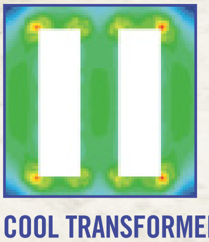
ANSYS

ADVANTAGE™

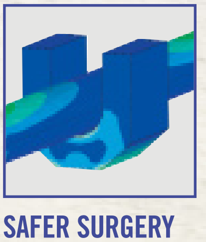
EXCELLENCE IN ENGINEERING SIMULATION

VOLUME I ISSUE 4 2007

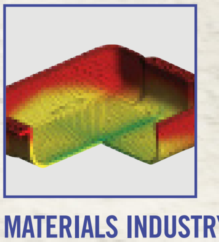
SIMULATION EASES THE LOAD
PAGE 10



COOL TRANSFORMERS
PAGE 4



SAFER SURGERY
PAGE 6



MATERIALS INDUSTRY SPOTLIGHT
PAGE s1

Help Us Navigate

and you could win a GPS

One year ago we launched *ANSYS Advantage*, taking the best from *ANSYS Solutions™* and *Fluent News* in an effort to bring you the most stimulating simulation stories.

Now we are looking for your help as we chart a path to the future. Tell us how to make *ANSYS Advantage* most relevant to you as a reader.

Take our 5-minute reader survey and have a chance to win a MAGELLAN TRITON 400 GPS AND OTHER NAVIGATION PRIZES.

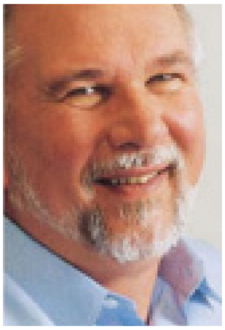
Help provide us with direction, and you may never have to ask for directions again.

Fill out our survey.

www.ansys.com/readersurvey

The Strategic Value of Multiphysics Simulation

A growing number of innovative products are being developed with technology representing real-world environments in which multiple types of coupled-physics interact.



For decades, commercial engineering simulation codes — most particularly those for finite element analysis (FEA) — have focused on predicting the effects of single physical phenomena: stress or deformation of parts under mechanical load, for example, or fluid flow around structures. In a growing number of applications, however, engineers must create simulation scenarios that more closely represent real-world environments in which multiple types of coupled physics interact. In fluid structure interaction, flow and the resulting structural deformation perturb one another, such as in aircraft wing flutter. Similarly in thermal–electromagnetic interaction, extreme temperatures affect the performance of devices such as electric motors.

In the past, these types of problems presented some real headaches for analysts, who might spend weeks setting up separate models, manually transferring data files between programs and running many sets of discrete solutions — often using different software codes. Today's multiphysics solutions address these hindrances by combining the effects of two or more physics in a unified manner. The ANSYS 22x family of elements, for example, solves coupled-physics problems in one solution pass with a single model, as explained in this issue's article "Multiphysics Simulation Using Directly Coupled-Field Element Technology."

Coupled-field technology is useful in the design of a range of products, such as sensors and microelectromechanical (MEMS) systems. An excellent leading-edge example of this type of application is covered in the article "Developing Sensors for Safer Back Surgery," which describes a project at the University of Alberta in which researchers used coupled-field piezoresistive elements to help design a MEMS device for measuring spinal loads in real time during surgery.

Another article, "Predicting Hot Spots Using Multiphysics," discusses an approach for predicting high temperature buildups in electrical transformers by making consecutive iterations between the CFD and the electromagnetic solvers. In this way, the approach calculates the coupled effects of temperature fields and heat losses together with related changes in electrical resistivity.

With a growing demand to capture reality more accurately in virtual prototyping applications, multiphysics simulation has gained considerable publicity in recent years and is inherent in Simulation Driven Product Development. With coupled-field physics a part of the core software for decades, ANSYS, Inc. is a pioneer and leader in multiphysics simulation, which builds on the breadth and depth of the company's structural, fluids, thermal and electromagnetic technologies. So multiphysics simulation is not really new for ANSYS, Inc. or its large user base, where companies have relied on this proven technology year after year in developing innovative products better, faster and more cost-effectively than is otherwise possible. ■

John Krouse, Editorial Director

For ANSYS, Inc. sales information, call **1.866.267.9724**, or visit **www.ansys.com**.
To subscribe to *ANSYS Advantage*, go to **www.ansys.com/subscribe**.

Editorial Director
John Krouse

Managing Editor
Chris Reeves

Art Director
Susan Wheeler

Ad Sales Manager
Beth Bellon

Editors
Marty Mundy
Erik Ferguson
Chris Hardee
Thierry Marchal

Production Assistant
Joan Johnson

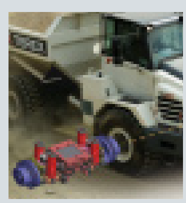
Editorial Contributors
Jeffrey Benko
John Dulis
Dan Hart
Siddharth Shah
William Wangard

Editorial Advisor
Kelly Wall

Circulation Managers
Elaine Travers
Sharon Everts

Designers
Miller Creative Group

About the Cover
Suspension systems in construction trucks contribute to greater productivity. Read about how Timoney Technology in Ireland is designing custom suspensions to address whole body vibration, on page 10.
Photo courtesy Terex.



Email: **ansys-advantage@ansys.com**

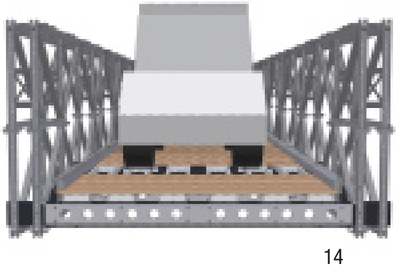
ANSYS Advantage is published for ANSYS, Inc. customers, partners and others interested in the field of design and analysis applications.
Neither ANSYS, Inc. nor the editorial director nor Miller Creative Group guarantees or warrants accuracy or completeness of the material contained in this publication.
ANSYS, ANSYS Workbench, CFX, AUTODYN, FLUENT, DesignModeler, ANSYS Mechanical, DesignSpace, ANSYS Structural, TGrid, GAMBIT and any and all ANSYS, Inc. brand, product, service, and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark licensed by ANSYS, Inc. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2007 ANSYS, Inc. All rights reserved.

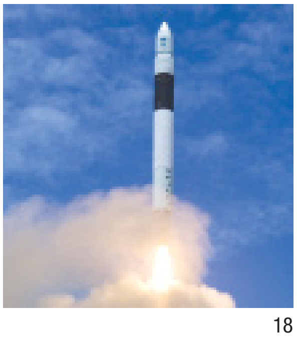
Table of Contents

6

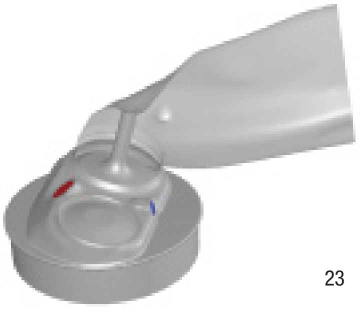
10



14



18



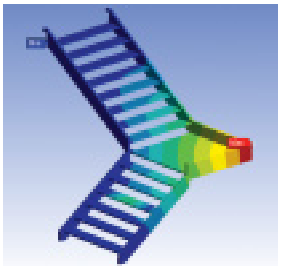
23

FEATURES

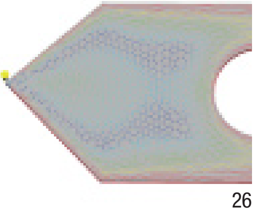
SIMULATION @ WORK

DEPARTMENTS

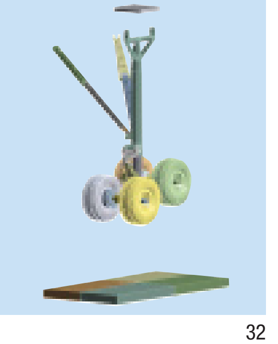
Spotlight on Engineering Simulation in the
Materials Industry



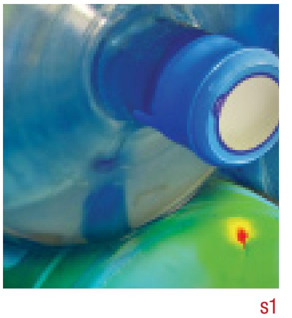
25



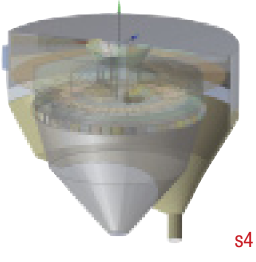
26



32



s1



s4

Predicting Hot Spots Using **Multiphysics**

Software coupling improves longevity and decreases power loss for electrical transformers.

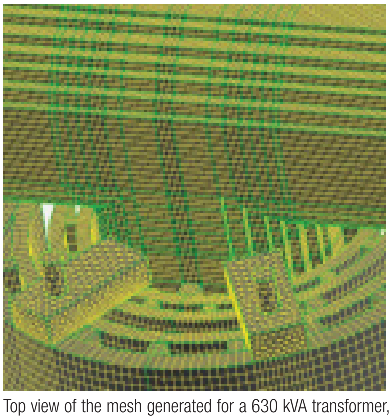
By Jacek Smołka and Andrzej J. Nowak, Silesian University of Technology, Gliwice, Poland

Electrical transformers transfer current from one circuit to another. Their primary purpose is to reduce the voltage in the electrical energy that is being delivered in order to supply devices at appropriate voltage level. Transformers normally consist of multiple sets of coils, also known as windings, which wrap around a ferromagnetic core. The coils are responsible for either inducing a magnetic field or assuming a transferred, induced current created by that magnetic field. This process results in high temperatures (hot spots) in the core and coils, which directly influence power losses and durability. While these components normally are cooled using natural convection, in the case of three-phase Y-Y dry-type power transformers used in mining applications, the transformer is further hermetically sealed in a tank. This creates a challenging situation when trying to encourage heat dissipation from the transformer. Simulation can be used to predict not only the high temperatures, or hot spot locations, that occur in the core and coil regions, but also to address cooling of the overall unit.

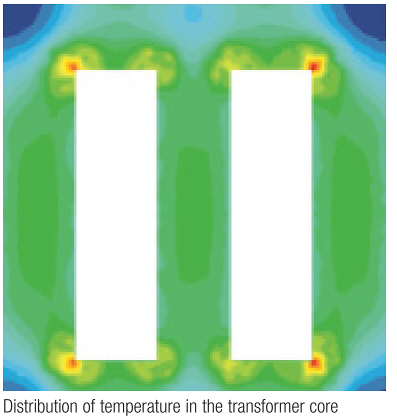
To accurately predict the temperature field, one must take into account the multidisciplinary nature of the physics involved. The fluid flow around the components is responsible for thermal management, whereas the electromagnetic field interaction with the components is both affected by and a primary cause of heating. To address these challenges, an effective numerical analysis must involve both computational fluid dynamics (CFD) and electromagnetic solutions. Combining FLUENT software with ANSYS Emag technology in an iterative, coupled manner provides that solution.

The coupling starts with a FLUENT calculation of the flow and temperature field around the device and a thermal field in the solid regions. The mathematical model uses the appropriate governing equations, source terms, boundary conditions and material properties specific to the device in question. The thermal conductivities are defined as anisotropic, since they have a preferred direction, for the core and coil elements. In addition, the coils' thermal conductivities were determined in a previous numerical analysis containing just the coils to be utilized in this main CFD model. After the FLUENT solver converges, the resulting temperature field is transferred to the electromagnetic solver to define the winding's temperature-dependent electric resistivities for the next solution stage.

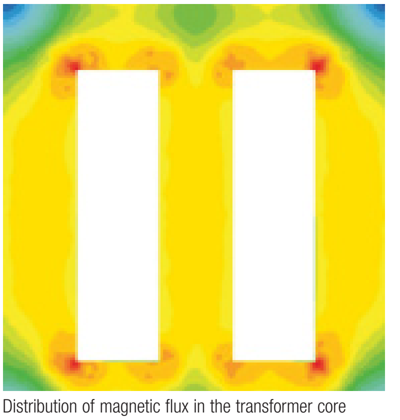
In the ANSYS Emag electromagnetic solver, the governing equations and the boundary conditions are specified to compute both the coil and the core transformer losses. The local core losses are calculated using a Steinmetz-based equation approach, while the coil losses, also known as



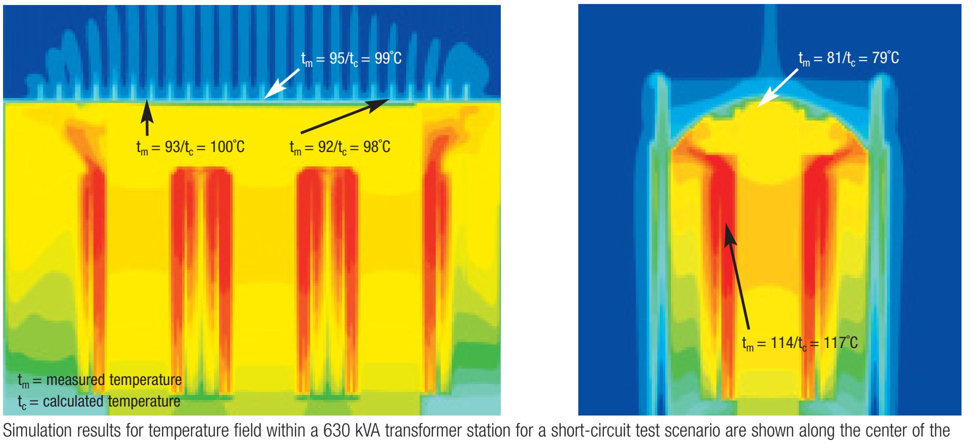
Top view of the mesh generated for a 630 kVA transformer, including the core, coils, screws and locating pads



Distribution of temperature in the transformer core (Locations of highest temperatures are shown in red.)



Distribution of magnetic flux in the transformer core (Areas of highest flux are shown in red.)



Simulation results for temperature field within a 630 kVA transformer station for a short-circuit test scenario are shown along the center of the core (left) and across the core (right).

Joule heat, are determined on the basis of a local current density and a local electrical resistivity. The electrical resistivity is defined as a temperature-dependent quantity using the FLUENT temperature field for its determination. As a result of this calculation, a local field of the transformer losses from the coils and core is obtained.

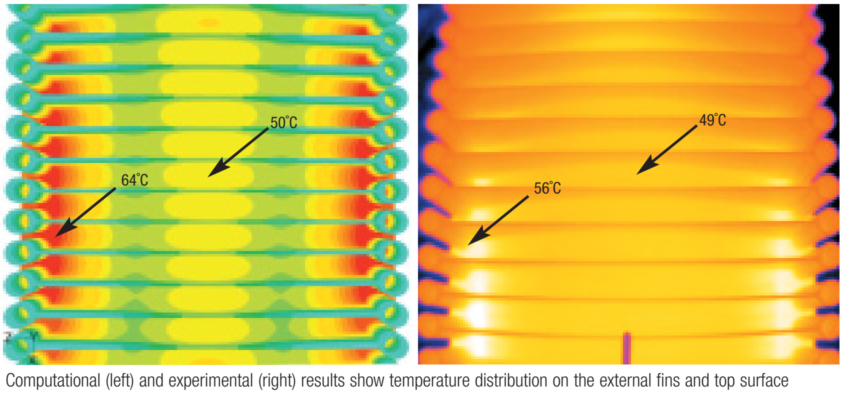
The calculated losses on an individual mesh cell basis are then transferred back to the CFD analysis, through the use of FLUENT user-defined functions (UDFs), as heat sources in the energy equation and the entire analysis process is reiterated. The cycle is repeated until the variations of the updated quantities become negligible. The typical case took two to three iterations between the FLUENT solver and the ANSYS Emag solver to reach convergence.

The largest transformer considered was a 630 kVA unit with a 2.13 cubic meter tank. The tank and transformer itself were naturally cooled by the surrounding air only. To effectively remove the heat from inside the tank, natural convection on the external walls of the station was enhanced by the addition of 25 flat fins welded to the top surface. Additionally, the surface area of the side-walls of the station was increased by the addition of 50 cooling pipes.

As was expected, free convection played a dominant role in the heat removal process. However, in the real device, the heat from the coils and the core was also dissipated via conduction to the transformer tank. In order to investigate the influence of the heat conduction on the total heat removal, the geometrical model included very small elements, such as the core screws and the coil locating pads. In order to accurately represent these components in the simulation, the generation of a fine mesh in GAMBIT software near these geometries was required. For the CFD computations, a mesh with 7.6 million elements was used in conjunction with parallel computing, which enabled the calculation of a solution in a practical amount of time.

To validate the numerical model, experimental transformer temperature tests for short-circuit, open-circuit and underrated parameter cases were performed according to the current European Standards for dry-type transformers. During the tests, wall temperatures were measured at selected points on transformer elements as well as on the overall external tank surfaces. In addition, information on the temperature field was captured for internal and external air surrounding both the device and the tank. The numerical results obtained confirmed that the prediction of the temperature distribution for the analyzed transformers and their surroundings was accurate.

The multiphysics mathematical model and procedures developed using coupled FLUENT and ANSYS Emag software have been applied to transformer units of different internal construction, positions in the tank and cooling system configurations. For each of the dry-type transformers considered, a number of potential changes were suggested to enhance the most effective heat transfer mode and lead to hot spot reduction. Moreover, the procedures developed for this group of devices can be easily extended to the more common oil-cooled devices by simply redefining the coolant material properties. ■



Computational (left) and experimental (right) results show temperature distribution on the external fins and top surface of a 630 kVA transformer tank for a short-circuit scenario.

Multiphysics Makes Spinal Surgery Safer

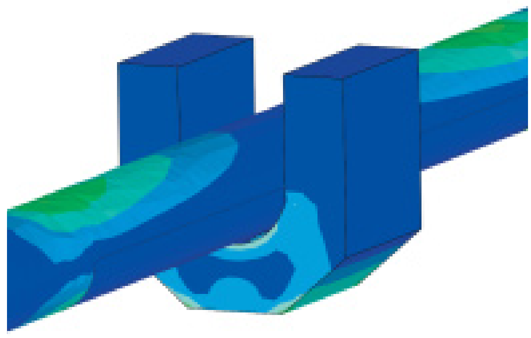
Piezoresistive sensors designed with the help of coupled-field analysis give physicians real-time feedback on applied loads during corrective spinal surgery.

By David Benfield and Walied Moussa, Department of Mechanical Engineering, University of Alberta, Edmonton, Canada
Edmond Lou, Glenrose Rehabilitation Hospital, Edmonton, Canada

Image © iStockphoto.com/geopaul

Scoliosis is a spinal deformity characterized by abnormal lateral curvature of the spine and axial rotation of the vertebrae. In severe cases, surgery may be required to correct this curvature. In this procedure, special connectors that either hook onto or screw into the vertebrae are inserted along the length of the spine where the curvature needs to be corrected. Surgeons then fit a metal rod into notches in the heads of these hooks and screws to realign the spine until to achieve the required correction.

This alignment process subjects the spine and the hook/screw assemblies to two types of 3-D loads: direct contact forces of the structural elements as they impact one another during the process and



FEA was used to determine contact forces on sensor strips in the surgical device.

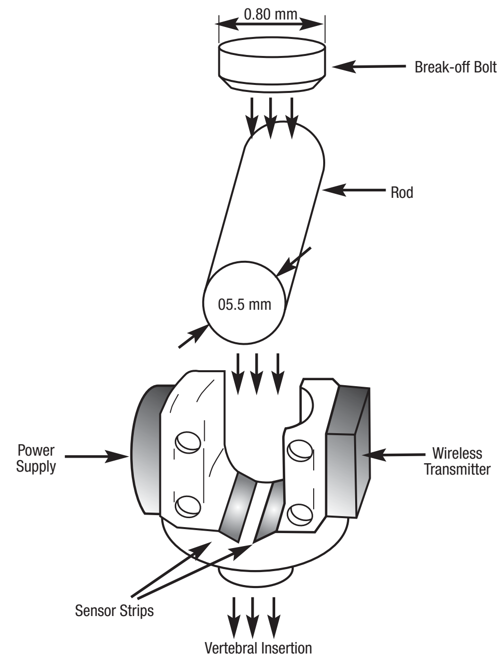


Diagram of hook/screw heads typically used in corrective scoliosis surgery. As seen here, they were modified with two piezoresistive silicon strips, a wireless transmitter and a power supply in order to sense and transmit loading information during surgery.

moment loads created by the leverage of the rods as they are installed. These loads are necessary in realigning the spine, but excess levels can contribute to bone breakage, as well as fatigue failure of the hooks, screws and rods.

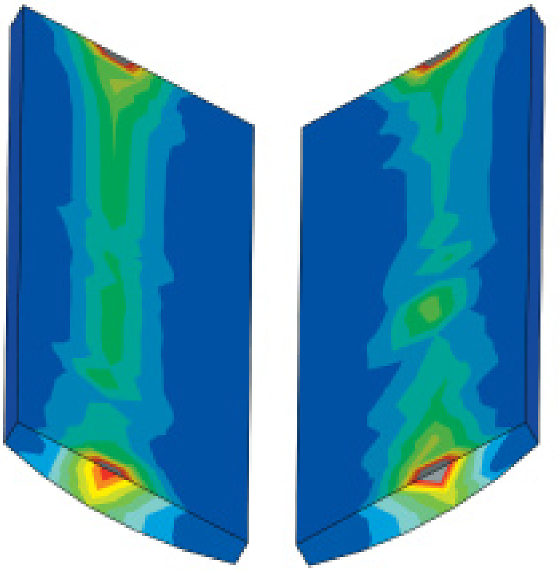
One recent research project at the University of Alberta in Canada is aimed at developing embedded micro-electrical-mechanical systems (MEMS) sensors to measure these forces and moments in real-time during surgery, thus providing physicians valuable feedback during the corrective procedure. For this application, modifications to conventional scoliosis hooks and screws were made. In addition, a power module and a wireless transmitter module were placed on either side of each of the hook/screw connectors, and two silicon sensor strips were placed at the interface between the rod and the hook/screw heads.

To detect 3-D forces and moments, each sensor strip must detect loads at more than one location along its length, so each strip has two sensor pads evenly spaced along the contact line of the corrective rod. Each pad consists of a deformable membrane onto which are mounted four piezoresistive gauges that are sensitive to contact forces in shear and normal directions. When combined, electrical output from the gauges on both strips indicate the 3-D forces and moments applied during surgery.

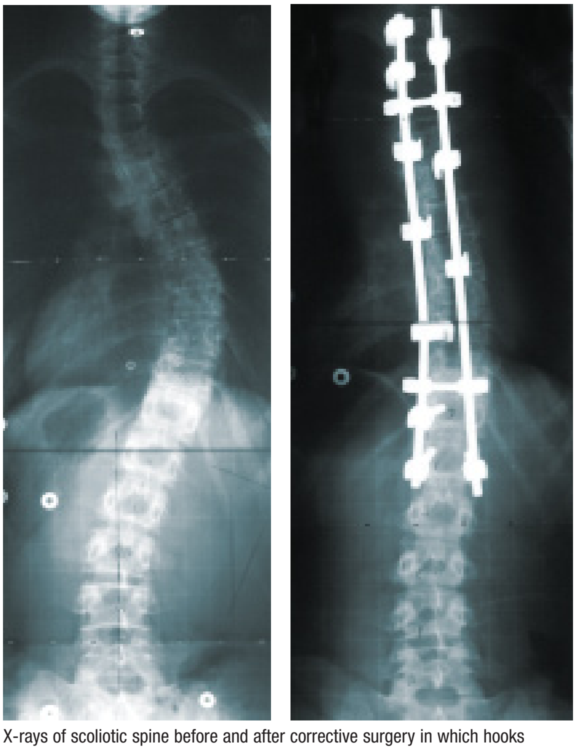
Multiphysics analysis using ANSYS Mechanical software was used to study three aspects of this configuration:

- Contact analysis used to predict loads transmitted between the rod and the sensor strips
- Structural analysis used to determine subsequent deformations of the anisotropic silicon membranes
- Piezoresistive analysis used to determine the output voltages from each piezoresistive gauge

ANSYS Mechanical technology provides coupled-field elements and multiphysics solution tools that enable all three regimes of this problem to be solved simultaneously in a single multiphysics solution. Using models that incorporate the element types and sizes for MEMS devices (see sidebar pages 8 and 9), output voltages from the sensor array were accurately predicted. Empirical data from other studies [7, 8] was used to determine a range of forces and moments that would be applied to each hook or screw.

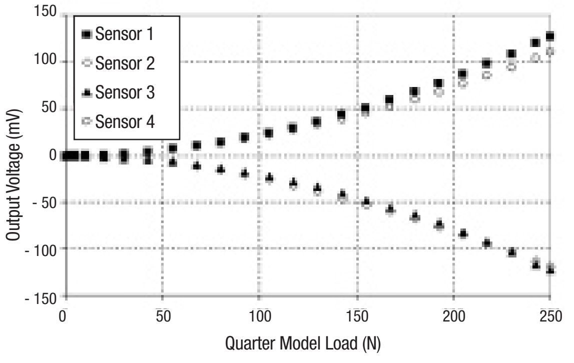


Contact stresses induced on sensor strips in a full hook trial

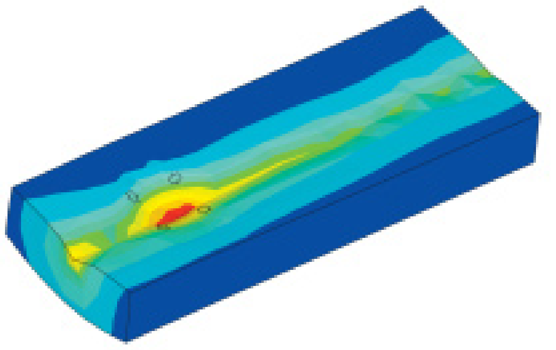


X-rays of scoliotic spine before and after corrective surgery in which hooks and screws are inserted into the vertebrae

Images courtesy Glenrose Rehabilitation Hospital.



Simulated sensor voltage outputs for the normal loading scenario



Numerical results for membrane deformation (μm units)

A full hook model with a rod contacting the sensor strips was built to determine the force distributions on the sensor strips for different load conditions. Using these force distributions, a symmetrical quarter-strip model was created to determine the membrane deformations and subsequent voltage outputs from the piezoresistive gauges. An excitation voltage of 3 volts was used to represent the power output characteristics of the power module.

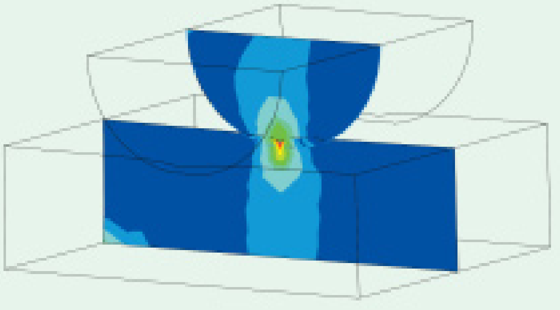
The flexibility of the FEA software allowed the voltage outputs for all 16 piezoresistive gauges to be accurately determined for the applied forces and moments. Voltage data from this analysis allowed calibration equations for the full sensor array to be developed utilizing simple superposition principles. These calibration equations enabled the 16 voltage outputs from each hook/screw assembly to be converted back into force or moment information representing loads that surgeons apply to the spine.

The numerical model is a powerful tool in creating these calibration equations, as it allows the problem to be broken down into symmetrical portions that are more easily analyzed than a full model. Moreover, ANSYS FEA simulation also can accurately predict sensor outputs,

Selecting and Sizing Finite Elements

ANSYS Mechanical software was utilized in each of three types of analyses and to establish appropriate parameters for the multiphysics simulations. This included using theoretical solutions to typical micro-electrical-mechanical system (MEMS) problems or simplified versions of the strip model to select the best element types and sizes for obtaining an accurate, fast solution. Element types and sizes were determined as follows:

Contact analysis of loads. The contact problem was the first to be evaluated for appropriate finite element (FE) parameters. A simple model of a cylindrical surface contacting a flat surface was developed, and the results of this model were compared to theoretical values obtained using standard Smith–Liu equations for determining contact stress distributions [1, 2]. At this point in the analysis, material properties are assumed to be linear for the flat silicon surface and for the stainless steel rod. The results of this analysis determined that when using a contact pair model (composed of ANSYS CONTA174 and TARGE170 elements) applied to 3-D solid elements, the mesh size in the contact area should be finer than one-half the theoretical contact width. In this analysis, the theoretical contact width is approximately 400 μm, so the optimal mesh size was determined to be smaller than 200 μm in order to obtain accurate analysis results.



Contact stresses produced in a trial analysis using a simple model of a cylindrical surface contacting a flat surface

Structural analysis of membrane deformation. Next, parameters were determined for accurately predicting deformations of the anisotropic silicon membrane produced by the contact forces calculated in the previous problem phase. For this analysis it was found that deflections predicted using ANSYS SOLID187 thus avoiding complicated testing of sensor components and related packaging.

Through preliminary manufacturing trials, it has been found that the manufactured devices have similar performance curves to the simulated models. Predicting the performance of a complex sensor array using multiphysics analysis without ever building a device is a significant advantage in terms of costs and time during MEMS product development. ■

References

[1] Shigley, J. E.; Mischke, C. M.; Budynas, R. G., "Contact Theory," *Mechanical Engineering Design*, 2004, 7 Ed., McGraw-Hill: Toronto, pp. 161-166.

[2] Smith, J. O.; Liu, C. K., "Stresses Due to Tangential and Normal Loads on an Elastic Solid with Application to Some Contact Stress Problems," *Journal of Applied Mechanics*, 1963, Vol. 71, pp. 157-166.

[3] Timoshenko, S.; Woinowsky-Krieger, S., *Theory of Plates and Shells*, 1959, McGraw-Hill: Toronto.

[4] Bao, M.; Wang, Y., "Analysis and Design of a Four-Terminal Silicon Pressure Sensor at the Centre of a Diaphragm," *Sensors and Actuators*, 1987, Vol. 12, pp. 49-56.

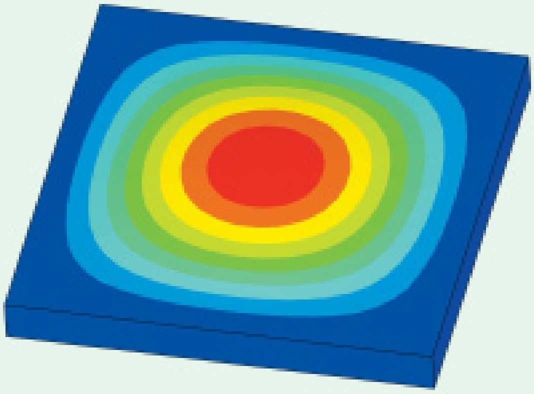
[5] Kanda, Y., "Optimum Design Considerations for Silicon Pressure Sensors using a Four-Terminal Gauge," *Sensors and Actuators*, 1983, Vol. 4, pp. 199-206.

[6] Bao, M.; Qi, W.; Wang, Y., "Geometric Design Rules of Four-Terminal Gauge for Pressure Sensors," *Sensors and Actuators*, 1989, Vol. 18, pp. 149-156.

[7] Lou, E.; Hill, D. L.; Raso, V. J.; Moreau, M. J.; Mahood, J. K., "Instrumented Rod Rotator System for Spinal Surgery," *Medical & Biological Engineering & Computing*, 2002, Vol. 40, pp. 376-379.

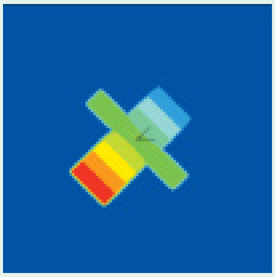
[8] Duke, K. K., "The Design of Instrumentation for Force Measurement During Scoliosis Surgery," *Mechanical Engineering*, 2001, University of Alberta, Edmonton, pp. 114.

elements and a mesh size finer than 2.5 percent of membrane edge length were in agreement with calculations for a typical square-membrane problem, as defined by Timoshenko [3]. For the 1 mm x 1 mm membranes used in this analysis, this corresponds to a maximum mesh size of approximately 25 μm. Fortunately, solutions produced by ANSYS Mechanical software are not subject to the thin-membrane limitation of theoretical solutions, which may not yield accurate results when membrane thickness is larger than 5 percent of membrane edge length.



Deflection of the 1 mm x 1 mm silicon membrane is predicted by structural analysis.

Piezoresistive analysis to determine output voltage. FE parameters for the piezoresistive part of the analysis were next determined for the four-terminal gauge subjected to uniaxial stress. The theoretical solution to this problem is outlined [4–6]. Conclusions from this literature agree with piezoresistive theory: that p-type silicon devices have maximum sensitivity when uniaxial stress is applied perpendicular to the device surface with the long axis of the four-terminal gauge at a 45-degree angle to this direction. The required mesh size for a 100 μm x 50 μm four-terminal sensor modeled with SOLID227 elements was also evaluated, leading to a determination that a 10 μm mesh is required to numerically simulate voltage outputs that are consistent with piezoresistive theory.



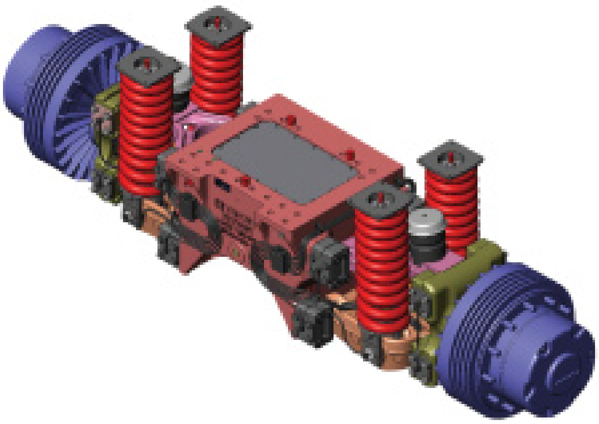
Voltage distribution of a simple four-terminal sensor under uniaxial stress

With a maximum payload capacity of 28 metric tons, the new Terex TA30 articulated dump truck is equipped with a Timoney independent suspension for increased comfort and safety.

Taking the Shakes Out of **Construction Trucks**

Timoney Technology designs independent suspensions that allow construction vehicles to travel faster on rough terrain without exceeding vibration regulations.

By Eoin Cashin, Technical Manager, Timoney Technology, Gibbstown, Ireland

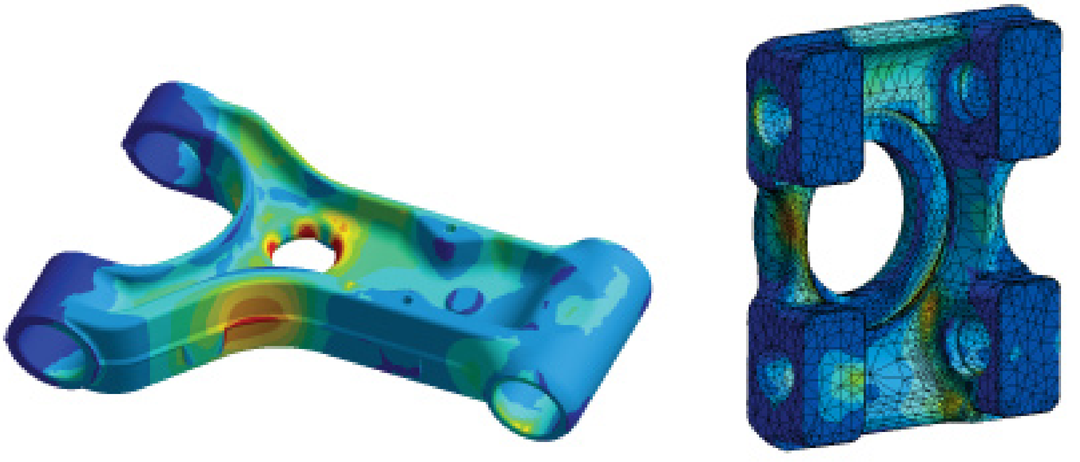


CAD geometry of a Timoney suspension system designed for trucks that see high vibration

To maximize productivity on construction sites, operators want to drive off-road hauler trucks up to maximum possible speeds. But traveling over rugged terrain at higher speeds can result in excessive whole body vibration (WBV): lateral and vertical forces that jostle drivers as the vehicle bounces over rocks, holes, bumps, ruts and other surface irregularities.

WBV is uncomfortable and can cause serious health problems, such as spinal degeneration. Also, accident risks are greater with elevated vibration levels, since drivers lack full control of vehicles as they are bounced around. For these reasons, a correlation between operator comfort and vibration magnitudes is provided in ISO 2631, and the European Community Human Vibration Directive 2002/44/EC establishes single-action limits for driver acceleration, as well as an overall daily vibration limit for operators.

Timoney Technology addressed the issue of WBV with a patented double-wishbone independent suspension that decouples asymmetrical wheel loads, thus reducing lateral and vertical acceleration. As a result, the smoother-riding trucks can safely travel over rough terrain at speeds greater than those with conventional straight beam-axle suspension



Stress contours are shown on the upper wishbone (left) and suspension knuckle (right), in which Timoney engineers used sphere-of-influence element sizing to increase mesh densities for studying areas of high stress in greater detail.

systems, thus increasing job-site hauling productivity and construction company profitability. Timoney has gained widespread recognition in the transportation industry designing and manufacturing such independent suspensions for off-highway vehicles — especially articulated dump trucks (ADTs), a relatively new class of hauler that is hinged with a universal joint between the cab and dump box for greater maneuverability on extremely uneven terrain.

Because of the structural complexities of ADTs compared to rigid-frame haulers, most of Timoney's independent suspensions are exhaustively custom-designed for each vehicle model. Engineers must tune suspensions by developing assemblies of parts to isolate severe loading on each wheel. Parts must be fatigue-resistant in withstanding these loads, yet cannot be over-designed with excess material because of cost and size considerations. Packaging is a particular focus because the suspension must fit within the overall vehicle width of 2.9 meters and between wider tires used to increase vehicle mobility. Moreover, designs must be completed efficiently, in order to meet tight OEM request-for-quotes and product delivery schedules.

Timoney engineers analyze stresses on all major parts of suspensions using ANSYS Mechanical software. Typically in an iterative process in which CAD geometry from SolidWorks® is imported into ANSYS Mechanical software, FEA meshes are generated, analyses performed and appropriate changes made to the CAD geometry to reinforce areas with high stress concentrations while removing material from regions with relatively low stress. The cycle is repeated until an optimal design is found — one that meets required performance specifications without excess material.

Automated modeling and analysis features of ANSYS Mechanical software, as well as tight integration with CAD, allow simulation iterations to be performed quickly in optimizing designs for performance and reliability together with size and weight savings. Native geometry for SolidWorks parts and assemblies was imported directly into the ANSYS Mechanical product. Analysis of major subsystems, such as the gearbox, was performed using features for modeling assemblies of parts that allow for representation of the relationships between the individual mating components.

Bonded contact element features were used to automatically detect joined surfaces instead of requiring the regions to be defined manually. Sphere-of-influence element sizing was used to increase mesh densities in localized regions for studying stress in greater detail. Sequences of solutions were performed automatically using a single FEA model for various load cases encountered on the suspension as the vehicle turns, brakes and traverses various types of terrain.

In one recent project, ANSYS Mechanical technology was instrumental in reducing the overall weight of a Timoney product from 4.3 to 2.6 metric tons — nearly a 40 percent weight saving that reduced manufacturing costs and increased the payload capacity of the hauler truck. This effective simulation process is essential in meeting OEM design requirements and deadlines for a wide range of vehicles.

Timoney has used software from ANSYS, Inc. for years as an essential tool, enabling engineering teams to develop suspension systems for some of the most complex off-road vehicles in the world — most especially ADTs now found on just about all major construction jobsites. An efficient iterative process between design and simulation is key to making rigorous trade-off decisions quickly in the development of custom-designed suspensions. ■

Bending for **Braking**

Dynamic simulation reduces the cost of automotive brake pedal design and manufacture.

By Jessica Song, FEA Specialist, and Randy Phinney, Product Engineer, GHSP, Michigan, U.S.A.

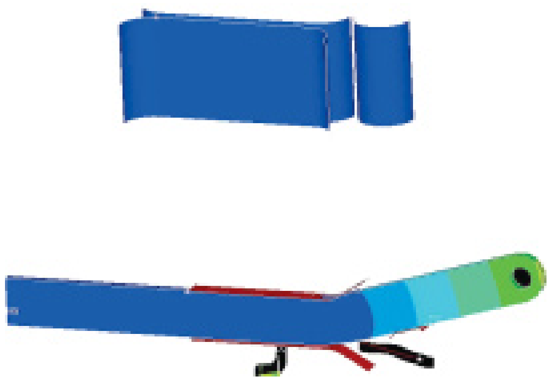
Conventional high-volume pedal beams for automotive applications are manufactured using a progressive stamping operation. The nature of progressive stamping results in a relatively high material scrap rate for many pedal beam designs. GHSP, a company that designs and manufactures driver controls, including shift systems, throttle controls and pedal systems, has modified this process to instead trim and bend, rather than stamp, a narrow strip of steel when manufacturing pedal beams. This change greatly reduces the material scrap rate and production costs.

The bender procedure is a continuous manufacturing process. It consists of a series of machines that perform sequential operations that transform a steel strip into a formed pedal beam. First, a basic stamping, or cutting, action occurs to form the pre-bend outline of the pedal beam. This pre-bend component has no bend in the structure whatsoever. The bending station then deforms the component to its final shape.

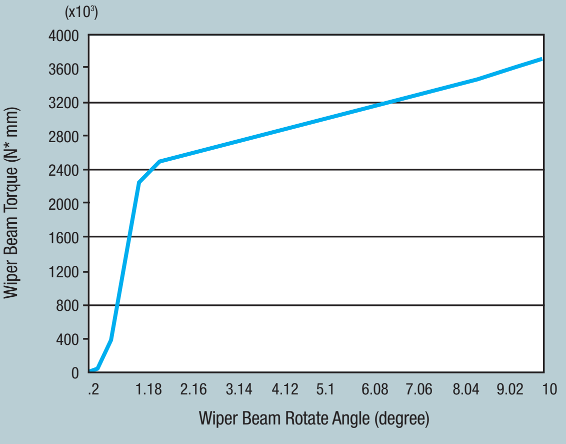
The bending station creates two kinds of bends: hard and easy. “Hard bend” refers to bending the steel strip along its length, against its greatest moment of inertia. An “easy bend” is normal to the hard bend and bends the component to the side, about the steel strip’s lowest moment of inertia. The bending station components that deform the strip to create the bends are called the bend heads and wipers. The bend heads hold the component in place and have guide curves that are designed to locate and guide the formation of the final bend geometry. The wipers press against the component to actually deform the part around the guide curves on the bend heads. In a bending process that requires multiple steps, typically — but not necessarily — the hard bend is performed first. A total bend sequence may consist of one single bend or a combination of many easy and hard bends.

Developing this new process required the design of bend tooling and procedures that would result in bend geometry matching the desired final product. Geometric and strength requirements for both of these components are of great importance. In order to provide a full analysis of the bending process, a dynamic finite element analysis (FEA) of the entire system was implemented.

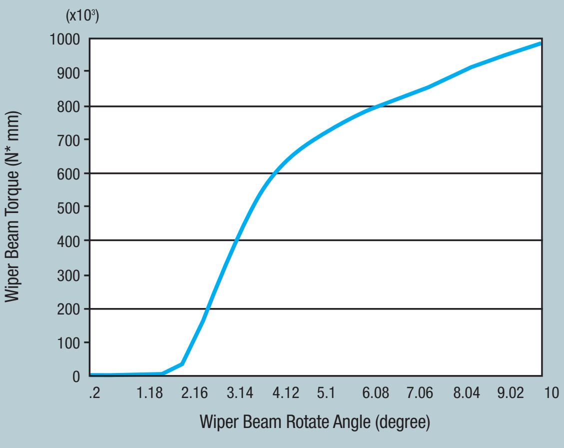
The full simulation included two sequential bend simulations: one hard bend action followed by an easy bend event. The model contained the trimmed steel strip to be bent, a hard bend head and wiper, and an easy bend head and wiper. The steel strip for the model was meshed with 3-D solid elements and modeled with the material properties for SAE950X, a very popular steel in structural automotive components due to its relatively high yield strength. The model, which was analyzed using ANSYS Structural



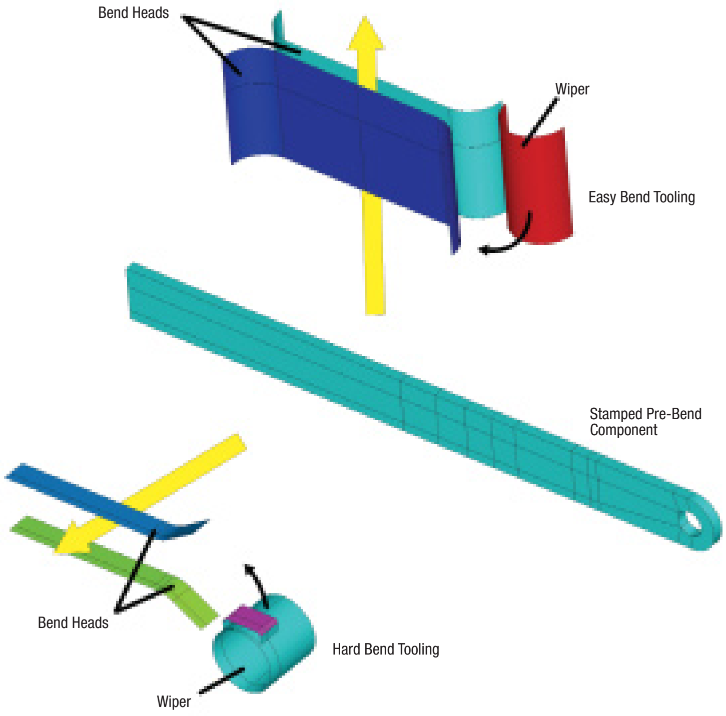
Bend heads (top) and steel strip during hard bend (bottom)



Hard bend torque, as seen by machine tooling, versus sweep angle



Easy bend torque, as seen by machine tooling, versus sweep angle



Components involved in a novel bending process developed by GHSP

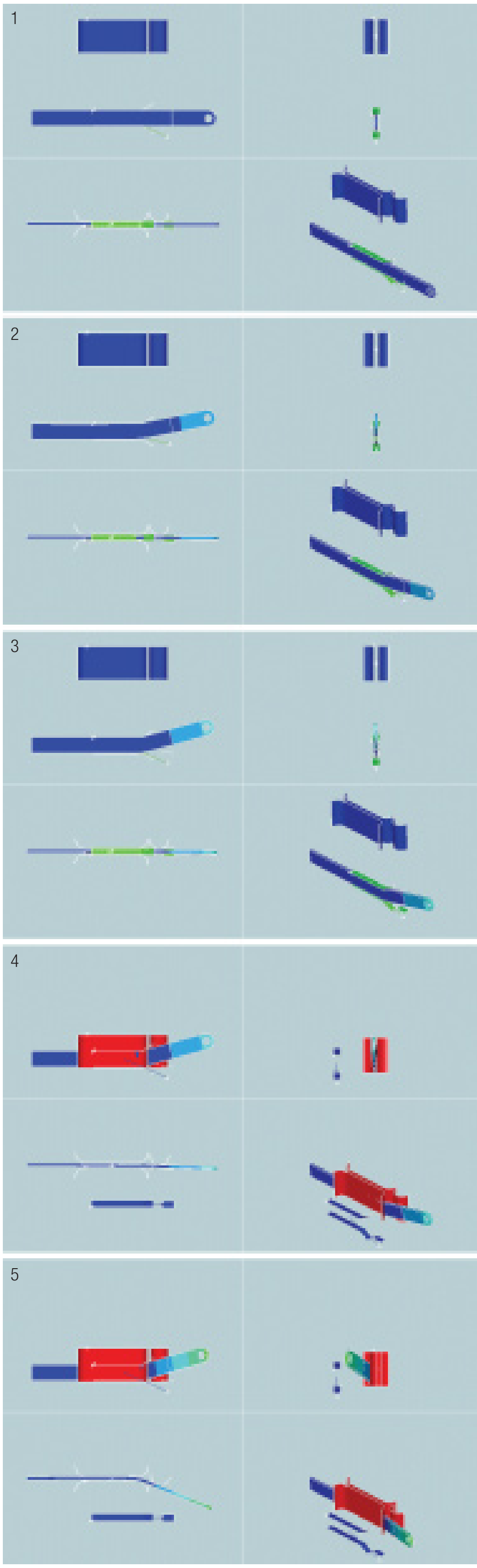
software, included contact, nonlinear material properties, large deformation and large rotation. Contact was defined between the steel strip and both the bend heads and the wipers. Both the bend heads and the wipers were assumed to be rigid. The hard bend wiper was defined with rotation. Both stages of the simulation were modeled as fully dynamic processes.

By using FEA in this way, the bending process could be examined in depth. The simulation results were used to study the final pedal beam geometry and strength properties, as well as the loading experienced by the tooling during the process.

The final bend geometry that resulted from the process was examined and measured to see if it matched the desired outcome. If it didn't, changes were made to the trimmed shape and to the geometry of the tooling and wiper placement, and the analysis was repeated until the process was verified to create the desired final component. The use of this methodology saved time and reduced costs that would otherwise have been expended when utilizing a trial-and-error method to create these components. The final nodal locations of the pedal beam were used to create a solid model of the finished beam geometry. An ANSYS parametric design language (APDL) macro was prepared to develop this geometry creation. This model was then sent back to design engineering to be used in the final pedal beam assembly.

Verifying that the manufacturing tooling was properly designed was equally important and also addressed by GHSP using this simulation. During the simulation, the torques and contact forces on the wipers and bend heads were calculated. By monitoring these values, GHSP ensured that the bender machine limits were not exceeded.

Tooling and time spent in trial, test and error proofing is expensive. Simulation allows GHSP to detect and correct potential problems with hard bend stability, sensitivity to edge conditions, tool wear, beam strain history, beam cracking, etc., before tooling design is finalized. The analysis demonstrated here is used today by GHSP to guide design and manufacturing, reduce cost and shorten the product development cycles. Using APDL, this analysis is being incorporated into end-user software tools to help designers evaluate designs and provide modification guidelines. ■



From top to bottom, these images depict the pedal bending process developed by GHSP. The top three images show the hard bend simulation of the beam advancing with time. The bottom two images display the easy bend process. Each image has four parts, which illustrate the side (top left), front (top right), top (lower left) and orthogonal (lower right) views of the apparatus.

Typical TMS temporary bridge structure in use after a flood

Bridging the Gap

Loading capacity simulation for temporary bridges assists in providing effective disaster relief.

By Pavel Manas, Department of Engineer Technologies, University of Obrany, University of Defence, Brno, Czech Republic
Tomas Rotter, Department of Steel and Wood Structures, Czech Technical University, Prague, Czech Republic

When natural disasters such as wide-scale flooding occur, transportation routes become critical in providing help to the afflicted area. Supplying a method for relief vehicles to safely cross rivers is extremely important during these periods. In the Czech Republic and many other countries, these relief efforts might involve assistance from allied countries within the North Atlantic Treaty Organization (NATO).

The temporary bridge tezka mostova souprava (TMS) was developed for military use between 1950 and 1960 in the former Czechoslovakia. The TMS bridge is a single-line, steel truss structure with a wooden deck well-suited for spans (distance between the supporting piers) of between 21 and 39 meters. The bridge design can be used in many configurations such as single-span, double-span, triple-span, and single-sided or double-sided, as well as single-story or double-story. This type of construction is commonly used instead of permanent bridges during bridge repair or for temporary bridging in an area affected by a disaster, such as a flood or landslide.

The NATO Standardization Agreement 2021 (NATO STANAG 2021) helps establish a standard method of computing the military classification of bridges, ferries, rafts and land vehicles. Under this agreement, every bridge can be assessed with a specific load capacity classifying number — military load class (MLC) — and every vehicle is assigned to a specific category from 16 MLC classes. If the vehicle category MLC is lower or equal to the bridge MLC, that vehicle can safely pass over that bridge. This methodology is independent of various national codes, such as EC standards from the European Union or DIN standards from Germany, and allows closer interoperability between NATO countries.

The University of Defence, in cooperation with Czech Technical University and funded by the Ministry of Transportation of the Czech Republic, set

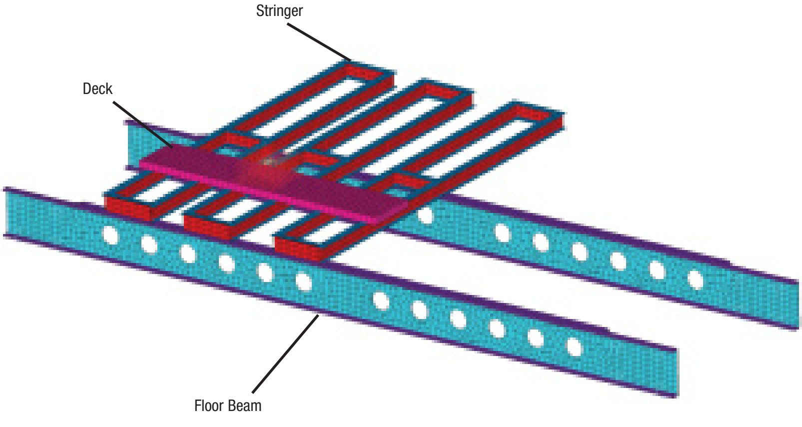
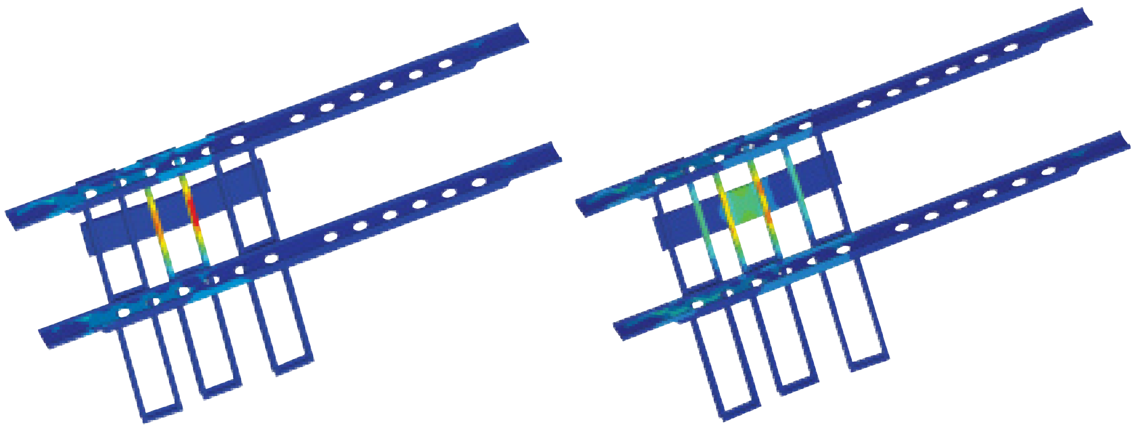


Diagram of a typical TMS bridge construction



Simulation of the wooden deck (left) showed minimal load distribution between stringers whereas, simulation of the steel grid deck (right) indicated significant load distribution based on a tire load of approximately 9.07 tons and a contact area of 400 mm by 400 mm.

out to certify the loading capacity of TMS bridges according to the NATO guidelines. ANSYS Mechanical software was used to assist in classifying various decks for existing bridges, as well as for new designs.

TMS bridges traditionally had wooden decks. Simulations revealed that the flexible wooden boards did not distribute tire loads effectively, causing the stringer directly under the tire to be highly stressed. As a result, the overall bridge rating was limited to MLC-70 (approximately 70 tons) when wood decking was used, even though the other structural components of the bridge could be rated for MLC-90 (approximately 90 tons) for some shorter spans. Consequently, three different decking systems — wooden decking, welded steel panels and a new design of welded steel grid decking — were analyzed to find a way to upgrade the floor rating to MLC-90, consistent with the rest of the structure.

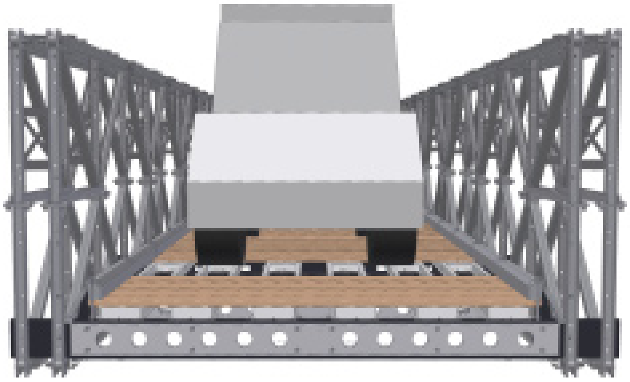
The simplest model consisted of decking and stringers. In the early stages of the study, this model used shell elements, but in later simulations the CAD geometry was meshed directly with solid continuum elements. ANSYS Mechanical software allowed for easy import of CAD data from Autodesk® Inventor™ and supported a legacy database from previous versions of ANSYS Mechanical software, so data could be reused. The model included contact, available through the ANSYS Workbench environment, between the decking and stringers.

The welded steel panel design developed about 10 years ago as an alternative to the wood decking and stringer system was modeled in Autodesk Inventor and imported into the ANSYS Workbench environment as an isolated component using solid elements. This model was analyzed as a linear elastic structure without contact. The welded panel had a high safety factor even under MLC-90 loading, but the expense of this design led to its rejection for military use, though it is still used in some urban areas, as it is quieter than a wooden deck.

In addition, a model of a single floor beam was analyzed to determine the effect of nonlinear responses, such as plasticity and lateral buckling, on its load-carrying capacity. This model used solid elements with bonded contact at bolted connections and frictionless contact at other interfaces between parts. The floor beam was found to be adequate for MLC-90 loading.

Finally, a complex model of deck, stringers and floor beams was analyzed to provide the most accurate interaction among all the components of the floor system. To minimize solution times, this model was analyzed using shell elements (no solid elements) and appropriate contact formulations. This model was investigated for both wooden decking and steel grid decking. The analyses revealed that steel grid decking effectively distributed tire loads to adjacent stringers, reducing the stress in the stringers and enabling the floor system to be rated at MLC-90. Based on these structural analyses, a new bridge deck composed of the welded steel grid was designed to enable better deployment of assistance to address a crisis situation. ■

This work was done with support of the Czech Republic Ministry of Transportation, project 1F44L/078/030.



CAD model of TMS temporary bridge construction and military vehicle

TECPLOT 360™

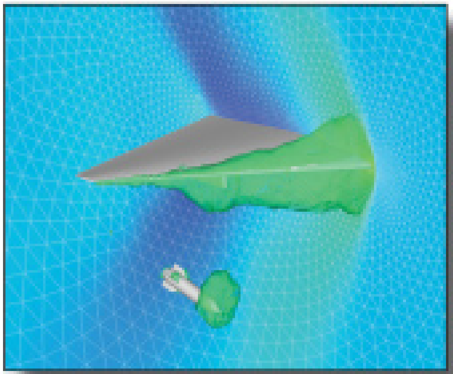
70.2% of engineers who do work with CFD use Tecplot® software to communicate their results to stakeholders. - Hebert Research, Inc. - 8/05

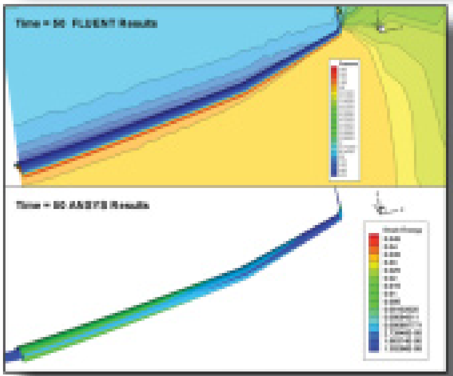
What do ANSYS software users like about Tecplot 360™ for post-processing results:

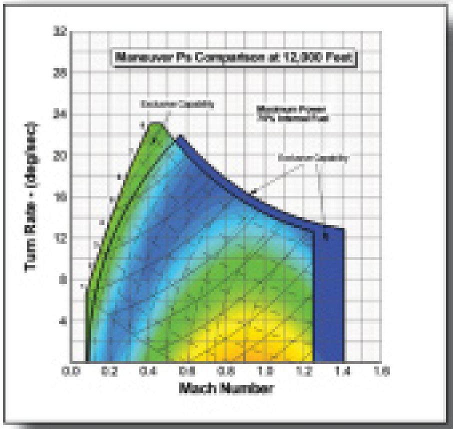
- Open gigabyte solutions on your desktop.
- Load native case and data files.
- Cut & Paste images and animations from Tecplot 360 directly into Microsoft® Word and PowerPoint®.
- Utilize time-linked, multi-frame workspace options.
- Explore transient solutions with a single mouse click.
- Extract critical flow features (vortex cores, shock surfaces, and more).
- Extensive XY and 2D plotting - control over 2,500 plot attributes.
- Drill down with CFD analysis options.
- Perform Fluid Structure Interaction (FSI) visualization and analyze ANSYS results as well as results from other structural solutions.
- Available on Windows®, UNIX, Linux, and Mac.

"Tecplot 360's greatest strengths are its ability to manipulate large files in a timely manner, good flexibility in layouts and frames, as well as its excellent graphical output."
- Steven Davie, Tetra Tech

"We have seen a major reduction in CFD post-processing time. This means that more resources can be devoted to running CFD simulations and analysis rather than struggling with data plotting issues."
- Mehul P. Patel, Orbital Research Inc.







Download the free whitepapers:

"Vortex Visualization" & "Visualization of Flow Separation"

at:

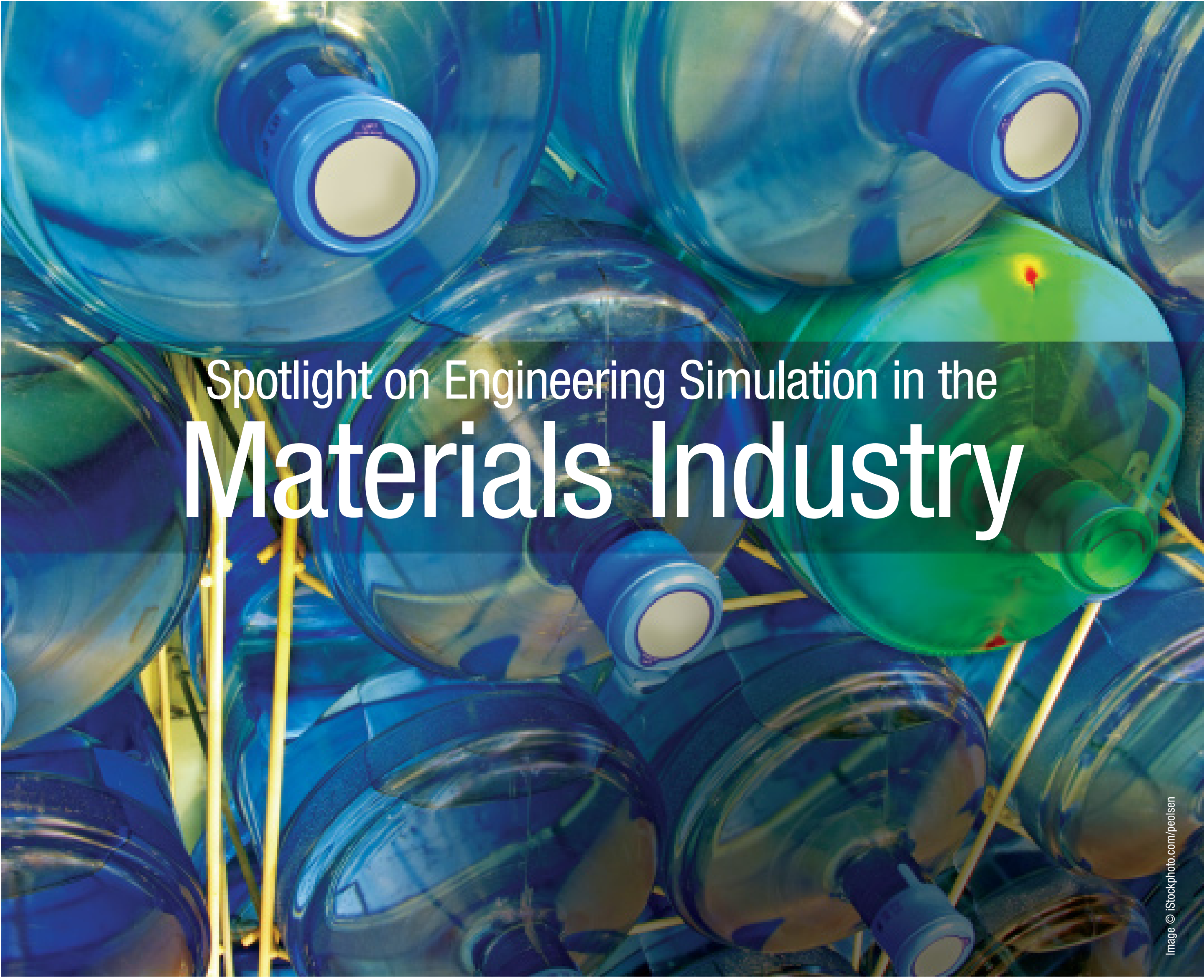
www.tecplot.com/flow

Tecplot®, Tecplot 360™, Enjoy the View™ and the Tecplot 360™ logo are registered trademarks or trademarks of Tecplot, Inc. in the United States and other countries. All other trademarks are the property of their respective owners.

Tecplot Enjoy the View™

1.800.763.7005

Spotlight on Engineering Simulation in the Materials Industry



POLYFLOW simulation of the blow molding of a 20-liter water bottle.

Pushing the Limits of Materiality: The Virtual Prototyping Solution

Materials are continually being improved to address the challenges of the 21st century in a fiercely competitive world.

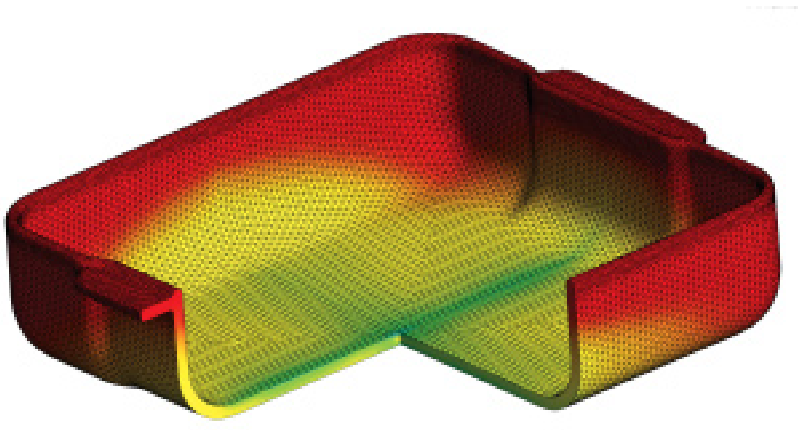
By Thierry Marchal, ANSYS, Inc.

The Challenges of the Materials Industries

Metals, glass, polymers, cement, wood, composites and other materials are the embryos of manufactured products — including airplanes, cars, electronic components, offshore platforms and medical devices. In supporting these products, the materials industries are at the core of the global economy. Getting better product performance often requires pushing for quality improvements of raw materials or switching to a totally new material. These industries, usually seen as employing traditional development processes, are in fact using some of the most advanced modeling techniques to continually improve manufacturing techniques and products and to create new materials with amazing properties. Numerical simulation has progressively demonstrated its importance as a critical technology for success in this fiercely competitive, though cautious, market.

Innovations and New Materials

The most impressive progress and breakthroughs have been the result of ideas that seemed, at the beginning, unlikely to lead to a major step forward. In today's extremely competitive environment, these innovations are more important than ever. In a risk-averse market, however, it is sometimes difficult to find the necessary flexibility and



Temperature distribution during the cooling of a dish. Using the Narayanaswamy model, it is possible to detect the peak stress related to the emergence of defects at the core of virtual glass products.

resources to demonstrate the potential of a new concept. Numerical simulation plays a role in the materials industries at every scale — from the study of how nanotechnology can touch the very structure of materials to the macroscopic-scale, hazard investigation of entire manufacturing facilities. Simulation is used to test potential innovative solutions to problems without jeopardizing business or safety. The emergence of new airplanes with much higher fuel efficiency [1], the design of a new race car or America's Cup yacht [2] and the development of advanced processes for extracting titanium [3] are all illustrations of this trend.

Seeing to the Core of the Manufacturing Process

Beyond these innovative activities, numerical modeling is used widely to improve, adjust, or troubleshoot existing processes, as well as for the routine inspection of existing device performance. These tasks are far from trivial, not because designers lack skills, but due to insufficient information such as local flow patterns or peak stresses at the core of the manufacturing device. Only supernatural powers or computer-aided engineering (CAE) modeling permit designers to actually see inside the solid structure to detect problems, such as possible excessive stress and risk of rupture due to fatigue, or specific flow recirculation that can lead to the dissolution of tiny additives in an aluminium furnace. Access to this type of information provides the designer with the necessary additional understanding to more easily improve a given process or fix an unfortunate problem.

Virtual Prototyping

While numerical modeling already has seen early successes in the materials industries, this solution alone is no longer sufficient to provide a clear lead in highly competitive markets. The innovators and successful players of the modeling world are now involved extensively in optimization analyses and parametric studies, not to mention Six Sigma analysis, in order to identify an optimized design before actually manufacturing any part. Experimental efforts are now part of the virtual world as well: Drop test, mechanical testing and behavior under normal or extreme service conditions are simulated on virtual prototypes. Virtual representations of new parts are completely designed, manufactured and tested before actually moving to the real world, with only the most promising prototypes making it into production.

Simulation Driven Product Development

The different steps described above have been used routinely in the materials industries with growing success, as the physics involved and the robustness of the solution were improving during the last few years. The trend that is clearly emerging is the need to integrate these different product development stages into a seamless virtual environment. Some solutions, such as the ANSYS Workbench platform, have been successfully developed to address this new requirement. These solutions are opening the door to the emerging concept of Simulation Driven Product Development. SDPD enables a new plastic part, a glass bottle or a concrete building structure to be modeled throughout an entire development process — from raw material production, to end-user part manufacturing, to final testing — in a common, unified environment.

We are at a threshold at which the virtual world will allow us to go seamlessly from conceptualization to end-product modeling, tracking every opportunity to produce better materials faster, cheaper, more safely and with a more environmentally friendly approach than ever before. ■

References

[1] "Software Delivers Faster Time-to-Part and Reduced Testing," *Reinforced Plastics*, Sept. 2007, pp 24-25.

[2] "The Simulation Race for America's Cup," *ANSYS Advantage*, Vol. I, Issue 2.

[3] "Tweaking Titanium's Recipe," *The Wall Street Journal*, Online, Sept. 10, 2007.

[4] "Modeling Dies for Rubber Parts," *ANSYS Advantage*, Materials Spotlight, Vol.1, Issue 4.

Shedding Light on **Auto Lamp Manufacturing**

Simulation saves time and resources in the development of an injection mold.

By Eliseu Sartori, ARTEB Industries S.A., Sao Paulo, Brazil
Cesareo de La Rosa Siqueira, ESSS, Florianopolis, Brazil

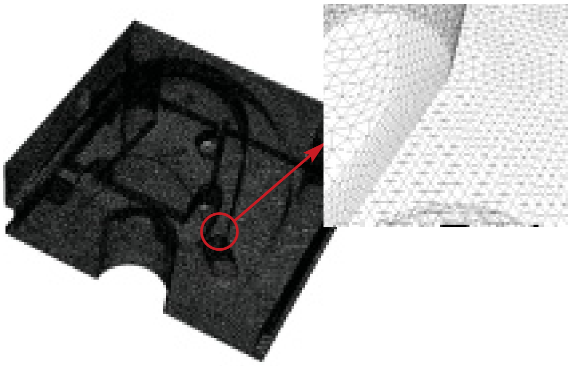
In the standard automotive lamp, reflector optics are responsible for distributing the light beam out of the lamp and toward the road. One common method for manufacturing the reflector section of the lamp is to use an injection molding process. When it comes to developing and optimizing the effectiveness of the molding process, one way to ensure consistent heating and shaping of the injected plastic is to incorporate thermal resistances into the mold itself.

The traditional method of placing these resistances relies on a trial-and-error procedure, based mainly on previous experience. Temperatures on the surface of the mold are then measured and compared to the allowable values. In order to break the paradigm of predicting the position of these resistances based on an almost empirical procedure, ARTEB Industries S.A., a Brazilian automotive lighting company, chose to use FLUENT software to pursue a simulation-based, thermal conduction study. Simulation offered the ability to expedite the development process, replacing the time-consuming traditional alternative with a more efficient option and providing engineers with a better understanding of the physical aspects involved in headlamp design.

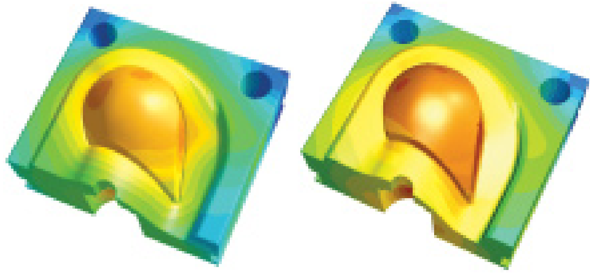
Using existing CAD geometry, a three-dimensional computational fluid dynamics (CFD) model was created. All the relevant aspects of the heat transfer phenomena were taken into account, and a conduction model was created based on the fact that the injection mold exchanges heat with the environment mainly through a natural convection mechanism. Mesh refinement was used near the mold surface in order to ensure a reliable discretization of the conjugate heat transfer mechanisms between the fluid and the solid structure representing the injection mold.

An initial simulation of the mold was pursued in order to examine a preliminary configuration of the thermal resistances. The analysis of the results showed an unacceptable temperature gradient at the surface of the injection mold in two locations. Such variation cannot be permitted, as it could cause thermal stress problems if this particular resistance configuration were to be used.

The surface temperature distribution that resulted from the initial configuration provided ARTEB engineers with the information needed to modify the position of these thermal resistances and create a new CFD model. The second analysis presented a temperature distribution with a much more uniform pattern along the entire reflector surface, decreasing the temperature gradient and creating a maximum observed temperature difference that was within an acceptable range over the entire mold surface. By using simulation, ARTEB designers found that they could reduce the time and resources needed to identify a reliable and accurate mold design. ■



Meshed geometry for an automotive headlamp mold



Surface temperature distribution on a headlamp mold shows initial resistance disposition (left) and a redesigned final disposition (right). The redesigned case showed a reduction in gradient across the headlamp region of the mold.

Image © iStockphoto/Lya Cattel

A New Spin on Cement

Simulation broadens understanding of dynamic separators in cement manufacturing.

By Lucas A. Kostetzer and Leonardo P. Rangel, ESSS, Florianopolis, Brazil
Joana B. Souza, Holcim Brazil, Sao Paulo, Brazil

For nearly all products, it can be said that the quality of the raw materials being used in creating those products affects the quality of the final item. In cement production, control of the incoming raw materials influences not only the end product, but also the efficiency of the manufacturing process and its associated costs. If not carefully managed, the size of the solid particles used in creating cement can have a significant effect on efficiency of the cement production line.

The regulation of particle size is handled by two processes in cement production: raw/cement milling, which grinds materials to decrease their size, and dynamic separation, which classifies and partitions particles by size. If particles are smaller than about 90 μm in diameter, the dynamic separator passes them on for further processing. Larger particles, which can have a detrimental effect on thermal and mass exchange during the later manufacturing stages, are returned to the raw milling stage for further grinding.

Holcim is a global supplier of cement, aggregates, concrete and asphalt products and services. To sustain its global positioning, the company has been trying to identify technology gains that can reduce operational costs and increase production efficiency. With the support of ESSS, Holcim Brazil chose to use computational fluid dynamics (CFD), specifically ANSYS CFX software, to analyze the gas–solid flow for one of their first-generation, traditional-design dynamic separators. The main objective of this simulation was to understand the separator performance, using computer modeling to replace expensive and time-consuming trial-and-error testing. This required that the particles' paths be fully simulated and classified as a function of particle diameter. The results of the simulation work were then used to suggest possible improvements in order to increase the separation efficiency.

The dynamic separator looks like a cyclone with adaptable deflectors, that are used to stop the flow of the larger airborne particles that are being swirled in the device. Ideally, the flow pattern in the separator consists of stratified, rotating patterns in which finer particles tend to reside in one area of the device (higher up) and the coarser ones tend to flow to another region (lower in the device).

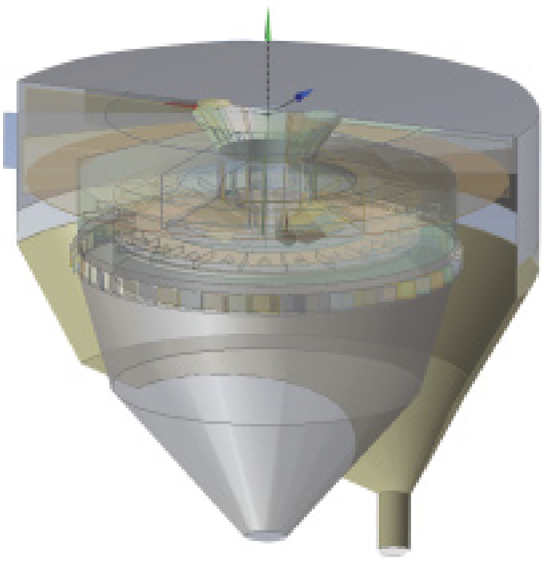
To effectively capture the larger particles and remove them from the flow pattern inside the device, particle classification and deflector positioning occur in strategic locations within the separator where the air velocity is of the same order as the particles' terminal velocities.

Due to the complex geometry involved, a mesh of 2.7 million nodes was generated using ANSYS ICEM CFD meshing software. The mesh was composed of tetrahedral elements with strategically placed prismatic element layers. Simulation of the stationary stage was performed using the ANSYS CFX SST turbulence model for the continuous phase. The dispersed phase was modeled using the Lagrangian approach with one-way coupling to the continuous phase. The dispersion of particles due to turbulence was also accounted for in the simulations, along with the particle drag and weight.

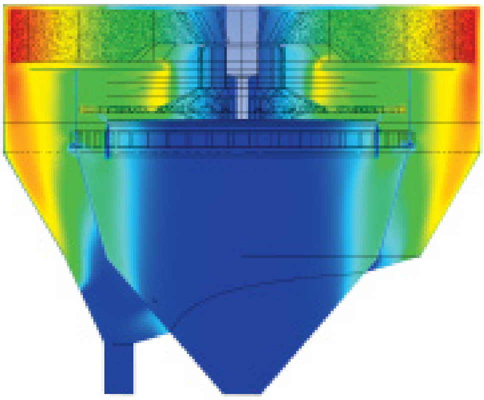
The particles' trajectories, operating pressures and mass flow values observed in the simulation agree with expected values. Particles can be observed as traveling along long trajectories inside the separator, influenced mainly by the blower rotation velocity (approximately 160 rpm) and its diameter (approximately 7 meters). Large recirculation zones have been predicted in places in which, eventually, small particles are retained and experience an increased residence time. While some trajectories of the larger particles appear to indicate that those solids have been dragged into the same current as the small particles, they eventually return to the large particle capture zones.

The CFD results allow Holcim engineers to compare the influence of particle classification on dynamic separator efficiency improvements. With simulation, they are able to gain an understanding of complex flow paths, influenced by changes in internal geometry and operating conditions, that would otherwise only be achieved through extensive trial-and-error testing.

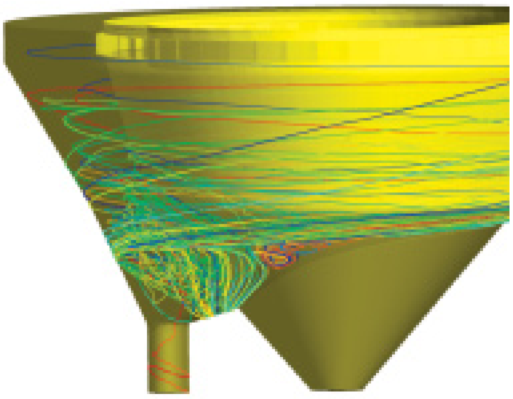
Prior to ANSYS CFX analyses, it had been impossible to understand completely the physical process occurring in the dynamic separator. With ESSS support, Holcim is able to effectively evaluate "what if" scenarios, supplement operators' knowledge and help managers to make decisions regarding costs and benefits of equipment and process modifications. With this methodology in place, Holcim Brazil can evaluate equipment modifications and identify improvements without repeated physical modification of existing equipment, thus reducing the cost of operational testing by approximately 30 percent. ■



The Holcim dynamic separator presented as a 3-D CAD geometry; particles enter through the top of the device and exit through the bottom.



Cross section of a dynamic separator showing contours of velocity. The top section is made up of a rotational bed, while the bottom region is a fixed bed with particle collectors.



Pathlines in a dynamic separator show recirculation zones representing the flow interaction and the residence time.

Layers of Strength

Simulation helps develop thinner composite materials using natural fibers.

By M. Nithiyakumar, Department of Textile Technology, SSM Academy of Textiles & Management, Erode, India
D. Gopalakrishnan, South India Institute of Fashion Technology, Coimbatore, India

Image © iStockphoto/jhorrocks

Fiber-reinforced composites have come a long way in replacing conventional materials like metals and woods. These types of composites are derived by combining fibrous material, which serves as the reinforcing material that primarily carries the load in the composite, with a matrix material, which bonds the fibers together, supports them and is responsible for transferring the load from fiber to fiber. The purpose of combining materials in this manner is to achieve superior properties and performance when compared to the individual materials. As truly engineered materials, designers of composites can select the composition to generate particular performance specifications based on individual application needs.

To improve the performance and durability of timber structures specifically, fiber-reinforced polymer (FRP) composites are one option that is increasingly used. Typical FRP composites found in the wood industry are composed of coir-ply boards with oriented jute (as face veneer) or coir combined with waste rubber wood (used as an internal layer). In these materials, phenol formaldehyde is often used as the matrix material.

To develop a design that meets the cost, weight and safety requirements for a specific application, a composite material's mechanical properties — especially its failure point — must be understood. In recent years, computer simulation has emerged as an effective approach for predicting load distribution and failure of composite materials.

Since each layer in a composite material may have different orthotropic material properties, special care is required in setting up the analysis. This includes choosing the proper element type, defining the layered configurations, and specifying failure criteria and individual layers' material properties. Solving fracture mechanics problems computationally involves performing a linear elastic or elastic–plastic static analysis and then using specialized post-processing commands or macros to calculate desired fracture parameters. Each of the products in the ANSYS family of mechanical simulation solutions allows for the modeling of composite materials with specialized elements, called layered elements, that support nonlinearities such as large deflection and stress stiffening. To learn more about FRP composites, researchers used these elements in the creation of five finite element (FE) models of varying thickness and layer configurations for analysis.

For each model, the tensile stress and deforming behavior were analyzed. Input data — such as density, Young's modulus, shear modulus and Poisson's ratio — was provided. The coir and jute, which was impregnated with phenolic resin, were considered as isotropic materials. Wood was considered as an orthotropic material.

The simulations were divided into two sets of material combinations. Set 1 included three simulations of FRP composites that were composed of an increasing number of layers of phenolic coir and wood. Set 2 used two of the three compositions from Set 1 but replaced a wooden layer in the composition with a phenolic jute layer.

The Young's modulus of the phenolic coir is only 4.3 GPa, while the modulus value of the wood is as much as 16 GPa in the direction of the grain — but only 1 GPa in the direction perpendicular to the grain. The modulus of jute is up to 40 GPa, and that of phenolic jute is 7.5 GPa, which is higher than that of phenolic coir. When deformed, the elongation of the coir is as much as 40 percent, while the elongation for jute is only 1.5 percent. This indicates that the introduction of the phenolic jute layers into the FRP composite should lead to significant improvement in tensile stress values.

The simulation results showed that the load was distributed to all the layers and throughout the entire parts. The edges were well gripped and there was no deformation. The deformations were controlled by the composite energy of the material. Even though the layers were individually separate, their movement or tendency for separation was controlled by the adjacent layers.

From the results of the simulation, it also appeared that, regardless of composition, the tensile stress gradually increased with thickness. The deflections seen in the models that did not include phenolic jute layers (those in Set 1) were higher than those seen in the equivalent models from Set 2. One thing that was noted was that the thicker of the two models in Set 2 used a thicker phenolic jute layer than the other to replace the wood layer. This was important because it explained what appeared to be a reduction in deflection that occurred with an increase in overall material thickness, which did not otherwise match the trend seen in the data.

The thickest composite model from Set 2 was found to have a tensile stress greater than its equivalent case from Set 1, though the deformation was the same. The same model from Set 2 was also found to have tensile strength equivalent to a thicker model from Set 1, but with the same deformation. This meant that, in comparison, the option from Set 2 could provide a decrease in the amount of material used without a significant variation in tensile strength and deformation.

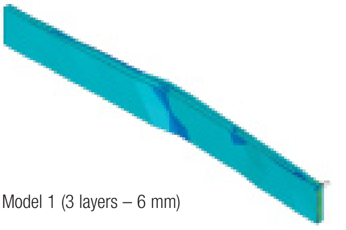
Using mechanical simulations and coordinated experimentation, it was concluded that even though the tensile stress values of computational models cannot be accurately compared to the real boards, due to the non-homogeneous nature of the real material, the deformation tendency is the same for both. In this way, mechanical behavior could be correlated. In this study, it was also demonstrated that inclusion of the phenolic jute layers gave composite boards higher stiffness and eliminated the need for one layer of wood in these particular materials. Simulation helped demonstrate that stronger mechanical properties could be obtained at lower thicknesses, leading to benefits such as reduction in cost and weight. ■

References

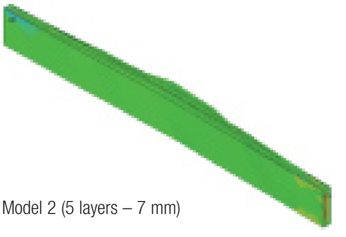
[1] Nithiyakumar, M.; Gopalakrishnan, D., "Development and analysis of jute and coir reinforced composites," www.fibre2fashion.com/industry-article/1/89/development-and-analysis-of-jute-and-coir-reinforced-composites1.asp, 2007.

Model	Set	Description	Total Thickness of Model (mm)	Max. Stress (Mpa)	Deflection (mm)
M1	1	3 layers (2 phenolic coir + 1 wood)	6	90.9	2.14
M2	2	5 layers (2 phenolic coir + 1 wood + 2 phenolic jute)	7	184	4.3
M3	1	5 layers (3 phenolic coir + 2 wood)	10.5	215	6.9
M4	2	7 layers (3 phenolic coir + 2 wood + 2 phenolic jute)	12.5	241	3.8
M5	1	7 layers (4 phenolic coir + 3 wood)	15	264	8

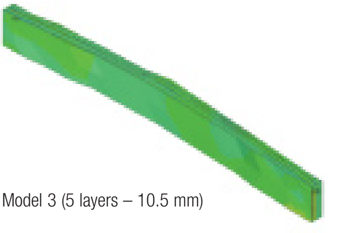
Description of simulation models and results for FEA simulations of FRP composites



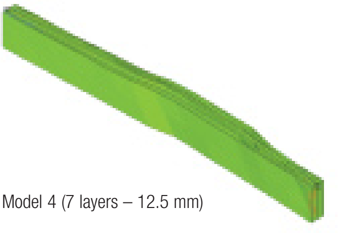
Model 1 (3 layers – 6 mm)



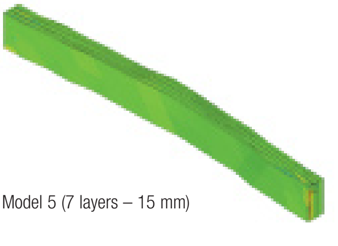
Model 2 (5 layers – 7 mm)



Model 3 (5 layers – 10.5 mm)



Model 4 (7 layers – 12.5 mm)



Model 5 (7 layers – 15 mm)

Tensile stress for the five models of jute and coir reinforced composites studied

Modeling Dies for Rubber Parts

Computer simulation helps reduce extrusion cost by 50 percent.

By Adnan Saeed, Derby Cellular Products, Connecticut, U.S.A.

Derby Cellular Products specializes in producing molded, extruded and fabricated ethylene propylene diene monomer (EPDM) polymeric seals for automotive, truck, agricultural, off-road, air filtration and appliance markets. The company recently found it difficult to profitably produce an EPDM environmental seal for a complex geometry. The initial plan was to produce the part using a single-cavity die. As a result of the complex geometry, approximately 20 trials on the extrusion line were required at a total cost of about $16,000. Rather than continue attempts to optimize the single-cavity die, Derby Cellular Products decided to use computational fluid dynamics (CFD) to evaluate whether or not the part could be produced using a two-cavity die. This application would also serve as an excellent platform from which to assess CFD's capabilities in general.

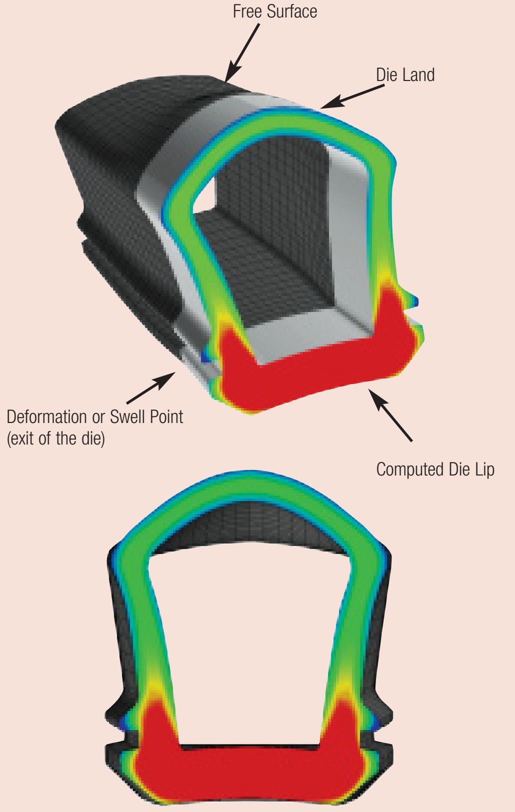
CFD simulation of extrusion processes provides information about flow patterns, including velocity, pressure, shear stress and temperature values, as well as hang-up areas within complex dies and manifolds. As part of the analysis, the die designer may investigate how changes in the die internals or feed ports affect the exit flow uniformity (die balance). CFD can also be used to calculate the extruded shape from a given die, facilitating detailed parametric studies that significantly reduce experimentation, design cycle times and costs.

Most CFD software is unable to handle complex polymer flow problems involving nonlinearities such as viscoelasticity, shear-thinning, viscous heating, free surfaces and irregular geometries. POLYFLOW software from ANSYS, Inc. was selected to perform the analysis because it provides the above capabilities, as well as a unique inverse die design feature. After specifying the desired profile shape and material properties, this feature instructs the software to compute the required die lip and adaptor shapes. POLYFLOW software also provides a library of rheological models capable of dealing with the material nonlinearities involved in polymer flows.

The dual-cavity extrusion die was modeled by importing a solid model of the initial concept into POLYFLOW software for die flow balancing. The geometry, which was in STEP file format, was imported into the GAMBIT pre-processor for geometry cleanup, identification of the fluid volume and creation of the volume mesh needed for CFD analysis. The boundaries for the 3-D CFD die model included a back plate wall containing two inlets, a middle transition or adapter plate wall and, finally, the die land wall and material outflow area. The two cavities begin in the die plate. The die land length plays a vital role in determining the die swell factor. During the mesh creation process,

Image @ iStockphoto/kickstand

Polymer seals produced by Derby Cellular Products are used in a variety of automotive, agricultural, filtration and appliance applications. One such application is the door seal for a dishwasher.

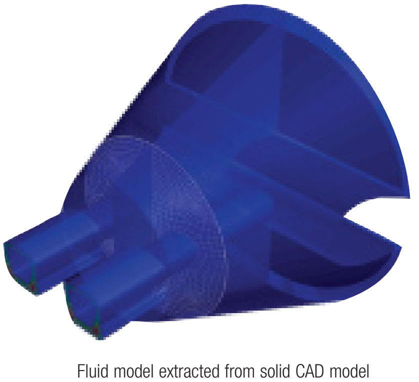


Velocity profile across a die land inlet section. Using the inverse die design capability of POLYFLOW software, the die land can be automatically created to generate the correct extrudate after all deformations.

particular care was taken to ensure that the mesh was fine enough to provide adequate coverage and solution detail of the thinner regions.

Initially, the flow of polymer through the die was simulated to balance the flow between the two cavities. Then, several cases were modeled in which the effect of modifying operating conditions on flow behavior through the die was examined. It was determined that increasing the die land length reduces the swell factor of the polymer because the additional residence time in the die land area enables the polymer molecules to properly align along the flow direction.

After balancing the die completely, an inverse extrusion analysis was run to determine the die lip geometry. First, simulations were run to examine the role of the wall slip factor on the resulting profile and to determine the wall slip coefficient, which plays a critical role in the inverse extrusion analysis used to calculate the correct amount of die swell. While classical Newtonian fluids consisting of



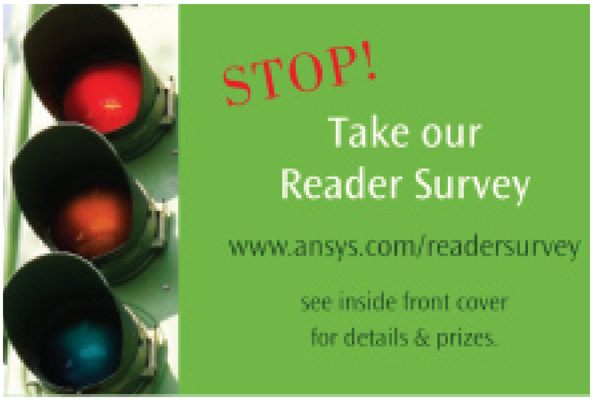
Fluid model extracted from solid CAD model

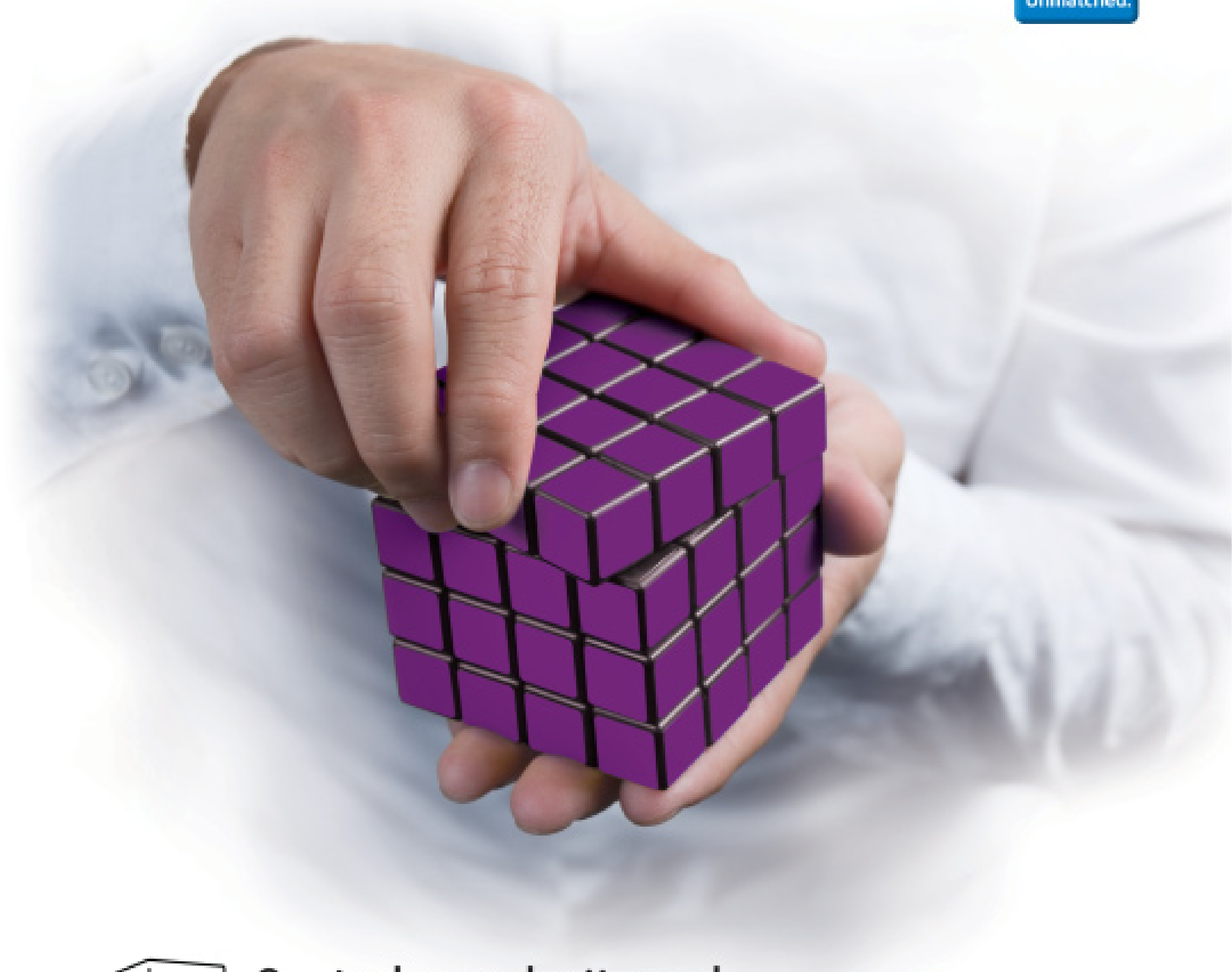
Dual extrusion of ethylene propylene diene monomer seal

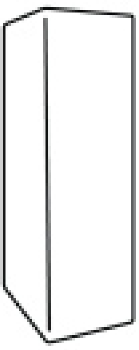
small molecules tend to stick to a wall, polymeric fluids consisting of long molecular chains have a tendency to slip along a wall at high pressures, and the amount of slip has an important effect on the extrusion process.

To optimize die lip geometry, a 3-D CAD model consisting of two sub-domains was constructed. One sub-domain contained the die land and the other contained the free surface that extended beyond the die lip area. An assumption was made that the polymer would slip along the die wall at a speed that is 25 percent of the average inflow velocity. Several slip coefficients were tried until one was found that achieved the desired 25 percent velocity ratio. This value was then used to run the inverse die extrusion and determine the lip geometry, while simultaneously determining the swell factor of the material coming out of the die lip and adjusting the die land geometry required to achieve the desired product shape. Using this die lip, the simulated adapter and back plate inlets were balanced for both cavities.

Finally, the dual-cavity die was fabricated and run on the extrusion line. The resulting profile shape was close to, but not exactly the same as, the desired geometry. As a result of the discrepancy in the profile shape, the polymer was slipping more than 25 percent against the die wall. Subsequently, the model was adjusted by changing the slip coefficient to a level that represented the actual conditions. The inverse die simulation was then repeated and a revised lip geometry was generated. Following this simulation, a new dual-cavity die was fabricated, and, when tested on the extrusion line, it produced a profile that matched the desired geometry. By using simulation to replace the time-intensive trial-and-error process employed previously, the cost of producing the part was reduced by approximately 50 percent. ■







Control complexity and complete projects twice as fast

When your applications all run in a single, powerful data environment designed specifically for engineering—projects can move from design to simulation, then on to completion, with ease. That's why we've built our storage and server solutions—including innovative Altix® XE servers based on the Quad-Core Intel® Xeon® 5300 processor—to work together and control complexities, even in the most challenging engineering environments.

Learn more at sgi.com/go/connectansys

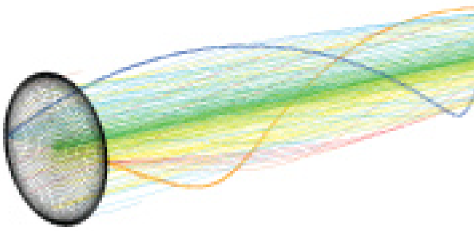
sgi
INNOVATION FOR RESULTS

©2007 SGI. All rights reserved. SGI is a trademark of Silicon Graphics, Inc., in the United States and/or other countries worldwide.
Intel, Intel Logo, Itanium, and Itanium Inside are trademarks or registered trademarks of Intel Corporation or its subsidiaries in the United States and other countries.

Turning Up the Cool Factor in **HVAC Systems**

Designers simulate flow through microfin tubes to analyze the effectiveness of heat exchangers.

By Yih-Jeng Teng, Chee-Onn Chan and Teng-Kiat Lim,
O.Y.L. R&D Centre Sdn. Bhd., Selangor, Malaysia



Pathlines of the particle flow inside the microfin tube colored by particle ID

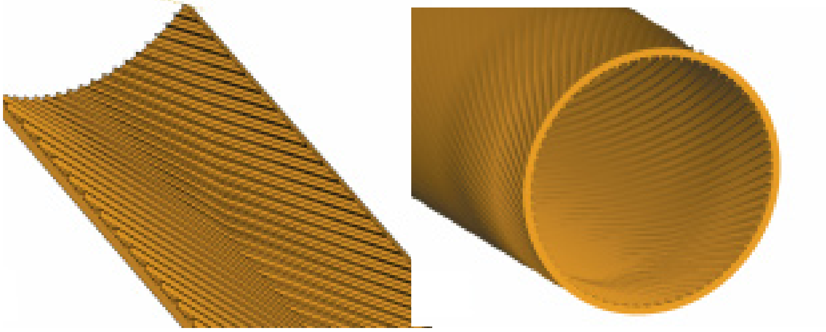
Manufacturers in the heating, ventilation and air conditioning (HVAC) industry are constantly looking for ways to enhance the heat transfer capabilities of their systems. In refrigeration and air conditioning systems in particular, the cooling fluid absorbs heat from the surrounding air by a process known as direct expansion evaporation. Heat from the surrounding air (the “hot” fluid) is thus transferred to the evaporating coolant via the walls of the metal tubes through which the coolant flows.

At the O.Y.L. R&D Centre in Selangor, Malaysia, designers have been analyzing coolant flow through microfin tubes within the evaporators. The microfins themselves are raised ridges less than one millimeter high that twist around the interior of the tubes in the pattern of a helix. The advantages of such tubes are that they increase the surface area that is available for heat transfer while causing a relatively small decrease in pressure. Tubes with microfin walls have been proven to be more effective in refrigeration and air conditioning systems compared to tubes with smooth walls.

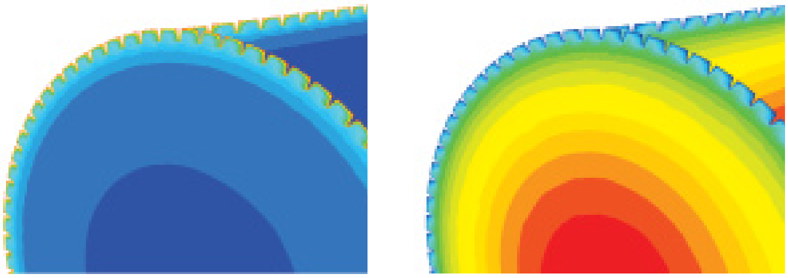
Using the FLUENT computational fluid dynamics (CFD) software package, O.Y.L. successfully simulated the flow of water inside a microfin tube. A constant heat flux was applied at the tube wall to provide the heating. Due in part to a tangential velocity component, the flow inside a microfin tube is relatively complex and involves swirl. The Reynolds stress model (RSM) was therefore chosen to model this turbulent flow. Only the length of one full helical revolution of the fins inside the tube was modeled because the flow was expected to repeat periodically. The inlet and outlet of the tube were thus set to have periodic boundary conditions.

The simulations showed that a swirling flow developed inside the microfin tube and became more significant near the wall region. Swirling flows enhance heat transfer in the tube, but at the same time also increase the pressure loss due to friction when compared to the smooth tube. The temperature and velocity profile of the microfin tube not only varied in the radial and longitudinal directions, but also in the tangential direction.

Both the friction factors and heat transfer coefficients calculated from the CFD results were validated with the experimental results obtained by O.Y.L. researchers in a previous study, and were within 15 percent of the measured values. The FLUENT models were thus able to predict the heat transfer and pressure drop performances with satisfying accuracy and serve as a very useful tool to analyze the flow inside a microfin tube. ■



Close-up views of microfin tube geometry including a cutaway section (left) and a full cross section (right)



Close-up view of contours inside a microfin tube, temperature (left) and velocity (right)

Keeping the **Space Race** from Heating Up

Coupled multiphysics simulation saves hundreds of thousands of payload-equivalent dollars per launch for SpaceX.

By Michael Colonno, Chief Aerodynamic Engineer, SpaceX, California, U.S.A.

Second demo flight of the SpaceX Falcon 1 launch vehicle

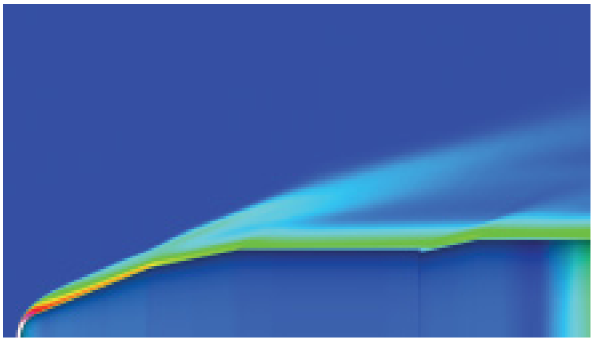
Space Exploration Technologies (SpaceX), a privately funded rocket venture founded by entrepreneur Elon Musk, is developing its Falcon family of launch vehicles from the ground up. SpaceX aims to change the paradigm of space flight by introducing a family of launch vehicles that will ultimately reduce the cost of space access by a factor of ten. As designers of the first launch vehicle developed entirely in the 21st century, Falcon engineers have the opportunity to take advantage of the latest design and analysis technologies.

One area of concern for the engineering design team was the amount of thermal insulation required to protect the payload and sensitive internal equipment from heat generated by high-speed atmospheric flight. Thermal insulation is required to protect the vehicle's metallic skin, in addition to sensitive electronic equipment that can malfunction at high temperatures. At higher temperatures, metals can lose critical structural performance as their material properties change. During the first demonstration flight, engineers erred on the side of safety by installing a conservative amount of thermal insulation. Optimizing the amount of insulation used for future flights is vital because every pound of excess insulation reduces the payload of the vehicle.

To optimize the insulation prior to the second demonstration flight, a simulation that integrated computational fluid dynamics (CFD) and finite element analysis (FEA) was performed to calculate surface and body temperatures expected during the flight. The main challenge of the coupled simulation was passing the heat loading for the surface of the launch vehicle, calculated by the CFD code, to the ANSYS Mechanical FEA model for structural and thermal analysis, and then passing the skin temperatures calculated by the FEA code back to the CFD analysis.

The temperature distribution on the launch vehicle throughout the entire flight was determined to ensure that sensors, instruments, propellant lines and other critical components were maintained at safe temperatures. Launch vehicle aerodynamics are uniquely challenging because there is no cruise condition. Instead, the conditions change continually and rapidly during the critical few minutes in which the rocket moves from sea level to the near-vacuum conditions at the edge of the atmosphere. The maximum heat transfer typically occurs high in the atmosphere when the speed of the launch

The Falcon 1 launch vehicle

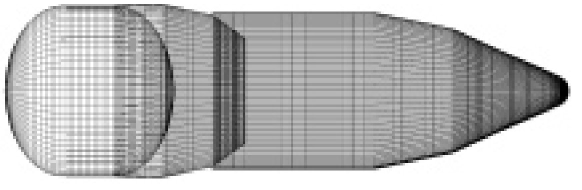


CFD contours demonstrate the heat load on the surface of the launch vehicle 152 seconds into the flight.

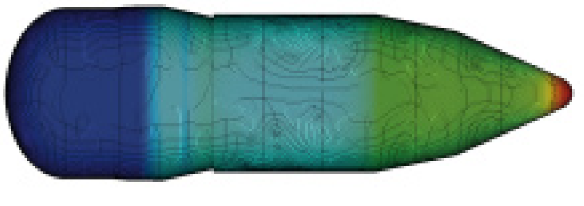
vehicle is very high and the density of air is very low. Heat transfer typically drops off to a much smaller value as the launch vehicle approaches its designated orbit.

With conditions changing so quickly during the atmospheric flight, multiple iterations of the simulation were required to capture the physics. Each of these iterations required both a fluid simulation of the air around the launch vehicle and a structural/thermal simulation of the launch vehicle itself. Since each code is dependent upon the results of the other, a series of iterations must be performed at each time step in order to converge to an accurate solution. Finally, since the amount of thermal insulation used also affects the heat transfer in the launch vehicle and the resulting vehicle skin temperatures, multiple repetitions of the entire simulation were required to examine the effect of varying the amount of insulation used.

SpaceX engineers used a CFD code designed for the high Mach numbers experienced during the launch vehicle's flight through the earth's atmosphere. The CFD code was used to calculate the increase in vehicle skin temperature that occurs due to the interaction between the vehicle body and the air through which it passes. The values for heat loading on the skin that are produced by the CFD code were then used as inputs for the ANSYS Mechanical simulation. This structural/thermal analysis then modeled the dissipation of the heat into the insulation and launch vehicle, which resulted in a new set of values for skin temperatures. These temperatures were then mapped back to the CFD simulation and the two codes run sequentially until they converged, such that the CFD results for heat loading were consistent with the skin temperatures determined by the structural/thermal code.



FEA mesh created for the Falcon 1



These contours represent the initial skin temperatures used for FEA simulation of the Falcon 1 launch vehicle.

SpaceX engineers developed a Matlab® routine, which automatically controlled the analysis by generating the appropriate input files for each code. These inputs included the correct atmospheric conditions and insulation amounts, in addition to mapping the results from each code to the other. The ANSYS Mechanical parametric design language (APDL) greatly simplified automation of the simulation process by providing a computer-aided design-based (CAD) application programming interface that enabled the integration code, in this case Matlab, to draw the model with a fraction of the number of commands — the launch vehicle model was defined with just a few lines of code. Using APDL, engineers were also able to create layered shell elements to represent the various layers of the skin of the launch vehicle, reducing solution time as compared to using solid elements.

After SpaceX engineers automated the process of running thousands of iterations, the minimum amount of insulation that would protect the launch vehicle was calculated to be approximately 50 pounds less than the amount used in the first demonstration flight. The temperature measurements taken during the second demonstration flight in March 2007 closely matched those predicted by the simulation, demonstrating that the reduced insulation performed as expected and within the requirements of the design.

As a result of simulation, the reduction in insulation weight makes it possible to increase the payload capacity of future Falcon 1 missions. With a number of launches scheduled over the next few years for the Falcon 1, and the larger Falcon 9 as well, SpaceX can now expand its range of potential customers and increase revenue on a payload mass equivalent basis. ■

Tiny Hearts and Lungs Get an Assist

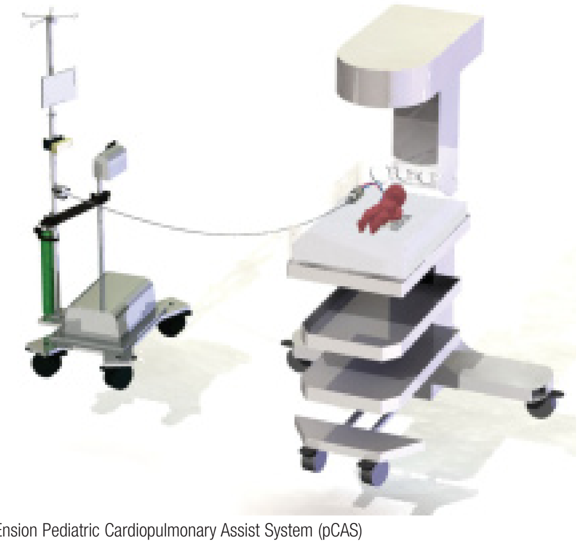
Blood-contacting portion of Ension pCAS with integrated blood pump and blood oxygenator components

Designers use simulation to improve pediatric circulatory support techniques.

By Mark Gartner, Ension, Inc., Pennsylvania, U.S.A.

In the United States, approximately two percent of all newborns require some form of corrective surgery for a cardiac or respiratory anomaly. A smaller yet significant number of these children require extended cardiopulmonary assistance for periods of days or weeks due to under-developed anatomy or as support until corrective surgery can be performed. The circulatory support technique known as extracorporeal membrane oxygenation (ECMO) has been used in more than 31,000 neonatal and pediatric patients worldwide with an overall survival rate exceeding 66 percent. However, with an international patient volume of less than 700 cases per year, technology investment has been limited. As a result, the devices used in ECMO have remained virtually unchanged since 1976.

In an attempt to stimulate advancement of pediatric circulatory assist devices and techniques, including ECMO, the National Heart, Lung and Blood Institute (part of the U.S. National Institutes of Health) awarded five contracts to develop a range of technologies aimed at improving the options for care of these smallest of patients. Ension, Inc. was awarded one of these contracts to develop a next-generation ECMO system that focused on ameliorating the shortcomings associated with current technology.



Ension Pediatric Cardiopulmonary Assist System (pCAS)

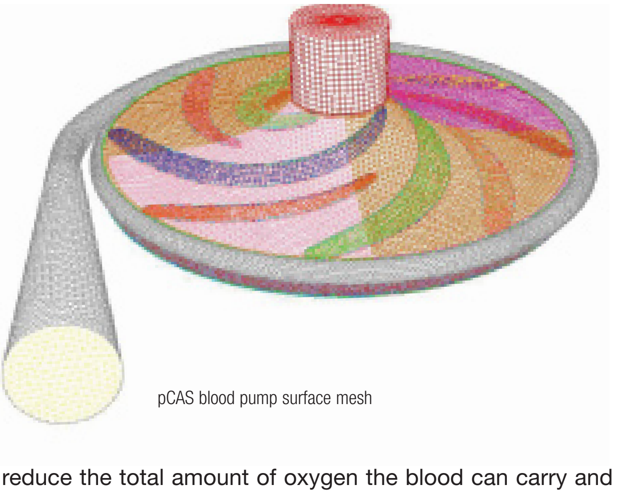
Ension's contract award led to the development of its pediatric cardiopulmonary assist system (pCAS). The pCAS device is mainly composed of an integrated blood pump and membrane oxygenator connected directly to the patient's circulatory system. Venous blood is removed from the patient and pumped through the pCAS device, where oxygen is added and carbon dioxide is removed. This oxygenated blood is then returned to the patient via an artery.

Extraordinary care must be exercised during the design of any blood-contacting component so that areas of flow recirculation, stagnation, excessive shear stress and residence time — all of which can damage the blood itself — are eliminated. Thus Ension designers desired to both predict the hemodynamic and mass transfer performance of pCAS prototypes and provide an estimation of device-induced blood damage prior to actual fabrication. As a result, the use of computational fluid dynamics (CFD) was proposed to aid in the design and analysis of the blood pump and membrane oxygenator portions of the pCAS. Due to the complexity involved in simulating the flow and mass transfer of blood, ANSYS, Inc. was engaged in a consulting role to assist with several facets of the program, including preparation of the initial CFD simulation files for the pump and implementation of user-defined function (UDF) routines.

Hybrid hexahedral and tetrahedral meshes of both the pCAS blood pump and membrane oxygenator were created using the GAMBIT pre-processor. After the completion of meshing, the miniature centrifugal pump component was modeled with the FLUENT software package, using the moving reference frame formulation to simulate pump rotation. Supplementary test conditions were simulated by Ension and verified against experimental results. Ension then modeled the pCAS hollow-fiber membrane oxygenator component using the porous media model in FLUENT software. A porous media approach was required because of the disparate length scales present in the device. In this instance, the microporous hollow fibers, which number about 3,000 in a pCAS prototype, possess a diameter of about 0.3 mm; however, the diameter of the pCAS oxygenator itself is over 50 mm. This physical characteristic, coupled with the lack of geometric symmetry, meant that direct numerical simulation of flow and mass transfer in the membrane oxygenator was impractical. Therefore, the porous media model was used to model the area containing the fibers.

After achieving a converged and experimentally validated flow solution, a correlation-based model for blood oxygenation and a basic diffusion model for carbon dioxide removal were implemented using the FLUENT UDF capability. UDFs allowed specification of important parameters such as blood temperature, pH and hematocrit, which is the proportion of blood volume occupied by red blood cells. This UDF enabled Ension to predict the spatial concentrations for both oxygen and carbon dioxide throughout the flow domain and also allowed for the evaluation of changes in device geometry prior to the complex and time-consuming fabrication process. Plots of the oxygen delivery and carbon dioxide removal versus the blood flow rate demonstrated good agreement between CFD-based mass transfer prediction and experimental data derived from a benchtop mock circulatory loop.

The last area of computational investigation was an estimation of device-induced blood damage in the pump. Blood hemolysis — the rupture of red blood cells — is a key factor affecting the success or failure of any blood-contacting device. Even moderate levels of hemolysis

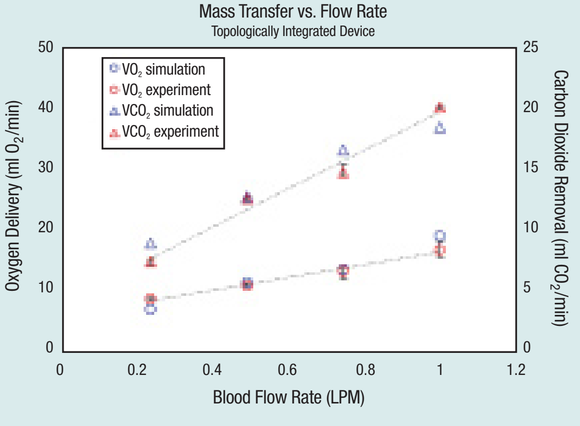


pCAS blood pump surface mesh

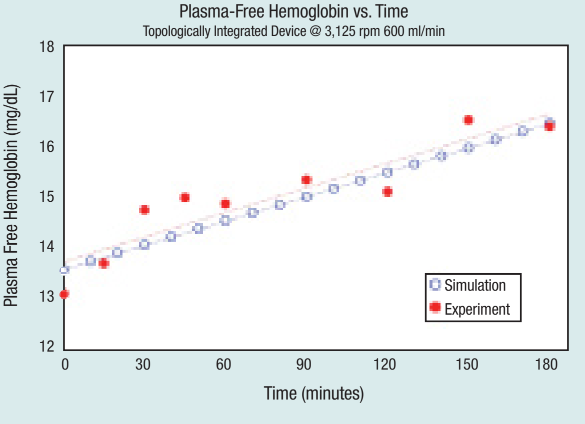
reduce the total amount of oxygen the blood can carry and can cause other kinds of deterioration. Hemolysis is usually calculated using particle tracking techniques, as a function of shear stress history and exposure time for a red blood cell. Once the converged and validated fluid flow solution was obtained, Ension used the discrete phase model (DPM) in the FLUENT product to track the hemolysis along the particle tracks traveling from the pump inlet to the outlet. A UDF was written to perform the necessary calculations along each particle track, which ultimately provided a prediction for the amount of blood damage experienced per pass through the device.

The predicted blood hemolysis again showed good agreement with experimental data for the pump speed and corresponding blood flow rate being studied. In the final analysis, a comprehensive CFD model was realized using the GAMBIT pre-processor, FLUENT simulation software and associated UDF functionality. The model served as a flexible tool that Ension could use as an adjunct to other methods for pCAS performance optimization. ■

This project has been funded in whole or in part with federal funds from the National Heart, Lung and Blood Institute, National Institutes of Health, under Contract No. HHSH268200448189C.



Predicted versus experimental mass transfer (oxygenation and carbon dioxide) for Ension's pediatric cardiopulmonary assist system

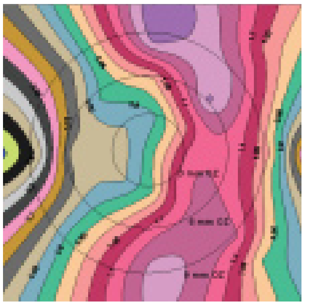


Simulation-predicted blood hemolysis versus experimental values for a cardiac assist device developed by Ension

Simulation for **Surgical Precision**

Modeling of LASIK plume evacuation devices increases accuracy of laser surgery.

By Fred Farshad and Herman Rieke, Chemical Engineering Department, University of Louisiana at Lafayette, Louisiana, U.S.A.
Leon C. LaHaye, M.D. CEO, Vision Pro LLC, Louisiana, U.S.A.

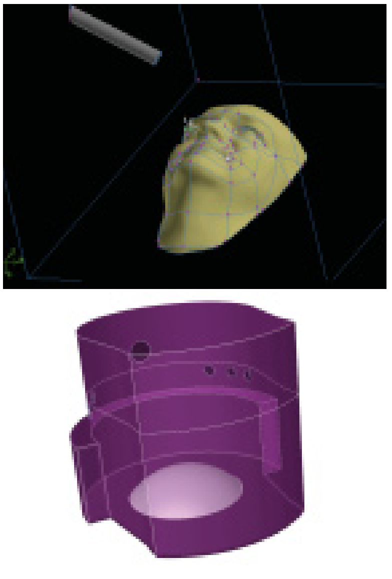


Contour map of the velocity field above the eye as measured experimentally for the proximal plume removal system

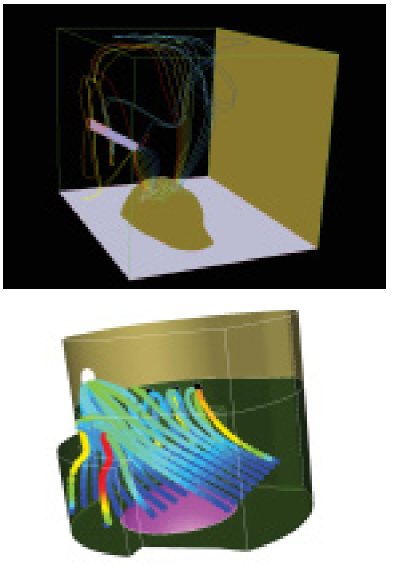
Laser-Assisted *In Situ* Keratomileusis (LASIK) eye surgery involves the use of an excimer laser to correct refractive vision errors. The laser burns off, or ablates, corneal tissue, essentially re-shaping the cornea in a way that increases the focusing power of the eye. During the procedure, the ablated tissue forms a plume just above the eye. Removing this plume during treatment eliminates obstructions from the laser beam path, thereby increasing the precision and accuracy of the laser ablation process.

The majority of the LASIK surgery systems on the market today use a distal plume evacuation system that removes the plume with a vacuum nozzle located above and some distance from the eye. The efficiency of these systems, however, is susceptible to ambient air flow patterns. As a result, the air flow in the surgical suite must be tightly controlled to get optimal performance. To address this issue, a team in the Chemical Engineering Department at the University of Louisiana at Lafayette developed the LA*Haye*SIK™ system. Instead of using a distal plume evacuation system, the LA*Haye*SIK™ employs a proximal plume evacuation technique that completely surrounds the eye, providing for a more controllable flow environment during the surgical procedure.

To compare and contrast the proximal and distal techniques, the research team used a combination of experiments and fluid dynamics simulations. The experiments utilized solid models of human faces to mimic the presence of a patient, in conjunction with a sensor that measured the air flow patterns induced by each device. For the proximal system, speed contours measured during the experiments demonstrated that there is a relatively even air flow distribution above the eye, a clear benefit of this approach.



The vacuum nozzle in the distal plume evacuation system is placed far away from the eye (top). The La*Haye*SIK™ device is a proximal system that surrounds the eye (bottom).



Particle pathlines are shown for the distal (top) and proximal (bottom) plume evacuation devices, colored by particle ID and time respectively.

An understanding of the velocity field was only the beginning, however. The next step predicted the efficiency of the device in removing ablated material. An experimental approach for evaluating plume removal would be difficult to implement in a reproducible manner. Instead, the discrete-phase model in FLUENT software was utilized to calculate the trajectories of particles seeded in the plume region for both the proximal and the distal techniques.

Simulation results showed that in the distal system the particles follow air flow patterns that sometimes do not result in removal and can lead to beam-masking during the surgical procedure. For the proximal system, however, the simulations indicated that the particles flow away from the path of the laser beam and are captured by the evacuation holes located to the side of the eye. The proximal system not only increased the efficiency of particle removal, but also reduced interference caused by the ablated particles. Furthermore, CFD analysis using the FLUENT software product helped to evaluate and improve instrument features during the development of the new evacuation device, leading to a better optimized design and, ultimately, more effective LASIK surgery for patients. ■

Fuel Injection Gets New Direction

CFD helps analyze fuel–air mixing in modern gasoline engines.

By Chris DeMinco, Delphi Automotive Systems, New York, U.S.A.
Yong Yi, ANSYS, Inc.

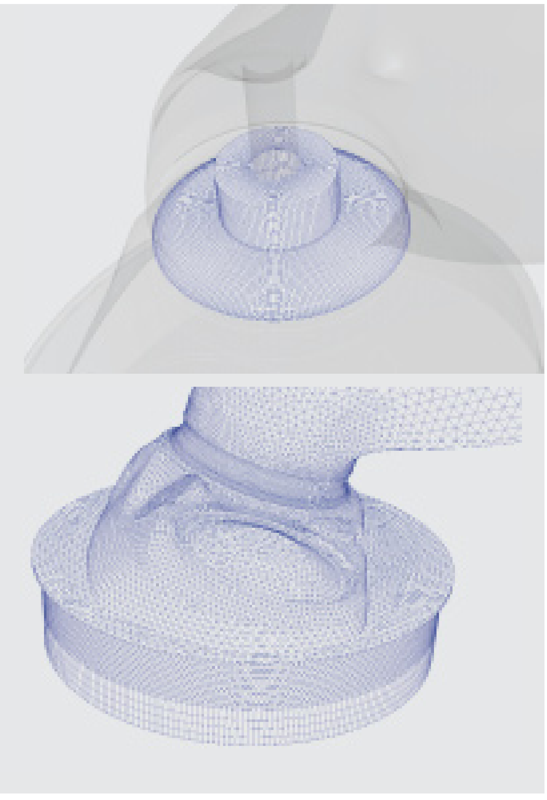
The geometry of the engine: the injector is located near the valves at the end of the engine cylinder as represented by the blue dot, while the spark plug is on the opposite side of the injector, located 3 mm inside the red dot.

Gasoline direct injection (GDI) engine technology has the potential to provide significant improvements in fuel efficiency while maintaining higher power output compared to port injection engines that are currently in mainstream usage. In direct injection, fuel is sprayed directly into the combustion chamber of each cylinder, whereas in multi-port injection, fuel is sprayed into the intake port from a location upstream from the intake valve.

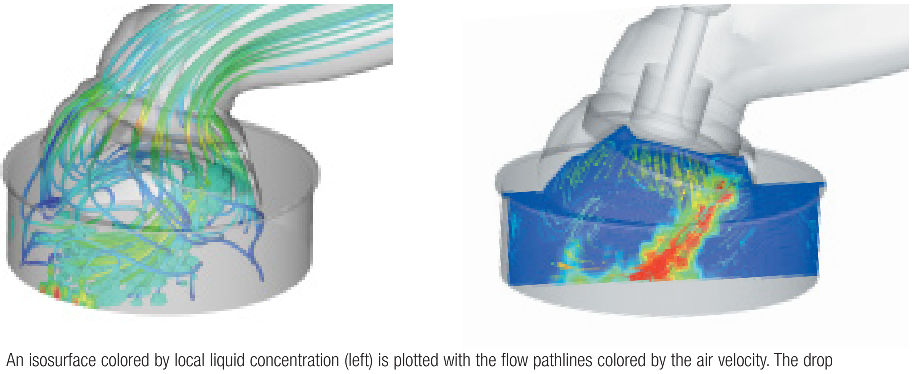
One benefit of using direct injection is that it produces improvements to fuel efficiency by offering more precise control over the fuel delivery process (amount and timing). One of the primary challenges with GDI engine design is to understand the behavior of the fuel injection and mixing processes, since the fuel spray process must be optimized to minimize evaporation time and distributed in a desired manner, under a variety of operating conditions and with a number of injection strategies.

To learn more about injection processes, a study was performed on a two-valve, four-stroke GDI engine provided by Delphi Automotive Systems based in Rochester, New York, in the United States. The study focused on understanding the injected fuel spray interaction with in-cylinder air motion. The effects of injection conditions on the ignition performance were considered by examining the mixture distribution in the combustion chamber and the equivalence ratio, which is the actual fuel-to-air ratio divided by the stoichiometric ratio at the spark plug location.

In order to simulate the engine's moving valve and piston accurately, a hybrid mesh was created using both GAMBIT and TGrid software products. The research team divided the mesh into two regions. The first region was a tetrahedral mesh in the upper portion of the combustion chamber near the valves. The second was composed of a hexahedral mesh used in the lower portion of the cylinder above the piston and in the port volumes, including the gap between the valve and valve seat. A layering model was used to simulate the valve motion: when the valve opens, extra cells are added to the grid volume between the valve and the valve seat, and when the valve closes, hexahedral elements are removed from the volumes between the valve and its seat. The team used a similar strategy to model piston motion.



Meshed areas for the valve (top) and combustion chamber (bottom)



An isosurface colored by local liquid concentration (left) is plotted with the flow pathlines colored by the air velocity. The drop size distribution (right) is plotted with velocity vectors of air on a plane cutting through the centers of the spark plug and the injector. Both plots show that the spray strongly changes the direction of the air flow.

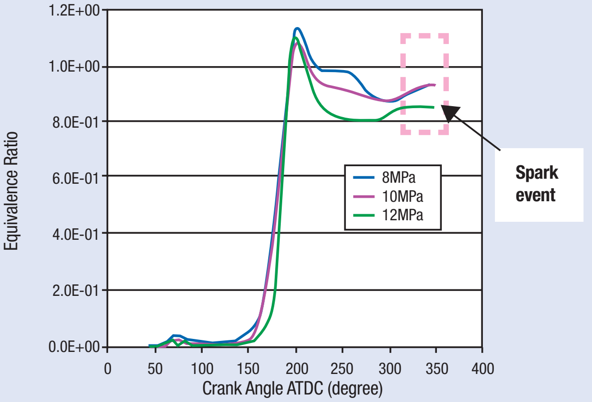
Once the meshing was complete, the motion of the intake valve and the piston was simulated using the dynamic mesh model in the FLUENT computational fluid dynamics (CFD) software package. The injector under analysis has six holes distributed in a U-shape and is located in the engine cylinder opposite the spark plug. However, the detailed structures of the injector and the spark plug were not modeled. Instead, the injector spray was simulated using the discrete phase model (DPM) and spray sub-models within the FLUENT package.

A uniform initial drop size distribution was computed from the measured discharge coefficient at the nozzle exit. The injection velocity was computed from the injection pressure and the discharge coefficient. The atomization and evaporation of the fuel jet was simulated using the variety of spray sub-models in FLUENT, including the drag, breakup, collision and coalescence, wall-impingement and evaporation models.

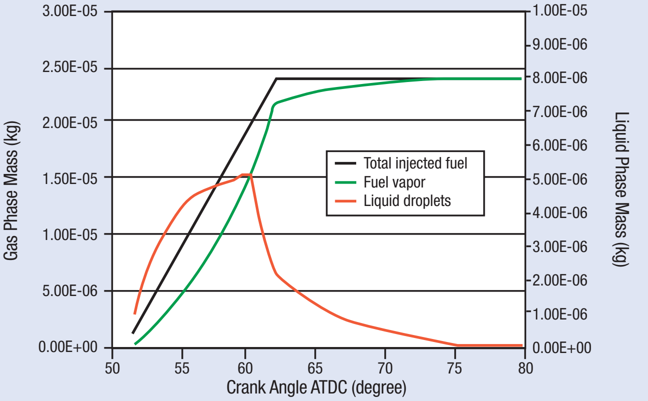
Since the fuel injection timing overlaps with part of the time during which the intake valve is open, the injected fuel will interact with the intake air flow. The FLUENT simulation showed that the fuel jet acts like a curtain and prevents a portion of the intake air from getting into the region of the cylinder below the spray. It also creates significant recirculation zones above and below the fuel spray. These flow patterns significantly impact the liquid evaporation and fuel–air mixing processes. As the liquid fuel spray evaporates, the fuel vapor mixes with air. The quality of the mixture, or the mass fraction that vaporizes, can be evaluated by examining the distribution of the equivalence ratio of the fuel–air mixture in the combustion chamber, especially near the spark location.

From the droplet distribution, the research team noted that many liquid droplets impinge on the piston and cylinder walls. This indicates the high possibility that a film of fuel forms on those surfaces. Due to the heat transfer between the wall film, the wall and its ambient gas, the wall film eventually disappeared. However, because of the weaker heat transfer behaviors and smaller liquid surface area for evaporation, it took much longer for the wall film to evaporate compared to the droplets in the free stream. Thus, the wall film has been shown to have a strong impact on the overall evaporation and mixing processes.

The spray wall impingement is an important factor in resolving accuracy in predictions of mixture quality. CFD models have provided a helpful tool for studying this phenomenon. The models also are proving to be an important asset in the advancement of GDI technology. ■



Equivalence ratio at spark plug location for injection pressure study. The dashed box indicates the possible spark timing. It is seen that all three injections form slightly lean, but still ignitable, mixture around the spark. The mixtures are moving to their stoichiometric ratios at about 180 degrees of crankshaft rotation, which indicates that later spark timing may be more favorable for a stable ignition.



Evaporation history of the fuel–air mixture. The mass of the wall film increases very rapidly at the start of injection since the piston is very close to the injector at that point. The wall film mass reaches the maximum value at a crank angle of 60 degrees after top dead center (ATDC), or about 10 degrees after the start of injection, after which the wall film starts to reduce.

Nowhere to Go but **Up**

A student uses simulation to reach new heights in secondary school education.

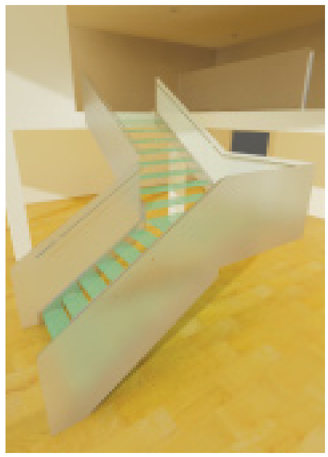
By Shane Moeykens, ANSYS, Inc.

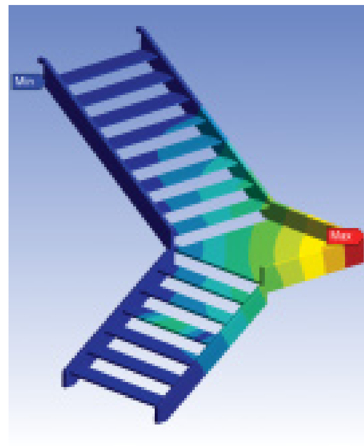
The Battle Creek Area Mathematics and Science Center (BCAMSC) is one of 33 regional centers in the Michigan Mathematics and Science Centers Network. Secondary school students attend these centers half of each school day for mathematics, science and computer education, and spend the remainder of the day at their home school for other course work. By the end of the four-year specialized program, every student will conduct at least one formal research project, and most students will engage in multiple research endeavors. Various courses are structured to accommodate a successful and unique research experience using problem-based inquiry so that each student can answer a question of interest.

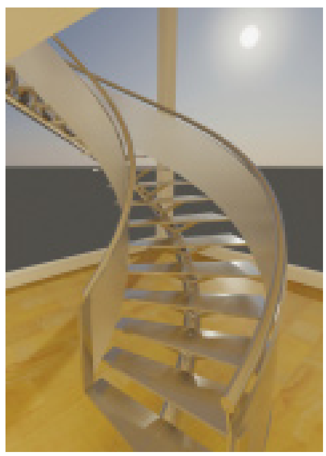
In his advanced physics class, Scott Taylor, a BCAMSC student with interest in architecture and structural engineering, decided to investigate floating staircase designs. "The primary focus of the investigation was to determine if the staircases would function," commented Taylor. "To answer this question, virtual prototyping seemed the obvious choice, recognizing that stress solutions allow you to predict safety factors, stresses, strains and displacement. My goals were to better understand design limits for staircases and to develop a working knowledge of a commercial FEA package."

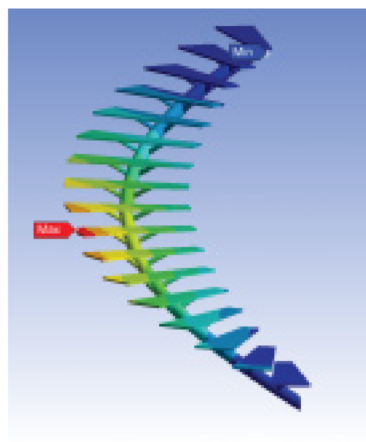
Taylor turned to ANSYS DesignSpace software (a feature of the ANSYS Academic Teaching Introductory product) to conduct this analysis, upon the suggestion from BCAMSC faculty. Two different designs were considered: an angled staircase and a curved staircase. The 3-D geometry models for each staircase were created using AutoCAD®. To simulate the effect of human loading, separate models were created in which a 60,000 pascal load, the equivalent of four to five people standing on a stair, was applied to individual steps along the length of the staircase. Deformation results from these human load cases were then calculated.

The Global Academic Program at ANSYS is a key enabler for schools such as BCAMSC to introduce students to engineering analysis tools. Karen Payson, Taylor's instructor in the advanced physics class at BCAMSC, commented, "It's great to see talented kids like Scott get excited about design by giving them the opportunity to gain a sense of how finite element packages work through hands-on application of the same analysis tools used by practicing engineers in the field. The obvious benefit of the ANSYS DesignSpace tool is that it allows students to answer their own questions via problem-based inquiry." Paul Lethbridge, Global Academic Program Manager for ANSYS, Inc., added, "The intuitive ANSYS DesignSpace product is a powerful enabler for exposing secondary education students to CAE tools. Analysis tools continue to permeate a much broader community than the core analysts, who were using this type of technology not all that long ago, largely driven by improvements in graphical user interfaces, workflow and CAD geometry interfaces." ■









As part of a high school research project, Scott Taylor analyzed loading variations for two staircase configurations: angled (above) and curved (below).

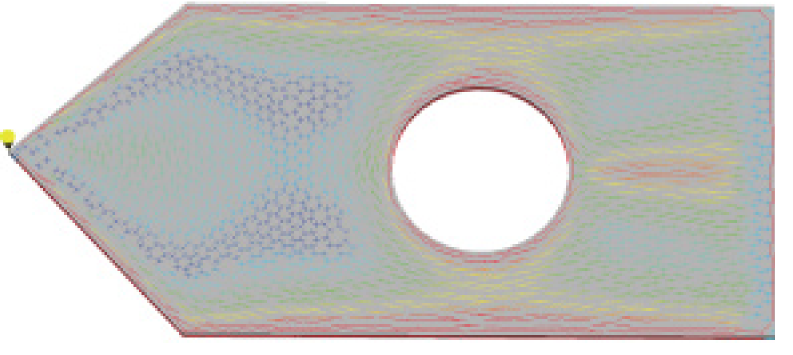
FEA results of staircase analysis

Making the Perfect Plastic Part

CAE analysis of material properties should be considered for metal-to-plastic replacement applications.

By Hanno van Raalte, MPI and MSA Product Line Manager, Moldflow Corporation, Massachusetts, U.S.A.

Because of the many inherent advantages in using plastic materials, there is an ongoing trend of replacing metal with injection-molded plastic parts in a wide variety of applications. More and more parts with critical end-use application requirements are becoming candidates for conversion to plastics. Plastics are lightweight, durable and corrosion-resistant; have a high strength-to-weight ratio; and, when used in transportation applications, for example, offer one of the easiest ways to increase fuel savings by making vehicles more lightweight.



Average fiber orientation in a simple injection-molded part is predicted by Moldflow simulation software. The fiber-filled plastic material is injected from the point on the left. Blue indicates more randomly oriented fibers, while red indicates highly oriented fibers, compared to the direction of material flow.

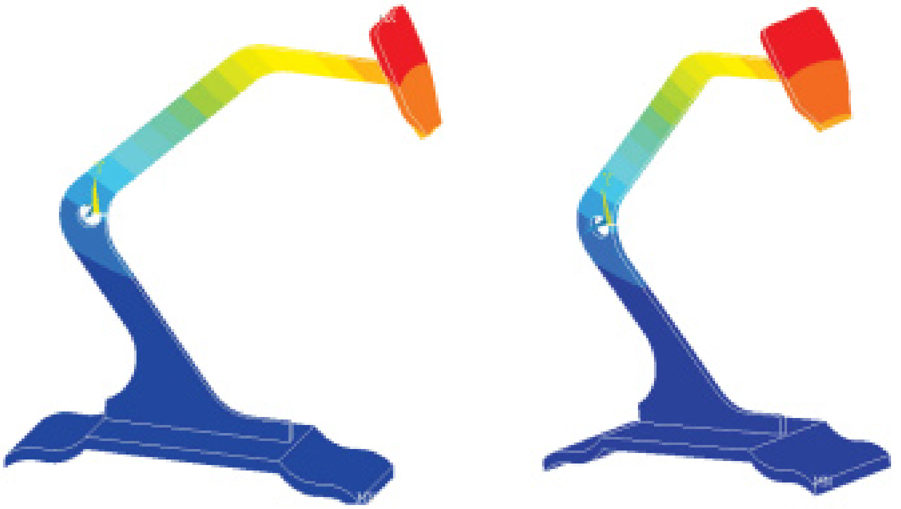
As plastics replace metals, the parts must be designed to take into account the properties of the specific plastic relative to the application requirements. One of the complicating factors for injection-molded plastic parts is that the properties of plastic materials effectively change during the manufacturing process. While this is not a problem in and of itself, problems can arise if the structural analyses are based on generic material data that does not accurately represent the actual properties of the molded part. These problems can include over-engineering of components, which can lead to unnecessary costs and material usage, or under-engineering, which can result in part failure.

Fiber-filled plastic materials are commonly used in metal replacement applications. When glass or carbon fibers are added to plastics, the elastic modulus can increase significantly with a negligible effect on part weight. This combination of low weight and high stiffness makes fiber-filled plastics ideal for high-performance applications.

The key to unlocking the potential of these plastics lies in the orientation of the fibers. The orientation direction and the degree of orientation of the fibers determine the mechanical properties of the molded part. In areas where fibers are strongly aligned, the material will have higher strength characteristics in that direction, but will be relatively weak in the perpendicular direction (across the fibers). In areas where the fibers are more randomly oriented, the material will not achieve maximum strength, though the strength properties will not depend as much on the loading direction, creating a more isotropic-like material.

During the injection molding process, the fibers in the plastic melt will orient in different directions under the influence of shear forces that are driven by the flow patterns. To demonstrate this, a simple case of flow across a plate with a hole in the center was simulated using Moldflow Plastics Insight® (MPI®) software. The results of this simulation show that the degree of orientation and the dominant direction of the fibers change throughout the part, especially in the area of the hole.

In the injection process, the polymer flow splits around the hole and then joins back together once past the hole, resulting in high fiber alignment and orientation just beyond the hole. The high orientation means that the material in this area has a relatively high modulus in the dominant direction of the fiber alignment (approximately 22 GPa).



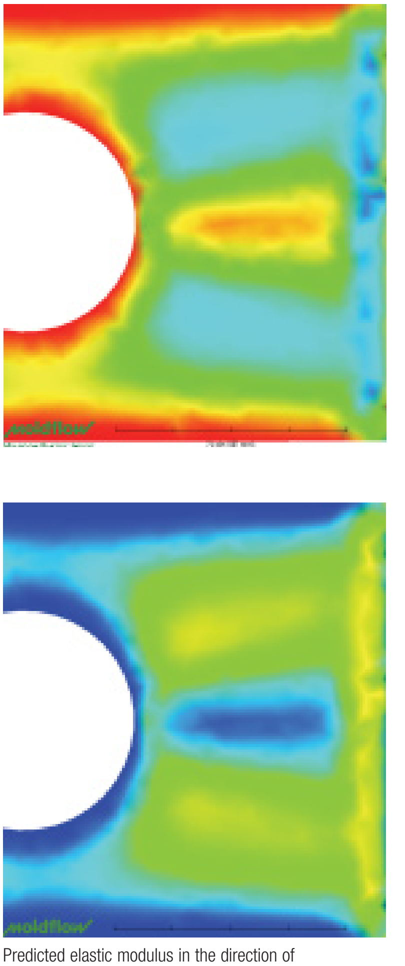
Predicted deformation using the data sheet modulus for this component, assuming uniform mechanical properties throughout the part (left), and using a distribution of mechanical properties as calculated by Moldflow injection molding simulation (right)

Perpendicular to this, however, the material has a much lower modulus of approximately 9 GPa, which is less than half of the modulus listed on the data sheet for this particular material (20 GPa). Because the injection molding process for fiber-reinforced parts can cause such great variation in strength throughout a part, the effects of the injection process should be considered in the design of such a part.

Design engineers typically solve the problem of using highly non-isotropic materials by applying generous safety factors to their designs, making them thicker than is actually required. As a result, money is wasted not only on raw materials, but also on cycle time. A simple rule of thumb for injection molding is that when the thickness is doubled, the production time quadruples. By coupling Moldflow's injection molding simulation tools together with finite element analysis (FEA) software, engineers can explore different design and manufacturing scenarios that will produce cheaper parts, while ensuring sufficient strength in highly loaded areas.

In order to enable product designers to incorporate the strength variations of fiber-reinforced, injection-molded components into mechanical analyses of those components, Moldflow has developed a new product called Moldflow Structural Alliance (MSA). MSA forms a seamless bridge between the Moldflow injection molding simulation tools (MPI and Moldflow Plastics Advisers®) and the mechanical simulation tools, including ANSYS Mechanical software from ANSYS. MSA achieves this by mapping the mechanical strength data calculated on a Moldflow mesh to the same part geometry with a 3-D mesh created by an FEA tool from ANSYS. This mapping technology enables a mesh type-independent link between the Moldflow and FEA domains, allowing each analysis to use the best technology suited for the task at hand.

Accounting for both the grade-specific material properties and the effects of the manufacturing process on the strength distribution in the molded part can make a significant difference in the interpretation of the results of structural analysis. Gaining a better understanding of the impact of the manufacturing process on molded plastic parts through the use of simulation software is crucial to the accurate design of parts for optimal structural performance and more efficient production. ■



Predicted elastic modulus in the direction of material flow (top) and perpendicular to the direction of material flow (bottom)

Analyzing **Bolt Pretension** in the ANSYS Workbench Platform

Convenient features enable pretension to be quickly and easily included in analysis of bolted joints.

By Doug Oatis, Senior Mechanical Engineer, Mechanical Simulation, Phoenix Analysis & Design Technologies, Inc., Arizona, U.S.A.

Analyzing a bolted flange used to be a serious undertaking, in part because of difficulties in including pretension loads produced by the installation torque of tightening the bolt. Analysts resorted to a variety of methods to account for pretension, including running a "dummy" thermal analysis to induce thermal expansion loads or creating beams and constraint equations on the flange to add equivalent compressive flange loads.

Pretension elements available in the ANSYS Workbench platform allow the analyst to more readily specify known axial loads or adjustments to groups of elements in accounting for these bolt installation loads. Indeed, bolt pretension is a great example of the user-friendly nature of simulation within the ANSYS Workbench environment.

One point of confusion, however, is in trying to scope a bolt pretension between the two "clamshell" faces of split cylindrical features imported from an external CAD package — for example Pro/ENGINEER® or SolidWorks®. A clamshell face is simply the result of a single cylindrical face being split in half. If both clamshell faces are selected for a single 'Bolt Load' object, a '?' will appear next to the 'Bolt Load.'

To get around this behavior, a little understanding of how simulation within the ANSYS Workbench environment applies bolt pretension is needed. First, the software divides a meshed body using the PSMESH command. Next, a pretension element effectively writes a constraint equation that relates the displacement of one cut boundary to the other.

Within the simulation environment, the face selected is used as a cutting guide for the PSMESH command. This means that you only need to select one clamshell face to define bolt load on an imported cylinder. The simulation tool will then use the middle of the selected face for the mesh division and pretension element creation. Bolt preload direction is determined automatically, as shown in Figure 1.

If you need more control over the preload application location, the bolt load can be scoped to a single body, as in Figure 2. You are then required to specify a coordinate system that defines the mesh slice plane and bolt load direction. This can be done through the default global or user-defined

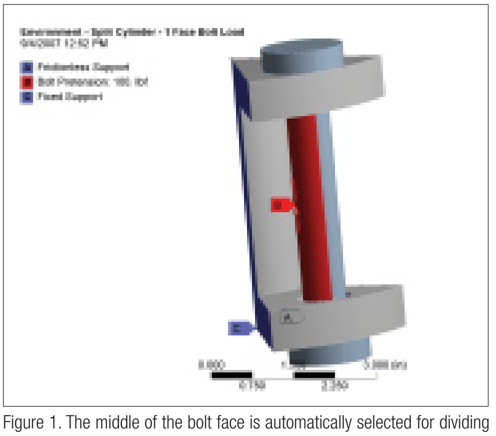


Figure 1. The middle of the bolt face is automatically selected for dividing the mesh and creating pretension elements.

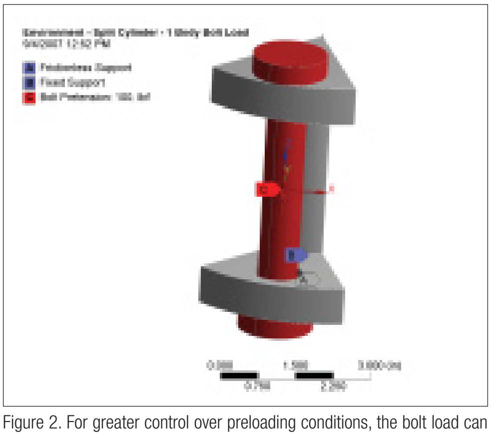


Figure 2. For greater control over preloading conditions, the bolt load can be scoped to a single body.

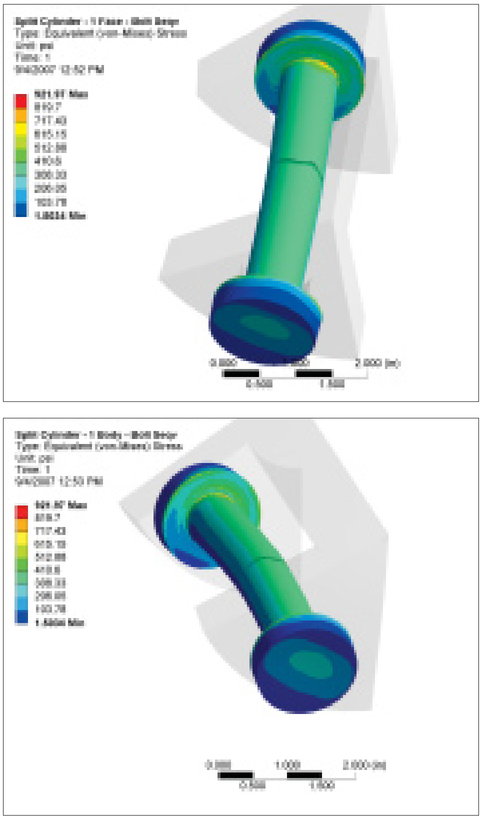


Figure 3. Stress contours of the one-clamshell-face (top) and the body selection methods (bottom) are identical.

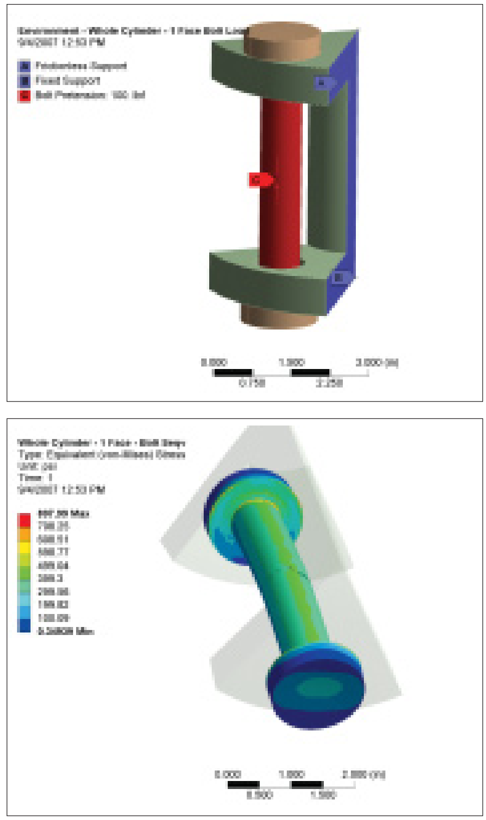


Figure 4. Geometry repaired and treated as a whole cylinder in the ANSYS DesignModeler tool (top) yields similar stress contours to the split cylinder case (bottom).

coordinate system. The XY-plane of the coordinate system defines the PSMESH slice plane, while the Z-axis defines the direction of bolt preload. You don't need to define the coordinate system "inside" the preloaded part.

When you compare the results of the one-clamshell-face with the body selection methods, they are the same, as shown in Figure 3. This is to be expected because the mesh division occurred at the same location for each environment.

Scoping the bolt load to a body and coordinate system not only lets you validate the one-clamshell-face scenario, but also allows the analysis of bolted flanges where the bolt is long relative to the flange thickness (e.g., the bolt midpoint is on the other side of the nut.)

If you create a cylinder using ANSYS DesignModeler software (the geometry modeling tool within ANSYS Workbench), it is defined as a single cylindrical surface. ANSYS DesignModeler capabilities can also be used to clean/repair imported geometry. If you use the 'Face Delete' tool (Create > Face Delete) on one of the two clamshell faces, the cylinder will be "repaired" into a single surface. Although not a required step, this allows you to clean up and simplify the geometry, as in Figure 4. This simplification could be done within a simulation by specifying a virtual cell from the two clamshells, though virtual cells do not support bolt loads.

The minor differences shown between the whole cylinder and split cylinder models occur at the contact interface, which was simplified as bonded. The reaction pretension adjustments were within .05 percent, as shown in Table 1.

When modeling bolted interfaces, the ease of using non-threaded solids to represent the bolt is increasingly attractive. Through a combination of automatic contact detection, multiple meshing controls and an easy-to-use bolt-loading interface, simulation using the ANSYS Workbench platform has made including bolt pretension intuitive and speedy. ■

Table 1. Pretension Differences at the Contact Interface for Variations in Clamshell Models

	Split Model		Whole Cylinder
	1-Face	1-Body	1-Face
Bolt Adjustment [in]	0.00019686	0.00019686	0.00019677

This article is based on a column from the technical newsletter *The Focus* (www.padtinc.com/epubs/focus) from engineering consulting firm PADT (Phoenix Analysis & Design Technologies).

Image © iStockphoto/switas

Multiphysics Simulation Using Directly Coupled-Field Element Technology

The 22x family of elements allows users to solve coupled-physics problems in one solution pass with a single model.

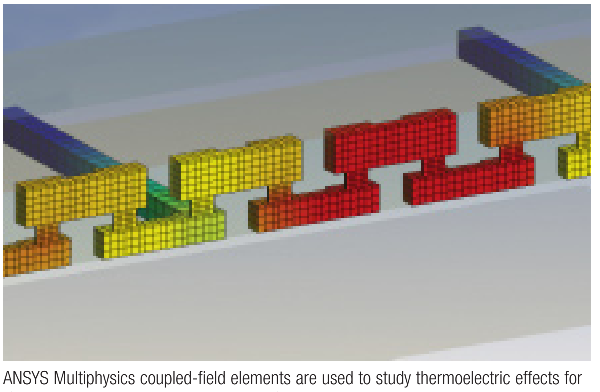
By Stephen Scampoli, Multiphysics Product Manager, ANSYS, Inc.

For an increasing number of applications, simulation of individual physics is not adequate for evaluating designs under real-world conditions, where engineers must account for the effects of multiple coupled-physics. ANSYS, Inc. provides two proven solution techniques to solve coupled-physics problems. The first is the ANSYS Multi-field solver (available with ANSYS Multiphysics software or with multiple physics product combinations), which performs sequential iterations between the physics disciplines until the solution converges. The second technique is to use directly coupled-field elements (available in ANSYS Multiphysics software) that solve for multiple physics in a single solution pass.

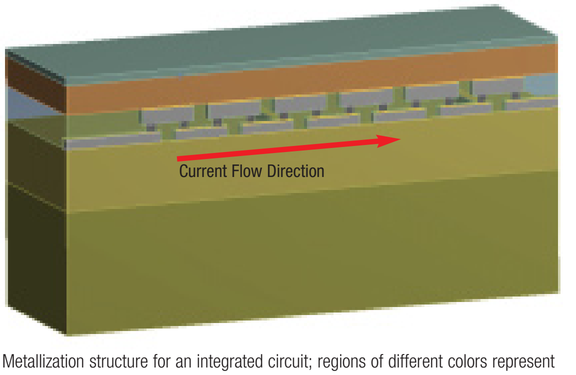
Coupled-field elements in ANSYS Multiphysics version 11.0 software handle a variety of analyses, including thermal–structural coupling, piezoelectricity, piezoresistivity, the piezocaloric effect, the Coriolis effect, electroelasticity, thermal–electric coupling and thermal–electric–structural coupling. The technology is useful in the design of a range of products such as electronic components, microelectromechanical systems, sensors, transducers, piezoelectric gyroscopes, accelerometers and thermoelectric coolers.

The ANSYS 22x family of directly coupled-field elements (PLANE223, SOLID226 and SOLID227) allow users to solve 2-D and 3-D coupled-physics problems by employing a single finite element model with the appropriate coupled-physics options set within the element itself. The coupled-field elements support up to five degrees of freedom per node, including displacement, temperature and voltage. Users can turn on or turn off these degrees of freedom depending on their application. This ability to select the appropriate degrees of freedom allows the flexibility to solve many different types of coupled-field problems.

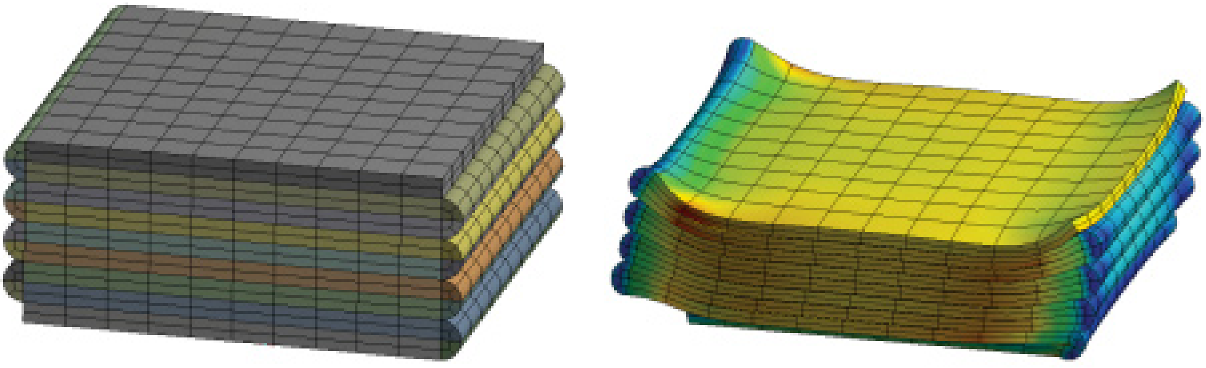
The coupled-field elements calculate the appropriate element matrices (matrix coupling) or element load vectors (load vector coupling) to account for the interaction between the different physics disciplines. For linear problems, coupled-field interaction is calculated in one solution iteration when using matrix coupling. Nonlinear



ANSYS Multiphysics coupled-field elements are used to study thermoelectric effects for IC circuits, as seen in this section of an IC's metallization structure.



Metallization structure for an integrated circuit; regions of different colors represent different materials that are present.



Mesh for a folded dielectric elastomer actuator (left) and contours of deformation (right), which results from electrostatic actuation

problems require an iterative solution employing the Newton-Raphson algorithm for both the matrix and load vector coupling options.

The ANSYS 22x family of elements offers several advantages. First and foremost, it allows for multiple physics solutions otherwise not possible with single physics finite elements. Coupled-field elements simplify the modeling of coupled-field problems by permitting one element type to be used in a single analysis model. This allows users to create a single mesh, apply all boundary conditions to one model and post-process just one set of results. Nonlinear solution convergence is robust, and coupled-physics solutions typically converge automatically without requiring manual intervention. Also, although 22x elements are not natively supported by the ANSYS Workbench platform, they are easily incorporated into this environment through the use of command objects. This enables users to perform many coupled-field solutions directly in the ANSYS Workbench environment and take advantage of functionality such as parametric models, sensitivity studies, design optimization and design of experiments for multiphysics solutions.

In working with coupled-field elements, users should note that, with up to five degrees of freedom per node, models tend to be larger for multiphysics solutions and thus lead to higher storage and memory requirements. Also, because coupling terms are often not symmetric, an unsymmetric equation solver must be used for the coupled-field solution.

Many companies use coupled-field elements to address their multiphysics design challenges. In the development of integrated circuits (ICs), for example, the technology is used in evaluating electromigration: the transport of material caused by the movement of ions in a conductor.

During an overload condition, high current densities and Joule heating can lead to electromigration in the metallization structure of an IC. The metallization structure forms the interconnections for an IC. Thus, electromigration in this component can decrease the reliability of an IC and possibly lead to failure of the circuit. For studying this type of critical problem, a coupled-field element can be used to couple the thermal and electric effects in one analysis model.

By incorporating the temperature dependence of the electrical and thermal properties into the nonlinear simulation, this type of study using coupled-field elements enables engineers to accurately predict temperatures during an overload condition, and thus determine the level of risk to the integrity of the IC.

In another example, coupled-field elements can be used in the evaluation of electroactive polymers, which exhibit a deformation when subjected to an electrostatic field. Often referred to as "artificial muscles," electroactive polymers are gaining increased application as electrostatic actuators and sensors, as well as in the emerging field of micro-robotics in which linear motion is required. One class of electroactive polymers is dielectric elastomers, which can be used as high-voltage insulators or electrostatic actuators.

Electrostatic actuators can be configured as a multi-layer stack, a helical or a folded sheet. In all configurations, the coupled-field elements can be used to solve for the deformation generated by the electrostatic field — a new feature of the coupled-field elements in ANSYS Multiphysics 11.0 software. This capability to perform coupled-field electroelastic analyses provides researchers with valuable data needed in the development of this growing class of products. ■

Flexible **Multibody Dynamics**

Models of flexible mechanisms incorporate large deformations and rotations, and nonlinear material properties to provide nonlinear dynamic response.

By Pierre Thieffry, Vikas Chawla and Srivatsa Sharma, ANSYS, Inc.

Multibody dynamics simulates motions and forces of parts interconnected to one another via sets of constraints modeled as joints. Multibody models are used in this manner to represent a wide variety of assemblies, such as suspension systems and landing gears, as well as complete systems including vehicles or aircraft.

While simulating the motion of an assembly constructed of purely rigid bodies is mature technology that is available from a number of software suppliers, such simulations cannot accurately account for the inherent flexibility of these components. Modeling a part's flexibility can be moderately important, or it can be critically important, such as when simulating a long, slender linkage that may buckle or bind, or when modeling parts that are flexible because of the material from which they are constructed, such as those made of elastomers.

The most widely used method to combine the time-saving characteristics inherent in a rigid body simulation with the flexible responses possible via the finite element method is the Craig–Bampton method. In this method, a modal FEA simulation is used to capture the linear dynamic response of a flexible assembly in the form of mode shapes and frequencies, or eigenvalues and eigenvectors. These flexibility matrices are used in conjunction with the reduced-order rigid representation to simulate a flexible assembly without running a completely flexible model with its inherently large number of degrees of freedom (DOF).

Though it is widely practiced, the Craig–Bampton is limited in the following ways:

- Method is time-consuming in practice (modal FEA followed by rigid multibody simulation)
- Method is limited to linear responses, so cannot be used to model:
  - Nonlinear material responses: hyperelasticity, plasticity, viscoelasticity
  - Nonlinear contact: rigid to flexible, flexible to flexible

ANSYS, Inc. offers an accurate and convenient approach for modeling such flexible multibody dynamics systems. While the method may require additional computational resources compared to standard analyses, it has the following significant advantages:

- The finite element mesh automatically represents the complete geometry of the system.
- Large deformation/rotation effects (including inertia) are built into the finite element formulation.

- Interconnection of parts via joints is greatly simplified by considering the finite motions at the two nodes forming the joint element.
- The resulting model supports static, modal, harmonic, spectrum, buckling and transient dynamic analysis types.

Modeling Flexible, Rigid Bodies and Joints

The multibody dynamics solution from ANSYS offers extensive versatility in handling the degree of complexity required. A user can create a quick initial approximation of the flexible and rigid parts of a mechanism using standard beam elements and rigid beams/links. Alternatively, detailed modeling of the flexible part can be performed using 3-D solid elements, and of the rigid part using contact capabilities.

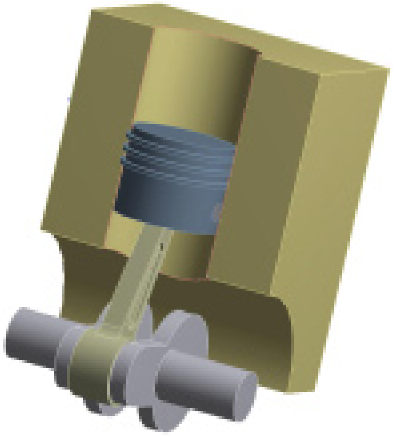
A library of beam, shell, solid-shell and solid elements is available for creating multibody dynamics models in combination with contact capabilities in order to model rigid parts and other contact conditions. Joint elements offer the required kinematic connectivity between moving parts. Rigid bodies can be connected via joint elements and connected to flexible bodies to model-mixed, rigid–flexible body dynamics. Relatively stiff parts can be represented by rigid bodies when stress distributions and wave propagation in such parts are not critical.

Various joint capabilities are provided to connect flexible and/or rigid components to each other. A joint element is defined by two nodes with six degrees of freedom (DOF) at each node. Relative motion between the two nodes is characterized by six relative DOF. Depending on the application, different kinds of joint elements can be configured by imposing appropriate kinematic constraints on any, or some, of these six relative DOF. The kinematic constraints in the joint elements are imposed using the Lagrange multiplier method. Joint types available for defining connections in multibody simulation from ANSYS include spherical, general, revolute, universal, slot, translational, cylindrical, planar, weld and orient.

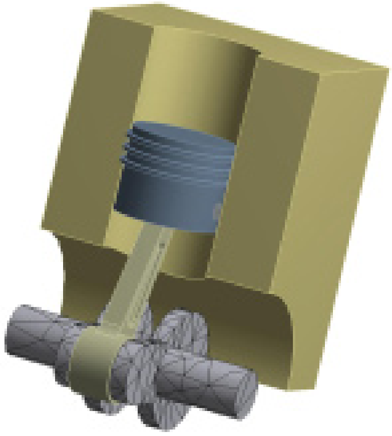
Through the use of the ANSYS Workbench environment, the setup of complex assemblies for a mixed flexible/rigid dynamics analysis has never been easier. The user only has to tag the bodies to be considered as rigid, since the default behavior is flexible. Contact conditions and joints are automatically detected for the assemblies, thus significantly reducing the time required to set up the full model.

Component Mode Synthesis

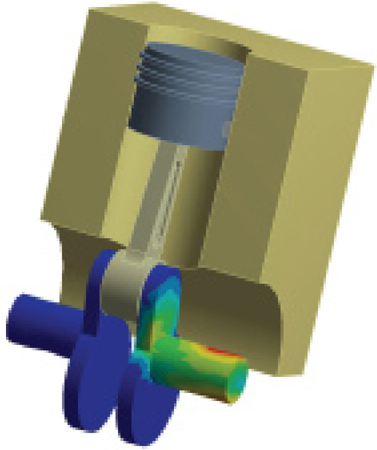
Determining flexible-body response to a dynamic motion event typically involves solving hundreds or thousands of time points — a task that can be tremendously time-consuming if many DOF are involved. To minimize run time, component mode synthesis (CMS) super-elements,



Joint specification and contact definition for engine assembly dynamic analysis



Multibody definition for an engine assembly with rigid and flexible bodies



Dynamic stress results on flexible components offer solutions for accurate fatigue predictions.

or substructures, can be used to replace thousands of flexible-body DOF with tens of DOF representing the dynamic response. The flexible body to be substructured is assumed to behave in a linear elastic manner and may undergo large rotations, but the strains and relative rotations within the body are presumed to be small.

Initial Settings and Damping for Transient Dynamic Simulation

Initial conditions define the state of the system at the start of the analysis. In structural finite element analyses, initial conditions are defined in terms of initial displacements, velocities and accelerations at all independent degrees of freedom. Because time-integration schemes (such as the Newmark method and the HHT method) rely on the history of displacements, velocities and accelerations, it is important to define consistent initial conditions. By default, a zero value is assumed for initial displacements, velocities and accelerations at DOF that are not otherwise specified.

Inconsistencies in initial conditions introduce errors into the time-integration scheme and lead to excitation of undesired modes. Accumulation of these errors over several time increments adversely affects the solution and very often causes the time-integration scheme to fail. Applying numerical damping or other forms of damping can reduce these effects.

Two types of damping can be specified in ANSYS multibody models: numerical and structural. Numerical damping is associated with the time-stepping schemes used for integrating second-order systems of equations over time. Numerical damping stabilizes the numerical integration scheme by damping out the unwanted high frequency modes. Larger numerical damping values are usually necessary for problems involving rigid body rotational motion, elastic collisions (dynamic contact/impact) and large deformations with frequent changes in substep size. Structural damping refers to physical damping present in the system. A user can specify the damping at the material level via viscous material models or dashpots.

Case in Point: Engine Assembly Simulation

Flexible multibody dynamic analysis of an engine assembly is important for accurate fatigue predictions in critical components. The flexible multibody solution from ANSYS provides tools for combining definitions for rigid and flexible parts in the same model. Rigid bodies are represented by using point mass approximation, with the properties for the parts accurately retrieved from the underlying CAD geometry. Flexible bodies are represented by traditional finite element mesh to accurately portray the geometry and calculate dynamic displacement and stress results for accurate fatigue calculation. Rigid and flexible components are connected using various joints that accurately represent the motion between the parts.

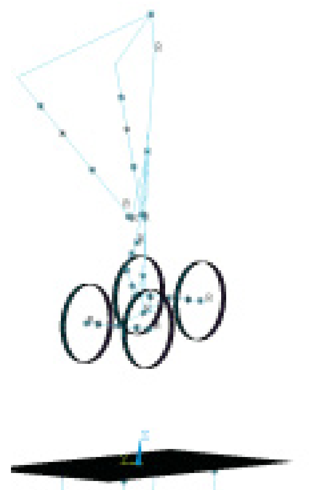
Case in Point: Drop Test Analysis of Aircraft Landing Gear

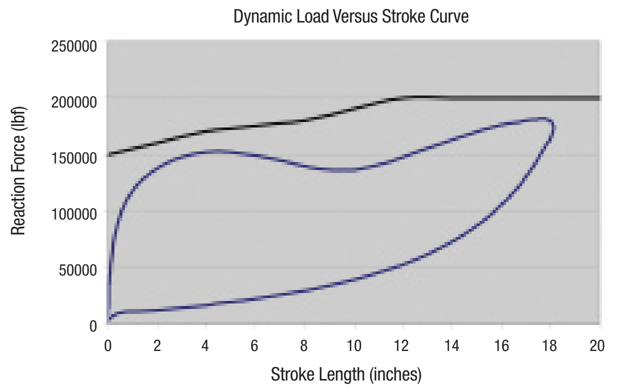
A drop test analysis of an aircraft landing gear involves simulating a landing in which the kinetic energy of a descending aircraft is absorbed by the landing gear shock strut. The simulation is used to develop damping characteristics of a drop test simulation. The damping characteristics need to be adjusted to ensure that the dynamic load/stroke curve always stays within the dynamic load envelope.

In the ANSYS multibody dynamics solution, a landing gear simulation is modeled by considering the landing gear components to be rigid and connected by various joint specifications between parts. The aircraft is assumed to be descending at a speed of 10 feet per second with the wheels rotating at the approach velocity of 150 miles per hour. Friction is modeled between the tires and the runway using contact definition. The dynamic behavior of the tires represented through stiffness and damping is modeled by springs and dampers below the runway. ■

The authors wish to acknowledge the development and technical support team at ANSYS, Inc. for their efforts and contribution to this article.


Joint specification and contact definition for landing gear drop test


Finite element representation of the landing gear assembly for transient multibody simulation



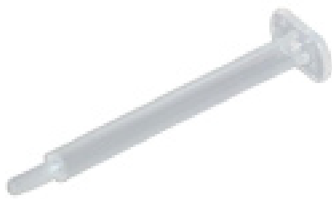
Dynamic load versus stroke curve (blue) and dynamic load envelope curve (black)

Validating your complex part design in Moldflow Plastics Insight
47 minutes

Mold trials
1

Cost of rejects
$0

Getting it right the first time
Perfect



NEW!

Introducing MSA—the Moldflow Structural Alliance program that allows ANSYS users to interface more easily with Moldflow software. Find out more at www.moldflow.com/msa.

If your company makes injection-molded parts, you need Moldflow software on your desktop.

Running a Moldflow analysis doesn't take much time, and it can shave days off every project, saving your company a lot of money. More than 7,500 of the world's top companies use Moldflow technology to speed products to market, reduce cycle time, and make higher quality parts.

Moldflow is inexpensive, easy to learn and easy to use. It also works with the CAD system you're using. Now would be a perfectly good time to call us.

Call 800.232.9363 Email **info@moldflow.com** Web **www.moldflow.com/ANSYS**

Moldflow Plastics Insight®
Moldflow Plastics Advisers®

moldflow®
plastics made perfect

HP helps you not only achieve your goals, but exceed them. HP CAE solutions deliver optimal performance, unprecedented reliability, and collaboration with experienced partners.

Innovation: HP innovations include the new HP BladeSystem, a groundbreaking product that reduces energy consumption by dynamically adjusting power and cooling.

Choice: Only HP offers a full range of industry-standard processors, operating environments, middleware, interconnects, and integration services—ensuring the optimal solution for your CAE applications.

Performance: The collaborative partnership between HP and ANSYS, Inc. produces highly scalable and reliable solutions for exceptional ANSYS®, FLUENT®, and ANSYS® CFX® results—on time and on budget.

HP Information: www.hp.com/go/CAE
Whitepapers/Presentations: www.hp.com/go/optimize-CAE
Contact HP: www.hp.com/country/us/en/wwcontact
Partner Information: www.ansys.com

ANSYS®

© 2007 Hewlett-Packard Development Company, L.P. The information contained herein is subject to change without notice. The only warranties for HP products and services are set forth in the express warranty statements accompanying such products and services.