Automotive companies must remain competitive through innovation, so engineering teams are pressured to design and deliver quality parts quickly. To succeed, they must develop processes to finalize design from the drawing board to the assembly line in the shortest possible time. A key element in reducing development time is engineering simulation that allows companies to iterate designs to meet product requirements quickly by predicting real-life behavior in a virtual environment long before testing is required. Analysis using high-fidelity, proven capabilities allows engineers to cost-effectively determine performance.
and reliability across a wide range of automotive applications.

The potential for simulation in the automotive industry is vast. The following represents only a fraction of the structural capabilities, applications and benefits of simulation for automotive engineers.

**ANTI-VIBRATION RUBBER**

Anti-vibration rubber has elastic and viscous characteristics. When applied to areas in which automotive parts are connected, it blocks vibration transmission. One method to model anti-vibration rubber is to employ the one-dimensional nonlinear spring-damper COMBIN14 element in ANSYS Mechanical. This element models the nonlinear behavior of rubber by directly using spring characteristics as well as data obtained through experiment. An example of how this technique can be used is a strength analysis of a suspension’s lower control arm for which the nonlinear spring is defined at two bushing positions.

However, a detailed evaluation of the performance of an anti-vibration rubber product requires analysis of a three-dimensional model of the product. It is important to correctly identify characteristics of this nonlinear material to ensure an accurate and converged analysis. To characterize the rubber material, ANSYS Mechanical offers more than 10 constitutive hyperelastic models (including the Mooney–Rivlin and Ogden models). The parameters included in these material models can be automatically determined from experimental data using provided curve-fitting tools. To overcome possible convergence difficulties, a special lower-order tetrahedron element that incorporates a mixed u-P method (for handling both displacement and pressure as variables) is available in ANSYS Mechanical for use with nearly-incompressible rubber materials. Employing this element facilitates solving large rubber deformation and complicated contact conditions that would be time-consuming or more difficult using other element types.

Some anti-vibration rubber products contain liquid that provides fluid resistance to optimize attenuation. An example is a liquid seal engine mount. Detailed analysis of such products must account for both structural and fluid characteristics. To help solve these problems, ANSYS offers powerful multiphysics technology that couples a structural solver (ANSYS Mechanical) and a fluid solver (ANSYS Fluent or ANSYS CFX). This technology enables a fluid–structure coupled solution (two-way fluid–structure interaction [FSI] analysis).

**TIRES AND SUSPENSION**

Tires are the only part of a vehicle that contact the ground, so they heavily influence an automobile’s performance — from safety to the vibration/noise level that determines ride quality. Tire simulation requires modeling technology for complex internal structures. ANSYS offers special reinforcement elements (REINF263–265) to model the many reinforcement structures within a tire.

Pneumatic pressures at work within a tire are not always constant with...
respect to time because of the deformation caused by contact with the ground. These pressure variations due to geometric changes can be expressed by defining a hydrostatic fluid element (HSFLD241–242) for the tire.

Suspension improves ride quality and steering stability. Together with the tires, the suspension is important in controlling the dynamic characteristics of a vehicle. The suspension fulfills its function only while the vehicle’s weight is on it, so the characteristics in the initial state are important. For simulations like this, the ANSYS Workbench environment offers an analysis process called linear perturbation. To perform a static/dynamic analysis of a leaf spring suspension, first add the divided load of the vehicle weight to the suspension model and determine deformation due to the dead weight (static analysis). In this example, this stage consists of nonlinear analysis involving contact between leaf springs in addition to tire modeling as previously described. Next, perform eigenvalue analysis using the same model (modal analysis). All the initial conditions required for this stage, such as deformation due to dead weight and resulting initial stresses,
As simulations become higher fidelity and contain more physics, a solid HPC structure will facilitate solutions.

For frictional heating, a heating effect should be defined for a contact element.

Frictional Heating
The use of coupled-field elements in ANSYS Mechanical (SOLID223, 226, 227) enables a single analysis that includes heating, heat transmission and structural deformation.

Frictional Wear
Frictional wear can be expressed by applying a wear model between contact elements. ANSYS offers the Archard wear model to calculate abrasion loss due to friction.

Brake Squeal/Noise
The generation of squeal noise is related to frictional vibration (self-sustained oscillation) that occurs between the disc and pads. One method to evaluate this vibration is via the stability criterion widely used in control theories. This criterion can identify whether motion is stable or unstable by solving a characteristic equation (complex eigenvalue problem). By setting a frictional co-efficient in the contact elements to ensure performance, safety and passenger comfort.
region between the disc and pads, ANSYS Mechanical can automatically determine stable and unstable modes. Instability is often associated with noise, vibration and harshness.

ENGINE
An engine comprises many parts, and only a small fraction of the possible simulations can be covered in this article.

Gaskets
Gaskets make a structure airtight by acting in a compressive direction. They exhibit significant nonlinear characteristics from the compressive load. Special elements (INTER192–195) and a gasket material model within ANSYS Mechanical can be used to model a complex gasket assembly — for example, to evaluate behavior of a gasket installed between the cylinder block and the head.

Bolts
Bolts are commonly used to connect parts in a wide range of machinery and applications. For example, bolts connect the cylinder block and head (with a gasket between them). ANSYS Mechanical contains a bolt-modeling capability that can be easily configured to represent the tightening force and subsequent change in preload due to other external loads.

Although a bolt thread can be accurately modeled using contact, this method usually requires a significant increase in calculation cost due to the inclusion of minute details. ANSYS Mechanical offers technology to simulate the influence of the bolt thread simply by using thread pitch information.

OTHER PROCESS COMPRESS

Password

CA

Bolt modeling in ANSYS Mechanical

Using the bolt section method to model bolts can speed simulation time by a factor of 10.

Mechanical is also equipped with acoustic analysis capabilities that can be used for powerful electromagnetic field–structure–acoustic multiphysics coupled solutions when used in conjunction with ANSYS Maxwell electromagnetic analysis.

Structural simulation is only one way that companies can expedite the design process for automobiles. Fluids and electronic simulation are also required, and by using all these physics together on a single-platform multiphysics environment, full insight into real-world performance can be gained. As simulations become higher fidelity and contain more physics, a solid high-performance computing (HPC) structure will facilitate solutions. Also, automated parametric studies and multiple physics generate large amounts of data that must be managed and shared, so a specialized knowledge support system is a must. As the challenges to produce and deliver better and faster products in the auto industry accelerate, ANSYS delivers solutions no matter the scale or scope.