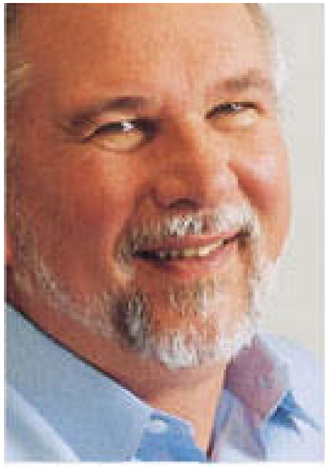
ANSYS

THE PREMIER E-ZINE FOR DESIGN INNOVATION

SOLUTIONS

Welcome to the Online Premiere Issue



We are pleased to send you this first issue of our new online publication; an e-zine focused on simulation-based applications, technology, processes and strategies. Our goal is to provide objective and authoritative information to help readers understand and apply finite element analysis and related technology, not only from an engineering standpoint but also from a business perspective.

The movement from a printed publication to an online e-zine enables us to bring you current information much faster than is otherwise possible.

The Internet and Web-based technology are playing a huge role throughout industry today, as ANSYS and we all know users are applying the tools together with simulation solutions in ways that were unimaginable just a few years ago. So it's appropriate for us to use this same digital communications medium to convey important information to you in a timely manner. The e-zine format also gives us more flexibility in terms of graphics, length, and mundane aspects of publishing a magazine such as deadlines and production schedules.

Under this new digital format, we'll continue to bring you industry news, overview articles, user stories, software profiles, product information, application tips, and other material directed toward professional analysts applying the technology as their main jobs, design engineers using simulation as part of their product development work, and managers overseeing these operations in their capacity of running the operations.

In ***ANSYS Solutions***, we'll be covering a broad spectrum including advanced analysis technology, virtual prototyping, design synthesis, process automation, application integration and CAE collaboration. These various types of solutions all have a technical foundation, of course, and we'll be covering them all from that angle. Beyond these technology stories is the business case for these tools: the benefits they bring to the enterprise in terms of time savings, cost reduction, quality improvement, product performance and design innovation.

These are the business drivers for implementing simulation and the overriding reason companies invest in these types of solutions. Indeed, the business benefits are so far-reaching that simulation-based product development is recognized as a critical competitive advantage at a growing number of forward-thinking companies.

We'll bring you management-oriented articles, columns, and commentaries on these business-drivers and related trends, issues and strategies. In this issue, managers will find the article "Who's Listening to the Customer?" by Howard Crabb of particular interest, discussing how manufacturers are shifting from product-push to market-pull development processes driven by customer expectations and consumer demand. See also Don's Brown's piece on "Emergence of the SuperDesigner" describing management issues related to the converging and overlapping roles and responsibilities of designers, engineers and CAE analysts.

We'd like to give our management readers the opportunity to contribute their stories in future issues. Material can range in scope from 600-word guest commentaries to full-length 1,500-word feature articles. Tell us in your own words how simulation technology is being implemented at your company and what impact it's having on your business and the way it operates. We invite you to share your experiences, ideas, and opinions on topics such as up-front analysis, managing designers and analysts, leveraging simulation throughout the product life cycle, simulation as a competitive necessity, integrating analysis with design, where simulation pays off, analysis across the supply chain, simulation as a business advantage, analysis versus testing, ROI, etc.

Our goal is to provide objective and authoritative information from an engineering and business perspective

We'd appreciate any input you can provide in helping us make ***ANSYS Solutions*** a valuable conduit of information for all levels of the enterprise. Considerable time and research goes into making this an authoritative source of useful information, and we want to be flexible in making sure our editorial content meets your ever-changing needs. We invite your feedback and want you to consider this your digital magazine. Together, we can build and evolve a format that will continue to make a useful contribution to your work and one we hope you will look forward to reading. ■

John Krouse
Editorial Director
jkrouse@compuserve.com

What's New in ANSYS 7.0

The ultimate scalable engineering solution for a multitude of engineering enterprises

By Ray Browell
ray.browell@ansys.com
Product Line Manager
ANSYS Inc.

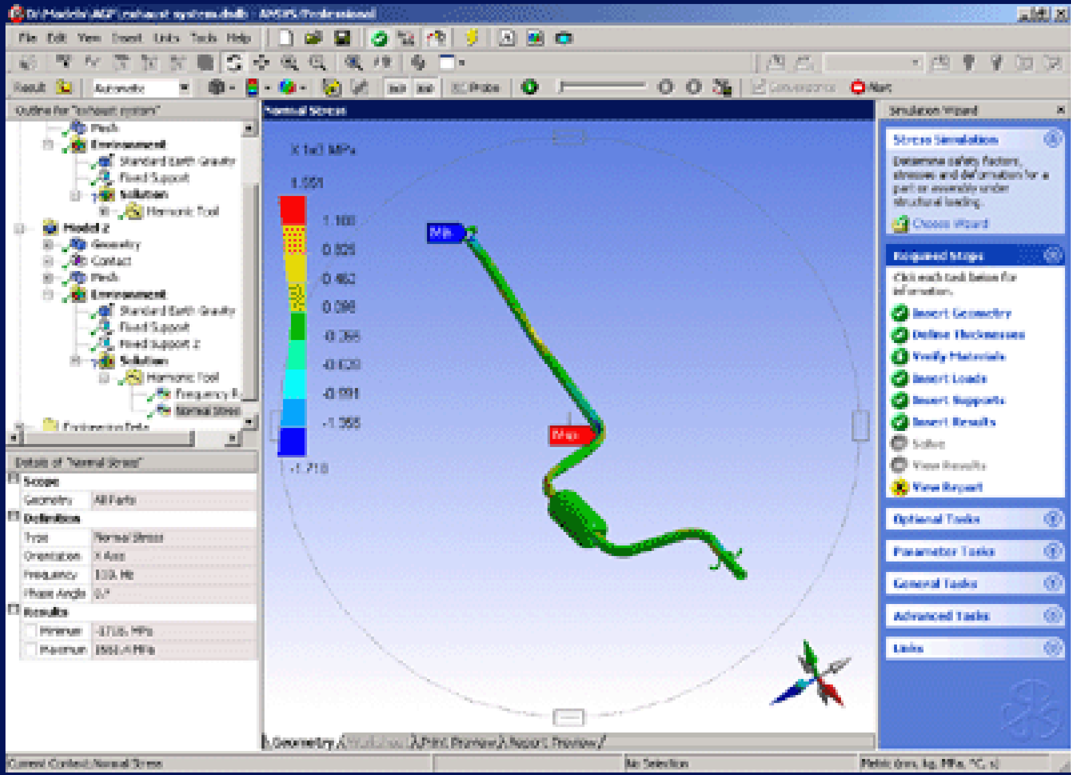
From elemental structural and thermal analysis in the aerospace and automotive industries to intricate fluid flow and electromagnetic investigations in the emerging biomedical and electronics industries, ANSYS is the premier, scalable engineering solution for a multitude of engineering enterprises. Plus, with new technology and capabilities constantly being added, ANSYS continues to keep pace with the increasing simulation demands of companies worldwide.

Adopting an ANSYS solution is more than simply selecting the right tool for the right job. It's about adopting technology designed to completely integrate with existing systems and legacy data to improve the overall efficiency of the product development process. ANSYS' scalable solution means that the same powerful technology can be extended to every tier of the corporate product development structure. From the entry-level CAD operator to the senior analyst, ANSYS efficiently increases productivity among engineering team members and improves product quality throughout the development cycle. This brings better products to market in far less time, resulting in lower overhead, increased revenue, greater market share and higher customer satisfaction.

Not only is ANSYS committed to bringing the most advanced functionality to its scalable CAE solution, it strives to improve accessibility for a growing user base. Throughout the next few years, expect major advancements in the power of ANSYS' features, as well as the intuitive operation of its user interface.

ANSYS 7.0 software offers engineers and scientists unmatched usability, technology and performance. A wide range of new features and enhanced capabilities has been added for simulating single as well as multiphysical phenomena. ANSYS – the catalyst for major changes in the product development process throughout its 30-plus year history. We'll continue to deliver the most sophisticated and innovative engineering applications today and tomorrow.

For more information, visit www.ansys.com



ANSYS FEMXplorer

ANSYS FEMXplorer module computes data vital to the improvement of product design performance. When combined with ANSYS simulation software, this powerful application allows analysts to make intelligent design directives given multiple competing objectives. FEMXplorer's proprietary Series Xpansion technology offers the capacity to study, quantify and graph the structural analysis response to alternative design parameters for linear elastic static analyses.

Finite element analysis results depend upon several input variables, such as:
- Material properties
- Physical parameters such as the thickness of a plate
- Number of ribs or stiffeners

In a traditional finite element analysis, each change of the value of any input variable requires a new FEA. To perform a “what-if" study where several input variables are varied in a certain range, a considerable number of finite element analyses may be required to satisfactorily evaluate the FE results over the space of the input variables.

Series Xpansion technology provides a much more efficient approach by providing a "response surface." A response surface is an explicit approximation function of the Finite-Element results expressed as a function of all selected input variables. Among other approaches known in literature, these approximation functions can be derived from Taylor series expansion approximation.

The accuracy of the Taylor series expansion technique depends on the order of the approximation function. Naturally, the higher the order of the approximation, the more accurate the approximation will be. The Series Xpansion technology implemented in ANSYS automatically determines the necessary order of the approximation based on the requested accuracy of the expected results.

To determine the response surfaces, it is necessary to evaluate higher order derivatives of the results with respect to the selected input variables, where the order of the derivatives corresponds to the order of the approximation function. It is a unique, key feature of the Series Xpansion technology implemented in ANSYS that all necessary derivatives of any order are calculated automatically within one single finite element analysis. Because the derivatives are also calculated, this "extended" finite element analysis may take longer than a regular solve. However, this one "extended" finite element analysis will take considerably shorter time if compared to the many solutions runs that are required for the mentioned "what-if" study. Depending on the analysis problem, typical acceleration factors may be in the order of 10 or even up to several thousand.

ANSYS provides two separately licensed Series Xpansion modules for ANSYS:

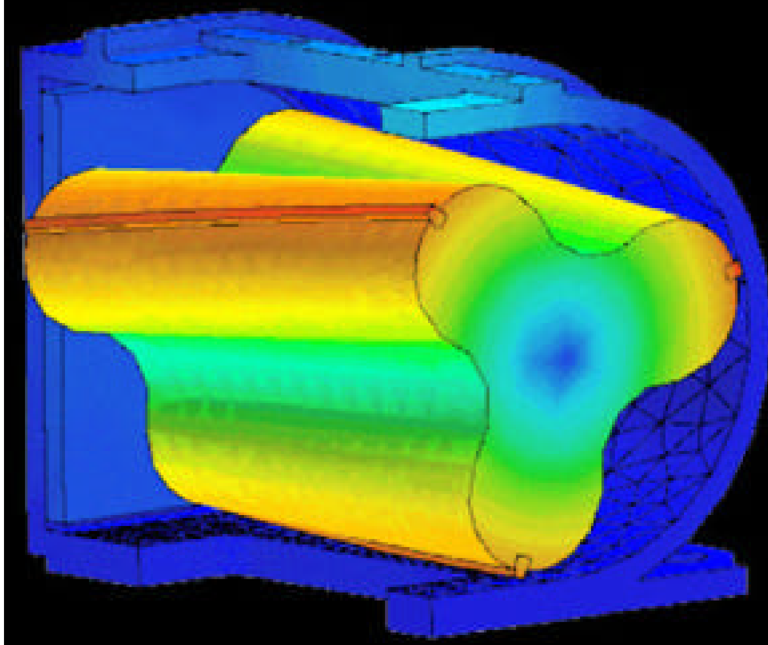
ANSYS Series Xpansion Frequency Sweep Module

ANSYS FEMXplorer Module

ANSYS Series Xpansion Frequency Sweep Module

Provides response curves of electromagnetic applications as a function of the excitation frequency.

Capable of performing a frequency sweep for high-frequency electromagnetics, this module is automatically included with ANSYS EMAX 7.0. This is also an add-on product to ANSYS Multiphysics only. It does not currently support any other type of frequency sweep except for high-frequency electromagnetics. Structural Frequency Sweeps will be supported in future releases.



ANSYS FEMXplorer Module

Provides a response surface of FE results for linear elastic static structural analysis.

ANSYS FEMXplorer provides a response surface of FE results for linear elastic static structural analysis. This version of the Series Xpansion technology works within ANSYS to select material properties, real constant values such as thickness, and a collection of elements to be active or inactive and then solves using Series Xpansion technology coupled with the ANSYS solver. This is an add-on product to any of the ANSYS products with structural capabilities.

- Based upon the series Xpansion (SX) technology acquired through the CADOE S.A. Acquisition
- SX technology provides a "response surface" by Taylor series expansion
- The SX technology implemented in ANSYS automatically determines the necessary order of the approximation based on the requested accuracy of the expected results
- The response surfaces are calculated within a single "extended" finite-element analysis
  - Solution time is typically 2x to 5x of a single solution
  - The “extended" finite element analysis is considerably shorter than typical DOE methods
  - Depending on the analysis problem, typical acceleration factors may be in the order of 10 or up to several thousands

Capabilities

- Supports analyses that are structural static analyses with linear elastic and isotropic materials
- Supports all of the 18x Series of Elements
- Works with the sparse solver
- Parameters that can be varied include:
  - Discrete (i.e. number of ribs) elements supported include:
    - BEAM188, BEAM189, COMBIN14
  - Material properties. Elements supported include:
    - LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187
  - Shell thickness through either real constants or section definitions. Element supported include:
    - SHELL181
- FEMXplorer is an add-on module to Classic ANSYS Environment

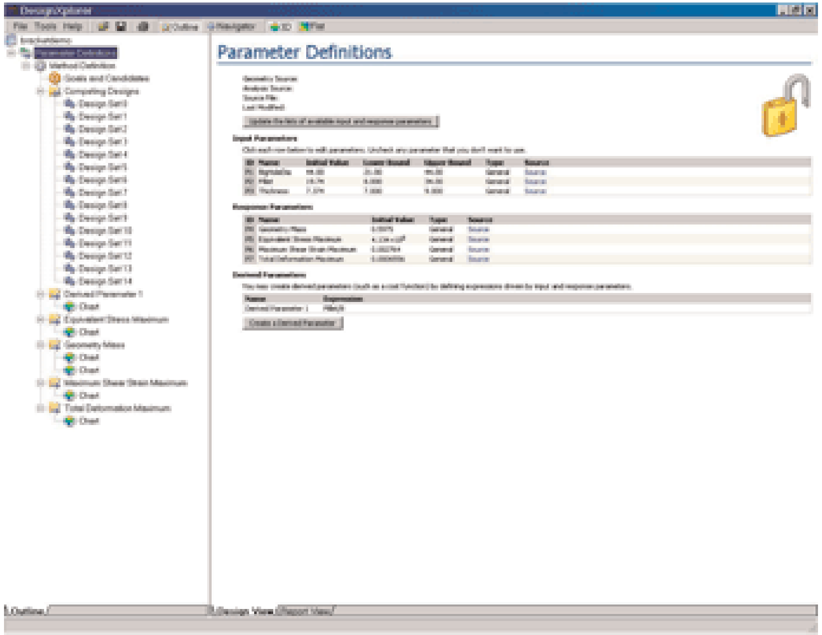
At the 7.0 Release, the FEMXplorer has a few limitations. Items not supported include:

- Temperature-dependent material properties
- Thermal loads
- Constraint equations (couples are supported)
- Elements using the mixed U/P formulation
- Limitation of only one FEMXplorer variable per individual element (e.g. you can not vary both modulus of elasticity and thickness on the same shell element)

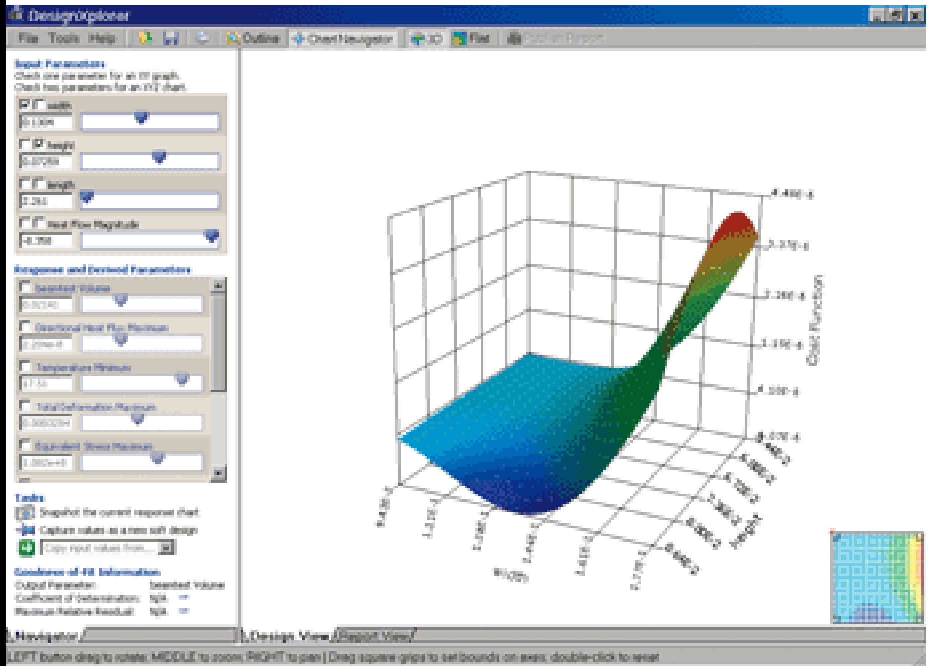
Discrete Parameters: DesignXplorer (DX) now supports discrete parameters. These are parameters that are not continuous in nature, quantities such as the number of holes in a part, or which type of material the part is made from. At 7.0, DX supports three different kinds of discrete parameters, they are:

.

Integer Discrete Parameters: Integer discrete parameters are parameters which can only have integer values and are meaningless for any value other than integer values. A good example of this is the number of holes in a part. Obviously, the number of holes is an integer values, but for this category, it is also important that non-integer values are meaningless. For instance having 2.25 holes does not make any physical sense.



Define parameters from multiple sources (i.e. CAD system, DesignSpace or within DesignXplorer itself) using the design parameter definition table.

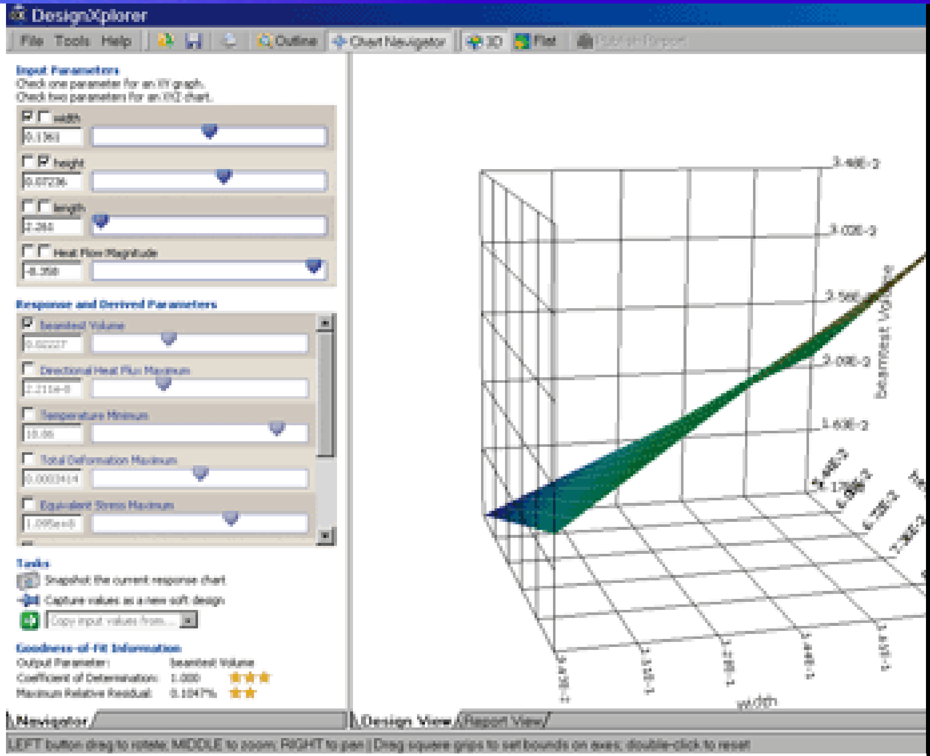


Easily modify design parameters using either the slide bars or manual entry options of DesignXplorer's parameter manager table.

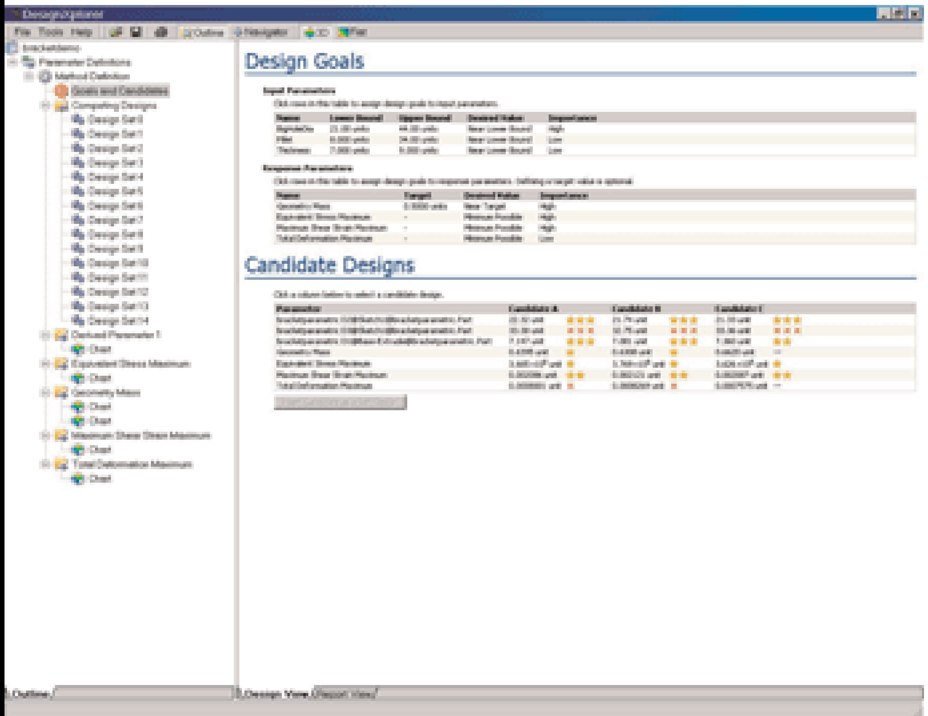
Scenario Discrete Parameters: Scenario discrete parameters are parameters that define an entirely new set of conditions that are completely independent from one another. Selection of the material is a scenario discrete parameter. Steel and aluminum are completely different materials and, therefore, each material has its own scenario with its associated set of geometry, boundary condition and loading parameters.

Continuous Parameters with Usability Constraints: Continuous parameters with usability constraints are parameters that are continuous in nature but are evaluated only at certain discrete values. An example is the diameter of a hole. It physically makes sense that a hole could vary in diameter and have an infinite set of possibilities, but due to manufacturing constraints, we would only evaluate hole diameters that correspond to drill bit sizes currently in use on the factory floor. Similar usability (or manufacturability) constraints may apply to items such as a fillet radius or a plate thickness.

Expanded Visualization Capabilities: The Parameter Navigator has been expanded to include several different views of the user's parameter space.



DesignXplorer's three-dimensional chart navigator provides instantaneous feedback via a 3-D response surface which corresponds to changes in design criteria (i.e. parameters) for a greater overall understanding of product behavior.



DesignXplorer's goal-oriented optimization menu allows you to find design parameters which correspond to optimized design by both traditional and non-traditional design optimization strategies.

The Parameter Navigator's expanded views include:

Enhanced Response Surface Chart: Providing the full suite of capabilities needed, DX has enhanced graphics within its Chart Navigator view.

For 3-D graphs, DX provides meaning full displays of discrete parameters vs. continuous parameters and discrete parameters vs. discrete parameters as well as the previously released capability to display continuous parameters vs. continuous parameters. When viewing a discrete parameter vs. a continuous parameter, DX only shows slices (isolines) corresponding to the discrete parameters, thereby only showing valid values of all parameters displayed. When viewing a discrete parameter vs. a discrete parameter, DX shows a 3-D histogram representing only valid values of both discrete parameters.

General Enhancements

- Ability to simultaneously mesh solids and surfaces in a single assembly*
- Option to export analysis results into Microsoft Excel spreadsheet*
- Option to erase mesh and solution data from database to conserve disk space*
- Simulation reactions displayed as parameters in detailed menu*
- Configured to support operation of Space Ball® and Space Mouse® 3-D motion controllers*
- Worksheet View displays contents of operation free folders in convenient Excel-like spreadsheet format*
- User's choice of Cartesian and cylindrical coordinate systems, in addition to stand and global systems, available in geometry definition menu*
- Memory option in boot.ini file boosts memory to 3 GB on Windows XP desktops
- ANSYS Connection for CATIA V5
- ANSYS Connection for Caltech Intermediate Format (.cif) for ECAD applications

ANSYS Workbench Environment

Improved Contact Robustness

ISolver Enhancements

Usability Enhancements

Nonlinear Diagnostics

ANSYS Geometry Healing Module

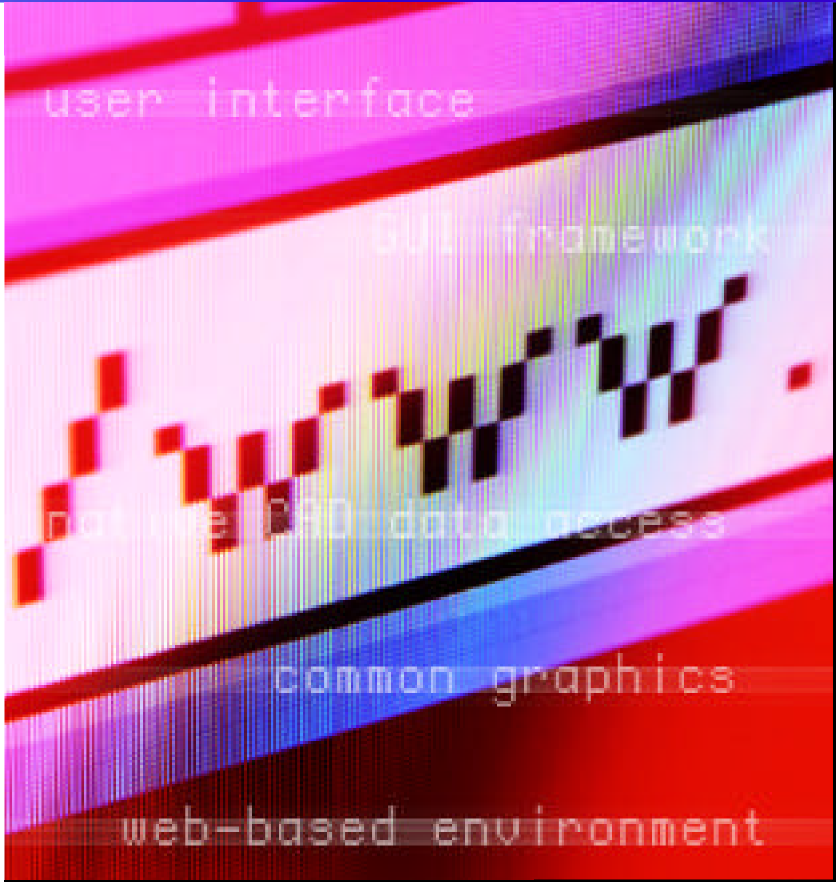
***Available within the ANSYS Workbench Environment only**

ANSYS Workbench Environment

New at Release 7.0 is the ANSYS Workbench Environment*, which is built upon a new platform that offers an efficient and intuitive user interface, superior CAD integration, automatic meshing, access to model parameters, and access to much ANSYS functionality, all within one environment. This feature also offers extensive customization potential and add-on modules such as DesignXplorer and Fatigue. Within the ANSYS launcher, you chose which environment to use: the Workbench Environment or the Classic ANSYS interface.

*Currently, this environment will be given to all TECS paying customers requesting ANSYS 7.0.

Why are we doing this?
Who is the customer?
What is the ANSYS Workbench Environment interface?
Won't jumping between the ANSYS Workbench Environment and the ANSYS applet seem clunky?
How would I use it?
How will it be different than the way I'm used to working?
What are the benefits – and the drawbacks – to this interface?
What are our expectations for ANSYS Workbench Environment?
What are the products we are releasing?
What is the future of ANSYS Workbench Environment?



Why are we doing this?

For our existing ANSYS customers, ANSYS Workbench Environment provides geometry and meshing benefits that they have been asking for.

Additionally the Workbench platform provides our customers (and their companies) with:

- Access to additional modules such as AGP, DesignXplorer, Fatigue, and other Workbench-based components
- Access to future technologies as the Workbench is the future platform for other technologies that we will deliver by developing, acquiring, or partnering
- ANSYS Inc. and its channel partners can deliver customization services based on Workbench

Who is the customer?

The existing ANSYS customer is the primary focus, particularly those that are struggling with geometry import and meshing and those that want to easily do parametric studies.

New customers who need more than what DesignSpace has to offer in terms of functionality will also be attracted to the product.

What is in the ANSYS Workbench Environment interface?

- Linear Static Stress
- Static Thermal
- Modal
- Linear Buckling
- Shape Optimization
- Automatic Assembly Contact
- Improved Cad Geometry Access
- Nonlinear Frictional Contact
- Harmonic Analysis
- Mesh Controls
- And More…

ANSYS functionality is then integrated into the Workbench via the "ANSYS Apple,t" and includes 3 pieces:

1. Pre-processing Command Builder

- builds, either via a GUI or manually, an ANSYS command input stream which is inserted into the input file ANSYS Workbench Environment creates to input into ANSYS for solving
- has GUI access to:
  - change element types and keyoptions
  - contact manager (change contact properties)
  - solution controls
  - material definition (nonlinear material properties)
- allows direct typing of ANSYS commands and APDL
  - therefore provides access to all ANSYS functionality
- these command "snippets" are maintained in the ANSYS Workbench Environment tree
  - saved in the ANSYS Workbench Environment database
  - can make a change and rerun, and the snippets are also re-inserted for the run

2. Post-processing Command Builder

- allows GUI access to POST1 post-processing via the Results Viewer
- allows direct typing of ANSYS commands and APDL
- allows image and listing capture back to the ANSYS Workbench Environment tree (and therefore into the ANSYS Workbench Environment report)

3. Load ANSYS

- takes current ANSYS Workbench Environment input into ANSYS and starts up the ANSYS classic enviroment for further processing
  - no geometry, but ANSYS Workbench Environment now allows creation of node components based on geometry for easier manipulation

Won't jumping between the ANSYS Workbench Environment and the ANSYS applet seem clunky?

Yes, to a degree – but this is still better than switching between most preprocessors and their analysis engine.

A way to think of it is that ANSYS will be embedded in an ANSYS Workbench Environment session much like an Excel spreadsheet can be embedded in a PowerPoint presentation. Clicking on the spreadsheet "jumps" you into the Excel environment – switching back to PowerPoint switches you again back into its environment.

Snippets can be pre-written and inserted into every analysis thereafter, so for cases where you work with a particular material or use a certain element option, you will not need to go to the ANSYS applet.

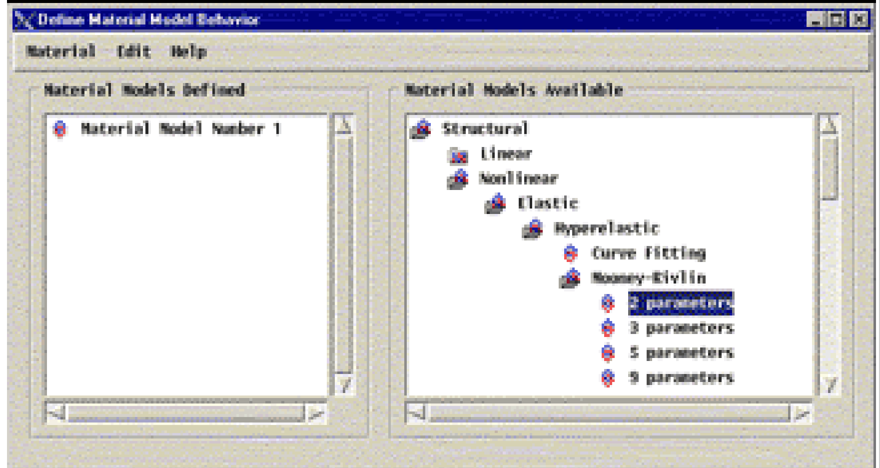
How would I use it?

If you are doing a nonlinear contact analysis with linear material properties – or any type of analysis that the ANSYS Workbench Environment supports natively – you will never venture into the ANSYS applet.

Here are a few scenarios where you would need to use the ANSYS applet:

1. Hyperelastic bushing. Here you have an assembly of metal parts with one of the parts being a rubber bushing. You would:

- attach (import) the model
- set up the parts to be the appropriate metal material apply the boundary conditions
- right-click on Environment, and Insert->Pre-processing Commands
- right-click on Pre-processing Commands and choose Pre-processing Command Builder
- once the ANSYS applet loads, select the part from the tree
- in the ANSYS material model interface, navigate the Material Models Available window to the hyperelastic folder and enter the properties
- exit the ANSYS applet and back to ANSYS Workbench Environment via a button
- solve and post-process in the native ANSYS Workbench Environment GUI as usual



In the ANSYS material model interface, navigate the Material Models Available window to the hyperelastic folder and enter the properties

How would I use it? (continued)

2. Extract contact forces. Here you have run a nonlinear assembly contact analysis fully from within the native ANSYS Workbench Environment GUI and you'd like to extract the force one body exerts on another:

- After solving, right-click Solution and Insert->Post-processing Commands
- right-click on Post-processing Commands and choose Post-processing Command Builder
- after the ANYS applet loads, type the ANSYS commands to select the part CMSEL) in the command input window
- use the Capture List button and enter the command "FSUM,,CONT" and a caption; this output is captured into the ANSYS Workbench Environment tree for inclusion into the report.

3. Perform a thermal transient. The native GUI allows for the setting up and solving steady-state thermal problems. To switch this to a transient:

- in the Processing Command Builder, add thermal capacitance material properties if not already defined
- back in ANSYS Workbench Environment, in the Preprocessing Command window, add the commands:

```
antype,transient
kbc,1                    ! step-apply the loads
time,…
```

- solve and post-process (the solution at the last time point) as usual
- to view solutions at other time points, or to obtain time-history (POST26) graphs, use the Post-Processing Command Builder

4. Model a bolted joint. Here you would do most of the pre-processing in the native GUI and then switch to ANSYS ("Load ANSYS") to define the bolt pretensions and solve and post-process:

- attach (import) the model including the bolts
- set up the parts to be the appropriate materials
- apply the boundary conditions
- using Named Selections, make components of the bolts
- right-click Environment and choose Load ANSYS
- use the PSMESH command in conjunction with component selects (made from the Named Selections) to pretension the bolts
- solve and post-process as usual in ANSYS

How will it be different than the way I'm used to working?

This is not just a new user interface; it is a new way of working. The ANSYS Workbench Environment interface has a very different user-interaction model that what you are accustom to with ANSYS. However, it is much more Microsoft-like and very similar to the more recent CAD modelers.

The biggest difference is that there is no longer a concept of a "log" or input file. Since the ANSYS Workbench Environment can work with parametric values (including the CAD geometry), and it remembers the state and influence of one object on another, it can update itself when a value of one object or parameter changes.

What are the benefits – and the drawbacks – to this interface?

Benefits:

- geometry import success rate
- maintains associativity with CAD
- meshing robustness
- parametric input
- access to ANSYS functionality and APDL
- AGP, Fatigue and DesignXplorer add-ons
- fosters collaboration between DS and ANSYS users
- extensible platform

Drawbacks:

- applet switching somewhat clunky
- no geometry transfer to ANSYS
- no handling of multiple load steps/multiple substeps (only one result object can exist)
- different element shape metrics

Improved Contact Robustness

Fewer contact issues will occur and the remaining ones will be easier to diagnose. The following capabilities have been added:

- Updating contact stiffness at each iteration
- Controlling the maximum allowable elastic slip parameter
- Monitoring contact chattering through contact status changes
- Ability to move contact nodes from inside the ICONT zone to the target surface
- Ability to extend the edge of a target surface
- Monitoring and controlling contact pair settings
- Detecting contact at the nodal level
- Detecting symmetric contact pairs
- Harmonic analysis support for surface-to-surface contact elements
- Monitoring and contour plotting Newton-Raphson residuals from equilibrium iterations
- Checking contact status before solution
- Checking of element shape parameters for deformed elements

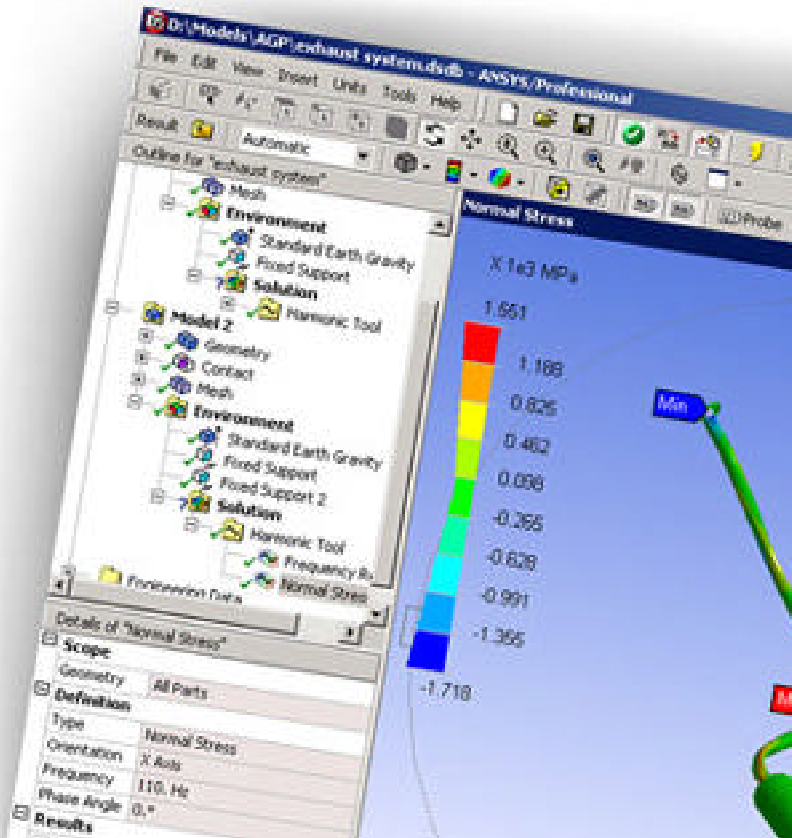
Solver Enhancements

- Sparse Solver may be employed during the generation of a substructuring analysis and to perform back substitution during the expansion pass
- BCS Sparse Solver now may be run in parallel for full harmonic analyses
- SOLID95 (also SOLID186) elements now may be used with the PCG Solver in addition to the SOLID92 (also SOLID187) elements
- Distributed Domain Solver (DDS) works for deformable-to-deformable, surface-to-surface contact elements from 169 to 170, and other node-to-node contact elements. It does not work for rigid-to-deformable or node-to-surface contact elements in general. It also does not support elements with U-P formulation or Lagrangian multiplier as their theoretical bases.

What is the future of ANSYS Workbench Environment?

The native ANSYS Workbench Environment will continually be enhanced to expose more and more ANSYS capabilities. The ANSYS applet GUI will be expanded (if required) based on customer feedback.

Eventually, over the course of several years, we will expose enough of ANSYS in ANSYS Workbench Environment that the need for the ANSYS applet will be minimized or even eliminated.



Improved Contact Robustness

Fewer contact issues will occur and the remaining ones will be easier to diagnose. The following capabilities have been added:

- Updating contact stiffness at each iteration
- Controlling the maximum allowable elastic slip parameter
- Monitoring contact chattering through contact status changes
- Ability to move contact nodes from inside the ICONT zone to the target surface
- Ability to extend the edge of a target surface
- Monitoring and controlling contact pair settings
- Detecting contact at the nodal level
- Detecting symmetric contact pairs
- Harmonic analysis support for surface-to-surface contact elements
- Monitoring and contour plotting Newton-Raphson residuals from equilibrium iterations
- Checking contact status before solution
- Checking of element shape parameters for deformed elements

Solver Enhancements

- Sparse Solver may be employed during the generation of a substructuring analysis and to perform back substitution during the expansion pass
- BCS Sparse Solver now may be run in parallel for full harmonic analyses
- SOLID95 (also SOLID186) elements now may be used with the PCG Solver in addition to the SOLID92 (also SOLID187) elements
- Distributed Domain Solver (DDS) works for deformable-to-deformable, surface-to-surface contact elements from 169 to 170, and other node-to-node contact elements. It does not work for rigid-to-deformable or node-to-surface contact elements in general. It also does not support elements with U-P formulation or Lagrangian multiplier as their theoretical bases.

Usability Enhancements

- CMLIST, CMPLOT and CMSLET commands ease component designation and usage specifications
- New NRRES, ESCHECK and CNCHECK commands investigate causes of non-convergence in nonlinear structural analyses along with revised CHECK and PLNSOL commands

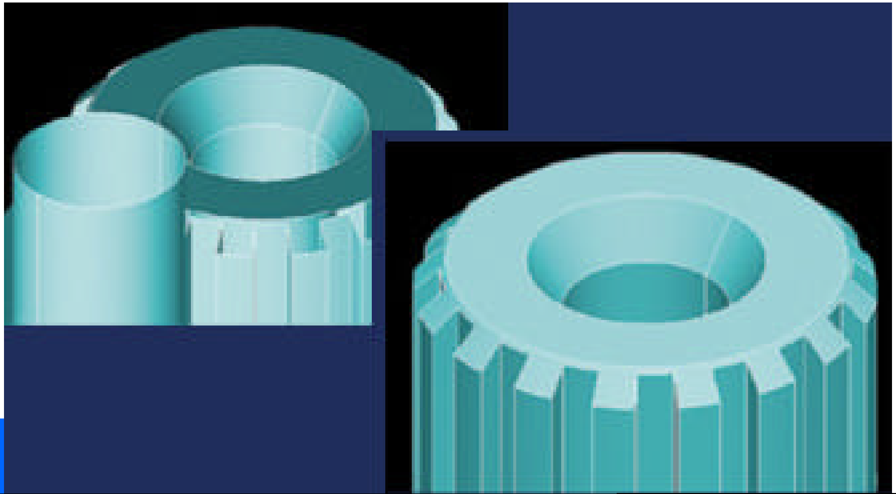
ANSYS Geometry Healing Module

ANSYS 7.0 features healing advancements in the ANSYS Geometry Healing Module. The Module now includes full GUI support and utilizes the full functionality of the latest CADfix technology, with dramatically increased success in healing. Improved ease-of-use and increased reliability allow the ANSYS Geometry Healing Module to resolve complex geometry issues.

Nonlinear Diagnostics

You can investigate and identify possible reasons for nonconvergence in many nonlinear structural analyses, including the following:

- Monitoring and contour plotting Newton-Raphson residuals from equilibrium iterations
- Checking contact status before solution
- Monitoring shape convergence



CAD model containing extraneous entities and missing areas–unsuitable for simulation

ANSYS Structural Enhancements

- Specify velocity and acceleration loads for rotating element components using the CMOMEGA and CMDOMEGA commands
- New tools for Nonlinear Diagnostics
- New Point/Edge-to-Surface Contact Element (CONTA175)
- Curve Fitting for Material Constants for Hyperelastic material models
- New Multi-Point Constraint (MPC) Rigid Link or Beam Element
- Hyperelastic Material Models for HyperFOAM and BLATZ-KO
- Cyclic Symmetry Buckling
- Cyclic Symmetry Traveling Wave Animation
- Viscoelasticity supported by BEAM188, BEAM189 and LINK180
- Allow SURF153 and SURF154 to have complex pressures
- Generalized Plane Strain Enhancements
- Substructure generation by Sparse solver
- Unsymmetric Sparse Solver can be run in shared memory parallel
- Element-by-Element PCG for SOLID95 (MSAVE)
- DDS solvers for contact and CE, CP

New Elements

Element Enhancements

Material Model Enhancements

ANSYS Thermal Electric Enhancements

- Electric current conduction bonded contact capability
- Electrostatic bonded contact capability
- Solution time for the radiosity solver method improved through storage of view factors in binary file format
- SHELL131 (4-node) and SHELL132 (8-node), 3-D layered elements, feature inplane and through-thickness conduction capability

Electromagnetics Enhancements

High-Frequency

- Fast frequency sweep capability
- Automatic extraction of S-parameter matrix
- Touchstone file format supported
- Pyramid element shape support in SOLID120

(Note: you must use the HF119 element for tetrahedral shaped elements in a mixed hex-pyramid-tetrahedral element shape domain).

Low-Frequency

- S-parameter calculation macro
- Adaptive meshing for High-Frequency EMAG element HF119
- New key options for SOLID97 and SOLID117 to ensure a solenoidal condition for current density
- Voltage-fed conductor option for SOLID117 and SOLID97 magnetic field elements
- LMATRIX support for voltage-fed conductors using SOLID117 and SOLID97

ANSYS FLOTRAN Enhancements

- New temperature field algorithm calculates film coefficients at walls in conjugate heat transfer problems
- Local wall roughness now may be input in a CFD analysis
- Transient analysis option now includes the Newmark time integration method

ANSYS LS-DYNA Enhancements

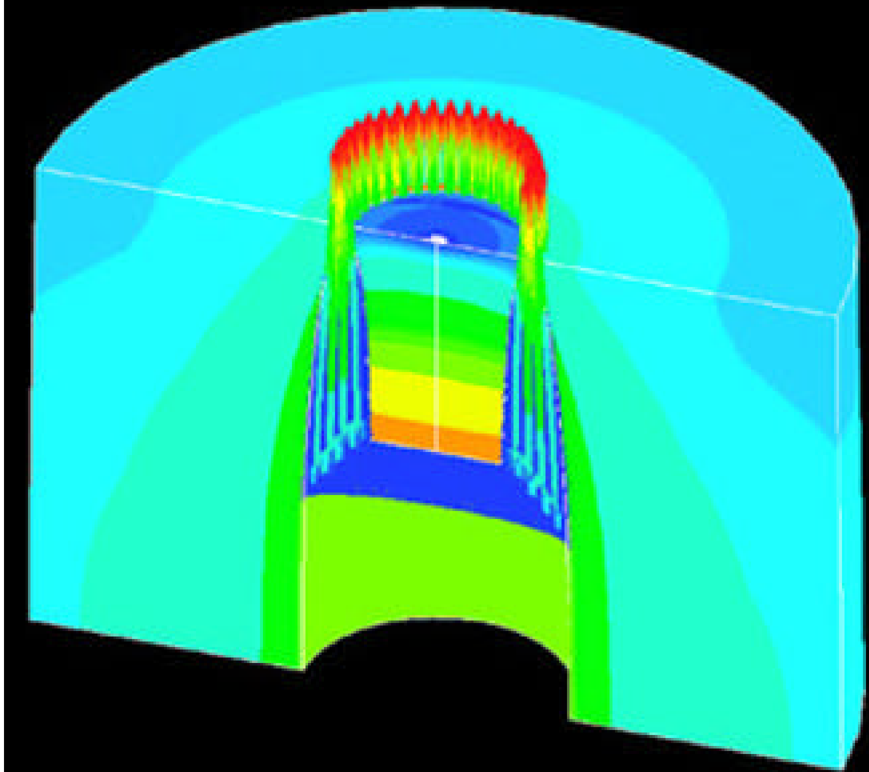
- New EDALE and EDGCALE commands support Arbitrary Lagrangian-Eulerian (ALE) formulations for very large deformations (e.g., extrusion, forging)
- Double precision capabilities now available on HP AlphaServer platforms – new EDDBL command enables feature
- Animations across multiple results files now possible through ANMRES command

Couple-Field Enhancements

- Reduced order modeling (ROM) tool solves coupled-field problems involving flexible structures
- Importation of 2-D Caltech Intermediate Format (.cif) files now possible for solving layered integrated circuit devices

ANSYS Multiphysics

- Newmark scheme for FLOTRAN, provides consistent time integration for FSI
- Reduced Order Modeling (ROM) tool for MEMS
- ANSYS Connection Caltech Intermediate Format (.cif) for ECAD applications
- ANSYS Frequency Sweep HP Module (see FEMXplorer)



Time-Saving Tool:
Automatic Geometry Healing Module

New ANSYS module, based on ITI TranscenData's CADfix technology, automatically heals and repairs model geometry.

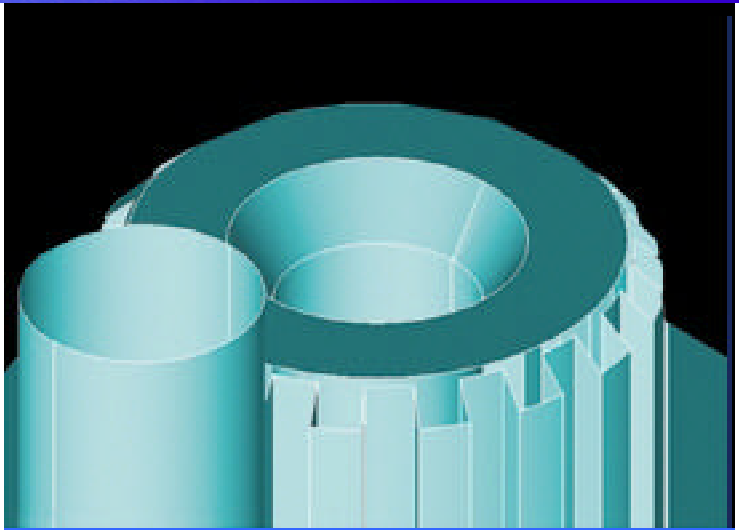
Jennifer Ferello
Manager, Global Vendor Solutions
ITI TranscenData
jenny.ferello@transcendata.com

In a market rife with change, consolidation and collaboration, the importance of working seamlessly with all forms of electronic data has never been greater. Nowhere is this more apparent than in the world of analysis. As enterprise-wide organizations struggle to find ways to reduce design cycle times, the communication between CAD and CAE becomes even more critical. Users need the tools to enable these changes as well as the ability to choose the best-in-class solution, and vendors need to focus their development resources on gains in their own products, free from the worry of interfacing with other products.

The National Institute of Standards and Technology (NIST) recognizes that CAD interoperability is a billion dollar problem for U.S. auto manufacturers alone and is currently funding the FIPER project aimed at promoting interoperability at every level. ANSYS has found a more immediate way to satisfy both their end users' needs and their own, by partnering with ITI TranscenData to provide CADfix-based healing and translation functionality for their flagship product, ANSYS.

The new ANSYS Automatic Geometry Healing Module, based on ITI TranscenData's CADfix technology, was recently released with ANSYS 6.1 and works with the current ANSYS Connection products. For even more robust geometry healing, the Module can be coupled with CADfix-based imports (IGES 5.3, STEP, CATIA v4, ACIS, Parasolid, VDAFS). Unlike CADfix for ANSYS, the Healing Module is a completely automatic healing process. It utilizes CADfix as background technology to heal and repair geometry from any of the connection products and tailors that geometry for re-use within ANSYS, communicating with ANSYS itself to check the integrity of the results by performing Booleans and/or meshing of individual faces.

The Healing Module tackles fundamental inconsistencies and inaccuracies, which complicate the translation of solid models from CAD systems to ANSYS; closing gaps, deleting voids, splitting surfaces and healing the data to a tolerance appropriate to ANSYS.

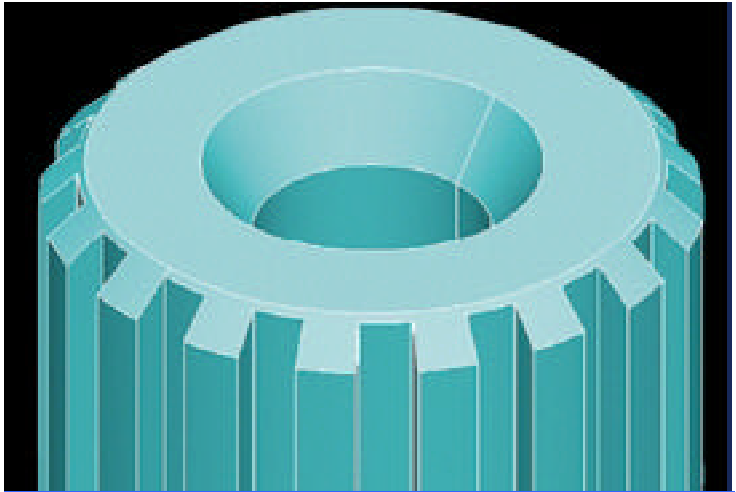


CAD model containing extraneous entities and missing areas is unsuitable for simulation.

Independent research has shown that up to 70 percent of the time spent on a finite element analysis project is spent reworking the original data so that it can be used within a CAE program. The automatic healing process can save an analysis engineer countless hours, even days, of unnecessary, non-value-added work. Eliminating this kind of wasted effort provides an immediate benefit to the organization by further compressing design cycles, shortening time-to-market and lowering overall costs.


ANSYS®
scalable integrated solutions
V7.0
version 7.0 is now available.

Using simple translation, Honeywell Engines was only able to achieve a 20-25 percent success rate. With the addition of the automatic healing, their success rate tripled to 60 percent, and by using the interactive CADfix, they were able to achieve an overall rate of 90 percent success, leaving very little rework to be done in CATIA. Using this example applied to the research above, this reduces the time spent in rework from 70 percent to 7 percent!



Revised CAD model, with extraneous entities removed and missing areas completed, generated through the ANSYS Automatic Geometry Healing module.

“Although we are always working to make our Connection Products the best they can be, we recognize that sometimes translation is not enough," said Ray Browell, ANSYS product manager. "Our users have been requesting automated healing technology, and we are happy to be partnering with ITI TranscenData to provide it. Relying on TranscenData's expertise in this area gives us the freedom to focus our development efforts on our core competencies thereby delivering improvements in core functionality, such as nonlinear mechanics, solver technology and multiphysics simulations, not to mention coupling our Workbench technology to ANSYS for unmatched usability." "The translation and healing capabilities of CADfix are unmatched in the market," explains Tom Makoski, vice president of sales and marketing at TranscenData. "Getting this technology into the hands of the ANSYS user base will allow ANSYS customers to take full advantage of the power the ANSYS product offers without having to worry about where the data originated."

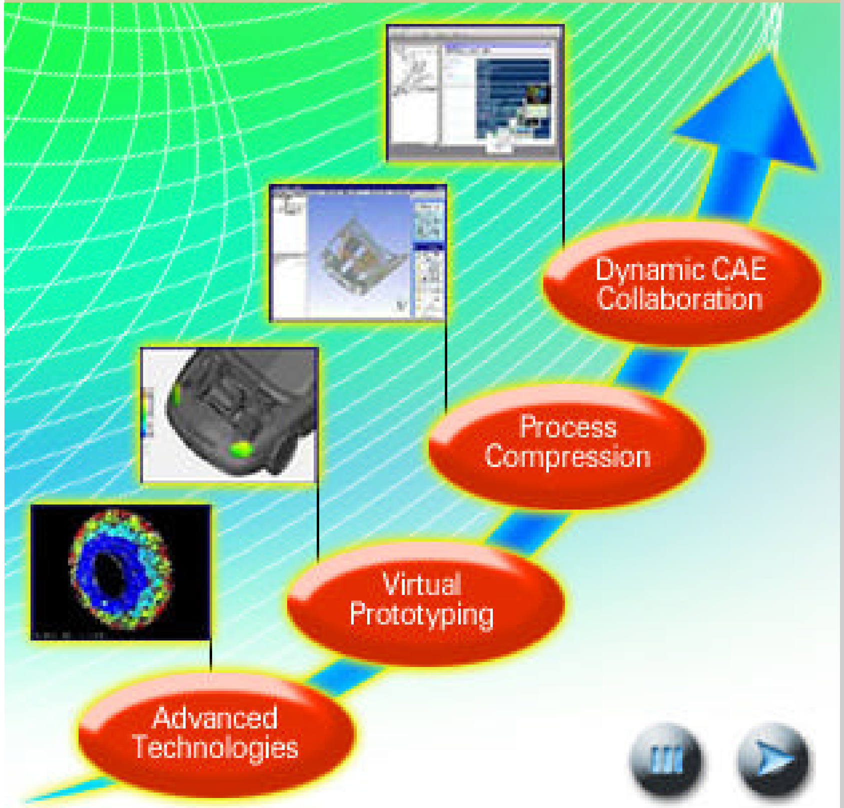
Note: The full, interactive CADfix product may be purchased directly from ITI TranscenData. For additional information, visit www.transcendata.com

Reaching the Full Potential of CAE

Companies can integrate an expanding range of CAE solutions into the product development process that yield greater levels of value for the overall enterprise.

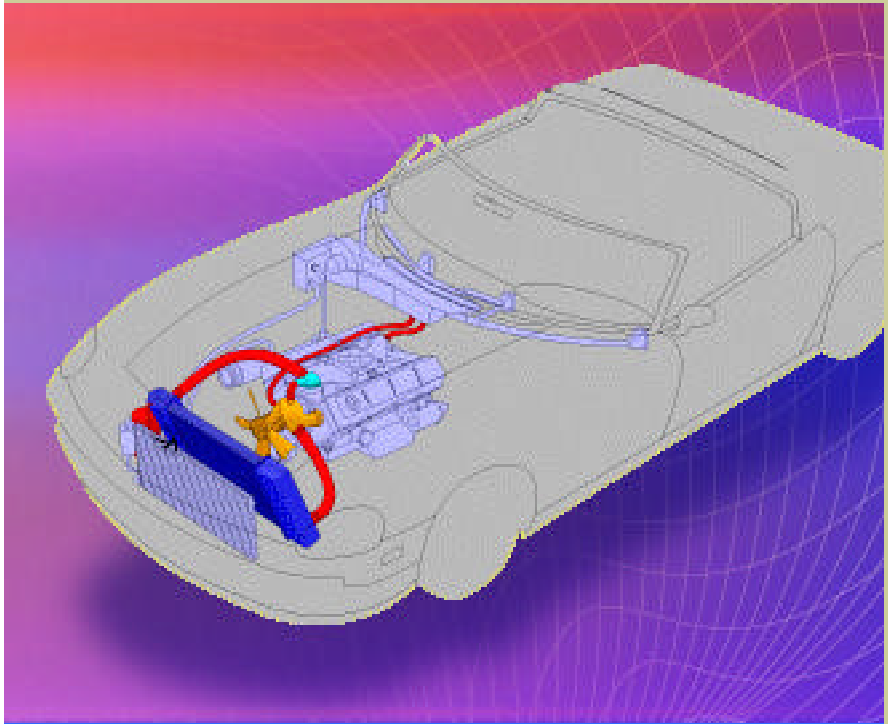
By James Cashman
President and CEO
ANSYS Inc.

In the 30-plus year history of the CAE industry, solution technology and applications have matured and grown significantly. Whereas in the past, companies had a comparatively limited number of tools and capabilities at their disposal, organizations today can choose from an expanding range of CAE solutions that yield greater levels of value for the overall enterprise by progressively integrating the solutions into the product development process. All companies are different, of course, and the manner in which CAE is applied depends on many factors unique to each organization including the type of products it designs as well as business goals and market objectives.

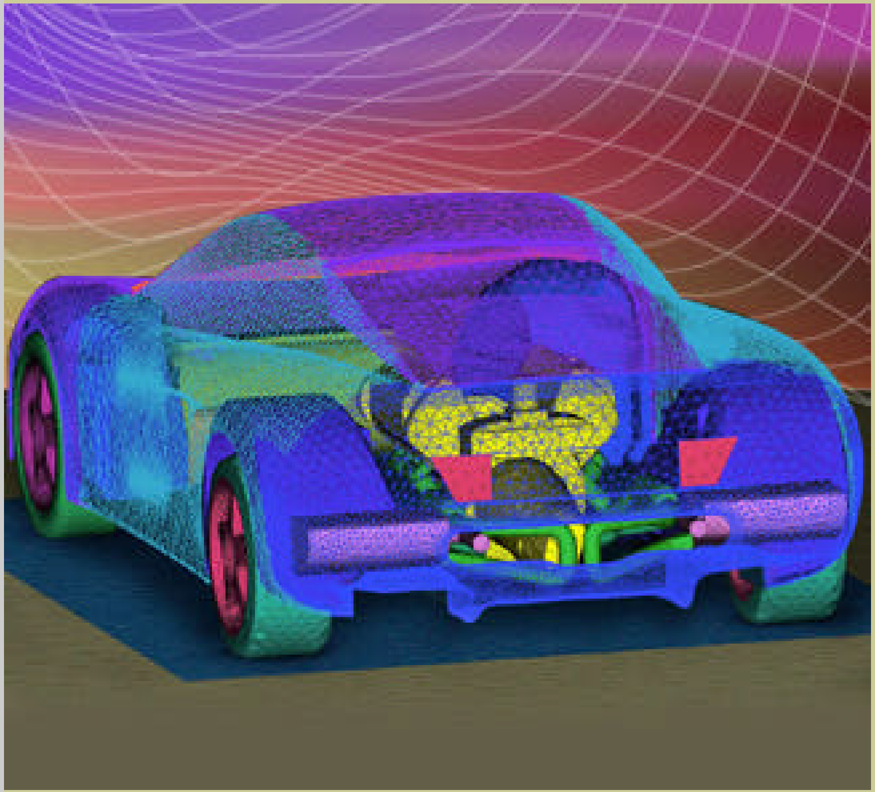


Advanced technologies are available to perform structural analysis and other detailed studies in solving linear and nonlinear problems, usually on individual components or assemblies that affect product performance. In these analyses, simulation tools utilize a range of technologies in finite element solvers, graphic pre-processing, modeling, material properties, part-to-part contact and other areas. Complex, real-world problems may include determining stresses in an automobile axle, for example, deformation of an engine-mount rubber bushing, or fluid flow in an ink-jet cartridge. These studies enable engineers to develop improved part designs that will function reliably throughout their lifecycle. Solving these types of problems is at the core of simulation applications and the foundation on which all further CAE application steps are based.

Virtual prototyping simulates complete products in their operating environments. Generally, these are multiphysics analyses that study the interaction of all factors encountered throughout the product during operation. Virtual prototyping studies predict the acoustics and ride of a railway locomotive as it enters a tunnel, for example, the effectiveness of an automotive cooling system on a hot day, or the deformation the body and wings as an aircraft lands on a runway. These are problems that 15 or 20 years ago were considered impractical to handle but today are now done routinely across nearly all manufacturing industries to study and refine real-world product performance.

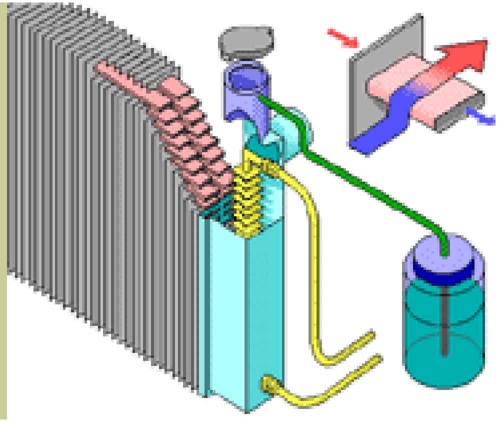


In implementing CAE, companies can take progressive steps, each integrating simulation solutions more into the company's product development process and yielding greater levels of value.



Process compression is possible by using analysis tools and virtual prototyping to simulate product performance and refine designs early on in development to spot and correct problems before they get to testing or manufacturing. The process can also be streamlined by automating routine tasks and customizing simulation tools to handle problems faster, with this time saving used to look at more alternatives and run through multiple "what-if" scenarios. Solutions are also available to balance often-conflicting attributes in synthesizing an optimal design. These capabilities let companies shorten time to market, reduce costs, improve quality, and create innovative designs by deploying simulation technology earlier in product development and making it an integral part of the design process. In work at Toyota, for example, development time and cost (including the number of physical prototypes) was reduced by between 30% and 40% in a front-end loading initiative where simulation was done early in design.

Dynamic CAE collaboration allows companies to leverage advanced simulation technologies and processes across the entire organization and throughout the design chain comprised of various groups with differing roles in the cycle. Collaborative capabilities built into simulation tools now enable people in disciplines beyond design and often in dispersed locations to work together in developing innovative products that are on target with market requirements. These embedded collaborative capabilities include functionality for efficiently capturing simulation information and sharing this intellectual capital across the enterprise, for example, and automatic reporting tools for readily extracting critical analysis data and communicating it throughout the organization. This all makes it easier for those not experienced in simulation to still fully utilize simulation results in the product lifecycle.

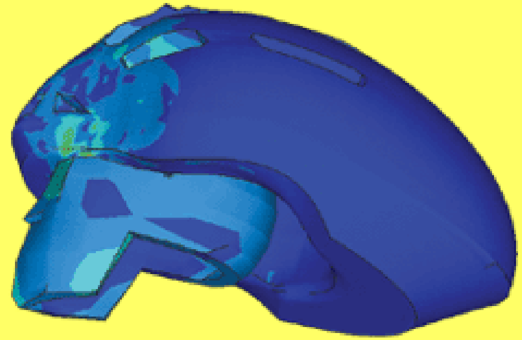


These progressive steps in implementing CAE require a development platform to serve as a backbone for the entire continuum: a framework that allows companies to bring together different applications, streamline processes and share data. All this comes together to make simulation an integral part of the product development process so companies can reach the full potential of CAE and make these technology-based processes a key element in long-term business strategies for rapidly delivering winning products to market.

Assessing Bicycle Helmet Protection:

Engineers accurately determine energy absorbtion of helmet design by simulating head impact.

Hubert Lobo
President
DatapointLabs



The study of product and component performance under impact conditions is vital to many industry segments ranging from automotive and aerospace to consumer products and toys. Considerable effort is devoted to testing finished products to determine their ability to meet the challenges they will face in daily life. The objective of this study was to show how virtual prototyping could help improve the efficiency of product development by providing answers about viable designs early in the cycle.

Protective sports equipment is in wide spread use in recreational activities ranging from bicycling to roller blading. The ability to protect the wearer from injury while remaining lightweight and comfortable to wear is an important feature of any design. ANSYS LS-DYNA was used to simulate head impact when a child rides a bike at 25 mph into a telephone pole. The child's bicycle helmet was the only barrier between the modeled human skull and the rigid telephone pole. The helmet, skull and pole were created using standard CAD tools (Fig. 1). It was meshed using TrueGrid, courtesy of Livermore Software Technologies Inc. (LSTC).

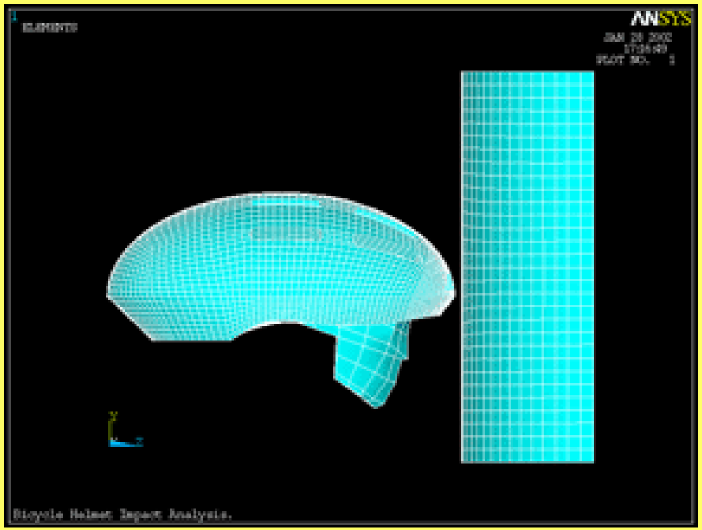


Figure 1

The bicycle helmet was made of a cellular energy dissipating foam covered with plastic membrane or skin. The material properties of the foam were derived from physical testing of an existing, medium cost helmet that is typically sold in toy and department stores. DatapointLabs, an ANSYS Inc. ESP partner, did the testing and produced material properties that were ANSYS input-ready, meaning that the analyst was able to simply take the data supplied by DatapointLabs, add it to the geometric and environmental simulation models, and run the explicit dynamics simulation. The properties were derived using the following procedures. Cylindrical test specimens were cut from the foam material. These specimens were then placed between compressive platens and subjected to compression under impact loads of two and four m/s in a Dynatup Instrumented Impact Tower. The resulting load-time data were converted to load deformation traces, from which, the stress-strain data were calculated. Compressive data were also developed using a conventional universal-testing machine.

Virtual prototyping helped improve the efficiency of product development by providing answers about viable designs early in the cycle

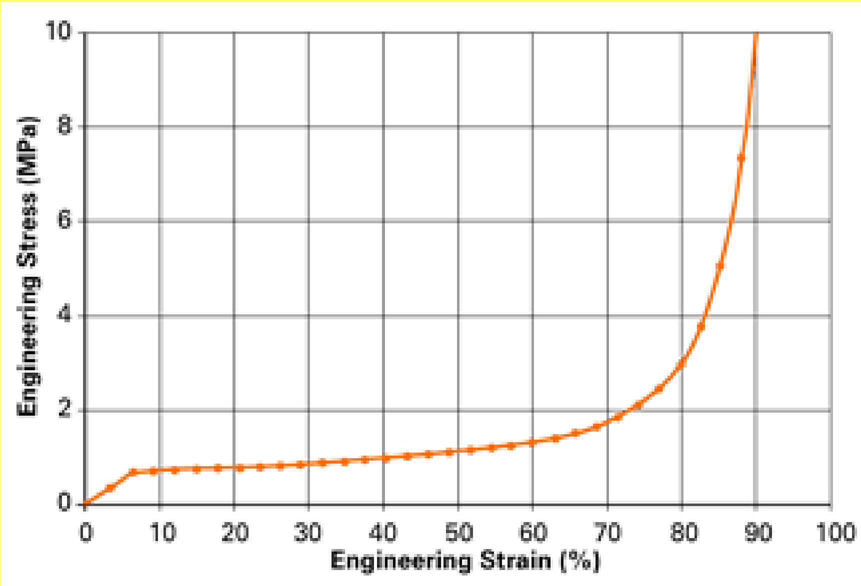


Figure 2

The data were fit to a crushable foam material model, which is capable of fitting the highly nonlinear stress vs. strain response of these materials. Note the behavior of a typical crushable foam (Fig. 2): there is a small linear elastic range, followed by a long crush range, then an exponential increase in stiffness as the foam reaches its crush limit. A viscoelastic material model was used to simulate the head response. The density was increased by a factor of three times in order to account for the brain matter that was not modeled. The pole was considered to be a rigid material since it is much stiffer than the helmet or head.

Contact was modeled between the skull and the helmet, the helmet and the pole, skull and the pole using general node- to-surface contact elements. A static friction coefficient of 0.2, dynamic friction coefficient of 0.1 and a viscous damping coefficient of 10 was used. For simplicity, the same values were used for all cases. The loading for the model consisted of an initial velocity of 25 mph (440 in/s) forward into the pole (+Z direction), and 6 mph (105.6 in/s) downward (-Y direction), applied to both the skull and helmet.

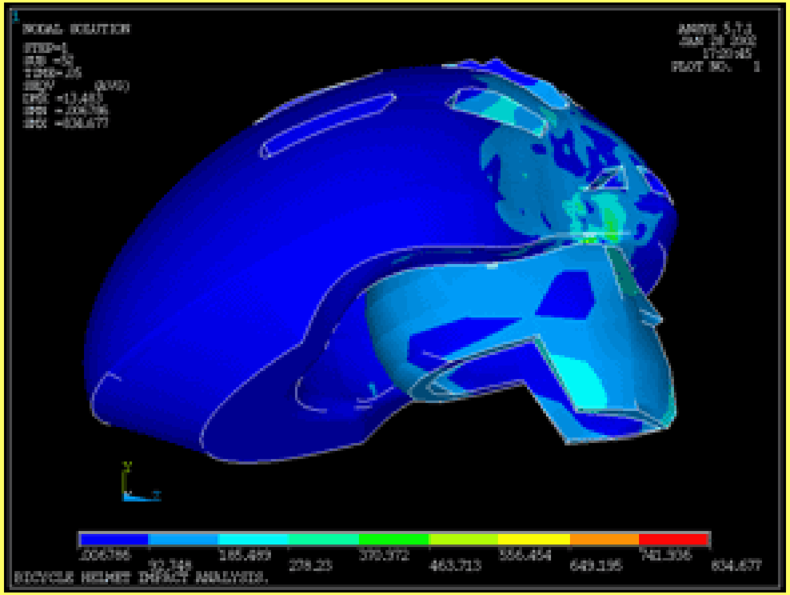


Fig. 3

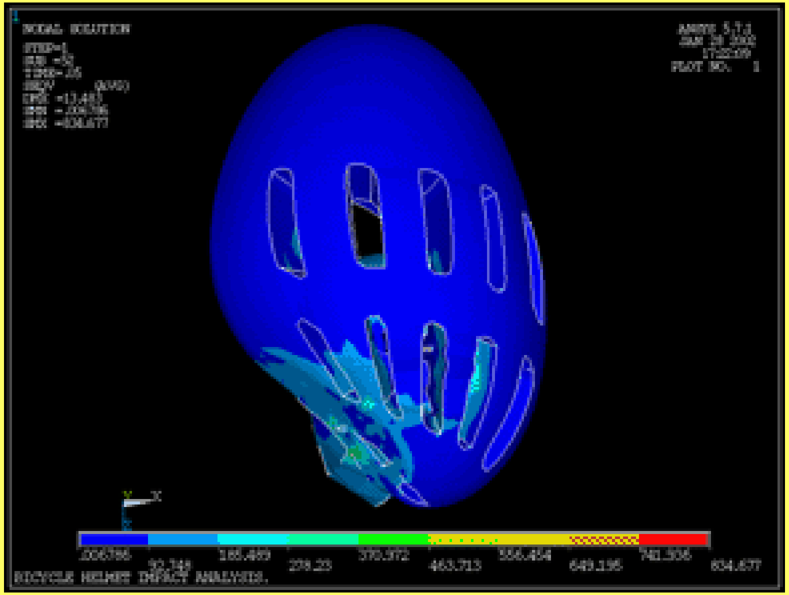


Fig. 4

The simulation run time on a single processor NT PC was estimated to be 30 days. The same problem was run on an SGI with eight CPUs, with results calculated in two hours. The results show that while the helmet does absorb a significant amount of the energy of the impact, a high amount of energy is transferred to the skull and a skull fracture is the predicted outcome (Figs. 3 & 4). While model refinement could have some effect on the calculated solution, the results suggest that a helmet redesign would be needed to dissipate more of the energy while preventing protective cavity intrusion by the pole. Alternatively, the helmet could be rated for a lower impact velocity, which would ensure helmet wearer survivability.

Nonlinear Analysis in the Aerospace Industry

FEM/FEA Helps Unravel Mysteries In Tragic Crash Of Boeing 747 Airliner Near New York's JFK Airport

Introduction

Painstaking reconstruction and analysis of the ill-fated 747-100 airliner that was Trans World Airlines Flight 800 has provided a great deal of information about how to build and maintain airplanes to avoid a possible fuel tank explosion. Moreover, this is testimony to the power of combining computer modeling and analysis with wreckage recovery and reconstruction, especially since the ultimate cause of the crash may never be known.

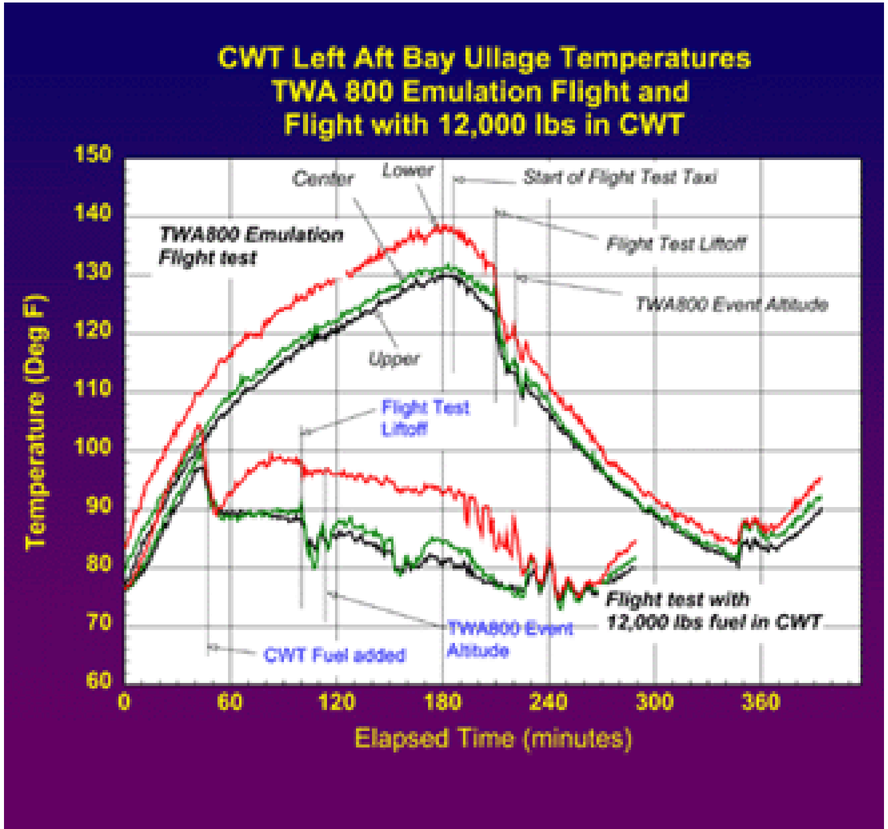


Chart courtesy of the FAA

Challenge

What remains untold is the story of the intensive and unprecedented finite-element modeling and analyses that were carried out as Boeing, TWA, the National Transportation and Safety Board and the Federal Aviation Administration strove to understand what happened to the 747-100's airframe after an explosion in the center wing tank (CWT). According to Boeing's final report to the NTSB, "This tragic accident launched the most complicated and comprehensive wreckage recovery, aircraft reconstruction, and accident investigation in the history of commercial aviation."

Analysts were faced with the challenge of determining the sequence of events after the explosion to understand why it happened and what needed to be remedied in terms of fracture mechanics to prevent a recurrence. They verified and incorporated data from similar events into not one but three separate types of analyses - kinetics, computational fluid dynamics (CFD) and structural. Managing the sheer size, complexity and mass of detail (some of it conflicting) in the models and in the analysis runs. About 37,000 elements and nodes were required adding up to about 110,000 DOFs, each of which is a variable to iteratively calculated.

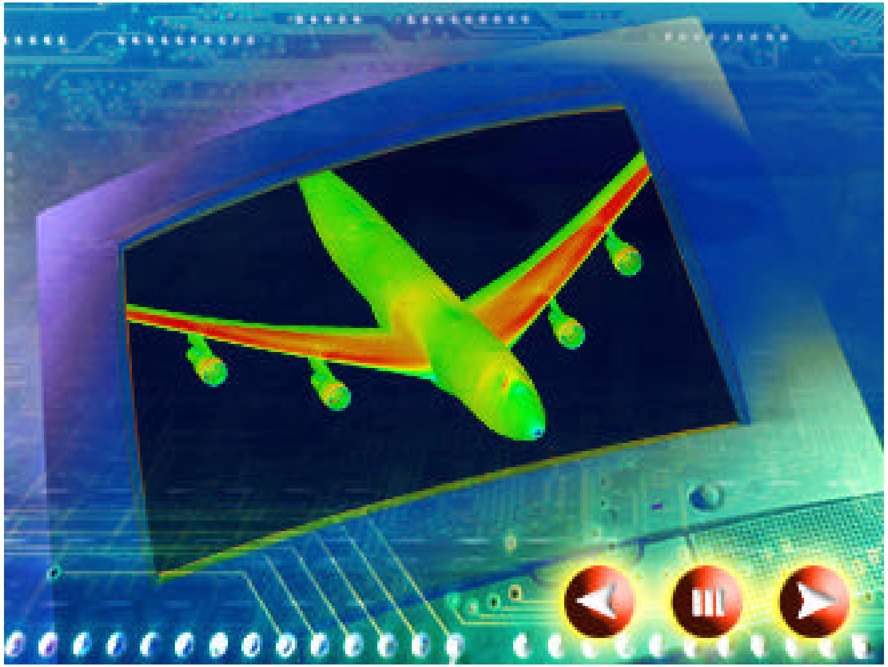
Solution

Using ANSYS (ANSYS Inc., Canonsburg, Pennsylvania, U.S.A.), analysts conducted structural analyses on a T-916 from Cray Inc. (Seattle, Washington, U.S.A.) rather than the usual, high-end desktop workstation. Installed at the Boeing corporate data center, the Cray has eight CPUs, each rated at a peak of 2,000 megaflops (millions of floating-point operations per second). It has two gigabits of random access memory (RAM).

Three of the analyses ran for 31 days and each used 533 hours - more than three weeks - of supercomputer CPU time. “By a factor of four or five, these were the longest analyses I had ever seen," said Patrick Safarian, former Boeing structural engineer who joined the FAA's Seattle office after the TWA Flight 800 investigation. "Other FEA runs in this study took from three hours to several days CPU time."

A team of Boeing Commercial Airplane Co. experts rather than engineers of the FAA or NTSB did virtually all of the computational analysis work. Once the center wing tank became the focus of the investigation, we ran out of easy answers for post-failure analysis. Traditional stress analysis was inadequate for these tasks, able only to provide rough approximations.

"The analyses that were performed were one of a kind. Such sophisticated analyses had never been done before in all the history of aviation," said Safarian. "Plus, there is no widely accepted, industry-standard way to integrate kinetic, CFD and structural calculations. We had to make lots of assumptions and then test them with a lot of 'what-if' analyses. This accounted for most of the complexity of the analyses."



The biggest effort in modeling the Boeing 747-100 center wing fuel tank that failed in the crash involved retooling an existing Boeing finite-element model originally created to certify the 747-400 freighter. In many ways, it was as much a CAD model as a finite element model.

“Despite the extensive modifications, we reached one of the first objectives: calculating the structural strength of the CWT with ANSYS," Safarian said. “The value we found with ANSYS in just 10 days could not be confirmed with the other FEA software package until two months later, which highlighted major advantages of ANSYS in speed and accuracy. As to verification and credibility, the answers differed by about 5 percent.

Only the structural analysis was done with ANSYS. The input to the ANSYS model was the kinetics and CFD outputs. But those two analyses were not very reliable; we needed answers and they were all we had. We used best estimates to put an envelope around the problem. The only part of the analytical study that actually gave reasonably accurate results was the ANSYS structural part."

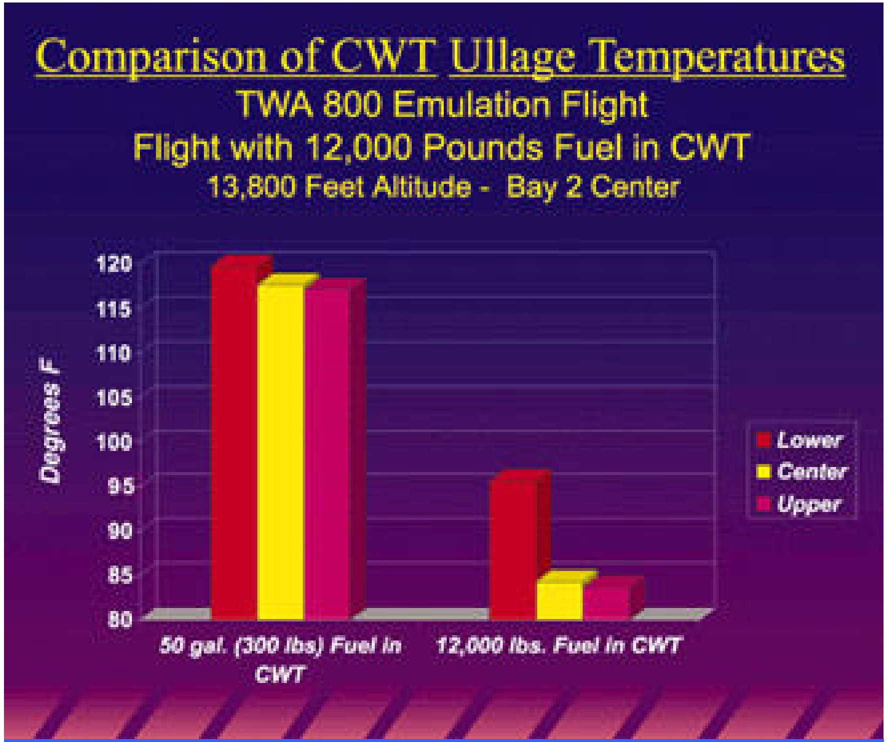


Chart courtesy of the FAA

Benefit

Numerous capabilities of ANSYS were helpful in this study. Because of the loss of life and the lack of a single obvious cause, time in this investigation was of highest priority while any compromises in the quality of the results were non-negotiable. ANSYS was able to deliver both accurate and timely analysis.

"Our foremost needs were speed and power, and the ANSYS capabilities for large deformation and plasticity in both static and dynamic models yielded highly accurate results in a comparatively short time," said Safarian. "ANSYS' overall flexibility, intelligent free-meshing capabilities, the excellent handling of very large-sized models, efficiency of the solvers for large deformations and plasticity, the well established convergence criteria, and the very efficient dynamic solution algorithm are among its benefits."

As evidence of the accuracy of ANSYS predictions, at least seven independent failure locations identified by ANSYS in the PAL 737-300 accident served as checks on the accuracy and validation of the TWA Flight 800 analysis. Identical behavior between the analysis and the documented Manila accident was shown. Comparison of the FEA results (failure/deformation locations and modes) with the actual wreckage from that CWT explosion provided excellent correlation. "This gave all of us a lot of confidence that we were going about this the right way," explained Safarian.

For other analyses, each capability of ANSYS was independently investigated. In every case, there was excellent correlation. In addition, the plasticity, large deformation and failure criterion employed in the ANSYS modeling was used to perform failure analysis of an actual built-up structure which had been physically tested earlier.

Who's Listening To The Customer?

Manufacturers are shifting from product-push to market-pull development processes driven by customer expectations and consumer demand.

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

Economic pressure and fierce competition are causing manufacturers to move away from product-push development processes where efforts are directed at marketing products based mostly on how well they are engineered. Stories abound about well-designed, reliable, and cost-effective products that flopped in the market because the public simply didn't like them or failed to see a need for them.

Such products may have been designed using the most advanced technology and meticulously tested to meet strict functional and safety requirements, yet development took place with little or no input from the customer.

In a major transition in manufacturing industries, many companies today are shifting to market-pull product development processes driven by customer expectations and consumer demand. Automotive companies started making this switch in the 1980s, and others have since followed suit. In the development of Boeing's 777 commercial aircraft, for example, major input on new systems and support procedures was provided by customers in the form of airlines, pilots, flight attendants and service personnel.

With consumer tastes more discerning and their demands changing more rapidly than ever, customers increasingly are now the focal point in product development. Market pull has replaced technology push and woe be to the manufacturer failing to respond. Studies have shown that recapturing customers once they are lost costs eight times as much as retaining them in the first place. Hence, the voice of the customer takes on strategic significance and is now a key element in the product development process. This major shift is truly a breakthrough.

Capturing the many aspects of the voice of the customer is no simple task, of course, and requires the efforts of many different groups in the manufacturing enterprise. Marketing activities in such organizations now include not only promotion and sales efforts to launch a product, but also up-front initiatives at the beginning of product development to provide input on customer purchase preferences and buying trends. Much of this information can be obtained from direct experience with customers, interviews, competitive benchmarking, focus groups and quality function deployment (QFD) studies.

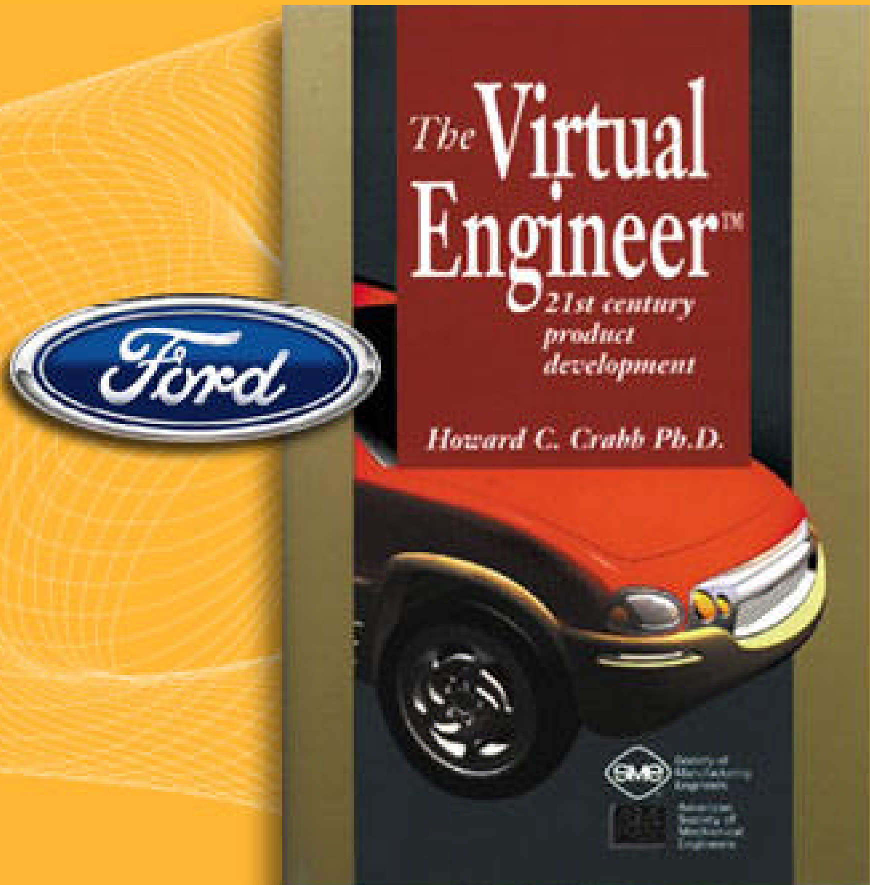
Likewise, other groups such as product support and customer service can provide valuable feedback on what customers want. Indeed, collaborative input from many different groups in the enterprise and supply chain is often necessary to factor in all the important customer requirements and balance the numerous often-conflicting attributes of a winning product.

In a broad sense, the most successful manufacturers include people and technology from across the enterprise to add value and strategic impact in product development. Each group contributes a unique, distinctive competency, with decision-making vested in cross-functional teams in the best position to use their expertise and most current information to determine what product attributes will work best in the market. Such design chains are absolutely critical in product development, because no one individual or group can possibly have all the answers. It's got to be a collaborative effort.

No longer can information on performance, trends, operations and other issues reside statically within individual departments or be gathered after the fact into historical archive reports. Translating market demands and customer needs into a final product requires efficient information sharing across the enterprise and into the product development process, especially in the early stages of design where the input can have the greatest impact.

Product development today extends far beyond the walls of the engineering department, even beyond the individual factory walls. In a globally competitive environment where one lost opportunity can sound the death knell for an entire company, basing customer-focused designs on input from groups throughout the extended enterprise and getting those products to market fast become overriding determinants of whether a company thrives, survives, or dies. ■

"With consumer tastes more discerning and their demands changing more rapidly than ever, customers increasingly are now the focal point in product development."



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

Dr. Crabbs book, ***The Virtual Engineer*** is available in hardback.

Predicting Long-Term Fatigue Life

Leslie Hogan
Marketing Communications
www.lmsintl.com

Durability analysis and FEA certify wind turbines to withstand 20 years of high winds and extreme weather.

Germanischer-Lloyd Windenergie GmbH (GL-Wind), a subsidiary of the German-based international classification and industrial certification society Germanischer Lloyd, is the world's market leader in wind turbine certification. The firm is accredited to certify turbines according to international and national regulations and standards including those for Germany, Denmark and the Netherlands. The procedures for certification are under continuous development and have made gains in accuracy and efficiency through the use of simulation technology.

Manufacturers of wind turbines are gearing up to double and triple production levels over the next five years to satisfy increased market demand for these power production systems.

The European Union Renewables Directive aims to increase wind-power capacity to the European grid from 24,000 to 64,000 megawatts by 2010. In addition, individual countries in the European Union have their own initiatives. Germany, for example, plans to add a total of 5,000 new five-megawatt offshore turbines by 2025 for its power generation needs.

To be certified, manufacturers must design a wind turbine that will reliably and safely operate for 20 years under often-extreme wind conditions. The critical loads acting on a turbine are mainly due to the fluctuations in wind speed and direction as well as those imposed by starting and stopping the system. These loads produce high levels of stress on internal components such as the rotor hub, which must be designed to maintain adequate fatigue resistance over its service life.

In the past, fatigue-life was evaluated based on calculations that did not fully take into account unpredictable forces hitting the structure at random intervals, the inter-relationships of multiaxial forces, or phase relationships between the individual loads. This oversimplification led to inaccurate predictions of mean stress. So the only way to secure the operational safety of the turbine was to overdesign it, thus increasing the cost of the systems and lowering operational efficiencies. Moreover, the calculations were done only for selected portions of the structure, limiting the engineers' understanding of the entire system and increasing the risk of overlooking potential trouble spots.

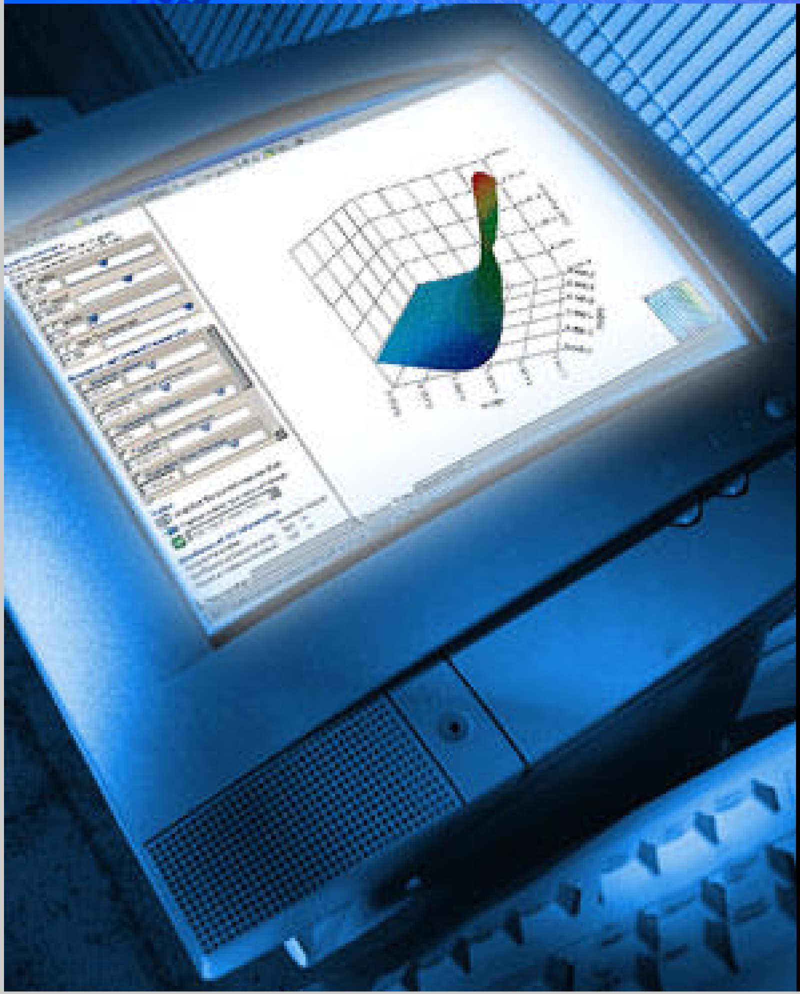

ANSYS®
V 7.0
scalable integrated solutions
version 7.0 is now available.

ANSYS

Now, GL-Wind is able to arrive at more accurate certification analysis through the use of ANSYS Structural simulation software in combination with the LMS FALANCS durability program from LMS International. Based on turbine design specifications and historical site weather information, GL-Wind uses a time-history domain approach where internal loads are applied to the ANSYS model. LMS FALANCS test schedules are then used to determine the fatigue life of components based on simulation of predicted events over the 20-year history. Phasing and mean stresses are accounted for in the simulation as well as strain curves based on material property data, wind speed parameters and material life curves.

The time history approach is significantly more accurate than previous calculations in determining fatigue life and is made practical by the speed at which ANSYS and LMS FALANCS perform the series of necessary simulations. The method includes all significant damaging events for a 20-year lifetime and accounts automatically for correct phase relations of load components and the correct mean stresses. Furthermore, the distribution of fatigue damage is displayed for the entire structure, thus giving engineers a valuable tool in assessing the response of the whole system and readily identifying potential problem areas.

The time history approach is significantly more accurate than previous calculations in determining fatigue life...



Instantaneous Design Feedback with DesignXplorer

Simulation response to parametric control helps engineers quickly produce optimal designs.

By Ray Browell
Product Manager
ANSYS Inc.
ray.browell@ansys.com

Virtual prototyping using simulation up-front in design goes a long way in reducing the number of build-test-redesign cycles and shortening product development time. However, these conventional tools handle only a very limited number of variables at a time. Users, therefore, must run multiple simulations to zero in an elusive solution to satisfy competing engineering requirements: strength versus weight, for example, or compactness versus heat dissipation.

Given today's complex products and multiple requirements they must meet, satisfying 10 or more of these variables at one time is not uncommon. To meet tight schedules and project deadlines, however, engineers don't have the time to make the large number of simulation runs necessary to arrive at an optimal design satisfying all these variables. So more often than not, they develop a design based on only one or two of the most critical variables and hope any conflicts with the others will be minor, or can be corrected later.

The problem is, conflicts between engineering requirements often cause major headaches downstream later in development when problems are more difficult, costly and time-consuming to fix. Plus, even in the best-case scenario with no problematic conflicts, designs are far less than optimal, just working and barely satisfying all the competing requirements.

Beyond Routine Optimization

Design optimization technology is available to automatically perform multiple analysis iterations, generally aimed solely at adjusting CAD geometry based on factors such as minimizing volume or weight. DesignXplorer takes this optimization technology a step further by determining the simulation response to those changes in the model's design parameters. The package works in conjunction with DesignSpace, allowing the user to make numerous critical adjustments to the design without having to constantly re-run it for performance evaluation.

In this way, DesignXplorer uses parametric control to provide instantaneous feedback on any design modifications. This ability comes in handy when optimizing parts for new conditions or creating new items within existing product lines, especially if users want to run "what-if" simulations to try out new ideas or compare alternative configurations.

Like DesignSpace, DesignXplorer is an easy-to-operate, front-end application. It associates bi-directionally with SolidWorks, Solid Edge, Autodesk's Mechanical Desktop and Inventor, Unigraphics and Pro/ENGINEER, and operates statically with CATIA. If using any other commercially available CAD system, DesignXplorer functions statically with standard Parasolid and SAT-formatted geometry files.

The typical interoperability configuration for a DesignXplorer system, in a standard Microsoft Windows (2000/NT/Me/XP) environment, looks something like this:

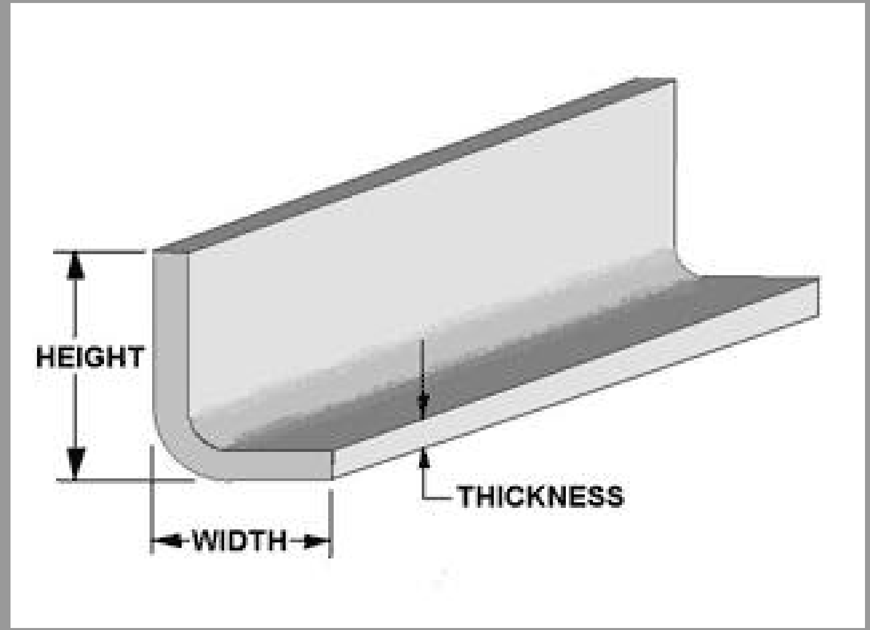
CAD System ↔ DesignSpace ↔ DesignXplorer

Utilizing a goal-driven optimization method, DesignXplorer's advanced parametric control allows users to formalize and manage all attributes of their designs. The design parameters for any given study may originate within the CAD system, DesignSpace or may be derived from within DesignXplorer itself.

Benchmark Study

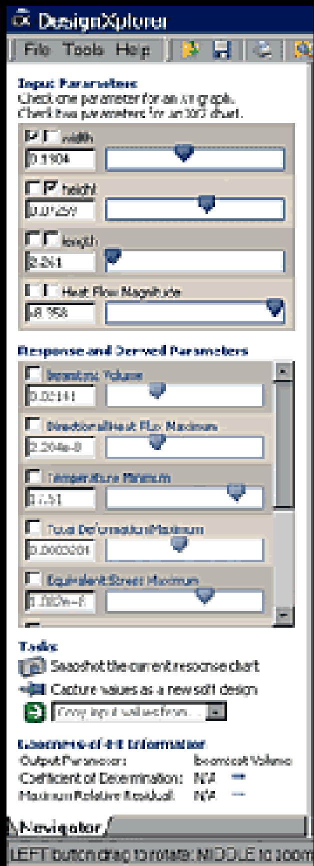
A benchmark study utilized DesignXplorer to calculate the stress, deformation and frequency responses for a standard L-shaped beam. The beam was calculated to possess a height of 25 to 35 millimeters, a width between 20 and 30 millimeters, and a thickness of 2 to 6 millimeters. DesignXplorer predicted the beam's optimum configuration under four possible conditions created by a variety of parametric input.

For the first calculation, a simple stress constraint was input to compute the minimum mass the beam required to maintain its structural integrity under a given stress value. This represents a traditional optimization strategy (i.e. only one input parameter) resolved through DesignXplorer. The subsequent two problems called upon DesignXplorer to employ non-traditional optimization (i.e. calculate multiple parameters) in order to maximize the beam's height configuration under the same stress constraint with both minimum deformation and maximum frequency parameters added.



A benchmark study utilized DesignXplorer to calculate the stress, deformation and frequency responses for a standard L-shaped beam.

DesignXplorer uses parametric control to provide instantaneous feedback on any design modifications.



DesignXplorer's advanced parametric control allows users to formalize and manage all attributes of their designs

Visualization Tools

DesignXplorer's visualization tools allow users to study, quantify and graph various structural and thermal analysis responses on all proposed component and assembly design modifications until the optimum configuration is achieved.

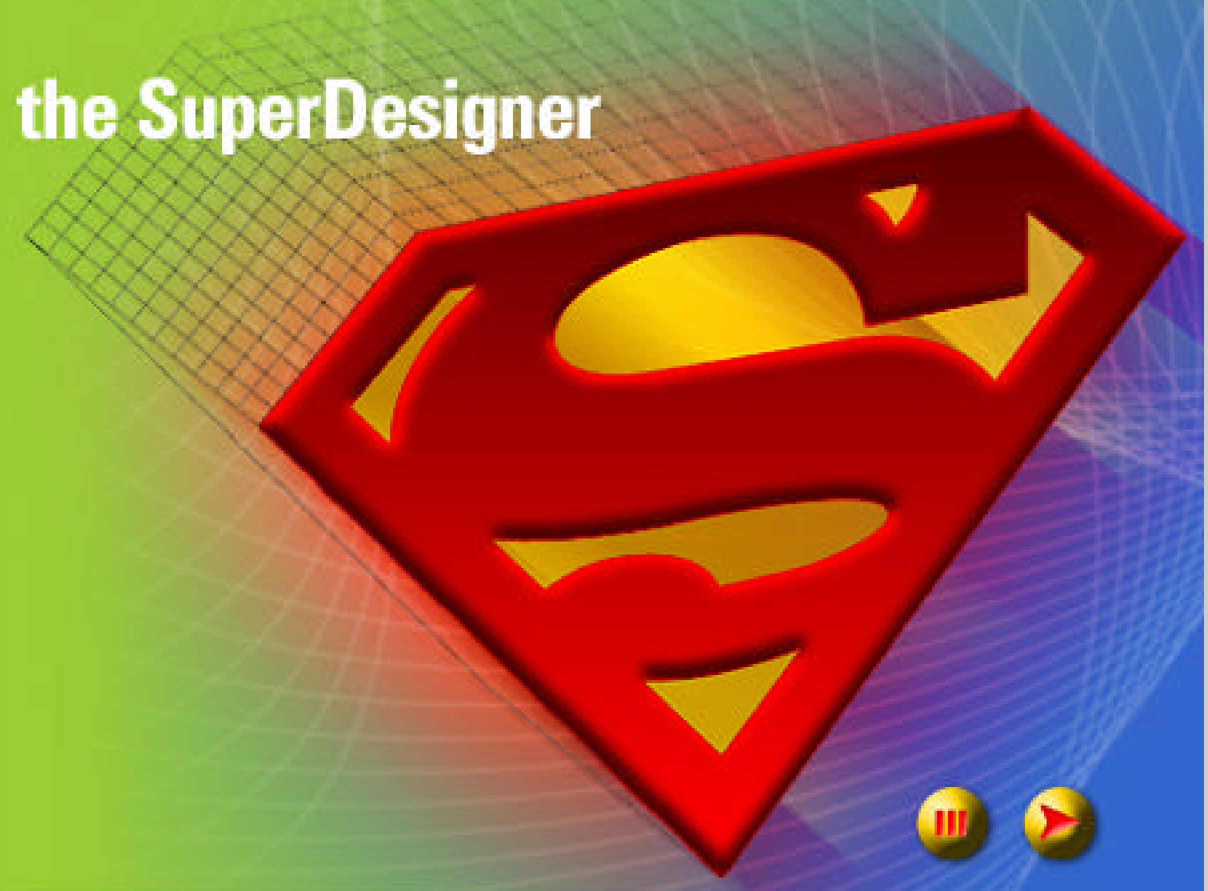
DesignXplorer generates 2-D design curves for quick verification of design changes as well as 3-D response surfaces for greater comprehension of overall product behavior. This is crucial for creating optimal designs in the early stages of development–when changes are easiest and most cost effective to make.

Early Recognition

Released this past April, DesignXplorer is already making a big splash within the CAE community. CADALYST magazine awarded DesignXplorer its **"Best of Show"** prize at the 2002 National Design Engineering Show, making it the only CAE product to receive such distinction.

To introduce DesignXplorer, ANSYS Inc. is extending the following offer: purchase a seat of DesignSpace Advansia combined with DesignXplorer and your order will be upgraded to DesignSpace Optima for no additional cost.

Emergence of the SuperDesigner



Don Brown
Manager, D.H. Brown Associates, Inc.
ITI TranscenData
dhbrown@dhbrown.com

More and more companies now drive greater, up-front simulation, supporting the analysis of conceptual designs early in the development cycle to iron out problems and optimize configurations. As a major characteristic that accompanies the trend, the roles of designers, engineers, and CAE analysts converge toward a “SuperDesigner” who assumes these responsibilities that now overlap. As a result, SuperDesigner programs at their base level–in simple terms–concentrate on educational programs that upgrade the skills of designers and broaden the software tools in use with a specific concentration on CAE.

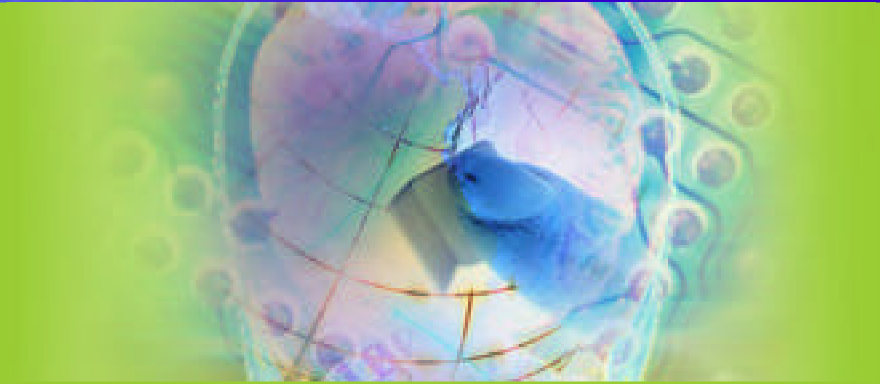
More broadly and fundamentally considered, however, the trend reflects a major effort to involve the entire product development community up-front, early in the design effort – not just design and engineering, but production and tooling, quality, marketing, and sales as well. Rather than hurriedly trying to develop a design that merely works and occupies space, SuperDesigner and related initiatives strive to concentrate first on basic design requirements. In other words, what customers expect in the end product. The collective experience of the group – the good, bad, beautiful, or ugly – must be considered early in the life and context of the particular product. Then, a whole parameterized system can be assembled to address the full range of performance issues and tradeoffs.

Designers may employ tools that focus on these main drivers to assess whether the design meets these performance targets, essentially the design goals of the product development process. The critical design variables (or parameters) typically involve a “control handle” in the design itself and may be exercised and evaluated through a variety of tools such as smart curves, automatic meshing, wizards, and templates to drive the analysis. Such a system provides clues as to what questions need to be asked and actions taken to ensure that potential problems are addressed.

This goal-oriented process of simulation and verification shows whether or not a design concept is worth further consideration. Today, a design may need to demonstrate that it is worth pursuing to be prototyped in hardware. Proof must be provided that it will work, and that it will meet market requirements, with virtual prototyping. In most cases, this involves the evaluation of many "what-if" scenarios and multiple simulations in which designs are iteratively evaluated and refined. The more times a design runs through such a virtual test environment, the closer the final product configuration is in meeting the design goals.

Naturally, the key focus in any SuperDesigner initiative relates to up-front simulation, which is often blocked by the organizational inertia of the traditional build-and-break cycle where prototypes are built, tested, fail, and then redesigned again and again until the product performs satisfactorily. In this type of cycle, engineers are often more likely to manage projects, make schedules, chase loose ends, and talk to vendors and suppliers. Little actual engineering work may occur at times, making the build-and-break process extremely inefficient in terms of engineering time and highly costly and time-consuming overall.

Changing behavior – changing the way business gets done – directly confronts the daily habits of people accustomed to this build-and-break mentality.



Overcoming this established way of operating is undoubtedly the greatest challenge in doing up-front simulation effectively. Workers in this entire process are very comfortable with the familiar build-and-break procedures.

Moreover, senior-level executives and managers, who absolutely need justification from a business standpoint for such a program, may regard up-front analysis with skepticism. Without the backing of management, which usually requires quantifiable results, such an initiative all too easily gets stuck in a bottoms up guerrilla phase. The advocates concentrate more on demonstrating benefits and internal marketing to peers, than in driving results.

The heaviest hitting priority in jump-starting an effective SuperDesigner program for up-front CAE relates to the assessment of engineering effectiveness. Tier one automotive supplier Eaton Corp. for example, reduced time and cost 30 percent to 50 percent in the design phase of development through up-front simulation on specific programs, with the biggest improvement coming from the reduction in the number of design iterations.

Their program focuses on designers performing first-pass analysis early in development, with the biggest improvement coming from a reduction in the number of design iterations. In a traditional CAE environment, four or five days were required to evaluate and analyze a product such as a clutch, for example. That cycle fell to only a few hours with designers performing up-front CAE. Eaton also registered a 36 percent reduction as compared to sending a design to a CAE core group. By having designers and engineers do first-pass simulation early, overall design time falls significantly.

By having designers and engineers do first-pass simulation early, overall design time falls significantly.

From a business perspective, such an improvement in engineering efficiency can easily translate into shorter time-to-market and lower development costs as well as improved quality. One of the areas where a SuperDesigner up-front analysis program can have a significant impact is in warranty costs, which is one of the biggest problems facing the automotive industry. Significant savings can also be realized in manufacturing, where tools and process can be established early and refined along with the product design. Otherwise, rework on the shop floor can voraciously consume time and resources. Likewise, up-front simulation can optimize a design to facilitate after-delivery product support and field service activities. In this way, the benefits of a SuperDesigner initiative extend throughout the entire product lifecycle.

D.H. Brown Associates Inc. collaborates with software vendors and leading end users to produce in-depth research reports on trends in engineering, manufacturing, and technical computing.

For more information, call 914.937.4302 or visit www.dhbrown.com.

Helpful Hints
to Keep in Mind When Writing Macros in APDL

John Crawford
Consulting Analyst

I write a lot of macros. My macros directory contains hundreds of them, and I add one or two new ones every week. Once in a while, I'll write a quick and dirty macro and throw it away when I'm done using it, but the majority of the macros I write are written with the intention of keeping them for future use, which means I try to write them in a reasonably professional manner. Here are some guidelines that I use that may be helpful to you.

Naming Macros

Try to use names that begin with a letter that relates to what the macro does or the type of entity it operates on. For example, a macro that splits a single PIPE16 element into "n" PIPE16 elements might be called E16SPLIT.MAC. You could just as easily call it SPLIT16.MAC, but six months later when you're looking for it in your macro directory you'll probably remember that since it works on an element that chances are good that the name starts with and "e", and this will be the first place you'll look for it. Using a convention like this isn't necessary if you only have 3 macros, but if you have hundreds you can easily spend 10 minutes trying to find a macro that you know is out there but you can't quite remember what you called it.

Introduction and Usage Information

At the top of the macro you should list the name of the macro, what it does, the arguments it uses, and any other information that would be useful to yourself or other users. Here's an example.

```
/nopr
!
! duhuh.mac A macro that generates a file containing
element type,
! nodes, and elements. Generates a file called "duhuh.txt".
!
! usage: duhuh,key,csys
!
! where: key =0 for unblocked NWRITE, EWRITE format
! 1 for blocked NWRITE, EWRITE format
! 2 for regular N and EN commands
! csys=coordinate system used for node locations
! (default is 0)
!
! written by: Alfred E. Neuman date: 4-1-02 version:
ANSYS 6.0
```

It's helpful to include your name, the date the macro was written, and the version of ANSYS it was written for. This will help you and others remember who wrote it and when it was written. This is useful when trying to track down the author or updating the macro to use new APDL commands and features that have been added since it was written.

While this may seem like more effort than it's worth, remember the first law of macro writing: The macro you write today will always mutate into the hieroglyphics of tomorrow.

What seems perfectly logical and obvious this afternoon will take you three hours to decipher next week, and in six months it will take an effort second only to the Manhattan Project to figure out what you were doing and why you were doing it. Spending one or two minutes writing an information header at the top of the macro will make your life easier as well as anyone else who uses your macros.

Comments

You can’t have too many comments. Following the First Law of Macro Writing, you should fully comment what you are doing and why you are doing it. Comments should be written as though the reader has never seen or heard of this macro before. In six months, that person will be you, and having your own personal Rosetta Stone included in every macro will save yourself and others a lot of time and effort.

Parameter Usage

Leading Underscore Parameters – These are reserved for ANSYS Inc. and are used for UIDL menus, tracking information that is written to CDWRITE and LSWRITE files, keyword status, and a host of other things. It is best not to use parameters with leading underscores to avoid overlapping present or future parameter names that ANSYS Inc. might use.

Trailing Underscore Parameters – These should be used by people who are writing macros that will be used by others. Almost all macros fall into this category, as most of us share macros with our coworkers. Every macro should clean up parameters before and after usage by using *DEL,,PRM_ at the beginning <<no mention of PARSAV or other>>and end of the macro. This will keep your macro from creating parameters that will conflict, overwrite, and otherwise cause problems with parameters that may already be in use.

Local Parameters – Anyone can use local parameter names like AR20-AR99 within their macro. Sometimes called local variables, they are only used within a given macro and automatically disappear when that macro is finished running. Local variables can be used in nested macros without problems.

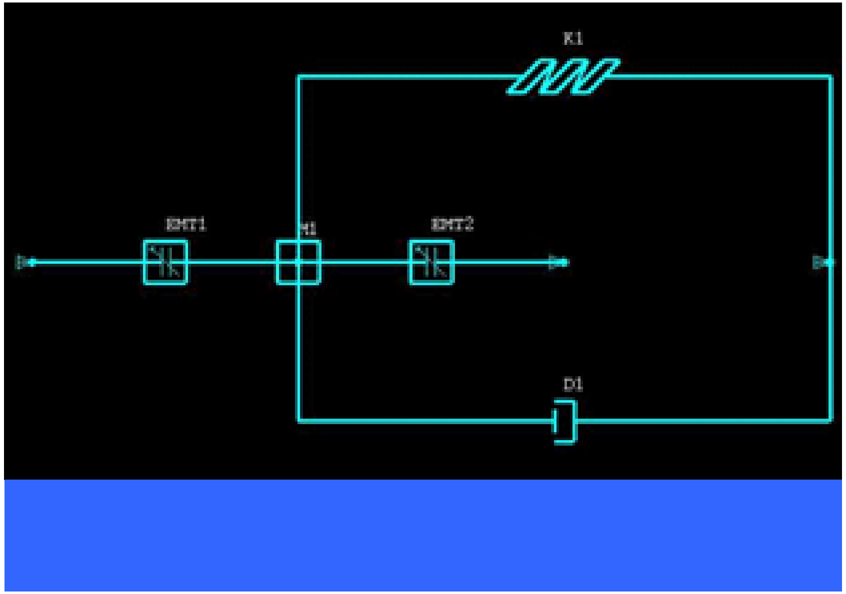
Regular Parameters – If your macro will never be used by anyone but yourself, feel free to use any parameter names you like. I rarely use regular parameter names because I know that sooner or later most of my macros will fall into the hands of others. If you're sure that no on else on Earth or any other planet will be using your macro, use any parameter names that you want.Using /NOPR and /GOPR

After your macro has been debugged and runs well, you can use /NOPR at the start of the macro to keep it from feeding unwanted information to the output window. When the macro is finished running, use /GOPR to turn the output window back on.Using *IF and *GO

Be sure to close all *IF statements with an *ENDIF. Don't use *GO to exit an *IF loop, use *EXIT. Feel free to use *GO as a way of hopping around within a macro, but keep in mind that it can be difficult for yourself or others to follow your logic at a later date.

Status Messages

At the beginning of each macro use *MSG to write a message to the output window that tells the user that the macro is running and what the input parameters are. For long macros, it is nice to print out additional messages to inform the user of the status of things as the macro progresses. You can also use the progress bar to graphically display the progress of a do-loop. <<site example>>



These are just a few of the things I keep in mind when I write a macro. Depending on the capabilities of the editor you are using and how creative you are, you can do much more. For example, if your editor supports colors you can assign everything within a given do-loop a unique color. Some people like to indent everything within do-loop. There are lots of other things you can do to ensure that your macros work reliably and are easy for yourself and others to read and understand. Spend a little time to think about this and I'm sure you will come up with ideas of your own.

Zeroing In On The Best Solution

Perform automated design iterations quickly and easily by linking ANSYS with a numerical optimization program

Dr. Shen-Yeh Chen
Senior Engineer, Honeywell Engines & Systems
shen-yeh@honeywell.com

The flexible ANSYS environment readily permits the integration of numerical optimization programs with finite element analysis. This automates design iterations so users can quickly zero in on the best solution, without having to manually configure the program for each run. The following chronicles the details of how this was accomplished at Honeywell Engines & Systems.

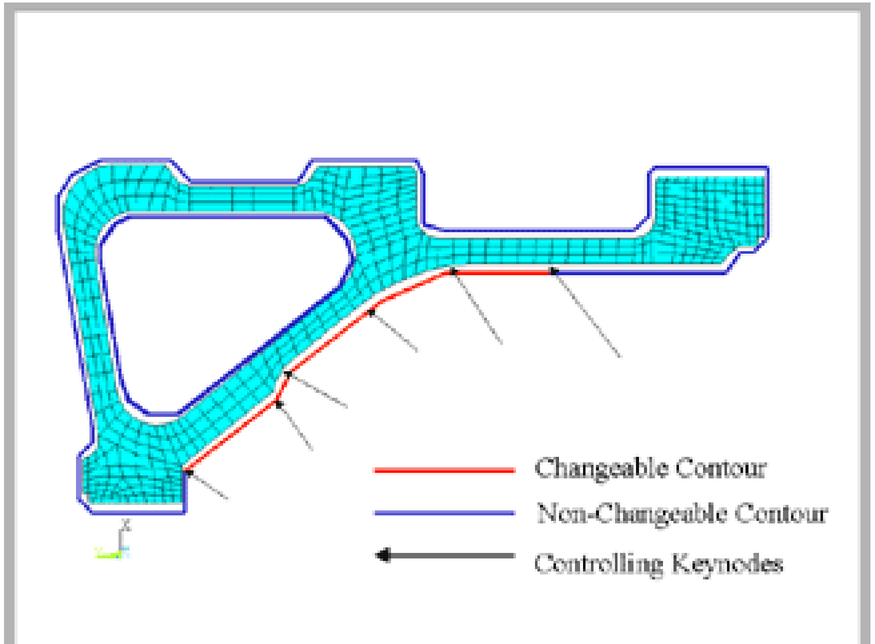


Figure 1. Finite element mesh and the contour controlling net.

CAD-Independent Parametric Modeling

It has been a common experience among analysts that parametric modeling is not easy to accomplish. De-featuring the original solid model takes time, which typically slows program completion. Building a parametric model—especially for FEA purposes—has proven to be especially time-consuming.

Moreover, the consistency of a mesh pattern cannot be guaranteed. If the mesh pattern cannot be maintained, there may be unexpected variations of stress, in addition to those caused by parametric changes. This is sometimes called the “stress oscillation” phenomena, and it will usually cause problems in the optimization process.

Engineers at Honeywell Engines & Systems have overcome this difficulty by utilizing a CAD-independent parametric modeling technique known as the “Contour Natural Shape” Function. The idea behind the function is that when only an FE mesh is available, the user should still be able to make parametric changes to the model without changing the existing mesh connectivity and pattern.

To illustrate this concept, consider a given mesh like Figure 1. First, the user divides the “contour” of an FE mesh into segments of controlling curves, without any CAD model information. The user input is not intensive since only a few key points control these segments.

The user then specifies the points to be moved and the magnitude of the movement. Using the parameters in ANSYS, this information can be saved and written out for external programs. The external program then reads the data, computes the new distribution of all nodes, and writes an ANSYS macro for defining the new coordinates. Finally, ANSYS reads the macro created by the external program and produces a new mesh, as shown in Figure 2.

The above procedure can be executed in one macro by using the flexible ANSYS Parametric Design Language (APDL). This model is indeed parametric since all of its modifications can be defined and saved as named parameters in ANSYS.

Design-Oriented Analysis

Parametric modeling is crucial for numerical design optimization. However, the ability to perform parametric modeling does not guarantee its use for optimization. Numerical optimization has its assumptions and limitations. Experience has shown that blindly coupling a parametric model together with an optimization routine will usually cause serious problems. This is why the above-stated methodology was developed.

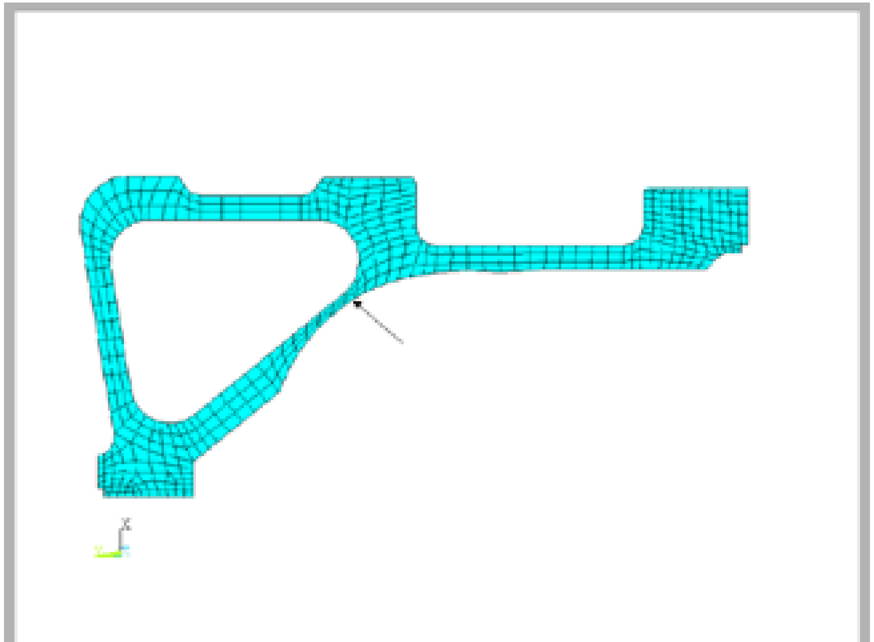


Figure 2. Perturbed mesh after moving the controlling node.

The concept of organizing and planning the global design process for the purpose of design automation is often referred to as “Design-Oriented Analysis.” With this concept firmly in place, Honeywell Engines & Systems is stepping forward to advance the analysis module so that it can be easily fitted into the design automation loop in the future.

Isolating Design Parameters

For the purpose of modularizing the FEA process, APDL can be utilized to separate the parameters from the modeling file and place them into another file. Other desired responses can be separated into different files, as well. This input/analysis/response model can immediately be fed into many stand-alone optimization programs.

For efficiency consideration, at Honeywell an in-house optimization code integrated with ANSYS was developed instead of using a standalone optimization package.

Using the named parameters in ANSYS as the interface and the APDL for model manipulation, the user can do as much number crunching as desired within ANSYS. In other words, the analysis procedure itself is programmable. This is almost impossible for most of the optimization packages available today.

Multidisciplinary Optimization

Another advantage of using ANSYS as an FEA solver in optimization is that it is multidisciplinary. With the architecture mentioned above, the optimization program does not need changes when an additional analysis discipline is added. If the users have a lot of in-house codes supporting ANSYS, with the right communication protocol, the in-house code can be added onto the optimization loop without additional programming or frustrating data transfer.

One example is the mesh perturbation module in the previous section. The module developed at Honeywell Engines & Systems can be used as either a separate parametric modeling tool through a macro, or in combination with the optimization loop by inserting this macro command into the ANSYS modeling batch file. This implies that it is possible to use ANSYS as an analysis platform rather than just as a module.

> “The flexible environment of ANSYS readily permits the integration of numerical optimization programs with finite element analysis.”

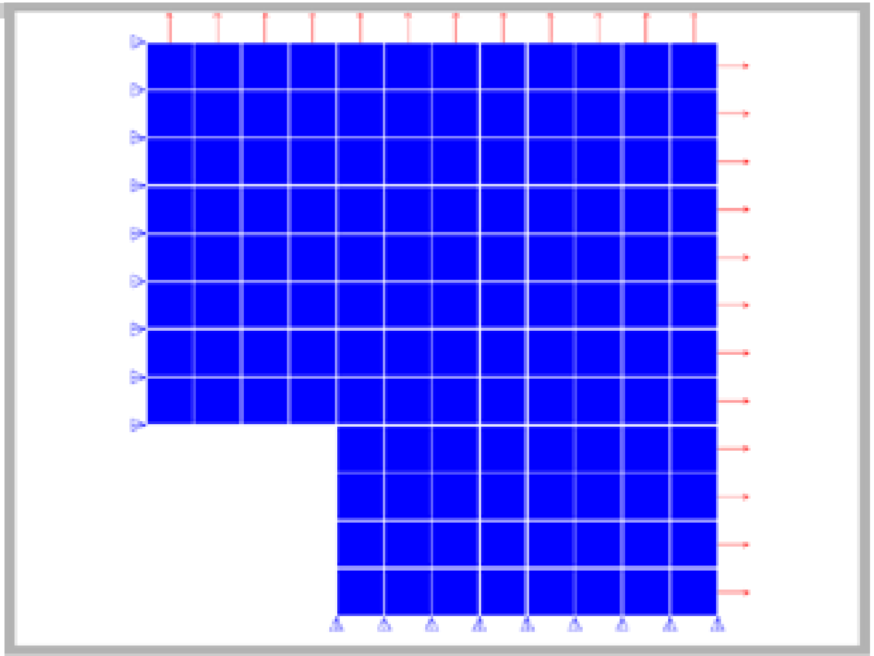


Figure 3. Finite element mesh and boundary conditions of the original geometry.

Reducing Engineering Costs

Finally, using the leading-edge numerical optimization technology provides the missing link from analysis response into a fast and optimum design. The immediate impact is the potential savings in engineering costs, since the iteration can be done in an automatic way.

For example, Figure 3 shows the FE mesh and the boundary conditions of a model. The goal was to minimize the weight, with the equivalent stress under a certain allowable value. The parameters for mesh perturbation took only a few minutes to set up.

An ANSYS batch file was then put into the optimization program and after about five minutes the optimum configuration—as shown in Figure 4 (next page)—was obtained (using a 2GB RAM HP C3000 workstation). Without the design automation tool, the same example would have taken more than one day to achieve the same result, lacking any prior knowledge of the physical significance.

Looking to the Future

Currently, Honeywell Engines & Systems has planned the incorporation of an even more flexible optimization algorithm, called the "Genetic Algorithm," into the in-house package. With this algorithm the package will provide more flexibility for the problems to be solved.

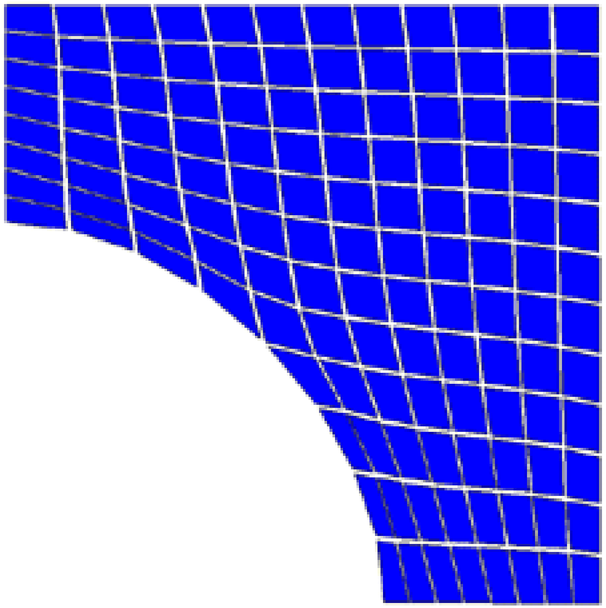


Figure 4. Optimal design and the final mesh.

Although this algorithm is more expensive due to its computational capacity, the growth of computing power has made it more practical. In the future, it is expected that the user will become more tolerant of these kinds of algorithms for handling complicated design requirements. This kind of non-gradient based algorithm is only made possible with today's ever-growing computing power and a robust analysis package like ANSYS.

These examples illustrate the potential for reducing engineering costs by integrating and automating the design process. Although it is still too early to expect a fully automatic design environment at this time, local automation is possible with a flexible analysis platform.

Surprisingly, such technologies have been around for decades, but questions of reliability have kept experienced engineers from investigating them. However, since 1995 there has been a significant increase in their acceptance among the various engineering communities. With proper education and effort, we can expect to see more and more successful design automation stories like this in the future.

Recent Announcements and Upcoming Events

ANSYS and Autodesk Enter Into Preferred Global CAE Partnership

I Autodesk — the world's leading design software and digital content company — selected ANSYS Inc. as their Preferred Global CAE Partner.

Autodesk and ANSYS will work closely together to develop mid-range simulation-based design performance tools for use by Autodesk Inventor customers, ensuring that sophisticated design engineering capabilities, including the new ANSYS DesignXplorer, are more accessible to product designers and engineers in need of easy-to-use, fast and accurate design performance simulation and optimization tools.

These front-end simulation tools will give product designers and engineers the ability to create more cost-effective and robust designs early in the product development process by facilitating quick and easy "what-if" scenarios, without the intricacies and the extensive learning curve of a sophisticated finite element analysis (FEA) solution.

This solution will help product development teams make more informed decisions earlier in the design process, allowing them to reduce costs and development time while developing better products.

"ANSYS has created a new generation of design tools that simulate product performance to meet functional requirements, study design trade-offs, and perform design synthesis much earlier in the product development process," said Dr. Joseph Solecki, chief technologist at ANSYS Inc. "This relationship with Autodesk will give our joint customers a competitive advantage by providing the ability to compress design cycles, to reduce risk and to enhance their ability to build the right products for the right market requirements."

"Autodesk is proud to be partnering with an industry leader like ANSYS," said Robert Kross, vice president of the Manufacturing Division at Autodesk. "There are many areas of synergy between our companies, including the opportunity to democratize simulation functionality, making it available to a broader mechanical design community. Our customers will surely benefit from this relationship."

ANSYS Inc. Joins Autodesk Inventor Certified Application Program

ANSYS Inc. recently joined Autodesk's newly unveiled Inventor Certified Application Program. The program certifies that DesignSpace for Inventor and DesignXplorer software applications have met Autodesk's standards for robustness, quality and the highest level of interoperability with its Inventor 3-D mechanical design system.

"We are proud to have a leading company like ANSYS join the Autodesk Inventor Certified Application Program," said Robert Kross, vice president of the Manufacturing Division at Autodesk. "Both of our organizations recognize the importance of delivering proven technologies to our customers. We view ANSYS as a key player in this new offering."

By meeting Autodesk's Inventor Certified Application Program requirements, DesignSpace for Inventor and DesignXplorer software packages will deliver to Inventor users ANSYS' proven high-end simulation technologies. DesignSpace for Inventor allows designers to simulate on the desktop the effects of real-world conditions on their 3-D parametric assemblies.

Complementing this functionality, DesignXplorer will allow them to perform "what-if" studies in real time and optimize 3-D designs for multiple objective functions, the functionality typically available only within high-end analysis applications.

"By participating in this program, ANSYS can deliver its proven high-end technology into the hands of designers, allowing them to perform valuable simulation early in the design process," said Michael Wheeler, vice president and general manager of the ANSYS Inc. Mechanical Business Unit. "As members of the Inventor Certified Application program, ANSYS will realize significant benefits not previously available to us through the MAI program. Additionally, the synergy that has been created between us - two of the most respected software design companies in the industry - will serve both our customers and investors well."

DRD Technology Offers ANSYS – Pro/ENGINEER Courses

DRD Technology Corporation, an engineering consulting firm and ANSYS Support Distributor, has developed two courses for ANSYS users who create ANSYS finite element models using PTC's Pro/MECHANICA in FEM Mode (previously called Pro/MESH) software. In both courses, students will devote most of the time to hands on workshops. DRD keeps both courses up to date with current ANSYS and Pro/E releases, and both courses are currently compatible with ANSYS 6, Pro/ENGINEER 2001, and the corresponding ANSYS - Pro/ENGINEER Interface.

A basic course is intended to enable engineers with rudimentary Pro/ENGINEER skills to develop proficiency at doing finite element analysis. Although this course is suitable for beginning FEA users, it has proven to be useful for experienced analysts who model assemblies of sheetmetal and steel plate parts.

Pro/MECHANICA in FEM Mode is used exclusively to construct finite element models. The course places special emphasis on midsurface creation and shell meshing of parts and assemblies. Additional topics include assembly meshing using Pro/ENGINEER end weld and perimeter weld functions, splitting surfaces, creating regions using datum curves, and definition of lumped mass, beam, spring and spar elements.

A more advanced course focuses on passing Pro/MECHANICA in FEM Mode generated finite element meshes to ANSYS and adding advanced capabilities to the model. Some of the advanced capabilities covered include plasticity, non-linear contact, large deflections, submodeling, geometry transfer and manipulation, transient thermal analysis, and response spectrum (seismic) analysis. Some time is spent introducing the student to the ANSYS Interface for selecting and post-processing.

DRD Technology schedules these courses on a demand basis, and conducts them at the company's Tulsa, OK and Addison, Texas (near Dallas) facilities and also at customer locations. For more information, visit DRD's Web site at www.drd.com, call 918.743.3013, or email at info@drd.com.

ANSYS and ITI TranscenData Announce Automatic Geometry Healing Module

ANSYS Inc. and ITI TranscenData recently launched the ANSYS Automatic Geometry Healing Module with ANSYS 6.1 software. Working in conjunction with the existing ANSYS Connection Products or with additional CADfix-based imports for IGES v5.3, STEP AP 203 and AP214, CATIA v4.x, VDAFS 2.0, STL, ACIS 6.3 and Parasolid 12.1, the ANSYS Automatic Geometry Healing Module allows users to heal and import problematic CAD models for analysis.

The ANSYS Automatic Geometry Healing Module provides ANSYS 6.1 users with a fully automated model import and healing solution designed specifically to meet data exchange needs. Utilizing ITI TranscenData's CADfix geometry repair tool as background technology, the new Healing Module is available to ANSYS users within ANSYS 6.1. Users requiring full CADfix functionality can contact ITI TranscenData for details.

“While ANSYS Connection Products offer direct model importation between the CAD system and ANSYS, users often encounter translation errors that can corrupt geometry files,” said Ed Wagner, vice president of marketing services for ANSYS Inc. “ITI has always offered superior geometry healing technology, and we are pleased to integrate this technology into our ANSYS 6.1 solutions.”

“Like all downstream product development applications, analysis is dependent on the uninterrupted access to clean, usable CAD data,” said Tom Makoski, vice president of sales and marketing for ITI TranscenData. “The ANSYS Automatic Geometry Healing Module provides ANSYS users with fast and reliable solution for analyzing problematic CAD models.”

ANSYS Inc. will be responsible for sales, marketing and customer support of this product. Those interested in the ANSYS Automatic Geometry Healing Module should contact ANSYS Inc. or their local ANSYS sales partner.

“The ANSYS Automatic Geometry Healing Module provides ANSYS users with fast and reliable solution for analyzing problematic CAD models.”

Cart3D CO-SELECTED AS NASA's "2002 SOFTWARE OF THE YEAR" ANSYS' ICEM CFD Cart3D Software Available for Commercial Distribution

ANSYS Inc. integrated NASA AMES' Cart3D product, recently recognized as NASA's "2002 Software of the Year," into its ICEM CFD product suite for commercial distribution. Cart3D is a high-fidelity, inviscid analysis package for external and internal flow simulations developed by NASA Ames Research Center and New York University for conceptual and preliminary aerodynamic design.

ANSYS has integrated Cart3D v1.1 as an addition to its ICEM CFD product suite, enabling the package to be licensed commercially as ICEM CFD Cart3D in five industries. In addition to its exclusive distributor status for NASA Ames' Cart3D solution to the aerospace, automotive and turbo machinery industries, ANSYS also has distribution rights for the electronics/electromagnetics and process industries. Moving forward, for non-government users, the Cart3D product only can be licensed through ANSYS or its authorized channel partners.

"Cart3D's novel algorithms and its state-of-the-art computational efficiency combine to provide designers with a new level of automation that reduces simulation time requirements by a factor of at least 250," said NASA Ames' Michael Aftosmis, one of the co-developers of Cart3D. "This level of automation encourages designers to test more vehicle variations over a wider range of flight conditions than previously possible." "We are honored to be selected as a NASA Ames partner," said Dr. Armin Wulf, ANSYS vice president and general manager of the environmental business unit. "The NASA Software of the Year co-winner - Cart3D - is the most advanced aerodynamic design tool available to designers and engineers. We are encouraged to offer ICEM CFD Cart3D as a part of our product suite and deliver this technology to a broader user base."

With the commercialization of Cart3D through ANSYS, users are able to receive dedicated support and training, as well as additional functionalities of the ICEM CFD product suite, such as a GUI for problem setup and post-processing, enhanced import of legacy geometry, direct import of CAD geometry, and the ability to automatically setup and run suites of simulations based on parametric changes to models. For more information about ICEM CFD Cart3D, visit: http://www.icemcfd.com/cart3d/index.html

Schedule of Events

December

Autodesk University

Dec 3-6

Las Vegas, Nev.

www.autodesk.com

If you would like your events posted, send a brief description including the name, date, location and registration URL to managing editor Jennifer Hucko at jennifer.hucko@ansys.com.

LMS to hold Conference for Physical and Virtual Prototyping

LMS North America will hold its Second Annual Conference for Physical and Virtual Prototyping on Sept. 18-19, 2002, at Michigan State University's Management Education Center in Troy, Mich. The event provides a forum for discussing trends, issues and applications for virtual prototyping, physical testing and the combination of the two technologies in hybrid engineering.

The keynote address on "The Critical Role of Virtual Prototyping for Innovation, Competitive Product Creation and Validation" will be presented by Dave Burdick, founder of the research and analysis firm Collaborative Visions and former vice president for Gartner Group's Manufacturing Applications Strategies Service Division.

> This approach enables users to understand and solve dynamic problems, turning upfront attribute optimization into a strategic competitive advantage.

"The central question in competitiveness is how corporations can develop and deliver continuous innovation that can be manifest in great products that customers want to buy over and over again. This problem is obviously broad in scope and requires a complete re-think about the processes and technology framework required," says Burdick. "Product lifecycle management (PLM) is a business IT strategy that is focused on addressing the 'big picture' problem of developing and delivering innovative products on a continuous basis. A key strategic cornerstone of PLM is constructing a robust virtual prototyping environment to rapidly create and validate high-quality product innovations."

LMS president and CEO, Dr. Urbain Vandeurzen, will discuss functional performance engineering in refining designs according to key attributes such as durability, vibration, acoustics, and fatigue lifetime. The goal of the approach is for product qualities to be designed in during the early stages of the cycle, refined throughout product development using analysis and optimization, and finally verifying and fine-tuning performance through in-depth testing on a reduced number of physical prototypes. This approach enables users to understand and solve dynamic problems, turning upfront attribute optimization into a strategic competitive advantage.

(continued)

ANSYS and CEO Jim Cashman Recognized as Leaders in the Technology Industry

ANSYS Inc. recently announced that Jim Cashman, president and chief executive officer of ANSYS was named CEO of the Year by the Pittsburgh Technology Council. ANSYS also was named to the Council's Sixth Annual Technology 50 list of the most successful companies in the region.

Companies are listed in five industry categories, and those that are named to the list demonstrate success in five key areas: advancement in product development or sales success; financial strength; corporate citizenship; job growth/retention; and innovation. ANSYS was recognized in the Information Technology category at a ceremony on Thursday, Sept. 26, 2002, at the Carnegie Museum in Pittsburgh, Pennsylvania. The Technology Council award to ANSYS comes on the heels of the company's listing in the ***Business 2.0*** Top 100 list, also announced in September.

"The Pittsburgh Technology 50 is a celebration of regional innovation and entrepreneurial successes, and it shines a spotlight on those leading the charge," said Steven G. Zylstra, president and CEO of the Pittsburgh Technology Council. "Jim Cashman embodies the types of attributes we were looking for in a CEO of the Year, particularly as it related to the quality of his leadership, his ethical standards, his focus on team-building and other characteristics that make a great leader."

"The Council's recognition endorses the ANSYS team's talent for implementing unique business strategies while offering best-in-class technology solutions that make our customers successful," said Jim Cashman, president and chief executive officer of ANSYS Inc. "We applaud the Council for highlighting Pittsburgh's leading businesses, as well as the group's dedication to enhancing the region's technology environment. We are honored to be among the few companies to repeat this recognition."

The Pittsburgh Technology Council determined the 50 fastest-growing firms by researching thousands of public and private technology companies headquartered in several Pittsburgh-area cities and towns. PricewaterhouseCoopers audited the Council's results.

ANSYS Recognized as One of the Fastest-Growing Tech Companies by *Business 2.0* Magazine

Further strengthening its reputation as a solid company, ANSYS Inc. announced it has been named to ***Business 2.0***'s premiere B2 100, a list of the fastest-growing tech companies. The list, compiled by Zacks Investment Research, included a review of 2,000 publicly traded companies.

"Outfits that have improved their financial strength during the grueling past three years have something to teach. The B2 100 firms are eloquent testimony to the power of innovation, focus, financial discipline, and a handful of other virtues, even in this dismal economy," editor-at-large Erick Schonfeld writes in the October issue of ***Business 2.0***.

"This recognition is particularly gratifying because it's a direct reflection of ANSYS' dedication to continually meet and exceed our customers' needs," said Jim Cashman, president and chief executive officer of ANSYS Inc. "We have maintained a strong business during uncertain economic times by consistently delivering leading-edge products that allow our customers to be successful. This recognition in the B2 100 is yet another indicator of the success of our strategy."

To arrive at the final rankings, in which ANSYS placed at number 86, ***Business 2.0*** narrowed the Zacks Investment Research list to those companies with at least three years' history as a public company, $50 million in revenues, and positive cash flow from operations in the most recently reported 12 months. In consultation with finance professors George Foster of Stanford University, Brett Trueman of the University of California at Berkeley, and Chip Ruscher of the University of Arizona, they created a ranking algorithm that combined the 12-month stock market return with revenue, earnings, and operating cash flow growth during the past three years. The first three measures each count for 20 percent of a company's overall ranking, the last for 40 percent.

The Chrysler Group selcts LMS Virtual Lab for Next Generaton, Computer-Aided Engineering Desktop Environment

LMS International announced that the Chrysler Group unit of DaimlerChrysler AG has signed an agreement for products and services, which will significantly improve the Chrysler Group's engineering productivity in the areas of Noise, Vibration & Harshness (NVH), Durability and Impact simulation.

Under the agreement, LMS will provide the Chrysler Group with LMS Virtual.Lab, a comprehensive software environment for functional performance engineering in the conceptual phase of product development. LMS will also team up with Chrysler Group engineers to evaluate the automaker's current processes, remove bottlenecks, increase value-added engineering time and improve the overall quality of designs.

The resulting software system will also integrate the computer-aided design (CAD), computer-aided engineering (CAE), and physical testing environments together for faster response rates to design changes and improved CAE productivity.

According to Dr. Jack Thompson, Director of CAE and Concept Development at the Chrysler Group, "Timely functional performance engineering for attributes like noise, vibration, durability and crash performance is one of the challenges we are tackling in our vehicle development process.

According to Dr. Jack Thompson, Director of CAE and Concept Development at the Chrysler Group, "Timely functional performance engineering for attributes like noise, vibration, durability and crash performance is one of the challenges we are tackling in our vehicle development process. With the help of LMS Virtual.Lab, which is specifically built for cross-disciplinary attribute engineering, we are able to better leverage our proven analytical tools to enhance the virtual design process."

In addition, LMS Virtual.Lab will enable the Chrysler Group to speed up the virtual design process, improve the accuracy and quality of virtual prototype simulations and standardize the types of software being used for NVH, durability and impact simulations. The evolutionary rollout of LMS Virtual.Lab at the Chrysler Group will be a multi-year process, ultimately reaching across all of the company's vehicle platforms and component teams.

"We are very excited about this strategic implementation of LMS Virtual.Lab," Chairman and CEO Urbain Vandeurzen stated. "The Chrysler Group decision is another confirmation of our vision and strategy to combine the strengths of both physical and virtual prototyping to meet the growing demands made on the engineering community to optimize mission critical product attributes and brand values. Our commitment to radical product and process innovation and the superior integration that comes with LMS Virtual.Lab were important factors for LMS to be selected by the Chrysler Group as a strategic CAE partner."

ITI ProjectView: Secure Data Management Collaboration

International TechneGroup Incorporated (ITI) announced the release of ITI ProjectView version 3.0. ITI ProjectView is an environment for collaborative project management through a Web portal providing secure yet seamless access to all types of business, manufacturing, engineering and project data

ITI ProjectView allows organizations to integrate and share data files, project plans, schedules, and other forms of business, product, and process information in a controlled secure yet effective environment. Additionally, users can schedule events or facilitate discussion forums through the site.

Security features include: User Name & Password, Secure Sockets Layer (SSL), Certificate Authentication, and I.P. Filtering. ITI ProjectView is currently available in English, Japanese, and German languages. Support for additional languages can be quickly developed.

"Because product development cycles are now being measured in days and weeks, timeliness and responsiveness are critical to success," explained Mike Lemon, Vice president and General Manager of ITI's Virtual Product Development business. "A major obstacle to greater collaboration is the incompatibility of information technologies used. The Internet has emerged as a practical and cost-effective infrastructure for linking manufacturers with key customers and suppliers," continued Lemon. "ITI ProjectView leverages the power of the Internet into tightly integrated collaborative global virtual enterprises."

To learn more visit http://www.projectview.com or call 513.576.3900 or 800.783.9199.

ANSYS Named to *Forbes* Top 200 Small Companies Listing

Stepping up to receive its third honor in 30 days, ANSYS Inc. recently announced it has been named to the ***Forbes*** 200 Best Small Companies, a list comprised of financially strong small-cap businesses. ANSYS ranked 56 among the 200 leading small companies in the nation.

"Despite a rough economy and other forces that took the NASDAQ to its 6-year low, this year's ***Forbes*** 200 Best Small Companies is a stellar bunch of companies," editor Katrina Keller writes in the October issue of ***Forbes***. "All of these companies are profitable, although some of them have experienced rough waters over the past few years, such as a slip in sales or earnings, their balance sheets indicate they have the muscle to remain viable entities over the long term. Each company on this list has earned its place and is, indeed, good enough to be called one of the best."

The list, compiled by William O'Neill & Co., Los Angeles, selected companies based on five key criteria. "The criteria we use to screen candidates is tough," said Lesley Kump, ***Forbes*** statistician. "We looked only at public companies with latest-12-month sales of $5 million to $600 million, a stock price above $5 per share and five-year average return on equity of at least 5 percent. Growth notwithstanding, profitability was a must - every nominee had to earn a net margin of at least 5 percent, excluding extraordinary and nonrecurring items."

"Achieving this award during one of the most difficult economic periods in the past ten years is a tribute to the ANSYS team, working every day to provide outstanding products, services and support for all of our customers," said Jim Cashman, president and chief executive officer of ANSYS. "Helping our customers succeed in business has been the key to ANSYS' success."

New to the ***Forbes*** 200 list this year was a review of relative stock price strength. The addition of this criteria enabled the reviewers to note companies that were doing particularly well or poorly relative to their peers and the market indexes.

THE PREMIER E-ZINE FOR DESIGN INNOVATION
SOLUTIONS

For ANSYS Inc. sales information, call **1.866.ANSYS.AI**, or on the Internet visit www.ansys.com.

Editorial Director
John Krouse
jkrouse@compuserve.com

Managing Editor
Jennifer L. Hucko
jennifer.hucko@ansys.com

Designers
Miller Creative Group
info@millercreativegroup.com

Art Director
Paul DiMieri
paul.dimieri@ansys.com

Ad Sales Manager
Ann Stanton
ann.stanton@ansys.com

Circulation Manager
Elaine Travers
elaine.travers@ansys.com

Editorial Advisor
Lynn Rowles
lynn.rowles@ansys.com

Subscriptions to ANSYS SOLUTIONS are free to qualified recipients. To continue to receive this publication, subscribe online at www.ansys.com/subscribe.