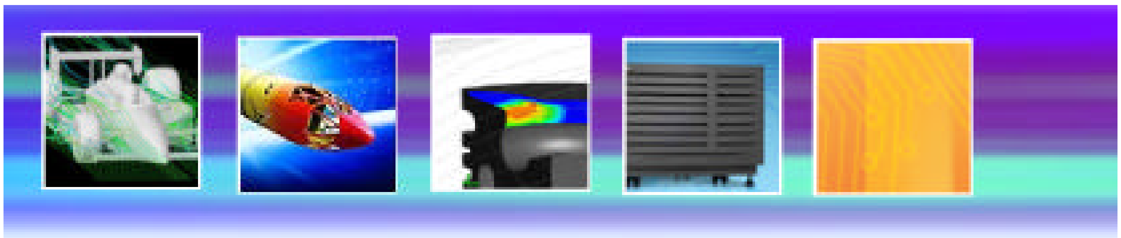
ANSYS SOLUTIONS

THE PREMIER E-ZINE FOR DESIGN INNOVATION

Fall 2003



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

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

Announced in 1996 to promote the development and flight of spacecraft for low-cost commercial transport of humans into space, the international X-Prize Foundation is providing a purse of US$10 million to the first competitor who can safely launch and land a manned spacecraft to an altitude of 100 kilometers (the international border of space), twice in a two-week period.

The first Canadian entry in this competition, the fully volunteer da Vinci Project (a wholly owned by ORVA Space Corp.) has put years of engineering research, design and developmental testing into the vehicle design, propulsion and flight guidance system.

A full-scale flight-engineering prototype of the manned rocket has been constructed. Detailed engineering and fabrication of the full-scaled manned rocket named Wild Fire Mk VI is currently underway. Flight-testing of the manned rocket and X-Prize competition flights are targeted to continue throughout 2004.

For R&D efforts on the project, a wide range of engineering software was utilized for CAD, basic engineering calculations, trajectory analysis, dynamics and mission control, supersonic external aerodynamics, and internal heat flow. Part 1 of this article series appearing in the last issue of ANSYS Solutions covered the use of ANSYS software for thermal analysis of the thermal protection system and on integration with CAD, MCAD and CAE software. This installment describes how ANSYS Classic and ANSYS Workbench/ DesignSpace were utilized to perform stress analysis of the rocket block.

Integrated Vehicle FEA Analysis with ANSYS

On May 13, 2003, the da Vinci project announced Kindersley, Saskatchewan (www.kindersley.ca) (see Figure 1a) as the launch site for Wildfire rocket-balloon. At the same time, our team (see Figure 1b) was finalizing flight mission profile and flight safety at its Toronto, Ontario research center located at the da Vinci Polytechnic Institute.

Figure 1a. Aerial view Kindersley airport, Saskatchewan

Figure 1b. Stress Analysis presentation at the Da Vinci Polytechnic Institute

ANSYS technology was widely utilized throughout all stages of the project and included external and internal loading of the space capsule, landing impact loading, parachute deployment, rocket block ground handling, vibration analysis and static and dynamic inertial loading of the flight hardware. Aeroheating, aerodynamic and capsule stress analysis were described in the paper presented at the third EADS International Re-entry Vehicle Symposium in Archachon, France March, 2003 (http://www.davinciproject.com/beta/Technical/TechnicalMain.html). In the present article, we will describe selected aspects of the rocket block inertial stress analysis.

The general outline of the rocket block (global model) structural skeleton is given on Figures 2a and 5a,b and also at http://206.210.04.185/fea_davinvi_text.html. Our main objective was to build integrated 3d computer prototype (global model), which could be utilized for the truss structure design and evaluation.

Our approach was implemented in several stages: we build separate vertical and angled truss models using ANSYS Classic 7.0 and ANSYS Workbench 7.0 interfaces, we compared results and developed good physical understanding of the stress distribution in the fundamental elements of the global model. We then utilized buckling theory to estimate buckling loads and we established truss thickness on the basis of load safety analysis. We then moved towards integrated rocket block FEA.

Our first bold attempt was to work with ANSYS Design Space/Workbench 7.0 to directly utilize bi-directional associatively with AutoDesk Inventor 7 or use SAT plug-in interface. However, we quickly realized that because of high truss thickness to length aspect ratio (typical for our structure) it is impractical to build accurate grids on any available PC hardware.

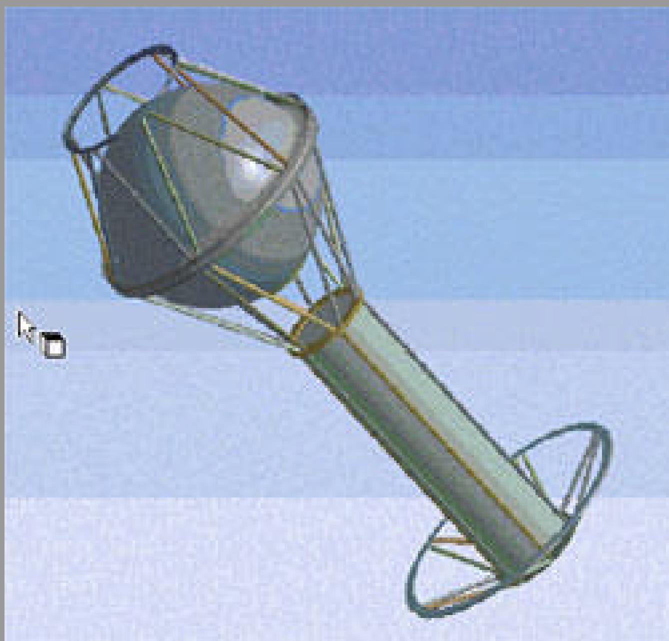


Figure 2a. Rocket block outline shown in ANSYS Workbench and DesignSpace 7.0 (shell is suppressed)

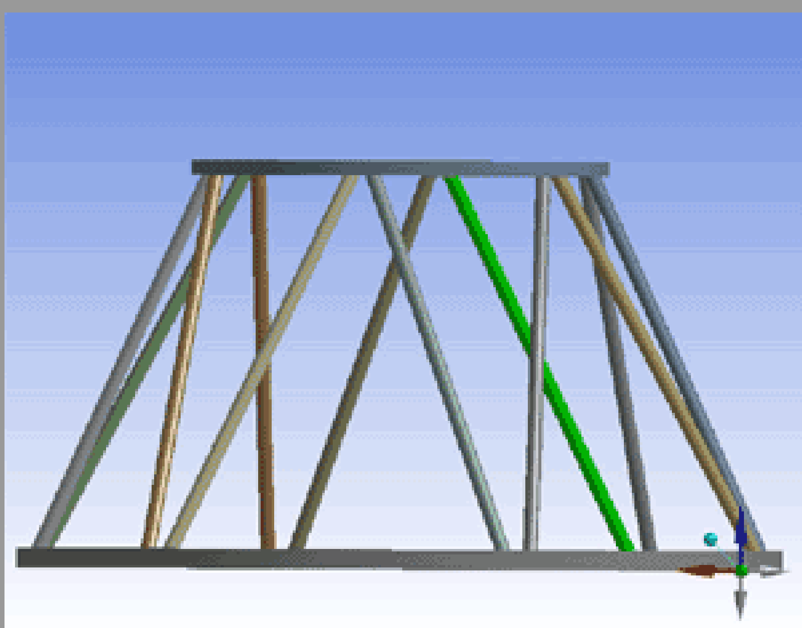


Figure 2b. Upper truss structure imported in ANSYS Workbench

We were working with desktop P4 class PCs and both memory and speed were not sufficient to convert and solve the entire geometry. Thus, we decided to utilize ANSYS Classic interface and to build parametrically controllable grid on a fixed geometry template and to use Workbench for the analysis of the individual components (latches, locks, bolts, individual trusses).

Our rocket block analyses were split into three phases. In the first phase we performed parametric studies of loaded trusses of various thicknesses, lengths and inclinations using both ANSYS Workbench and ANSYS Classic (see for example Figure 12b). In the second we worked with ANSYS Classic beam and shell elements (Figures 7a,b) and estimated moments, forces and deformations (Table 1, Figures a,b). These results were subsequently used to perform truss buckling analysis (Table 2). During the third phase we build 3D brick model of the truss structure. Shell elements were utilized for the aero-shell and inertial loading (from the space capsule and fuel tank) was estimated and applied at the appropriate locations. Parametric 30 deg (1/12th of the model) grid was build and then rotated around the circumference. We utilized full 360 deg. geometry to account for possible three-dimensional loading. However in some cases a half-model can be sufficient.

Our grid size estimates and first analysis (utilizing 3d brick model) quickly demonstrated limitations of Pentium4 PC hardware available to us, many junctions were gridded with insufficient resolution and demonstrated staggered patterns, especially visible on element solution outputs (see Figures 4 a,b). RAM memory limitations, memory management model and overall bus throughput were clear show-stoppers. We evaluated alternative available hardware platforms and approached Sun Microsystems Canada to provide us with its scalable Sun Blade Server technology, either stand-alone or a cluster of two SunBlade 2000's dual-processor workstations. It features a high-performance, crossbar-switch system interconnect that provides high bandwidth (up to 4 GBps) for ultra-high-speed processors and graphic subsystems, running on Solaris 9. Each workstation has 8Gb of shared RAM, with two workstations offering 16GB and Server offering 32GB. System is powered by 64-bit 1200-MHz UltraSPARC III Cu, 8-MB external cache per CPU. ANSYS provides superior parallel scalability in both memory sharing (AMG solver) and distributed memory (DDS solver) regimes, so our engineering department can effectively implement flexible processor loading. Our subsequent experience demonstrated great potential of scalable shared memory servers when coupled with algebraic multi-grid (AMG) parallel solvers (Solvers International and ANSYS).

Figure 3a. Blade 2000 Workstation

Sun Microsystems entered into a technology partnership agreement with da Vinci Space Project which made it possible for us to utilize Sun Microsystems hardware and to build much more accurate three-dimensional models. In this article we demonstrate work-in-progress results obtained on Sun Blade 2000 (See Figures 3a,b).

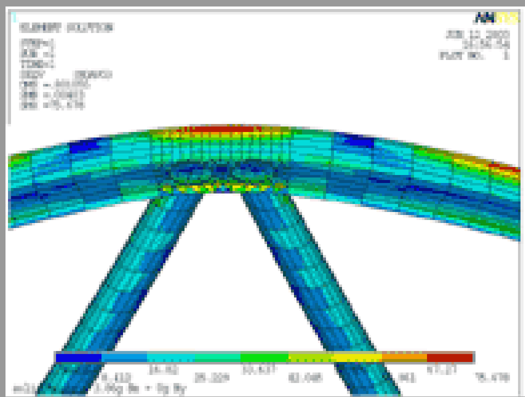


Figure 4a. Initial simulation results using PC, iteration 1 (60K model)

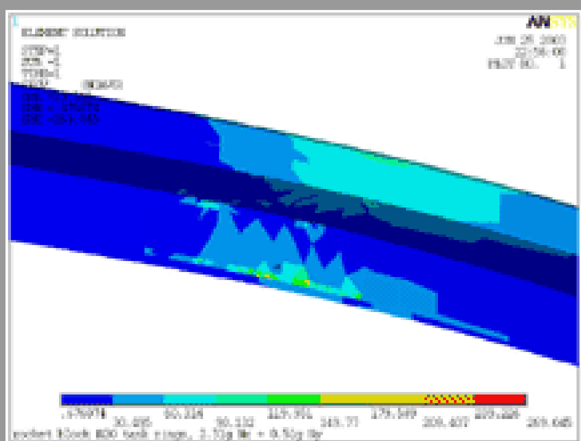


Figure 4b. Initial simulation results using PC, iteration 2 (130K model)

Rocket Block Loading Conditions and Constraints

Inertial and aerodynamic load variation was estimated using three-dimensional trajectory analysis and was implemented on Mathworks Matlab 6.5/Simulink 13 and Maple 8 Mathcad software platforms. Sample trajectory results were demonstrated in the EADS paper and were shown (axial and lateral load components) in the Part I of this article. The da Vinci Wildfire rocket is launched from a giant helium balloon, so it will be difficult to secure precise vertical launch angle, thus a window of +- 30 degrees has been considered, which can result in significant variation in lateral loads and poses serious challenge to ensure safe structural design. Truss and shell structure also needs to be rigid to secure that bend curvature angle is kept below 0.5 deg. Increase in lateral loads will contribute to lateral distortions and associated bending moments on the structural components. Global model presents many challenges, first among many is to select proper constraints and zero displacement boundary conditions to fix structure statically. Rockets and airplanes do not really have restriction points, they sort of 'hang' in the air. Airplanes are in static balance during the horizontal flight and rockets move with acceleration, thus experiencing inertial loads.

ANSYS offers two types of restraints. Conventional are being used for static systems. An equivalent free body analysis is performed if static analysis and inertial relief are used. This is a technique in which the applied forces and the torques are balanced by inertial forces induced by an acceleration field. Displacement constraints on the structure should be only those necessary to prevent rigid body motions. We utilized both approaches and performed free body analysis manually to estimate equivalent inertial loading (see Figure 5b) and this was build-in our flight simulator software. Axial loading stress diagrams showed that compression load (-) (inertial mass force) is relieved into tension (+) as we traverse axially into the internally pressurized engine chamber. For static analysis we fixed inner portion of the engine ring thus limiting its axial and radial displacements. All other rocket block components were allowed to deform as appropriate.

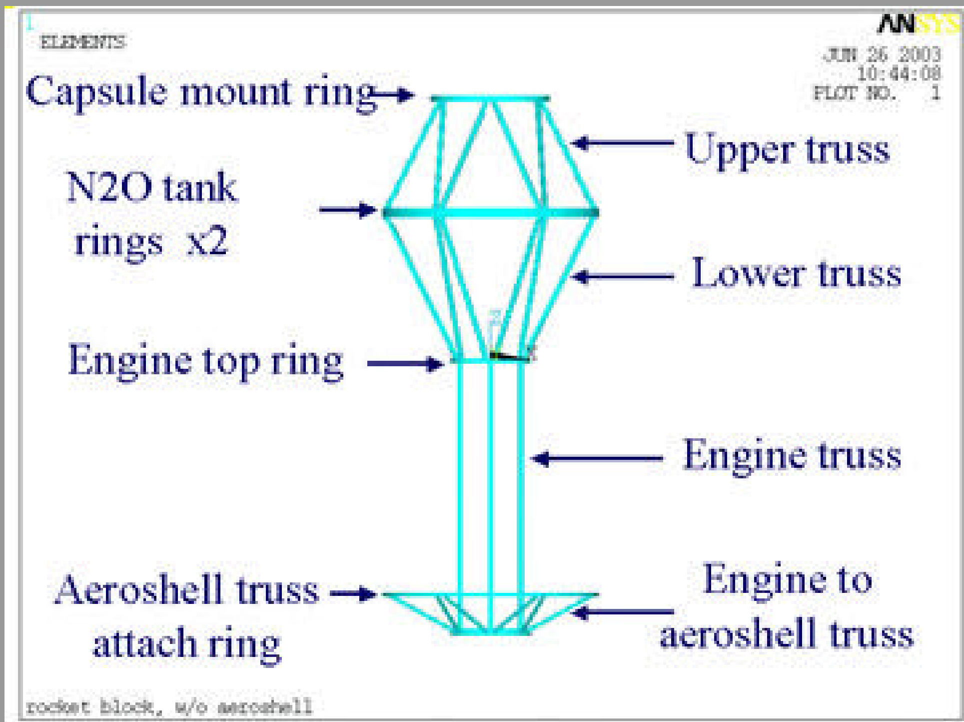


Figure 5a. Functional elements of the structure

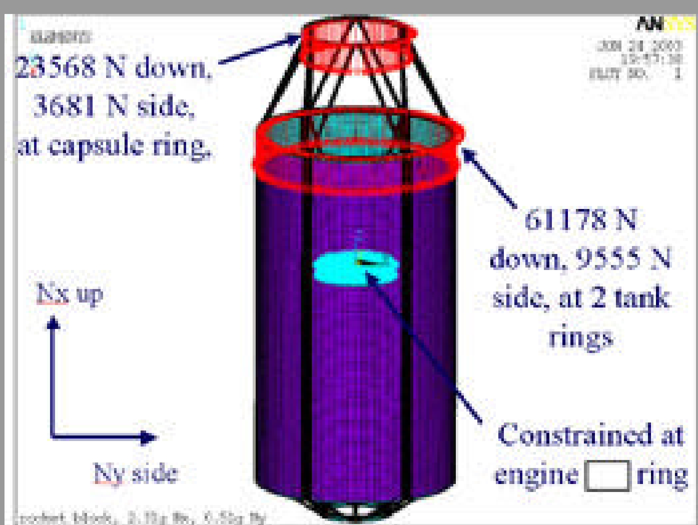


Figure 5b. Loads on the rocket block

We evaluated several different trajectories and several characteristic time moments including engine launch, engine cut-off, and maximum dynamic loads. In this article we present on-design lateral load scenario that takes place during 15 deg off-vertical launch and presumes that reactive control system (RCS) returns vehicle to a vertical flight profile. We considered 2.31g N_x (axial) and 0.5176g N_y (lateral) at trajectory time of 2 sec from the engine launch.

This leads to the following boundary conditions (see Figure 5b). Force applied from the space capsule equals to 7112N x(2.3139+1)=23568 N is directed axially down, and laterally 7112 N x 0.5176=3681 N (see Figure 5b) . Forces from the N2O fuel tank are applied at cap mounting rings (3300+15161 N(N2O fuel))x3.3139=61178 N and are directed axially down, (3300+ 15161)N x0.5176=9555 N in lateral direction, at tank ring. In our analysis we assumed 4130 steel properties, truss O.D. =38.1 mm, thickness =1.6 for upper and lower trusses, O.D. =25.4 mm, thickness = 1.2446 for engine to aero-shell trusses.

Integrated Truss Beam Element and Shell Model

In the preliminary design phase, a simple beam type element (BEAM4) model was build to simulate all truss tubes and rings structures, and shell type element (SHELL 63) was implemented to simulate aero-shell structure. It had 6300 elements and took 1.5 min to run on P4 PC computer. The main objective was to obtain the axial forces and bending moments in tubes and rings. This information can then be used for the design of the appropriate sections. Maximum stresses can be re-estimated using detailed brick element global model. Obviously, the tubes were under compression forces and experienced end moments. Thus trusses must resist compression buckling and flexural bending at the same time. Therefore tubes should be designed and evaluated as beam-column structures. Required design loads can be easily obtained from the beam element type. The appropriate design formulas can be found in steel structures design code or textbooks, see for example Bruhn, Analysis and Design of Flight Vehicle Structures, Chap. C8 and Handbook of Steel construction (p1-27). For a beam-column, under compression and end moments (internal loads), the secondary bending is considered. The section design is based on the interaction equation:

$$f_a/F_a + C_m\ C_a\ f_b/F_b < 1$$ (from Canadian code)

F_a : buckling strength

f_a : compressive stress

F_b : bending strength

f_b : bending stress

C_m : reduction factor

C_a : magnification factor

F_a = compressive stress = P/A

F_a = buckling allowable strength = _AFy(1+_^{2n})$^{-1/n}$

Where : _ =0.9, n=1.34, r=(I/A)$^{1/2}$$\lambda$= (KL/r) (Fy/2E)$^{1/2}$

C_m = 0.6-0.4(M1/M2)_ >= 0.4, if no transverse loads between both ends, M1/M2 positive for opposite directions

C_m = 0.85 if both ends hinged, with transverse loads

C_m = 1.0 if both ends fixed, with transverse loads

Fe´ = Euler buckling stress = π2E/(KL/r)1/2Fb = allowable bending strength = 0.9 Fy for tube.

Resulting beam element model safety factors are presented in the Table 2. They point that lower truss and engine aero-shell truss are weakest links in the present designs. Additional effort is required to modify truss thicknesses and junction design to ensure safety factor uniformity. This can be accomplished utilizing new ANSYS 7.1 variational technology capability or through manual iterations.

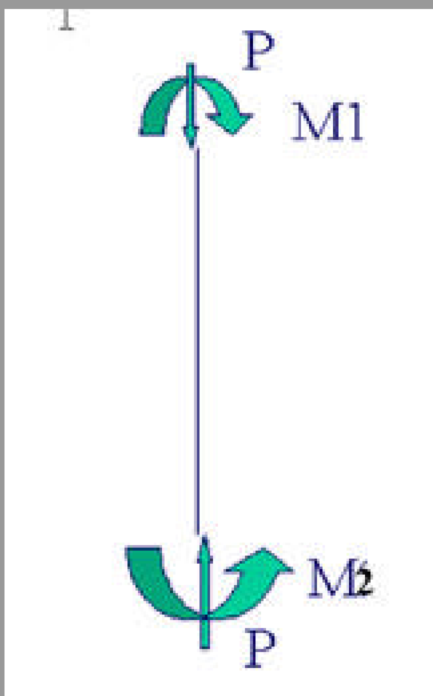


Figure 6. Beam column P & M

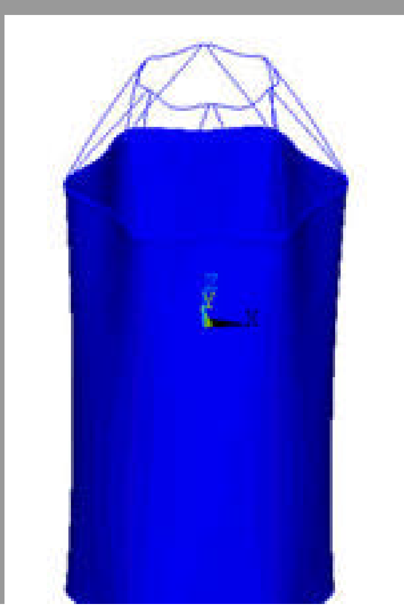


Figure 7a. Axial load 3.86g

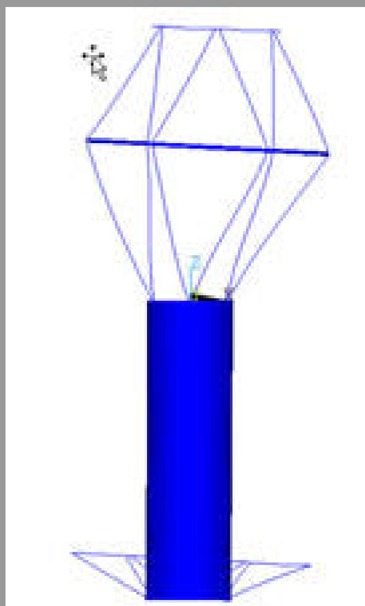


Figure 7b. 2.31g axial, 0.51 lateral

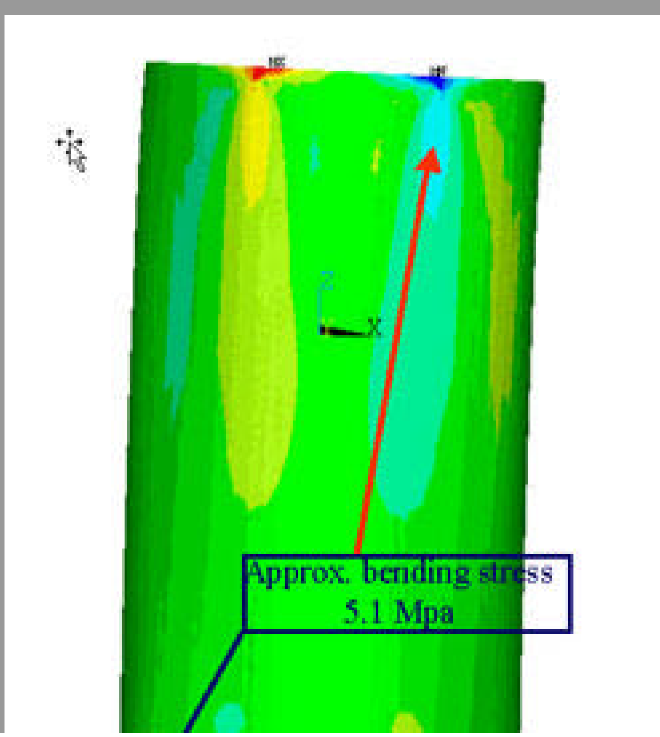
After tube and ring thickness were established detailed brick element model was created to simulate truss tubes and rings. Aero-shell was still modeled utilizing shell elements. Because of high stresses at near tube and ring junctures beam element stress results are not very accurate in these areas. Therefore a brick element (8-node SOLID45) model is required to comprehensively evaluate safety of proposed structural design.

Table 1

Forces and Moments Table					
Case 1: 3.86g Nx	axial force	My -I	Mz -I	My -j	Mz -j
upper truss	-3224	3472	-17094	-4423	11916
lower truss	-4176	-6938	-4196	7903	2751
engine to aeroshell truss	-873	11102	8262	-176	-7003
capsule mount ring	-2861	68831	207130		

N2O tank ring	1787	-242930	4801		
aeroshell attach ring	856	-107910	5444		
case : 2.31g Nx 0.51g	axial force	My -I	Mz -I	My -j	Mz -j
upper truss	-3888	-3158	8912	-8207	-12935
lower truss					
engine to aeroshell truss	-3066	273	24154	39960	28859
capsule mount ring	-2467	61969	-161694		
N2O tank ring	3349	-529720	21369		
aeroshell attach ring	3672	-388460	17713		

Table 1 data summarizes moments and force components in all key system elements and were extracted from ANSYS visual outputs (see Figure 8a, lower truss and Figure 8b for the aero-shell).



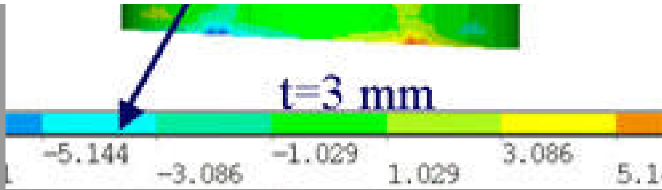


Figure 8b. Aeroshell bending stress

Table 2

Safety Factors for each item			
Items	Safety Factor I	Safety Factor II	Safety Factor III
upper truss	10.9	4.1	4.1
lower truss	1.56	1.3	1.3
engine to aeroshell truss	2.59	2	2
capsule mount ring		2.3	2.3
N2O tank ring		1.2	1.2
aeroshell attach ring		-1.4	1.4
aeroshell (3 mm thick)	7.4	2.7	2.7

Safety Factor I is based on manual buckling analysis done by Canadian code. Safety Factor 2 is based on 130K element brick model assuming the yield strength of 500Mpa. All estimations included load factor uncertainty of 1.5. From the results presented in the Table 2 (both beam and brick models) we can conclude that present design exhibit considerable non-uniformity, safety factors on several elements are close to the threshold and that buckling is not likely to occur.

Brick Model

Brick model implementation survived three iterations and we are presently working on iterations four and five. First two iterations were 60,000 (Figure 4a) and 120,000 (Figure 4b) elements in size. However, due to machine speed and memory limitations resolution near tube/ring junctions in high stress areas was not sufficient (Figure 4a). So refined brick element model (third iteration) was build utilizing ANSYS 7.1 release and was run on SunBlade 2000 workstation. Model included 348,000 elements (Figure 9a,b) and was tested using iterative (PCG), direct sparse matrix and iterative AMG (parallel) solvers. Single processor direct sparse solver took 35 minutes to run, iterative PCG took 90 minutes. Total RAM memory allocation was equal to 2.6GB (3.47Gb for AMG solver). ANSYS estimated following rating for Sun Blade: 161 MIPS, 125 Scalar MFLOPS, 188 Vector MFLOPS (ANSYS defaults are 80/20/40). Using ANSYS runtime statistics we estimated that 550,000 element model will require approximately 4GB of RAM and with memory saving options (MSAVE) we can run 1 million element model on Sun Blade 2000 workstation. If we link two workstations together we can increase total memory pull to 16GB and run it in 2-3 hours in distributed domain mode.

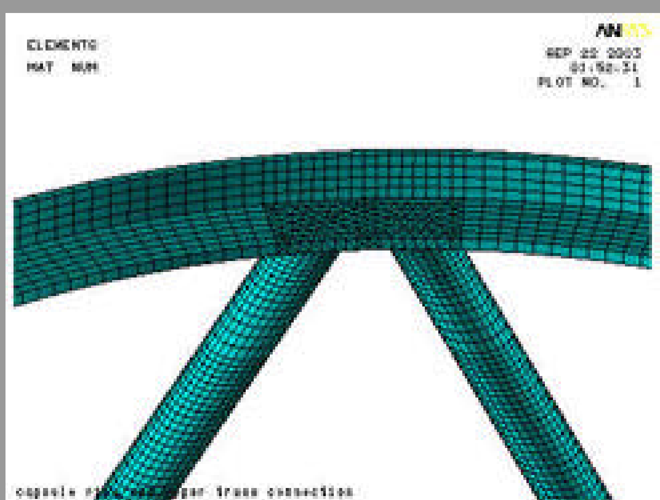


Figure 9a. Capsule ring, 348K element model

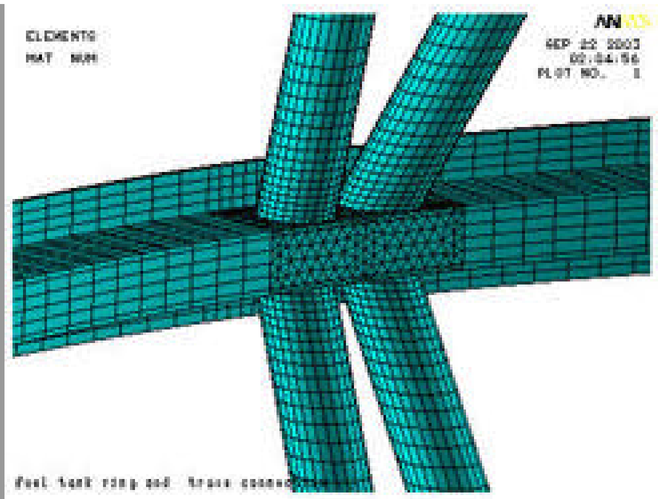


Figure 9b. N2O tank ring, 348K element model

Iterative PCG (1.e-6 accuracy) and direct sparse solver that demonstrated very close stress and deformation results. Lateral deformation (bending) of the structure is very important limitation as it ultimately leads to engine trust misalignment (Figure 11a) and can lead to the failure trajectory shown in Figure 10a. Summary of trajectory analysis (body rotation around its center mass) are presented on Figure 10b. They clearly demonstrate that allowable misalignment range is within 0.5 degree. Figure 11a shows distribution of RB lateral deflection, with maximum equal to 18 mm (0.018 m). Same figure also demonstrates two approaches to estimate angle vs. deformation relationship. Nonlinear estimation takes into the account spatial deformation curvature and requires lateral deformation to be within 11 mm (0.11 m). This suggests further need to increase structural stiffness of the rocket block despite the fact that entire design has safety factors above unity (see Table 2). Figures 11b,c also demonstrate deformed shape (top view) and side view. Considerable lateral deformation takes place, especially in the upper truss/capsule ring side. Von Mises stress distributions at several characteristic truss and ring locations are shown in Figures 12a,b,c,d,e and Figures 13 a,b,c,d. Figures 12a,b compare stress loads calculated using 130K element model and 348K element model (252/242 MPa respectively). On Figure 12e we show Workbench results that were obtained using simplified stand-alone truss model (239 MPa).

These are highest stress areas of the rocket block, and all models demonstrate close results, however higher resolution model provides much better quality field data with smooth gradient transition. On Figures 12d,e we show stress distribution at the N2O ring-to-truss junction location (truss is not shown). Figure 12e shows the backside of Figure 12d. Same location is exemplified on Figure 4b (130K element PC based model). Notice considerable improvement in the stress distribution smoothness and considerable reduction in maximum stress on the ring, i.e. 269 MPa (old model) and 150.39 MPa in the new model. We can also notice that peak stress (in old model) was located near the lower truss attachment point. Upon closer review of the same location (Fig 12e) we notice that ring stress at the truss attachment point is equal to 97MPa, i.e. it is almost 2.5 times smaller. Such finding will be further investigated and to be confirmed using local grid dependency study, however it clearly shows considerable potential design benefits and radical improvement in solution quality and accuracy. Smaller stress will lead to thinner and lighter parts or to higher safety factor for the given component.

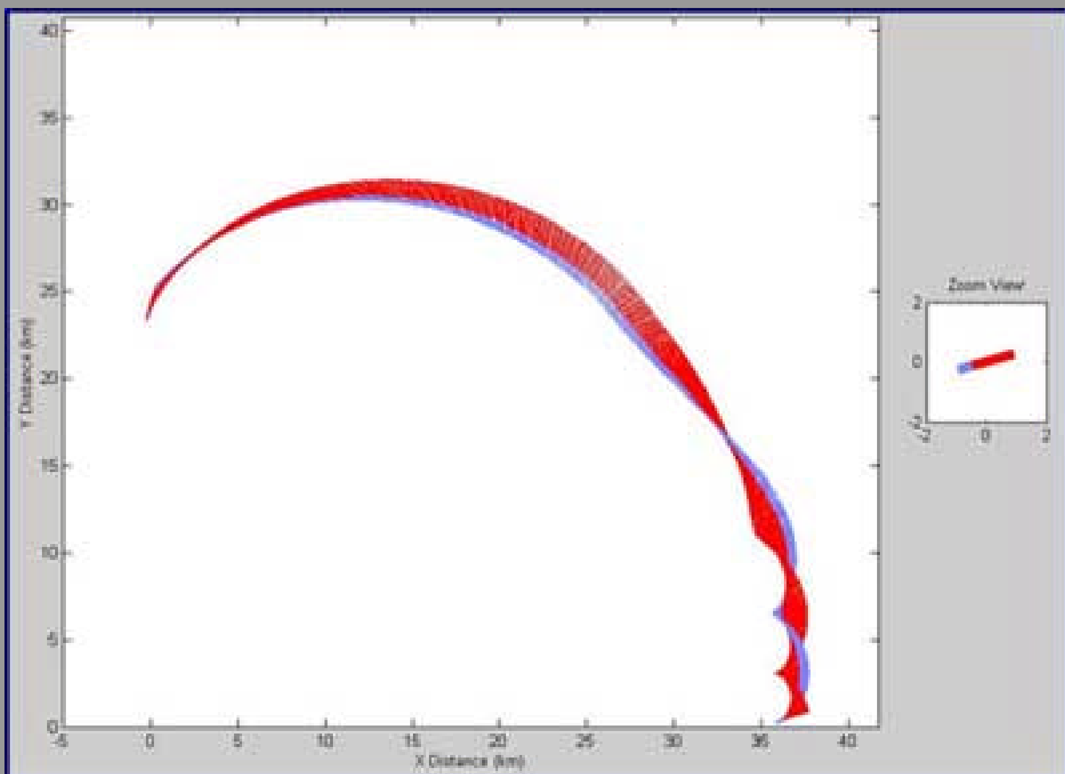


Figure 10a. Trajectory simulation with 1deg engine trust vector misalignment

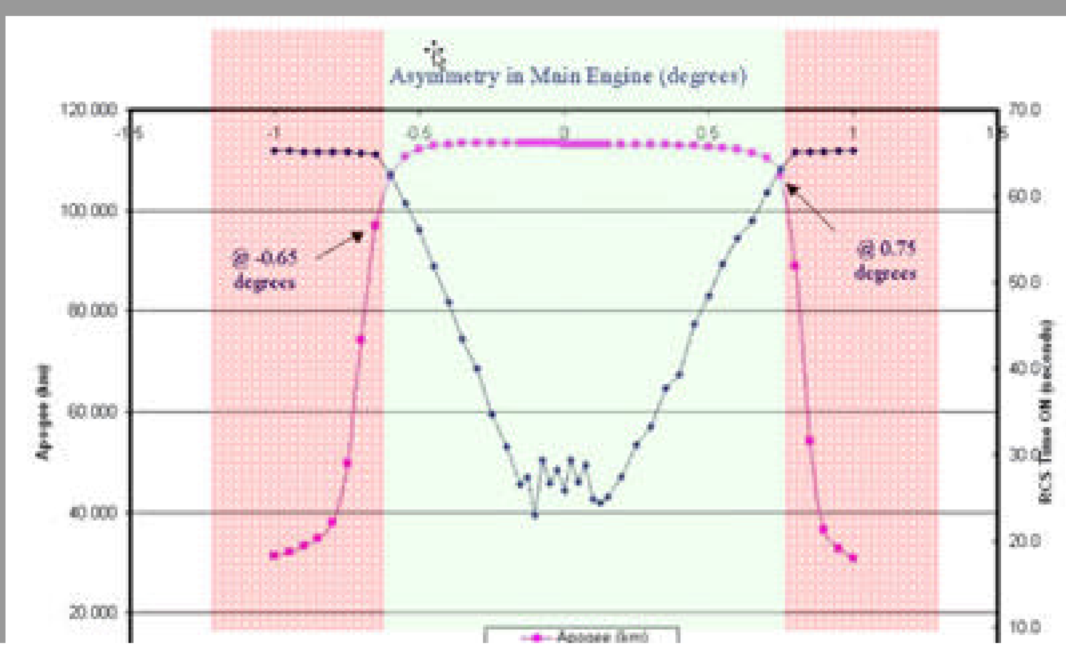


Figure 10b. Applicable misalignment range

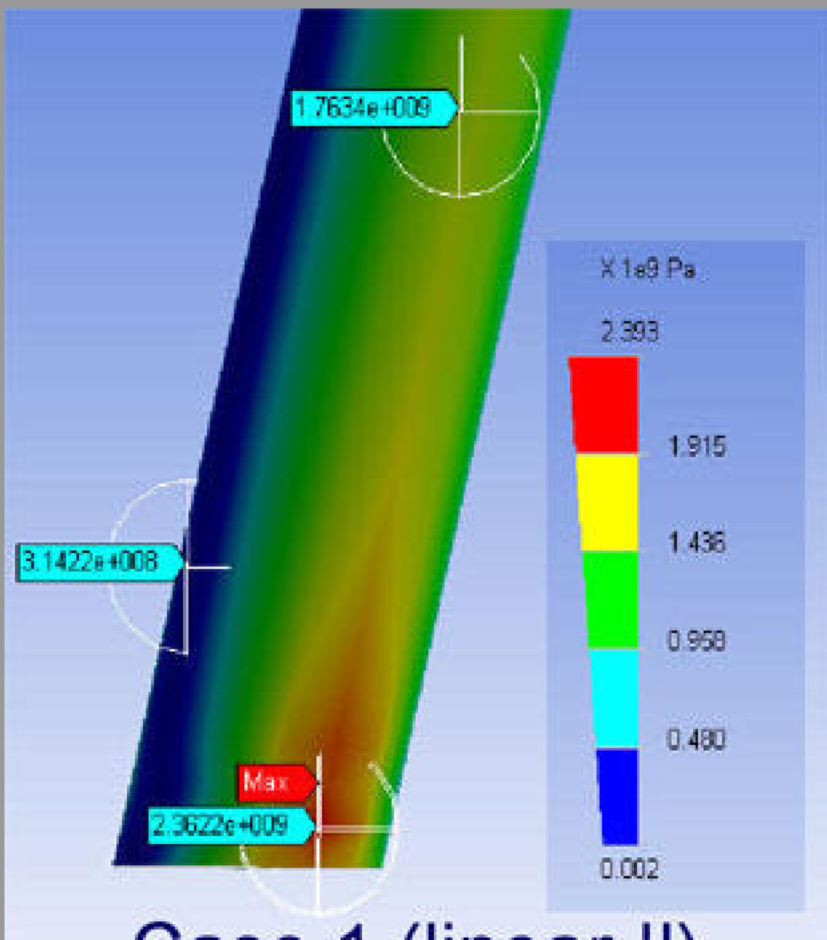


Figure 12b. Lower Truss Truss/ring junction (375K model)

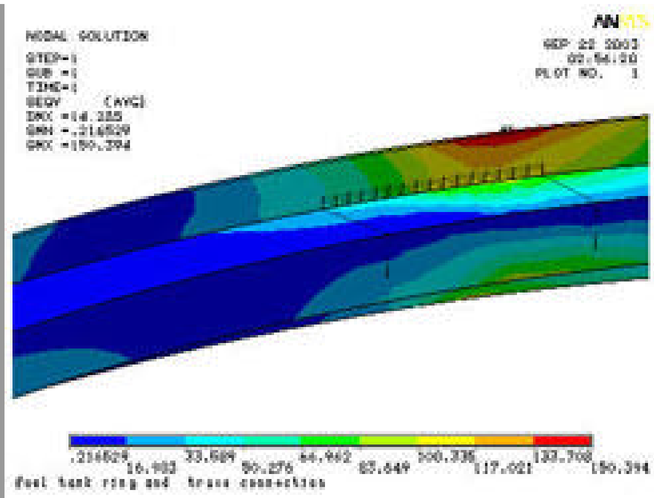


Figure 12c. Truss results analysis using Workbench, maximum stress 239 MPa

We are concurrently working on the 4th and 5th generation brick models which will consist of 700,000 and 1,000,000 elements and will be implemented on Sun Blade 2000. Fourth generation model will utilize shared memory (single node, 2 processor) AMG implementation and is expected to consume 7.5GB of RAM and fifth generation model will use over 10 GB of Ram and will be implemented utilizing ANSYS DDS distributed domain solver.

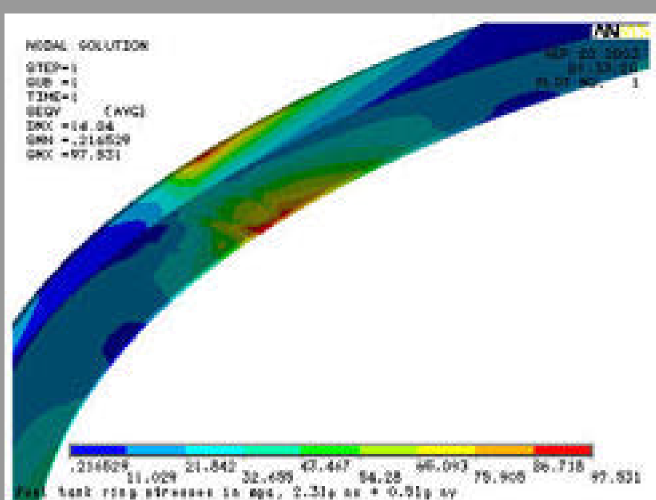


Figure 12d. Ring model details

Additional examples of von Mises stress distributions in various key system components and assemblies are shown in Figure 13a,b,c,d.

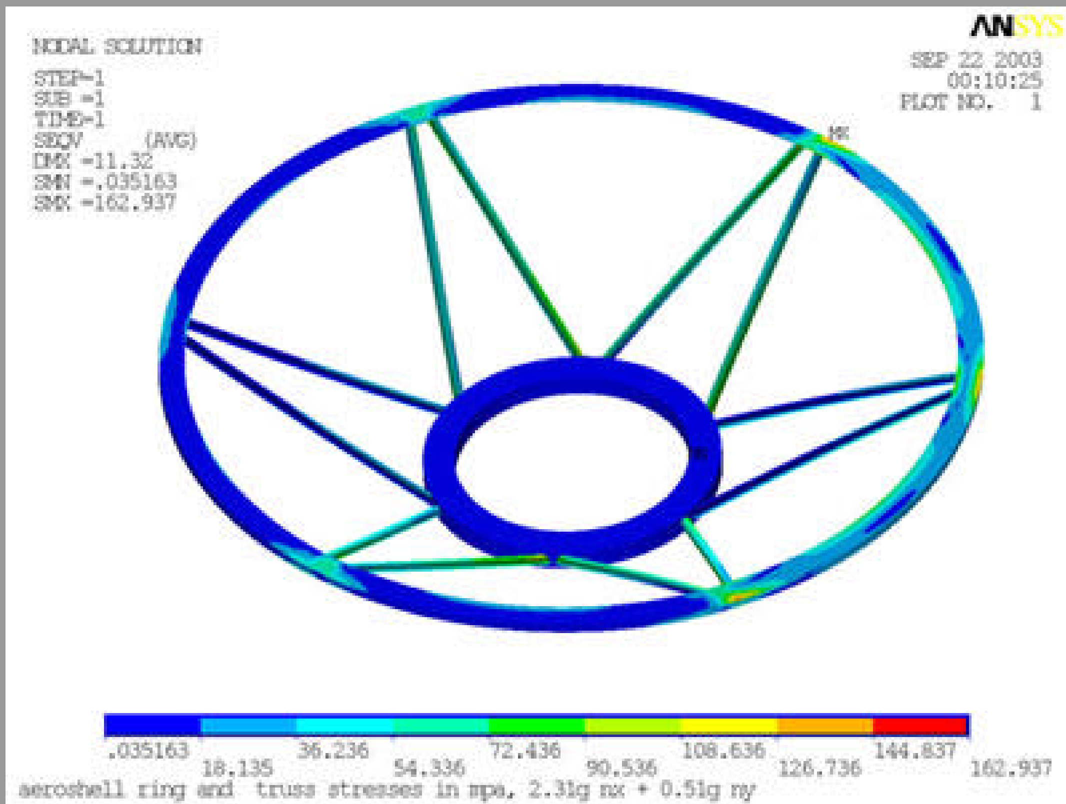


Figure 13 a. Engine to Aero-shell Truss and Aero-shell attachment ring, maximum stress 162 MPa

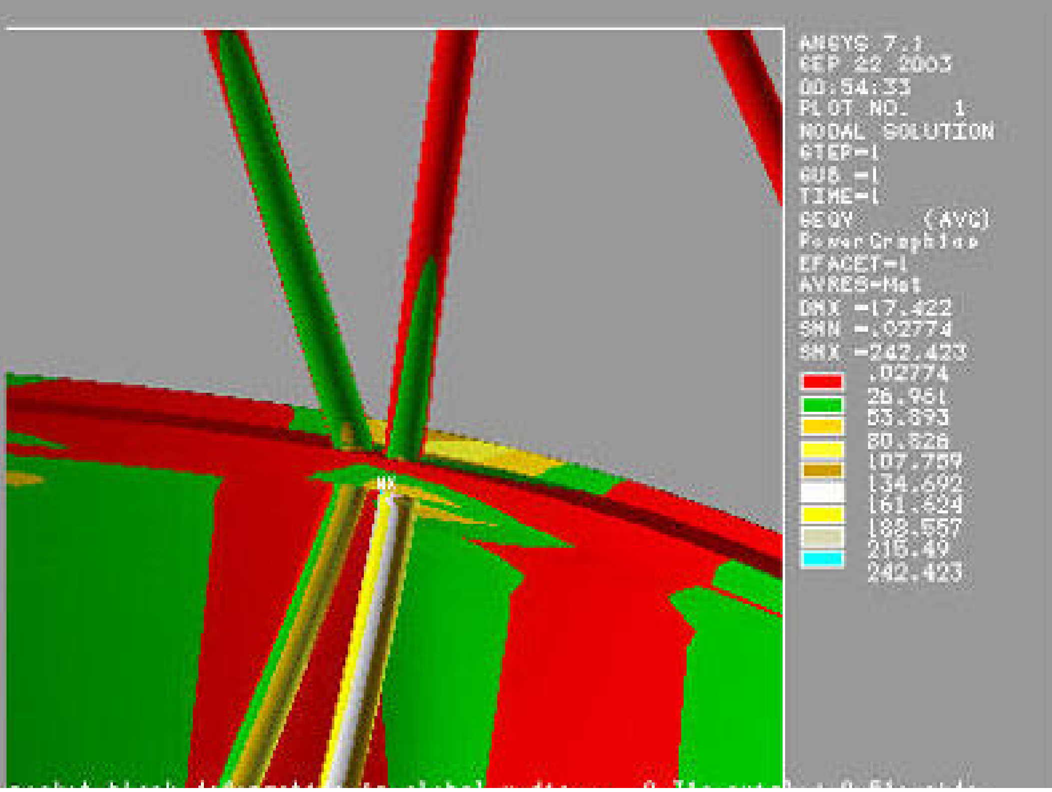


Figure 13 b. Stress distribution in truss/ring/shell junction, maximum stress 242 MPa

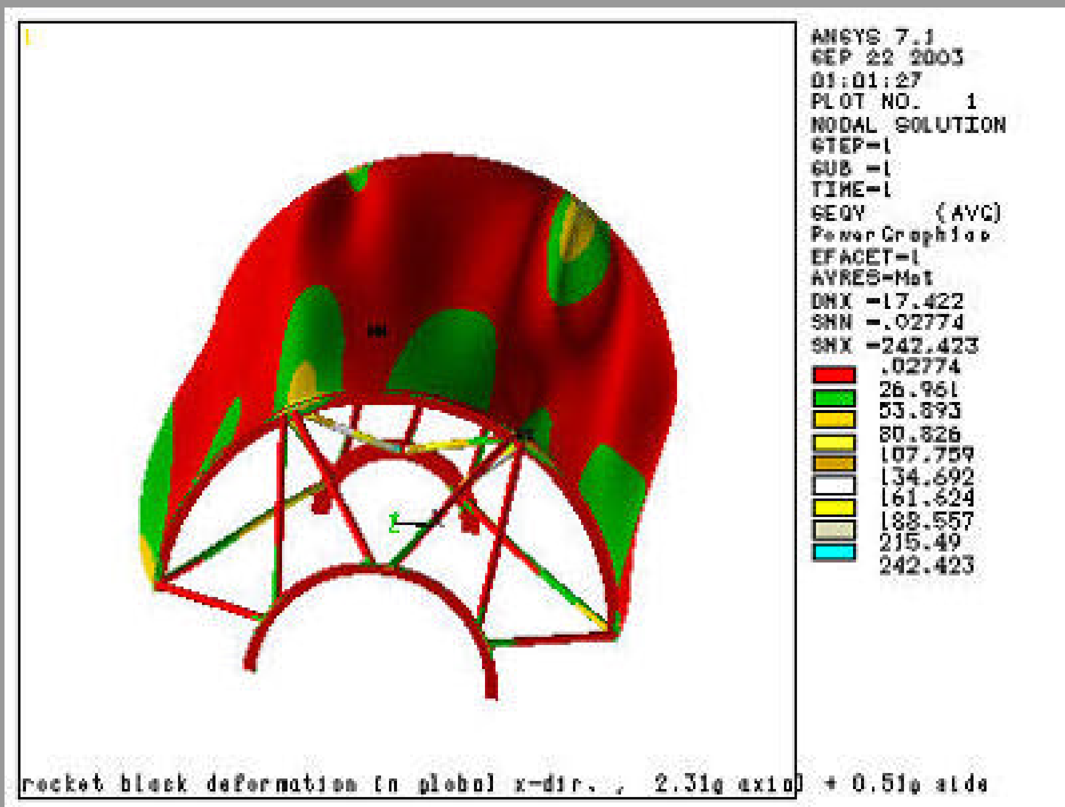


Figure 13d. Stress Distribution in the half-geometry, showing maximum von Mises stress of the entire structure, 242MPa

Maximum nodal stress and yield strength safety summary is presented in the Table 3.

Table 3

Component		
Items	Safety Factor	Stress, MPa
upper truss	4.9	98.9
lower truss	1.9	242.4
capsule mount ring	5.9	81.8

N2O fuel tank ring	3.2	150.4
aeroshell truss attach ring	2.9	162.9
engine to aeroshell truss	2.9	162.7
aeroshell	2.4	200.2
engine truss	8.9	53.8

348K element model results (for von Mises stress) clearly point that lower truss structure has highest stresses and present the weakest link. We further verified these results by running a 525K element model that demonstrated the same maximum stress and lateral deformation. This confirms the conclusion that came out of beam element model. We recommended to further increase the lower truss thickness and we anticipate that it will increase overall system structural stiffness thus helping to reduce lateral deflections to 12 mm (0.012m).

Conclusions

ANSYS Software tools were heavily utilized by da Vinci project engineers throughout all phases of research, design and manufacturing. They allowed effective integration with parametric Autodesk Inventor 7 CAD package and Matlab 6.5/Simulink 13 to form computational design analysis (CDA) process streamline. Using comprehensive set of computational solution technologies from ANSYS we analyzed multiplicity of thermal and stress problems that were encountered by our project team. These technologies included (but were not limited to) Workbench bi-directional parametric CAD interface, variational design technology and superior parametric grid generation capability of ANSYS Classic. Wide set of utilized features included beam, shell and brick element models and a selection of iterative and sparse matrix serial and parallel solvers.

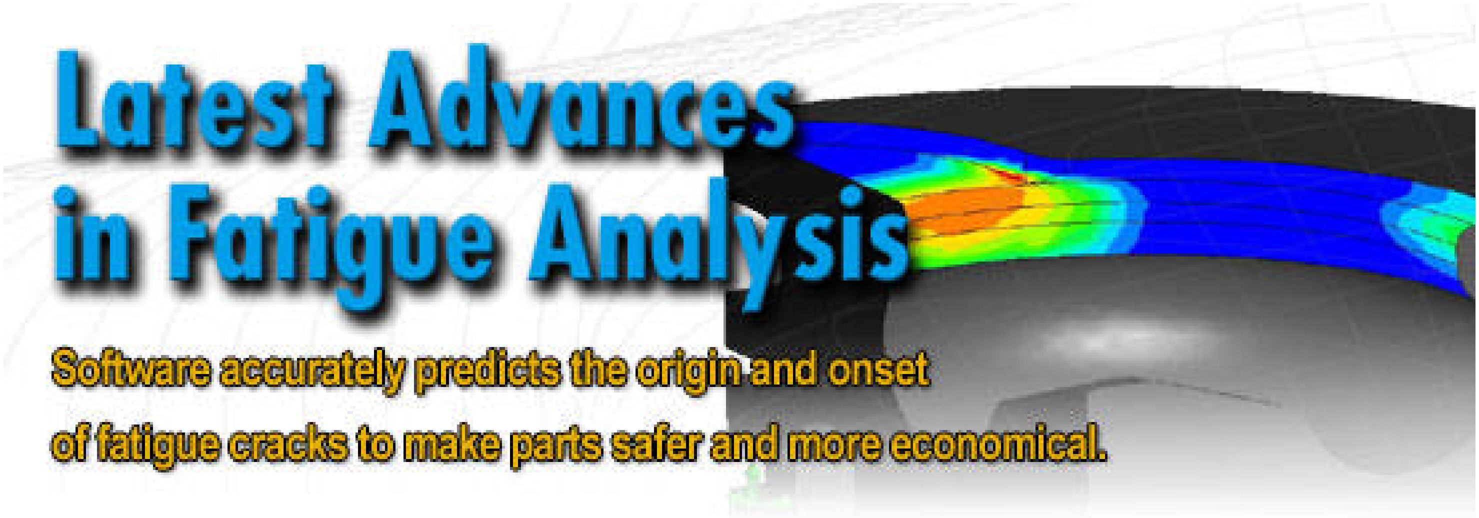
Not all finite-element results were created equal. If PC hardware imposes considerable modeling limits (model size, resolution, and high aspect ratios) then it inevitably raises the bar of required human expertise, slows down CDA process and requires considerable number of extra engineering resources to estimate and validate uncertainties. That in turn leads to lower safety or product over-design. Implementation of powerful scalable workstations allowed us to shorten design process and required engineering time considerably with 3 months to 1 week ratio.

ANSYS tools had performed at their best when were coupled with the scalable state-of-the-art Sun Microsystems Blade 2000 workstations-servers under Solaris 9 64-bit operational system. We found shared parallel memory mode to be easiest and fastest to work with. Advanced hardware became the enabling environment that allowed bringing the best in both ANSYS Classic and ANSYS Workbench environments.

Space will never be the same.

Acknowledgements: to Dav1d Grossman for aerial photographs of Kindersley airport.
For more information on the volunteer da Vinci Space Project, visit http://www.davinciproject.com

---



By Professor John Draper
Managing Director
Safe Technology Ltd.

Engineers have known for more than 150 years that metal can fail in fatigue, but “fingers crossed” used to be the design rule for metal fatigue. Advanced CAD packages could provide accurate stresses and dynamic modeling could predict loads and vibration effects with impressive accuracy. But the final and most important calculation - How long will it last in service? - was an over-simplified, almost back-of-the-envelope calculation surrounded by uncertainty. Now engineers can perform fatigue analysis on machined, forged and cast components in steel, aluminum and cast iron, on complex assemblies containing different materials and surface finishes and most recently can even incorporate the effects of high temperature and time-dependant creep-fatigue in the calculation using state-of-the-art software represented by fe-safe v5 from Safe Technology Ltd.

Fatigue used to be a black art that produced financial black holes. Premature fracture of engineering components costs Europe and the US about 4% of GDP each year. Potential errors were hidden behind the euphemism of “safety factors.” And manufacturers paid the price for overweight components that cracked prematurely, a seemingly endless series of prototype developments, unpredictable warranty claims and loss of customer confidence. Traditionally, fatigue failures have been fixed by over design. But increasingly, engineers are under pressure to “design down” to save weight and material costs; over design is no longer a viable option and the need for sophisticated fatigue analysis tools becomes increasingly apparent.

How Cracks Initiate

In 1850 the I.Mech.E. discussed the results of fatigue tests on wrought iron, where an iron bar was subjected to repeated cycles of loading to simulate 90 years in the service life of a

railway axle. They considered that the raised shoulders used to locate the wheels on the axle contributed to the failures, and that failures would not occur providing the stresses did not exceed the elastic limit.
So three essential features of fatigue design were discussed in 1850 – that failures are caused by cyclic loading; that stress concentrations at changes of section reduce the fatigue life; and that there is a safe working stress below which failure will not occur (although its definition is not quite as simple as that proposed in 1850).

We now know that fatigue cracks initiate in weaker grains or grain boundaries in metals under the action of repeated stress cycles and that these small cracks may propagate, also as a result of cyclic stresses, until the material fractures. But to calculate the fatigue life we must consider every significant load in the service life. The complexity of service loading, and the complex stress states which can occur during the service loading, means that fatigue analysis is much more challenging than simple "design to withstand maximum loads," and it has taken the past 150 years to produce the algorithms needed for confident fatigue design.

Technology Advances

The past fifteen years have seen the transformation of fatigue analysis into a sophisticated computer-aided design tool with an accuracy at least equal to other software in the design and analysis process. Fatigue analysis software combines component loading, FEA stresses and materials data and performs advanced multi-axial fatigue analysis. The software interfaces directly to FEA software and is supplied complete with a database of material fatigue properties to which users can add their own data. It calculates where and when fatigue cracks will occur - the fatigue hot-spots - the factors of safety on working stresses for rapid optimization and the probability of survival at different service lives - the 'warranty claim' curve. The results are presented as contour plots of fatigue lives, stress safety factors and probabilities of failure and plotted using standard FEA viewers and graphics software.

The accuracy of modern fatigue analysis is based on research that originated in the 1950's, and really started to be applied during the 1980's when in-vehicle load measurement became available together with analysis software and low cost computers. This early software analyzed measured strain histories and was used for prototype assessment and post-failure investigation. Local stress-strain or "critical location" fatigue analysis uses a mathematical model of the material's response to each loading event, including the effects of in-elastic stresses that may occur locally in fillet radii or other stress concentrations. This integrated approach works equally well for high-cycle components and for components where some of the fatigue damage is caused by less frequent high loads. The traditional separation of fatigue into "high-cycle" and "low-cycle" is no longer necessary.

Materials databases contain reliable fatigue properties for a large number of materials, and material testing to published standards is widely available at low cost. Local stress-strain fatigue methods were originally developed to analyze strain gauge measurements, often taken from prototypes.

During the past 15 years, the fatigue analysis techniques have been extended to analyze results from finite element models. FEA analysis is computationally intensive. Using elastic-plastic FEA to model a component's stress-strain response to a long time history of service loading is unacceptably time-consuming. For this reason, techniques have been developed to take a single set of results from an elastic FEA, scale the stresses by the time history of loading and calculate the extent of any plasticity that may be developed at each node on the model. This takes place in the fatigue software where the analysis is much more computationally efficient. Multi-axially loaded components can be analyzed by simple superimposition of the elastic results before estimating plasticity effects.

Until recently these methods could only be applied to uni-axial stress states. This was because of the complexity of calculating cyclic plasticity effects for multi-axial stresses. Suitable techniques of acceptable accuracy are now available, and it has become common practice to treat all components as multi-axially stressed. Principal stresses that change their orientation are handled using critical plane analysis, a search technique to find the direction of crack initiation at each node on the model. Figure 1 shows the fatigue life contour plot of an aluminum alloy suspension component, generated using modern fatigue analysis software working from a finite element computer model. The software correctly predicted the failure locations and gave excellent correlation with the test lives.

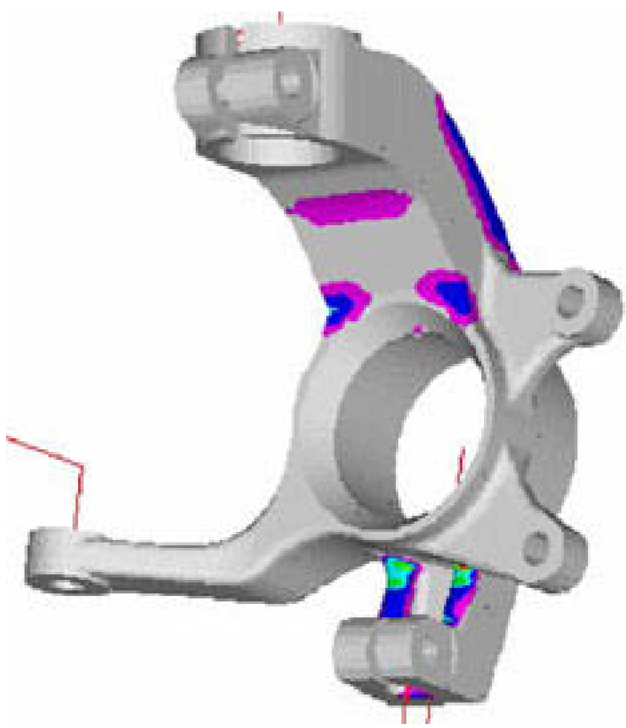


FIGURE 1. FATIGUE LIFE CONTOURS FOR A SUSPENSION COMPONENT

Moving Away From Design-Test-Redesign

During design, when the component exists only in a 3D CAD system, potential failures can be corrected cheaply. Once metal has been cut and moulds or dies produced, fatigue failures become much more expensive, and the planned production time-scale is in jeopardy. Costs really escalate if prototype testing fails to reveal a potential failure and the component is put into production. There is then a real possibility of warranty claims, recalls and accident liability. If manufacturers are to escape from the costly cycle of “design-test-redesign,” they need to invest a little more in design.

Figure 2 shows what happens if durability assessment is built into the design process to assess and validate virtual components before cutting metal. True, initial design costs are higher because the analysis is more sophisticated, but compensation comes in the form of increased accuracy, which gives the confidence to remove 'virtual metal' from areas that are over-strength in fatigue. Lower weight may reduce dynamic forces and produce further benefits in other components. There will also be immediate savings in material costs in production.

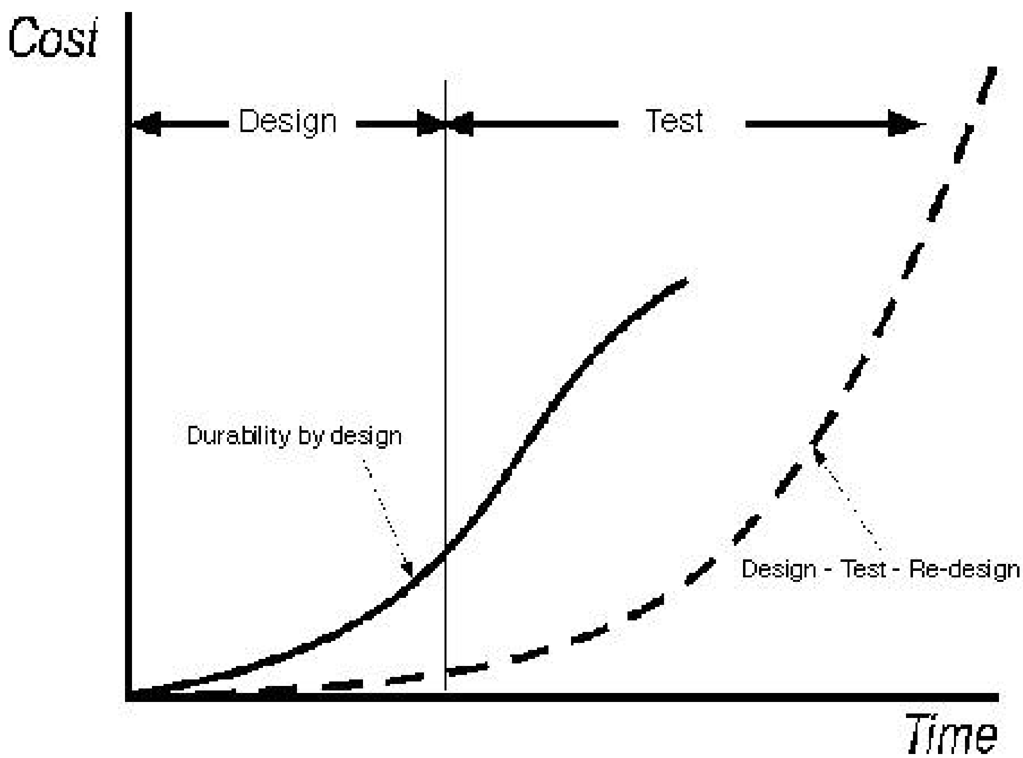


FIGURE 2. COST AND TEST BENEFITS FROM FATIGUE ANALYSIS DURING DESIGN

Further payback comes during prototype testing, since it may not be necessary to test at all. More probably, a prototype could be required by the customer or demanded by legislation. But because of the extra effort and sophisticated analysis at the design stage, there is now a much greater probability that the prototype will be right first time. Test times are reduced and are more predictable, so they can be integrated into the project plan. And the product gets to market faster. In addition, there are some other less obvious benefits. Fatigue analysis software can identify which loads are important at the predicted failure points, so again prototype testing can be simplified and inadvertently missing out some important loading can be avoided.

Fatigue analysis software is also able to estimate the future warranty claim curve, and in particular how it might vary for different user profiles. An example is shown in Figure 3.

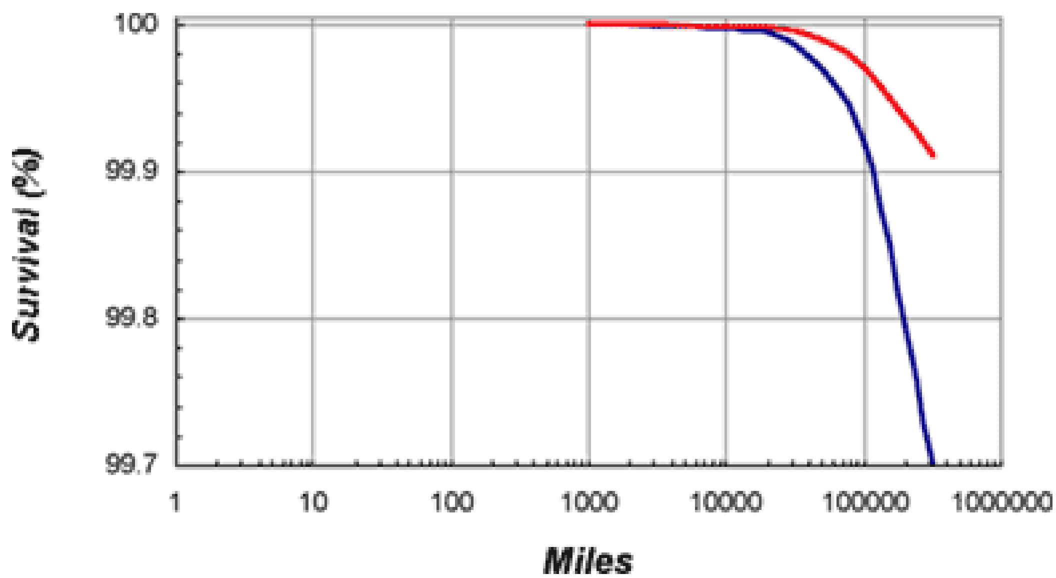


FIGURE 3. EFFECT OF CUSTOMER PROFILE ON IN-SERVICE FAILURES

This shows the effect of two different user profiles on the calculated in-service failure rate, and from this the design could be refined or exclusions could be put in the warranty. Either way, the information is available to assess both options and make the most cost-effective decision.Continuous Development

With these techniques now widely accepted, Safe Technology's fe-safe software is being extended into the much more demanding areas of high temperature and time dependent creep fatigue . These features are vital considerations in engine design, for example. The main considerations here are:

- Stress-strain response, which depends on instantaneous strain rate and temperature
- Bulk relaxation of stresses with time
- Strain-aging of the material
- Phase relationship between stress and temperature, which can vary from cycle to cycle

Although quite specific materials data is required, the results justify the effort, and very successful life estimates are being obtained. Figure 4 shows the results of high-temperature fatigue analysis of a prototype automotive piston. The premature failure of the prototype was predicted accurately by the fatigue analysis software, and a number of computer-aided-design iterations were used to develop the successful design. It is interesting that one of the design changes was to increase the radius of the fillet at the point of crack initiation – a solution which goes back to the earliest days of the fatigue design.

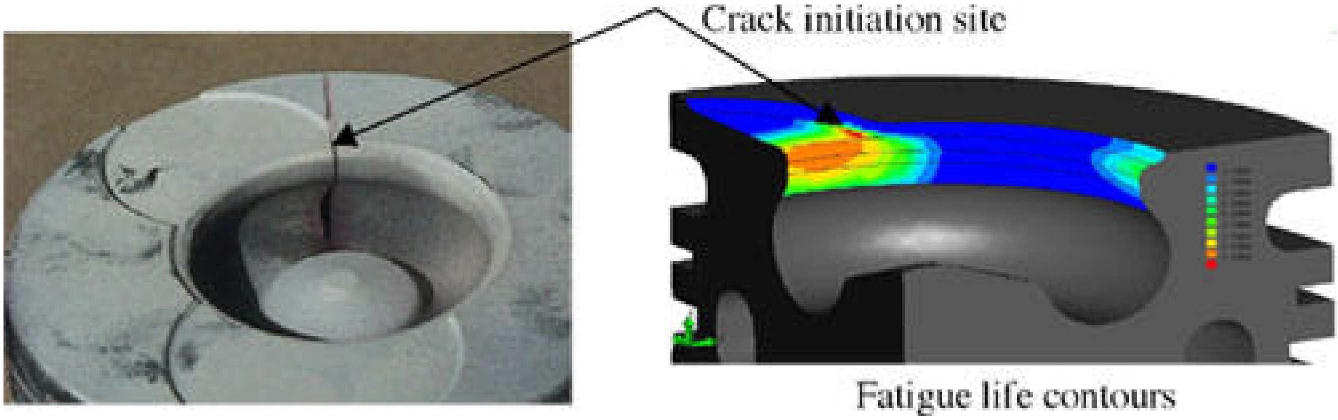


FIGURE 4. HIGH TEMPERATURE FATIGUE ANALYSIS OF A PISTON

Conclusion

Fatigue test lives are usually plotted on a logarithmic scale. Fatigue progress also seems to follow a log scale. 150 years ago engineers were defining the initial rules for fatigue design. 15 years ago a simple uni-axial fatigue analysis from a few strain gauges was considered a success. Today we can achieve a much more accurate fatigue analysis for complex multiaxial stress states, for real service loading, with effects of temperature and manufacturing process for a million nodes in a finite element model and highly detailed fatigue life contour plots can be produced. Safe working stresses for design optimization can be calculated and warranty claim curves can be estimated. Advances in computer power and faster software algorithms allow durability analysis to be integrated even more tightly into the design process. Successful fatigue analysis for increasingly demanding components is now a practical proposition. As more products are designed “right” rather than just “strong enough” to meet ever more stringent weight and cost saving targets, under tighter production schedules, fatigue analysis can no longer be ignored or done with your fingers crossed.

John Draper worked as a fatigue design specialist in the UK aircraft industry, then as a project manager for fatigue research and failure investigation projects in the British Railways R&D Division. He founded Safe Technology Limited in 1987. He is also Chairman of the Engineering Integrity Society and an Honorary Visiting Professor to Sheffield Hallam University, England. For more information on Safe Technology and fatigue analysis software, visit www.safetechnology.com.

Acknowledgement: The author wishes to thank Metaldyne International, Federal Mogul Technology and MIRA for all images supplied.

Lynn Lewis
HP CAE Alliance Manager
Hewlett-Packard Company

Obtaining maximum value from a company's investments in CAE demands the highest levels of performance and capability from the computing environment. Extraordinary advances in computing price and performance now enable users to increase model resolution and complexity for deeper insights into improving products.

At the forefront of these advances is technology from Hewlett-Packard Company, which provides a wide range of computing hardware as well as the expertise to help companies select the right solution for their needs. HP solution architects will evaluate a company's CAE requirements, working with users and the HP CAE team to navigate the complex range of options available today:

The recent release of HP Integrity servers and clusters gives CAE users new choices with compelling benefits. These systems support all ANSYS Inc. products and ANSYS Inc. partners applications on the HP-UX operating system. Many of these applications are also available for Linux systems. This article provides guidance on how these Itanium2-based systems meet the toughest CAE requirements—without compromise.

Server Solutions for CAE Applications

Integrity Server Configurations. HP Integrity servers deliver powerful throughput, flexible operation, and easy system management supporting hundreds of CAE users running a complete suite of CAE applications. Based on Intel Itanium2 processors, these server architectures have comparatively lower memory latency, massive memory capacity, and a fast I/O subsystem.

- The two-processor HP Integrity rx2600 entry-level server provides fast and affordable performance for complex technical computing.
- The four-processor HP Integrity rx5670 supports larger models with up to 48 GB of main memory and increased scalability for SMP parallel processing.
- The high-end HP Integrity Superdome, with up to 64 Intel Itanium2 processors, delivers record-breaking raw performance and job throughput required by large engineering organizations.

Operating Environments. Running the HP-UX 11i version 2 Technical Computing Operating Environment (TCOE), HP Integrity servers support virtually all CAE solver codes at the fastest possible performance level and with SMP or MPI scalability up to 64 processors. Current HP-UX customers on PA-RISC systems achieve seamless integration of Integrity servers into their existing high-performance computing infrastructure. Current SGI IRIX, Tru64 UNIX, Sun Solaris, and IBM AIX customers can readily migrate to HP's UNIX for economy and operating system independence.

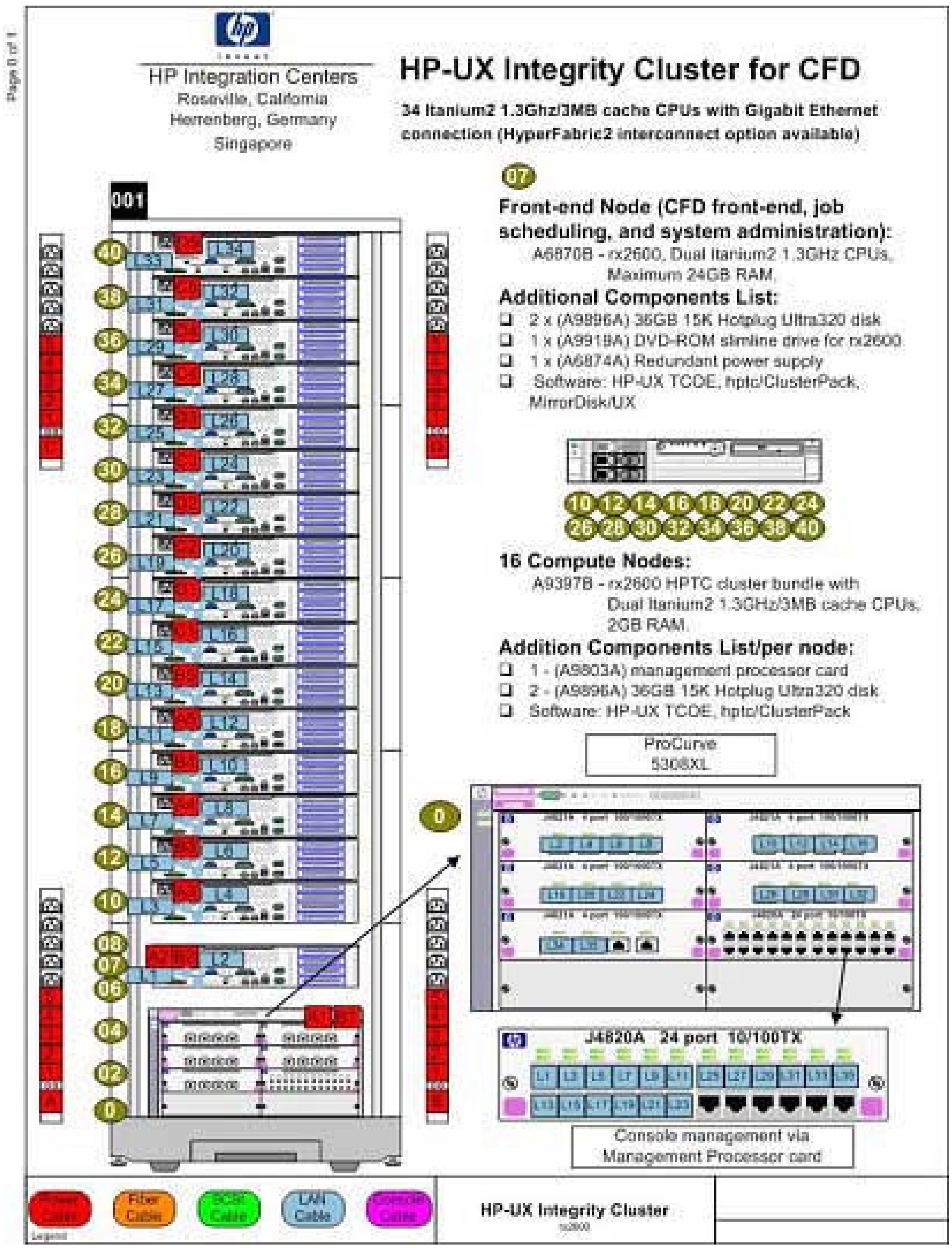
Running the 64-bit Linux operating environment, HP Integrity servers support most leading solver codes, with more being released periodically. Performance scaling is achieved with up to four processors on entry-level Integrity servers today, or 64-bit Linux clusters of these nodes can be built with up to 128 CPUs for MPI applications such as computational fluid dynamics (CFD).
Integrity Server Advantages.

- Superdome runs multiple CAE jobs from many users simultaneously with little degradation in time-to-solution.
- 2- and 4-CPU servers may be racked as a loosely coupled "farm" running multiple simultaneous jobs with no contention.
- All SMP server memory is available for pre- and post-processing tasks such as grid generation and domain decomposition/discretization.
- Servers run all solver codes (SMP or parallel MPI) for operational efficiency.
- Single operating system image on one large server provides the easiest system administration.
- HP Services performs on-site installation and management (visit www.hp.com/hps for details).

Investment Protection. HP Integrity servers provide best-in-class performance, an open architecture roadmap, aggressive acquisition prices, and low operating costs, all of which help maximize a company's return on investment in CAE. HP also offers the proven AlphaServer series for CAE solutions, ideal for current Alpha customers. Based on the Alpha EV7 processor, AlphaServers have an impressive record for performance, scalability, availability, and management. CAE solver applications are supported on Tru64 UNIX operating environments through 2006.

ANSYS Support for Tru64. From feedback received from its customers, ANSYS Inc. has reconsidered discontinuing support for the HP Compaq Alpha UNIX Tru64 as early as ANSYS

8.1 and now plans to support the Tru64 version until at least ANSYS 9.1 (Spring 2005). The ANSYS Inc. letter information customers of this decision is at www.ansys.com//hardware_support.



HP Integrity cluster HP Integrity Superdome

Cluster solutions for Optimization or Multi-Physics

Cluster Configurations. Clustering multiple SMP nodes to form a large, scalable system is increasingly popular among CAE users. With recent advances in interconnect technologies, system management software, and distributed-memory CAE applications, clusters provide attractive and cost-effective solutions for the high performance and capacity requirements of fluid and explicit analysis. HP offers complete cluster solutions including servers, interconnects, software, management tools, integration, implementation, customer services, and training, all from HP and trusted HP resellers. The HP-UX Integrity cluster is typically based on the HP two-processor rx2600 Integrity servers.

hptc/ClusterPack Software. The HP-UX Integrity cluster comprises multiple compute servers controlled by one head node (or management server), which directs the cluster by means of an integrated software package: hptc/ClusterPack, which provides a comprehensive set of integrated tools that simplifies the tasks of cluster configuration, cluster system administration, and distributed job management. Multiple interconnect technologies are supported under the HP-UX Integrity server cluster. For many CAE applications, the industry-standard Gigabit Ethernet has proven cost-effective in connecting a moderate number of servers. The HP ProCurve Switch 5300xl series is well-suited for a 16-node cluster, with the possibility of scaling up to 32 nodes on a single ProCurve Switch 5308xl. For CAE applications more dependent on low latency or high bandwidth, HP offers the HyperFabric2 interconnect supporting three messaging protocols: TCP, UDP, and HMP (HP's HyperMessaging Protocol).

Integrity Cluster Advantages.

- Enjoy attractive price/performance benefits while maintaining a 64-bit architecture for large models and solution accuracy.
- Add or subtract nodes easily for maximum flexibility and scalability.
- Achieve superior rack density in crowded IT spaces.
- Run MPI applications (see page 4).
- Choose your preferred private switched-fabric interconnect—HyperFabric2 for HP-UX, Myrinet for Linux, or Gigabit Ethernet for both.
- HP Services performs on-site installation and management (visit www.hp.com/hps for details).

Linux clusters. HP also offers flexible Linux-based solutions. Linux clusters are available using both HP Integrity servers with Intel Itanium2 processors and HP ProLiant servers with Intel Xeon processors.

Server Farm for Structural Analysis Solvers. Available configurations: multiple rx2600 or rx5670 servers, each with direct attach SCSI or FC disk array having 5+ spindles per disk controller for striping scratch files. HP-UX 11i / TCOE with vxfs file system. Tuning info available. Benefits include: multiple users submit 1, 2, and 4-way jobs simultaneously for fast job turnaround and little contention. 64-bit address space for models > ~2 million DOF. Solvers gain price/performance benefit of Itanium2 for ROI.

Helpful URLs

Explore the following websites for further information about products and services provided by Hewlett-Packard Company.

HP solutions for CAE
www.hp.com/go/hptc
www.hp.com/techservers/cae
www.hp.com/go/integrity

Integrity Cluster Quote Requests and Information
www.hp.com/techservers/clusters/linux_clusterblocks

HPTC Consulting and Integration
www.hp.com/techservers/products/consulting_and_integration

Lynn Lewis is HP CAE Alliance Manager at Hewlett-Packard Company, a technology solutions provider to consumers, businesses and institutions globally. The company's offerings span IT infrastructure, personal computing and access devices, global services and imaging and printing for consumers, enterprises and small and medium businesses. For the last four quarters, HP revenue totaled $71.8 billion. More information about HP is available at www.hp.com

---

Getting the Most from the ANSYS DDS Solv

Load Share Facility (LSF) optimizes resource allocation to leverage parallel processing solvers

By Arend Dittmer
Integration Architect
Platform Computing, Inc.

Responding to the industry trend toward handling increasingly complex models in shorter timeframes using multiple solvers, ANSYS Inc. introduced the parallel processing solver DDS (Distributed Domain Solver) with ANSYS version 5.7. Part of the Parallel Performance for ANSYS add-on module, DDS solves large static or transient nonlinear analyses over multiple systems (distributed memory parallel, DMP) and/or over multiple processors on a single machine (shared memory multi-processor, SMP).

To fully leverage parallel processing technology, distributed resources have to be managed effectively. Platform Computing's Load Share Facility (LSF), a sophisticated distributed resource management system, increases the throughput of DDS jobs by optimizing the host allocation for distributed DDS jobs. Moreover it ensures that DDS jobs are controllable and that accounting information is available. LSF makes it possible to run and manage parallel MPI based distributed applications as if they were running on a single system.

Load Share Facility

LSF is a solution that is highly complementary to distributed applications such as ANSYS DDS. LSF launches jobs on nodes that meet given resources requirements and chooses the least loaded systems in case multiple candidate nodes qualify. On Non Uniform Memory Architecture (NUMA) based systems, LSF supports an automated dynamic allocation of CPU sets, optimized for short memory access times.

Moreover, LSF facilitates the enforcement of business priorities through centrally defined policies for coordinated resource allocation. Examples of supported resource management policies are the preemption policy, the fair share scheduling policy (which allows "fair" access to CPU resources based on recent usage history) or the FCFS (First Come First Serve) policy. Through the integration of LSF with MPI libraries MPI-based applications such as ANSYS

DDS can be submitted to LSF transparently, without the need to edit MPI specific host files.
Job Control and Accounting

A key problem inherent to distributed applications like ANSYS DDS is the lack of control over the application at runtime. With every distributed DDS run, DDS processes are launched on multiple hosts. The Message Passing Interface (MPI) standard defines an interface for passing messages between these processes, but does not define a method to control all the distributed processes that belong to the same DDS run. It is not possible for an end user to kill or suspend a DDS run spanning multiple hosts with one command. MPI based applications are typically “out of control” once they have been launched. This has several implications:

- In order to control a DDS run, an end user has to log into multiple hosts and manually identify and kill processes
- Without an additional control mechanism some LSF scheduling policies cannot be enforced. It is for example not possible to let a higher prioritized parallel DDS job suspend a DDS job of a lower priority, to use its resources. The signal for job suspension is not sent to all processes of the lower prioritized DDS job.
- An exit of one process due to an error condition may result in “runaway” processes wasting CPU cycles. As all the distributed processes of a DDS run depend on each other, an exit of one DDS process should result in the exit of all processes that belong to this application run. Lacking a central point of control this behavior cannot be enforced.
- There is no easy way to get accounting information (e.g. CPU time usage, memory usage) for distributed DDS jobs.

In order to make it easier to control parallel applications and allow the enforcement of scheduling policies, LSF provides a tool that acts as the single point of information and control for parallel runs of MPI based applications: The Parallel Application Manager (PAM). PAM tracks all processes of a distributed application and forwards signals to all processes of an application instance. Moreover PAM is aware of processes that exit due to an error condition and automatically kills parallel jobs if any one of its processes exits.

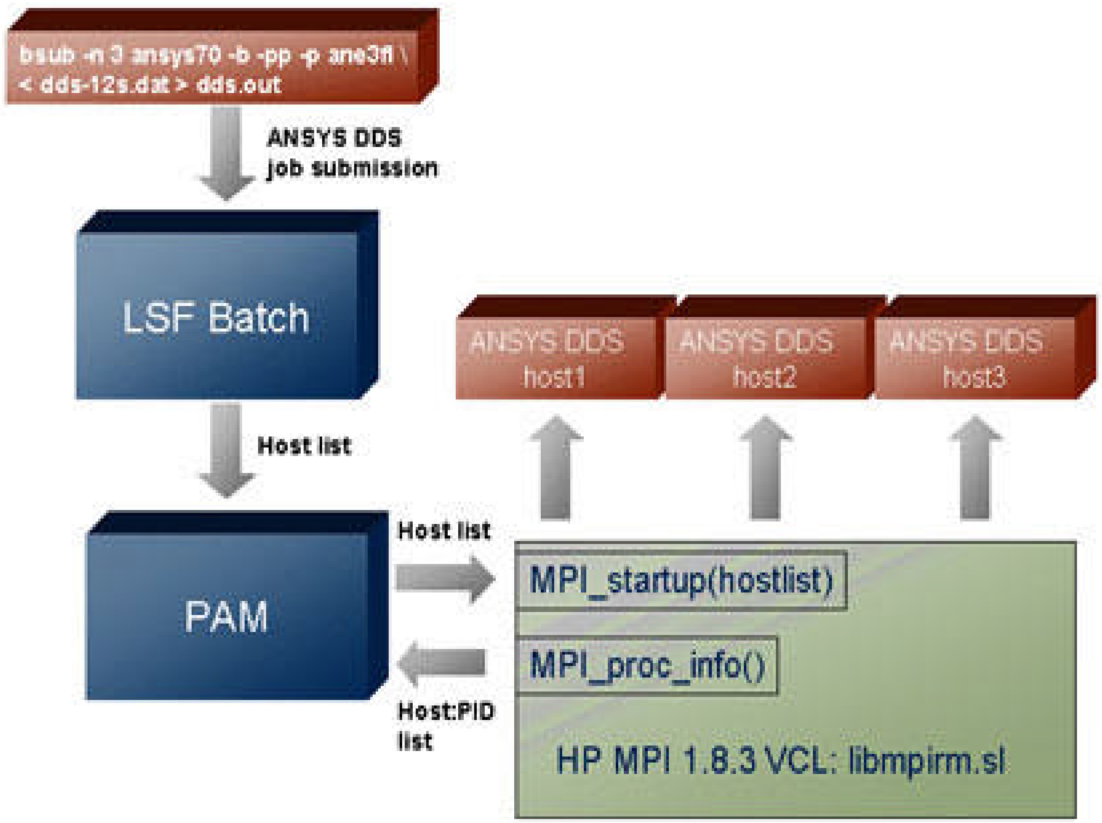
With the automated forwarding of control signals through PAM it is also possible for LSF to enforce scheduling policies for distributed application runs. In the preemption scenario mentioned above, PAM enables LSF to suspend all processes belonging to a distributed application run of a lower priority to “give way” to higher prioritized jobs. With PAM as a single point of control LSF can enforce resource limits, for example a wall clock runtime limit or a CPU-time limit.

Beyond providing job control capabilities for distributed parallel application runs, PAM also facilitates the collection of resource usage information. The accounting information collected by LSF on each host, is aggregated by PAM and is accessible to end users as well as administrators. In tandem with solutions like Platform Intelligence this accounting information can be used to charge for utilized compute resources.

LSF and PAM are bundled in Platform Computing's HPC package and work with MPI implementations from the major system vendors HP, IBM, SGI and Sun. A generic implementation of PAM that works with all MPI implementations is also available. Particularly in the context of low cost, high performance compute environments it is important to note that LSF/PAM are available for building IA32 and IA64 Linux clusters.

Integration with Vendor-Specific MPIs

The following figure shows a simplified high level overview of the startup procedure for distributed ANSYS DDS runs on HP-UX 11 systems, using the HP 1.8.3 MPI libraries.



Launching distributed ANSYS DDS jobs through Platform LSF

A job is submitted to LSF by prepending LSF's bsub command to the command line for launching a distributed DDS job. According to the number of requested CPUs, LSF allocates hosts that are to be used for the distributed launch. For host allocation LSF considers defined scheduling policies, specified resource requirements (no requirements were specified in this example) and the dynamic load situation on candidate hosts. After identifying suitable execution hosts, LSF batch launches PAM on one of the identified execution hosts. In this example, PAM loads an HP specific Vendor Component Library (VCL), which is part of HP's MPI package. The VCL contains a startup function that launches the MPI application on the allocated hosts. Through the MPI_proc_info function PAM is able to retrieve the process id and host of each process belonging to the parallel ANSYS DDS run. After executing this function PAM has the information that is required to forward job control signals or gather accounting information for DDS jobs launched through LSF.

Arend Dittmer is Integration Architect with Platform Computing, Inc. The company plans, builds, runs and manages enterprise grids that optimize IT resources according to core business objectives. Platform is at the forefront of grid software development and has helped more than 1,600 clients gain powerful insights that create real, tangible business value.

A defining feature of the CFX-5 CFD software is its high performance coupled multigrid solver. While it is true that coupled multigrid is a key technology to achieve high performance, many additional technologies are involved. High performance CFD requires a cohesive strategy and excellence in six technology areas: meshing, accuracy, reliability, speed, physics, and flexibility. As with most technologies, overall success is limited by the weakest link. CFX-5 delivers high performance CFD by a unique combination of strategies in these six areas, as outlined below.

Meshing Strategy

The CFX-5 meshing strategy is one of flexibility. No single strategy is ideal for every case. Choice is the key. Four element types, hexahedral, tetrahedral, wedge or prism, and pyramid, are available. From these building blocks almost any style of mesh is possible. CFX-5 supplies its own state-of-the-art hybrid meshing technology which exploits a mixture of element types, as well as automated curvature, edge and 3-D proximity detection, boundary orthogonality, and smooth transition scales. The result is near-automatic best-practice CFD meshes that resolve the geometry and the boundary layer. CFX-5 also supports a large number of external grid formats. More choice. Finally, combinations of multiple mesh styles in a single analysis is easy to do using the General Grid Interface (GGI) technology in CFX-5, developed and perfected over a 12 year period in CFX-TASCflow. For example, a hex mesh created with ICEM Hexa can be connected to a hybrid unstructured mesh created with the CFX-5 mesher.

Accuracy

Every company has finite computing resources, no matter what the size. It remains essential to get the most accurate answer possible on your mesh, be it 20,000 or 20 million nodes.

CFX-5 delivers high accuracy per node by a combination of several key factors. Element based discretization is employed, unlike the “volume filled” methods in most other commercial CFD codes. High accuracy per node is the result. On a tetrahedral mesh, each integration volume in CFX-5 consists of 60 surface “integration points”, on average. In comparison, competitor volume-filled methods consist of a meager 4 surface “integration points” per volume. High accuracy per node does not mean high cost per node. The CFX-5 element-based method is inherently efficient, providing a natural “condensation” of effort. For example, on a pure tetrahedral element mesh there are 1/5th the number of linear equations to solve compared to the same mesh used in a volume-filled method.

Rapid reliable reduction of numerical error as the mesh is refined is critical. Numerical errors occur in the key transport terms advection, diffusion, and in sources. The CFX-5 element technology yields accurate local gradients for diffusion, and sub-grid source resolution. Because the advection term error is often dominant, CFX-5 offers all of the industry-accepted first and second order advection schemes. Our premier second order scheme is robust and accurate. CFX-5 is the only commercial CFD software that enables second order discretization by default.

In any CFD prediction there are model errors in addition to numerical errors. The turbulence model is often a major source of model error. Choice is available (over 16 different turbulence models from RANS to DES), but for some this is daunting. In response, CFX-5 has established a new workhorse turbulence model, SST. This model is as economical as the k-; model, but offers much higher fidelity, giving excellent answers on a wide range flows and near-wall mesh conditions. For more complex flows, CFX-5 offers full Reynolds Stress models based on the v-equation with automatic wall treatment and a newly developed zonal Detached Eddy Simulation (DES) formulation based on the SST model.

Reliability

A CFX-5 simulation involves the solution of a set of coupled non-linear equations. The non-linear solution is obtained by repeatedly updating and solving a set of linearized equations. It sounds obvious, but one of the best ways to achieve reliable convergence is simply to “solve the linear equations well”. There are many approaches but one method stands out, coupled multigrid. The CFX multigrid method has evolved over a 20 year period, rooted first in CFX-TASCflow and now in CFX-5. It has been used to solve literally millions of simulations. This is the only linear solver available in CFX-5. It is fully automatic. It is fully scalable (linear increase in CPU time with problem size). It is insensitive to mesh aspect ratio. It greatly reduces sensitivity to the time step and relaxation factors. It is fully parallelized. It is fully implicit. It solves the linear equations tightly. All of this means the user experiences far fewer problems in getting the desired computation from start to end. Many “coupled solvers” and “multigrid solvers” exist in other codes, but none like that in CFX-5. CFX has relied on this technology, solely, for the past 10+ years.

Speed

Any number of computers can be combined to perform a given analysis: parallel computing. The CFX-5 parallel implementation combines the memory and CPU resources of many machines to reduce wall-clock time, and to make larger simulations possible. All physical models, features, modes, and options in CFX-5 work in parallel, no exceptions.
CFX-5 makes it is as easy to perform a simulation in serial or parallel, using a multi-processor computer, a network of single processor computers, or any combination. Parallel performance maintains both CPU scalability and memory scalability, even for large numbers of processors. The CFX-5 parallel performance is due, in part, to the fact that parallel and serial simulations follow the same convergence history. It is easy to parallelize a CFD code, but much harder to parallelize while preserving its convergence performance. And with the release of CFX-5.6 even the native mesh generation process is parallelized. Another industry first.

Physics

A wide range of physical models is needed in any modern CFD software package. The fidelity of a simulation is linked directly to the choice of physical models available. CFX-5 has seen a huge expansion in its physical models in recent years. Steady and transient turbulence models (16+), frame change models (3), mixture models, multiphase models (homogeneous or inhomogeneous, N phases), free surface models, interphase mass transfer models like boiling, condensation and cavitation, Lagrangian phases, combustion models (8+), real fluid models, high speed flow models, radiation models (3+), transient models… the list goes on. CFX-5.6 is model rich. A key point is that these models all inter-operate with each other, and in conjunction with the key technologies already discussed: for all element types, across all GGI connection types, using the coupled multigrid solver, in parallel, with accurate numerics and advanced turbulence models. This inter-operability is referred to as a “full feature matrix”, and is a significant benefit to CFX-5 users.

Flexibility

Finally, high performance is only possible when the software system is flexible and open. This is achieved in CFX-5 through a variety of approaches. A highly flexible and powerful User Fortran system is available for custom development. The user interface system is programmable and scripted. The pre-processing, solver and post-processor all use a uniform, simple, state language, as well as embedded Perl language support. All functions run interactively or in batch mode. The result is that CFX-5 can be customized and embedded into your end-use system, in a clear and reliable manner.

In the future you might hear it said that “CFX-5 performs well because of coupled multigrid”. Maybe your response could then be “yes, that’s true, but that’s not the whole story”. High performance CFD is achieved in CFX-5 by application of a cohesive strategy and excellence in all six of the key technology areas: meshing, accuracy, reliability, speed, physics, and flexibility.

CFX1427 , CFX1426 - Automatic 3-D proximity detection in the CFX-5 hybrid mesher. The mesh density increases automatically where objects are close to each other.

GGI2.eps - Dis-similar meshes can be connected together using 'General Grid Interface' technology in CFX-5. The mesh structure and shape can change. The GGI connection is fully automated for the user.

LES.eps - Large eddy simulation, 'LES', of flow behind a cylinder. This fully transient turbulence model directly predicts and resolves the largest turbulent structures.

graph.eps, **graph2.eps**, **dragpolar.eps**, cfx1961, cfx1966 - AIAA Drag Prediction Workshop results obtained with CFX-5. Supplied hex mesh involves 5.8 million nodes and has grid aspect ratio in excess of 5000:1. The predicted lift and drag are stable in about 110 timesteps. This transient calculation runs on 16 Linux PC's in about 20 hours. AIAA Drag Prediction Workshop results obtained using CFX-5 were computed within the EU Flomania project.

Challenge:

To improve energy efficiency in the design of refrigerant-cooled hermetic motors for water- and air-cooled chillers.

Solution:

Implement CFX to simulate heat transfer phenomena in high-power induction motors and refrigerant flow through the hermetic motors.

Benefits:

Enabled engineers to explore large-scale unsteady flow phenomena to obtain the highest energy efficiency in a wide range of applications. Minimized environmental impact by improving reliability of the products, increasing service intervals, and reducing the need for disassembly of the units. Also eliminated possible emission of refrigerant associated with these processes.

Using CFX to analyze the unsteady flow field inside the chiller, track down the response of the unit structure, and predict air motion around the machines, allowed engineers to deliver units with lowest noise radiation.

Introduction

As living standards rise precision manufacturing proliferates around the world, air-conditioning equipment accounts for more and more energy consumption. Stricter environmental regulations and increasing environmental consciousness of air conditioning equipment users demand more efficient compressors with less impact on the environment. The Trane Company, a leading worldwide supplier of indoor comfort systems, is using the latest technology -CFX software from ANSYS, Inc. -to achieve these goals.

Improving energy efficiency is a key issue in the design of chillers for an air-conditioning system. Efficient chillers not only reduce the operating cost but also reduce green house gas emissions by reducing power consumption. A one-percent increase in efficiency brings substantial savings over the life of a medium size chiller.

Trane centrifugal chillers in the chilled water plant at the University of Arizona, Tucson.

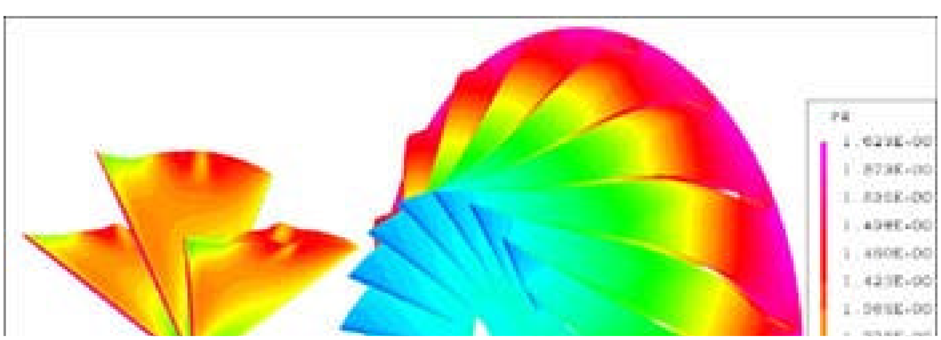
The social and environmental benefits are even greater. Trane engineers are using multi-objective optimization to improve chiller efficiency. Different from the traditional trial-and-error technique, Trane engineers analyze multiple variables and effects simultaneously and optimize the design to achieve the maximum benefit.

Challenge

A refrigerant-cooled hermetic motor is an integral part of Trane water- and air-cooled chillers. This technology provides high efficiency and extends the life of high-power induction motors. In these hermetic refrigeration machines, coolant flow is needed to carry away the heat generated by high power motors to improve energy efficiency and motor durability. However, the coolant flow also induces the windage loss to the systems.

Two competing mechanisms need to be quantified simultaneously to achieve the optimized design of these hermetic systems. Trane engineers use latest computational fluid dynamics (CFD) technology to simulate heat transfer phenomena in high-power induction motors and refrigerant flow through these motors concurrently.

These simulations bring many benefits compared to traditional build-and-test techniques: temperature distribution in the motor is evaluated to eliminate hot spots and obtain uniform cooling; through-flow induced torque and loss are determined; the impact of rotor surface geometry and speed are assessed; and the required mass flow rate for effective cooling is predicted. As a result, hermetic refrigerant machines are designed with optimized configuration for the best energy efficiency.



CFX calculates force and torque from pressure and viscous shear stress on the surface of inlet guide vanes and impeller of a Trane centrifugal compressor stage.

To improve energy efficiency, design and analysis move from the individual component to the entire system. Compressor aerodynamic performance is no longer limited to single parts, but extended to include the interaction of multiple components.

This integrated analysis ensures the maximum performance of each component as a part of the compressor system at various working conditions.

Solution

Trane has developed a Virtual Laboratory for the design of refrigerant compressors. It is based on CFX, a CFD software package that Trane has found to be ideally suited to modeling indoor comfort systems and which includes a real-gas equation of state for the refrigerant. The result is that engineers can easily obtain overall performance and local flow field details for
complete compressor stages without building a prototype.

Different from single-component analysis, the Trane Virtual Laboratory simulates the entire compressor, readily providing information on the interactions between the components. Since it quantifies the impact of design changes of a single part on the performance of the whole compressor, this feature has special value for the overall improvement of machines. This capacity proved very useful when Trane investigated the options to use different diffusers in compressors.

The impact on the overall compressor performance was analyzed in detail, with the simulations indicating that the change would lead to different flow fields inside the upstream and downstream components. The individual performance of these components was altered when the diffuser was changed. The information obtained was of great value to designers, guiding product improvements and avoiding unnecessary design iterations.

Transient simulations in the Trane Virtual Laboratory provide further physical insights into compressor performance and flow field unsteadiness, reduction of which is critical to improve efficiency and reduce vibration and noise levels. Pressure fluctuation distributions inside the impellers and diffusers can be obtained for different compressor designs and loss mechanisms inside the flow field can be studied thoroughly.

CFX allows Trane engineers to explore large-scale unsteady flow phenomena. Removing these instabilities has the benefit of obtaining the highest energy efficiency in a wide range of applications. To minimize the impact on the environment, Trane also develops technology to improve the reliability of our products, increase service intervals, and reduce the need for disassembly of the units. Possible emission of refrigerant associated with these processes is eliminated.

For the compressors designed by Trane, the force and torque on components are quantified to ensure safety and reliability. The force and torque are calculated from the pressure and viscous shear stress obtained from flow field simulations on the surface of the components. These force and torque calculations are also used to predict the vibration and noise of the system. The possibility of failure associated with fatigue of the components is therefore reduced. All this analysis is conducted using a whole system simulation under strict dynamic loading conditions. This cutting-edge technology helps Trane chillers earn a reputation as the world's most reliable refrigeration machines.

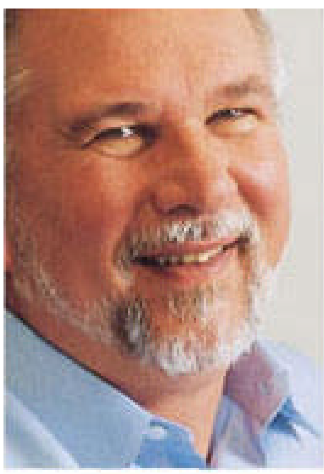
In recent years, the sound produced by HVAC equipment has attracted more attention. Since the equipment is typically located near building occupants, noise radiation must be controlled. Reducing acoustic impact is particularly important for hospitals, schools, and music halls. The machinery sound from HVAC equipment is caused by temporal variation in the flow field. The internal flow field vibrates the machines, setting the air around them in motion. This unsteadiness reaches the human ear in the form of noise.

Benefits

By using CFX to analyze the unsteady flow field inside the chiller, track down the response of the unit structure, and predict air motion around the machines, Trane engineers deliver units with the lowest noise radiation. Since Trane developed the industrial first scroll compressor, the Trane 3-DR scroll compressor, in 1987, these devices have dominated the market for small tonnage air-conditioning equipment.

Compared to the reciprocating compressors, where intake, compression, and discharge occur in discrete steps, scroll compressors conduct intake, compression, and discharge phases of operation simultaneously in an on-going sequence. Their smooth operating characteristics reduce force and torque variation inside the compressor, making scroll compressors quiet and reliable. Lubrication is very critical to improve the durability of scroll compressors since the oil pump is an integral part of the compressor. Trane engineers developed the technology to use the latest particle tracking techniques to predict the oil circulation rate inside the scroll compressors.

The oil coming from different regions inside the compressors is tracked through the operating process. Oil circulation features of different designs and oil droplet sizes are obtained at different operating conditions. CFX helps designers develop compressor designs with adequate lubrication and an ample supply of oil. The fully lubricated moving components extend the durability of the system and reduce the need to replace parts and service the units.



John Krouse
Editorial Director

Finite-element analysis (FEA) and related engineering simulation technologies bring significant benefits to user companies. How this software is applied and what role it is given in overall operations have a direct bearing on the value of the technology to the enterprise and the ultimate justification making such an investment.

As analytical and predictive tools, simulation can be used to study stress, deformation, fluid flow, heat distribution and other critical attributes in much greater detail and far less time than running physical tests on hardware prototypes. On this level, simulation brings a lot to the table in improving designs of parts and assemblies, and as a foundation of virtual prototyping in evaluating the behavior of complex products before any hardware is built.

Benefits are cranked up a notch when such simulation is applied early in the cycle when critical decisions are made regarding the configuration of the product. Through such frontloading of the product development process, engineers can make changes and refine designs easily and inexpensively compared to finding and fixing problems later in development when making changes is far more expensive and time-consuming. Up-front simulation requires changes in the way analysis is performed. Greater cooperation and closer communication are required between the design department and analysis group, for example, as are better data exchange technologies. Such changes make it much easier to do analysis alongside a relatively narrow stretch of design.

Huge gains are possible by taking this a step further and using simulation continuously within the design process throughout the entire product lifecycle - from concept and engineering through prototype testing, hardware production, and after-delivery support. This utilizes the power of simulation as a design tool not only before the project is handed off to production but also as design continues throughout the life of the product as engineering changes are made, often hundreds or thousands of them. This trend allows analysis to be done more seamlessly within routine engineering through the use of optimization trade-off studies, for example, that balance competing requirements such as weight, stress, and manufacturability.

In this manner, simulation technology is used to compress the design cycle and even change the product development process to reduce time and lower costs. The effective use of simulation technology can thus have a significant impact on how operations are performed in the engineering department, where blending the tools into the overall product development process enables engineers, designers, and analysts do their jobs more efficiently. These benefits alone generally are more than sufficient to justify an investment in simulation tools.

The value of simulation for the enterprise extends far beyond this role of the technology as a cost-cutter and time-saver in the engineering department, however. Smart companies are finding that they can actually leverage simulation in changing their overall business focus.

Companies with experience in engineering simulation are using the tools to develop innovative new products, and organizations that formerly never used simulation in their engineering operations are investing in the tools as a way to take advantage of emerging market opportunities, shifts in customer demands, new competitive pressures, and radically different economic conditions.

Manufacturers that fabricate products for general applications might want to penetrate specialized markets such as defense or biomedical, for example, which often demand well-documented results of rigorous analysis that can be provided only with simulation tools. Likewise, subcontractors that once considered themselves strictly parts manufacturers are now shouldering more responsibility for design and find themselves concerned with analyzing stresses, evaluating deformation, and predicting overall product behavior. Simulation also is a critical element in mass customization, design for manufacturability, engineered-to-order product development, and other types of business approaches that represent a shift in the traditional way a company may operate.

Such creative applications of simulation technology requires out-of-the-box thinking, of course, especially from smart executives who today must be on their toes not only in running a business but also in re-engineering the company to take advantage of analysis tools that less astute competitors might not ever have never heard of. For this reason, engineering simulation is now being discussed more in boardrooms than ever before and the technology is assuming a critical role in the strategic plans of some of the world's most progressive companies likely to be leaders in the coming years as many of their less imaginative peers get left behind in the dust.

John Krouse is president of Krouse Associates and editor of the industry newsletter *Engineering Process Journal* covering enterprise-wide strategies for integrated product development. Krouse Associates provides

public relations counseling and writing services for technology-based companies in the CAE, CAD, CAM, PDM, PLM and related markets. Previously with Penton Media for 18 years, Krouse was CAD/CAM editor of *Machine Design* magazine, and editor and publisher of its sister publication *Computer Aided Engineering* magazine.

ANSYS Presents at SG Cowen 31st Annual Fall Technology Conference

James E. Cashman III, President and Chief Executive Officer of ANSYS, Inc., presented a company overview at the SG Cowen 31st Annual Fall Technology Conference in Boston, Massachusetts on Thursday, September 4. A live audio Webcast and archive is available on www.ansys.com/newsrooms/investor.htm.

Siemens Broadens Relationship with ANSYS

ANSYS, Inc. recently announced a deal anticipated to be worth more than $5 million over the next three years with Siemens. The three-year contract includes more than 200 seats of ANSYS software, as well as seats from the ANSYS ICEM CFD Product Suite and services.

This agreement speaks to the long-standing relationship that ANSYS has established with Siemens. ANSYS has provided Siemens with computer-aided engineering solutions for more than two decades. During that time, Siemens implemented ANSYS core simulation and virtual prototyping applications - such as ANSYS Mechanical - within the design process of a large number of different products.

"Siemens relies on ANSYS for its superior engineering software capabilities," said Gerhard Muller, Siemens PG IT. "Incorporating the ANSYS product family with the CAD-systems we are using has allowed us to maximize our design process while containing design costs."

"The various industries that Siemens supplies closely mirrors the markets served by ANSYS," said Joseph Fairbanks, vice president of worldwide sales and support at ANSYS. "Our ongoing relationship with Siemens signals their great confidence in the solutions ANSYS has provided to enhance their product development process. We value our relationship with this global leader, and we will continue to provide the premier technology solutions that they've depended on for 10 years."

Structural Reliability Technology Announces the Release of a 3-D Fracture Mechanics Pre- and Post-Processor for ANSYS

In July 2003, Structural Reliability Technology (SRT), located in Boulder, Colo., released an ANSYS-compatible version of its FEA-Crack software, which is a 3-D crack analysis program with automatic mesh generation. In the past, creating a 3-D finite element model that contains a crack has been tedious and time consuming, but FEA-Crack simplifies this process considerably.

FEA-Crack is a Windows® - based program where the user inputs basic information about geometry, dimensions, material properties, and boundary conditions. The automatic mesh generator then creates ready-to-run ANSYS input files (*.ans). If desired, FEA-Crack can be used as a driver program to run multiple ANSYS jobs in batch mode. The FEA-Crack post-processor module extracts data from ANSYS results files and computes various fracture mechanics parameters including the J-integral, stress intensity factor, crack opening area, and the failure assessment diagram (FAD). These results are presented in spreadsheet tables and x-y plots. FEA-Crack also has 3-D graphics capabilities, which enable viewing meshes, color stress maps, and deformed shape.

Until recently, FEA-Crack analyses were restricted to stationary cracks. However, a re-meshing fatigue crack growth module has been added to FEA-Crack. Other crack growth capabilities are currently in development.

Fatigue Analysis Software Suite Offers Increased Neutrality For FEA Users

Safe Technology announced the release of another FEA interface for fe-safe, further increasing the neutrality of their suite of durability analysis software for finite element models.

fe-safe v5.00-11, is now available with a direct interface to the NASTRAN OUTPUT2 file format. This is a major sub-release of fe-safev5, released by Safe Technology in January. The new interface can be downloaded by NASTRAN users direct from the company's Web site at www.safetechnology.com.

This new development means that fe-safe now uniquely offers direct interfaces to the following FEA file formats:

- ABAQUS .ODB and. FIL

- ANSYS .RST files
- NASTRAN .OP2 and .FO6
- IDEAS .UNV
- BEASY
- Pro/Mechanica
- Hypermesh
- PATRAN
- FEMSYS
- CADFIX
- MTS RPCIII
- Adams

Other new developments also now incorporated in fe-safev5.00-11 are:

Influence coefficients - evaluation of the individual contribution of a unit load on the strains or stresses in a particular direction

Gauge outputs - this facility enables the strains or stresses in a particular direction to be exported

This is in addition to the extended capabilities and substantial increases in speed of analysis and ease-of-use already incorporated in fe-safev5. These include:

- complex loading conditions and advanced multi-axial fatigue capabilities
- fatigue analysis of cast iron
- fatigue analysis of welded structures
- high temperature and creep fatigue analysis
- fe-rotate (quick analysis of axi-symmetric components)
- thermo-mechanical fatigue analysis
- comprehensive signal processing capabilities (safe4fatigue)
- default algorithms (automatic selection of the most appropriate method of fatigue analysis based on the selected material, ensuring ease of use for even the non-specialist engineer).

ANSYS CEO Named to Pittsburgh Technology Council Board Executive Committee

In September, the Pittsburgh Technology Council named ANSYS President and CEO, James E. Cashman III, to the executive committee of its board of directors.

During Cashman's 30 years experience in finance and operations, he has held management positions at Structural Dynamics Research Corporation in the areas of international sales, major account development and production planning. Programs instituted under his supervision resulted in a 678 percent growth in international sales over a two-year period, a 183 percent increase in international sales revenue and profit growth of 190 percent.

Cashman holds a bachelor's degree in mechanical engineering and master's degrees in mechanical engineering and business administration from the University of Cincinnati in Ohio. He was voted *CEO of the Year* during the Pittsburgh Technology Council's Tech 50 awards last year, and he has served on the Council's board since June 2000.

HP Leads in Worldwide Server Revenue in Fastest Growing Market Segments — No. 1 in x86, Linux, Blades and Windows Servers

Driven by continued strong customer demand for its ProLiant servers, HP reaffirmed its No. 1 position worldwide in the markets for x86, blades, Linux and Windows® servers - the fastest growing segments of the server market—both in terms of factory server revenue and unit shipments, according to figures released today by IDC.(1)

HP also continued its strong position in the UNIX® server market and remained No. 1 in terms of worldwide total server shipments, with 30.8 percent market share for the second calendar quarter of 2003.

HP held the following market leadership positions according to numbers released by IDC:

- HP is No. 1 in the worldwide x86 server market, both in terms of factory revenue and unit shipments, driven by its industry-standard Intel®-based HP ProLiant servers;
- Capitalizing on the continued strong growth of Linux, HP held a firm lead in worldwide server revenue for the Linux market with 28.9 percent market share;
- HP ProLiant BL blade servers lead in worldwide revenue and shipments for the x86 server blades market, with 31 percent market share in terms of unit shipments and 32.9 percent of factory revenue (an increase of more than 10 percentage points from the previous quarter);
- Driven by brisk HP Superdome server sales, HP leads in worldwide revenue for high-end enterprise servers (servers priced $500,000 or more) with 29.7 percent. HP also tied for the lead in the UNIX midrange enterprise server market (servers priced from $25,000 to $500,000) with 33.7 percent share of factory revenues worldwide; and
- HP is No. 1 in the worldwide Windows server market segment with 33.6 percent share of factory revenue, based on record ProLiant server sales worldwide during the second calendar quarter of 2003.

"HP ProLiant systems remain, quarter after quarter, the world's best-selling industry-standard servers, which shows that customers demand standards, value and innovation," said Scott Stallard, senior vice president and general manager, HP Enterprise Storage and Servers. "Together with the new 64-bit Integrity systems, HP offers the strongest industry-standard based server lineup of any vendor in the marketplace today."

From the industry-standard ProLiant and Integrity servers, to the high-end Superdome and NonStop systems, HP offers enterprise customers the broadest server portfolio in the industry, along with the industry's leading storage and management solutions.

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

For the second consecutive year, ANSYS, Inc. has been named in *Business 2.0*'s B2 100, the magazine's ranking of the fastest-growing technology companies. ANSYS ranked 81 on the list, climbing five places from last year. The list of 100 was winnowed down from an original group of 2,000 publicly traded tech firms.

To make the B2 100, companies had to meet rigorous financial requirements. Criteria for making the final list included at least three years of trading on a major U.S. stock exchange, at least $50 million in annual revenue, and positive cash flow over the most recently reported 12 months. *Business 2.0* editors then ranked the companies with the help of Zacks Investment Research of Chicago, using a combination of four financial criteria: growth in revenue, profit, and operating cash flow during the past three years, and the 12-month stock return.

Cash flow growth counted for 40 percent of a company's ranking; each of the other criteria counted for 20 percent.

In announcing this year's list, *Business 2.0* editor Josh Quittner observed that "companies on the 2003 B2 100 have shown remarkable results in a most challenging business environment. But what the list shows is who to watch, who to watch out for, and what we can learn from the leaders. Technology is alive and well, as proven by the performance of these companies."

"Manufacturing continues to be an extremely competitive field where time and cost are both crucial factors that can make or break a product. ANSYS is providing technology that helps manufacturers compete more effectively, by accelerating product introduction, and driving product innovation," said Jim Cashman, president and CEO at ANSYS, Inc. "Our simulation and virtual prototyping tools let you measure design performance directly on the desktop at any stage in the product lifecycle. We've succeeded because our technology works and because we can easily demonstrate its benefits to the bottom line."

The B2 100 is featured in the October 2003 issue of the magazine, on newsstands September 22.

Upcoming Events

ANSYS Seminar Series
On-going
Locations throughout North America

ANSYS CFX, UK User Conference 2003

12 November 2003
Coventry, UK

German CFX Conference 2003
4 - 6 November 2003
Garmisch-Partenkirchen, Germany

PowerGen 2003
9-11 December
Las Vegas, NV

Nordic Update Seminar
30 October 2003
Helsinki, Finland

Nordic Update Seminar
31 October 2003
Copenhagen, Denmark

Nordic Update Seminar
3 November 2003
Oslo, Norway

Nordic Update Seminar
4 November 2003
Stockholm, Sweden

Benelux User Meeting
5 November 2003
Breda, The Netherlands

French User Conference
6-7 November
Beaune, France

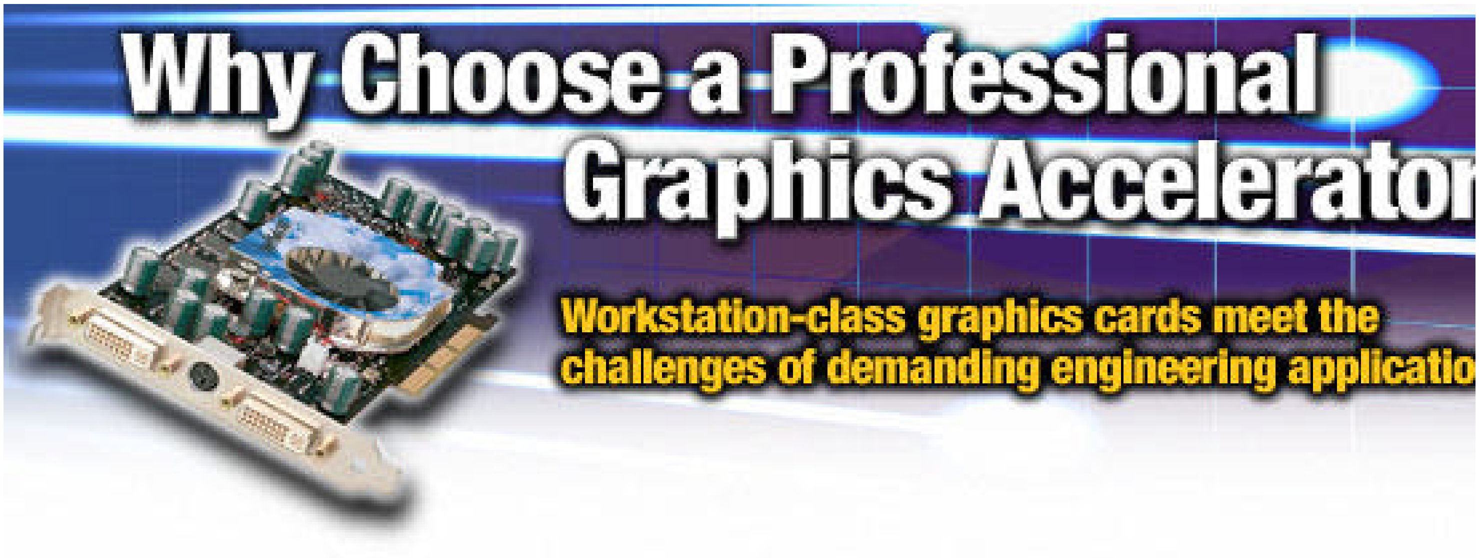
Spanish User Conference
10 November 2003
Madrid, Spain

UK User Conference
11-12 November 2003
Coventry, UK

21st CAD-FEM Users' Meeting
International Congress on FEM Technology
12-14 November 2003
Potsdam/Berlin, Germany

2003 Japan ANSYS Conference

20-21 November 2003
Crowne Plaza Metropolitan
Tokyo, Japan



By Brian Yu
Marketing Manager
3Dlabs

Professionals use applications like ANSYS to increase productivity. Similarly, a professional's hardware also should improve workflow. The core components of a Windows or Linux-based workstation include a motherboard with support for the latest components, large amounts of memory, a fast CPU, ample hard drive capacity, and most importantly, a robust graphics accelerator. A workstation-class graphics accelerator – one used by professional designers and other engineering professionals – is required by demanding applications such as ANSYS to provide the latest features; offer a stable work environment; be seamlessly compatible with other software, and have powerful performance. In this way, the graphics card should be a transparent addition to project workflow, allowing users to work without having to worry about hardware performance or driver stability. With a workstation-class graphics card, one that has been built from the ground up with professionals' needs in mind, users need not worry about overloading their system and potentially losing hours of work.

Unlike cards adapted from PC games, professional-grade graphics accelerators such as the Wildcat VP990 Pro card from 3Dlabs are developed to meet demanding analysis and engineering applications running on technical workstations.

Workstation graphics accelerators are designed from the start with the intent of improving the overall user experience for the customer with faster throughput and efficiency. Features like image quality and stability are paramount, whereas graphics cards originally designed for playing computer games on PCs (unlike true professional cards) are designed with one thing in mind - speed. With gamers begging for more frames per second (FPS), gaming card manufacturers (and those who manufacture so-called professional cards based on gaming technology) take shortcuts and cut corners on rendering quality in order to hold the title of the fastest card in the industry. That distinction in no way helps the professional user, as pixel dropout is usually the result at higher FPS. Occasionally, the gaming enhancements cause professional applications to stutter and sometimes stall. Consumer cards are not designed to take on the heavy workloads that workstation applications impose on the graphics processor. Professionals in CAD and CAE disciplines cannot afford to sacrifice image quality and stability for the sake of speed.

Professional graphics accelerators are engineered to handle the rigorous demands of professional CAD and CAE software. The models associated with these demanding applications require a high-level of hardware stability to ensure users productivity and efficiency. Qualities such as visual accuracy and precision, speed and performance, and stability are examples of what to look for an in a graphics card – the very qualities that set professional accelerators apart from the gaming-bred variety. Add industry leading technology, rock-solid drivers, and relevant, professional software certifications to that list, and you'll get the card that can hold up under the stress of demanding simulation applications, such as the Workbench Environment or ANSYS.

Brian Yu is a Marketing Manager at 3Dlabs, the only workstation graphics company solely focused on designing professional-grade graphics accelerators to run graphics-intensive applications. These professional graphics accelerators go through rigorous testing, both internally and with application vendors, to ensure seamless compatibility with the host hardware, the operating system, and other concurrently running applications. Information on the company and its products can be found at www.3dlabs.com.

ANSYS Helps Tata Reduce Front Axle Weight by 40%

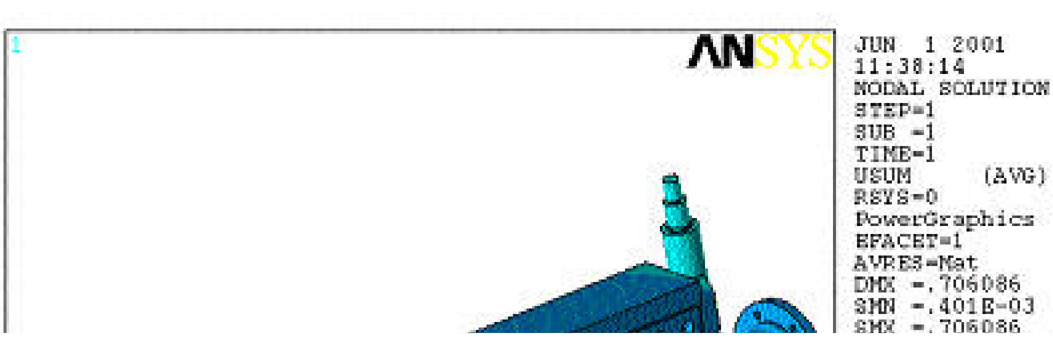
Tata used ANSYS Mechanical to design a new front axle for a commercial tractor, equal in strength and reliability to the existing one, but with a significant reduction in weight.

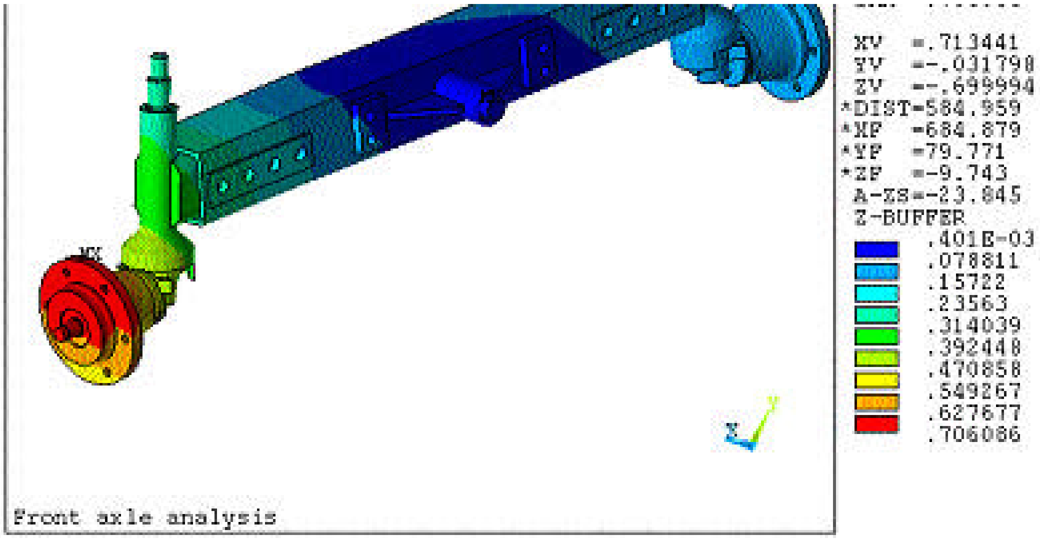
Introduction

The rapid developments in numerical simulation techniques, faster computing ability, and greater memory capacity, are allowing engineers to create and test industrial equipment in virtual environments. Through finite element analysis (FEA), these sophisticated simulations provide valuable information for designing and developing new products, as well as perfecting existing ones. Manufacturers have found this method eminently useful, as it helps them to achieve better productivity at a lower cost per unit, and develop engineering components that are easy to manufacture, and which make the most economic use of their materials.

Tata Consultancy Services (TCS), of Mumbai, India, is at the forefront of FEA. Dilip K. Mahanty, Ph.D, Group Leader of the Design & Analysis Center of TCS' Engineering Services Group, and an expert in FEA, has used this method to perform numerous design evaluations and modifications on components for everything from household appliances to locomotives. In order to attain the utmost accuracy, Dr. Mahanty always uses ANSYS, a leader in Finite Element Analysis software.

International Auto Limited (IAL), one of the foremost automotive component and aggregate systems manufacturers in India, supplies sheet metal stampings and precision machine parts to various original equipment manufacturers throughout the country.





Challenge

IAL's objective was to find a way to reduce the weight of the front axle of a commercial tractor. Since the tractor had never been known to fail in the field, the design of its front axle was to be used as the basis for the axle of the new vehicle. As the front axle undergoes the heaviest load conditions of the tractor, it would take some intense testing and pinpoint modifications to equal the success of the existing axle, while making it lighter.

Mr. S.S. Udgata, Research & Development Manager for International Auto, Ltd., called Dr. Mahanty to collaborate with his team in the evaluation and modification of the design for the new tractor.

Solution

Once again, Dr. Mahanty turned to ANSYS to solve the problem. The team members devised a series of 13 grueling certification tests, including seven major cases, and six sub-cases, through which they would put a geometric model of the front axle of the tractor, and use FEA/ ANSYS to evaluate it under different load conditions.

They built the model in Unigraphics, a CAD package, and then transferred it to ANSYS. Using HP Workstation C3000, each iteration took about 30 minutes to solve the finite element model for the given load and boundary conditions. ANSYS Mechanical software was used for pre-/ post-processing and solving.

First, they performed the Drop Test, wherein a 35/55-hp tractor was run on a hard course. At 35 kmph, the tractor approached a pit measuring 5 x 2 x 2.5 feet, and one of the front wheels was allowed to fall into it. The tractor was pushed, under its own power, to the end of the pit, where it stayed until the engine stopped.

Next came the five-phase Torture Test, in which the tractor was put on level and V-shaped test tracks that simulated various unfavorable road conditions, including pits (potholes), small humps, and large humps.

During alternating segments of this test, both front wheels fell into the pit; one wheel was in the pit, while another went over a hump; one was on a plane, while the other was on a slope; then, it was operated on a V-shaped road with small and large humps.

During both phases of the Wide Open Test, the front axle extenders were fully extended, while the tractor ran, first with one wheel on a slope, and the other on a plane, then on the V-shaped road with a hump, at 15 kmph.

For the Pit Test, the tractor went down one side of a 10-foot-deep pit with a 20-degree slope, and up the other, at 30 kmph.

In the Eight (8)-Shaped Track Test, the tractor ran at 35 kmph on an 8-shaped track, with three medium-sized humps positioned at 120o to each other in each half of the track. While negotiating a curve, the steering wheel was turned until it reached its locking position.

The Impact Test had one side of the tractor crashing into a wall at 35 kmph.

In the Worst Load Test, all of the worst load conditions were applied to the axle, and then analyzed.

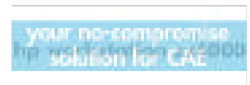
The results of the 13 load cases on the current model indicated that the structure was safe overall. Although there were a few high-stress areas, it was apparent that they were localized, since the stresses died down within an element or two. These results were used as the basis for comparison with the five proposed models that had evolved, each of which emphasized weight optimization and easy manufacturability.

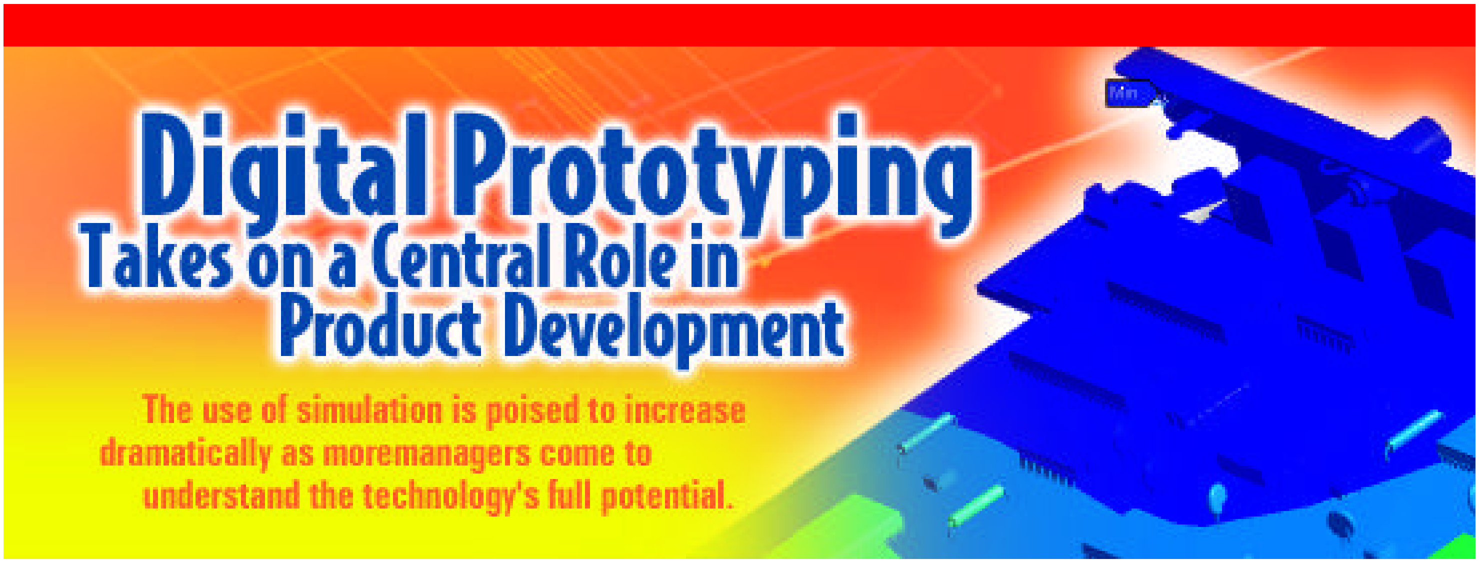
The proposed designs were then evaluated under the same selected worst load cases as the existing design. The analyses of all five models yielded displacements and stresses close to those in the existing design. The displacement increases were insignificant, while the stress increases were close to 15 percent. All of the new designs met the structural requirements. Ultimately, the U-box with extender was chosen as the best of the proposed models.

Benefits

Not only did the proposed designs have a weight reduction of approximately 40%, but they could also be produced without much welding, which meant marked savings in manufacturing costs. The need for smaller components, such as bearings and knuckles, also made them considerably less expensive.

Dr. Mahanty says, “This analysis work showcases the use of finite element analysis as a method for reduction of cost in terms of materials and manufacturing. The benefits provided are many. We could change the design and see the impact in a very short time, which helped in fixing a feasible design. Also, due to analysis results obtained from ANSYS, it was very much possible to check the design parameters, such as maximum stress level limits. And the reduction in the cost of production and weight significantly reduced the cost of the new design of the front axle.”


your no-compromise
solution for CAE



By Charles Foundyller
President
Daratech, Inc.

Digital prototyping and simulation have greater near-term potential than perhaps any other class of information technology to help manufacturers in a wide range of industries compete in today's environment. Developing best-selling products at lower cost, on shorter schedules, at higher quality—few manufacturers we talk to believe these imperatives can be met without putting digital simulation and digital performance models at the core of product development and validation.

To meet these urgent business goals, the world's leading manufacturers are using digital simulation technology in conjunction with work-process restructuring to shorten product development cycles, manage increasing product complexity and variety, lower development costs by reducing physical prototype counts, better control subjective product attributes that create brand value and buyer appeal, and reduce warranty exposure, recalls and product failures.

Helping accelerate the move to math is the dramatically improving price/performance of high-performance computers, together with higher-quality, better-managed deployment, implementation and training. Also key is growing confidence among both practitioners and management in simulation accuracy and correlation with physical test results. However, bringing simulation tools and processes into the mainstream of product development, and making their capabilities and results available to support every decision point—what some describe as simulation-based design—remains supremely challenging, even at companies where program managers, discipline specialists and designers have all committed to making such initiatives succeed.

Even more important than shortfalls in the technology, we believe, are the management challenges for manufacturers. Program managers, department heads and engineers alike repeatedly describe how the greatest barriers to broader, more effective use of simulation are not technological but instead organizational, cultural and psychological. Indeed, the culture clash encountered by digital prototyping's advocates is summed up in an adage some of these pioneers like to repeat: "Everyone believes the test results except the test engineer—and no one believes the analysis results except the analyst." What most everyone agrees on is that the capabilities offered by today's technologies are substantially ahead of manufacturers' ability to configure their product development processes so that the full potential of these technologies is realized.

Our best-practices research among the industry's most successful implementers of digital prototyping indicates that, to remedy this, the first step is for management to gain a sound understanding of the capabilities as well as limitations of today's technologies, and put in place organizational structures and processes that exploit what is available today while also having the flexibility to continue adopting new, improved capabilities as they become available.

Management also needs to take a leading role in fostering cooperation and collaboration among divisions, departments and disciplines that may have had little or no direct contact with one another in the past, and that often resist change and new ways of working. In this regard, management can profit by leading the effort to build and implement process and platform frameworks that coordinate and tie together different groups and organizations, and then extend these frameworks to be increasingly inclusive of suppliers and partners. Here, best practices have demonstrated that bringing in new technologies can be used to aid restructuring of work processes, and serve to incent and involve people in the new ways of working.

One notable gauge of digital prototyping's increasingly central role is the significantly higher growth expected for this market than for mechanical CAD/CAM. By our forecast, expenditure on digital prototyping and simulation will grow an average of 11.5 percent annually to nearly $2.5 billion by 2007. By contrast, mechanical CAD/CAM expenditure is projected to increase only 3 percent annually over the same period, to $5.1 billion.

Moreover, the breakout segment of digital prototyping that we term systems performance modeling will enjoy dynamic growth of 14 percent annually to more than $1.4 billion by 2007, we project. Systems performance includes CFD, crash, dynamics and motion, forging and mold design, durability and fatigue, heat transfer and thermal interactions, NVH, control systems design and the like. The other main market segment, structural analysis, is essentially the traditional CAE domain—solvers plus pre- and post-processors for structural finite-element analysis. We project structural analysis expenditure will grow 6 percent annually to nearly $1 billion by 2007.

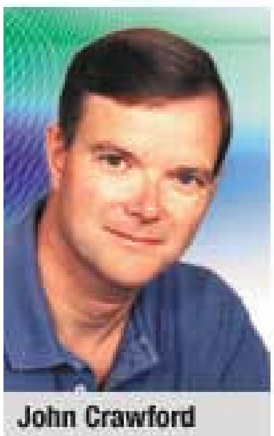
Another mark of digital prototyping's ascendancy is seen in the automotive industry, which traditionally leads others in the use of simulation technology. We've seen that in automotive vehicle programs, expenditure on physical test is declining, while expenditure on digital prototyping and simulation is on the increase. In 1985, most customer investment in the test, measurement and verification domain was on physical prototyping and test. In 1995, manufacturers were still investing heavily in physical testing. But today, many organizations are spending as much on digital prototyping and simulation as on physical test and measurement. While total spending on digital and physical methods together has not changed dramatically, we believe the total is poised to increase as more executives and managers come to understand the full potential of digital prototyping and simulation, and the technology continues to meet more and more of practitioners' critical needs.

Charles Foundyller is founder and president of Daratech, Inc., a market research and technology assessment firm specializing in CAD, CAM, CAE, PLM and related areas. The company hosts a variety of user forums and technology workshops throughout the year and publishes a range of studies, newsletters, market reports, sourcebooks, and industry statistics. Daratech's upcoming management-level conference on Digital Product Simulation Strategies for Aerospace and Defense will take place Nov. 3-4, 2003, in Anaheim, Calif., focusing on reducing costs, accelerating schedules, and improving quality and performance through strategic application of digital simulation and validation technologies. For more information, call at 617.354.2339 or visit www.daratech.com.

Guidelines for Good Analysis

A step-by-step process for obtaining meaningful results

By John Crawford
Consulting Analyst



Every so often I have the opportunity to introduce a new analyst to finite element analysis. I usually begin by presenting a framework for how one views engineering analysis in general and finite element analysis in particular. I then talk about the steps one goes through when doing an analysis and interpreting the results. It is important to think about the entire process up front because it's very easy to get wound up in the details of doing an analysis and lose sight of the big picture.

The list below outlines the steps that I follow in my daily work and that I share with others who are just getting started with finite element analysis. While I have used examples and terminology from structural problems, these guidelines listed are equally valid for heat transfer, electromagnetics, CFD, etc.

1. **Thoroughly understand the actual problem.**
   The first step in any analysis is to understand the problem. Don't accept someone else's interpretation of the problem. Look at the components and figure out for yourself how it works and what the real issues are. You'll know that you understand a problem when you can successfully explain it to someone else. If you can't explain it to another person, chances are good that you don't understand it yourself, and if you don't understand the problem you certainly aren't going to be able to analyze it properly and understand whether your answers are correct or not.

2. **Predict what you think the answer will be.**
   Once you understand the problem you should try to estimate what you think the answer will be and how the system will behave. Identify regions where you think high stresses will occur, estimate what the deflected shape of the structure will be, and so forth. Develop an image in your mind of what the component will look like after loads are applied and use this to determine how you'll set up the model. More than another other step in the analysis process, this one depends on your engineering intuition to

lead you in the right direction.

3. **Decide if finite element analysis is a reasonable method for analyzing this problem.**
While finite element analysis is a very powerful tool, it isn't the only way of analyzing things and sometimes isn't the best way, either. Some problems are solved more efficiently using classical techniques and others are best understood via experiment. Make sure that finite element analysis is appropriate and reasonable before you progress any further. If you can find a better way of solving the problem, use it.

4. **Determine the type of analysis needed to obtain reasonable answers.**
This is the most crucial part of the analysis process because you will make almost all of the critical decisions that will define the path you will follow as you make the model, solve it, and postprocess the results. The real world is three dimensional, transient, and nonlinear, while the FEA world almost always involves some simplification of one or more of these. Is a static analysis sufficient, or is a transient analysis necessary? Will you need to do a heat transfer analysis to obtain a temperature distribution before you do the stress analysis? Are nonlinear material properties needed? Can you take advantage of symmetry to reduce the number of elements in the model? Will an axisymmetric model provide satisfactory results? What kind of meshing techniques are best suited for this geometry? Will you use free meshing, mapped meshing, sweep meshing, or a combination of these? If dissimilar meshes are unavoidable, will you use constraint equations or bonded contact elements to tie these regions together? How will you apply boundary conditions? By answering these questions you will define a blueprint for how you will do the rest of your work.

5. **Determine the type of elements you will use.**
Once you have decided on an approach you will need to choose the elements that you will use to obtain the desired results. This is why it's so important to fully understand the problem and visualize how the components will behave. As an example, let's consider a cylinder that is fixed at one end and has a load on the other end that causes bending. We can model this several ways. The easiest and simplest way is to use pipe elements (such as PIPE16). These elements will do a very good job of simulating the way in which the cylinder will bend, but they assume that the cross section of the tube does not change shape. If we model the tube using shell elements (SHELL63, SHELL93, etc) we'll be able to see if the tube changes cross section from circular to elliptical. Moving another step closer to reality, we could use solid elements (SOLID45, SOLID95, SOLID92, etc) to see if the wall of the cylinder changes thickness. Depending on what we think is important in our problem we can choose the best way of getting an answer that we believe will satisfy our needs in an efficient and effect manner.

6. **Determine the geometry needed to generate the elements.**
The geometry you'll need to generate a mesh depends on the elements you have chosen and the techniques you will use to mesh the model. While everything in the real world is a 3D solid, the FEA world isn't necessarily 3D or solid. In the case of the previously mentioned tube, if we are using pipe elements we'll only need a series of lines and arcs that define the centerline of the tube. If we are using shell elements we have two possible paths to follow. One would be a series of lines that defines the

center line of the tube and a circle at one end that we can drag down these lines to generate areas. Another is to make or import areas and skip the dragging operation. If our tube will be meshed using 3D solid elements we could map, sweep, or free mesh a volume, or we could use a center line and drag a 2D ring down it and generate the volumes and mesh it at the same time. The geometry you need is a function of the elements you will make. You might also find that you need to move the geometry to a particular location to better suit how you plan to analyze it. As an example of this, when using axisymmetric elements, ANSYS requires the global Y axis to be the axis of symmetry and the elements must be located on the Z=0 plane and on the positive side of the X=0 plane.

7. **Create the geometry within ANSYS or import it from another source.**
   Once you have determined the geometry you will use to generate the mesh, you can create the geometry within ANSYS, import it from another source, or do both. The path you choose depends on the complexity of the model and whether another source of geometry is available. For relatively simple geometry it might be faster to generate it within ANSYS. Complex geometry might be better made in a CAD program and imported into ANSYS via IGES, one of the Connection products, or a third party translator like CADfix. If you are importing geometry from another source you may have to alter it to suit your needs. You may only need a segment of the geometry, a planar cut through 3D geometry, or nothing more than a series of lines. You may also choose to remove details from the geometry that you think are insignificant and would add unneeded complexity to the model. Sometimes the CAD geometry is missing features we believe are important, like fillets on inside corners. One thing for sure is that the CAD geometry frequently needs to be modified to make it suitable for analysis. We might alter the CAD geometry to allow us to take advantage of symmetry or other simplifications, or we may subdivide the geometry so we can apply boundary conditions properly or use certain meshing schemes. One idea to keep in mind is that it is usually easier to mesh a group of smaller, simpler geometries than it is to mesh a single, more complicated geometry. Plus, if it is decided at a later date to make a modification to the geometry and rerun the analysis, having a number of volumes means that you'll only have to clear and remesh a small part of the model instead of the whole thing.

8. **Create the attributes needed to define the elements.**
   All elements in ANSYS are defined by their attributes, which are the tables that contain the information that describe the element and it's behavior. There are five type of attributes; TYPE (defines the element type), REAL (defines physical constants), MAT (defines material properties), ESYS (defines the coordinate system the element is aligned with) and SECNUM (defines cross section information). Usually, 2 or 3 attributes are all that are needed to define most elements. It's convenient to assign attributes to each geometry entity you will be meshing because they are automatically applied to the elements as they are generated. It also allows you to remesh the geometry without having to worry about which attributes are currently active.

9. **Set element sizes.**
   Assign what you think are realistic values for the element edge length in various regions of the model. Use your prediction of how the model will behave to help you

determine where the elements need to be small enough to obtain accurate results and where they can be large and still provide reasonable answers. Another factor to consider when setting element size is the geometry and whether ANSYS will be able to mesh each region successfully. You may find it beneficial to adjust the element size in certain regions to improve the likelihood of successful meshing. Another meshing parameter is the rate at which element size increases from the outside to the inside of the model. It is common to have larger elements in the middle of areas and volumes because high stresses usually occur on the outside surface. If you think that the number of elements in your model might present a problem during solution, you can increase the rate at which elements increase in size from outside to inside.

10. **Mesh the geometry and create any other elements that are needed.**
Meshing can be as easy as executing a single command, or as time consuming as almost any other part of model building. You can use mapped meshing, sweep meshing, free meshing, or explicit element generation. A commonly followed procedure is to begin with mapped meshing and then use sweep meshing and free meshing as needed. This is followed by generation of special elements like contact elements, point mass elements, spring and damper elements, surface effect elements, and so forth.

11. **Apply boundary conditions.**
Boundary conditions can be applied to solid model geometry or directly to nodes and elements. It is good practice to apply boundary conditions to solid model geometry whenever possible in the event that you might want to remesh part or all of a model later on. Not all boundary conditions can be applied to solid model geometry, so it is common for a model to also have boundary conditions applied directly to nodes and elements.

12. **Set the load step controls.**
There are a number of solution controls that can be set to enable a more efficient or more accurate solution. You may wish to choose a specific solver for your problem, or you might control the time step size or the amount of data that is written to the result file. You can also control the number of substeps that will be solved in a given load step and much, much more.

13. **Write the load step files.**
After you have applied boundary conditions and defined the controls for a given load step you can write a file that contains this load step information. It’s not always necessary to write a load step file, especially if you only have to solve a single load step for your problem. One benefit of writing a load step file is that it acts as a record of the boundary conditions and solution settings used to run the analysis. You can open a load step file with an editor and see all the ANSYS commands that control the analysis, which is a handy way of making sure that the boundary conditions really are what you think they are. Another benefit of using load step files is that you can rerun the analysis using a given load step file and be sure that you have exactly the same boundary conditions and solution settings as before.

14. **Solve the load step files.**

Load step files can be solved either individually or as a group using the LSSOLVE command. You can also choose to solve the currently applied boundary conditions and solution settings using the SOLVE command. During solution it is frequently beneficial to keep an eye on the output window and see how the things are progressing. Depending on the analysis being done, ANSYS may plot the convergence criteria and how the solution is converging in the graphics window. If you watch the available solution output while the program is solving you can occasionally detect and diagnose problems that may occur during solution.

15. **Review the results.**
    Look at the results and see if there is anything obviously wrong. Are all the load steps that you thought were being solved present in the results file? Are there any obvious errors in the results? Do the results compare favorably with your understanding of the problem?

16. **Interpret the results.**
    All too often the postprocessing of finite element results is done quickly and with hardly a second thought, but one of the most important steps in the analysis process is to look at the results and interpret what they really mean. If a singularity is present in the model do we ignore it or do we modify the model to include the real world geometry at this location? Another question we must ask ourselves is whether the mesh is refined enough to provide answers that are accurate enough for our needs? By viewing the averaged results, the unaveraged results, the Powergraphics results, the full graphics results (along with SMXB values), and the estimated error, we should be able to determine whether the mesh is adequately refined and what the real answer might actually be.

17. **Compare the results to your original prediction.**
    When you look at the finite element results you should ask yourself if they make sense and appeal to your understanding of how the system works. Are the highest stresses in regions that seem reasonable? Are the answers close to what you initially thought they would be? This is a vitally important part of the analysis process because reviewing each result and comparing it to what you thought it would be will help you sharpen your engineering intuition. The intelligent analyst will always try to fit the answers he is seeing on the screen into his understanding of how things work. By doing this for each analysis you will become a better engineer and a more valuable and productive analyst.

18. **Iterate as needed to obtain a satisfactorily accurate answer.**
    What are the odds that your first answer is sufficiently accurate? If you have done a good job of interpreting the results you will have a reasonable idea of how much error is present in your results. Use your estimation of the actual stress values as the final answer from your analysis. When you present results you should present the values that were calculated in ANSYS, your interpretation of what they really mean, how much error is included in them, and what your final estimate actually is.

While all of the steps are important and must be done properly to obtain valid answers, the most difficult and crucial ones from the list above are 1-3, and 16-17. These require engineering insight and understanding. You must understand the problem, use your intuition and experience to predict the behavior of the system and the answer you area likely to obtain, determine the finite element representation that will give you this answer, and interpret the results of the analysis into a sensible and accurate engineering assessment of how the system behaves.

Perhaps the most interesting step is 17. By comparing the finite element results to what you thought the result would be, you will revisit your understanding of the problem and see if you have overlooked something. If the results are not what you thought they would be, there are three possibilities that might explain this. One is that you made a mistake when doing the analysis and the problem you solved was not the problem you actually intended to solve. Another is that the problem was set up properly, but ANSYS did not analyze it correctly. Such occurrences are few and far between, but no program is perfect and there is a small but finite chance that you may have run across a problem with the program.

Take a look at the Class 3 error reports and see if anything has been reported that could explain why your answers don't look quite right to you. Finally, if the results from your analysis do not compare favorably with your understanding of the problem, maybe your understanding of the problem was incorrect or incomplete. As you look at the results and think about things, sometimes the light bulb goes on and it all suddenly makes sense. These moments of enlightenment are one of the highlights of being an engineering analyst.

Using SECFUNCTION to Define Varying Shell Thicknesses

Specify thickness as a tabular function instead of entering a separate real constant for each element.

By ANSYS, Inc. Technical Support

Q: Is there an easier way to define varying shell thicknesses than entering a separate real constant for each element?

A: A simpler method is to use shell sections instead of real constants. A more general way of defining shell construction than the real constants option, shell sections allow the user to define the thickness of a set of elements as a function of X,Y, and Z.

The thickness of elements SHELL131, SHELL132 and SHELL181 can be defined by a section. Shell section commands allow for layered composite shell definition and provide the input options for specifying the thickness, material, orientation and number of integration points through the thickness of the layers.

Note that shell sections provide additional flexibility in defining single-layered shells as well as layered composites. They allow the use of the ANSYS function builder to define thickness as a function of global coordinates and the number of integration points used.

Shell section data is defined by the SECTYPE, SECDATA, SECOFFSET and SECFUNCTION commands:

- SECTYPE,SecID,Type,Subtype,Name,REFINEKEY associates the section type information with the section ID number. SecID is the section identification number; Type is set to SHELL to specify a shell section (other types of sections are BEAM and PRETENSION). Subtype only applies to type=BEAM, Name is the name of the section and is optional, and REFINEKEY only applies to type=BEAM.
- SECDATA,TK,MAT,THETA,NUMPT describes the geometry of the section. TK is the thickness of the shell layer for constant layer shells; MAT is the material ID for a shell layer. Theta is the angle of the layer element coordinate system with respect to the element coordinate system and NUMPT is the number of integration points in a layer.
- SECOFFSET,Location,OFFSET1,OFFSET2,CG-Y,CG-Z,SH-Y,SH-Z defines the

section offset for cross sections. More details on this command can be found in the ANSYS Commands Reference.

- SECFUNCTION,TABLE specifies the shell section thickness as a tabular function. TABLE is the table name reference for specifying tabular thickness as a function of global XYZ coordinates. To specify a table, enclose the table name in percent signs (%). Use *DIM to define a table before using this command. This table defines the total shell thickness at any point in space, and can be entered by the user, or defined by ANSYS through the Function Builder.

The table used by SECFUNCTION can be created in three ways: entered as a TABLE in ANSYS, as a table imported from a spreadsheet, or as a function created in the Function Builder. Building the table explicitly requires you to specify thickness values for different points in space, and ANSYS will interpolate thickness values for locations not explicitly defined. The Function Builder allows you to specify the thickness as a mathematical function of X, Y and Z entered into a calculator format. It also lets allows you define equations for different regimes, e.g. X<0 or X>0.

The function builder is accessed through the GUI by choosing Utility Menu>Parameters> Functions>Define/Edit. Choose either a single equation or a multi-valued function. If you choose multi-valued function, you enter the name of the regime variable. For both methods, enter the equation using the primary variables (in this case, X, Y and Z) and the keypad. If you are using a multi-valued function, click on the different Regime tabs on the Function Builder and define each regime and the equation for that regime. When you are finished building your equation, choose Editor>Save, and save the function file.

After you have built your function, you need to load it in (building it does not make it active). Choose Utility Menu>Parameters> Functions>Read from File and browse to your function file. You will then need to enter a table parameter name. You will use this name when you specify your function for the shell thickness. Save your new table. The function is saved as an encoded table, and ANSYS automatically dimensions the table.

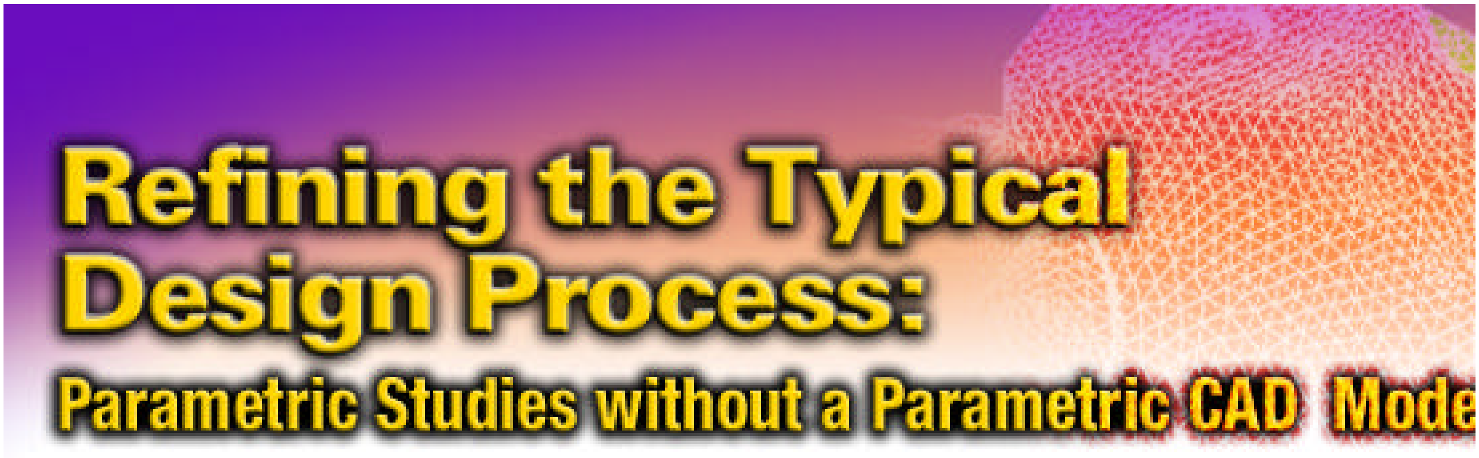
If you need to define your thicknesses as discrete values instead of a function, you first need to define a table parameter, either by the *DIM command, or by choosing Utility Menu>Parameters> Array Parameters>Define/Edit. Be sure to identify the variables for the table, or the shell thicknesses will not be read. If the table rows correspond to the x-coordinate and the columns the y-coordinates, be sure to specify that variables 1 and 2 are X and Y, respectively.

The table can be entered in the GUI by choosing Utility Menu>Parameters>Array Parameters>Define/Edit, or by using *SET or by importing the table. You can define your table in Excel or in comma delimited format, then read it in either by Utility Menu> Parameters>Array Parameters>Read from File or by using *TREAD. This command reads the first element of the table as the first value of variable 3, then reads the rest of the first row as the values of variable 2, and the rest of the first column as the values of variable 1. So, if your variables are defined as X, Y and Z in order, you would set up the first line of your table as: z1, y1, y2, y3, y4, etc. The second row of your table would be x1, thickness(x1,y1,z1), thickness(x1,y2,z1), thickness(x1,y3,z1), etc. When you have defined all of the values corresponding to z1, your next line would be: z2, y1, y2, y3, y4, etc., and each line following would have the thicknesses for each of the x and y locations for that value of z. If you are using the same X and Y values for each value of Z, you still need to list their values for each value of Z. An example follows:

0, 0, 5, 10
0, .1, .15, .2
5, .1, .17, .21
10, .1, .18, .22
5, 0, 5, 10
0, .2, .25, .3
5, .2, .27, .31
10, .2, .28, .32
10, 0, 5, 10
0, . 3, .35, .4
5, .3, .37, .41
10, .3, .38, .32

The X, Y and Z values are all 0, 5, and 10. The first four rows define the thicknesses for X and Y values at Z=0. The following four rows define the thicknesses for X and Y values at Z=5, and so on. Each value of Z has a 2D table associated with it, with Y values in the first row, and X values in the first column. Even though the first row and first column of each subtable is the same, you still need to define these values for each value of Z.

Once the function table is defined, the shell section is defined. If using the GUI, choose Main Menu>Preprocessor> Sections>Shell>Add/Edit. In this window, choose the material associated with the shell section, and choose the function, then save the section. If using command format, use SECTYPE, SECDATA, SECOFFSET and SECFUNCTION. After you have defined your section(s), you can assign the appropriate section ID to areas or elements during meshing the same way as you assign real constants. You can verify that your thickness function was done properly by choosing Utility Menu>PlotCtrls>Style>Size and Shape, or use the /ESHAPE command. ANSYS will create a 3-D visualization of your shells.



New Design Modification Tool Works Directly on an Existing Mesh

[Steve Pilz](#)
ANSYS Inc.

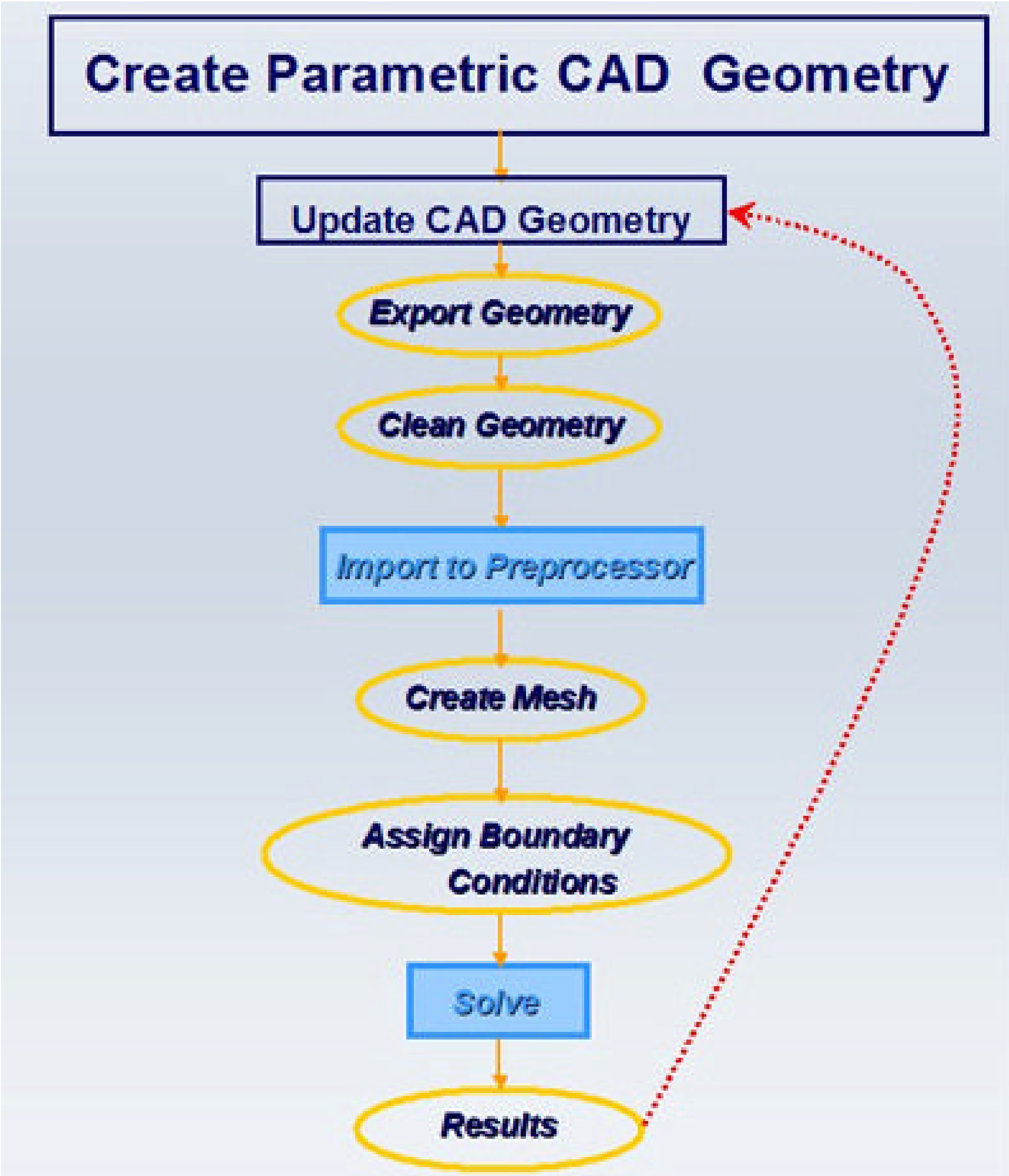
[Jean-Daniel Beley](#)
CADOE S.A.

What is the optimal shape for your design? How do you find this shape? What tools do you use? How many people need to help you? Critically important — How long does it take to get to this final shape?

The typical design verification process is iterative and requires multiple people with different software systems to drive it. The process starts with a CAD model that was designed with intuition and user-accumulated experience. The CAD model can be parametric, or non-parametric, which we'll explore below. An instance of the CAD model is then exported and provided to the finite element analyst. A well-known loop then begins: geometry cleaning (geometry healing, feature suppression, geometry abstraction), meshing, boundary conditions, solve, post-processing, and finally, delivery of results to validate or invalidate the design. If the design is not acceptable, the analyst goes back to the designer for a discussion. If agreeable, the CAD model is then modified to take on a new shape. The design/analysis loop iteration is then repeated.

With a well-constructed and robustly rebuildable parametric model, and close cooperation between the design and analysis engineers, this iterative process can work well and has been serving product development companies for 30 years. But it's an expensive and time-consuming process that's ubiquitous enough that there is no distinct competitive advantage to using it.

The traditional design verification process is an iterative loop that's expensive and time-consuming.



The traditional design verification process is an iterative loop that's expensive and time-consuming.

Time Sinks

This traditional process has inherent time sinks:

- Building a fully parametric, rebuildable and exportable CAD model.
- Rebuild CAD model for each verification driven iteration. CAD models, when fully parametric, aren't normally very forgiving of analysis lead design change suggestions.
- Data exporting and importing iterations. A data import problem — in one iteration — is likely a problem for all process iterations.

While examining these and other time sinks built into the traditional process, two kinds of CAD models have to be considered: parametric models and non-parametric models. With a fully parametric model, modifying one dimension can be straightforward if the model has been built so that each modification leads to a valid model. However, building a fully parametric design is always a hard task that is very time consuming and expensive. Many companies, despite the intentions of their aggressive plans, find that in practice, it is entirely too expensive to build fully parametric models all the time. Modifications of the geometry such as mid surface extraction are seldom part of the CAD history tree, for instance, and often the parameters controlling the shape or function of the design are not created with future design simulation in mind.

Two other cases make it difficult to go back and forth between CAD and FEM. First, when using legacy models, CAD is not always available and even less often parametric. Second, getting robust parametric models for large assemblies often is difficult, especially when different divisions within the same company design the parts or when parts are coming from different suppliers.

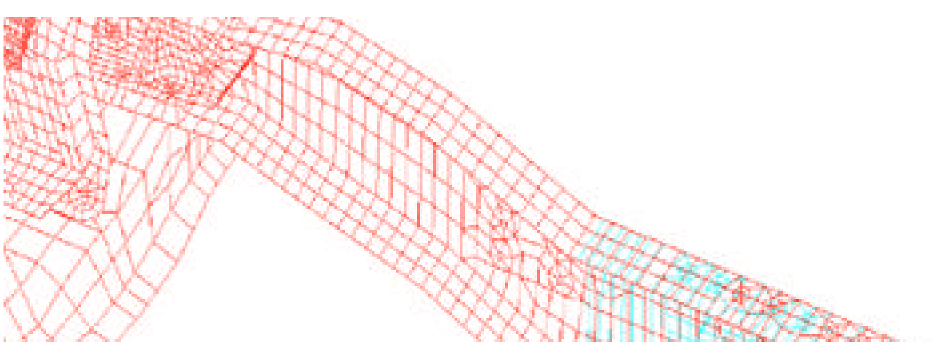
The time spent on geometry cleaning also is strongly dependent on the quality of the model and also on the files used for the communication between CAD and FEM experts. IGES format, for example, usually requires geometry healing before being able to mesh.

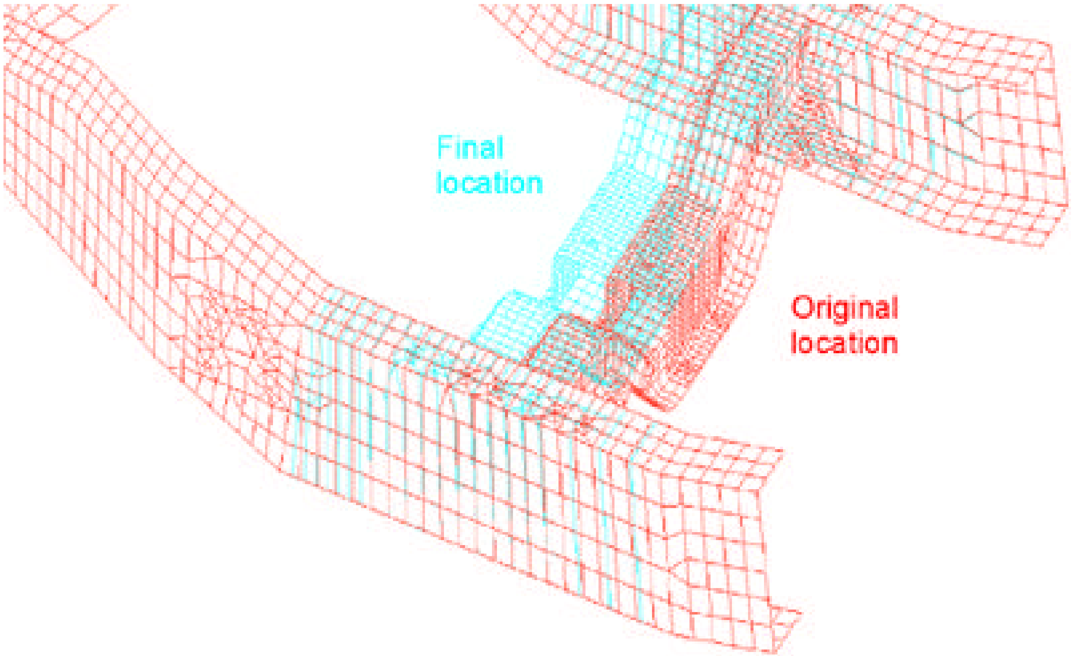
A Better Way

There has to be a better way. What if there was a product that let you work with one model, but allowed you to change it over and over to investigate multiple “what-if” scenarios without having to go through all of that import/export work? What if you had a tool that made a finite element model flexible, useful, and parametrically modifiable? What if it didn’t matter where that finite element model came from, or how old it was?

These challenges are met with ParaMesh: a parametric mesh modification tool that directly works on an existing FE mesh to perform shape modifications. It is discipline-neutral (structural, CFD, electromagnetic, multiphysics...) and also mesh-type neutral, so it can handle solid meshes (tetra, hex as well as pyramids and wedges), shell meshes (quad or triangles) and a mix of solid and shells. The software began shipping to customers September 24, 2003.

With ParaMesh, you read in a meshed model, and you shrink, stretch, move, indent, bulge, thicken, thin and generally do what you want to the mesh, as many times as you want -- all without that painful CAD model updating/import step.





Modifying a truck cross member: the central assembly is slide 300 mm to the left without using a CAD system.

How ParaMesh Works

Input to ParaMesh is simply a node and an element file. Formats allowed include structural analysis solver input files such as ANSYS, NASTRANs, PATRAN, CFD solvers input files such as StarCD, Fluent, CGNS, and general text files.

ParaMesh uses the data from these files as a starting point. Features, such as holes, ribs and protrusions are identified and made able to move, or morph to a different shape. Surface offset, translations, and rotations, or more complex features, such as emboss shape creation are associated with a ParaMesh parameter, giving you parametric control over the mesh changes requested.

ParaMesh works directly on an existing finite element mesh to perform shape modifications quickly.

ParaMesh works directly on an existing finite element mesh to perform shape modifications quickly.
ParaMesh modifies the nodal locations to stretch and shrink the mesh, without changing anything else. This means that the rest of the analysis input isn’t harmed in any way, that is, no boundary conditions need to be updated, deleted or reapplied.

Mesh Morphing Algorithm Details

The node moving, or mesh morphing process is optimization based. ParaMesh technology is based on a global smoothing technique, which allows large transformations of the mesh while still maintaining solution accuracy.
The smoothing algorithm seeks to find new node locations maximizing mesh quality, such that: $\underset{elements}{Max}\left(\underset{\text{int egpoint s}}{Max}\left(\underset{qualitycriteria}{Max}\left(\frac{C}{C_0}\right)\right)\right)$, where C is one of the quality measures on one integration point of the current element, and C_0 is the quality measure on the initial mesh.

This optimization technique works to maximize individual and global element shape metrics subject to millions of quality measures, with hundreds of thousand parameters that are geometrically tied.

This problem is complex and computationally intensive to resolve, which is why most previous mesh local optimization algorithms have attempted this optimization on an element basis, rather than process the entire mesh at one time, tuning global element quality as a whole.

For large modifications of the mesh, local algorithms have had a lot of trouble getting a useful result, producing some elements that were very distorted, and others, such as those close the modified boundaries, were often inverted.

ParaMesh’s global optimization algorithm processes all the elements simultaneously, allowing larger modifications of the mesh by taking advantage of the computational processing power available today.

Finally, the mesh can be re-exported out of ParaMesh, generating an input file that is similar to the initial input file, but with new coordinates. If the initial input file is ready to solve, the updated input file also will be ready to solve.

Legacy Models

Many companies maintain a library of previously created, or legacy, finite element models. Collectively, these models at once represent a huge financial investment as well as a treasure trove of digital information.
Previously, these legacy models were mostly worthless. Because they were not parametric or geometrically modifiable, most companies, using the traditional process, started any geometric changes or iterations at the CAD model level, despite the large costs.

ParaMesh is adept at reading these legacy models, extracting the pertinent mesh information, modifying the mesh parametrically to take the shape desired, an exporting the new mesh back to the surgically modified input file of the legacy model. ParaMesh, in many ways, gives life to previously dead legacy models.

Summary of Benefits

- ParaMesh modifies only nodal coordinates, and nothing else. After the modification of the node coordinates, the input file is ready to solve: material properties, physical properties, solution controls remain unchanged. Running design of experiment (DOE) or optimization studies after parameterizing the mesh with ParaMesh is straightforward, and very cost effective.
- ParaMesh does not require geometry. No CAD license and/or experts are needed.
- Every modification will be done on the original mesh, without a new CAD model, without healing, without remeshing, without reapplying boundary conditions. Most of the shape modifications can be done in a few minutes with ParaMesh's intuitive interface, and its powerful mesh manipulation tools. ParaMesh also can change the model easily in ways that a CAD model would not appreciate.
