Virtual Proving Ground – A CAE Tool for Automotive Durability, Ride & Handling and NVH Applications

Arthur Tang - Vice President, Business Development
Engineering Technology Associates, Inc.
1133 E. Maple, Suite 200
Troy, MI USA
E-mail: atang@eta.com

Nasser Tamini – Senior Project Engineer
Engineering Technology Associates, Inc.
1133 E. Maple, Suite 200
Troy, MI USA
E-mail: ntamini@eta.com

David Yang – Project Engineer
Engineering Technology Associates, Inc.
1133 E. Maple, Suite 200
Troy, MI USA
E-mail: dyang@eta.com

Keywords:
Virtual Proving Ground, durability, ride & handling, NVH, time domain, frequency domain

ABSTRACT

The Virtual Proving Ground approach has been developed for simulation of dynamic nonlinear events as applied to automotive durability, ride & handling and noise/vibration/harshness applications. This finite element analysis technique provides a unique method to create and analyze vehicle system models, capable of including vehicle suspensions, powertrains and body structures in a single simulation. Through the development of this methodology, event-based simulations of the vehicle performance, over a given three-dimensional road surface can be performed.

The development of methodologies and approaches for performing this type of analysis will be discussed, which make up a "Virtual Proving Ground" environment.

Case studies will be presented to show the application of this methodology to a full vehicle system for vehicle durability, ride & handling and noise/vibration/harshness applications.

The results of this case study will highlight the potential applications of this approach, as well as the challenges associated with the method.

INTRODUCTION

Automotive finite element analyses aimed at predicting vehicle durability have been traditionally performed using a simplified, segmented approach. The approach is made up of three components:

1. Road-Load Generation
2. Static Linear Finite Element Analysis
3. Fatigue Life Analysis

When performing these studies, various assumptions are made inherent to software tools in use and have been discussed in numerous technical papers, and journals. Simply put, these assumptions are:

- Vehicle suspension system components behave in a rigid manner with translations and rotations occurring only at joint locations.
- Vehicle structures behave in a linear manner with loads applied as a static condition.
- Vehicle constraints are applied to allow for the solution of the static analysis.
- Full reversals of the peak stresses or strains occur for each loading condition.
- The initial stress state in an element is zero.
- Specialized, separate software must be used to predict the behavior of mechanisms and predict the operating stresses of mechanical systems.

Many different analysis techniques have been developed to minimize or compensate for the effect of these assumptions. These techniques have resulted in a somewhat standardized approach to solving vehicle durability problems in use at many auto manufacturers today.

Methods have been developed that are currently in use, which allow the use of unit loads applied at the vehicle suspension mounting points. This method determines preliminary stress results, which are then multiplied by measured load data, to provide a fatigue life estimate of the vehicle structure [1].

This method, while providing a reasonable basis for the prediction of vehicle fatigue life, is limited in the application due to the lag between design period and prototype vehicle testing.

Combinations of software packages have been presented which address the individual components of the analyses, such as kinematics/dynamics using ADAMS, combined with NASTRAN to solve the stress condition [2], [3], [6], [8].

This approach, while addressing the main problem of providing simulation of "events", is limited in use by the small-displacement restrictions of NASTRAN element formulations. It is also limited in the sense that the ADAMS analysis does not have the overall vehicle structure response included in the loads generated.

Additional limitations are present due to the fact that two separate software packages are required to perform the simulation. This requires additional levels of expertise to be possessed by the engineer. In many organizations, different departments possess these areas of expertise.

The approach presented here relies on the integration of CAE disciplines to shorten the time required to develop a fatigue life estimate and provides that information using a system approach.

This new methodology also addresses the issues brought forth by the use of multiple software packages. This methodology requires single commercially available software for the solution of dynamic full vehicle simulation.

Full vehicle simulations also require the modeling and analysis of the vehicle as a system. This vehicle system includes all structural components as well as rubber bushings and mounts, springs, and shock absorbers. In addition, this vehicle system requires tire models which can transmits inputs from the road to the structure.

While tire modeling and development has been presented by Kao, et. al., the application of such tires to full vehicle simulations has not been considered [9].

The methodology presented here includes the vehicle system components such as bushings, springs, and tires to allow for the interaction of the tire and road surface. The VPG method, while developed by the author and colleagues, has been subsequently presented independently as an “Analytical Proving Ground”. The potential of this technology has been identified and shown through application examples [8].

APPROACH

Methodology Development

The authors in previous work [3], [6] have presented the concept of using a dynamic non-linear analysis technique to perform kinematic and dynamic analysis.

The previous work discussed an approach to simulate kinematic analyses using the software LS-DYNA. This work compared the results of such a kinematic analysis to both ADAMS analyses and theoretical calculations.

The conclusion of this work was that such an approach can be useful to provide component stress and deflection data in a dynamic manner.

This idea has been expanded into a methodology to perform full vehicle simulations using a dynamic nonlinear analysis approach.

The methodology makes use of dynamic nonlinear simulations to simulate both the dynamic nonlinear behavior of the suspension system, as well as the material and geometric nonlinear behavior of the body structure.

The “Virtual Proving Ground” approach grew from simple mechanism simulations that were used to evaluate the performance of flexible mechanism components. It was shown that LS-DYNA has the capability to perform such simulations and the results of these analyses are useful and accurate for the evaluation of mechanical systems. The “Virtual Proving Ground” methodology now can be shown to predict the forces that are generated in vehicle tests and calculate the stresses generated during such events.

The Virtual Proving Ground Concept

The concept of the Virtual Proving Ground evolved from the limitations and frustrations of attempting to explain the limitations and correct usage of analysis results to engineers not experienced in simulation techniques.

The many assumptions that are used as disclaimers in traditional analyses result in the loss of respect or appreciation for the analysis results. An approach that would limit the assumptions being made, addressing many of these concerns and providing an event-based simulation is thought to increase the acceptance of the analysis results to the designer.

The ability to perform an event-based simulation results from the ever-increasing computer processor speeds, the relatively low cost of these processors, and, most importantly, the ability of commercially available analysis programs to efficiently solve this type of multi-discipline problem.

Integration of these disciplines: road load generation, vehicle modeling, stress analysis and fatigue life prediction, make it possible to predict vehicle loading and stresses simultaneously and use these results to predict a fatigue life.

The development of the Virtual Proving Ground required the creation of several components. These components are used in the development of the models and the processing of the analysis results.

Figure 1 shows the overall modeling and analysis process and contrasts it with the traditional automotive analysis process.

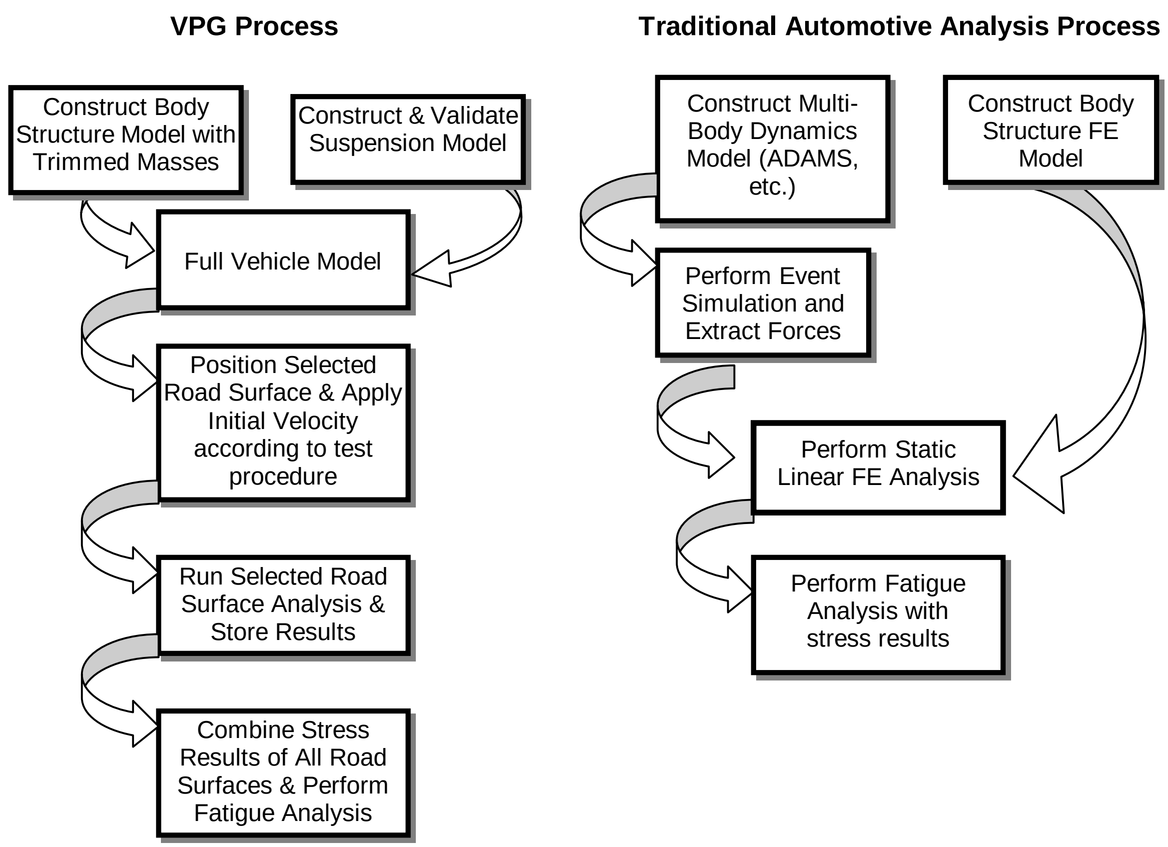

Figure 1. Traditional Automotive Durability Analysis Methodology Process vs. VPG Process

Vehicle Modeling. In the Virtual Proving Ground Methodology, vehicle structure models are constructed in much the same manner as traditional durability finite element models. These models contain sufficient detail to predict stress levels in local areas of the structure and while maintain an overall constant mesh density.

All structural components are modeled and connected by means of rigid beam elements simulating spot welds.

Since the analysis technique used is based on explicit nonlinear finite element techniques, the use of small elements is discouraged to speed the analysis. These small elements result in increased computation requirements that will increase the overall solution time.

Material properties are defined for each model component. Bilinear material stress-strain characteristics are typically used to describe most structural members. Components that are nonstructural may be defined using a rigid material definition.

Lumped masses are added to the model to account for all non-structural masses, as well as powertrain components, fuel tank, spare tire, tools, etc.

The vehicle structure model is combined with the suspension and tire models to form the complete vehicle system model.

Tire Modeling and Development. Since the tire is the primary load transfer mechanism between the road surface and the vehicle, it is a vital component for performing VPG simulations. Efficient and accurate models of tires must be implemented that have the ability to accurately transmit the road profiles with the correct amplitude and frequency content.

The tire model used in this methodology was developed using published test data for the radial and lateral stiffness, as well as the dynamic behavior of the tire/wheel combination. This data was incorporated into the model, providing a tire stiffness that correlates to actual tire and empirical data [4].

The tire model is generated using a software utility developed for this specific purpose. The software tool utilizes tire geometry and inflation pressure to automatically generate a three-dimensional finite element model of the tire.

The sidewall of the tire is modeled using plate elements. The tread area of the tire is modeled using brick elements. A cross section of the tire model is shown in Figure X. This basic tire geometry is used to create all tires used in this method with appropriate scaling of the dimensions to meet the specific application. In order to simplify and reduce the overall size of the model, no plies of the tire are modeled and no detailed tread patterns are included. While this simplification may limit the application of the tire for handling and noise and vibration applications, this tire has been shown to provide a reasonable and practical means of transmitting loads from the road surface to the vehicle structure.

A Moody-Rivlin material constant is supplied for the rubber material.

Air pressure inside the tire is considered and modeled using an “airbag” modeling approach. The tire pressure is a constant and the tire is considered to have a constant volume.

Road Surface Development. The road surfaces used in the Virtual Proving Ground approach are three-dimensional finite element models of actual road surfaces. The road surfaces used to perform the VPG simulations are obtained from proving ground facilities and represent the significant durability components of the vehicle durability tests.

The road surfaces contain sufficient detail to describe the major vehicle inputs. Details such as localized cracks and road surface deterioration are not modeled.

The typical road surfaces that are found in vehicle durability schedules have been modeled and include:

Pothole Track
Alternate Roll Surface
Cobblestone Road
Body Twist
Ripple Tracks
Washboard Roads
Chatterstrips

The road surfaces have various intended purposes in vehicle development and when combined, form a full vehicle durability test schedule.

Figure 2 shows the road surfaces in use for a typical proving ground analysis.

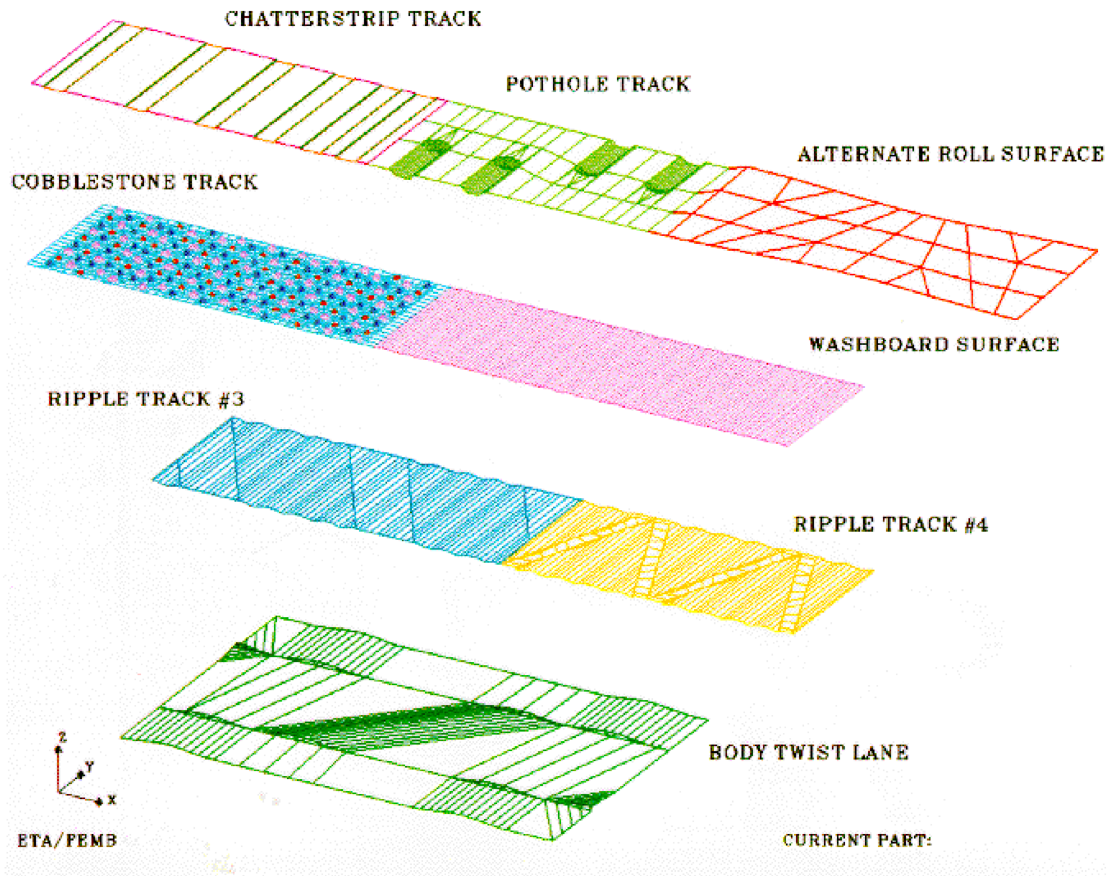

Figure 2. Road Surface Finite Element Models

Suspension Modeling. Virtual Proving Ground models may include suspension system models of varying complexity. Suspension systems may be modeled using either a rigid body, a flexible body approach or a combination of flexible and rigid bodies.

In the rigid body approach, the suspension members are modeled as rigid members attached via joints and spring elements.

The joints simulate the actual joints in the vehicle suspension, while the springs simulate the suspension bushings. Joints available in LS-DYNA for such applications are revolute, spherical, universal, cylindrical and translational joints.

This modeling approach is similar to those found in many multi-body dynamics programs and provides basic information about the force distribution in the suspension components and the load inputs into the vehicle body structure [5].

However, the rigid body approach does not account for component flexibility and does not allow for the direct calculation of component stresses.

The flexible body approach makes use of component finite element models, assembled using joints and spring elements.

The flexible body approach has the advantage of accounting for component flexibility and provides the ability for calculation of stresses during the analysis event. An additional advantage of inclusion of the flexible bodies is that it may provide the basis for a true full vehicle model, collecting real time stress data for each of the proving ground events. Inclusion of the flexibility has been shown to provide higher levels of correlation when compared to a rigid body modeling approach [7].

One disadvantage to the flexible body approach is the added level of complication to the model and the computational time required to solve the more complex representation of the components.

Material properties of the suspension components may be specified using the nonlinear stress-strain characteristics of the material. Alternately, the finite element models of the components may be used, but using a rigid material type. This provides an accurate mass and inertia representation of the component with adding significant computational requirements.

Suspension bushings, springs and dampers may be specified using nonlinear force-displacement characteristics. These specifications may be made using a load curve.

Local coordinate systems are included in the model to identify the initial positions of the joints and the orientations of the springs and dampers.

Analysis Approach. Once the full vehicle model is assembled, the analysis is carried out using a dynamic nonlinear finite element analysis approach. The commercially available program LS-DYNA is used as the general solver for these studies.

The LS-DYNA program has the ability to simulate component contacts, and allows the transmittal of forces between these components. This ability allows for the analysis of large displacement events, such as a vehicle driving over an uneven road surface, traveling at operational speeds.

The methodology used to analyze the proving ground events requires that the appropriate tire be added to the vehicle model.

A frictional contact is defined between the road surface and the tire to account for tire dynamic friction. For purposes of most analyses, a typical value is used. However, the frictional coefficient may be altered to simulate a specific driving condition.

The vehicle model is placed on the desired road surface model at the appropriate position. The vehicle tire and springs are then preloaded by allowing the vehicle model to reach a steady state under gravity loading.

Once the vehicle has reached an equilibrium position, a velocity is prescribed by applying an initial velocity to the model. This velocity represents the vehicle velocity prescribed in the physical proving ground test.

The analysis is then conducted, allowing the vehicle to traverse the road surface.

Analysis Results. Since the results of VPG simulations are in the time domain, this provides us with a unique opportunity to simultaneously evaluate stresses as well as the vibration of the structure. In order to make use of this feature, separate strategies must be used to post process the analysis results.

Durability Analysis. Since the primary evaluation tool of the durability analysis is the prediction of stresses and the subsequent fatigue analysis of stresses, the results of the VPG simulation may be in the form of real time stresses, strains and displacements at any desired point in the model.

Typical analysis outputs may be the body stresses with respect to time and model node displacements. These outputs are used to evaluate the vehicle structure, identifying the areas of interest or concern with respect to vehicle performance.

Analysis results may also be obtained to perform suspension parameter analysis. The results may be used to calculate parameters such as toe and camber angles during the event.

The stress results are used as input into a fatigue post processor, providing fatigue life predictions on an element-by-element basis.

The prediction of fatigue life is performed using a fatigue program developed to accept the stress output of the analysis solver.

The fatigue life program performs a rainflow counting of the stress data to identify the key damage events and stress amplitudes. This data is then used to calculate an element fatigue life.

Loading conditions can be combined and the individual cases counted multiple times to account for the number of loading cycles in the proving ground test procedure.

For example, the pothole road surface may be traversed 1,000 times during the proving ground durability schedule. The peak stress occurrence would then be counted 1,000 times in the cumulative fatigue life calculation.

This combination and multiplication of the counted peak stress amplitudes are used to develop an overall view of the vehicle structure or suspension components.

Vibration Analysis. Automotive vibration analysis, commonly known as NVH analysis, evaluates the response of various components and systems. This is used to provide sufficient isolation between inputs and vibrational modes of the systems.

Since the results of VPG analyses are in the time domain, these real-time responses of the structure may easily be viewed using animations and deformed shape plots of the structure. However, these have limited value due to the potential to excite many of the structure's modes of vibration simultaneously.

NVH analysis of the VPG simulation may be made by monitoring the displacement, velocity or acceleration of points of interest in the model. In addition, signal processing, such as Fast Fourier Transform, may be used to convert time domain results to the frequency domain, identifying the frequencies at which the highest amplitudes occur. From this data, operating mode shapes of the structure may be reconstructed.

Significant time saving may be gained through this feature, since it allows for the evaluation of durability and NVH using a common model. It also allows for the prediction of NVH characteristics using the road surface model as an accurate input and provides a means for performing NVH analyses using a nonlinear approach.

DISCUSSION

Application Examples
Four application examples, a durability analysis, a vibration study, cork screw rollover analysis and a tire cleat test will be presented as a demonstration of the various types of simulations that can be carried out using the VPG analysis technique. These simulations are typical applications of this method.

Durability and Fatigue Analysis
As an example of an application of this methodology, a compact front wheel drive sedan was studied. The vehicle is intended to be marketed worldwide with focus on U.S. and European markets.

The durability analysis was to be conducted to identify the areas of concern for the proving ground durability analysis. The Virtual Proving Ground concept was chosen to perform this analysis, prior to any prototype testing was to take place.

The Body-in-White (BIW) finite element model was created from CAD surface data, using a typical element size of 25 millimeters on a side. Plate elements were used for each of the sheet metal components of the body structure. Figure 3 shows the BIW model created for use in this example.

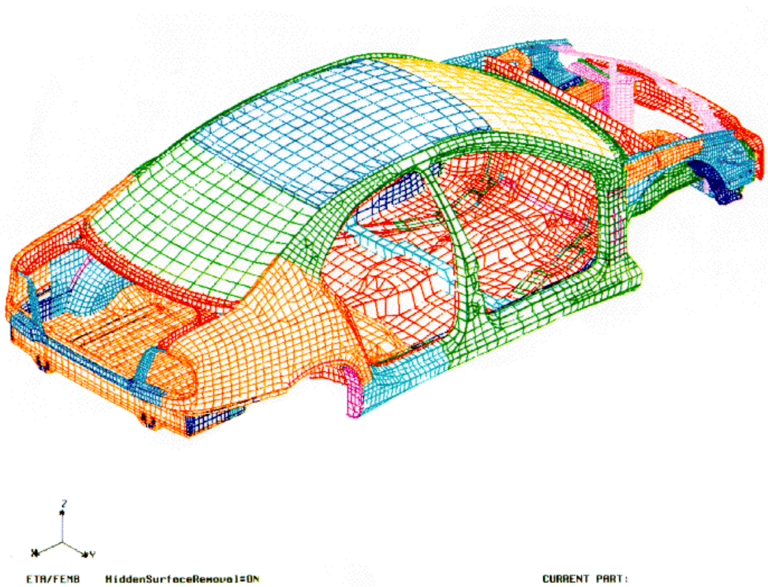

Figure 3. Body-in-White Finite Element Model

The suspension components were modeled using a coarse mesh. A typical element size of 25 millimeters was used as in the body structure model. However, in the suspension components, many of the design features must be neglected.

Figure 4 shows the finite element models of the front and rear suspension systems.

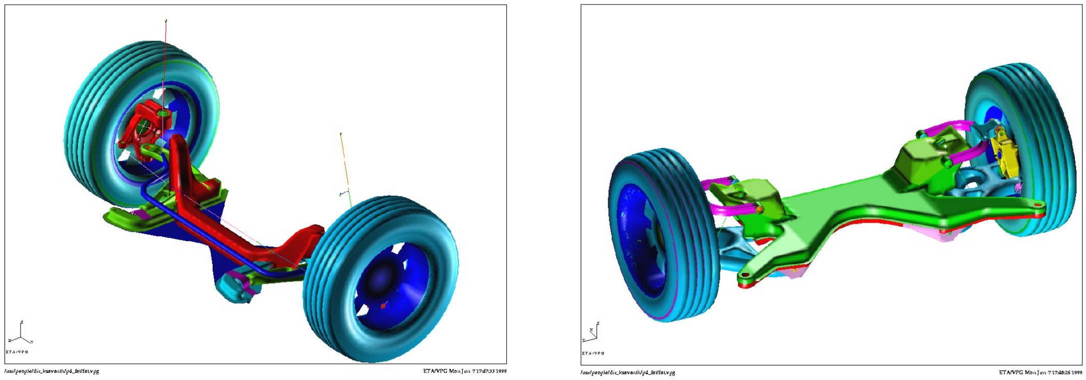

Figure 4. Front and Rear Suspension Models

Modeling considerations for the Virtual Proving Ground study were primarily the elimination of small elements, which may result in a longer analysis time. This must be balanced with the need for accurate analysis results.

To meet this requirement, the model was meshed finer in the areas of anticipated high stress. These areas were the shock tower and suspension mounting areas, as well as the cowl and hinge pillar areas.

Trimmed body masses were added to the assembled body/chassis model. These masses represent the major nonstructural masses in the vehicle. Typically, masses over 2 kilograms are modeled with the remaining masses accounted for using an even distribution of lumped masses.

Joints were defined between all suspension members with nonlinear spring rates defined for the bushings in six degrees of freedom.

The struts were modeled using a spring and damper combination with nonlinear curves supplied for the damping characteristics and the spring constant.

Tire models were added to the suspension system. A P215/60R15 tire model was included and attached to the hubs via revolute joints.

The final model contained 49,830 plate elements, 13,635 solid elements, 18 joints, and 92 springs.

Analysis Results. An analysis condition simulating a proving ground road surface was performed. The road chosen for this analysis was a pothole condition. The analysis is performed in a manner that reduces the overall computational requirements. This was accomplished by identifying repeating sections of the road surfaces, and then limiting the analysis to that section. In short, the repeating sections of the actual road surface are modeled and used in the simulation.

The analysis results came in the form of real time stresses and deflections of the vehicle structure and suspension components. These stresses vary with respect to time, reflecting the varying forces in the vehicle during the proving ground event.

Typical stress results for one time are shown in Figure 5. This figure shows the results of the pothole track at 0.7 seconds into the event simulation.

Forces, velocities, and accelerations can be monitored at any node location in the vehicle and plotted vs. time for the entire event.

These results allow the analyst to view the behavior of the vehicle system over the entire event.

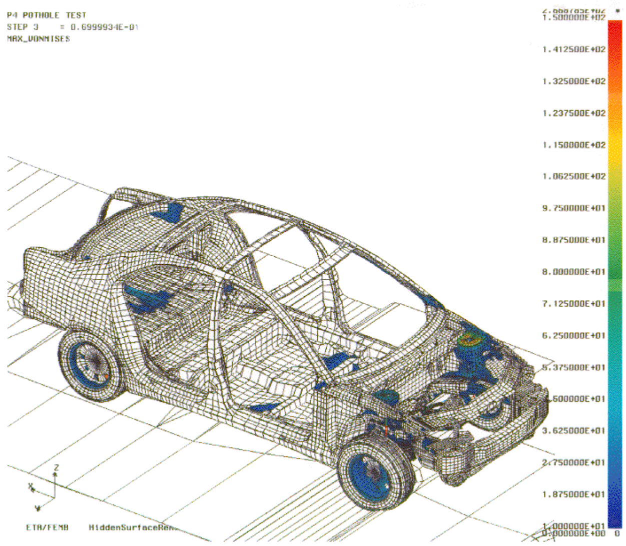

Figure 5. Pothole Track Stress Results at 0.7 seconds

Fatigue Life Analysis. The 20,000 Mile General Durability Schedule used in this study has a known content of road surfaces and a specific number of cycles of each road surface. From this schedule, a number of cycles for each analysis road surface can be determined.

The analysis results for each of road surfaces is saved in a binary file format. These results are input into a fatigue life analysis program, along with the values for each of the road surface repeating sections.

The results of the fatigue life analysis are in the form of tabular data for each element. This data can then be read by a finite element post processor and plotted in a contour plot format. Figure 6 shows the fatigue life contour plot for the example analysis. The lowest areas of fatigue life are identified using this technique.

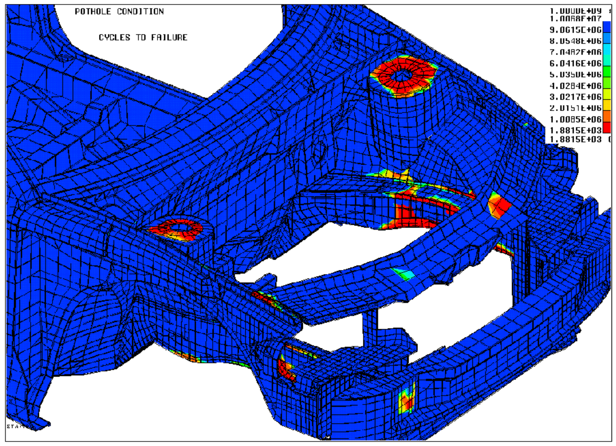

Figure 6: Fatigue Analysis Results for Pothole Condition

Vibration Analysis
As an example of a full vehicle vibration study using the VPG method, a vehicle traveling over a rough, uneven road surface will be studied. The vehicle under study is a small, European/Asian market mini-car with a vehicle weight of 1280 kg.

The vehicle structure was modeled using shell elements with a uniform mesh distribution. Figure 7 shows the vehicle structure model used in this study.

The vehicle suspension was modeled using beam elements to represent the suspension components. These beam elements are comprised of both rigid and flexible materials.

Response nodes are identified prior to the analysis to extract the necessary accelerations for subsequent signal processing. This is usually a small set of nodes that are sufficient to describe the mode shapes. These nodes are then connected using display elements so that the user may easily discern the deformed shape of the vehicle structure. These display elements are typically referred to as “plotels” or plot elements.

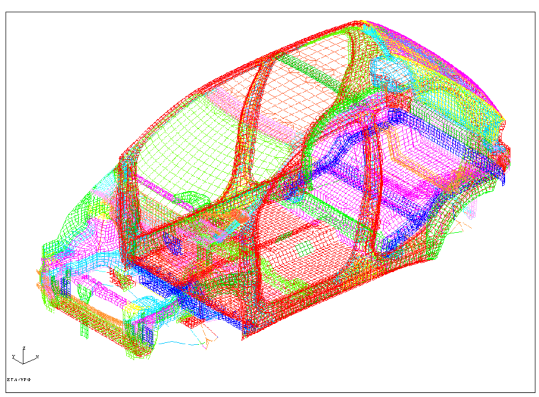

Figure 7: Vehicle Structure and Suspension System for NVH Analysis

The display model used for this study is shown in Figure 8.

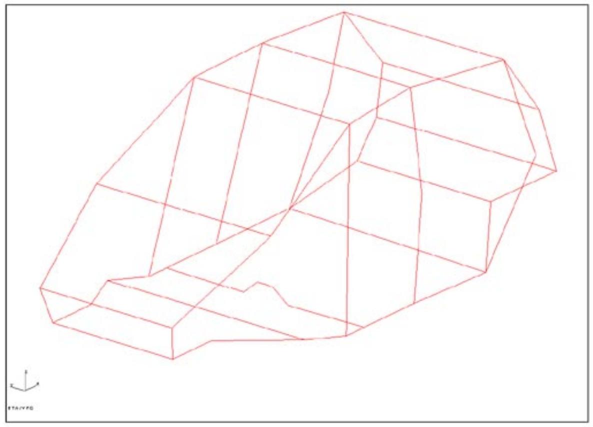

Figure 8: Display Model for NVH Analysis

The vehicle model is placed on a rough road surface and a vehicle speed of 30 kph is prescribed. The vehicle is allowed to travel over the road for a time of 2 seconds. During this event, the accelerations of the nodes are obtained in 3 dimensions. This data is shown in Figure 9.

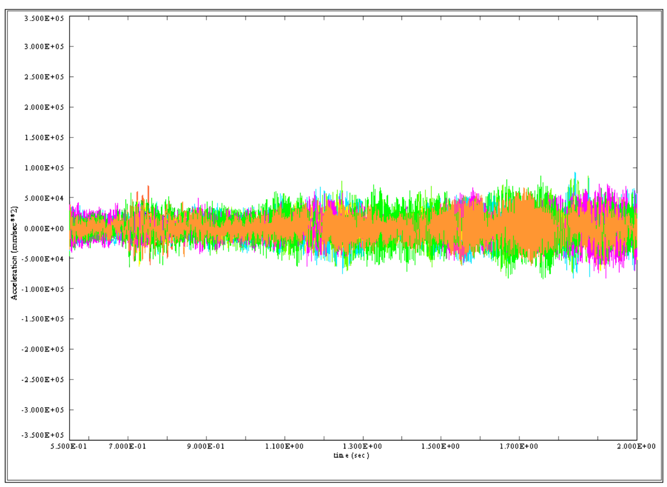

Figure 9: X,Y,Z Acceleration vs. Time for Belgian Block Road

The Operating Deformed Shape (ODS) is then reconstructed from the amplitude and phase angle information extracted from the simulation. This is performed using a Discrete Fourier Transform technique.

The ODS is examined to identify typical modes shapes of the vehicle, such as bending and torsion modes. The PSD vs. frequency curve (Figure 10) is used to help identify peak amplitude frequencies. The ODS at each of the dominant frequencies is noted from the curve and then animated using the eta/VPG software post processing features.

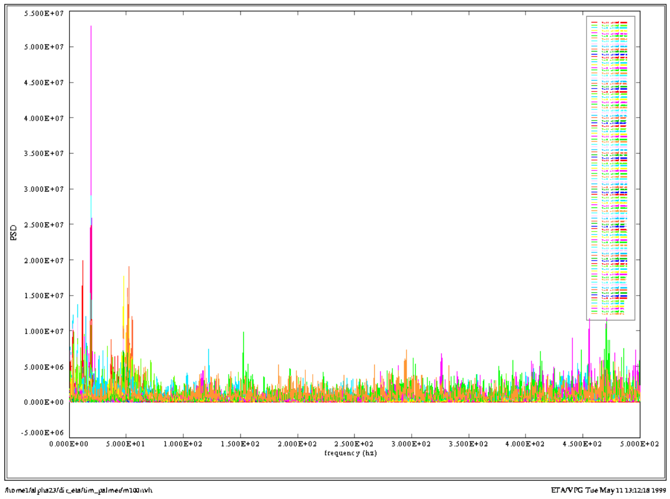

Figure 10: PSD vs. Frequency for Belgian Block Road

The results of the case study vehicle are shown in Figures 11 through 15. These results are the mode shapes for the vehicle, as excited by the road surface. The modes displayed were chosen arbitrarily, to demonstrate the ability of the simulation to predict typical modes, as well as those occurring at frequencies higher than that usually considered in linear dynamic finite element analysis.

These results may be used to identify the response frequencies of the vehicle under this specific loading. Additional points in the structure may be monitored to identify key driver/vehicle interface locations such as the steering column response and the seat mounting locations.

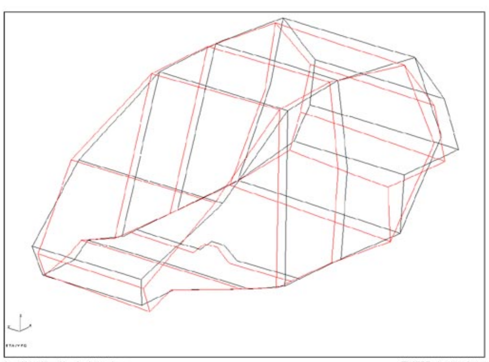

Figure 11: Front Lateral Bending , Vehicle Pitch @ 10.2 Hz

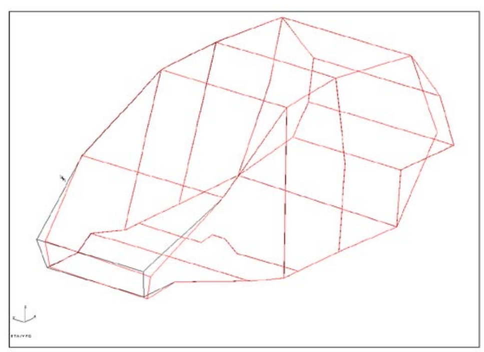

Figure 12: Front End Lateral Bending @ 19Hz

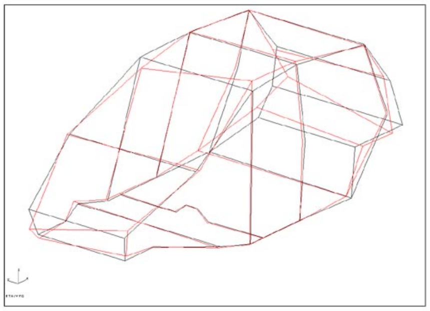

Figure 13: Full Torsion @ 35 Hz

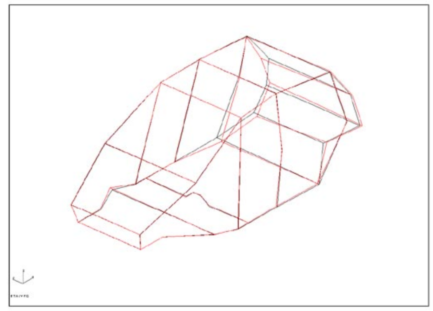

Figure 14: Rear Lateral Bending @ 51 Hz

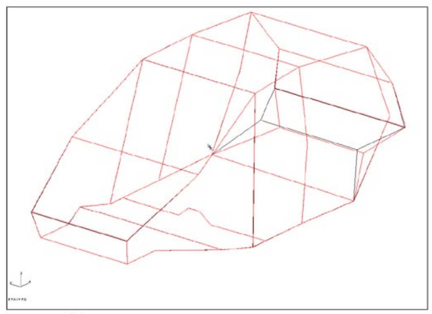

Figure 15: Rear Lateral Bending at 152 Hz.

Cork Screw Rollover Analysis

A Sport Utility Vehicle (SUV) passenger car has been studied to examine the effectiveness of using the VPG method to evaluate the vehicle stability of rollover, dynamic performance of the suspensions, estimating the body damages due to the collision with ground and tackling the many difficulties associated with this problem.

It is widely understood that the SUV have almost double the rollover risk of other types of vehicles. The rollover crashes are one of the most significant safety problems and it is more serious for light trucks such as SUV. The design characteristics which affect the rollover propensity are static stability coefficient, suspension compliance, tire sidewall stiffness, etc. The SUV designed initially for off road use. The vehicle sits high off the ground and has softer suspension system than most cars.

In this analysis, the SUV vehicle passenger car travels over a 23 degrees inclined ramp to the horizontal ground as shown in Figure 16. The vehicle forward velocity is 44 mph (20 m/sec.). The overall length of the ramp is about 3.89 meters. In this analysis, the vehicle first travels on a flat road to reach its steady state ride condition and then the right hand side of the vehicle travels on tripped off-road (a 23 degrees ramp) rollover to simulate the cork screw rollover.

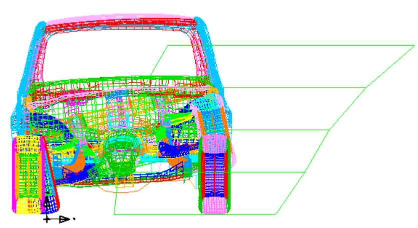

Figure 16. Rear View of SUV

The resulting animation is shown in Figure 17a - e at different times as the vehicle travels over the ramp.

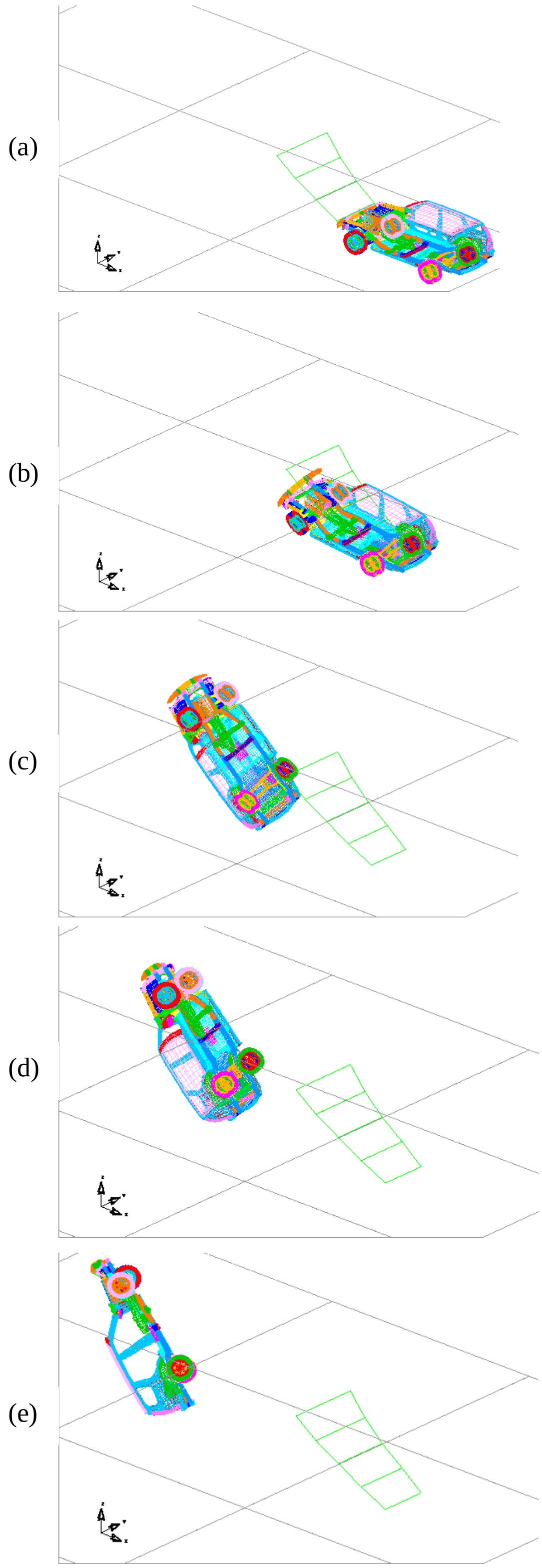

Figure 17. Resulting Animation of Rollover Analysis

Due to the existence of a huge angle of the ramp, the vehicle passenger front suspension is suppressed extremely and rotates upwards around the lower control arm joints. The rear tire of driver side is almost flat since the vehicle load transfers from front to rear left-hand side and the normal loads of two passenger side tires are kept increasing while travelling over the ramp as long as they contact with the ramp surface. These normal loads raise the vehicle CG center and together with vehicle forward velocity will cause sufficient longitudinal rolling velocity to build up and result in the rollover of the vehicle finally.

Figures 17d and 17e illustrate that the vehicle rolls and twists in the air and lands on the road with the rear bumper hit the ground first, which absorbs a lot of kinematic energy and hence increasing the occupant safety.

The analysis results very clearly demonstrate the feasibility of VPG application and its effectiveness to assess the suspension components. Furthermore, the method provides adequate data for evaluation of the body crash with ground. This method may be extended to evaluate the occupant safety during the rollover. Finally, this method is capable to carry out all these assessments in simulation.

Tire Cleat Test

Using the VPG method, the tire cleat test of a rolling tire has been numerically simulated to study the transient responses of the rolling tire impacting a cleat.

The tire in this study is P215/60R15, as shown in Figure 18. The tire is a three-dimensional finite element model consisting of solid and shell elements. The tire mass is 12.56 kg and the pressure is 35 psi. The rubber material properties of the tire were defined as a Moony-Ravilin hyperelastic material.

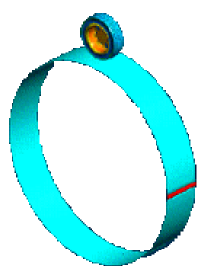

Figure 18. Tire Cleat Test

Initially, the tire was tuned to ensure the vertical, lateral and longitudinal stiffness of the tire model corresponds to that of the test

On completion of the stiffness validation, the simulation of the tire rolling on a drum (with 3 m diameter) that contains a cleat (76.2 mm long and 19.1 mm height) were conducted using Explicit Three-dimensional Nonlinear Dynamic Finite Element Code LS-DYNA.

The simulation includes:

- A revolute joint with fixed axle to roll the tire along the periphery of the drum.
- A pre-load of 4500 Newtons of the tire against the drum.
- The progressive rotation of the drum until the tire model reaches the steady state rotational speed corresponding to 15 mph.
- Finally, impacting of the cleat on the tire.

The computed vertical and longitudinal forces at the joint extracted from the simulation were studied and compared against the measured test data as shown in Figures 20 and 21.

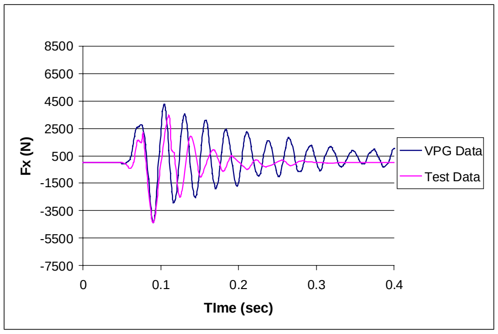

Figure 20. Time vs. Force of X

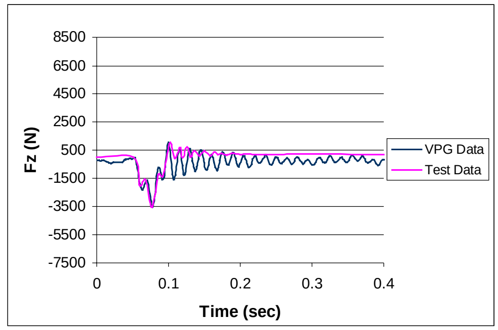

Figure 21. Time vs. Force of Z

Comparing the VPG simulation results and the test data, there is a clear indication that the tire cleat test simulation using the VPG approach is feasible and may offer the tire manufacture a simulation tool, in addition to the traditional test approach.

CONCLUSIONS

A finite element based analysis tool has been developed to provide a greater understanding of the full vehicle system under simulated proving ground road inputs. The methodology can be applied to full vehicle systems to identify areas of high stress or reduced fatigue life and can be applied to vehicle vibration analyses.

This type of simulation may make use of different types of road inputs can be effectively simulated, allowing the analyst to study the entire event.

Analysis results can be evaluated on an individual basis or combined and analyzed using a fatigue life analysis program.

The method presented here demonstrates that one model could be used for various functions that are typically completed using several different analyses. This method also allows the engineer to study the complete behavior of the vehicle, considering the nonlinear dynamic behavior, more accurately simulating the real world conditions of the vehicle usage.

ACKNOWLEDGEMENTS

The authors would like to acknowledge Daewoo Motors and specifically David Wheeler of the Worthing Technical Centre for the permission to publish the results of the case study. The authors would also like to thank Tim Palmer, formerly of Engineering Technology Associates, for his work in preparing this paper.

REFERENCES

1. KUO, E. Y. and KELKAR, S.G. (1995). "Body Structure Durability Analysis." Automotive Engineering, July pp. 73-77.

2. PINTAR, S. and MCGUAN, S. (1992). "Flexible Vehicle Simulation or Modeling Vehicle Suspension Compliance at Ford Motor Co. using ADAMS/FEA", Proceedings of the 1992 International ADAMS User's Conference.

3. ASLANI F., YATHEENDAR, M., VISANTAINER, R., ROHWEDER, D., and deALDA, J.L. (1994). "Simulation of Proving Ground Events for Heavy Truck Cabs Using ADAMS, MSC/NASTRAN, and P/Fatigue." MSC 1994 World Users' Conference Proceedings.

4. LEE, C.R., KIM, J. W., HALLQUIST, J. O., ZHANG, Y., and FARAHANI, A. (1997). "Validation of a FEA Tire Model for Vehicle Dynamic Analysis and Full Vehicle Real Time Proving Ground Simulations." SAE Paper Number 971100.

5. TANG, A., FARAHANI, A., PALMER, T., and BAOHUA, X. (1995). "Kinematic and Dynamic Non-Linear Finite Element Analysis of a Multi-Body Linkage System using LS-DYNA3D", 3rd LS-DYNA3D World User's Conference.

6. ZHANG, Y. and TANG, A. (1998). "New Approach for Vehicle System NVH Analysis – Part I: Concept and Vibration Analysis." SAE Paper 981668.

7. ZHANG, T., XIAO, P., PALMER, T., and FARAHANI, A. (1998). "Vehicle Chassis/Suspension Dynamics Analysis – Finite Element Model vs. Rigid Bodel Model" SAE Paper 980900.

8. VISANTAINER, R.H., WATTS, D. E., DOHADWALA, S., LEE, P. , SHAHDI, B. K., ASLANI, F., CHUANG, C. H., and HUANG, J. (1997). The 42nd L. Ray Buckendale Lecture "CAE Methods and Their Application to Vehicle Design." SAE Paper 973187, pp96-104.

9. KOA, B. and KAMOULAKOS, A. (1998) "Transient Dynamics of a Tire Rolling over Small Obstacles – A Finite Element Approach with Pam-Shock." Tire Science and Technology, TSTCA, Vol. 26, No. 2, April-June, pp. 84-108.