56-39
187837
p. 14

N94-19471

Numerical Simulation of Vehicle Crashworthiness and Occupant Protection

Nripen K. Saha
Ford Motor Company
Dearborn, MI

PRECEDING PAGE BLANK NOT FILMED

VEHICLE CRASHWORTHINESS DESIGN

Objectives

To design the vehicle structure for optimum impact energy absorption, and to design the restraint system (seatbelts, airbags, bolsters, etc.) for optimum occupant protection.

Approach

A major part of the impact energy is to be absorbed by the vehicle structure. The restraint components will provide protection against the remaining crash energy.

Certain vehicle components are designed to deform under specific types and speeds of impact in a desired mode for sound energy management.

Structural components such as front side rails, rear rails, door structure and pillars undergo large amounts of deformation.

With properly designed geometry and material these components assist in mitigating the effects of impact.

Role of CAE

CAE is playing a vital role in crashworthiness design of automotives.

Instead of being dependent on numerous prototypes, the design process is rapidly becoming CAE guided.

This new approach allows engineers to examine many alternatives in a shorter time period.

Also, CAE models are complimenting prototype tests.

Models are used to predict crash performance of:

Components
Subassembly, and
Full-vehicle systems.

CAE Tools for Crashworthiness Analysis

Four types of CAE models are generally used in the industry. These are:

- Concept models (lumped masses and springs)
- Occupant simulation models (rigid body type)
- Hybrid models (concept and FEA combined with test data)
- Detailed finite element models.

Use of these models depends on the stage of vehicle development and accuracy of results desired.

CPU usage and modeling turnaround time is also a key factor.

Finite Element Models

Currently at Ford Motor Company nonlinear finite element models are used to simulate:

Front Crash
- Frontal perpendicular barrier impact
- Frontal angular barrier impact
- Frontal pole impact

Side Impact
- Side impact simulation

Rear Impact
- Vehicle rear-end impact with moving rigid barrier
- Car-to-car rear offset impact

Roof Crush
- Quasi-static roof crush of vehicle structure

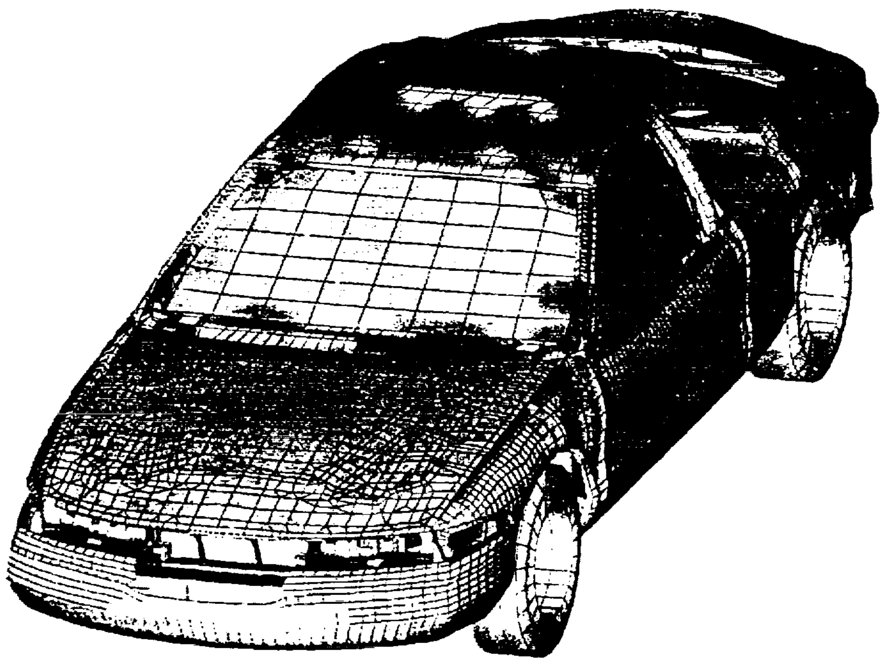
Frontal Crash Models

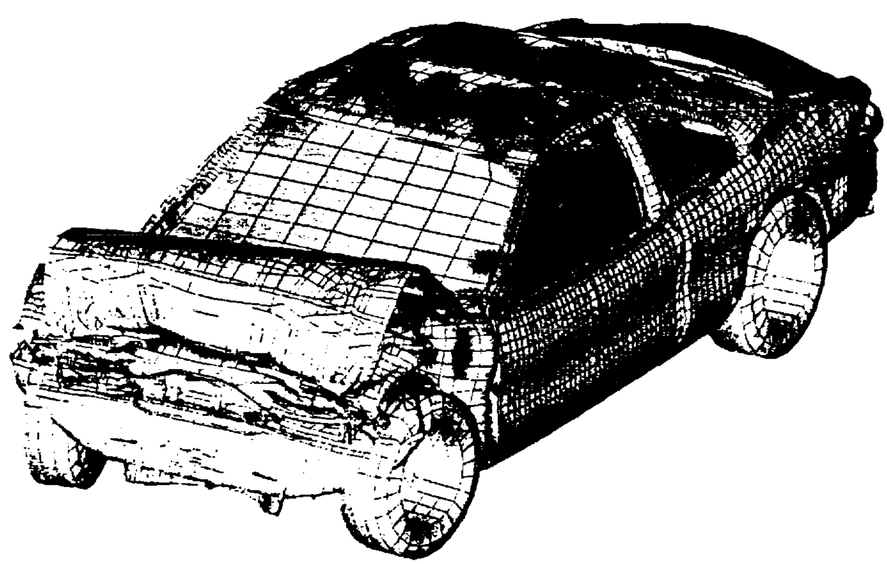
Applications:
- Front-end structural design of the vehicle
- Provide design directions for airbag sensor development
- Provide input for occupant simulation models

FEA Model Output:
- Total vehicle and component collapse pattern
- Engine compartment energy absorption
- Front bumper and side rail load-displacement
- Engine block movement and dash wall intrusion
- Deceleration pulse in passenger compartment
- Displacement and velocity histories at critical locations

Frontal Crash FEA Model

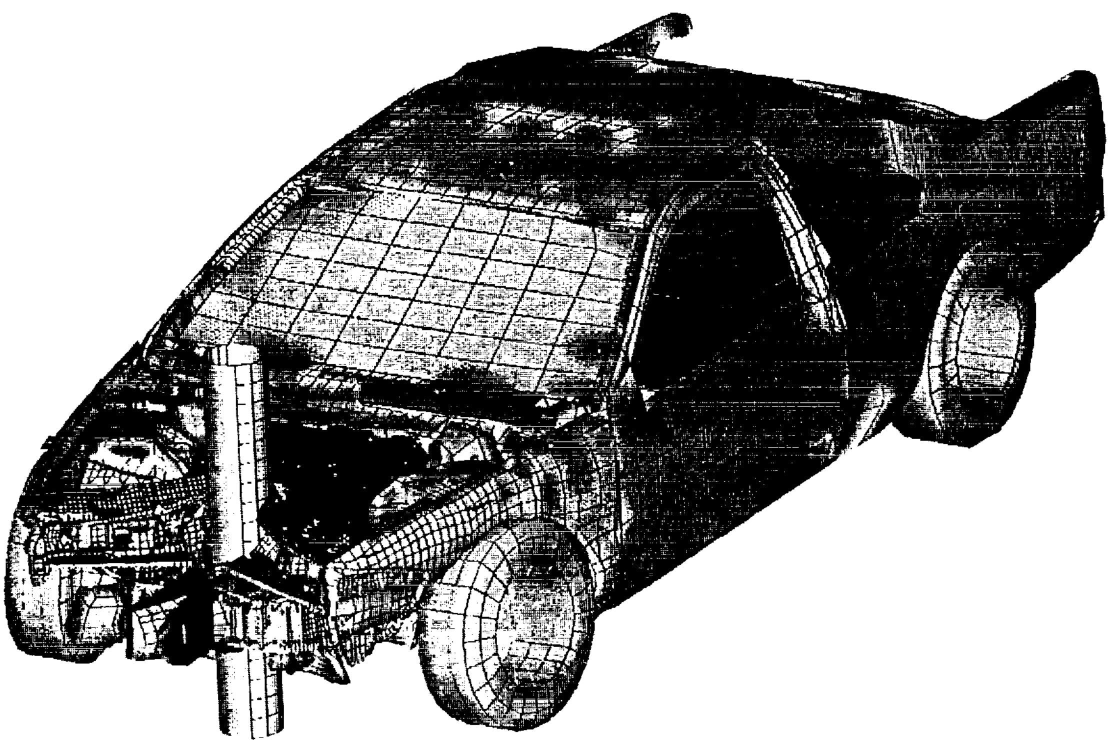
The original and deformed geometry of a frontal barrier impact model is shown below. Typically, model size is approximately 25000 elastic plastic shell and solid elements. Computer runtime for 100 milliseconds of crush is about 20 hours on CRAY YMP. The analysis is performed with a commercially available code, RADIOSS, from Mecalog, France.



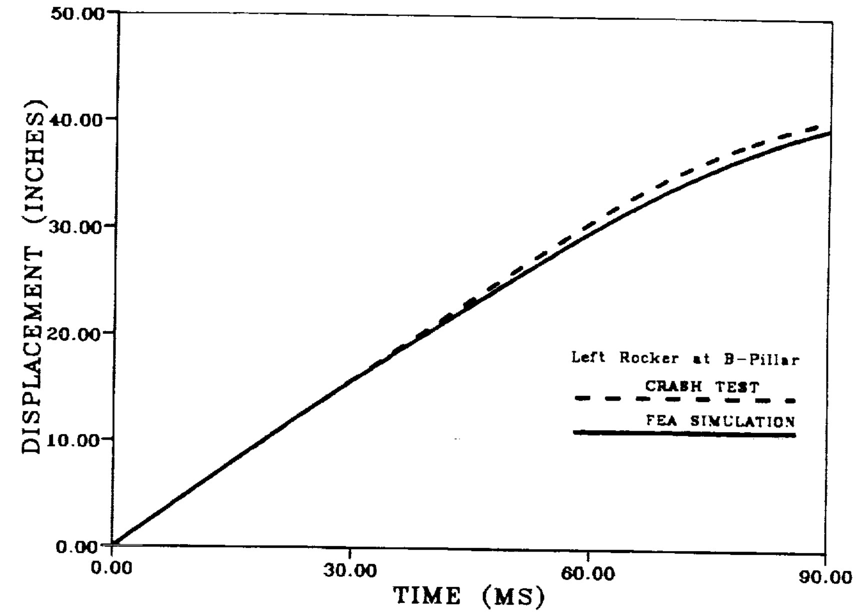


Frontal Pole Impact Model

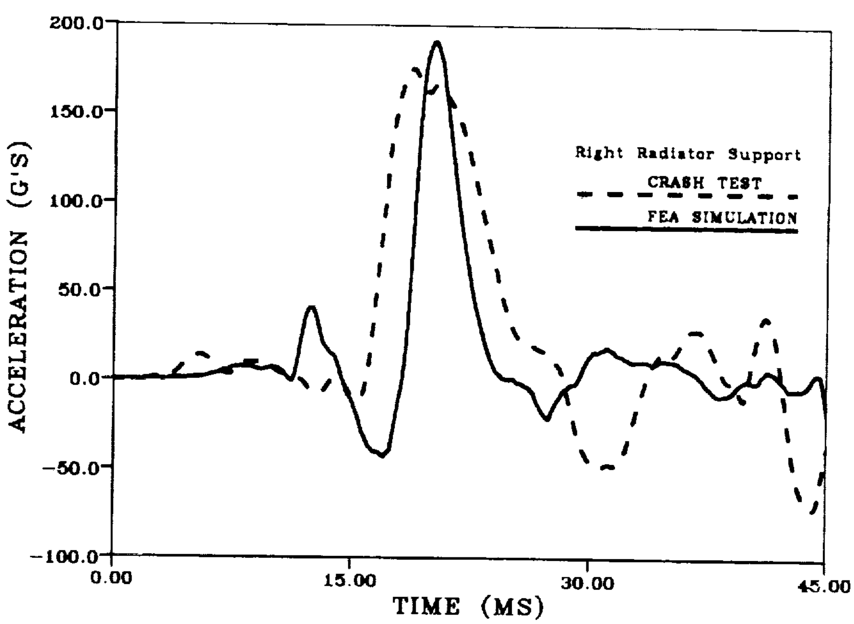
Finite element model of a vehicle impacting a rigid pole at 31 mph is shown below. The model has approximately 28000 elements. The deformed geometry shows that the pole is wrapped around by the front bumper beam and the engine block is pushed rearward.



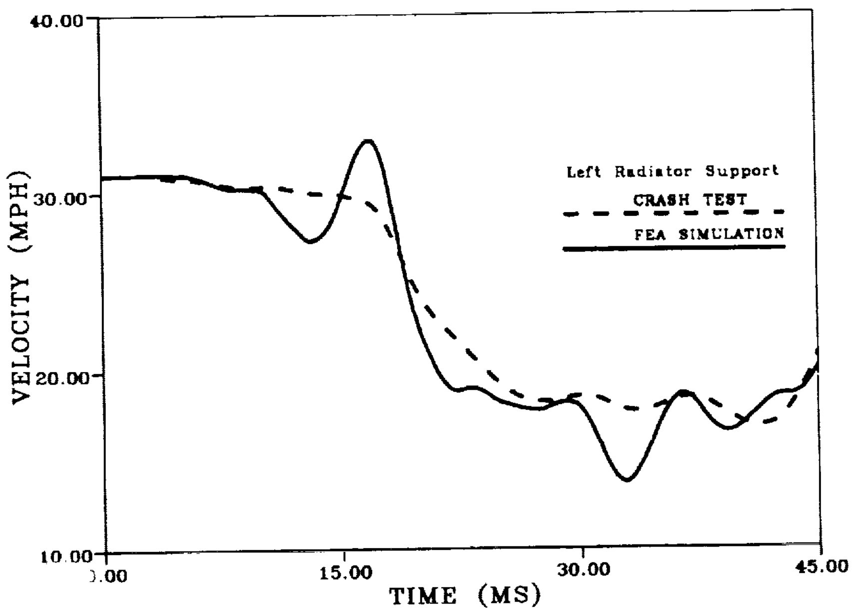
Comparison of B-Pillar/Rocker Joint Longitudinal Displacement with Test Data



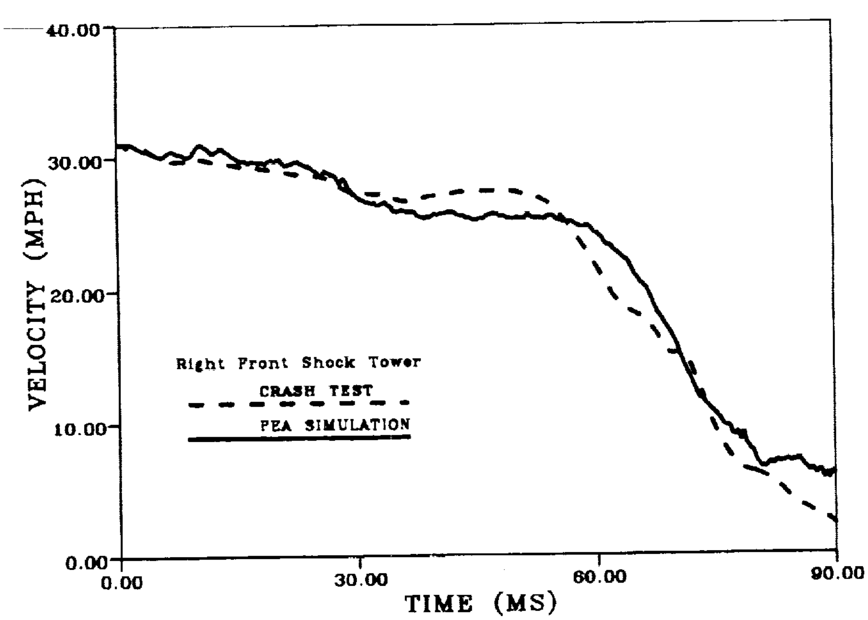
Accelerations at Right Side Radiator Support From Model and Test



Comparison of Predicted Velocity at Left Side Radiator Support with Test Data

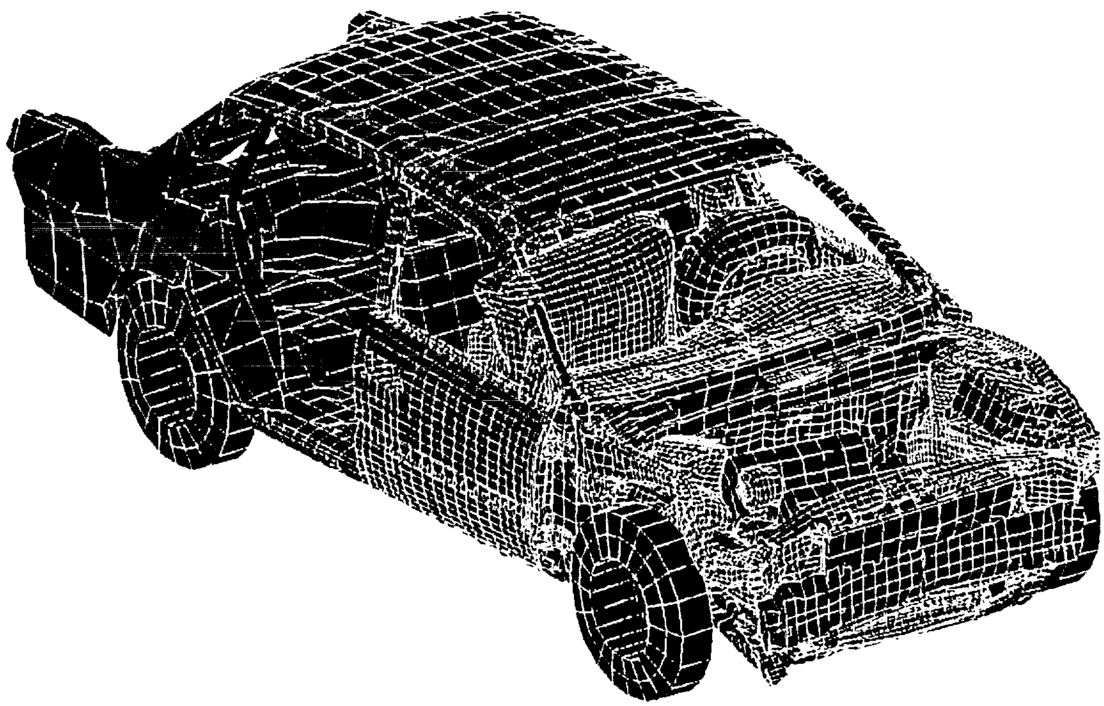


Comparison of Predicted Velocity at Right Shock Tower with Test Data



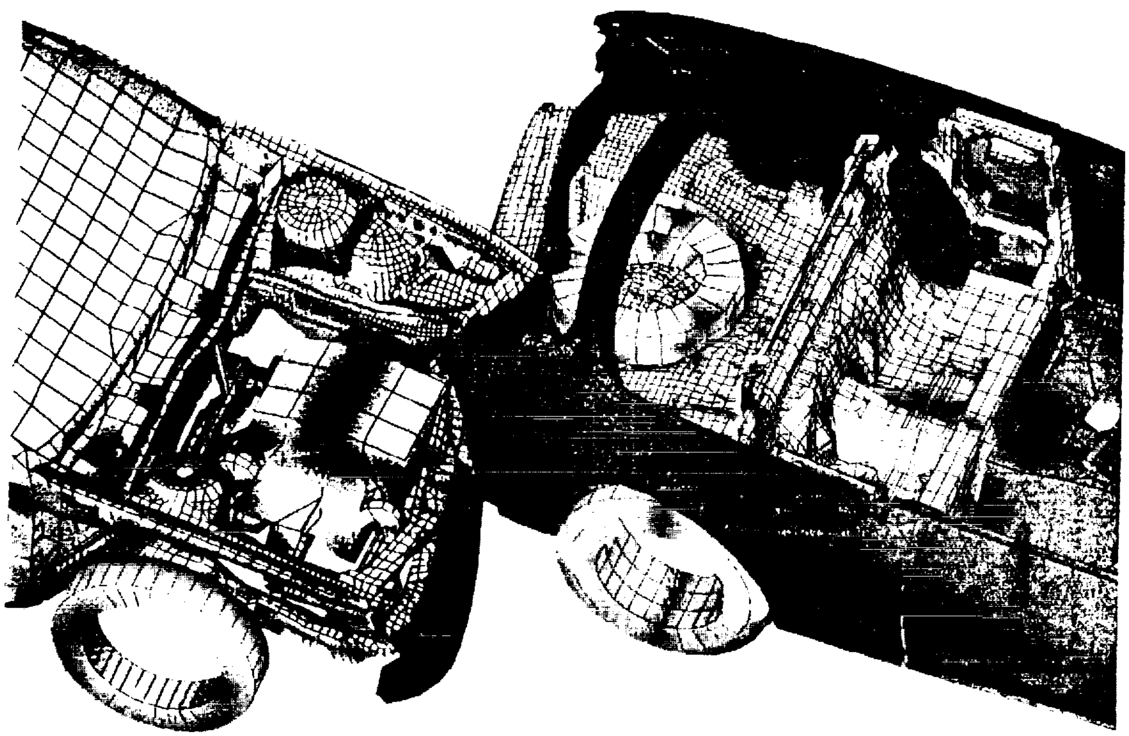
Front Crash System Model

Integrated system model consisting of vehicle, occupant, airbag, steering column systems, and other interior environment to predict both structural and dummy responses from the same run.



Vehicle Rear Impact Simulation

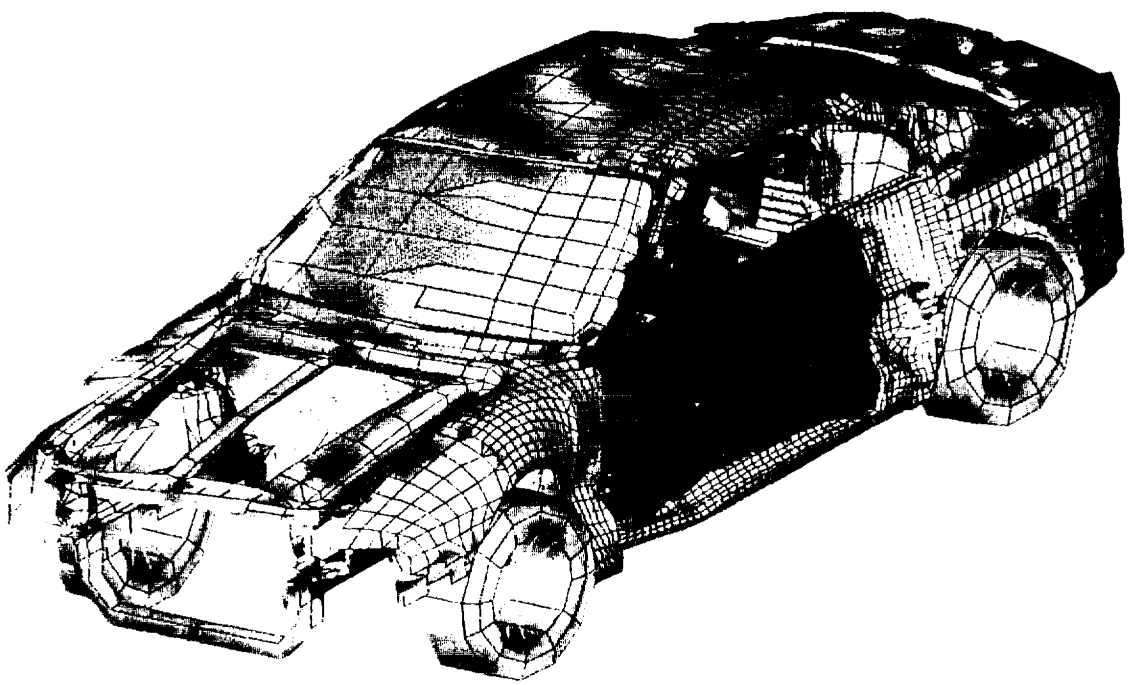
The car-to-car offset impact model is developed to predict strength and energy absorption of vehicle rear-end (rear rails, rear bumper system, lower back, quarter panel and rear floor). The main tasks are to minimize fuel tank deformation under severe impact, predict relative motion of bullet and target vehicles and displacement, velocity and deceleration histories all over the vehicles.



FEA Dummy-Structure Side Impact System Model

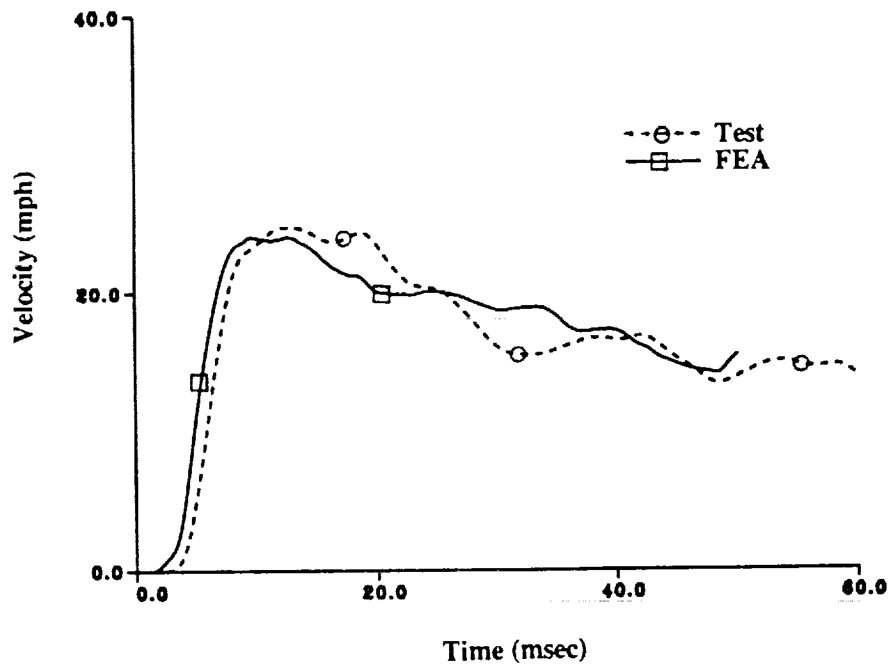
The side impact system model consists of a moving deformable honeycomb barrier, side impact dummy model and the vehicle structure. Approximate model size is 16000 elements and CPU needed on CRAY YMP is about 15 hours.

The detailed FEA model predicts strength and energy absorption by side structure (door, pillars, quarter panels) and energy absorption by trim and bolster as well as front and rear seat dummy responses (pelvic, spine and rib accelerations).

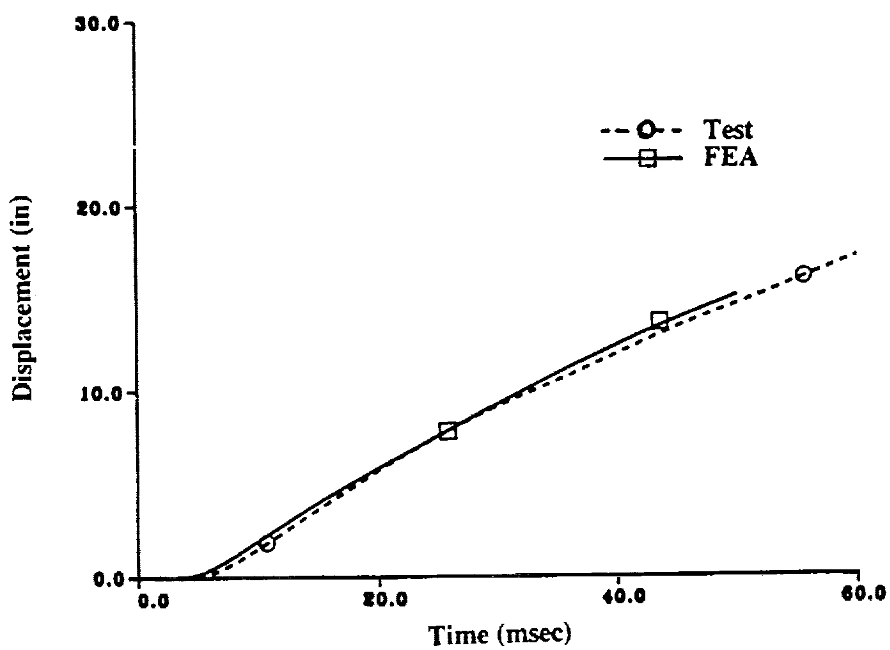


Vehicle Structure Response From Side Impact Model

Comparisons of model predictions of B-pillar lateral velocity and displacement histories with corresponding test data.



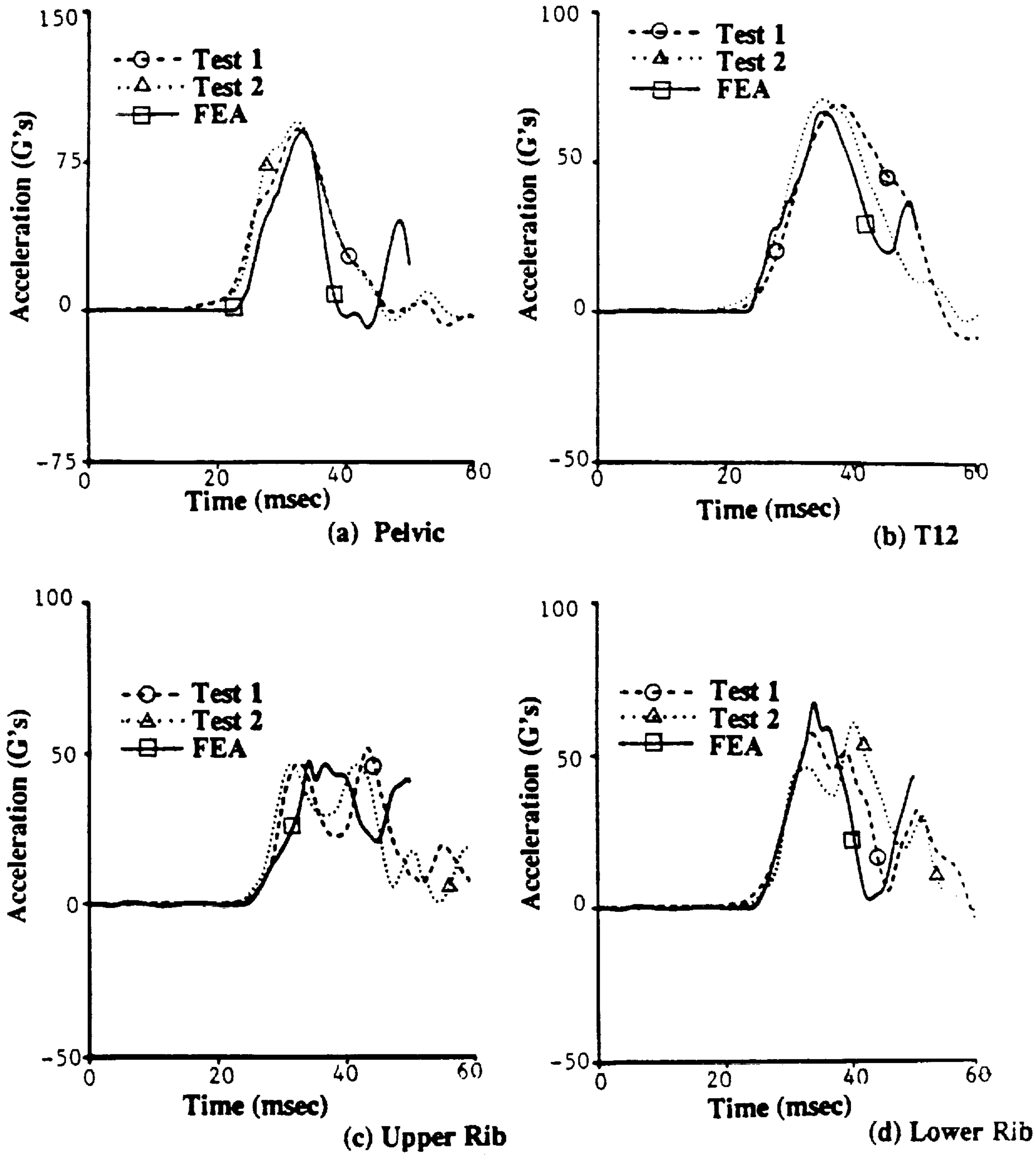
(a) B-pillar Velocity



(b) B-pillar Lateral Displacement

Occupant Responses From Side Impact System Model

Comparisons of front dummy pelvis acceleration, T12 acceleration and upper and lower rib accelerations from model and tests.



PUBLICATIONS ON CRASHWORTHINESS

1. Saha, N. K., Wang, H. C. and Achkar, R. E., "Frontal Offset Pole Impact Simulation of Automotive Vehicles," ASME Int. Computers in Engineering Conference, San Francisco, CA, Aug. 2-6, 1992.

2. Saha, N. K., Mahadevan, S. K., Midoun, D. E. and Yang, J. S., "Finite Element Structure-Dummy System Model for Side Impact Simulation," ASME Winter Annual Meeting, Atlanta, GA, Dec. 1-6, 1991.

3. Saha, N. K., "Design of Automotive Components for Crash Energy Management Using Nonlinear Finite Element Techniques," Proceedings of the 21st Midwestern Mechanics Conference, Houghton, MI, Aug. 13-16, 1989.

4. Devries, R. I., Saha, N. K., et al., "Structural Optimization of Beam Sections for Minimum Weight Subject to Inertial and Crash Strength Constraints," Proceedings of the Sixth SAE Int. Conference on Vehicle Mechanics, Detroit, MI, April 22-24, 1986.

5. Mahmood, H. F., Saha, N. K. and Paluszny, A., "Stiffness and Crash Strength Characteristics of Thin-Walled Plate Components," Proceedings of the ASME Int. Computers in Engineering Conference, Boston, MA, Aug. 1985.