Flexible Body Suspension System Modeling and Simulation Using MD Nastran SOL700 in VPG Environment

Casey Heydari, Ted Pawela
MSC.Software Corporation
Santa Ana, California
U.S.A.

Tim Palmer, Arthur Tang
Engineering Technology Associates, Inc.
Troy, Michigan
U.S.A

Keywords :
FEA, Dynamic Non-Linear, Rigid Body, Flexible Body, Road Load, Fatigue, Durability, Proving Ground

Abstract

Automobile durability and fatigue life prediction depend on road load generation approaches including proving ground testing, laboratory measurement and CAE simulations. Traditionally, the road load simulations are done with ADAMS using rigid body and modal flexible body approaches. With the newly available MD Nastran SOL700 capability, a new approach of using FEA-based flexible body modeling for suspension systems will be an attractive supplemental solution.

MD Nastran SOL700 is an LS-DYNA based explicit solution which is capable to correctly simulate FEA flexible body structure, motions, and kinematics while accurately accounting for the material and geometric non-linearity of a suspension system.

This paper will focus on the methodology development to enable efficient conversion from an ADAMS model in XML format into MD Nastran SOL700 or LS-DYNA format in ETA's VPG environment. In the VPG environment, a rigid body suspension model can be easily changed to FEA based flexible body model for either MD Nastran SOL700 or LS-DYNA solution.

A demonstration case will also be provided to illustrate the process and approach.

Introduction

Automotive structure development relies on the interaction of various disciplines within the vehicle development activity to share data and pass critical information between various design attribute development activities. One such cooperative arrangement in the vehicle development scope is the relationship of load prediction and the durability of the vehicle body or suspension structure.

The loads must be identified early enough to accurately develop an optimum structure – one that is not too weak, yet meets all of the requirements for mass and stiffness. Therefore, early knowledge of the loads for durability events allows engineers to meet this optimum balance of mass and strength.

While this process relies on the sharing of data to achieve this result, the acquisition of the loading data is troublesome. In most instances it requires that a prototype system be created and tested, measuring the forces at strategic locations during operation. In the current, time-compressed product development process, the time and resources required to create and test such a prototype is impractical and unavailable.

The need for sharing data between applications such as multi-body dynamics software and finite element analysis, provides the engineer with the opportunity to streamline the product development process by accurately predicting the forces and passing this information to the finite element program for calculation of the resulting deformations and stresses.

Various factors limit the usefulness of this data exchange. One such limitation is the requirement of the product to be tested as a system, with all of the component interactions and nonlinear behaviors of mounts, bushings and connections. Another is the consideration of the component deflections during the operation of the system.

A solution to these limitations is available in the method known as Virtual Proving Ground, where the system in question is studied completely in a finite element analysis, not only as a matrix embedded in the multi-body solution.

Recent developments in Nastran have allowed users to fully consider systems as a collection of flexible components, with complete interaction of the various components as well as consideration of geometric and material nonlinear effects. This results in the ability to completely simulate the behavior of a system using a single model, and a single solution sequence.

This paper will describe this process of assembling a model for a system level simulation, and discuss the results of such a simulation, as well as compare the results of a rigid system simulation with a flexible system simulation and a corresponding physical test result. It also discusses how graphical tools and solver interfaces allow for sharing of data for which there may not be a direct correspondence, such as multi-body dynamics data and finite element data.

MD NASTRAN – Solution 700

MD Nastran is an enterprise class solution that will provide simultaneous multidiscipline system level simulations to accurately predict the real life scenarios. MD Nastran combines the best-in-class technology platforms including MSC.Nastran, Marc, Dytran and LS-Dyna into one fully integrated multidiscipline solution for the enterprise. This is a paradigm shift from single point solution tools and it can dramatically reduce the time-to-solution up to 50% when compared with bundled single-point simulation tools because now one single common data model can be used in place of multiple models for uncoupled discipline analysis using multiple single point tools.

SOL 700 is the powerful Explicit Nonlinear Solution available in MD Nastran and offers unprecedented new technology to analyze transient dynamic events of short duration with severe geometric and material nonlinearities.

MD Nastran SOL 700 allows users to work within one common modeling environment using the same Bulk Data interface. The NVH, linear and nonlinear models can be used for explicit applications such as crash, crush, and drop test, blade out and bird strike simulations. This dramatically reduces the time spent to build different models for implicit and explicit analysis and prevents the users from making mistakes because of unfamiliarity between different programs.

Applications such as durability and simulation of flexible body suspension systems demonstrates how point solutions from MSC.Adams, Nastran and LS-DYNA can be fully integrated in to a multipledisciplined enterprise solution with eta/VPG and MD Nastran SOL 700.

Interface Development in eta/VPG

In order to efficiently use the SOL700 software for simulation of a system level model, it is necessary to create a graphical user interface which supports all of the modeling tasks and data structure of the solver.

Taking advantage of the existing interface with LS-DYNA, and the tools existing within the software, eta/VPG was adapted to support all of the required MD Nastran data. This was accomplished by re-structuring the existing LS-DYNA and NASTRAN interfaces, to completely support all of the entities which were added in the MD Nastran SOL700 software.

In addition, an interface was developed which allowed the importation and re-use of existing MSC ADAMS data. This data is in the form of either an ADAMS (*.adm) data structure, or the XML format data created from MSC ADAMS pre-processing environment. Automatic reading and conversion of this data allows the user to efficiently re-use the data, with minimal intervention, resulting in a FE-based model which has similar topological structures, bushings, springs and dampers that are needed to accurately model a suspension system.

The ADAMS interface is a crucial component in the capability of creating complex suspension system models for use in MD NASTRAN. The automatic conversion of data structures is necessary due to the complexity in the various connections, especially at the joint or bushing connections. These connections may require multiple coincident nodes, which become difficult to visually identify and manage via the user interface. The automatic conversion of this data provides a simple way to reliably convert the data without requiring the user to be an expert in either the multi-body or finite element software.

Conversion of ADAMS XML Data

The VPG user interface allows users to directly import ADAMS formatted in an XML standard. This import function creates a corresponding finite element entity for the XML data, and populates the data according to the parameters defined in the XML data. For instance, a PART definition in the ADAMS file would be converted to a *Part_Inertia, using the part center of gravity, mass and inertia properties found on the PART definition in ADAMS. Similarly, JOINT definitions would be translated to a corresponding *CONSTRAINED_JOINT definition in VPG, with NODES created for the proper definition of the joint. Local coordinate systems may also be automatically created, depending on the joint definition options found in the XML file.

Since the ADAMS XML data is used to define suspension systems, the VPG interface identifies the specific suspension type which the XML data defines, and creates a corresponding topology to which the XML data is converted into.

As an example, an XML data may define a McPherson Strut suspension, as identified by the XML Tag. This prompts VPG to then define a topology or “template” into which the McPherson Strut suspension data is translated. Various options are also identified in the XML, which then creates corresponding topologies in the VPG data. An example of this optional configuration would be the existence of a stabilizer bar, and further, whether that stabilizer bar is attached to the lower control arm or the knuckle. The VPG interface has been developed to automatically import and convert the following suspension types: SLA, McPherson, Multi-Link, Trailing Arm, and Rear Coil Spring suspensions.

Since the XML data could possibly be incomplete or inconsistent with the requirements of a finite element model, several data checks have been implemented into the translation process. While multi-body models may be run “kinematically”, where no component masses or inertias are used in the calculation, the finite element models require that the components have an appropriate mass and inertia property. Other warnings are generated when the XML data structure is incomplete or has customized parameter definitions, as in a user-defined subroutine.

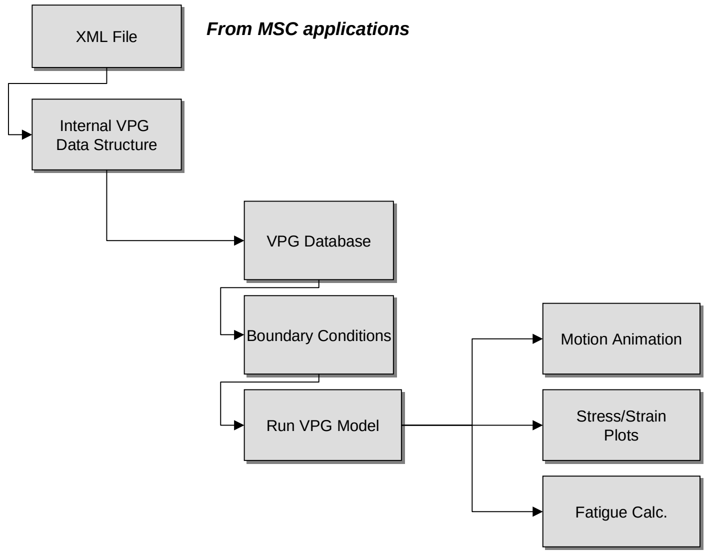

Figure 1: Data Flow for ADAMS XML Translation

Once the data has been imported successfully, VPG allows for boundary condition definition created by the user, and also provides a toolset of standard loading conditions which can selected from a library and applied to the suspension system.

At the completion of the import and translation operation, a complete, functional finite element model of the suspension exists in MD NASTRAN format.

VPG allows the user to perform operations on the imported data to edit any of the suspension joint locations, the bushing stiffness and orientations, and the damping parameters. These are all accessed through the eta/VPG Suspension Menu. This menu also provides tools to visualize the suspension. During the visualization eta/VPG creates temporary surfaces which are then shaded for the viewing of the suspension components.

Since the ADAMS model is typically “grounded” at the suspension to body attachment points, eta/VPG automatically creates a rigid body definition for the suspension to be connected to. The user may conveniently delete this part when connecting the suspension to a complete system model.

Examples of the resulting suspensions are shown in Figure 2.

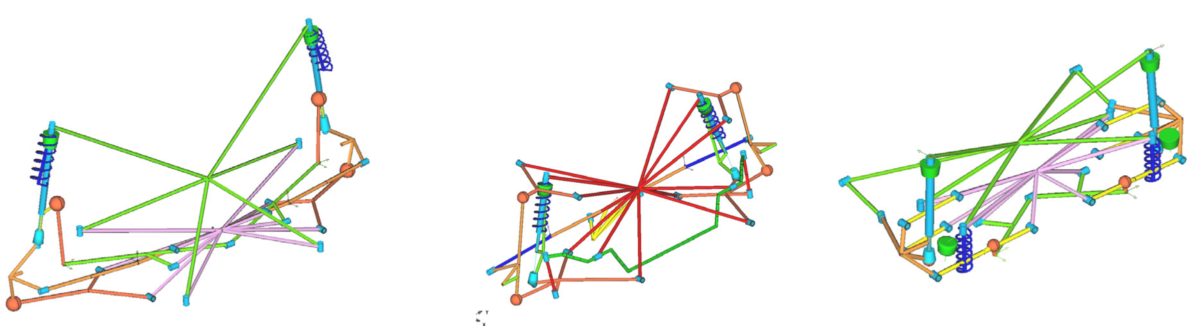

Figure 2: McPherson Strut, SLA and Rear Multi-Link Suspensions

Case Study: Translation of a McPherson Strut Suspension System into MD NASTRAN SOL700 using eta/VPG

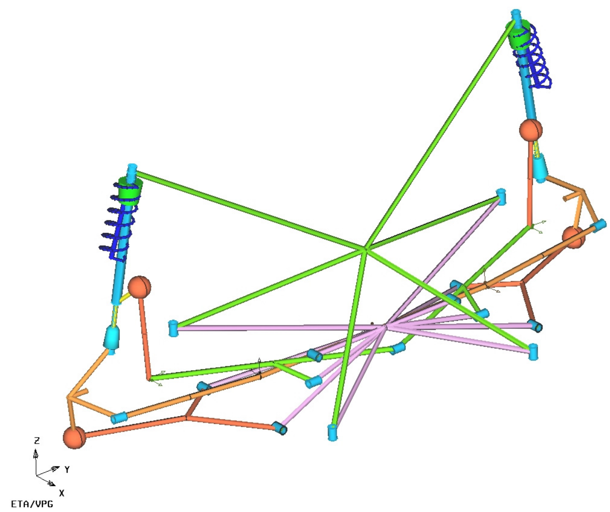

Figure 3: Translated McPherson Strut Suspension ADAMS Model in VPG

The case which will be presented to demonstrate the process of model conversion, and the performance of MD NASTRAN in system level simulations is a McPherson Strut suspension system. This model, shown in Figure 3, was originally created for simulation in ADAMS and exported via ADAMS/CHASSIS.

The ADAMS model was imported into VPG via the XML data, where the components were all defined as rigid components, with the appropriate mass and inertia properties. Connections were made between the components via discrete elements (NASTRAN ELEMENT TYPE). The resulting model corresponds directly to the ADAMS model from which it was translated.

In order to verify that the model was successfully translated into a finite element structure, a series of simple suspension articulations were performed. The same simulations were performed using MD NASTRAN. The forces generated in the suspension joints and bushing were compared. These results are shown in Tables 1 and 2. As shown in these figures, the ADAMS and MD Nastran results are in agreement.

Results

Several cases were considered for evaluation of the translated suspension system. The 2 cases shown in this paper are 3G load applied in the fore/aft direction, at the wheel center. The second simulation considered was a 4G load applied in the fore/aft direction, at the wheel center.

Table 1: Results for 3G Fore/aft Loading at Wheel Center

3G Vertical Input	Fmag (N)		Fx (N)		Fy (N)		Fz (N)	
	ADAMS	MD Nastran	ADAMS	MD Nastran	ADAMS	MD Nastran	ADAMS	MD Nastran
LCA Frt Bushing on Subframe	3145.94	3342.99	-246.96	-333.16	-3071.23	-3256.90	635.19	676.17
LCA RR Bushing on Subframe	177.53	292.06	-104.90	-136.32	-143.22	-257.50	0.54	20.30
Outer Tierod End	412.03	493.52	-55.32	-66.73	-408.21	-478.64	8.16	100.05
Upr Strut Mount	5785.67	5995.27	430.08	406.90	2079.51	2094.90	5381.88	5602.60

Table 2: Results for 4G Fore/Aft Loading at Wheel Center

4G Fore/Aft Input	Fmag (N)		Fx (N)		Fy (N)		Fz (N)	
	ADAMS	MD Nastran	ADAMS	MD Nastran	ADAMS	MD Nastran	ADAMS	MD Nastran
LCA Frt Bushing on Subframe	23839.75	25535.08	3382.85	2605.70	-23598.50	-25401.0	30.21	199.80
LCA RR Bushing on Subframe	19570.21	21290.61	9422.54	11069.00	17152.10	18187.00	119.19	16.30
Outer Tierod End	6603.40	6984.48	887.77	651.07	6542.84	6945.40	-89.79	-347.20
Upr Strut Mount	3021.51	3306.22	2904.96	3140.30	-823.86	-1034.20	109.47	-3.40

The computer resources required to make these calculations is relatively small. All simulations were conducted using an 2.6 GHz Intel Pentium 4 processor, with a Windows XP operating system and 1 gigabyte of memory. The simulations in this study were performed in 124 seconds of CPU time, and 125 seconds.

Conclusions / Summary

As a natural extension of model data exchange requirements, the common data requirements of MD NASTRAN and LS-DYNA has allowed for the development of an interface which accepts models created in MSC.ADAMS software for simulation of mechanical systems.

It has been demonstrated that the multi-body data of an MSC.ADAMS model can be converted into equivalent finite element entities, which can then simulated using MD NASTRAN. It has also been shown that the results of these simulations are within acceptable ranges, demonstrating agreement between the simulations.

Future work will include the inclusion of fully flexible bodies into the simulations, to demonstrate the ability of MD NASTRAN to simulate system-level models.

Acknowledgements

The authors would like to acknowledge Dr. Liang Tang of Ford Motor Co, and Mr Shawn Zhang, formerly of ETA for their contribution and guidance in this work.