MANUFACTURING COMPANIES RELY ON MARC

The World is Nonlinear

SIMULATING MANUFACTURING PROCESSES
Solving Problems & Improving Processes

THE HADLEY GROUP
Simulation Helps Increase Sales by $4M
Nonlinear FEA Validates New Cold Roll Forming Process

COMTES/PILSEN STEEL
Getting to the Root Cause of Cracks
Heat Transfer Analysis Helps Solve Tough Forging Problem

INNOVATING MOMENTUM IN GROUND VEHICLES
New Tire Development & Optimized Engines

BIG TYRE
Designing the Right Tire
Nonlinear FEA Accelerates Development of Non-Pneumatic, Non-Solid Tire for Mining Industry
EDITORIAL PREFACE

The World is Nonlinear
Srinivas Reddy, MSC Software

CUSTOMER SPOTLIGHT

Simulation Helps Increase Sales by $4M
Nonlinear FEA Validates New Cold Roll Forming Process
Hadley Group

Getting to the Root Cause of Cracks
Heat Transfer Analysis Helps Solve Tough Forging Problem
Comtes/Pilsen Steel

Designing the Right Tire
Nonlinear FEA Accelerates Development of Non-Pneumatic, Non-Solid Tire for Mining Industry
Big Tyre

Optimizing Engine Performance
Simulation Saves Millions per year by Getting Design Right the First Time
Litens Automotive Group

SPECIAL SPOTLIGHT

What Engineers Can Do with Marc Nonlinear FEA
Srinivas Reddy, MSC Software
As engineers can attest, designing better products requires understanding of materials, loads and boundary conditions, environmental conditions, and component interaction in assemblies. Rise in the use of engineered materials like composites, elastomers and plastics, and special alloys, and more stringent market based or regulation based requirements necessitate deeper knowledge of product behavior well before being introduced into the market. This means that the inherent nonlinear response of the structural systems and the physical processes cannot be ignored during development phases.

Companies seeking to gain further insights into product behavior are going beyond virtual product testing by adopting simulation to understand and improve manufacturing processes. Since manufacturing processes are designed to introduce permanent deformation or change material characteristics, accurate simulations require robust handling of all the types of nonlinearities, namely: Geometric, Material, Boundary condition/contact, and Multiphysics.

The nonlinear simulation solution of Marc from MSC Software has been providing the advantage of accurate nonlinear modeling for four decades through its broad set of capabilities highlighted at the end of this document. With its powerful technology, Marc has also been addressing the manufacturing problems in various processes, like forming, welding, cutting, extrusion, curing, and more.

This supplemental issue highlights engineering ingenuity of some of our customers and the way they address their manufacturing challenges using nonlinear simulation. In the following customer stories, you see how the power of innovation effectively comes together with the right tools to solve complex design and manufacturing problems.

- **Hadley Industries**, whose core business is manufacturing cold roll formed products, used in building construction industry, developed and patented a unique cold forming process to significantly improve mechanical and structural properties of strip steel. By using Marc to simulate the manufacturing process, they expect to save on considerable tooling costs (which could range from $30,000 to $150,000 for each design), and estimate to increase sales volume estimated to $4 million over the next three years offering a superior product in the market.

- **Big Tyre**, a manufacturer of solid wheels primarily used in underground mining, is focused on improving safety by avoiding punctures and related damage and cost and improving performance. While the mining industry is moving towards solid tires, it comes with its challenges in the form of ground rutting, leading to further ride problems. Big Tyre has developed a non-pneumatic, non-solid segmented tire to overcome these problems by using composite leaf springs that are four times as flexible as steel. Since creating physical prototypes and testing multiple designs is a very expensive process, Big Tyre has been using Marc to test and perfect their design.

- **Litens Automotive Group** is a full service design and manufacturer of engineered power transmission systems and components. Its patented TorqFiltr crankshaft vibration control system incorporates a series of components that transmit power through frictional contacts rather than fixed connections. The product must be frequently customized to deliver optimal performance for a specific automotive engine. To avoid the high costs and time-consuming trial and error process, Litens has used Marc to accurately predict the design behavior, saving millions of dollars per year across their product line.

- **Pilsen Steel**, a leading producer of castings, ingots and forgings, located in Pilsen, Czech Republic, experienced difficulties with ingots cracking in a forging operation. COMTES FHT, contracted to investigate the problem, used Marc to analyze the process of heating the ingots in the furnace, which helped them to identify the root cause of failure. COMTES team also identified a solution to overcome this problem using Marc, which was then verified during the actual manufacturing process. This has helped reduce reject rate of the product and increased confidence in the process parameters.

I have truly enjoyed learning about some unique ways our customers have been using our products, and I’m sure you will too.

*If you are new to nonlinear analysis, a great introduction to it can be found in the white paper “When f≠Ku, An Introductory Guide to Nonlinear Analysis,” published by MSC Software, which is available at the following link: [www.mscsoftware.com/f_not_equal_ku](http://www.mscsoftware.com/f_not_equal_ku)*
Simulation Helps Increase Sales by $4M

Nonlinear FEA Validates New Cold Roll Forming Process

Hadley Group | Based on an interview with Dr. Martin English, Design & Development Manager

Hadley Industries PLC’s core business is manufacturing cold roll formed products, primarily to the building and construction industries. The company developed and patented a unique cold forming process known as UltraSTEEL® which significantly improves mechanical and structural properties of the strip steel by imparting a dimpling pattern prior to the roll forming operation. The geometry is much too complex to determine the structural properties of the end product based on analytical calculations alone, and it could cost $30,000 to $150,000 for tooling to manufacture the part so its properties could be physically measured.

Hadley addressed this challenge by using Marc nonlinear finite element analysis software to predict the highly nonlinear changes in geometry and material properties that occur during the UltraSTEEL® process, cold roll forming and secondary processes. “Marc overcame problems seen with other finite element software packages such as nonconvergence and provided reliable and consistent results that matched experimental measurements,” said Dr. Martin English, Design and Development Manager for Hadley. “The ability to accurately simulate the process and quickly determine its performance in customer applications has been responsible for a substantial increase in sales volume estimated to total over $4 million over the next three years.

Challenges

Complex Geometry and Process Creates Simulation Challenge

When evaluating any new cold roll forming application, the potential customer needs to accurately investigate the performance of the finished product by estimating section properties such as stiffness and load-carrying capacity in order to make a buying decision. Standard cold roll formed products have a uniform cross sectional geometry, so theoretical calculations can be relatively easily performed to determine their section properties. The geometry formed by the UltraSTEEL® process is much more complicated and the material properties vary over the geometry, so theoretical calculations cannot be used to accurately analyze its performance. “In the past, the only way to fully understand the behavior of a section produced by the UltraSTEEL® process was to invest in a complete set of tooling, produce prototypes and perform physical testing,” English said. “The high cost of this approach was a major obstacle to trying out the new process.”

Hadley has worked for a number of years in developing the capability to simulate the dimpling process as well as the subsequent cold rolling forming and secondary operations with the goal of developing the capability to predict the performance of the finished product. The company tried one popular finite element analysis software package only to discover that the results did not correlate well with physical testing. So Hadley assessed three leading developers of nonlinear finite element analysis software by using them to simulate a complex nonlinear problem involving compression of a thin-walled column. “Marc demonstrated its capabilities to solve applications involving highly nonlinear changes in geometry by providing by far the most accurate results on this difficult problem,” said Bac Nguyen, Research and Development Engineer for Hadley.
The ideal approach is to simulate the entire geometry and material data of the dimpled flat steel strip into a dimpled strip, 2) the cold roll forming process that produces the desired product, other processes and applications.

The geometry and material data of the dimpled strip are transferred from one process to the next in a closed loop. This approach is practical for small sections of dimpled products and optimizing the dimpling process itself. However, the models of the rolls and dimpled strip can contain tens of millions of elements for larger models so transferring the stress/strain data between each of the stages becomes very complicated and time-consuming, resulting in high computational costs.

Hadley Group addresses this challenge by using a simplified method to compute the properties of large dimpled products. This approach begins with simulating the deformation of a flat steel plate by producing a single dimple. This general dimple geometry is used to generate a dimpled strip. 3D elements are used for smaller products and shell elements are used for larger products. It’s important to note that only the geometry of the dimple is transferred from the dimpling process. The material properties of the dimple are provided from a separate tensile test on a dimpled steel sample. This test can be quickly and inexpensively performed without tooling. The next step is simulating the cold roll forming process that develops the dimpled strip into the desired dimpled product, other processes and applications.

The reliable and consistent results provided by Marc make it possible to accurately assess the applicability of UltraSTEEL® for existing and new products in a short time frame at a low cost, English concluded. “The accurate simulations have enabled Hadley to make and substantiate technical claims regarding the benefits of the process. As a result, the company has increased its sales of UltraSTEEL® products and also generated additional revenue by increasing licensing of the process amounting to an estimated $4 million over the next three years.”

The solution Transferring Geometry and Material Data from One Process to Next Complex and interrelated nonlinear changes in contact, geometry and material properties occur during the UltraSTEEL® process and subsequent section forming and secondary operations. “The simulation challenge involves both accurately simulating these processes as well as applications under loading,” Nguyen said. “This requires simulations that connect to previous or subsequent simulations to perform continuous processes while taking into account the changes in geometry, material and structural properties of the materials. An important advantage of Marc in this regard is its PRE STATE procedure which can be used to transfer the geometry of the dimpled strip together with its material data including stress/strain data generated from the dimpling process into the subsequent simulations such as the cold roll forming process.”

The ideal approach is to simulate the entire chain of processes as a sequence from start to finish: 1) the dimpling process that deforms a flat steel strip into a dimpled strip, 2) the cold roll forming process that produces the desired section, and 3) additional processes such as shear cutting and applications such as products under tension, bending, compression loads, etc. The geometry and material data of the dimpled strip are transferred from one process to the next in a closed loop. This approach is practical for small sections of dimpled products and optimizing the dimpling process itself. However, the models of the rolls and dimpled strip can contain tens of millions of elements for larger models so transferring the stress/strain data between each of the stages becomes very complicated and time-consuming, resulting in high computational costs.

Hadley Group addresses this challenge by using a simplified method to compute the properties of large dimpled products. This approach begins with simulating the deformation of a flat steel plate by producing a single dimple. This general dimple geometry is used to generate a dimpled strip. 3D elements are used for smaller products and shell elements are used for larger products. It’s important to note that only the geometry of the dimple is transferred from the dimpling process. The material properties of the dimple are provided from a separate tensile test on a dimpled steel sample. This test can be quickly and inexpensively performed without tooling. The next step is simulating the cold roll forming process that develops the dimpled strip into the desired dimpled product, other processes and applications.

Results/Benefits Simulation Results Match Experimental Data “Both simulation approaches have been extensively validated by experimental results and have demonstrated the ability to accurately represent the dimpling, cold forming and secondary processes,” Nguyen said. In a typical example, a computer aided design (CAD) model of the top and bottom rolls was imported into Marc for preprocessing. An elastic-plastic material model was used with a Young’s modulus of 205 GPa and a Poisson’s ratio of 0.30. The top and bottom rolls were modeled as rigid bodies that rotated around their central axes. The plain strip was generated in Marc and placed in a pre-defined position between the two rolls. Five layers of elements were used through the strip thickness to model both bending and stretching phenomena. Two different meshes were evaluated, one with 37,980 solid elements and the other with 149,810 solid elements. The dimpling simulation showed that the maximum plastic strain and stress developed in the two meshes differed by less than 7% and 4% respectively so the coarser mesh was used for the balance of the study.

The top and bottom rolls had an overlapping gap of 0.40 mm between the mating teeth. The sheet was fully fixed at one end and initially fed to the rotating rolls with a velocity equal to the linear velocity at the tip of the roll teeth. When the roll teeth just grasped the strip, the fixed end was released and the strip was deformed by the rotating rolls. The original plain sheet and the dimpled sheet were used in tension and bending simulations. The engineering stress and strain data of the plain steel sheet were obtained from tensile tests. The dimpled sheet was merged into the new model in order to start the new analysis and the PRE STATE option was employed to directly transfer result data from the previous dimpling process into the tensile and bending simulations. “The simulation was validated by comparing the 3 dimensional geometry of the predicted shape with scanned data of the physical sample,” Nguyen said. “The geometry of the predicted dimpled sheet differed from the actual process by less than 1.7%. The distances between adjacent dimples on the simulated sheet also matched the actual dimpled sheet. The predicted plastic strain distribution correlated well with experimental results obtained with micro-hardness tests. In the tensile test the predicted yield and ultimate forces in the plain sheet were 2% and 4% greater than the experimental results and the predicted yield and ultimate force in the dimpled sheet were about 1% and 5% less than the experimental results. The predicted and experimental values for ultimate load in the bending test were also close with a maximum difference of 0.4% and 4% for the plain and the dimpled sheets respectively.”

“The reliable and consistent results provided by Marc make it possible to accurately assess the applicability of UltraSTEEL® for existing and new products in a short time frame at a low cost,” English concluded. “The accurate simulations have enabled Hadley to make and substantiate technical claims regarding the benefits of the process. As a result, the company has increased its sales of UltraSTEEL® products and also generated additional revenue by increasing licensing of the process amounting to an estimated $4 million over the next three years.”

![Image](https://via.placeholder.com/150)

![Image](https://via.placeholder.com/150)

![Image](https://via.placeholder.com/150)

![Image](https://via.placeholder.com/150)
Getting to the Root Cause of Cracks

Heat Transfer Analysis Helps Solve Tough Forging Problem

Comtes/Pilsen Steel | Based on an interview with Dr. Filip Tikal, Computer Modeling Specialist

A leading producer of castings, ingots and forgings, Pilsen Steel, Pilsen, Czech Republic, recently experienced difficulties with ingots cracking in a forging operation. The company contracted with COMTES FHT to investigate and determine the root cause of the formation of longitudinal cracks on a diverse load of 34CrNiMo6 steel ingots in a forging operation. Previously the ingots were cooled after casting in water to between 500°C and 600°C. The ingots were then placed in the forging furnace which is at a temperature of 1100°C to 1200°C. It was unclear whether the root cause of the cracks was heating the ingots or the forging operation.

COMTES used MSC Software’s Marc nonlinear finite element analysis (FEA) software to analyze the process of heating the ingots in the furnace, which was suspected as the root cause of the cracks.

The results showed that increasing the temperature of the ingots by 100°C prior to putting them into the furnace reduced thermal stresses to acceptable levels. Pilsen Steel implemented this change and it eliminated the cracking problem.

Challenge

COMTES FHT was tasked with investigating the root cause of the formation of longitudinal cracks on a diverse load of 34CrNiMo6 steel ingots in a forging operation.

In their current process, Pilsen Steel was cooling the formed ingots in water to between 500°C and 600°C. The ingots were then placed in the forging furnace which is at a temperature of 1100°C to 1200°C.

Once cooled, the ingots longitudinal cracks contained a series of undesirable longitudinal cracks. It was unclear whether the root cause of the cracks was heating the ingots or the forging operation.

“Marc is the only finite element analysis software I am aware of that is capable of handling all of the complexities of this analysis problem,” Tikal said. “Performing thermal analysis on the complete ingot workload requires determining the radiant heat transfer between the furnace and each of the ingots with shading effects taken into account. Marc excels at this type of challenging multiphysics problem which is why it is our finite element analysis tool of choice.” Marc is an implicit nonlinear FEA software program that simulates static, dynamic coupled physics problems.

Marc eliminates the need for the simplifying assumptions that are required with linear FEA, making it possible to accurately simulate complex real-world behavior under realistic environments and operating conditions.

Solution

Pilsen Steel provided the geometry of the ingot and furnace in the form of 2D drawings.
COMTES FHT researchers reproduced the ingot geometry in SolidWorks. Then they created a simple 2D axisymmetric model of a single ingot and the furnace wall. The material model for the simulations was obtained through experimental testing including thermal capacity, thermal conductivity, heat expansion coefficient and phase transformation in the COMTES FHT mechanical testing laboratory.

The researchers ran a transient analysis for a period of 5 minutes starting with when the ingot was placed in the furnace. Marc calculated the radiant heat transfer from the furnace to the ingot then converted the thermal gradients into mechanical stresses on the ingot. The results showed very high stresses in the ingot so the 2D axisymmetric model was expanded to a full 2D model also consisting of a single ingot.

The researchers then expanded the 2D model by adding additional ingots to determine the effects of the placement of the ingots in the furnace and of the shading of ingots from radiation heat transfer by other ingots. The actual layout and materials of the ingots in the work load were monitored in the plant for several months to ensure accurate modeling of the ingot placement.

Next, the 2D model was extruded to a 3D model consisting of the furnace and a typical load of ingots. The 3D model provided verification of the 2D analysis as well as a more detailed prediction of the location of the cracks, which was used for validation by comparing the analysis results with inspection data.

These results confirmed that heating the ingots in the furnace generated thermal stresses that later caused cracks to form during forging. Based on the analysis results, researchers were able to predict which ingots would develop cracks and the location of those cracks. In both the analysis and in production, cracks were primarily seen on ingots placed near the wall of the furnace.

It was clear that the problem was caused by thermal gradients during the heating operation. The temperature of the furnace could not be reduced so the researchers wondered whether raising the ingot temperature would eliminate the cracking problem and, if so, what was the lowest ingot temperature that would eliminate cracks? To answer these questions, COMTES FHT researchers ran a series of analyses while varying the starting temperature of the ingots in 50°C increments.

A high performance computing (HPC) cluster with 32 cores was used to provide a fast turnaround on this more complex analysis.

The simulation showed that the cracks originated in areas where high thermal stresses were generated as the ingots were heated. COMTES researchers next ran a number of additional analysis runs that evaluated the impact of adjusting the temperature of the ingots prior to inserting them into the furnace.

### Results/Benefits

The results showed that raising the ingot temperature to 700°C completely eliminated the cracking problem. COMTES FHT researchers worked with Pilsen Steel engineers to modify the soaking process used to cool the ingots after casting to provide assurance that ingots would be at least this temperature when they were placed in the furnace.

As predicted by the analysis, when these changes were implemented they eliminated the cracking problem.

### About Pilsen Steel

Pilsen Steel produces steel, ductile and grey-iron castings, ingots and finished machined forgings for a wide range of industries such as power generation and shipbuilding as well as for further processing by rolling mills. The company performs the complete production process under one roof including steel making, casting, forging and round and finish machining. Two thirds of the company’s production is exported. Pilsen Steel delivered the wheel shaft and other castings totaling more than 200 metric tonnes for the London Eye Ferris wheel. Pilsen Steel is also the world’s leading producer of wind turbine shafts and one of the largest suppliers of large crankshafts for 4-stroke diesel engines.

### About COMTES FHT

COMTES FHT offers a wide range of services to metal producing and metalworking companies including physical testing, material analysis, computer simulation, process design and development, and prototype manufacturing. COMTES FHT’s customers include steelmakers, rolling mills, forge shops and nonferrous metal producers. The company’s capabilities include fracture-toughness testing, high- and low-cycle fatigue testing, high temperature testing, strain measurement, material analysis with electron microscopes, electron backscatter diffraction (EBSD) analysis and energy dispersive X-ray analysis (EDX) analysis of chemical composition. COMTES FHT cooperates with industrial partners, primarily in Europe, and participates in research and development projects with research institutes and universities all over the world. COMTES FHT employs more than 65 researchers, technicians and other employees.
Designing the Right Tire

Nonlinear FEA Accelerates Development of Non-Pneumatic, Non-Solid Tire for Mining Industry

Big Tyre | Based on an interview with Bruce Louden

Designing the right tire for large wheeled vehicles used to haul coal and other materials in underground mines presents an enormous design challenge. Pneumatic tires present risks because of the danger of an explosion in a confined space while solid tires are associated with relatively large vibrations experienced by the driver. Big Tyre, a company that specializes in producing solid tires for mining vehicles, is developing a unique alternative which uses arrays of leaf-springs, typically made of composite materials, to provide performance similar to pneumatic tires without the risk of blow-outs. The company began designing the new tires in 2002. They first designed and built drive tires for a racing go-kart - these prototypes, built in 2006, proved the concept very well. Next they designed full-size prototypes for mining vehicles using finite element analysis but found that the software was incapable of modeling the rubber once it was bulging under load, causing the analysis of the wheel to always terminate early while carrying much less than full load. This seriously inhibited their ability to design the wheel – consequently the first full-size prototype for mining, while again confirming the concept, did not meet the design criterion they had set.

Challenge

Underground Tire Challenges

Big Tyre is a manufacturer of solid wheels that are primarily used in underground mining vehicles. “Many mining companies have switched from pneumatic tires to solid wheels because of the dangers presented by pneumatic tires underground,” Louden said. “Mines underground have bolts sticking out of the walls that can easily cause punctures. Tires on heavy vehicles are inflated as high as 170 psi, so when they are torn or punctured a considerable amount of force is released. Due to space constraints underground, workers are often in close proximity to the tires so the potential for injury when a tire is torn or ruptures is a major concern.” Because of these concerns, many mining companies have switched to solid rubber-tired wheels or solid tires comprising a pneumatic tire whose interior is filled with foam. These tires eliminate the risk of punctures but their dampening properties are inferior to pneumatic tires so operators experience considerably more vibration. In some mines, operators are restricted to spending no more than 90 minutes a day on machines with solid wheels due to the effects of vibrations. Another problem is that foam-filled solid tires don’t provide as even a distribution of pressure on the ground as pneumatic tires. The highest pressure is usually seen in the center of the tire which can lead to the formation of ruts on the floor of the mine where the mine is working on soft strata. Big Tyre has developed a non-pneumatic, non solid segmented tire which it hopes will overcome these problems and potentially revolutionize the underground mining industry. The tire uses composite leaf springs that are more than four times as flexible as steel. The leaf springs are configured in opposing arrays so the wheels are completely balanced. The inner end of each leaf spring is connected to a steel hub, while the outer end is connected to steel or fiber segments that are bonded to a rubber tread. This design eliminates the need for the tires to be inflated so it eliminates the risk of punctures, reduces heat entrapment and is also expected to provide greater load, speed, haulage distance and longer life relative to pneumatic tires. Compared to solid wheels, the new design provides reduced vibration, increased ride comfort, lower contact pressure, improved even pressure distribution, and improved lateral stability on slopes. While prototype wheels are open so that the springs can be viewed, the production models will be sealed on both sides for safety reasons and to prevent the tire from being contaminated by particles and debris from the mine.

Solution

Introducing Simulation Into The Design Process

The design concept provides the flexibility and challenge of defining various design parameters including the number of springs in an array, thickness of springs, curvature of springs, length of springs, material properties of springs, geometry and material properties of the segments that the springs attach to on the outer diameter of the wheel, as well as many others. The design criterion is to provide a very efficient vertical loading for the size of the wheel while providing similar if not equivalent suspension to a pneumatic

We were pleasantly surprised that Marc was able to handle such a demanding analysis problem in such a short period of time and deliver results that closely matched test results on the prototype.”
tire, with excellent torque capacity and lateral stability. “When you are designing something that has never been done before it can be frustrating,” Louden said. “Using the traditional build and test method, without suitable FEA software, we would have had to build prototype after prototype, which would have been very expensive and time-consuming. It was clear that we needed to simulate the performance of the tire but this was a very challenging simulation problem.”

In order to simulate the performance of the new tire, Big Tyre engineers needed to model the 3D contact between the springs and the segments and between the rubber and the road, the glued contacts between the segments and the rubber and the self-contact of the springs. They needed to address the hyper-elastic material behavior of the rubber, the large deformation and (deliberate)buckling of the springs, and the elastic plastic material properties with failure for the composite leaf springs. They also needed to model the pin-joints between the segments while allowing for large deformations and driving the wheel. Big Tyre engineers tried to evaluate the performance of the new tire using the software package they had previously successfully used to analyze structures. “This software package is excellent for analyzing structures or assemblies with small deflections but could not cope with the highly nonlinear behavior found in the new tire,” said John Shaw, Managing Director of Solidtech Engineering Services, a consulting company that provides engineering services to Big Tyre. “We were only able to solve about 40% of what we wanted and we were suspicious of these results because we were not sure that the nonlinear properties of the wheel were properly taken into account. Most important, we were unable to determine the failure point of the wheel which is the most critical design specification.”

“We talked to Compumod because we discovered an article on the web that described how MSC FEA software was capable of re-meshing during an analysis to handle excessive strain in materials such as rubber,” Louden said. “When we described the problem, they told us that Marc was the right simulation tool to address it.” Big Tyre commissioned Compumod to simulate the initial prototype. Big Tyre provided 3D geometry of the wheel and Compumod imported the geometry into Patran to create the finite element model. Hexagonal elements were used for the springs and rubber and tetrahedral elements were used for the steel segments. Symmetry in one plane was used to reduce the size of the model. The hinges between the different segments and between the springs and segments were modeled with rigid RBE2 elements that allowed free rotation around their own axes parallel to the wheel axis. 3D contact was defined between the ground and rubber. The rubber was glued to the segments through a special glued contact condition. Normal contact was defined between the springs and the segments and between the springs themselves. Big Tyre provided data for the material properties of the rubber and springs by conducting force deflection experiments in their laboratory.

**Results/Benefits**

**Nonlinear Analysis with Marc**

Compumod first conducted a nonlinear static analysis on one spring to correlate the model material properties with experimental data. The material properties were tuned to replicate the measured reaction force in the experiment. Then a nonlinear analysis was performed on the entire wheel to assess its strength. The wheel was given an enforced displacement of 150 mm which was solved in 100 nonlinear increments. The reaction force was then measured on the ground and graphed against displacement. The first negative slope indicated failure of the wheel at 252 kiloNewtons or 25.7 metric tons, which is well over the target of 16 metric tons. After the first collapse of the wheel, the contact between the springs and between the springs and segments added stiffness to the wheel and the reaction force increased again for increasing displacements. “We were pleasantly surprised that Marc was able to handle such a demanding analysis problem in such a short period of time and deliver results that closely matched test results on the prototype,” Louden said.

“After seeing the benefits of the software, we decided to purchase Patran and Marc,” Louden said. “Compumod organized training for us in their Sydney office and handed over the models they created in the consulting project. We very quickly began designing the second full-size version of our design, and have been able to improve the design at a much faster pace than in the past. It even allows us to simulate driving maneuvers of the vehicles, including obstacles on the road.”
Optimizing Engine Performance

Simulation Saves Millions per year by Getting Design Right the First Time

Litens Automotive Group | Based on an interview with Steve Jia, Chief Engineer

Litens Automotive Group’s patented TorqFiltr crankshaft vibration control technology uses an arc spring isolator mechanism to decouple the accessory drive system inertia from the engine torsional vibrations. The product is dimensionally rather small but incorporates a complex mechanism consisting of a series of components that transmit power to each other through complicated frictional contacts rather than fixed connections. The product must frequently be customized to deliver optimal performance for a specific automotive engine. In the past, this involved a time-consuming and expensive trial and error process.

Recently, Litens has developed the ability to accurately simulate the operation of the TorqFiltr, making it possible to accurately evaluate performance of design alternatives and iterate to the optimal design before building the first prototype. “MSC Software’s Marc nonlinear finite element analysis (FEA) software has been used to accurately predict how the design behaves, how components move and react against each other and what happens under dynamic loading conditions,” said Dr. Steve Jia, Chief Engineer, CAE Technologies and Materials Engineering, for Litens. “It is difficult to accurately estimate the cost savings we have obtained through virtual product development (VPD) but we are certain that it amounts to millions of dollars per year across our complete product line.”

Dynamic Tension Control

High static and dynamic accessory belt tensions reduce belt life, reduce accessory component bearing life, increase noise, waste fuel and add weight and cost through coping strategies. Litens specializes in delivering engineered control of pre-set (static) and running (dynamic) belt tension, resulting in accessory drive systems that are efficient, quiet and transmit maximum power under all conditions.

Litens TorqFiltr controls the system resonant frequency by tuning the spring stiffness to the system inertia. Because the spring stiffness is softer than traditional rubber isolators, vibrations from the engine are mostly absorbed before being transmitted to the accessory drive belt. This results in isolation of all components in the accessory drive, and any accessory drive system resonance has very small peak amplitudes since there is little excitation. TorqFiltr springs are made of steel and do not deteriorate like rubber. Additionally, the built-in automatic clutch system eliminates belt squeal associated with resonance of rubber dampers so no separate one-way clutch is needed.

The TorqFiltr device connects to the engine crankshaft through four bolts that connect to the holes in the driver shaft shown in the top half of Figure 1. The driver shaft has two wing tabs that compress the arc springs shown in red on the drawing. The arc springs connect to two shells shown in black in the drawing. Half of the shells have been removed for display purposes. The shells in turn connect to the clutch springs which are shown in gray in the drawing. The clutch springs have a frictional engagement with the pulley that drives the accessory belt.

Very complex engine vibration loading drives the device. The arc springs absorb most of the angular vibration energy of the engine and the clutch transmits power in only one direction, serving to decouple the engine from the accessory drive system. It’s interesting to note that none of the components in the load path have a fixed connection to each other and torque is transmitted only through frictional contacts rather than fixed connections. In addition, the contact conditions including the magnitude, location and direction of the contact forces are continually changing as torque varies or the device rotates.

Challenge

Design Challenge

“This device provides an enormous design challenge,” Dr. Jia said. “We need to fully understand the behavior of the design under dynamic loading conditions in order to ensure that we will deliver the right products the first time. We need to determine the magnitude, location and direction of the action-reaction forces and stress and deformation/deflection on each component and to investigate the contact mechanism in order to achieve...”

Marc not only substantially reduces development time and cost, but also is the only tool that enables us to investigate how the proposed design works.

Marc not only substantially reduces development time and cost, but also is the only tool that enables us to investigate how the proposed design works.
experiments is limited because there are no sensors available at a reasonable cost that can tell us what is going on inside that small assembly,” Dr. Jia continued. “We are still left guessing as to what is happening inside such as contact locations and forces and stresses and deflections of the individual components.”

**Solution**

**Picking the Right Simulation Technology**

Litens has evaluated a number of different simulation technologies. Large displacement dynamic simulation systems such as MSC’s Adams software do a great job of simulating complex mechanisms, however, they are not designed to handle the elastoplastic nonlinearities seen in this application. There are a number of finite element analysis software programs on the market but most are limited to solving within the limits of linear material properties and displacements, small strains and small rotations. Some software packages claim to have nonlinear capabilities but are not able to consistently and reliably solve problems involving continually changing contact conditions between components, large rotational sliding frictional contacts and elastoplastic material behaviors.

On the other hand, Marc was built from the ground up to consistently obtain converged solutions for highly nonlinear problems involving nonlinear materials, large strain and displacement and contacts. Marc also provides multiphysics capabilities, enabling engineers to simulate coupling between structures, thermal, fluid, acoustics, electrical and magnetics.

Litens analysts receive a detailed computer aided design model from the company’s design engineers. They import the geometry into Patran, the company’s pre-processor of choice. The biggest challenge in pre-processing is to reconcile the detailed meshing required in areas such as where the clutch inserts into a slot in the lower spring shell with fillets of 0.2 mm with the need to keep mesh size as large as possible in areas with smaller transients to reduce solution times. Patran gives Litens analysts complete control over the mesh distribution. Analysts typically manually create a surface mesh in critical areas and use the automesher to fill in the less critical areas. The model is then sent to a high performance computing computer with 32 CPU/cores and 256 gigabytes of RAM for solution.

**Results/Benefits**

**Understanding How the Design Behaves**

The simulation results enable Litens to understand how the design behaves, how the components affect each other and what happens as the product rotates through large angles of displacement. “We can see how every component moves and reacts against other components as the product rotates through its full range of motion,” Dr. Jia said. “We can determine contact locations, contact forces, stresses and deflections, many things we need to know to optimize the design of the product. For example, our clutch was originally designed in an S-shape based on previous experience, but the FEA results showed us that a C-shape provided much better performance at no increase in cost. Simulation results with Marc are consistently within 5% to 10% of physical testing results, and even less than 5% in some cases, giving us confidence to use simulation to drive the design process.”

Figure 3 shows the maximum principal stresses at one point in the rotation. The figure shows that the highest stress is in the arc springs. The ability to view stress on each location of each component makes it possible to identify hot spots so they can be corrected to avoid premature failures. At the same time, areas where stresses are low present the opportunity to remove material to save costs. By the way, for this plot the scale was set to 900 Megapascals in order to easily view stresses in the arc spring in relation to other components. Litens analysts lower the scale in order to distinguish differences in stresses among the components with stresses that are so close to each other that they all show up as green in Figure 3.

Figure 4 highlights the ability of Marc to determine the contact locations and forces. The colored areas show the location of the contact and the magnitude of the contact forces. Analysts can determine these values at any point in the rotation or can generate an animation that visually shows the contact locations and forces changing as the device rotates. “It’s not practical to obtain this type of information using physical testing,” Dr. Jia said.

“Marc has been widely used in our everyday VPD to simulate the complete mechanism and to virtually measure everything we need to optimize the designs. Marc not only substantially reduces development time and cost, but also is the only tool that enables us to investigate how the proposed design works and how the product behaves when a physical part is not available during conceptual development stage or a physical experiment is not practical or cost-prohibitive,” Dr. Jia said. “Over the years, we have developed great confidence in both the Marc software platform as well as our own ability to apply the tools in an accurate and consistent method. Obviously this approach saves money and time and has become such an embedded part of our engineering process that I cannot see us developing any new product without this capability.”
Static Analysis
- Perform linear and nonlinear static analysis to virtually test your designs
- Include advanced nonlinear material models for metals, composites, elastomers, plastics, ceramics, powder metals, soils, concrete, shape memory alloys, glass and more
- Incorporate both large deformation and large strain behavior
- Accurately model nonlinear boundary conditions including follower force effects, foundations, and contact
- Perform creep simulations to determine the long term response of the structure.
- Perform post-buckling analysis to perform stability studies
- Determine the inertia relief force to balance free structures
- Perform steady-state rolling analysis of tires
- Perform mechanical wear analysis due to friction
- Export or import DMIG files for compatibility with MSC Nastran
- Perform global-local analysis to better capture local behavior

Dynamic Analysis
- Perform natural mode analysis of structures to determine structural stability under dynamic loads
- Conduct frequency response analysis subjected to harmonic loads or random vibrations to analyze structural performance
- Include advanced damping models that incorporate frequency and deformation dependent damping observed in rubber and plastics
- Obtain insight into dynamic performance of structures through transient analysis
- Gain improved accuracy through accurate modeling of contact, nonlinear materials, and loading conditions
- Create Modal Neutral Files (MNF) that may be shared with Adams including nonlinear preload

Heat Transfer
- Perform steady-state and transient analysis for one-, two-, and three-dimensional bodies
- Obtain temperature distributions in a structure for linear and nonlinear heat transfer problems
- Model nonlinearities including temperature-dependent properties, latent heat (phase change) effect, heat convection in the flow direction, and nonlinear boundary conditions (convection and radiation)
- Compute radiation view factors faster and more accurately
- Simulate thermal degradation of Thermal Protection Systems (TPS) with advanced pyrolysis model
- Perform ablation analysis for space systems, brakes, and bio-medical applications
- Compute heat fluxes across multiple components that come into contact

Thermomechanical Coupling
- Analyze structural response due to temperature changes in the environment and thermal gradients in the structure
- Model heat generation due to plasticity and friction between different components for accurate physics
- Incorporate heat generation due to curing in composite manufacturing.
- Simulate the influence of annealing
- Simulate the effects of changes to thermal boundary conditions due to large deformations

Electrostatics & Magnetostatics
- Evaluate electric fields and magnetic fields in a body or medium
- Compute electric potential field, electric displacement vectors, magnetic induction, magnetic field vector, and more to gain insight
• Model infinite domain with semi-infinite elements for improved accuracy
• Determine the capacitance between electrical conductors in electrostatic analyses
• Calculate the inductance due to wires or coils in magnetostatic analyses

**Electromagnetic Analysis**
• Perform transient and harmonic fully coupled electromagnetic analysis to calculate electrical and magnetic fields subjected to external excitation
• Compute magnetic permeability as a function of magnetic field in a transient analysis
• Calculate magnetic flux density, magnetic field vector, electric flux density, and electrical field intensity along with potential, nodal charge, and current

**Piezoelectric Analysis**
• Simulate piezoelectric effect of coupling of stress and electric field in a material
• Solve for nodal displacements and electrical potential simultaneously
• Perform static, transient dynamic, harmonic, and eigenvalue analysis to better understand material response
• Couple with heat transfer analysis to perform a coupled thermal-piezoelectric analysis

**Coupled Electrostatic-Structural Analysis**
• Simulate the influence of Coulomb forces on structural components and deformation influence on electrostatic field
• Model contact between different bodies and simulate influence of their interaction on the field

**Coupled Electrical-Thermal-Mechanical Analysis**
• Simulate structural response due to Joule heating effects
• Account for nonlinearities that arise due to convection, radiation, and temperature-dependent thermal conductivity and specific heat
• Simulate structures with nonlinearities due to geometric and material behavior
• Use contact analysis to analyze interaction between multiple components

**Coupled Magnetostatic-Structural Analysis**
• Simulate the influence of Lorentz forces on structural components and deformation influence on magnetostatic field
• Model contact between different bodies and simulate influence

**Coupled Thermal-Electrical Analysis (Joule Heating)**
• Compute heat generated due to electric flow in a conductor
• Model temperature dependent resistance and internal heat generation as a function of the electrical flow
• Determine the resistance of the device

**Coupled Electromagnetic-Thermal Analysis (Induction Heating)**
• Compute induced current which generate heat and heat flux
• Incorporate temperature dependency for material data for improved accuracy

**Coupled Electromagnetic-Structural-Thermal Analysis**
• Simulate induction heating with staggered approach of harmonic electromagnetic analysis followed by thermal-stress analysis
• Compute induced current which generate heat and heat flux
• Incorporate temperature dependency for material data for improved accuracy
• Accurately determine the surface strains and stresses
• Utilize either adaptive meshing or a dual-mesh approach to model large motion through the air
• Predict the amount of power required for surface treatment
Simulating Manufacturing Processes
Analysis of Cold Roll Formed Products and Processes
Simulation of Failure and Crack Propagation
Introduction to Simufact.welding
Improving Safety and Reliability with Simulation of Elastomers
Advanced Contact Modeling for Accurate Simulations

www.mscsoftware.com/webinars

ADVANCE YOUR ENGINEERING SKILLS AND YOUR CAREER.

Master FEA.

THE INTERNATIONAL FEA MASTERS PROGRAM

The FEA Masters Program is offered by UNED - National University of Distance Education - the international leader in online finite element analysis education. In 2013, the FEA Masters Program is celebrating twenty years of educating over 3,000 FEA Masters graduates in FEM & CAE. Join the next generation of CAE users. Enhance your engineering knowledge & refine your CAE skills.

Program Partners:

www.uned.es

ANNUAL REGISTRATION OPENS
SEPTEMBER 2, 2013

www.ingeciber.com/uned-masters-registration
MSC’s Solution Portfolio

MSC Software makes products that enable engineers to validate and optimize their designs using virtual prototypes. Customers in almost every part of manufacturing use our software to complement, and in some cases even replace the physical prototype “build and test” process that has traditionally been used in product design.

MSC Products

Simulating Reality, Delivering Certainty

<table>
<thead>
<tr>
<th>Integrated Solutions</th>
<th>Solver Solutions</th>
<th>Mid-Sized Business Solutions</th>
<th>Simulation Process &amp; Data Management</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Adams</strong></td>
<td><strong>MSC Nastran</strong></td>
<td><strong>MSC Nastran Desktop</strong></td>
<td><strong>SimManager</strong></td>
</tr>
<tr>
<td>Multibody Dynamics</td>
<td>Structural &amp;</td>
<td>Multidiscipline Simulation</td>
<td>Simulation Process &amp; Data Management</td>
</tr>
<tr>
<td>Simulation</td>
<td>Multidiscipline</td>
<td>for the Desktop</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Actran</strong></td>
<td><strong>Dytran</strong></td>
<td><strong>SimDesigner</strong></td>
<td><strong>MaterialCenter</strong></td>
</tr>
<tr>
<td>Powerful Acoustic</td>
<td>Explicit Nonlinear &amp; Fluid Structure Interaction</td>
<td>CAD-Embedded Multidiscipline Simulation</td>
<td>Materials Life Cycle Management</td>
</tr>
<tr>
<td>Simulation Software</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Digimat</strong></td>
<td><strong>MSC Fatigue</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>The Nonlinear Multi-scale Material and Structure Modeling Platform</td>
<td>Fatigue Life Prediction</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Sinda</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Advanced Thermal</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Easy5</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Advanced Controls</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Simulation</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Marc</strong></td>
<td><strong>SimXpert</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Advanced Nonlinear &amp; Multiphysics</td>
<td>Multidiscipline Simulation Environment</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

MSC Products

Simulating Reality, Delivering Certainty

Integrated Solutions

- **Adams**: Multibody Dynamics Simulation
- **Actran**: Powerful Acoustic Simulation Software
- **Digimat**: The Nonlinear Multi-scale Material and Structure Modeling Platform
- **Easy5**: Advanced Controls Simulation
- **Marc**: Advanced Nonlinear & Multiphysics
- **SimXpert**: Multidiscipline Simulation

Solver Solutions

- **MSC Nastran**: Structural & Multidiscipline
- **Dytran**: Explicit Nonlinear & Fluid Structure Interaction
- **MSC Fatigue**: Fatigue Life Prediction
- **Sinda**: Advanced Thermal

Mid-Sized Business Solutions

- **MSC Nastran Desktop**: Multidiscipline Simulation for the Desktop
- **SimDesigner**: CAD-Embedded Multidiscipline Simulation
- **Patran**: FE Modeling and Pre/Post Processing
- **SimXpert**: Multidiscipline Simulation Environment

Simulation Process & Data Management

- **SimManager**: Simulation Process & Data Management
- **MaterialCenter**: Materials Life Cycle Management

Simulation Process & Data Management
Optimized compute platforms for CAE workloads
Remote graphics and batch scheduling enabled
Maximize efficiency of available licenses
Simplified integration within existing infrastructures

sgi.com/solutions