Virginia Tech Uses Finite Element Models to Predict Suspension Member Loads and Compliance in a Formula SAE Vehicle



A racing vehicle suspension system is an elastic structural linkage that defines many of the handling characteristics of the racecar while supporting it under complex loading scenarios. The Formula SAE team at Virginia Tech uses Abaqus to determine suspension member loads and compliance to design a safe and effective suspension system.

An Abaqus finite element model was developed for the right front suspension corner to determine member loads and suspension compliance. The finite element model was initially developed with beam elements, allowing axial, bending and torsional loads, and connector elements to represent kinematic degrees-of-freedom of the joints. Welded control-arm members on the upright, simplified pull-rod (PR) mounting brackets on the upper control arm (UCA) and the toe-rod (TR) were also represented. The pull-rod member actuates the spring damper through the bell crank to the wheel assembly, previously mounted on the upright. The front-corner model used several design load cases, revealing reduced axial loads and increased bending moments reducing the critical buckling load of the compressive members.

The Abaqus front-corner suspension model incorporates steering loads from the driver, input as prescribed displacements at the toe-rod (TR). The steering linkage displacement changes the upright orientation angles and wheel load directions applied to the suspension. Including steering inputs in the FE model results in significant changes in member loads for load cases involving slip-angle effects used to determine the effective steer angles during cornering. Quasi-static design loads are used to predict the vertical movement at the wheel center and associated member loads. The suspension designer uses these results to determine if changes to spring rate or anti-roll bar stiffness will result in a more desirable wheel movement for a given loading condition.

Determining which suspension components contribute most to camber and toe compliance in the suspension system requires a more refined finite element model. The purpose of the suspension compliance model was to match camber and toe stiffness goals and breakdown stiffness on a component level. A three-dimensional finite element model of the suspension linkage parts was developed from a full-car solid model, importing the front suspension corner parts into Abaqus. A discrete rigid tire model was used to avoid confounding the suspension compliance with the tire elastic response. Assembly constraints were used to locate the suspension parts, defining sets and contact surfaces in the part models. The model makes extensive use of partitioning and sets to parameterize part meshing using C3D10M elements.

Continued on page 21

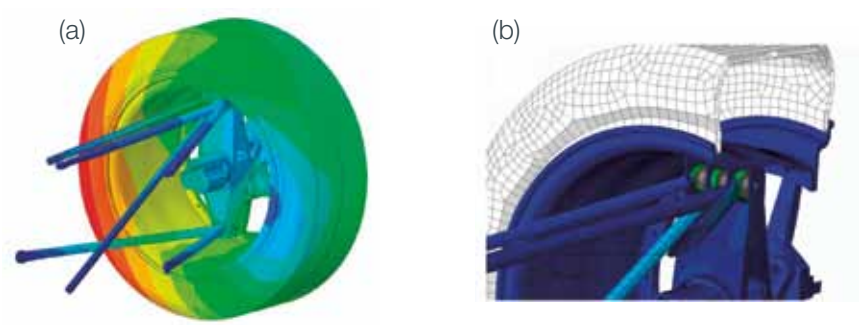

Figure 1. Right front corner compliance model, (a) displacement magnitude for the lateral load under acceleration load case, (b) von Mises stress in the rod-end connections.

Contact-pair modeling between joint components was as realistic as practical, using shear and frictional forces between contact pairs to mimic bearing and clamp loads. Driving condition load cases were applied at the tire contact patch with boundary conditions applied at the inboard suspension points on the chassis. Two load cases were studied for suspension compliance, the 5g bump and pure lateral load under acceleration cases. Figure 1 shows the displacement magnitude for the lateral load under acceleration load case and the stresses in the rod-end connections. The compliance at the wheel center was used to estimate the overall corner compliance. The displacements at the joints were queried to determine the load distribution and relative displacement in the rod-ends, including whether or not the joints reached the hard stops in the spherical bearing.

Virginia Tech FSAE uses Abaqus extensively for design, analysis and simulation for chassis, suspension and driveline subsystems. Previous modeling emphasis was associated with load transfer through contact modeling in assemblies and subsystems, suspension, chassis mounts, bell-crank design and integrated composite structure. The immediate future design and modeling efforts are to support dynamic load simulation and model correlation with vehicle system test. The Abaqus simulation environment allows Virginia Tech FSAE to evaluate integrated vehicle subsystem and systems design, in addition to component level analysis. Effective use of integrated multi-step simulation is key to making better design trade-offs and optimizing vehicle performance.

Lane Borg
Associate Engineer, Performance Test Driver
Goodyear Proving Grounds,
San Angelo, TX
Lane_Borg@goodyear.com

Johnson Miles
Associate Engineer Crane and Large Derrick
Product Development
Daleville, VA
Johnson.miles@altec.com

Bob West
Associate Professor
Mechanical Engineering Department
Virginia Tech
Blacksburg, VA
westrl@vt.edu

For More Information
www.vtmotorsports.com

Introducing the SIMULIA Online Learning Community



SIMULIA has always understood the importance of developing a strong presence in the academic community. Since the earliest releases of Abaqus, we have made our software available to researchers. Abaqus quickly became the simulation software package of choice because of its unique nonlinear capabilities and effectiveness solving difficult, real-world problems. This wide-spread adoption helped Abaqus technology stay on the cutting edge of many areas of academic scientific and engineering research.

Expanding the SIMULIA academic community helps to increase availability of talented engineers qualified to tackle tomorrow's challenges in realistic simulation. In support of this goal, we developed a formal academic program in 2003 with a team dedicated to promoting SIMULIA products to academic institutions in the United States and around the world. Since then, we have delivered the Abaqus Teaching Edition license for classroom use, the extremely popular Abaqus Student Edition—available for personal use at home, on campus, or in the office—and engaged in countless projects aimed at expanding our overall academic presence around the world.

As part of our ongoing commitment to engineering education and to developing strong ties between Academia and Industry, we would like to announce the availability of the online SIMULIA Learning Community. Our community platform is a brand new way for you to experience SIMULIA technology; a place where our user community can collaborate across industry and academic boundaries to share examples, tutorials and methods related to simulation and analysis.

In the community site you can:

- post and read blogs
- access tutorials, examples, and research papers
- ask and answer questions
- post job openings
- create your own network of similar-minded members to share ideas and activities

For example, you can access video tutorials outlining the complete process of creating and running a simulation in Abaqus/CAE. If you're curious about how Abaqus is used in the Aerospace industry, you can watch a presentation created for new aerospace engineering students at Old Dominion University. For those affiliated with an academic institution in North America and Europe, you will find a link that (with the approved credentials) will allow you to download a free copy of Abaqus 6.10 Student Edition.

While we are just getting started, we invite you to join our community to share your interests and leverage the wealth of knowledge and resources available in the SIMULIA users' ecosystem!

A user account must be created to join the learning community.

For More Information
swym.3ds com/#community:73