Altair® MotionView® and ABAQUS for Direct Suspension Bushing Tuning

Jon Quigley, Mike White, Jeff Liu, Andy Rocha

Altair Engineering, Inc.

Automotive development engineers analytically tune suspension bushings using a multi-step process of changing the bushing rate, measuring the effect of the suspension changes, and confirming with the bushing designer that the desired rate can be achieved. A single process to simulate bushing tuning would have the key advantages of eliminating the step between the multi-body dynamics assessment and the FEA analysis of the bushing properties, thus leading to a simulation containing both mechanical systems as well as finite element analysis. This paper introduces Altair MotionView and ABAQUS as an integrated solution to accurately tune rubber components in a Multi-Body Dynamics system model. The process and solution impact will be discussed.

1. Introduction

Suspension analysis has traditionally been performed by decoupling the study of the mechanism from the study of the connector entities between the suspension components. In the case of a bushing between a control arm and subframe for example, the bushing is typically described as an empirical fit of test data or results of a separate FEA analysis, either of which require an additional step in the process of predicting suspension behavior. This paper describes how HKS/ABAQUS and Altair® MotionView® can be used to generate one model containing both the multi-body representation of the suspension as well as the FEA representation of the bushing. The MBD (multi-body dynamics) portion of the suspension utilized the new connector elements available in ABAQUS v6.1.

2. Model and Analysis Description

The suspension used in this study was a short-long arm (SLA) which is a fairly common automotive suspension (Figure 1). The description of the multi-body topology, FEA bushing representation and applied analysis is described in this section.

2.1 Multi-Body Representation

The topology and data that describe the SLA suspension was represented in MotionView’s MDL (Model Definition Language). The model was constructed from the suspension library available with MotionView and defines the entities of the mechanical system, their connectivity, data properties, and graphics (Figure 2).
The primary interest of the multi-body representation is the connectivity of the suspension. Kinematic connectivity is represented in MotionView by entities such as *BallJoint, *RevJoint, and *InlineJoint. This type of connectivity defines constraints and degrees of freedom between two bodies, such as a control arm and knuckle for example. Compliant connectivity is represented in MotionView by entities such as *Bushing and *SpringDamper and apply to an attachment between a control arm and subframe for example. For compliant entities, the stiffness and damping properties can be linear or non-linear.

For this study, MotionView utilized a prototype ABAQUS solver writer in conjunction with an API to map the native MDL elements to the ABAQUS connector elements (Figure 3). Out of eight bushings in the suspension, seven used this traditional compliant connectivity (the model was designed to be asymmetric to highlight the difference between the two approaches of modeling a bushing). The remaining bushing was represented as a full FEA model as explained below.

2.2 FEA Representation

The difference between the suspension model used for this study and traditional suspension models is that the front left bushing was represented as a full finite element model in ABAQUS. Instead of representing the bushing stiffness as an empirical fit to force/displacement data, this bushing contains the physical description of the bushing itself (Figure 4) which includes property data and dimensions.

2.3 Loading

The suspension model was actuated using the MotionView *ActionOnlyForce to apply a quasi-static lateral force to both sides of the suspension. This type of loading, when distributed through the suspension components, results in displacement of the control arm bushings (Figure 5). As with the connector entities, MotionView mapped the MDL force to the ABAQUS input deck syntax. The quasi-static simulation solves the equilibrium condition within a user defined tolerance for a set of sequential time steps.

2.4 Combined Representation and Model Export

This section will describe the generation of the final ABAQUS model both at the prototype level for this study as well as the methodology targeted for final implementation in MotionView.

The prototype ABAQUS writer required the FEA portion of the final model to be exported by HyperMesh and subsequently re-used through a MotionView Templex template. The template entity in MotionView provides users with the capability to export parametrically substituted syntax directly into a solver input deck such as ABAQUS.

The entities represented in MDL mapped to the ABAQUS solver at the time of model export through the ABAQUS solver writer. The resulting ABAQUS model therefore contained both MBD connector elements as well as the FEA model of the bushing. This file was submitted to the ABAQUS solver.

Note: The final implementation of the MotionView coverage of ABAQUS will allow each FEA component of the final model to be represented as an Altair H3D file that gets generated from HyperMesh. This file contains all of the information required for export of the FEA portion to ABAQUS as well as information required for graphics of the FEA model to show in the pre and post processing windows of MotionView (Figure 6, Figure 7).
3. Results and Post-Processing

The model was submitted to ABAQUS and the solver predicted the motion of the system that contained MBD and FEA elements, with the primary source of compliance coming from the FEA model of the suspension attachment bushing.

4. Potential Applications

The capability of combining MBD & FEA modeling using MotionView and ABAQUS allows for a new set of potential market applications. In the field of vehicle dynamics and chassis modeling for example, one can now simulate multi-axial loading of bushings as well as non-linear flexible components such as twistbeams. The non-linear capabilities of ABAQUS provide functionality over the traditional CMS (component mode synthesis) based flexbodies due to CMS being applicable only to linear components.

5. Challenges

As is typical in the CAE industry, there exists a trade-off between accuracy and performance when considering the type of modeling approach to take. In the case of mixing FEA based components and rigid body MBD components and connectivity, one has to compare the accuracy gained with the potential increase in simulation time.

6. Summary

When used together, MotionView and ABAQUS can provide analysts with the capability to generate and simulate a combined multi-body dynamics and finite element model. This functionality can be applied to disciplines such as prediction of suspension behavior and combined loading of rubber elements for example.

7. References

- Fundamentals of Vehicle Dynamics, Gillespie (1992)
- SuspensionGen, Altair Engineering, Inc.
8. Figures

Figure 1. Example SLA suspension
Figure 2. MotionView MDL suspension model.
*BeginMDL( the_model, "MBD Model" )
 *Body( b_0, "Body 0", , , , )
 *Body( b_1, "Body 1", , , , )
 *BallJoint( j_0, "Joint 0", MODEL.b_0, MODEL.b_1
 , MODEL.P_Global_Origin )
 *EndMDL()

**|------------------------------|
**|  PART/30301                |
**|------------------------------|
**| RIGID BODY, REF NODE="PART/30301 CG ORIGIN", TIE NSET="PART/30301 LPRF POS"
**|------------------------------------------
**| Define rigid body and set CG as the reference node
**|------------------------------------------
**| NODE, NSET="PART/30301 CG ORIGIN"
 | 9000001,  2100.,  0.,  1250.
 *ORIENTATION, NAME="PART/30301 CG AXES"
 | 1.,  0.,  0.,  0.,  1.,  0.

Figure 3. MDL language and ABAQUS syntax
Figure 4. Bushing represented with FEA model.
Figure 5. Example loading.
Figure 6. Pre-processing using MotionView
Figure 7. Post-processing using MotionView