13 Th European ADAMS User's Conference, November 18-19, 1998

Study of Component Flexibility Effects on Full-system Performance via Mixed Body Dynamics Approach

F. Palčák,M. Hulla.

ABSTRACT

In this paper the description of a new CAE method application for tight interaction between computer aided design (CAD), component analysis (FEA) and system simulation (MSS) technology is presented. Investigation of the full vehicle Virtual Prototype (VP) of light truck vehicle is the case when designers must not neglect the frame flexibility due to its significant impact on overall vehicle performance during ride maneuvers. The possibility of two-way model data exchange served to transfer model coordinates, element connectivity, stiffness, damping and mass matrix from ANSYS to the ADAMS environment after analysis of the superelement. The resulting flexible body derived from its modal representation was incorporated into a full vehicle model and exhibited correct coupling between overall motion and small elastic vibrations. The applied loads, joint reactions, acceleration, and velocity - dependent loads were computed and saved at each point in the simulation. Obtained results, along with corresponding constraints, were merged with model data in the FEA environment for calculation of nodal loads and subsequent component analysis.

PREFACE

Engineering teams greatly reduced their reliance on costly physical prototypes despite of investmens can be considerable. Well-established companies have proven that the payoff in the form of time savings, cost savings, and quality improvements can more than offset expenditures when the technology is applied effectively.

An MSS simulation of mechanism containing only rigid bodies can provide useful loads data for component designers. But that may be just part of the complete picture, as component flexibility can have a significant impact on load distribution. It is without doubt, that the inclusion of complex flexible structures in the full vehicle VP can have significant impact on its behavior. Co-simulation of rigid and flexible bodies has advantages for design on the system level, namely concurrent engineering concepts, provides new advances in finite element mesh generation.

The goal of our full vehicle VP analyses was to determine dynamic loads acting by wheel suspensions, steering mechanism and further subsystems in the attachments on the vehicle frame. For purpose of comparison of rigid-body and flexible VP analyses it was necessary to develop suitable models in MSS environment.

RIGID BODY VIRTUAL PROTOTYPE.

The study of rigid-body model basic modal properties preceded assessing of the more complex VP containing flexible frame model. VP was build up from logical independent subsystems like wheel suspensions with the viscous dampers, tires, the pneumatic springs with rubber bumpstops bordering the jounce and rebound, steering mechanism and truck box.

Strong nonlinear springs and viscous dampers characteristics were experimentally measured. MSS technology do not lay claims to be an excellent 3D modeller so the inertial properties of each individual part was necessary to obtain from I-DEAS CAD software.

The subsystems of the full vehicle VP were separately investigated before assembly. After verification of steering mechanism functionality, this link mechanism was joined on the frame and spindle, then prescribed motion was applied on it. For dynamic ride analyses the steering motion controller was acting on the input steering lever to follow specified track shape. Spindles of the rear wheels were constrained by links to the frame with purpose to fix their swings relative to the vehicle body in ride direction.

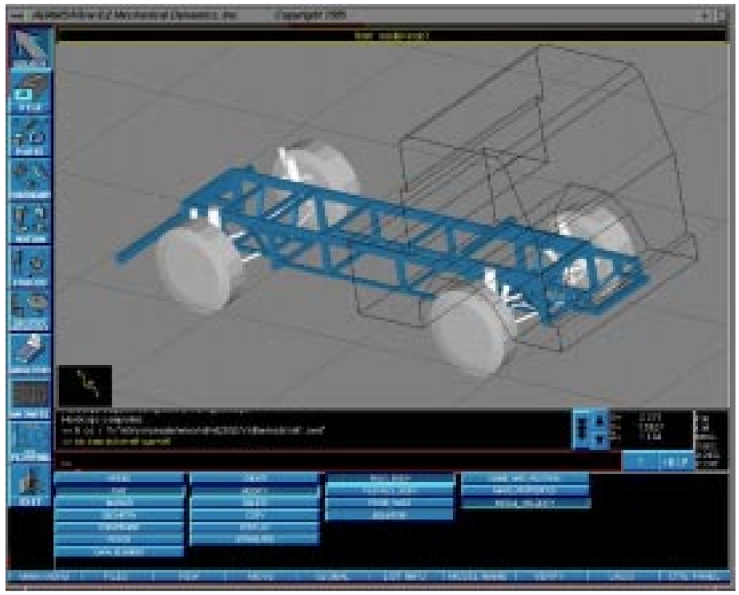

Fig.1 View of the complex virtual prototype of the vehicle.

The frequency properties of the silentblocks joining the independent suspension onto the frame were verified by modal analysis. The full loaded truck box was in VP represented by one rigid block joined to the frame by six flexible bushings. The drive line was not considered because of its non-significant effect on loading of the observed flexible frame.

The tire-road contact was modeled by the combination of the internal ADAMS/Tire module element and user written IMPACT function. This description of tire/road interaction is based on the slip characteristics of the tires, which are sources to compute tire contact forces.

The kinematic and dynamic behavior of subsystems was assessed as sufficient for this light truck vehicle so the assembly process of these structural components with verified properties to one complex vehicle VP was final development stage .

VIRTUAL PROTOTYPE OF THE VEHICLE INCLUDING FLEXIBLE FRAME.

As second stage it was necessary to consider flexibility of the welded thin-walled tube frame. Initial VP was created by ADAMS beam elements via Timoshenko's theory. During estimate analyses we recognise that such complex beam VP is non-effective.

More effective was to include the flexibility of the frame via the modal expansion approach integrating the MSS Technology and Finite Element Method (FEM). The FEM model of the vehicle frame was developed in the I-DEAS/FEM preprocessor environment using 8 nodes shell elements. The lumped-mass method used in eigenmodes extraction enabled desired accuracy of mass distribution in flexible VP. Extraction of the frame modal properties was performed on the FEM frame model imported to the ANSYS solver environment. We extracted its 20 eigenfrequencies.

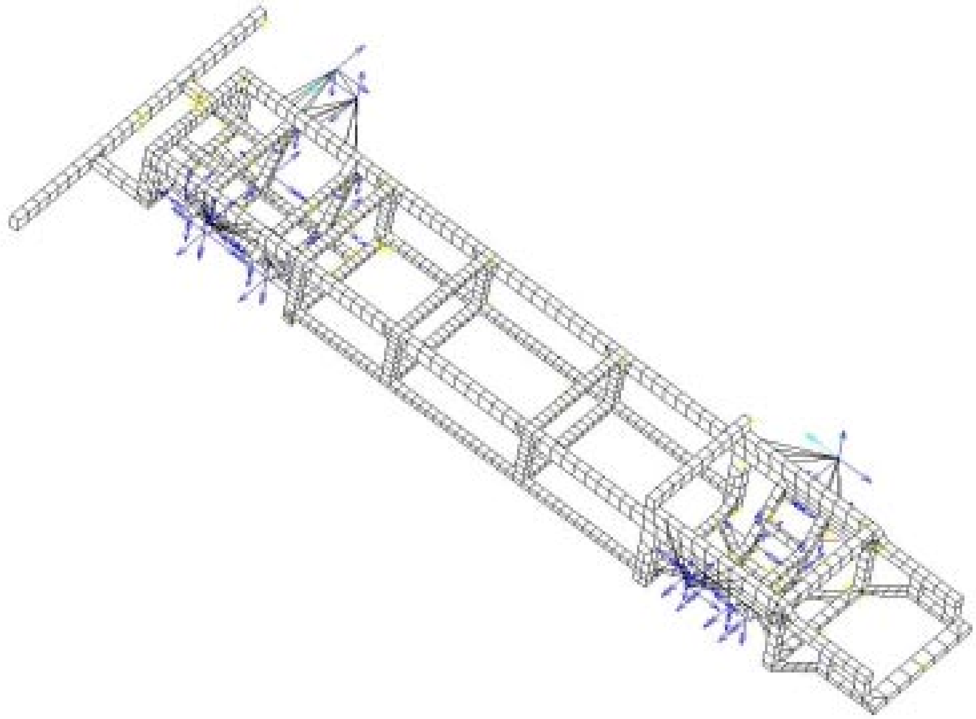

Fig.2 View of I-DEAS FEM model of flexible frame.

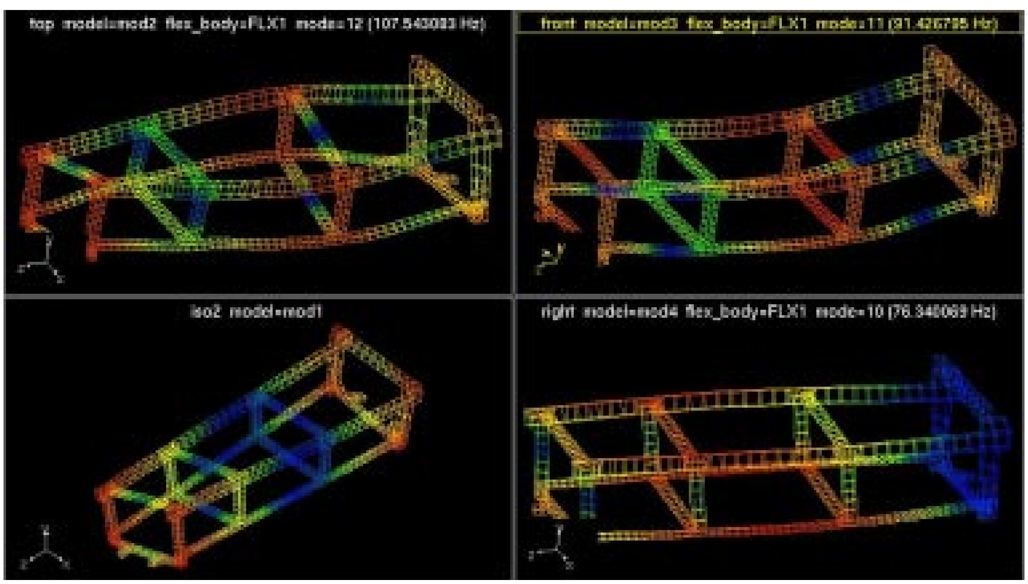

Fig.3 Some significant eigenmodes of considered part of flexible frame in simulation

The frame modal representation was imported to the ADAMS vehicle VP. Such a transferred flexible body behaves in ADAMS as one individual part with flexible properties corresponding to the FEM properties. The steering, suspensions, tire, truck box and other subsystems were joined with flexible frame via fastening elements and kinematic constraints.

DYNAMIC LOADS FOR VEHICLE FRAME FEM STRESS VALIDATION

Like the real physical prototype the VP had to undergo a series of standard tests which should have the most expressive loading influence on the investigated vehicle frame.
For intended evaluation and comparison of dynamic loading static analyses were realized and determined the dynamic coefficients for both models with rigid and compliant frame.

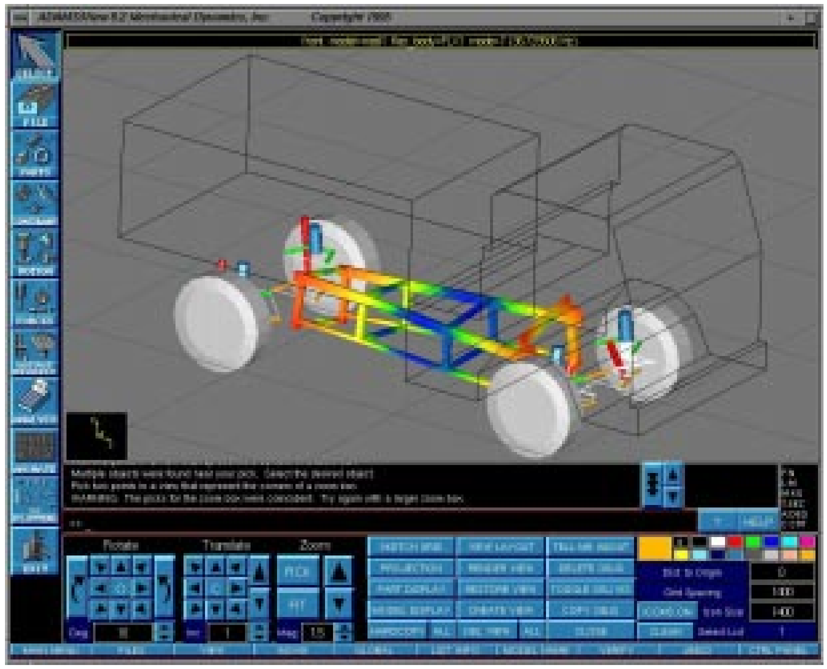

Fig.4 View of the complex virtual prototype with flexible frame.

The most important vehicle parameters in test procedure for transient yaw response are lateral acceleration, roll angle, steering-wheel angle and yaw velocity as a function of time after the steering-wheel angle input is made by the driver. During the lane change maneuver simulated at velocity 70 km/h, oversteering of vehicle was observed. This property was caused also by the mass center shifting to the rear, because load on the vehicle box was taken into the consideration. The dynamic loads on the frame are not so significant in this maneuver.

The next test maneuver consist of driving over the 150mm high ledge, which produce short force shock on the tires which are converted in the wheel suspension system and finally to the frame. Simulations showed that rear axle of vehicle model with compliant frame is more dynamically loaded like rigid frame was considered. On the Fig.5 there is plotted comparison of front wheel bearing dynamic loads generated by flexible (dashed curve) and rigid (solid curve)VP.

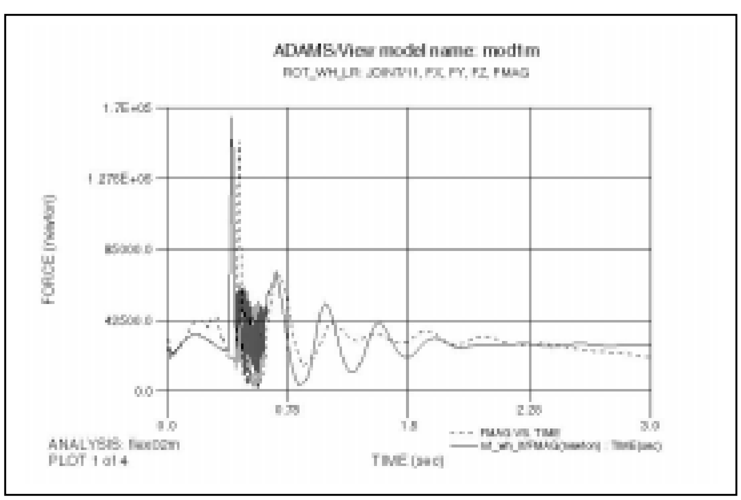

Fig.5 The front wheel dynamic loads courses during passing obstacle.

The intensive braking test procedure with applied maximum normal forces confirmed the expected result that it generated the most representative forces from front suspensions to the front section of the vehicle frame.

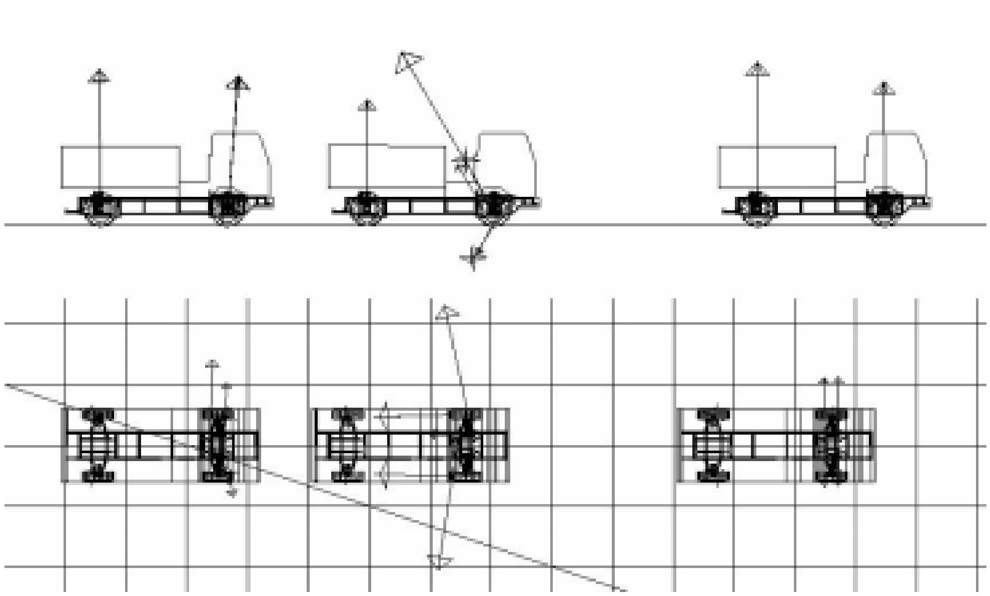

Fig.6 Starting, maximum breaking forces and ending phases of braking.

The comparison between simulation and measurement can bring further refinement of VP. A more significant aspect, however, is the fact that VP reflect the principal interactions of the vehicle components involved, which allow us to identify their individual contributions and to determine their influence.

VEHICLE FRAME FEM STRESS VALIDATION

After we obtain dynamic loads via simulations of several manoeuvres of mixed rigidbody-flexible virtual prototype, the stress validation was performed. We checked for adverse vehicle configuration at the discrete moment of the simulation.We focused critical locality in frame construction and then, the arrangements for construction changes we designed.

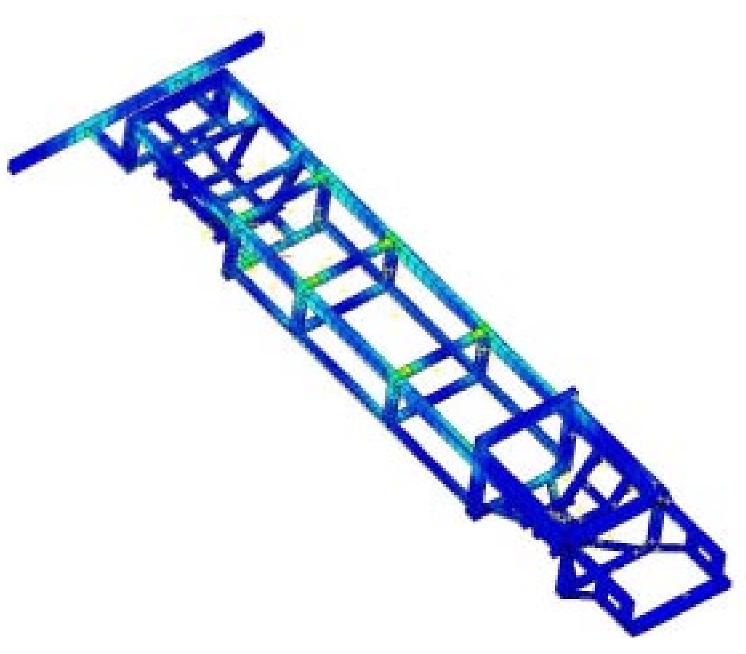

Fig.7 Stress validation of the I-DEAS FEM model of the complete frame.

CONCLUSIONS

During development of VP the new method of compliant body replacement by its modal representation was used. The significant vehicle ride regimes were simulated and the influence of flexible frame to the global behavior from vehicle dynamics point of view was investigated.

The Craig-Bampton component mode synthesis (CMS) with modeshape orthonormalization and multiple reference frames contribute to the accuracy and reliability of analysis results and to very realistic dynamic behavior of the model. It helps assure that interaction between the component and the complete mechanical system is accurately modeled. An assembly of modal flexible bodies captures rich dynamic behavior with few degrees of freedom. Time-variable modal damping on a mode-by-mode basis reduces numerical effort by keeping inactive high-frequency response from adversely affecting solutions. Correct make-up of the mode shape basis tailors the body for use in a particular frequency range.

A rigid frame approximation leads to the considerable error in estimating lateral acceleration and vehicle with a rigid frame follows a smaller-radius turn. Including a flexible frame in this design dramatically changed the dynamics of the full vehicle. It was confirmed that imported flexible structure caused the 16% decrease of dynamical loads.

The main goal of our investigation was to determine the effects of the vehicle frame flexibility on its dynamic loads. Computed dynamic loads of the vehicle frame were used as FEM boundary conditions. Dynamic loads was used for vehicle frame stress validation .

This paper show that coupling CAD-FEM-MSS technology helps designer asses the effects of flexible components on full system performance, improve the accuracy of simulations and such bring him closer to the system-level design. Integrating system-level motion simulation and component-level FEA improve the reliability of load prediction by supplying FEA model with complete time histories of component deformations. Providing this data to designer can give added confidence in subsequent analyses of stress, fatigue and other desired component variables.