in diameter. Its structure will be an aluminum truss skeleton, designed to transfer axial loads from the engine through to the capsule mount ring. Additional structural rings will be located at the circumference of the fuel tank, at the top and bottom of the solid motor housing, and at the base of the exterior body of the rocket block; the aero-shell. Torsional stiffness will be created by the semi-structural aero-shell and by the angled truss sections. The rings at each of the junctions of the truss form mounting points for the primary propulsion system, and ensure that the entire spacecraft is self supporting. The primary engine which will propel the spacecraft to its apogee of over 100 km will be a hybrid, using N20 as the oxidizer, and a fuel grain of Hydroxyl Terminated Poly-Butadiene (HTPB).

The rocket block will carry its own guidance control and telemetry system, which will be under the pilot's control via a link to the capsule during ascent. Once separation occurs, the rocket block's re-entry systems will be operated in a fully automatic mode, under control of the guidance computer. Steering during the main engine burn will be carried out by the rocket block's RCS a thruster acting in concert with the capsule's set. During ascent to apogee the rocket block will be stabilized and oriented for an ideal descent by its onboard thrusters.

Upon firing of the explosive bolts, and capsule separation, the faring between the two bodies will be cast aside and allowed to burn up on re-entry. The rocket block features a blunt conical nose which has high drag characteristics and will serve to slow the body while it is descending through the atmosphere. During descent, the rocket block will be slowed by a parachute deployed from the base of the rocket, and will be suspended for a nose-down landing.

Thermal Protection System Evaluation Analysis

One of the critical safety systems onboard the da Vinci spacecraft is the Thermal Protection System (TPS) for both the capsule and propulsion sections. A preliminary analysis of the TPS helped us determine the appropriate materials and configuration for the TPS. With a sub-orbital target altitude of 100 km, the spacecraft will re-enter the earth's atmosphere under much lower thermal and pressure loads than a full re-entry from orbit (such as what the space shuttle experiences). Using CFD analysis, we predict a maximum thermal load, at the stagnation point on the re-entering capsule's surface, of 921.09 K [647.94 °C]. The general thermal load experienced by the capsule (as well as the propulsion section) goes through a cold-to-hot-to-cold cycle.

We generated a list of the materials that matched our load characteristics (a maximum temperature of 921.09 °K [647.94 °C] and an aerodynamic flux of 75,000 MW/m2). Materials included Acusil II, Norcoat 4011, Norcoat Liege, Aleastrasil, Prosial, Nextel 312, Nextel 440, and LTC-HSA. We then compared the thermal diffusivity of the various materials to get an initial indication of how well each material would perform in the subsequent FEA modeling.

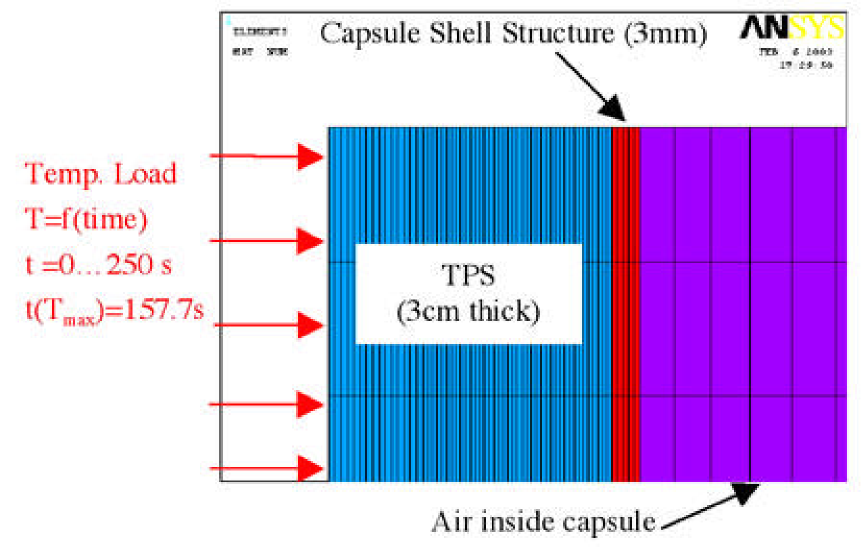


Figure 2. ANSYS finite element model in for thermal heat wave propagation analysis

Using ANSYS Multiphysics FEA software, 2D finite element block models (shown in Figure 2) were constructed as the basis of a comparative thermal heat wave propagation analysis. Several models were also constructed to facilitate a layered TPS structure, enabling us to determine the efficacy of combining different materials together in various configurations. The initial temperature of the blocks was set to 300 °K [27°C] and the model was analysed over a total trajectory time of 250 seconds, starting at apogee. This is a sufficient time span to fully cover the re-entry phase of the flight. We then compared the heat wave at various times in the capsule trajectory (50s, 100s, 150s, 200s, and 250s). Selected reference points, at various depths within the TPS were also compared.

A 2-D half circle finite element model was created to represent a cross-section of the spherical capsule. This was used to evaluate variation in the heat wave propagation along the circumference of the capsule. As the 2-D block comparison used the maximum transient thermal load from the stagnation point, the actual load along the circumference of the capsule is varied; up to 17 % lower in certain sections (largest temperature difference being 150 degrees). The skin temperature distribution was used to approximate a temperature profile over the perimeter of the full capsule cross-section shown in Figure 3.

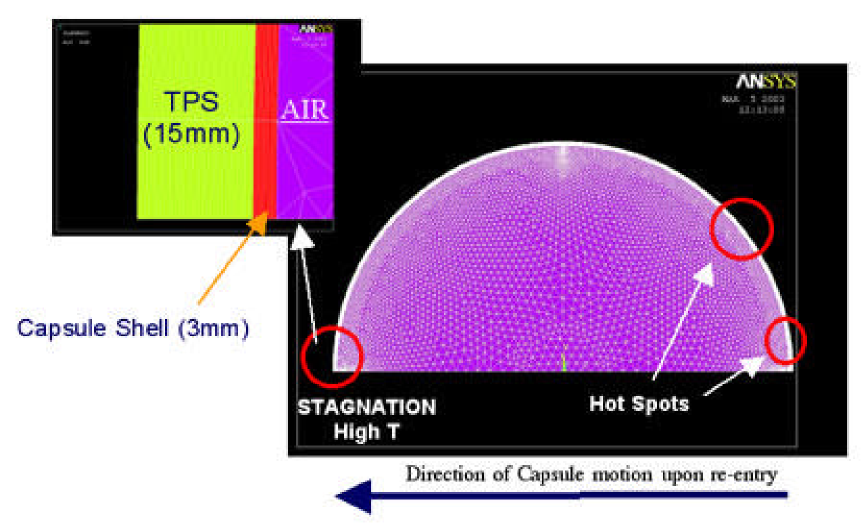


Figure 3. Capsule cross-section finite element model

The maximum temperature in the thermal load is seen at 157.7 seconds into the trajectory. The plot at 200 seconds demonstrates the heat wave that resulted from the high temperature portion of the thermal load. Both the Nextel and Norcoat Liege materials had no temperature change past a depth of 1.0 cm. All the other materials had significant rises in temperature past the 1.0 cm point. The heat wave in Aleastrasil propagated to a depth of 2.5cm.

All of the reference point plots showed that there was the least temperature change with Nextel 440, Nextel 312, and Norcoat Liege (in order of increasing temperature change). The best results were achieved with those materials that had the lowest thermal diffusivity; as to be expected. Comparing materials by thermal conductivity alone is not sufficient. The LTC-HSA (Low Thermal Conductivity – High Surface Area) material had the second lowest thermal conductivity but did not provide an adequate thermal barrier relative to the other materials. This was due to its low density which contributed to a higher thermal diffusivity.

The 2-D block results obtained enabled us to gauge how much (thickness of TPS) of a certain material would be necessary in order to limit significant thermal propagation from our predicted loads. Nextel (312 & 440) and Norcoat Liege were the least sensitive to their thermal environment and provided the best thermal protection with the least thickness. At a depth of 1.0 cm all three materials would sufficiently provide adequate thermal protection. However, there are other equally important factors that must be considered such as the total mass of the system.

The selection of a final structural capsule shell material will have an impact on the TPS design by altering the target maximum temperature for the outside of the shell. A material like Aluminum will be much more sensitive to the thermal environment than Kevlar and would thus lower the target maximum temperature that must be designed for; this could in turn increase the TPS thickness. The best materials when factoring in strict mass constraints are Norcoat Liege, Acusil II, and LTC-HSA.

Even though LTC-HSA had the lowest mass of only 35.3 kg, its ablative properties would not allow it be exposed to the outer environment upon re-entry and therefore any TPS with LTC-HSA would need to have an outer layer. One such configuration could be with Nextel 440. The preliminary results show that there is a mass savings when combining these materials. The single 5.33 mm layer Nextel 440 TPS had a mass of 58.9 kg with an outer shell maximum temperature of 325 °K [51.85°]. Where as the 1N-13L configuration had an outer shell maximum temperature of only 304.3 °K [31.15°C] and only weigthed 41.6 kg. A mass savings of 17.3 kg, which is fairly substantial when we take into account the capsule's strict mass constraints. The outer shell maximum temperature was also lower by 20.7 degrees. Decreasing the LTC-HSA layer thickness to allow for a greater outer shell maximum temperature would decrease the mass even further. The rest of the layered models also showed similar reductions in overall mass. The Nextel-LTC-HSA layered approach yields better results over single layer models with the same materials. Further layered trials will be conducted to evaluate this approach.

To confirm the accuracy of our ANSYS results, two comparisons were performed using the explicit finite-difference method. For the first comparison, an example was used from a heat transfer text of a copper block with a constant heat flux load applied to one side. The text provided various solutions including an exact solution. An ANSYS result of the same problem was compared to the finite difference method programmed with Maple software, and the exact solution. It was found to have very good agreement between the methods; with the largest variation being 0.43%. The second comparison was of 1-D heat conduction through a 3cm block of Acusil II with a constant temperature load applied to one side. The variation between the ANSYS result and the explicit finite-difference model was 0.16%. These validation results raise our confidence level in the results that we are getting from ANSYS. Future validation work will also be performed through experimental results.

Data Connectivity

The da Vinci project is working as a small integrated team and is leveraging software and computer technology to maximize the effectiveness of its members. By producing an integrated work system, we built up a rapid design analysis capability.

Data is passed between the three software packages (Matlab 6.5, AutoDesk Inventor 7 and Ansys 7) through the use of Microsoft Excel spreadsheets as illustrated in the figures below.

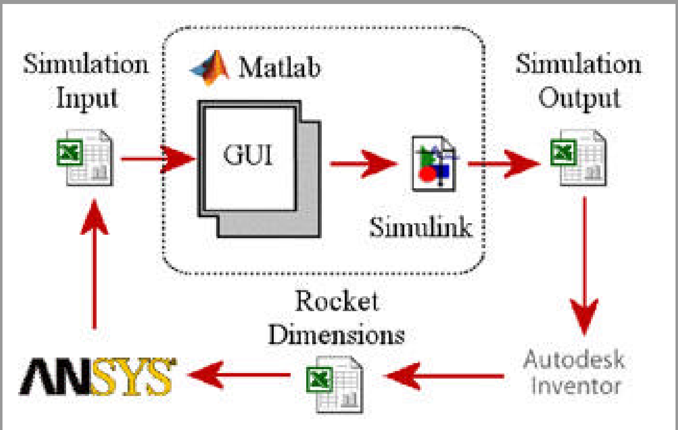


Figure 4. Data Communication path between the three software packages used in the design process.

Figure 5. Vehicle Configuration

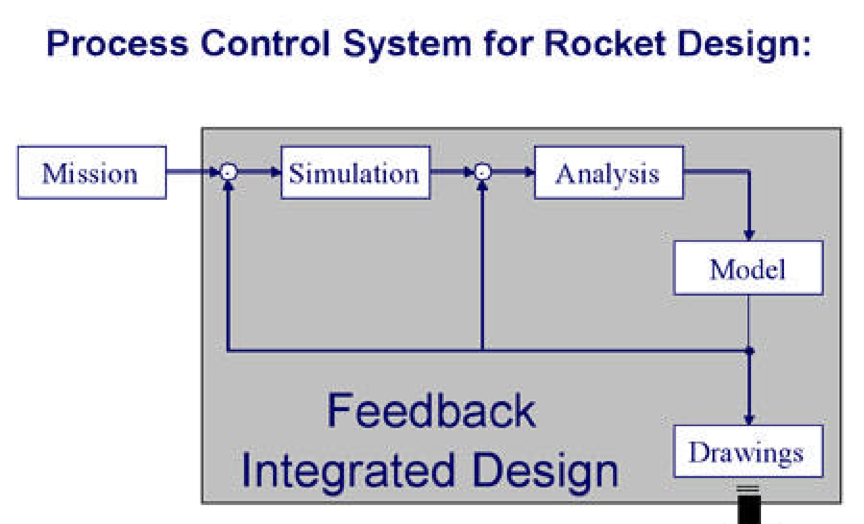
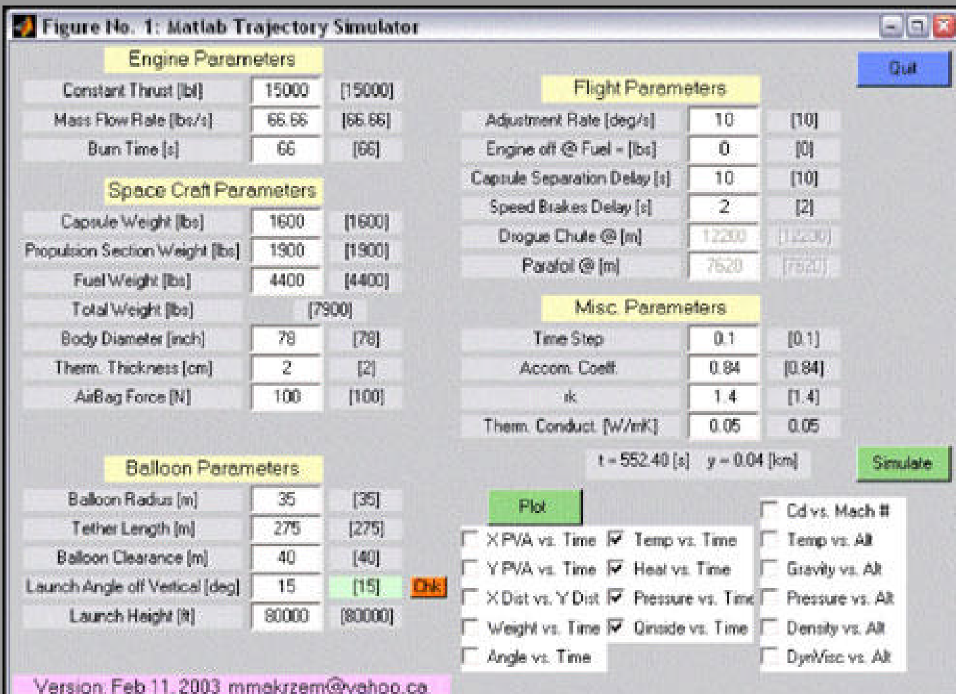
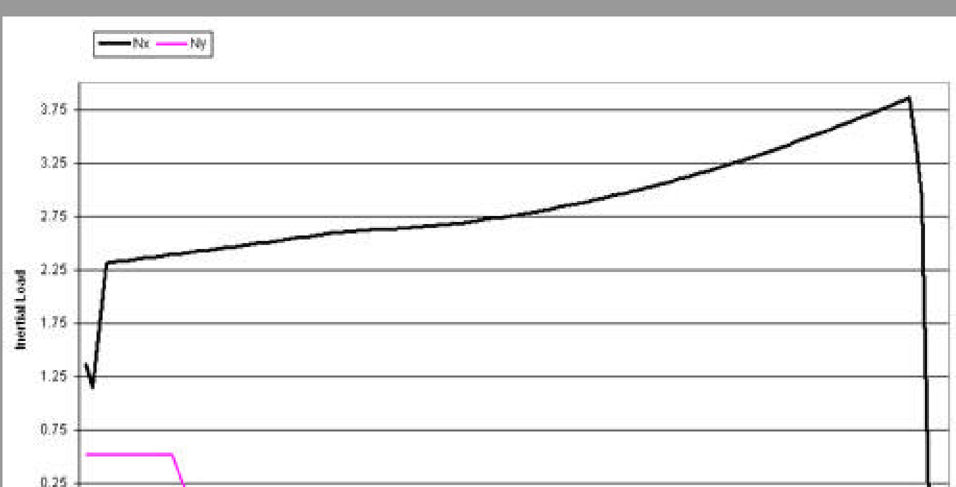


Figure 6. Data path for connected simulation, analysis, and modeling software. Arrows within the gray box represent fully automated information exchange.

The design loop begins with an Excel spreadsheet that contains all the parameters required to simulate the rocket flight. These parameters include rocket dimensions as well as mission specifications such as launch height, direction and wind speed to name a few. All these parameters are loaded into the Matlab flight simulation environment and displayed on a Graphical User Interface (GUI) before each simulation.





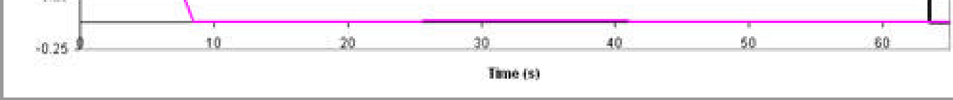


Figure 7. Matlab Graphical User Interface and Axial (Nx) And lateral (Ny) time dependency

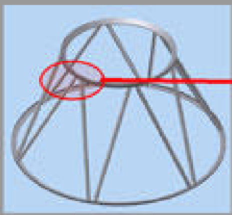
After the simulation has finished running, its output is saved in another Excel spreadsheet. Load data produced in the simulator is passed on to our CAD model (in AutoDesk Inventor) via the spreadsheet. The spacecraft's maximum acceleration in the axial direction is used by the spreadsheet to calculate appropriate member thickness.

Maximum Nx	=	3.8	G	Parameters to Simulator Interface.xls
Inertial Overload	=	4.8	G	io=a/g+a
Recommended Wall Thickness	=	1.141	mm	
	=	1.0000	1/16 in	number of sixteenths of inches, Rounded Up
	=	**1/16**	**in**	Based on Euler Load

Maximum Nx	=	5.8	G	Parameters to Simulator Interface.xls
Inertial Overload	=	6.8	G	io=a/g+a
Recommended Wall Thickness	=	1.688	mm	
	=	2.0000	1/16 in	number of sixteenths of inches, Rounded Up
	=	1/8	in	Based on Euler Load

Figure 8. Updating the axial load (Nx) from the simulation causes the truss member wall thickness to be re-calculated and passed to the CAD Model

The thickness is passed to AutoDesk Inventor as a parameter, and the model is updated as soon as the simulator updates the predicted loads. The updated structure can then be validated by using Ansys Workbench to perform FEA on the structure.



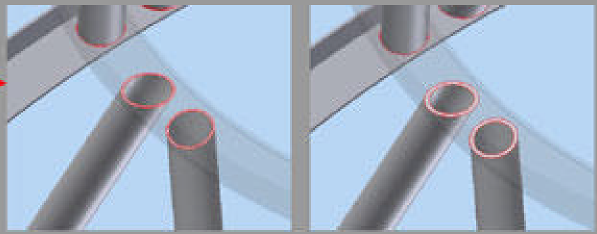


Figure 9. Propulsion Section structural truss (Built in Autodesk Inventor)

Figure 10. Member thickness in the Inventor Model is increased to withstand change in load predicted by simulation.

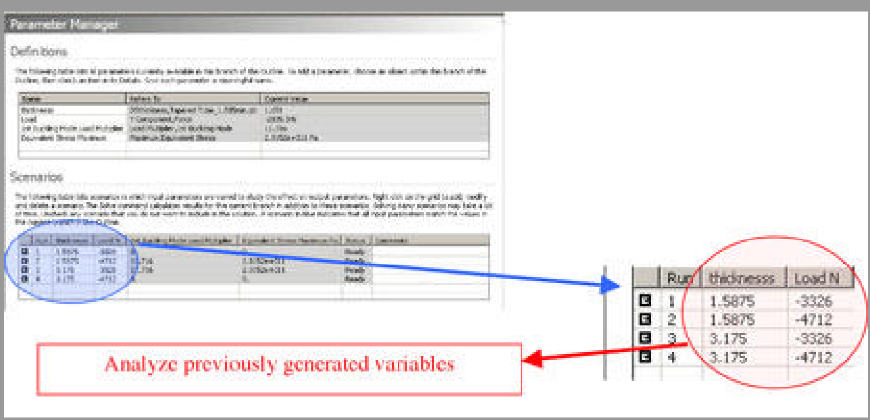


Figure 11. Parametric Analysis using Ansys Workbench v. 7.0

The final step consists of the FEA solution using Ansys Workbench. The loads from the Simulator and the varied thickness from Inventor are read into Workbench. Inside Workbench the “Parameter Manager” enables us to parametrically analyze different combinations of variables that were generated from the Matlab Flight Simulator and from Autodesk Inventor CAD package. The results are checked and design iterations are made.

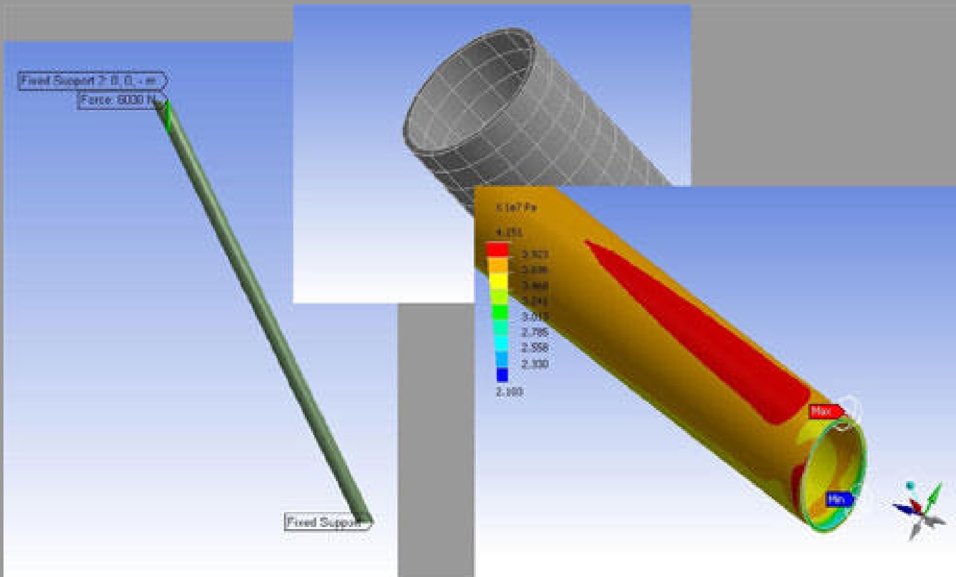


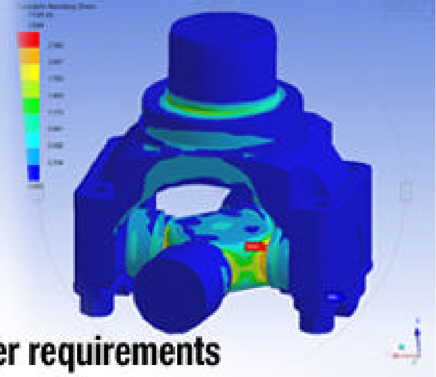
Figure 12. Environment, Mesh and Load Results examples from Workbench.

ANSYS Classic and ANSYS Workbench/Design Space environments were utilized to perform stress analysis of the rocket block, space capsule and various critical components. This will be the subject of the Part 2 of this article. For more information on the da Vinci Project, visit http://www.davinciproject.com

Print this article

Email this article to a friend

New Features in Workbench Environment



Latest enhancements meet expanding range of customer requirements

The ANSYS Workbench Environment in the 7.1 release provides unique automation, performance, and knowledge capture technology that integrates simulation activities with parametric CAD systems; all built upon the world-class ANSYS solver foundation.

Some of the many highlights of the release are summarized below. Click here to see all of the new features (including those in the Classic Environment).

General Workbench

ANSYS Workbench Start/Project Page, Project Tabs for managing projects including defining preferences for CAD importing, and selecting the main project task module (DesignModeler, Design Simulation, or DesignXplorer).

DesignModeler

- Open profiles (i.e., chains of sketch edges that are not closed) are now supported for all of the basic 3D features: Extrude, Revolve, Skin/Loft, and Sweep.
- In DesignModeler 7.1, you can now insert features into the middle of the feature tree.
- The Edit Selections capability has been expanded to include most features.
- The Thin/Surface Feature supports Thickness > 0, even if the selected faces are part of surface bodies to allow for the "thickening" of imported surfaces.
- The Thin/Surface has now a third "type" for the Direction property, namely: Mid-plane, allowing to apply half of the given thickness to both sides.
- DesignModeler 7.1 has a new Slice feature that improves the usability of the DesignModeler as a tool to produce sweep-able bodies for hex meshing.
- The Concept Modeling Lines From Points feature allows the creation of Line Bodies in DesignModeler that are based on existing points.
- The Concept Modeling Lines From Sketches feature allows the creation of Line Bodies in DesignModeler that are based on base objects, such as sketches and planes.

- The Concept Modeling Lines From Edges feature allows the creation of Line Bodies in DesignModeler that are based on existing model edges.

- The Concept Modeling Split Line Body feature allows the division of a line body edge into two sections when not used in conjunction with the Surfaces From Lines feature.

- The Concept Modeling Surfaces From Lines feature allows the creation of Surface Bodies in DesignModeler that use Line Body edges as the boundary.

- With Concept Modeling you can apply 11 different cross sectional attributes to line bodies, including: Rectangular, Circular, Circular Tube, Channel Section, I Section, Z Section, L Section, T Section, Hat Section, Rectangular Tube, and User Integrated.

- DesignModeler supports CATIA V5 releases 2 through 10 and CATIA V5 assemblies (* .CATProduct).

- DesignModeler now supports IGES versions 4.0, 5.2, and 5.2.

- New Body Operations for Mirror, Move, Copy, Delete, Scale, Cut Material, Slice Material, Imprint Faces.

- Target Bodies allows you to specify which bodies are to be operated on during the Cut, Imprint, Slice Material, and Slice by Plane operations.

- Merge Topology feature will "optimize" the topology of feature bodies, by removing so-called "redundant" topology.

- DesignModeler 7.1 will automatically save backup files (so-called “auto-save” files) of the model, after a specified number of Generate clicks.

- DesignModeler 7.1 can group bodies into parts using the Form New Part tool, when transferred to Design Simulation they become parts consisting of multiple bodies with shared topology.

- The sketching toolboxes now contain tools for creating regular polygons and offsetting sketch edges.

- Multi body parts from SolidWorks can be attached to DesignModeler in this release.

Design Simulation

- Reaction moments are now reported in the Details View of a support object (fixed support, Given Displacement, etc.).
- Reaction heats are now reported in the Details View of any convection load.
- Useful for performing free body analyses or can be used as an alternative to using weak springs to stabilize the body, inertia relief may be enabled for a structural analysis.
- Coordinate systems will automatically be created if they are defined when geometry is read in via DesignModeler or SolidWorks.
- A new drop-down button on the Graphics Toolbar is available for selecting items (vertices, edges, etc.) through single selection (as was the only method previously), or through box selecting.
- The following legend enhancements are included in this release:
  - Right-clicking on the legend displays a menu for resetting to the default values.
  - User legend settings are retained when a figure is dragged to another result object.
  - Double-clicking a color in the legend allows the color to be changed.

- A color now appears that represents beyond the result maximum value.
- Adding new results and re-solving does not reset legends in figures.
- Shape checking is now included as an Advanced control with both a Standard setting, proven to be effective for linear, modal, stress, and thermal analyses; or an Aggressive setting, recommended for large deformation or material nonlinear analyses performed within the ANSYS Classic Environment.
- Options are now available for applying radial, axial, or tangential cylindrical supports, either individually or in any combination.
- A CAE template has been added that writes local coordinate system definitions through the use of the ANSYS LOCAL command.
- Material data sets in Autodesk Inventor, Pro/ENGINEER, and Unigraphics can now be imported into Design Simulation where the imported material data is isotropic and can include Young's modulus, Poisson's ratio, mass density, specific heat, thermal conductivity, and thermal expansion coefficient.
- The following new contact options are available in the Details View for a selected Contact Region:
  - Contact formulation [pure penalty, augmented Lagrange, or multipoint constraint (MPC)]
  - Initial interface treatment of the contact pair
  - Contact offset
  - Thermal conductance coefficient
- Line bodies are now supported for structural, modal, buckling, and harmonic analyses with a variety of loads and supports including applied forces and moments, fixed and simple supports with only the structural degree of freedom results being available for post-processing.
- Multi Body Parts, which are a collection of bodies, are now available.
- A remote force load is now available, equivalent to a force applied on a surface with the origin of the force being remote to the model.
- Vector result displays are now available for result plots of principal stresses and strains, deformation and heat flux.
- Local Coordinate System Creation now has two additional methods for creating a local coordinate system, which includes selecting multiple vertices, or by selecting a cylinder.

DesignXplorer APDL Support

DesignXplorer, which is based on the Design of Experiments (DOE) optimization method, supports the parameters associated with ANSYS Parametric Design Language (APDL). This means that legacy ANSYS APDL input files may take full advantage of the DesignXplorer capability, including the creation of design and analysis graphs, sensitivities, and optimization. See APDL Input files in DesignXplorer for further details.

DesignXplorer VT

DesignXplorer VT, using Variational Technology, provides a response surface based on a single finite element analysis. It relies on mesh morphing for CAD geometry parameters and a Taylor series expansion applied to the input and response parameters. The result is a speed increase of potentially several orders of magnitude over the DOE method, depending on the analysis problem. Additionally, DesignXplorer VT also includes the Design of Experiments (DOE) method, which DesignXplorer uses. DesignXplorer VT within the ANSYS Workbench includes the following capabilities:

- Analysis types:
  - Structural static, linear, elastic
  - Normal modes: You can compute the evolution of the eigen frequencies with respect to the input design parameters mentioned below (pre-stress modal is not included)

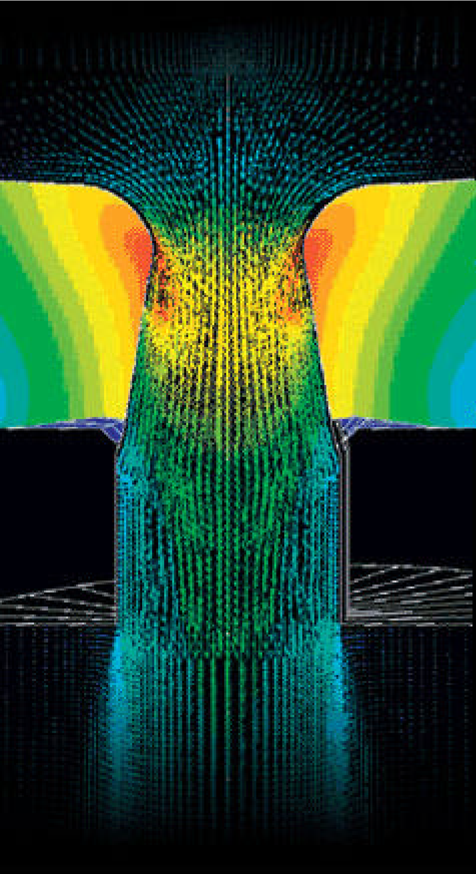
- Supported input design parameters:
  - Geometric shape parameters from supported CAD package interfaces and from DesignModeler
  - Material parameters (Young's modulus, Poisson's ratio, density)
  - Sheet thickness

- Spot welded sheet assemblies (no contact elements)

Fatigue

- Equivalent Alternating Stress, which is essentially the stress used to query the S-N curve to find life, is now available as a result item.
- Used to model cases where the model alternates between two completely different stress states (for example, between bending and torsional loading), Non-proportional loading is a new loading type designed for non-proportional, constant amplitude fatigue loading.
- If the fatigue loading is non-proportional, users can choose in the Details View, either the average or standard deviation of biaxiality.

Print this article | Email this article to a friend



Coupled-Field Solutions

Create Competitive Advantages

By Mark Troscinski, Multiphysics, Product Manager
Mechanical Business Unit, ANSYS Inc.

Now more than ever, companies compete in a furious race to market. By the time one company designs, tests and re-designs a product, another company has gained the lion's share of the business. Often times, the companies lagging behind are those that insist on running product designs through multiple, unrelated finite element analysis tools. And while they are piecing their results together, the successful companies have already begun investing in their next innovative product, thankful that they used a single, integrated FEA simulation tool to design and verify their products.

This idea of turning coupled-field solutions - two or more different, yet interrelated physical phenomena within one, unified simulation environment - into a competitive advantage gives companies the power to innovate, and improve the performance and quality of their products.

As the leader in multiphysics simulation, ANSYS Inc. offers the most comprehensive coupled-field computational tool available by combining the best structural, thermal, CFD, acoustic, as well as high and low-frequency electromagnetic capabilities into its flagship product, ANSYS Multiphysics. ANSYS Multiphysics is a state-of-the-art, multidisciplinary analysis tool that allows engineers and scientists, from all industries and disciplines, to gain a solid understanding of their own product performance characteristics before developing costly physical prototypes. Currently the only software tool on the market that provides all major physics simulation capabilities within one, unified environment, without need to purchase add-on modules, ANSYS Multiphysics is truly the premier virtual prototyping tool developed to complement the physical stages of the design process.

In addition to having outstanding single physics simulation capabilities, within the ANSYS Multiphysics product lurks a phenomenal tool for Fluid-Solid Interaction. FSI occurs where a fluid flowing around or within a structure causes it to move, spin or even change shape due to flow-induced pressure and shear loads. FSI is often a transient occurrence, where the structural motions are dynamic and continuously vary with time. In certain cases, FSI can be steady-state where the loads induced by the fluid are exactly balanced by the structure's reaction forces, and the structure reaches a displaced equilibrium position in the fluid stream. In either case, FSI simulations involve multiphysics coupling of fluid and solid mechanics.

As companies continue to race toward market victory, the winners know the fastest way to get there is by incorporating ANSYS Multiphysics within their product design process. With coupled-field simulations that more closely match reality, all managed from within a single, unified graphical user interface, ANSYS Multiphysics can also help you win the "checkered flag" in the race to market for your own products.

The All Together Now Featuring ANSYS Multiphysics seminar is heading your way!

For locations, dates, and times
www.ansys.com/seminars.htm

Learn how leading companies are benefiting from ANSYS Multiphysics solutions in ways that include...

- More accurate and realistic simulations that lead to superior designs
- Fewer software tools to purchase, learn and maintain
- Process compression through more flexible and efficient design workflow

Who should attend?

The All Together Now seminar is designed for engineers, product designers and managers who play a role in product development and verification at their company.

What will be discussed?

Using real-world examples, the All Together Now seminar will highlight ANSYS' multiphysics capabilities and their impact on high-fidelity design. Best practices, other implementation issues and critical success factors for a cost-effective and quick deployment will be discussed as well as a live demonstration of ANSYS Multiphysics' pre-processing, solution and post-processing features. Plus, you'll have the opportunity to discuss your applications and related challenges one-on-one with our experts.

For locations, dates, and times: www.ansys.com/seminars.htm

Print this article | Email this article to a friend

Fewer Prototypes for Naval Shock Mounts

ANSYS simulates underwater explosive conditions to optimize vibration-damping system

BAE SYSTEMS' Engineering Dynamics team has more than 50 years of experience in structural design and simulation. Its team of mechanical engineers has used finite element analysis extensively, to assess the effects of non-contact underwater explosions on ships and submarines, designed using composite materials, structural optimization, and dynamic simulation. The team uses ANSYS, composite design software, and in-house codes to simulate static, dynamic, and non-contact shock loading.

Non-contact underwater shocks from mines or other sources can cause severe damage to naval vessels and their equipment.

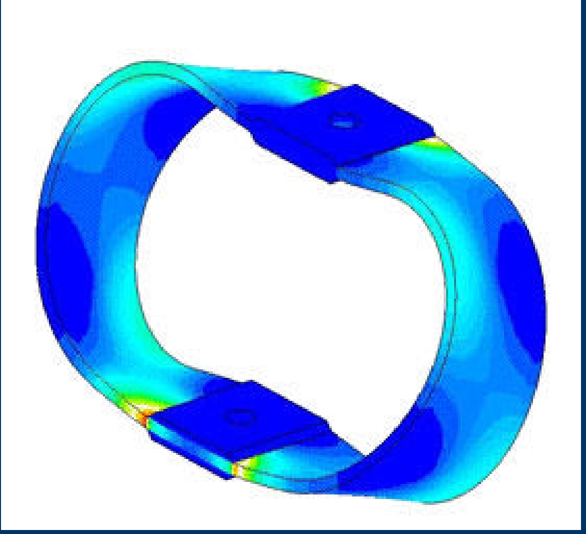
Naval ships and submarines are exposed to many types of shocks, whether at war, or during peacetime. Non-contact underwater shocks, from mines or other sources, can cause severe damage to these vessels and their equipment. In many cases, however, precautions taken during the design stages can minimize the damage. This is where shock simulation is paramount.

For the past two years, Scott Lafferty, a Consultant Engineer for BAE SYSTEMS, has been working to develop a ‘Composite Shock and Anti-Vibration Mount’ for the Naval Fleet of the British Ministry of Defence. This mount will significantly reduce acceleration levels and transmitted vibrations to the machinery it supports during severe non-contact underwater explosions and normal operation.

Design Challenges

For weapons and other vital shipboard equipment to remain functional during combat, they must be able to withstand severe shocks. Unfortunately, designing such shock-resistant equipment is costly and difficult, especially because of space and weight constraints.

The mount deflects freely throughout the shock motion to transfer force gradually and thereby lower acceleration levels that could damage equipment.



The best way to protect the equipment is to affix it to shock mounts, which will lessen the force of the shock, by transferring it to the equipment gradually, making it more of a quasi-static event than a dynamic one. During a shock event, significant deflections occur, so the mount must be able to deflect freely throughout the shock motion. Otherwise, the acceleration levels may be greater than the initial shock, which can damage the equipment.

The ship or submarine may also have noise requirements for quiet running, which make it necessary for some equipment to be flexibly mounted. Although there are many anti-vibration mounts on the market, few meet the deflection capability requirements of a shock mount. Shock mounts have occasionally been used in tandem with anti-vibration mounts, but those systems are expensive to design, purchase and fit.

The Ministry of Defence Sea Technology Group commissioned BAE SYSTEMS' Engineering Dynamics Team to design a cost-effective shock and anti-vibration mount. Among the extensive list of specifications and requirements, was the stipulation that it had to be capable of being retrofitted to replace existing shock mounts. This immediately restricted the possibilities in design, because it meant that most of the dimensions were already fixed.

Another challenge was that the mount had to possess specific vertical and horizontal stiffness that would be high enough to safely support equipment and low enough to isolate vibrations, while having the proper damping characteristics to absorb shock and vibration.

Simulation Solution

To meet the timeframe, budget, and specifications of the Ministry of Defence, the Engineering Dynamics Team immediately turned to ANSYS simulation software. At the outset, a series of simplified design iterations showed that it would be best to design the mount using composite materials modeled in ANSYS. The team has used this procedure many times; for static, dynamic, and shock analyses of substantial composite structures, including mine hunter machinery rafts, support brackets, pipe, and pipe joints.

The team first chose a material, Epoxy/E-glass, based on cost, mechanical properties, chemical and environmental resistance, temperature stability, and the fire-retardant properties of its resin. The material's manufacturer conducted initial tests to identify the properties of a single layer of the material, which were used within ANSYS to represent the lamina layer properties.

"Over the period when the manufacturer was performing the lamina tests, a mount was modeled in ANSYS, with the existing mount dimensions, using isotropic material properties," explains Lafferty. "We then set up ANSYS to automatically perform design optimization on several of the minor dimensions. The objective function was set as the stress level at a defined location, the state variables were set as the required stiffness, and the design variables were set as the minor dimensions. From this assessment, the trends in mount stiffness, stress, and geometry were developed from which the optimum mount shape was identified." He adds that, "As in most design cases, what we call the optimum configuration is purely a compromise, and represents the best combination of factors, such as stress, stiffness, weight, and cost."

Next, a finite element (FE) model was developed, using the elements with the lamina properties specified by the manufacturer. A starting lay-up was identified, then modeled in ANSYS, with its sequences and fiber orientations set to vary automatically, using the design optimization algorithms in ANSYS. Design trend graphs were then developed, showing the variations in mount stiffness and stress that would be caused by modifying the parameters, and identifying a lay-up that would give the proper vertical and horizontal stiffness, while it minimized the stress state within the mount.

In addition, a series of 3-D sub-models was developed to find a range of possible configurations, fastening methods, and materials for the mount fixings, which are crucial to the design because of their effects on stress and stiffness. The results led the team to a design that would minimize these effects, and which could be manufactured consistently and economically.

A detailed 3-D FE model was then developed with the optimized shape, lay-up, and fixing design. Analysis showed that it would meet all of the specifications and safety factors required by the Ministry of Defence.
For the non-contact shock loading simulation, the team modeled equipment supported on the composite mount, and applied a shock load to the base of the mount. The equipment response results showed that, because of the low damping ability of the composite material, the shock could have damaged the equipment, and, perhaps, injured personnel.

The next step was to perform a series of tests to ascertain an acceptable level of damping. The team still had to come up with a damping mechanism that would be inexpensive and easy to manufacture, without affecting the static/quasi-static characteristics of the composite mount. After investigating many options, they selected a material that is normally used to reduce structural resonance and impact-induced noise on large flat panels.

A 3-D harmonic model of the composite mount, damping material, and equipment was developed in ANSYS to determine the mount's damping characteristics. Analysis established the optimum quantity of the damping material and its positioning on the mount, to achieve the best damping characteristics. The results indicated that this design would increase the damping by 200%.

Using the design details established from the ANSYS analyses, a batch of prototype mounts was manufactured. Results of quasi-static and dynamic tests on these prototypes, in comparison with the original design specifications and the FE models, showed that the stiffness of the mount in the vertical direction was within 2% of the result calculated in ANSYS. The horizontal stiffness was within 5%, an error that was slightly higher because of the complexity of the horizontal loading conditions. No audible or visible matrix/fiber failure was found in any of the tests. The dynamic tests indicated that the introduction of the damping material had, in fact, increased the mount's damping by 202%, which was impressively close to the 200% that was predicted.

ANSYS Benefits

Throughout the project, the use of ANSYS in design has drastically reduced the number of prototypes and tests needed on the mount. ANSYS made it possible for the team to simulate complex explosive conditions that could not have been duplicated in a laboratory. It also allowed the engineers to modify designs quickly and easily, and perform numerous design iterations, which would have been very expensive to test on prototypes.

As Lafferty notes, "The Engineering Dynamics team uses this method of design by analysis extensively, mainly to meet shorter timescales and to remain commercially competitive."

The final, and defining, shock test of the mount is scheduled for mid-July. All of the simulations indicate that it will be successful, and that the acceleration of the mounted equipment will be significantly less than the shock input, meeting all of the requirements stipulated by the Ministry of Defence.

This mount was designed to support a specific mass range. During the next phase of the project, it is expected that the design will be extended to support an increased range of equipment masses. At the same time, a set of composite mounts will be fitted to a Navy vessel for onboard trials.

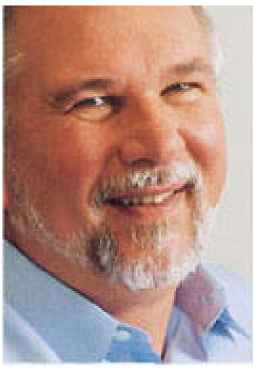
Lafferty says, “The composite shock and anti-vibration mount has been designed and developed using primarily ANSYS software. Extensive analyses have been performed using the static, nonlinear static, harmonic, nonlinear transient dynamic, and optimization algorithms. The use of design simulation using ANSYS has allowed us to reduce timescales and supply prototype mounts at minimum cost. The resultant mount performs as ANSYS predicted, and we will, therefore, use ANSYS extensively for the next phase of the project.”

Print this article

Email this article to a friend

ROI for Simulation

The greatest business value isn't incremental bottom-line savings but top-line revenue growth.



John Krouse
Editorial Director

Companies use a wide range of engineering simulation tools for studying stress loads, fluid flow, heat distribution, electromagnetic fields, fatigue life, and other critical product attributes. The software is becoming indispensable in designing a broad spectrum of manufactured products from automobiles, aircraft, and heavy machinery, to cell phones, recreational equipment , toys, and home appliances.

Decades ago, these programs - for the most part - were quirky, esoteric codes used only by handfuls of analysts to troubleshoot isolated problems. If a part failed during final testing or if too many service reports came in from the field, the problem was “thrown over the wall” to the analyst, who modeled the component, ran an analysis, and issued a report - often weeks later - to the waiting engineers.

The expense of the software, computers, and skilled personnel was justified based on cost savings, with a return on investment (ROI) determined by comparing these amounts with what a company would expect to pay otherwise - if solely testing physical prototypes, for example, did the job.

If a single mock-up of a product takes three weeks to build and costs $50,000, then eliminating just one of these testing cycles lets you shave that amount of time and money off the development cycle. Multiplying that times the number of products tested in a year comes out to some substantial annual savings - a no-brainer as far as the financial people are concerned.

Simulation software and the ways companies implement it have come a long way since then. Programs can handle much larger problems considerably faster, of course, due to advances in hardware and software technology. Moreover, many packages can now be used directly by engineers in performing some analysis tasks on their own. Also, programs are much more closely integrated - not only to other simulation packages but also to software for design, manufacturing, and product data management.

Through these capabilities, simulation can now be more seamlessly woven into the entire product lifecycle, with the technology used in upfront studies during conceptual design and throughout development. Analysis data resides not just in the engineering department but is used in groups across the entire product lifecycle, including manufacturing, customer service, and field support.

In this way, simulation has become an integral part of the development process in its entirety and is a key element in the triad of people, process, and technology needed to develop innovative products, as is noted in Mike Wheeler's article “Simulation Tools for Design Innovation” in this issue. The article covers some of these powerful solutions including first-pass tools for performing analysis early in the design cycle, advanced optimization technology for refining product designs, and virtual prototyping methods for evaluating how products will perform in actual operating conditions.

Without simulation tools such as these, companies would be hard-pressed to bring innovative products to market quickly and cost effectively. Simulation enables engineers to perform what-if studies and evaluate alternatives that would be impractical otherwise. It allows product behavior to be evaluated in greater detail and designs optimized automatically according to specified attributes. And it predicts how products will operate before a single piece of hardware is built.

Simulation has thus evolved to a new role in the manufacturing enterprise. The technology is no longer exclusively a cost-cutter and time-saver, although those qualities alone are generally more than sufficient to justify an investment in these tools. Rather, simulation is now key at a growing number of smart companies in their processes to develop winning products. As Dave Weisberg points out in his column "Recognizing the Critical Value of Engineering Software" in this issue, these types of products bring money into the company and are critical for a manufacturer to define the brand value of its products, capture greater market share, and strengthen its competitive position. Without winning products, no amount of cost savings matter.

In this respect, the greatest business value simulation provides is not incremental bottom-line savings but its contribution in driving top-line revenue growth, which for many companies could make the critical difference in their surviving a brutal economy.

Industry News

Recent Announcements and Upcoming Events

ANSYS Provides Increased Accessibility to Advanced CFD with CFX-5.6

In June, ANSYS announced the release of CFX-5.6, which provides increased accessibility to advanced computational fluid dynamics (CFD). CFX is used by companies around the world in a wide range of industries including aerospace, chemical processing, power generation, automotive, oil and gas, heating, ventilation and air conditioning, and many others to improve processes, solve performance issues and vastly reduce product time to market.

CFX-5 is the technology leader with its coupled multigrid solver that solves the full system of hydrodynamic equations simultaneously. This methodology, in addition to robustly providing highly accurate results, also is the basis for industry-leading parallel efficiency, thus allowing companies to solve these computationally intensive problems in far less time than ever before. Further, CFX-5 delivers near total interoperability between its wide range of models providing the realistic picture that leads to informed design decisions.

“The parallel processing capabilities, as well as the unstructured solver capabilities of CFX-5 have allowed us to perform analyses in as little as three days on complex geometries that used to take as much as two weeks,” said Jason Kopko, Aerodynamic/Thermodynamic Engineer, Dresser-Rand. “This substantial time savings allow us to perform more iterations, as well as more detailed analyses than would have been possible in the past.” The latest release of CFX-5 includes CFX-Pre, an intuitive physics pre-processor that dramatically simplifies simulation set-up. Its interactive interface provides easy definition and quick navigation of all aspects of the CFD problem definition. Graphical and quantitative post-processing capabilities also have been extended through the addition of turbomachinery-specific post-processing to allow quick evaluation and improvement of rotating machinery applications. While presenting an intuitive interactive user interface, all components of the flexible CFX-5 system also offer advanced programmability and customization for automation and integration of CFD into the engineering design process.

“CFX-5.6's new physics pre-processor has tons of useful features that make problem set-up very easy. I am particularly impressed with CFX-Pre's ability to import and assemble meshes very quickly," said Peter Bostwick, Engineer Specialist - CFD/Heat Transfer, Emerson Motor Company.

In this release, CFX-5 expands its rich range of physical models with Lagrangian particle tracking, advanced combustion models, access to CFX's proven radiation solver, real gas thermodynamics and interphase mass transfer with phase change. Examples of expanded industrial applications include improved pump performance and wear through the prediction and suppression of cavitation, or occupant comfort and room ventilation in HVAC applications.

"We continue to build CFX-5 on our foundation of advanced CFD technology to provide the superior speed, performance and usability that the industry requires to be competitive," said Dr. Michael Raw, vice president of product development for ANSYS' fluids business. "The addition of CFX-Pre provides a tool to allow both new and experienced users to quickly set up problem physics and obtain solutions that are both fast and accurate."

ITI TranscenData Announces Newest CADIQ Customer

MILFORD, OH; May 5, 2003 - ITI TranscenData, the product data interoperability business of International TechneGroup Incorporated (ITI), today announces that Motorola Inc. (NYSE: MOT) has selected CADIQ to help improve product development speed and efficiency. Motorola's Personal Communications Sector (PCS) division, providers of wireless subscriber and server equipment, will utilize ITI TranscenData's CADIQ model quality software to inspect CAD models for hidden defects and errors thus assuring that quality data is being provided to downstream suppliers.

"Time spent reworking CAD models because of hidden defects can spell the difference between first to market and missing the window of opportunity," explained Don Hemmelgarn, President of ITI TranscenData. "Model quality programs allow organizations like Motorola to address downstream interoperability issues in the early stages of design. As a result days or weeks of delay, rework and associated costs are eliminated," continued Hemmelgarn. "We are delighted to be entrusted with the CAD model interoperability success of Motorola."

Front-End Simulation Simplified with ANSYS 7.1

New Variational Technology and Concept Modeling Tools Promise to Accelerate Time-to-Market

With the aim to encourage design innovation and improve product quality in companies, ANSYS Inc. announced the availability of ANSYS® 7.1, featuring two technologies that will make designing and analyzing new products faster and easier, permitting more design innovations, and ultimately driving higher-quality products to market faster.

ANSYS DesignXplorer VT™ brings the power of variational technology (VT) to the designer's desktop, giving users the ability to visualize the effects of a multitude of simultaneously varying design parameters in a single solve. DesignModeler™ is a new concept-modeling tool that gives users a better way to approach the early stages of design for large, CAD-intensive structural projects. ANSYS 7.1 is the first product suite in the computer-aided engineering (CAE) industry to offer these types of capabilities to users.

These robust, easy-to-use tools support engineering design and simulation capabilities early in the product development process, when design innovations are explored and most of the manufacturing costs are determined.

Meanwhile, the heart of ANSYS 7.1-ANSYS Workbench Environment™ offers a newly improved user interface, superior CAD integration, automatic meshing, and access to model parameters and other ANSYS features.

“ANSYS 7.1 provides product engineers with more data to make smarter decisions - and smarter decisions mean better products,” said Mike Wheeler, vice president and general manager of the mechanical business at ANSYS. “Our combination of variational technology and concept modeling is an industry first. We are proud to offer complete, integrated solution that shorten the product engineering process and help bring better products to market faster.”

ANSYS 7.1 Adds Variational Technology and Concept Modeling to Product Lineup With ANSYS 7.1, design engineers and analysts have new tools to speed product development. In June, ANSYS announced the availability of ANSYS® 7.1, featuring the company's Variational Technology (VT) within DesignXplorer VT(tm), and its concept modeling tool DesignModeler® . ANSYS 7.1 will help designers and analysts rapidly prototype, and test new and existing products.

The latest program components from ANSYS allow designers to visualize the effects of varying parameters on a design early in the product development process as well as speed time-to-market and improve product quality. ANSYS 7.1 is the first product in the CAE industry to offer these types of capabilities to users. More details on the two newly integrated products follow:

DesignXplorer VT

ANSYS 7.1's DesignXplorer VT brings the power of variational technology (VT) to the designer's desktop, giving users the ability to visualize the effects of a multitude of simultaneously varying design parameters in a single solve. DesignXplorer VT's proprietary variational technology offers the capacity to study, quantify and graph the structural analysis response to alternative design parameters for both linear elastic static stress and normal modes analyses. With VT, users can approach product design decisions much more efficiently. Depending on the analysis problem, VT can provide acceleration factors between 10 and several thousand.

When combined with the ANSYS® Workbench Environment(tm), this powerful application allows designers and analysts to make intelligent design directives given multiple competing objectives. Changes can include geometric CAD parameters as well as material properties and thicknesses. The combination of DesignXplorer VT and the ANSYS Classic Environment(tm) gives analysts have the ability to gain a detailed understanding of their designs.

DesignModeler
ANSYS DesignModeler is a new concept-modeling tool that gives users a better way to approach the early stages of design for large, CAD-intensive structural projects. Concept Modeler allows the designer to work with mathematical abstractions of the physical world. Supporting both beam and shell abstractions, concept modeling can begin with just lines and surfaces, as full solid models are not required. The concept-modeling product supports the following cross sections for beam:

Ability to apply 11 different attributes to Line Bodies:

- Rectangular
- Circular
- Circular Tube
- Channel Section
- I Section
- Z Section
- L Section
- T Section
- Hat Section
- Rectangular Tube
- User Integrated

ANSYS 7.1 Improvements

Workbench Start/Project Page, Project Tabs
Upon initiating ANSYS 7.1 in the Workbench Environment, an intuitive Start Page now is displayed that is intended as a management point for creating or editing projects. The Start Page has provisions for creating and naming new projects, selecting existing projects, defining preferences for CAD importing, and selecting the main project task module (DesignModeler, Design Simulation, or DesignXplorer). Once a user is inside a project task, tabs are displayed along the top that represent each of the project task modules for the session, as well as a Project tab. Clicking any of the module tabs will move the project to that module (i.e., from Design Simulation to DesignXplorer). From the Project Page, users also can move to another module, including opening the analysis in the ANSYS Classic Environment.

Contact Elements now have an Internal MPC Algorithm
A new internal multipoint constraint (MPC) algorithm is available for the surface-to-surface and point-to-surface contact elements. When using this method, ANSYS 7.1 builds MPC equations internally based on the contact kinematics. The internal MPC approach can overcome the drawbacks of the traditional contact algorithms and other multipoint constraint methods. When combined with bonded contact, the MPC approach facilitates the following types of contact assemblies and kinematic constraints:

- Solid-solid assembly
- Shell-shell assembly
- Shell-solid assembly
- Beam-solid assembly
- Rigid surface constraint
- Force-distributed over an arbitrary surface

Fluid-Solid Interaction (FSI) Analysis Using MpCCI
User now can perform fluid-solid interaction analyses using ANSYS and a third-party CFD code. In this method, the Mesh-based Parallel Code Coupling Interface (MpCCI) couples ANSYS with the CFD code. MpCCI is able to transfer quantities across a fluid-solid interface even if the meshes are dissimilar.

“ANSYS 7.1 provides product engineers with more data to make smarter decisions - and smarter decisions mean better products,” said Mike Wheeler, vice president and general manager of the mechanical business at ANSYS. “Our combination of variational technology and concept modeling is an industry first. We are proud to offer a complete, integrated solution that shortens the product engineering process and help bring better products to market faster.”

At the heart of ANSYS 7.1 is the ANSYS Workbench Environment, offering an efficient and intuitive user interface, superior CAD integration, automatic meshing, access to model parameters, as well as access to advanced ANSYS functionality.

ANSYS Announces AI*Environment

As a powerful new generation pre- and post-processing tool, ANSYS' latest product provides a direct path from CAD to analysis. Inc. (NASDAQ: ANSS), a global innovator of simulation software and technologies designed to optimize product development processes, today announced the release of AI*Environment™, a robust pre- and post-processing tool to support complex and advanced simulation problems faced by engineering analysts. By applying the company's ICEM CFD Engineering technology to the structural modeling problem, AI*Environment is designed to provide a direct path from CAD to analysis and reduce time and efforts required in preparing data for large simulations.

As a FEA-centric pre- and post-processing tool, AI*Environment works with several geometry sources, including a parametric link to leading CAD systems. AI*Environment provides extensive geometry repair tools that are useful in preparation for meshing, as well as comprehensive healing, world-class meshing and mesh editing capabilities. This powerful combination of tools leads to a software product that significantly reduces the time for simulation compared to traditional pre-processing tools used by analysts.

“With our proven track record as the #1 pre-processor for CFD modeling, we know how to rise to a challenge," said Devendra Rajwade, general manager of the Environment Business Unit at ANSYS Inc. "We see similar needs in the structural analysis environment. Combining our strengths in automation, high-quality meshing, and robust geometry handling and an easy-to-use interface, we have created a tool that will appeal strongly to any engineer while doing everyday FEA analysis to advanced multidisciplinary simulation."

"Automotive and aerospace customers deal with some of the most interesting -but also some of the largest and most complicated - engineering simulation problems known to man," said Steve Pilz, product manager for ANSYS Inc.

"AI*Environment extends the well proven meshing methodology - top-down and geometry independent meshing - used extensively by designers and analysts from CFD simulation to FEA simulations. With the flexibility and robustness provided by AI*Environment, the engineering analysts will be able to produce results for complex simulations in a fraction of the time and play a leading role in product's development cycle."

AI*Environment boasts key engineering advantages such as robust geometry handling with parametric link to commercial CAD packages, extensive geometry repair/feature suppression tools, mid-plane extractions/extension, powerful mesh editing, multiple solver support, batch meshing, advanced boundary conditions and reliable large model post-processing, on both Unix and Windows operating systems. Additionally, patch independent meshing methodology enables users to capture the geometric detail based on a tolerance or minimum element edge length setting. AI*Environment patch independent meshers can rapidly compress previously unsolvable problems down to produce same day solutions, keeping the product development schedule intact.

ANSYS Inc. Recognized as a BusinessWeek "HOT GROWTH COMPANY" and Named to CNN Money's "Fabulous 40"

For the fourth time, ANSYS Inc. earned recognition as one of BusinessWeek's 100 best "Hot Growth Companies" as well as one of CNN Money's "Fabulous 40." BusinessWeek annually evaluates organizations based on sales growth, earnings growth and return on invested capital. Previously number 55 on the list, ANSYS Inc. moved up to 48. CNN Money's "Fabulous 40" tracks the shares of technology stocks since the close of trading on March 10, 2000 and recognizes those whose stock is trading at a higher price as of May 27, 2003, than they were on the Nasdaq's peak of March 10, 2000. The article is available at http://money.cnn.com/pf/features/lists/technology_stocks/.

"It all stems from our dedication to providing customers with the broad range of simulation software tools they deem critical for ensuring they are ahead of their competition and stay ahead," said Jim Cashman, president and CEO, ANSYS Inc. "ANSYS' response to our customers' evolving engineering needs results in innovative, superior-quality products and leads to savings in time, costs and productivity. ANSYS software positively affects their business, which in turn is reflected in our continued solid financial performance. This type of recognition would not be possible without the loyalty of our customers, or the dedication of our employees and our channel partners, to provide the most comprehensive computer-aided engineering solutions available."

Autodesk Improves Homeland Security Efforts for City of West Palm Beach

Autodesk, Inc. announced that the City of West Palm Beach, Florida, has selected Autodesk MapGuide® software to manage and distribute GIS information to all city employees and the general public. The city is using Autodesk MapGuide software to improve the sharing of homeland security and emergency information among city departments, and to power a GIS Web site that allows visitors to select from more than 100 types of maps.

The City of West Palm Beach has 11 departments that must communicate and share a wide variety of information ranging from hazardous materials sites, to parcel owner records and infrastructure management. Because the city's GIS department has just one staff person, West Palm Beach required a mapping distribution solution that easily incorporates existing data without conversion, and is easy for non-technical users to understand and access.

"Using (Autodesk) MapGuide, I was able to make GIS information available citywide within 30 days of implementation - about three months sooner than expected," said Nestor Navarro, GIS Coordinator for the City of West Palm Beach. "As the city's only GIS employee, I needed a solution that would not require a significant investment of my time. (The Autodesk) MapGuide authoring tools, and its ability to publish city maps on the Internet, allowed me to meet these challenges."

Autodesk MapGuide Software Helps Identify, Manage Emergency Sites Prior to the Autodesk MapGuide implementation, West Palm Beach's departments typically shared GIS information using static paper maps, since data from one department's database usually didn't correspond with data from another department. Public access was limited and slow: Individuals had to request maps through the City Clerk's office, which then passed on the request to the GIS department. Owing to the time required for creating such maps, the public was charged a materials fee.

After the implementation, city departments not only began using GIS data for daily operations, such as emergency planning and permit issuance, they began providing the GIS department with richer information for the entire mapping system. When the GIS department first introduced online maps via its Web site, it offered 63 types; today, more than 100 types are available. The public can now access these maps via the city's GIS Web site, without making appointments with the city clerk or paying fees.

West Palm Beach police and fire departments also use online maps for homeland security initiatives, put into place after the Sept. 11, 2001, terrorist attacks. For instance, police and fire officials use mapping data to identify sites that might be terrorist targets, as well as those containing hazardous materials. In addition, emergency officials in West Palm Beach use online maps to plot evacuation and debris removal routes in the case of natural disasters such as hurricanes. Autodesk MapGuide software is frequently used by state and local governments to manage GIS information for emergency situations. The Florida Emergency Operations Center, which oversees statewide disaster response activities, will be using Autodesk MapGuide in order to provide information to the public on hurricane preparedness and response.

“Autodesk MapGuide software's ability to bring together GIS information from diverse sources and diverse platforms makes it the logical choice for local governments needing fast and cost-effective GIS solutions,” said Richard Neiman, founder of CADD Centers of Florida, the Autodesk reseller that supplied West Palm Beach with the Autodesk MapGuide application. “The short implementation time and easy-to-use browser interfaces in (Autodesk) MapGuide guarantee a fast return on investment in the solution.”

SGI Altix 3000 Gains Momentum in the Manufacturing

Industry with Major MCAE Software Now Available

SGI recently announced that industry-leading mechanical computer-aided engineering (MCAE) software applications are immediately available on the SGI® AltixTM 3000 family of servers and superclusters, the first high-performance Linux® HPC environment capable of scaling to hundreds of processors with global shared memory. A variety of industries, including automotive, aerospace and defense, and general manufacturing, rely on MCAE applications to improve design quality and reduce design-cycle time and costs. By combining the leading HPC technologies of the Intel® Itanium® 2 processors, the open-source Linux operating system and SGI® NUMAflexTM system architecture, the SGI Altix 3000 family of servers and superclusters offers the capability to meet the essential needs of MCAE simulation environments for advanced product development.

Manufacturing companies and research organizations employ commercial MCAE software from a variety of independent software vendors. The list of MCAE applications immediately available for SGI Altix includes ABAQUS® from ABAQUS Inc., LS-DYNA® from Livermore Software Technology Corp., MSC.Nastran™ and MSC.Marc™ from MSC.Software Corp., and STAR-CD™ from CD adapco Group. Over the next several months, additional MCAE software is undergoing development and is expected to become available soon. This software includes ANSYS® from ANSYS Inc., CFX-5™ from CFX (a business unit of ANSYS Inc.), FLUENT® from Fluent Inc., PAM-CRASH® from ESI Group, PowerFLOW® from Exa Corp. and RADIOSS™ from Mecalog SARL. “Today, more than ever, manufacturing industry solution providers such as SGI need software application partners that can help them deliver total solutions to the market faster,” said Larry McArthur, senior director, manufacturing industry marketing, for SGI. “Manufacturing companies and research organizations of every size want powerful yet flexible applications and solution infrastructures that will help them bring improved efficiencies and innovations to product development. SGI's relationships with key software partners are built on complementary technology strengths that can help our customers advance their engineering capabilities.”

In January, SGI announced the SGI Altix 3000 family of servers and superclusters, which combine SGI® supercomputing architecture with Intel Itanium 2 processors and the Linux operating system. SGI Altix 3000 is recognized as the first Linux cluster that scales up to 64 processors within each single Linux OS based image node and the first cluster of any variety to allow global shared-memory access across nodes. Since its launch earlier this year, the SGI Altix 3000 family of Itanium 2 processor-based systems has shattered scalability and performance records on high-performance computing industry-standard benchmarks.

“SGI has put forth a compelling vision for supercomputing, an area where edge-of-the-art hardware performance is still critical. Its server strategy gives customers the freedom to choose between their existing MIPS and IRIX environment and an Intel Itanium processor and Linux OS-based product,” said Bruce Jenkins, executive vice president, Daratech, Inc. “By leveraging the Intel platform and continuing to work with leading MCAE developers to optimize applications performance, we foresee SGI making high productivity computing solutions more broadly available to manufacturing organizations than ever before.”

With the introduction of the SGI Altix 3000 family of servers and superclusters, SGI offers industry-leading price/performance for application of advanced MCAE simulations. Configurations include the SGI® Altix™ 3300 server, a midrange configuration with 4, 8, or 12 Itanium 2 processors and up to 96GB of memory, and the SGI® Altix™3700 supercluster, a large, single-node configuration with up to 64 Intel Itanium 2 processors and up to 512GB of memory. Altix 3700 can scale to 2,048 processors with multiple 64-processor nodes in a cluster configuration.

Upcoming Events

Coal-Gen 2003
August 6-8
Columbus, OH

Daratech iDPS Aerospace Event
September/October 2003
Los Angeles, CA

PowerGen 2003
December 9-11
Las Vegas, NV
Discovery Tour: All Together Now featuring ANSYS Multiphysics™

Print this article   Email this article to a friend

Calling All Techies

Biennial ANSYS Users Conference Offers Users the Unique Opportunity to Showcase Engineering Successes.

Continuing what avid ANSYS users consider a CAE tradition, ANSYS Inc. has announced its Call for Papers for the 2004 ANSYS Users Conference and Exhibition to be held from May 24-26, 2004 in Pittsburgh, Pennsylvania, U.S.A.

Preparations are underway for the biennial event, and ANSYS, ICEM, CFD and CFX-related technical paper abstracts are being accepted until August 15, 2003. The technical paper portion of the conference provides users the opportunity to tell their story to colleagues and showcase their organization.

In addition to the personal rewards, ANSYS distributes awards for the best paper in each technical session. The best overall conference technical paper author will be awarded conference travel compensation, hotel accommodations, and free admission to the conference, while second, third and fourth place authors will be awarded free admission to the conference.

"Technical paper topics address the complete spectrum of engineering applications from biomedical to aerospace," said Bill Bryan, conference chairman at ANSYS Inc. "Suggested topics include accuracy in FEA, coupled-field analysis, fluid structure interaction, MEMS simulation, and probabilistic design."

All technical paper abstracts will be accepted electronically. To download the forms, please visit www.ansys.com/conf_2004.

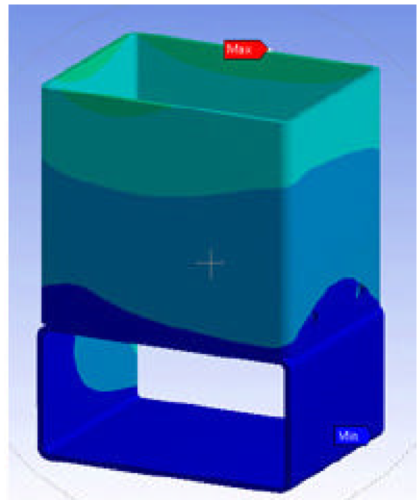
About the 2004 ANSYS Users Conference and Exhibition

Since 1983, ANSYS Inc. has hosted the ANSYS Users Conference and Exhibition to showcase the advances in computer-aided engineering and related technologies. The 3-day conference addresses the complete spectrum of engineering professionals including engineers, analysts and engineering managers. Conference attendees have the opportunity to participate in a variety of breakout sessions, seminars, and roundtable discussions.

Print this article | Email this article to a friend

Reaching New Heights

Champion Elevator finds that ANSYS DesignSpace helps them reach for new and higher standards



ANSYS DesignSpace™ screen image of the redesigned car stand for Champion Elevators, Houston, Texas. Redsign increased strength while saving weight.

In the heavily regulated world of high-rise construction, rack-and-pinion elevators that move workers, tools and materials to upper stories on a tower must meet two different sets of standards. One set of standards is comprised of building and electrical codes and are essentially regulatory. The other is safety standards and these are essentially physical properties.

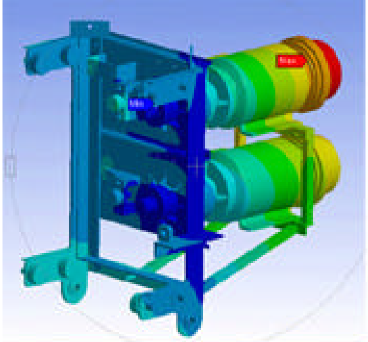
Located just west of Houston's Hobby Airport, Champion is the acknowledged leader in the design and installation of rack-and-pinion driven elevators. Aside from construction, its biggest market, Champion elevators are on offshore oil and gas rigs, refineries and power plants; in port facilities and shipyards and on ships; in or on the outsides of buildings, towers, iron and steel mills and bridges. Six Champion elevators are on New York City's George Washington Bridge. Champion elevators are also in mines and tunnels.

Champion Elevators' basic design is a tower assembled from standard segments, rigid ties to connect it to the exterior of the structure, electrically powered rack-and-pinion drives, sheet metal elevator cars, automatic braking (in case of power, over speed or safety device failures) and spring-like buffers at the bottoms of the towers. The towers are rectangular tubes with channels and stiffening grids fabricated in five-foot sections. The rack for rack-and-pinion drive is pre-mounted on the tower's exterior.

After almost three decades in the business, the people at Champion Elevators Inc., Houston, know that conformance to building codes is a must. Given the obvious risks, engineers at Champion run every job through analysis.

Challenge

Assuring safety and conformance to the codes and regulations fall into two very different types of engineering analyses. Safety assurances of the elevator — essentially measuring maximum stresses and ensuring adequate safety margins — is handled with ANSYS DesignSpace® software for finite element modeling and finite element analysis (FEM/FEA) from ANSYS Inc., Canonsburg, Pa., U.S.A.



ANSYS DesignSpace screen image of the redesigned elevator car drives and undercounterweighted mountings. Redesign increased strength and made ride smoother while saving weight.

Code and regulatory conformance of the tower is verified with beam modelers designed with specific regulatory codes at their analytical hearts. Sometimes called “stick modelers,” these packages allow for rapid modeling and analysis of the towers.

Like DesignSpace, beam modelers support finite element modeling and analysis. “Here at Champion, the physical values generated for the codes by the beam modelers are used as input to DesignSpace,” explained Bradley D. Oliver, P.E., Senior Engineer. “We use the deformation information for verifying conformance to codes because that is how codes are expressed,” he noted. “The forces generating those deformations have to be derived for each installation. They are always different.”

The beam package used by Champion does static and dynamic analysis in steel, concrete and composite-materials designs. It can calculate section properties of custom and built-up shapes including area, moments of inertia, section modulus, center of gravity, shear center, and torsional constants. The package is built around the codes of the American Society of Civil Engineers (ASCE) and is configured to show conformance to specific sections of this most fundamental construction-engineering code.

Without 3-D solid meshing, however, beam modelers cannot generate sufficient data for the stress calculations that lie at the heart of margins of safety. Nor can they be used very effectively for design verification, the task for which ANSYS originally conceived DesignSpace.

A typical Champion Elevator installation on a refinery stack near Houston.

Champion Elevator runs all its analysis software on a Dell Computer Corp. D-530 workstation with dual Intel Corp. Xeon CPUs totaling 3.8 gigabits of random-access memory (RAM) running at 1.5 gigahertz. Disk drive capacity is 39 gigabytes. Operating systems are Microsoft Corp. Windows 2000 and NT server.

What sets Champion Elevators apart from competitors is its ability to custom engineer unique products. From a manufacturing and operations standpoint, it is a custom fabrication shop. There are no assembly lines. Each product is made to rigid specifications, as determined by the client and its own engineers. The company employs 150.

Champion sees the high-rise construction part of its business as “commercial.” Embracing refurbishing and demolition, these installations are almost always temporary. In most cases these systems are rented. Champion has one of the industry’s largest such “fleets” of equipment.

Many of the safety challenges of high-rise construction also apply when inside buildings and underground but the business environment is substantially different. Champion categorizes these more or less permanent installations as “industrial.”

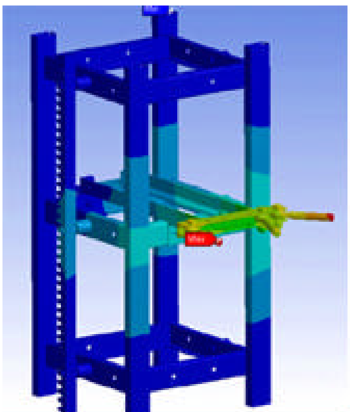
Codes and Design Refinement

In addition to ASCE, the list of regulatory codes to which elevators must conform is long:

- American Institute of Steel Construction (AISC) for steel fabrication.
- American Gear Manufacturers Association (AGMA) for rack and pinion drives.
- American Welding Society (AWS) for anything that is welded.
- American Society for Testing and Materials (ASTM) for anything that is galvanized.
- The U.S. National Electrical Code and its international counterpart in Europe.
- American National Standards Institute (ANSI) for permanent construction and temporary elevators.
- Uniform Building Code (UBC) for earthquakes

There are also countless local building codes, Champion has to verify with the city and county where the system will be installed.

There is also weather with which to contend. “Along the coasts we have to meet hurricane standards,” Oliver pointed out. “We have to determine and verify that our installation will withstand sustained loading, parallel and perpendicular, of winds of 125 miles per hour (MPH) and 150 MPH on Guam. Wind-load details are generated in the beam modeler with ASCE codes.



ANSYS DesignSpace screen image of the wall ties that link elevator car track to existing walls. This kind of analysis is performed on virtually every job done with Champion's products.

Added Bob Meiresonne, Engineering Manager: “We analyze every job. Our products may be standardized but the applications always vary. Both code conformance and margins of safety have to be verified,” he added. “Customers doing due diligence often ask for the standardized DesignSpace reports that are generated automatically.

“On commercial jobs we usually just do worst-case analyses and provide the customer with specific loading criteria for the building ties,” Meiresonne added. “Some of these ties are standard but some are specially engineered to the condition of the building’s exterior, its age and what is being done to it.” For offshore oil rigs and ships, Champion submits its analyses to the American Bureau of Shipping (ABS) or Det Norske Veritas (DNV). Without an okay from one of these industry “classification societies,” insurance coverage will be denied.

“The truly interesting thing with DesignSpace is that you can really refine a design,” said Meiresonne. “In a particular installation, you can determine where the problems are most likely to be. It will show you where you need to strengthen the design versus just adding steel, which adds weight,” he pointed out. “The beam modelers cannot do that.”

Analyzing a multitude of points of stress became vital when the company began re-engineering its products in 2000 to reduce weight and cost and simplify manufacturing. “DesignSpace shows you that the stresses are not always where you might think they would be,” he added, pointing out that this is why DesignSpace requires specific numerical values and not just standard data from the codes. “In other cases it exposes bad assumptions and keeps you from just throwing metal at a problem. Around here weight is very important.”

Oliver elaborated. “Our analysis jobs always start with a mathematical calculation package,” he said. “We use the math package to derive the forces for entering into DesignSpace. We do the analysis with the actual numbers taken from the codes. The math package is used because the codes just give us results, specific values that must be met. We have to work backwards to get the relevant numbers for each part of the code.”

The beam modeler lets him create simple models quickly and run them past a solution engine, which compares the design with the relevant codes. The result is a yea or a nay. “This is sufficient for building codes, windstorm resistance, ABS and DNV,” Oliver noted. “The beam modeler analyzes our tower design and forces acting on buildings and structures,” he continued. “This is non-meshed modeling, but it goes quite far beyond sticks and simple solid elements, straight line forces and beams.”

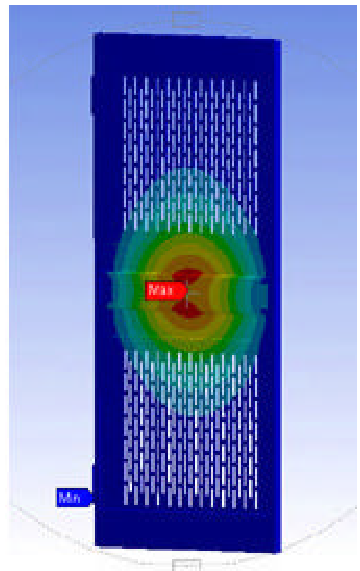
Oliver also likes the DesignSpace capability of displaying all materials used, the specifications of the steel, and dimensions of all the components. “We like the report generator and we use it a lot for due diligence,” he noted. “First, we get a good solid model file, then we import it right into DesignSpace,” he added. “We usually get dimensions and specifications that way.”

Modeling is done in Champion’s software for computer-aided design [EDS/Unigraphics Corp.’s Solid Edge] but, again, without meshing. All of Champion’s DesignSpace analyses are for static loading, not dynamic, and all are linear, not nonlinear.

Design Specifics

Champion’s design efforts focus on cutting weight and cost from its elevator cars. The goal was two-fold: to increase payloads and therefore customers’ profits while cutting its own fabricating and shipping costs. Once the design is done, Meiresonne noted, the basic tower and elevator car structures do not change.

In addition, Champion redesigned its landing gates, the doors at the tops and bottoms of its elevator towers; the wall ties that attach the elevator towers to the customer’s structure, and the elevator cars’ fabricated steel feet.



ANSYS DesignSpace screen image of the redesigned elevator car landing gate. Redesign increased strength while saving weight and simplifying manufacturing and meeting building-site. safety codes

The major re-engineering effort went into the structure of the elevator car to reduce its weight, simplify its manufacturing processes and cut costs. “But we ran into a few little problems,” Meiresonne said, “which we worked out with DesignSpace.”

One of the biggest of the challenges was not inside the car but underneath it, the buffers that ensure a safe, soft landing regardless of what might go wrong. For more than 20 years Champion had relied on four coil springs formed from heavy steel rod, 3/4 inch or more in diameter. Over the years, the springs’ prices kept going up. The car redesign replaced the four springs with two urethane shock absorber systems. They weigh less, cost less, and simplified manufacturing since only two buffer attachment points were needed rather than four spring pods.

“However, having just two contact points beneath the car rather than four doubled the stresses on the contact points,” Meiresonne pointed out. “And for various reasons, the new contact points were not beneath the cars’ centers of gravity. This created eccentric loadings. We used the ‘microanalysis’ capabilities of DesignSpace to redesign the car bottom with tapered trusses.”

“The maximum buffer loads which are experienced under conditions defined by the Elevator Safety Code were up to the yield points of the steel structurals that make up the underside of the elevator car floor,” Oliver recalled. The two buffers were offset toward the tower. This generated a cantilevered load that Champion did not have before. “Some redesigns of the trusses were required,” he said, “but in one case we just added a simple fish plate as a stiffener.”

A new landing gate door was designed not by engineers armed with computers and finite-element software but by a long-time employee who did his calculations by hand. Billy McCoy, General Superintendent, created the design on his own initiative and showed it to Founder and President Walter Manning. Manning liked what he saw and the new design soon landed on Oliver’s desk for verification.

He tested it for load deformation. The applicable codes required the door panel to withstand a load of 1,125 pounds, deforming no more than 0.75 inch. DesignSpace and physical tests showed that Champion's door panel would deform no more than 0.3 inch, well within the standard's allowable limits. Oliver also analyzed the new hinge mountings and frame supports.

A mundane steel fabrication is, in Oliver's and Meiresonne's view, the best illustration of DesignSpace use. It is the car "feet," the forklift lifting point and test stand mounting, four to a car. "The redesign made them made smaller and stronger," Oliver said. "In fact, we got a much better safety factor so lighter steel could be used. We used DesignSpace to make sure the margins of safety were protected.

"We saved 45 pounds of steel per elevator car," he added. While this might seem trivial, those 45 pounds no longer have to be hauled up and down the side of every Champion tower countless times.

Benefits

"What we like about DesignSpace is that it works the way a design engineer works with materials, geometry and specifications," Oliver said. "We use stresses from DesignSpace for safety factors. We use the report generator to print out all the steps taken and results generated whenever customers ask for it. We put in lots of comments into the reports, in a format not unlike PowerPoint, as to where we get the values we used in the modeling and calculations."

Champion's analysis needs are simplified by the fact that it buys its drives, braking systems and rack and driving pinion components. Suppliers are responsible for those analyses, noted Oliver, adding, "we specify that they design in an eight-to-one safety margin for the racks and pinions." Once assured these specifications have been met, Champion only has to analyze and verify the drive systems' alignments and mountings.

The only concern with tower-mounted racks, he and Meiresonne observed, is precise alignment of tower components for smooth rides up and down. This is as much an on-site installation concern as one of fabricating in Houston. A jerky, bumpy or clattery ride will be rightly perceived as a poor installation or a bad design in the first place.

Having tightened up its manufacturing and shipping costs, privately held Champion is thriving. In contrast to most U.S. manufacturers, it is shipping systems to the low-manufacturing-cost countries on the Pacific Rim. An office in South Korea was opened in early in 2002.

Employment is 150, up from about 35 ten years ago. In the shorter term, construction— Champion's core market—is to some extent recession resistant. Builders traditionally take advantage of low interest rates to build in advance of the economy's inevitable recovery and upturn. Clearly, thanks to product redesign with DesignSpace, and to on-going analysis, Champion Elevators is ready.

Champion's product line also includes temporary use elevators, four-post systems, non-counter weighted elevators, transport platforms, mast climbing work platforms and ladder lifts. Elevator car capacity ranges from 600 to 9,000 pounds.

Champion Elevators feature rack-and pinion drives for safe, reliable operation. The pinion gear is coupled directly to the electric motor's drive shaft and this complete assembly is mounted on top of the elevator car. The pinion mates with the tower rack allowing the car to be driven up or down.

A combination of control systems including a multiple-disc electric brake is used to stop car travel. Champion Elevators safety systems set the brake automatically in the event of a power failure and also allows for manual lowering of the car by its

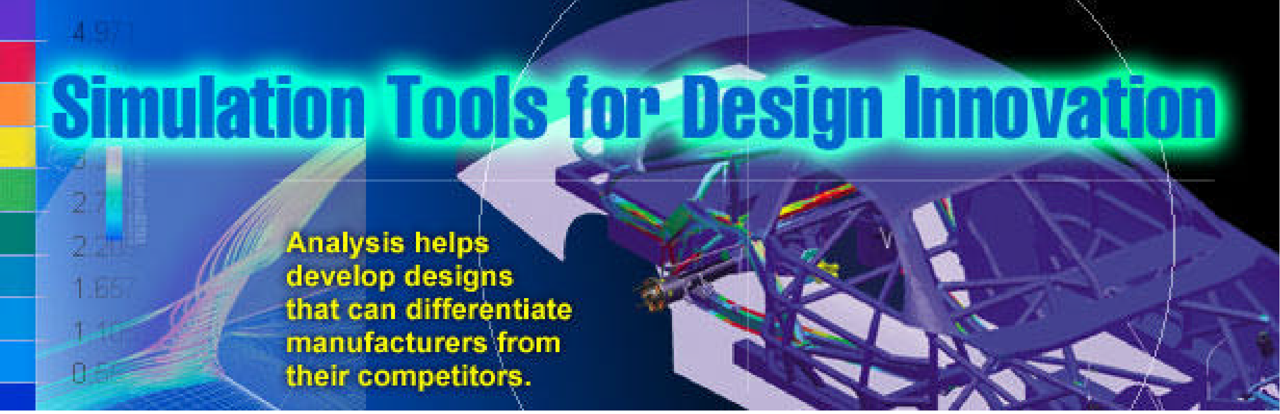
occupants if power is not promptly restored.

The heart of its quality control system, and its safety, is a state-of-the-art variable frequency drive available on all its elevator models. The variable frequency drive system maintains control of the motor at all times, provides the maximum amount of motor-developed torque through all ranges of operation, eliminates the need for a reversing starter and flywheel, and allows for soft, smooth starts and stops. The variable frequency drive also has adjustable starting/stopping speeds and, when used with encoder systems, achieves optimum landing accuracy. Cost savings are realized from very low brake disc wear and reduced brake maintenance.

The Champion exterior-mounted approach eliminates the need for wire rope sheaves, space-wasting machine rooms and dedicated hoist-ways inside the customer's structure.

Print this article

Email this article to a friend



By Michael Wheeler
Vice President and General Manager, Mechanical Business Unit
ANSYS Inc.

In today's turbulent economy, manufacturers are doing all they can to maintain their competitive edge by developing innovative designs. In many cases, companies use design innovation to improve on existing products. Other times they create a whole new class of products and dominate a market segment as competitors scramble to catch up.

The focus today is on innovation: products that clearly differentiate themselves from others while also being affordable, reliable, and fast to market. To sustain sales growth and market position, successful manufacturers must have strategies for developing products to meet customer needs innovatively without driving up costs, sacrificing quality, or delaying product delivery.

Where do these innovative ideas for successful customer-focused products originate? Most manufacturers consistently launch innovative winning products through a combination of people, process, and technology to translate a knowledge of customer requirements into viable products.

Manufacturers have a wide range of such simulation tools to deploy throughout the product development process. Some of the more powerful of these solutions include first-pass tools for performing analysis early in the design cycle, advanced optimization technology for refining product designs, and virtual prototyping methods for evaluating how products will perform in actual operating conditions.

Companies are finding that such tools can be used most effectively to facilitate design innovation if they are blended seamlessly within the product development cycle rather than used alongside the process. Simulation performed as an integral part of this process - rather than separately off to the side - continuously verifies the design and guides the configuration of the product.

First-Pass Tools

First-pass analysis tools facilitate product innovation by enabling designers and engineers with a limited understanding of analysis technology to perform rough simulation early in the development cycle to evaluate concepts, conduct what-if studies, and compare alternative ideas.

Such tools overcome the limitations of traditional product development processes where engineers hand over design models and drawings to a central FEA group that must study and interpret the design, create simulation models from scratch, run the analysis, interpret results, and manually generate reports on the problem. The cumbersome process of exchanging information, building separate models, and manually recreating data often is slow and error-prone. Days or even weeks may go by before analysis results are available to designers, so simulation is often used mostly as a troubleshooting tool to hurriedly fix problems late in product development.

By performing their own analysis with first-pass tools, designers and engineers have more time to devote to creative aspects of the product design, and they have greater freedom to explore innovative ideas that otherwise would be bypassed in favor of more conservative configurations. In this way, first-pass tools are valuable in helping direct design in the early stages of product development when ideas are taking shape and innovative concepts can be investigated. Later in the cycle, specialized optimization routines can be used to refine designs, and more advanced simulations can be performed to study product behavior in greater detail.

One of the original and more advanced first-pass tools for performing finite-element analysis is the DesignSpace software from ANSYS Inc. Wizards guide users step by step in performing routine tasks for structural, thermal, dynamic, weight optimization, performance optimization, vibration mode, and safety factors. Design and analysis are tightly integrated through powerful bi-directional associativity that allows users to make design changes to their CAD model without having reapply any of the loads and or supports in DesignSpace, and conversely to automatically update CAD geometry from DesignSpace based on the simulation.

At tier one automotive supplier Eaton Corp., DesignSpace was used in a pilot program focused on designers performing first-pass analyses early in development. At Eaton's Design Innovation Center reports, the program demonstrated a 30% to 50% time reduction in the number of design iterations in evaluating products such as a clutch, for example. By providing rough "back of the envelope" analyses, filtering out poor features before they become imbedded, and performing what-if studies, first-order tools such as DesignSpace keep the design headed in the right direction and serve as effective guides during the process. They help engineers determine which ideas are most promising and provide insight into why some are good or not. In this sense, the first-order simulation tool can be a valuable decision-support tool in helping guide the design process of developing innovative products in a timely and cost-effective manner.

Optimization Technology

A major challenge in developing innovative designs often is the number and complexity of competing engineering requirements. Automotive components must be lightweight for the highest possible fuel economy yet strong enough for maximum crashworthiness, for example. And engine assemblies must be compact while maintaining adequate airflow for proper cooling. Many of today's products involve a dozen or more such competing requirements. All are important, and neglecting just one can result in a missed opportunity in the market.

However fast single-solution simulation package operate, however, the tools are generally intended to handle only a limited number of variables simultaneously. Users are thus faced with the tedious and time-consuming task of painstakingly running multiple simulations in attempting to iteratively zero in on an often-elusive good solution satisfying most of the requirements. More often than not, engineers develop a design based on only one of the most critical variables and neglect the rest, hoping any conflicts can be corrected later in the cycle. The result is usually not an overall optimal or innovative design but rather one that just works and barely performance and market requirements.

A variety of technologies are coming together in providing a new class of simulation tool that automatically optimizes designs based on boundary conditions and ranges of variables entered by the user. Design of experiments (DOE) technology performs numerous iterative simulations using various sampling and statistical methods, including probabilistic design and Monte Carlo simulation. Instead of performing multiple simulations, another approach called variational technology (VT) uses series expansion to make all necessary calculations much more efficiently using a single finite-element solution. Depending on the problem, VT can arrive at solutions ten to several thousand times faster than conventional DOE approaches.

Users can make adjustments to any variable and see how these changes effect the optimized design.

Some of the more advanced design optimization software combine these technologies with CAE simulation methods and parametric CAD into an integrated solution. Such tools define the optimal dimensions of a part so that stress or weight is minimized, for example, or that a resonant frequency is below a certain level. The result can be a numerical listing of values for a recommended design, simulation response plots showing the trends and influences of each set of variables, or an actual solid model of the optimal design as determined by the software. Also, users can make adjustments to any variable and see how these changes effect the optimized design. By clearly showing the relationship of multiple parameters and their effect on performance, design optimization guides the process of arriving at a configuration that might not otherwise have been considered with pure point-solution simulation.

In one of the first commercially available advanced solutions of this type, DesignXplorer from ANSYS Inc. has a slider bar for each key variable is provided for users to dynamically interact with the model, changing parameters and seeing how this affects the overall design. Feedback is immediate, so engineers can run through multiple ‘what-if’ scenarios that would otherwise be too time consuming to perform with conventional tools. Moreover, because the underlying mathematics of the solution does not limit the number of variables to be considered, factors such as manufacturability and other issues can be taken into account which otherwise would wait until after the design was completed.

In a goal-driven approach using this software, users can study, quantify, and graph various performance simulation responses as a function of design parameters for parts as well as assemblies. Bi-directional associativity with CAD packages allow designs generated through the system to be immediately translated into solid models, with the speed of performance simulation iterations matching the rapid pace of parametric CAD iterations. This speed saves time in arriving at optimal designs. Furthermore, the dynamic interactivity of the process provides users with greater insight into the problem and serves as a catalyst for exploring new ideas.

Virtual Prototyping

Virtual prototyping simulates an entire system or subsystem in its operating environments to study and refine real-world product performance. These often are multiphysics analyses for studying the interaction of all factors encountered throughout the overall product during operation, including real-world factors that radically influence performance such as temperature, fluid flow, vibration, and fatigue, for example. Such simulation may determine the deformation the body and wings as an aircraft lands on a runway, for example.

This approach overcomes the historic “build-test-redesign” problems by evaluating designs through computer simulation and analysis, earlier in the product development process and reducing reliance on validation testing late in the cycle. Often performance problems are encountered this late in the product development cycle necessitate repetitive redesign cycles until satisfactory performance is achieved, with several testing iterations usually required. This adds considerable time and cost to the development cycle, with automobile mock-ups costing $300,000 to $500,000 each and requiring months to build, for example.

Studies have shown that the cost of change increases exponentially with each stage of development, thus making changes costly during detailed design, very expensive during prototype testing, and tremendously high during production. Moreover, correcting errors after the product is sold can be prohibitive in terms of recall and warranty costs, sometimes causing economically catastrophic consequences for manufacturers. Also, designs are often far less than optimal, with quick-fix changes to meet scheduling demands solving isolated problems by usually detracting from the overall design. Components may be grossly overdesigned with needless weight and bulk, for example, to strengthen failed assemblies. When the product is finally launched, the window of market opportunity may have closed, or performance may not satisfy customer demands and expectations.

By using simulation as part of the design process in the early stages of product development when concepts are just starting to take shape, engineers avoid such difficulties later in the cycle by exploring various product configurations, evaluating different part geometries and materials, and examining all the many tradeoffs inherent to product development.

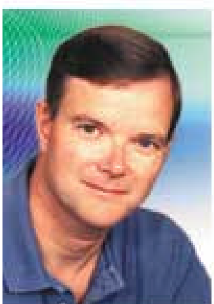
The aim in virtual prototyping is not to entirely eliminate physical testing but rather to use a simulation-driven product development approach to guide the design and reduce the dependency on physical testing for troubleshooting problems late in development. This approach leads to fewer, but better, hardware prototypes that serve to verify a refined design at greater levels of sophistication. This can result in significant time reductions, cost savings, quality improvement, and product design innovation.

By shortening the cycle needed for physical testing near the end of design, virtual prototyping gives engineers added time earlier in development to explore and investigate innovative concepts. Moreover, identifying and correcting problems through virtual prototyping before designs are committed to hardware ensure these design innovations are carried through in the final product configuration. Otherwise, companies would tend to stick with familiar approaches rather than risk encountering unforeseen problems with new and untried product configurations and manufacturing processes.

A Closer Look at Bolted Joints

Using FEA to Gain Greater Insight Into What We Thought We Understood

By John Crawford
Consulting Analyst


John Crawford

One of the neat things about doing finite element analysis is that every so often you gain an unexpected insight into how things work. I'm not talking about finite element things; I'm talking about ***real*** things. Some time ago I was given the task of analyzing a bolt and determining the loads that it was subjected to. At first glance, this doesn't seem to be an especially difficult job. Engineers have been analyzing bolted joints for a long time, and you would think that it would be pretty well understood by now. It's the sort of thing that is presented in undergraduate machine design courses, yet bolted joints are more complex than the rudimentary coverage given them in school might indicate.

I was to learn a bit more about how they work by writing a parametrically driven macro that analyzes a bolted joint and compares its results to those obtained using traditional means. Bolted joints appear to be pretty simple, but as you begin to think about them and start to understand how they work, it becomes apparent how complicated they really are and how difficult it is to precisely quantify their behavior. It's the sort of thing that finite element analysis is made for: solving a messy real world problem for which a precise classical solution is not readily available.

The Classical Approach

My undergraduate machine design class used what is one of the most respected and commonly used textbooks in mechanical engineering education, Joseph Shigley's ***Mechanical Engineering Design***. It contains a wealth of information on a wide variety of mechanical design subjects and is a truly fine book. My copy is dog-eared and beaten from being referred to on a regular basis for more than 20 years. Shigley's presentation of bolted joints is comprehensive and concise. He says that the tension load on a bolt is a function of its preload, the elasticity of the members being clamped, and the pull load on the flange. If one were to plot the bolt load and the flange clamping load as a function of the pull load applied to the flange, you would get two curves like those shown in Figure 1.

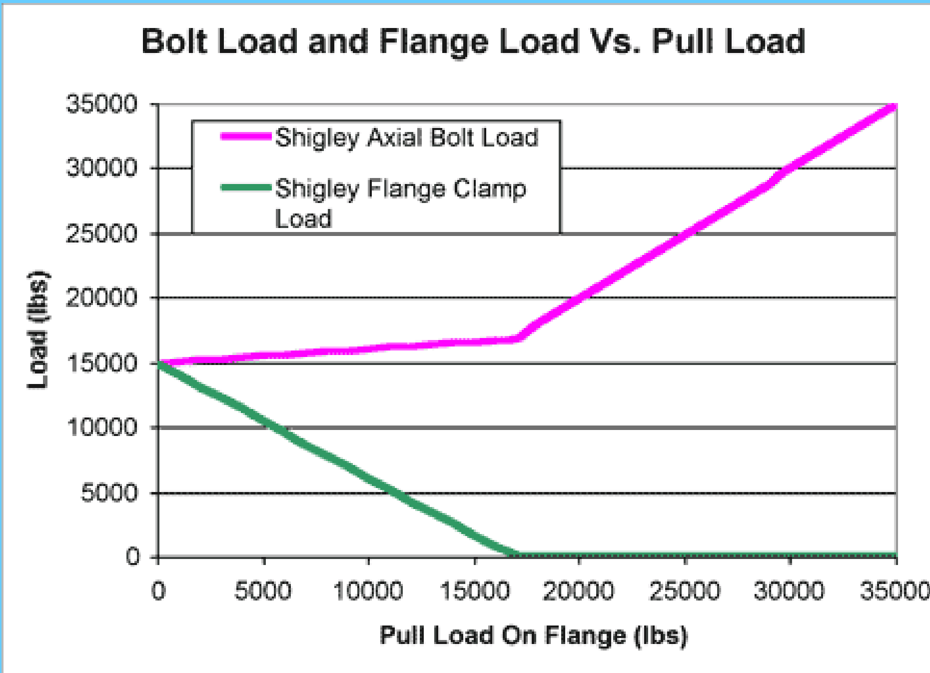


Figure 1: Axial bolt load and flange load as a function of flange pull load, calculated using Shigley's equations.

As the pull load applied to the flange increases, the load on the bolt increases in a bilinear manner. It has a gentle slope up to the point where the flanges separate, and a steeper slope thereafter. As the flange load increases and decreases the load on the bolt varies up and down. The stresses that result from this mean and alternating loading are used to calculate the fatigue life of the bolt. This sounds simple and straightforward, but sometimes the simple approach isn't as accurate as we have been led to believe.

The Finite Element Approach

The first step in any analysis is to understand the problem and determine the approach that will provide the type of answer you are looking for. Depending on the type of problem it might be sufficient to use a hand calculation to obtain a reasonably accurate answer, while others might require a more detailed finite element analysis. If it takes me less time to model it in ANSYS than it does to dig up a textbook and solve it using classical means, I usually grab the keyboard and solve the problem with ANSYS. This sort of approach may make traditionalists cringe, but my experience has been that simple equations work for simple problems, while more complicated problems frequently require more capable analysis techniques to analyze them with precision. If you're building battleships or bridges, classical techniques may be more than sufficient for your needs. If you're building airplanes or race cars, finite element analysis allows for a much more complete understanding of a problem and opens the door to meaningful optimization. It simply comes down to how complicated your problem is, how accurate an answer you want, and how optimal you want your design to be.

After looking at my bolted joint problem for a little while, it struck me that Shigley's approach didn't look quite right. I couldn't put my finger on it, but it seemed to me that the picture painted by Shigley wasn't complete. It was too simple and did not take into account the true flexibility of the flange or the bolt head. I decided to write a macro that would generate a finite element model of a bolted joint and compare its results to those obtained using Shigley's equations. As with Shigley's approach, I ignored the stress concentrations around the threads and also assumed that the bolt head is round. These assumptions allow the problem to be modeled using axisymmetric elements rather than 3-D solid elements. Contact elements with friction were placed between the bolt and the flange, as well as between the flange and the gasket.

A critically important part in analyzing anything is to fully understand the path it follows as it progresses from a zero stress state to the condition it is in when it operates. In the case of a bolted joint, we assume that the bolt is initially in a stress-free state and during installation it is stretched by turning a nut down its threads until a certain tension load is achieved. Usually the tension load is 75-90 percent of yield, depending on whether the bolt will be discarded after one use or will be used over again. This problem required a number of load steps that mimic the way in which the load is applied in real life, which are listed in Table 1 below.

Load Step 1	Apply an initial displacement to the bolt that is close to what we think will be needed to get the desired bolt preload.
Load Step 2	Replace the bolt displacement with the desired bolt preload force.
Load Step 3	Obtain the amount of bolt stretch that results from this preload force and apply it as a fixed displacement to the bolt. Remove the preload force that was applied in Load Step 2.
Load Steps 4-End	Increase the flange load in a series of load steps until the desired final flange load is applied.

Table 1: Load steps used in the bolt analysis.

The macro bolt.mac can be found at www.ansys.com/solutions. It generates and meshes the geometry and applies initial boundary conditions as shown in Figure 2, solves a series of load steps, and postprocesses the results of each load step. It also writes two files, ***boltresults.txt*** that is a summary of the analysis, and ***bolt4excel.txt*** that is suitable for reading into Excel for plotting purposes.

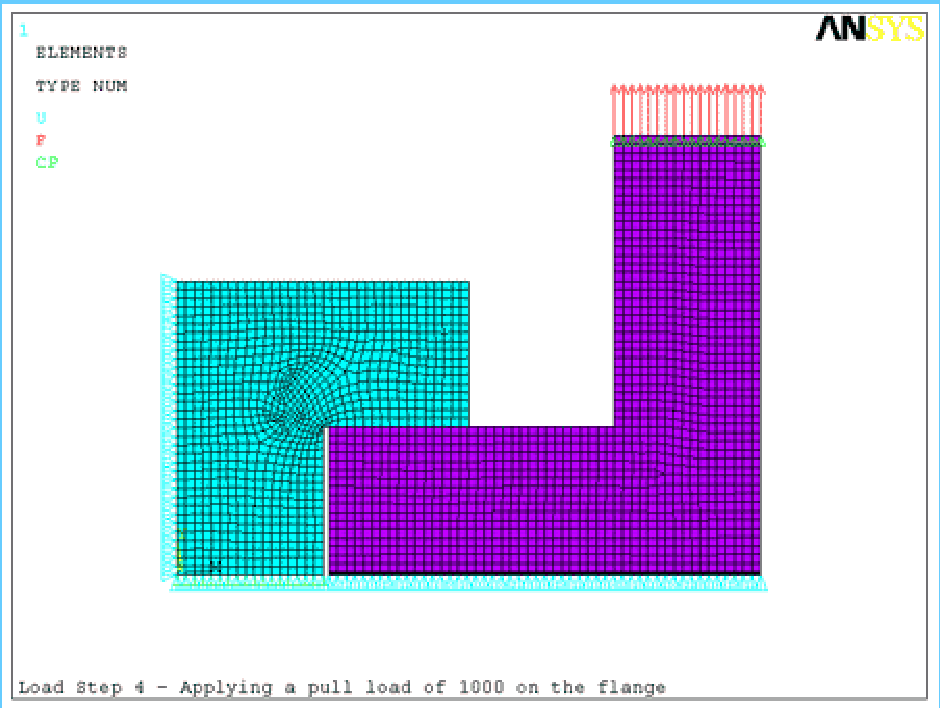


Figure 2: Finite element model generated by bolt.mac.

FEA Results

For the example problem a bolt preload of 15,000 lbs was used and a flange load from zero to 35,000 lbs was applied. The finite element results show that the manner in which the axial bolt load increases as a function of flange pull load is quite different from what was predicted by Shigley. For pull loads up to 10,000 lbs the axial bolt load is much lower, which means that if Shigley's results were used to predict bolt fatigue life the resulting calculation would be conservative and might result in an unnecessary change to a larger bolt. Conversely, for a very high pull load Shigley's results are low.

The fact that the finite element model varies from Shigley's equations is not surprising, but what I had not anticipated was that the bolt load initially can initially decrease as the pull load is applied. I had always assumed that the bolt load increased as the pull load was applied, but the finite element model indicated otherwise. How could this happen?

When we compare Figures 3 and 4 we see that when the bolt is initially loaded that most of the load is carried by the inside of the bolt head. As the pull load is increased, the bolt load moves from the inside of the bolt head towards the outside. When the bolt load is carried by the inside of the bolt head the flexibility of the outer portion of the bolt head plays a lesser role in the overall stiffness of the bolt. As the pull load increases, the flexibility of the bolt head become more significant and results in a load path that decreases in stiffness. The result of this is that the load on the bolt decreases a small amount when the pull load is initially applied. The assumptions made in Shigley's work resulted in an analytical approach that completely overlooks this phenomena.

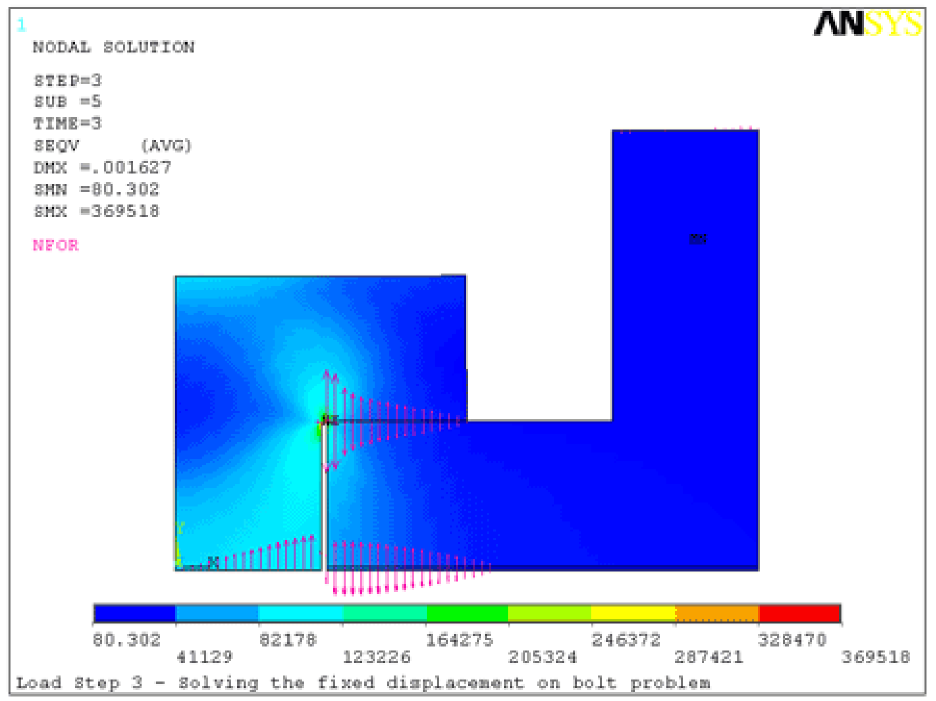


Figure 3: Von Mises stress and reaction loads that result when the bolt is tightened. The high stress is due to the very small fillet radius located at the inside corner of the bolt head. A larger fillet would have resulted in a lower stress. The reaction arrows between the bolt head and the flange show that most of the bolt load is transferred to the flange near the inside of the bolt head.

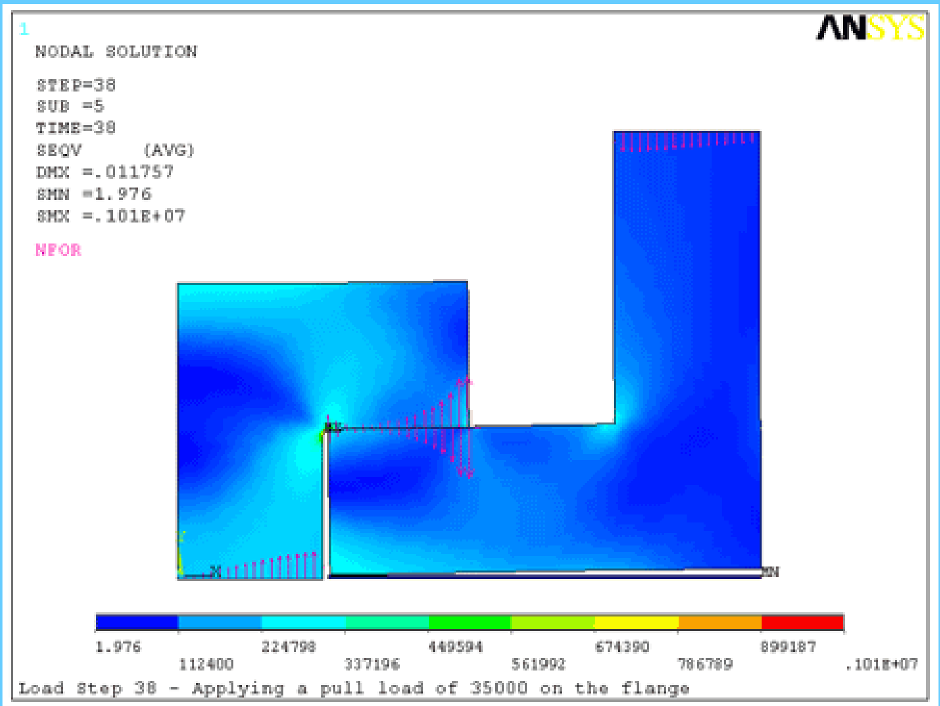


Figure 4: Von Mises stress and reaction loads that result when a large pull load is applied to the top of the flange. The absence of reaction symbols on the bottom

of the flange indicates that the flange has lifted completely off the surface of the gasket. The reaction arrows between the bolt and the flange indicate that the load path has moved to the outside of the bolt head.

The results from every analysis should be reviewed with an intuitive eye. Do the results make sense? Are they what you expected? If not, why are they different? Perhaps there was a mistake in the finite element model, or maybe a boundary condition was overlooked when the problem was initially set up. After checking an analysis and not finding a mistake that would explain why the results were not what we had expected, we have to review our understanding of the problem and see if we are missing something. While I had expected the finite element answer to be different from Shigley's answer, I did not anticipate that bolt load would initially decrease as flange load is increased. Turning on the reaction force arrows revealed how the load path changes as the pull load on the flange increases and helped me understand what was really happening. Figure 5 shows the finite element results plotted with Shigley's results. The data used to create these curves are displayed in Table 2.

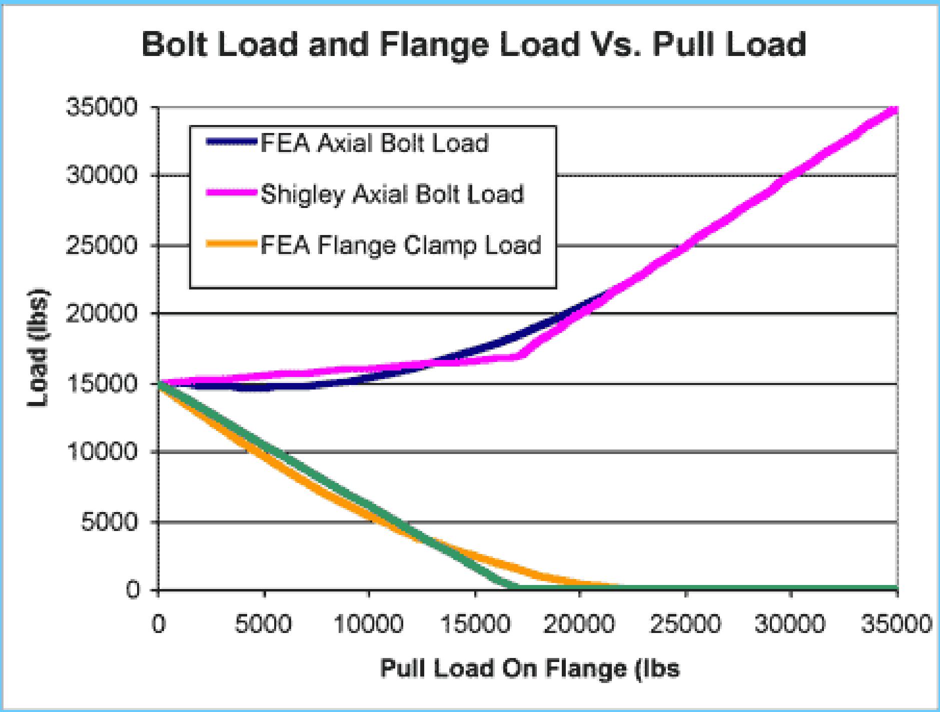


Figure 5: Results from the finite element model are added to the results from Shigley's equations. Notice that the finite element results show a much different load path until the flange has lifted off the gasket. A surprising result of the finite element analysis is the revelation that the bolt load actually decreases a small amount as the pull load is applied, and then increases rapidly.

The example problem illustrates why finite element analysis is such a great learning tool, whether you're a college student or have been a practicing engineer for many years. Sometimes you get results that are not intuitively obvious, and by looking at stress contours, animating deformed shapes, and viewing reaction force arrows you can begin to grasp what is really going on. In the case of bolted joints, one can use the macro bolt.mac to simulate a wide variety of bolt configurations and begin to develop a feel for how bolted joints behave. For example, after I noticed that bolt load initially decreases, I varied the flange thickness and saw that the amount that the bolt load dips below its initial preload decreases as flange thickness increases. Similar insights can be gained by changing the bolt head diameter, thickness and stiffness of the gasket, and so forth.

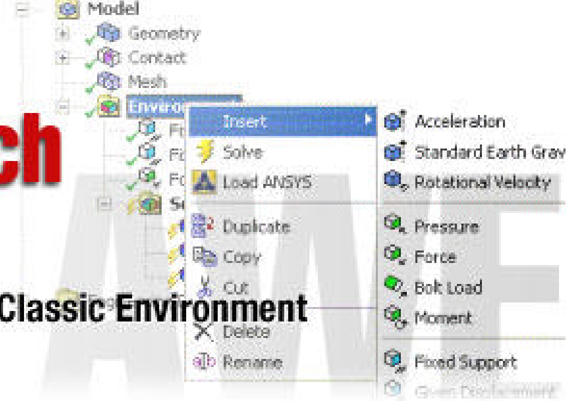
The value gained from an exercise like this is more substantial than just an answer to a specific engineering problem. By viewing the results of a finite element model and then varying the inputs and how they affect the results, one begins to develop a feel for how a particular mechanism operates. As one analyzes more and more problems, you begin to develop a feel from how things work, which is one of the most rewarding aspects of being an analyst.

Pull Load	FEA Bolt Load	Shigley Bolt Load	FEA Flange Load	Shigley Flange Load
1000	14917.37	15111.11	13917.37	14111.1
2000	14832.62	15222.22	12832.62	13222.21
3000	14760.55	15333.33	11760.55	12333.33
4000	14710.85	15444.44	10710.85	11444.44
5000	14703.07	15555.56	9703.069	10555.55
6000	14735.38	15666.67	8735.379	9666.659
7000	14818.46	15777.78	7818.464	8777.77
8000	14959.2	15888.89	6959.201	7888.882
9000	15161.05	16000	6161.051	6999.993
10000	15420.49	16111.11	5420.486	6111.104
11000	15724.35	16222.22	4724.348	5222.215
12000	16070.26	16333.33	4070.256	4333.326
13000	16476.43	16444.44	3476.432	3444.437
14000	16911.28	16555.56	2911.276	2555.548
15000	17393	16666.67	2392.996	1666.659
16000	17943.67	16777.78	1943.666	777.7704
17000	18512.24	17000	1512.24	0
18000	19084.9	18000	1084.903	0
19000	19789.62	19000	789.617	0
20000	20518.53	20000	518.531	0
21000	21256.64	21000	256.6376	0
22000	21999.4	22000	-0.5952	0
23000	22999.9	23000	-0.0964	0

24000	23999.71	24000	-0.2901	0
25000	24999.69	25000	-0.3117	0
26000	25999.75	26000	-0.246	0
27000	26988.89	27000	-11.1098	0
28000	27994.28	28000	-5.7224	0
29000	28988.73	29000	-11.2706	0
30000	29993.55	30000	-6.4458	0
31000	30996.82	31000	-3.1783	0
32000	31999.43	32000	-0.5652	0
33000	32999.61	33000	-0.3918	0
34000	33999.5	34000	-0.4991	0
35000	34999.41	35000	-0.5891	0

Table 2: Results from the finite element model and Shigley's equations.

Working in the ANSYS 7.1 Workbench Environment



Some behavior might differ from the ANSYS Classic Environment

By ANSYS Inc. Technical Support
ANSYS Inc.

Q: What type of behavior in the new ANSYS 7.1 Workbench might not be expected by a user more familiar with ANSYS Classic?

A: Here are ten common ways you'll see ANSYS Workbench behave differently from ANSYS Classic:

1. Element shape checking in solution is off by default. Also, the default shape metrics used to create the mesh in AWE are not the same as those applied in solution. If solution shape checking is activated (by inserting SHPP,ON in the Preprocessing Command Builder) element shape checking should be set to "aggressive" under "advanced" Global mesh control when creating the mesh.

2. The default value of "Save ANSYS db" is No (see the menu in the lower left hand corner when Solution is highlighted in the tree.) This may limit the user's options upon returning to ANSYS Classic. For example, the nodal and element components created from "Named Selections" in Workbench will not be available.

3. In a harmonic analysis only the "scoped" harmonic results are written to the ANSYS Classic results file. As a result, these will be the only results which can be postprocessed in ANSYS Classic, unless additional results are requested via the Preprocessing Command Builder [OUTRES].

4. When a preprocessing command object is added to the Workbench tree outline, several commands are added by default. One of these may be SOLCONTROL,ON. If a number of frequencies have been specified for a harmonic analysis, the SOLCONTROL,ON command has the unintended effect of resetting the number of frequencies back to one and should therefore be removed.

5. The default contact behavior in Workbench is different than the default behavior in ANSYS Classic. The settings are intended to minimize the need for user intervention to obtain a converged solution. You'll note differences from ANSYS Classic in the following Workbench defaults, most of which can be changed in the contact details view:

- Symmetric contact pairs are created at all part interfaces
- The default contact behavior is intended primarily as a way of connecting parts. It is linear and does not change status ("Bonded Always" KEYOPT(12)=5).
- With linear contact, solution control is turned off and the number of equilibrium iterations is set to 1.

- With linear contact, initial penetration, gaps or offsets are excluded (KEYOPT(9)=1.)
- If a model contains only linear contact behavior, the normal contact stiffness factor, FKN, is increased from 1 to 10.
- With nonlinear contact, initial gaps and interference are included using ramping (KEYOPT(9)=2.)

6. When “Load ANSYS” is selected in ANSYS Workbench, the model comes into ANSYS Classic with the "load replace vs add" option set to “add.” This is not the default in ANSYS Classic and is a rarely used option. It results in loads for multiple load steps being cumulative [See FCUM and SFCUM commands at the top of the ds.dat file.]

7. The default solver working directory is “Use System Directory.” This can be changed to “Use Project Directory” to more closely match ANSYS Classic behavior.

8. To obtain contour results in ANSYS Classic which match ANSYS Workbench displays, use the following graphics settings: Nodal Solution, and Full Graphics.

9. Only material properties believed to be relevant are sent to solution by ANSYS Workbench. For example, if the ANSYS Workbench tree outline defines a steady state thermal analysis, which is switched to a transient thermal analysis in the Preprocessing Command Builder, the material properties for density and specific heat will not be sent to the solver. In this case, the Preprocessing Command Builder input should contain the density and specific heat material property definition.

10. It is not possible to change materials in the details pane (they will be grayed out) after a Preprocessing Command Builder branch is defined. This is to avoid redundant specification of materials.

Recognizing the Critical Value of Engineering Software

Under funded and unappreciated, engineering software is the best hope of survival for many companies

By David Weisberg
Chief Industry Strategist
Cyon Research Corp.

Starting in the mid-1990s, overall industrial productivity has increased significantly. Computer and communication-based technology has been credited with powering a substantial portion of this improvement. But, often, these discussions fail to recognize the huge contribution of engineering software technology: the tools and processes related to CAD, analysis, simulation, document management, lifecycle management, and collaboration.

Engineering software technology has had a profound impact on productivity in the automotive industry, for example. In many cases, design cycles have gone from 50 or 60 months to less than 20 months. Not only has the time to bring a new automobile to market decreased, the quality of the typical automotive product has improved dramatically in terms of functionality, styling, manufacturability and reliability. Unfortunately, the technology used to create these improved products, enhance productivity, and increase the profitability of the companies using the technology receives little credit for its accomplishments.

A good example of the tremendous value of engineering software technology is the use of crash simulation programs in the automotive industry. Not only does this software eliminate the need to build numerous costly, physical prototypes, simulation takes less time and permits design engineers to evaluate a proposed vehicle under more conditions than does traditional physical testing. Yet when the subject of automotive safety comes up, rarely do we see this technology discussed in the media.

One indication of this lack of respect is the fact that seldom do we see articles about design, analysis, and product data management technology in the mass media, and only occasionally in the business press. Before it became history, ***Red Herring*** magazine published an annual list of the 100 companies it believed would shape the future. Among the nearly 200 companies in its 2001 and 2002 lists, not a single engineering software company was included. Yet advanced design creation and analysis tools were the key in bringing new products to market, including cellular telephones with built-in cameras, video game boxes, notebook computers, digital cameras, and large commercial aircraft. The list could go on and on.

This lack of appreciation for the value of engineering software in product development — particularly on the part of company executives who control the corporate purse-strings — results in engineering technology being grossly under-funded in relation to its potential impact on top-line revenue growth and corporate profitability. Unfortunately, C-level managers in the ranks of CEO, CFO, CIO, COO often continue to focus only on justifying nearly all forms of engineering technology on cost savings, as has been done for decades. “If it takes 20 designers today for that task, how many fewer can accomplish the same job if we invest in new technology?” So goes the argument in all too many boardrooms.

Taking costs out of systems is crucial when systems are “fat.” Today, however, most operations are already lean. Reducing costs is just part of the overall equation and certainly not the most important part when greater enterprise objectives are considered. Far more significant is the business value of creating innovative, competitive products that customers want to buy, and doing so in a timely manner. These types of products bring money directly to the top line and pump life into companies that otherwise would be struggling. Manufacturers can reduce expenses all they want, but without winning products, no amount of cost reduction matters.

Engineers can’t do a better job designing products just by working harder or longer hours, they need the right tools to leverage their expertise, and that demands investment in the latest software technology.

Engineers can’t do a better job designing products just by working harder or longer hours, they need the right tools to leverage their expertise, and that demands investment in the latest software technology. Making the right investments in engineering software and applying the tools intelligently in the product development process is critical for a manufacturer to define the brand value of its products, capture greater market share, and strengthen its competitive position.

For many companies, engineering software applied intelligently to the product development process represents their best hope — and in many cases their last hope — for continued survival in a brutal economy. No longer is there any tolerance for stragglers in the market or those too frugal to invest in their own future. More forward-looking companies with enough sense to adequately fund the product development initiatives that are their bread and butter will brush them aside.

From a broader perspective beyond corporate profitability, engineering software certainly brings benefits to society through innovative, affordable products that improve and enrich all our lives. In this respect, I truly believe that we are better off today with the engineering software technology tools that are available than we would be without them.

Dave Weisberg is chief industry strategist of Cyon Research Corp., an analyst and consulting firm serving the engineering technology market. The organization hosts the Congress on the Future of Engineering Software (COFES), an annual industry event that brings together executives from manufacturing, construction and plant design companies with high-level vendor representatives and industry analysts to discuss business strategies, barriers to growth, and approaches to implementing engineering software technology. Weisberg also is the founder and former publisher of the Engineering Automation Report, a monthly software industry newsletter that provides insight, analysis, and news for mechanical engineering managers, users, and vendors.

Print this article | Email this article to a friend

CADfix Speeds CAD/FEA Data Translation at Dana Corporation

Automatic defeaturing reduces mesh complexity without sacrificing analysis accuracy

The route from CAD model to FEA analysis and back again is traditionally time-consuming and marked by compromise. There is a fine line between accuracy and solvability as designs are simplified to avoid file corruption but at the risk of losing the accuracy of the FEA results.

While some companies are content to accept this compromise, an increasing number are turning to dedicated data interoperability solutions such as ITI TranscenData's CADfix.

One such company is Dana Corp., one of the world's largest suppliers of components, modules, and complete systems to a wide variety of vehicle manufacturers and their related aftermarkets. Founded in 1904 and based in Toledo, Ohio, the company operates hundreds of technology, manufacturing, and customer service facilities in 30 countries and employs approximately 60,000 people worldwide.

Dana Corporation's Commercial Vehicles Systems Division (CVSD) designs and manufactures heavy truck components in tune with the company's "under-the-chassis and under-the-hood" focus. The division serves some of the world's largest truck, bus and motor home OEMS, designing and testing steer axles, drive axles, suspensions, brakes and other key components.

Interoperability Problems

As Dr. Prasad Managalaramanan, Lead CAE Specialist at CVSD explains, data interoperability problems between CAD and FEA systems were slowing the design to manufacture process considerably. "A CAD designer typically designs a component with a number of objectives in mind, such as manufacturability, packaging, casting drafts and clearances. Naturally, some of these objectives interfere with the needs of a finite element analyst, i.e., a simple model that can be easily meshed."

Traditionally, CAD models were defeatured before translating them into IGES and finally into the FEA system – a lengthy and unsatisfactory process. "The defeaturing of CAD models solved our data translation problems to a certain extent, but cost us heavily in terms of FEA accuracy," explained Dr. Mangalaramanan. "IGES is also not the best method of data transfer and the imported geometry was often corrupted. So even after a successful data import, there was not much we could do with the information."

CVSD's design engineering model mirrors that of many similar organizations. Its CAD modeler is Pro/ENGINEER, with IGES as its neutral file format and Pro/Mechanica and ANSYS for its finite element analysis. Usually an IGES file of the solid model is imported into ANSYS and in some cases ProMesh is used to transfer the finite element mesh from Pro to ANSYS.

"We needed a way to mesh complex CAD components without spending too much time in defeaturing, to retain as much of the original geometry as possible and to improve the accuracy of our FEA analysis," says Dr. Mangalaramanan.

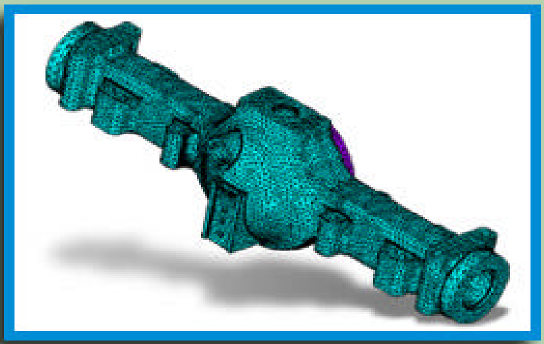
Time-Saving Solution

“CADfix offers us exactly what we were looking for,” says Dr. Mangalaramanan. “The defeaturing functionality removes the complex details that have been included by the designers but are not significant from an engineering point of view. We can then run the FEA with the reassurance that the results will be accurate.”

Dr. Mangalaramanan is delighted with the results: “Geometries that would take us weeks to mesh now take only days, and the package is very simple to use, yet powerful. For the first time I have seen a sales person delivering me more than he could convey in words. I have used almost every single feature of CADfix and it is now an indispensable part of the engineering process at CVSD.”

ANSYS and CADfix

ANSYS has taken this process a step further, by incorporating the strength of CADfix’s automatic healing and translation process, into the ANSYS Geometric Healing Module. This module, which works either with ANSYS own connections or additional CADfix based imports, allows ANSYS users to automate as much of the CAD to CAE integration process as possible, while the interactive CADfix product is still available from ITI TranscenData (and many ANSYS resellers!) when more customized or intensive healing is required.



CADfix provides significant time savings in important complex CAD models into ANSYS for analysis.

Print this article

Email this article to a friend