ELSEVIER

Finite Elements in Analysis and Design 30 (1998) 279–295

FINITE ELEMENTS IN ANALYSIS AND DESIGN

Experiences during development of a dynamic crash response automobile model

J.G. Thacker, S.W. Reagan, J.A. Pellettiere, W.D. Pilkey[1,]*, J.R. Crandall, E.M. Sieveka

Automobile Safety Laboratory, Department of Mechanical, Aerospace, and Nuclear Engineering, University of Virginia, Charlottesville, VA 22903, USA

Abstract

A finite element automobile model for use in crash safety studies was developed through reverse engineering. The model was designed for calculating the response of the automobile structure during full frontal, offset frontal, or side impacts. The reverse engineering process involves the digitization of component surfaces as the vehicle is dismantled, the meshing and reassembly of these components into a complete finite element model, and the measurement of stiffness properties for structural materials. Quasi-static component tests and full-vehicle crash tests were used to validate the model, which will become part of a finite element vehicle fleet. © 1998 Elsevier Science B.V. All rights reserved.

Keywords: Automobile modeling; Non-linear finite element modeling; Reverse engineering

1. Introduction

Environmental pressures have provided the incentive to look for ways to reduce the need for energy and to reduce the pollutants that are often associated with its generation. This is particularly true in the automotive industry, which is closely associated with the world's most extensive usage of fossil fuels. One of the best ways to reduce fossil fuel consumption by vehicles is to reduce their weight drastically. However, this creates a problem for the near-term, since these "New Generation Vehicles" (NGVs) will be very different from the rest of the fleet in terms of both weight and structure. How the NGVs will behave in crash situations is of concern to industries and governments, which are, presumedly, committed to the task of improving highway safety while, at the same time, improving energy efficiency. To meet both goals in the United States, the

* Corresponding author.

[1] This paper corresponds to a lecture given by W. Pilkey at Melosh Competition, Duke University.

PII: S0168-874X(98)00043-2

government's highway safety agency the National Highway Traffic Safety Administration (NHTSA) has begun an initiative called the Partnership for New Generation Vehicles (PNGV).

The goal of the PNGV is to improve the competitiveness of auto-makers as they strive to create NGVs. One way to do this is to help industry assess the safety issues related to their new cars. This is normally done through government prescribed crash testing into standard barriers. However, this method contains the assumption that other vehicles with which the test subject might interact are of similar design. A computer model is one potential way to study the safety issues of vehicles which are radically different in design from those which are presently being sold.

The NHTSA's answer to this is to develop detailed, dynamic finite element (FE) models of representative vehicles from today's highway fleet. These models can then be used as "crash partners" for corresponding NGV models developed by the automobile manufacturers. For standardization, the NHTSA has chosen to do all FE modeling with the LS-DYNA program [1]. LS-DYNA has full, 3-D capability and is designed to handle highly dynamic events with large deformations. It is based on the public domain DYNA-3D code and is already widely used by industry.

2. Reverse engineering: Building a finite-element "fleet"

Creating a fleet of FE "vehicles" is not a simple task of collecting existing models. While CAD programs and FE programs are standard tools for today's manufacturers, they do not all use the same software nor do they readily hand out their proprietary models, even to government agencies. What must be done is to "reverse engineer" complete finite element models (FEMs) from actual vehicles and shop manual drawings. A new vehicle is disassembled until all relevant components have been measured. The measurements include mass, centers-of-gravity, and geometry. Representative samples are also chosen for destructive strength testing to obtain material properties. Having been converted into numerical data, the components of the car are then reassembled into a complete FE vehicle. The complete reverse engineering process consists of four steps [2]:

- Data collection
- Finite element model construction
- Model validation
- Model implementation

2.1. Data collection

This project, as visualized in Fig. 1, involved the creation of a complete geometrical and material model of a 1997 Honda Accord. A complete data characterization was done that included a visual and dimensional inspection of the intact car. Each component was identified, labelled, and the material evaluated. Data that could be efficiently extrapolated from existing sources were collected. Decisions concerning the modeling of each part were made as the automobile was disassembled. These decisions included the assumption that cabling and moulding had negligible mass and stiffness.

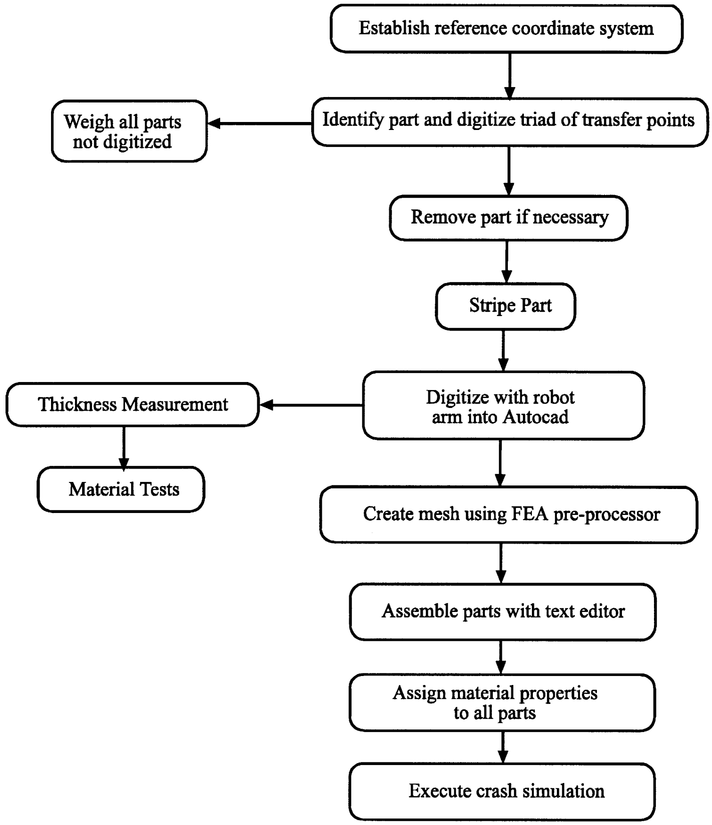

Fig. 1. Flowchart of project.

The parts were then inspected to determine the number of overlapping metal layers of any components and locations where symmetry could be applied. It was determined that most structural components (front bulkhead; A,B, and C pillars; doors; outer shell, etc.) could be assumed to be symmetric with respect to the car's centerline. Exceptions were the underside of the hood, the floorpan, the firewall, and the engine with related attachments.

2.1.1. Component digitization

The digitization phase of data collection is the process by which a numerical representation of the vehicle's undeformed geometry is obtained for finite element mesh generation. The primary tool used for digitization was a segmented, articulating measuring instrument (Fig. 2). This device interfaces with a computer and is capable of accurately recording the position of a probe at the end of the arm. The particular arm [3] used during this project had three points of articulation, a reach radius of 1.2 meters (m), and an accuracy of 0.3 mm. The points measured by the digitizer are collected on a PC workstation and pre-processed with a geometrical visualization program [4] to facilitate later processing by the finite element mesh generator.

The first step in the digitization process was to select the origin and orientation of the vehicle coordinate system so as to minimize the number of times that the digitizer had to be repositioned.

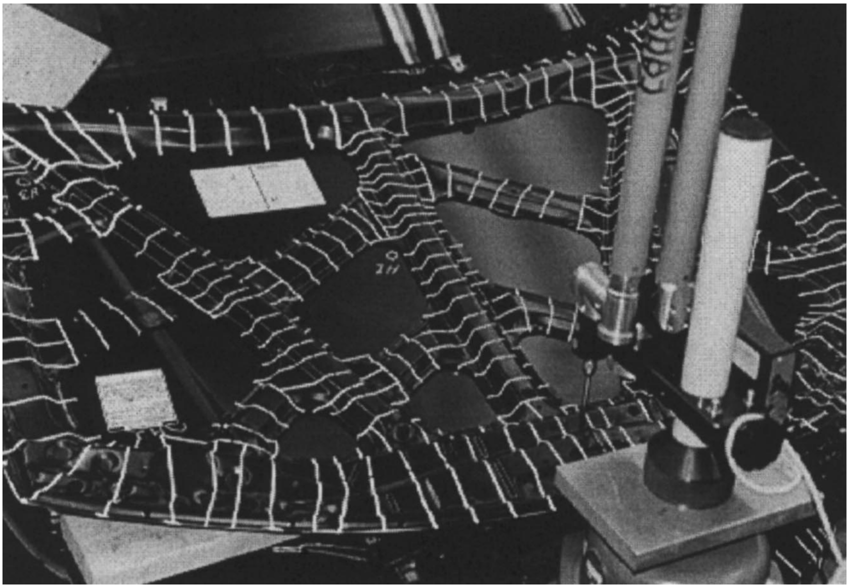

Fig. 2. Digitization arm being used on underside of hood.

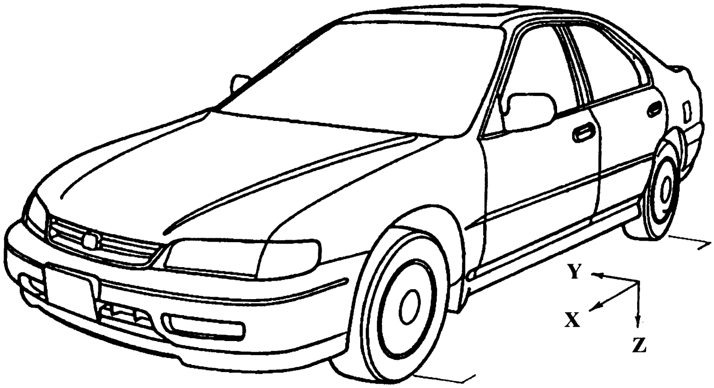

Fig. 3. Cartesian coordinate system for automobile.

The chosen system is shown in Fig. 3. When either the digitizer or the subject part had to be moved, data continuity was maintained through the use of reference point triads. Prior to movement, three reference points were marked and measured in the original coordinate system. After movement, these same points were redigitized and their new coordinates were referenced to the original coordinates. This permits the software to calculate all new measurements in the original system. By establishing new sets of reference points before each move, it is possible to leap-frog the digitizer around the vehicle and still have all of the data referenced to the same coordinate system. Obviously, the three reference points must all be within the reach of the digitizer before and after it

Fig. 4. Initially, striping tape is placed as a visual guide.

is moved. In the case of parts which must be removed from the vehicle for better access, reference points are measured in situ prior to removal.

To guide the movement of the digitizer over the often complex surfaces of the vehicle, thin, white striping tape was used to highlight holes, boundaries, and gradient changes (Fig. 4). Figs. 2 and 5 show the underside of the hood after preparation with the striping tape. The geometry of this part is complex and considerable data was required to characterize it properly. On the other hand, relatively flat surfaces such as the outerside of the hood and the roof could easily be described by a small number of points.

In addition to surface complexity, modeling limitations also affect the collection of geometry data. As will be discussed later, the minimum feasible element dimension in these models is 5 mm. In practice, the smallest average element size used in the present model is 15 mm (Table 1). For this reason, holes and surface features smaller than about 20 mm were generally ignored.

2.1.2. Mass and center of gravity

The center of gravity of the vehicle was determined from the weight distribution of the wheels (Table 2) and the front and rear track widths of 1515 and 1500 mm, respectively (Fig. 6).

2.1.3. Material properties

The materials used in the FEM were high, moderate, and low strength steels (structural components), glass (front and rear windshield), aluminum (engine block and assorted parts), plastic (bumpers), and rubber (tires). Tests of structural components removed from the vehicle revealed that three different steel alloys were used. The door beams were made of a high strength steel, the

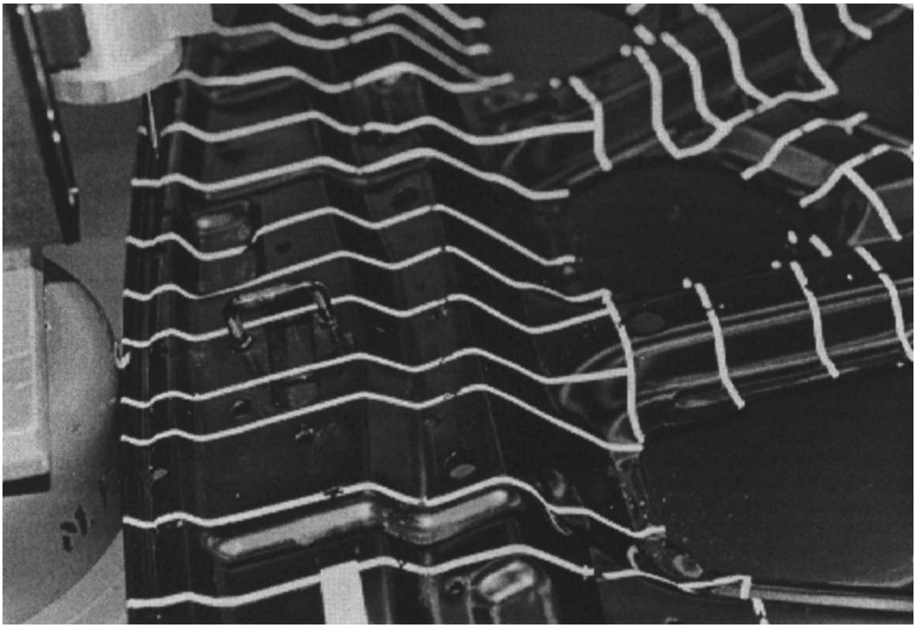

Fig. 5. A close up of the underside of the hood.

Table 1
Characteristic element size distribution

Location	Nominal element dimension (mm)
Front Bulkhead	15
Engine Cradle	25
Firewall	25
A pillar	25
B pillar	30
Floor	50
Outer sheet metal	75

Table 2
Weight distribution of received car

Wheel location	Weight N
Front, left tire	3339
Front, right tire	3830
Rear, left tire	2276
Rear, right tire	2307

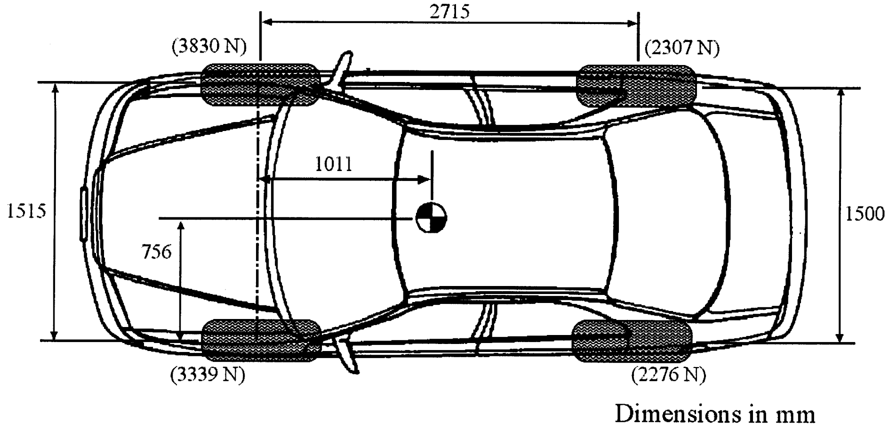

Fig. 6. Center of gravity for deliverd car.

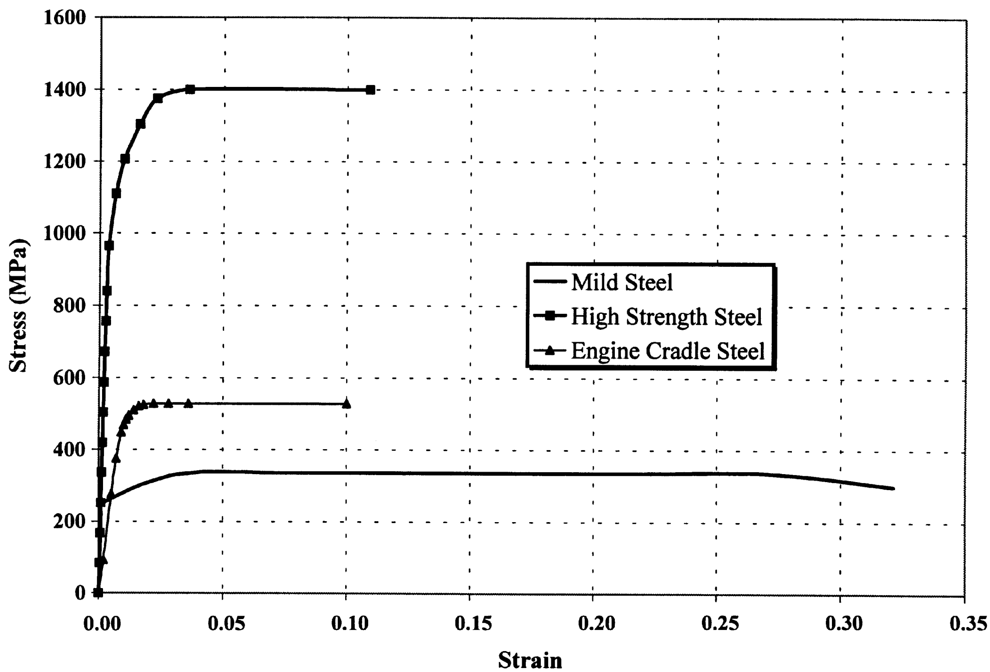

Fig. 7. Comparison of FEM steel material properties.

engine cradle of a more moderate strength steel, and all remaining tested components were manufactured from a low strength steel. Individual material samples were cut from various locations and physical testing performed to determine a stress–strain curve. It was determined that the yield strength of most frame components had a variation of about 20% using a three-point bending test. The stress–strain curves utilized for the model are shown in Fig. 7. Once the FEM had been assembled and all rigid bodies, connections, and materials specified, the second stage of the reverse engineering process (model generation) was completed.

2.2. Finite element model construction

2.2.1. Meshing and connecting

The finite element mesh for each part was constructed from the digitized line and surface data. Ultimately, each individual part is a continuous mesh with a single thickness and material property. However, the digitized data for a given part was usually subdivided into several sections, or "patches". Meshing a part, therefore, means meshing and connecting patches.

Each patch is described by a set of polylines or by a single AutoCAD surface. Meshing from polylines is a manual process of selecting pairs of lines and having the mesh generator create elements between them. Fortunately, most of the patches were described by a surface function which could be fed directly into an automesh algorithm. For the outer sheet metal, a characteristic element dimension of 75 mm was used. For the remaining parts, element sizes varied depending on the part location and expected deformation. The engine compartment was finely meshed, for example, and the element size generally increased towards the rear of the automobile. Table 1 shows approximate values for element size distribution.

With all of its component patches meshed, a complete part was assembled by merging nodes along coincident edges of adjoining patches (Fig. 8). Care was taken in the merging process to avoid creating excessively warped elements.

As adjacent parts were meshed, they were connected together to reassemble the vehicle. In general, parts were connected with spotweld or rigid-body constraints. Only in special cases, where excessive warping would not occur and where contact thickness constraints would not be violated, were nodes merged between separate parts. Spotwelds and rigid-body constraints are a type of imposed nodal constraint and cannot share nodes in common with each other or with other nodal constraints or rigid bodies. The primary difference between the spotweld constraints and the rigid-body constraints is the ability to specify a failure criterion for spotwelds. However, through inspection of several automobiles that had undergone severe deformation, it was determined that few spotweld failures occur. Because of this, the failure option for spotwelds was not used.

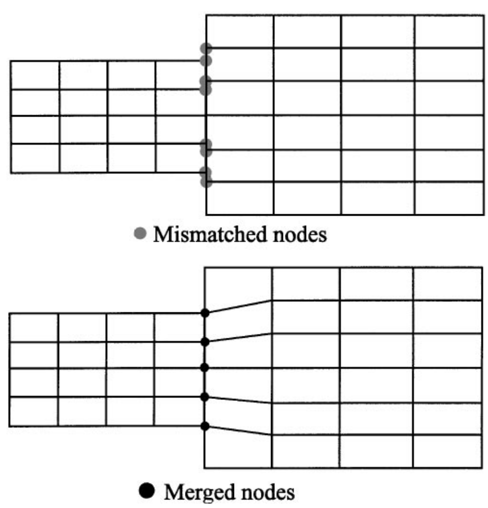

Fig. 8. Merging nodes for element continuity.

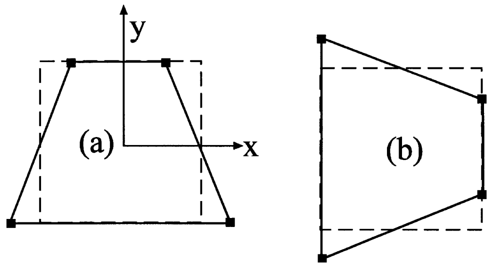

Fig. 9. Hourglass modes for bilinear shell elements.

The completed model contains approximately 177 parts, 40 materials, 88 000 elements, and 93 400 nodes. Structural components and specific element types used in the model include:

- Solid elements (≈ 2000) – engine, radiator, tires.
- Belytschko–Tsay shell element (≈ 83 000) – all pressed metal parts, glazing, bumper.
- Hughes–Liu beam element (≈ 50) – door beams (for frontal impacts), suspension components.

2.2.2. Shell element particulars

As described above, the bulk of the model is composed of shell elements. This is a natural choice since most of the vehicle's structure is formed sheet metal. LS-DYNA offers an extensive library of shell elements which differ in complexity, computational speed, and accuracy, but the simple Belytschko–Tsay (B–T) element is used almost exclusively in this model.

The standard B–T element is a four-node quadrilateral with a single Gaussian integration point at the center. In static analysis, four Gauss points are normally used to prevent the occurrence of zero-energy hourglass modes (Fig. 9) which are associated with under-integration [5]. However, the size of the computational task in dynamic finite element analysis prohibits the extensive use of fully integrated elements. Not only is the model structurally large (≈ 100 000 elements), but at typical integration time steps of approximately one microsecond, a 100 ms simulation run requires that the model be reevaluated approximately 100 000 times. This 10^5 increase in the computational load over the static solution of a similarly sized model forces the use of the most efficient element possible.

Fortunately, the large number of elements and the complexity of the interconnections inhibits the development of hourglassing to some extent. When it does occur, however, it is manifested in the dynamic environment as an undamped oscillation within groups of nodes (Fig. 10). To cope with this problem, LS-DYNA introduces an artificial viscosity to damp out the hourglass oscillations. The providers of the code suggest that the energy absorbed by this artificial damping should not exceed five percent of the total crash energy. Monitoring this "hourglass energy" term is one of the quality control checks used during dynamic simulations.

2.2.3. Time step considerations

The timestep of an explicit finite element analysis is determined as the minimum stable timestep in any deformable element of the mesh. In general, the Courant–Friedrichs–Lewy (CFL) condition

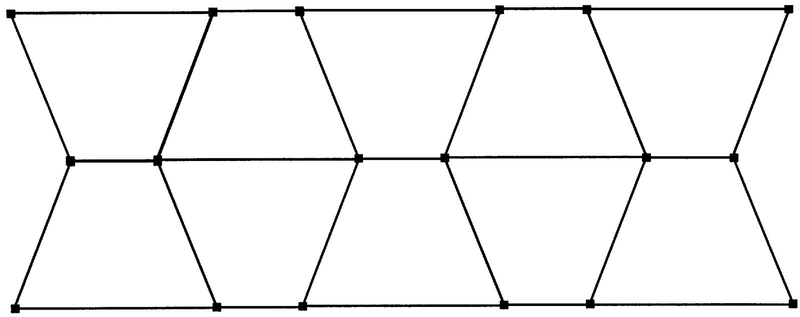

Fig. 10. Single shell elements combined to form hourglass shapes.

[6] gives

$$\Delta t \leqslant \frac{l}{c}, \tag{1}$$

where l is the characteristic length of the element and c is the acoustic wave speed of the material. Physically, this requires that the numerical timestep must be smaller than the time needed by the physical wave to cross the element. The CFL condition thus tells us that we cannot numerically calculate the effects of a stress wave in locations physically unreachable in the elapsed time. Spring elements have a timestep determined as

$$\Delta t = \sqrt{2m/k} \tag{2}$$

if both of the nodal masses are equal. The quantity m is the nodal mass and k is the stiffness. This can be seen to be equivalent to considering the spring as a truss element with

Mass:

$$m = \frac{Al\rho}{2}. \tag{3}$$

Stiffness:

$$k = \frac{EA}{l} \tag{4}$$

giving

$$\Delta t = \sqrt{2m/k} = \sqrt{l^2\rho/E} = \frac{l}{c}, \tag{5}$$

where E, A, and ρ are the elastic modulus, cross-sectional area, and material density, respectively. For shells we have

$$c = \sqrt{E/\rho(1 + v^2)}. \tag{6}$$

For mild steel, the primary material in most automobile bodies, $c = 5000\,\text{m/s}$. Thus, a minimum characteristic element size of 5 mm leads to a stable timestep of 1.0×10^{-6} s. or 1.0 μs. The target timestep in most crash analyses is 1.0 to 2.0 μs. Since a reasonable timestep leads to minimum element side length of about 5.0 mm, automotive body geometry cannot usually be represented entirely.

The analysis software allowed increasing the minimum stable timestep by adding "virtual" mass to the critical elements. This is termed mass scaling and is routinely used to control the timestep and to reduce computational time in crash simulation. From (Eq. (5)) it can be seen that increasing m will increase the minimal timestep. This additional mass may be up to 2–5% without introducing detrimental mass effects [7].

2.2.4. Rigid-body parts

The engine and transmission are massive objects and play a significant role in any crash event. However, they experience very little deformation relative to the surrounding sheet metal. It is reasonable, therefore, to model the engine and transmission as a rigid-body. The important features in this case are the surface geometry, the inertia properties, and the attachments to the vehicle structure. Proper modeling of these features ensures that the engine and transmission will load the surrounding structure correctly during the crash event.

2.2.5. Contact modeling

In dynamic finite elements, particularly when used for crash modeling, deformations are much larger than those typically seen in static analysis. Accurately modeling the stresses within a structure is not sufficient. One must now describe the inter-part contact between different parts of the model as well as intra-part contact when a part buckles in upon itself.

Fortunately, LS-DYNA is well-equipped to handle the contact problem. For a complicated model with many parts, such as a full vehicle model, the automatic contact algorithm is preferred. At the start of the run, LS-DYNA checks the spacing between parts and activates contact between nearby neighbors. As the structure collapses, the contact table is periodically updated. If an expected contact is missed by the automatic routine, it can be set explicitly by the user.

One consequence of assembling a model from many parts is that some initial penetrations can occur. This is due to accuracy limits associated with the digitizing process, as well as to the faceting of the part's surface by the finite mesh spacing. The program can handle small initial penetrations by adjusting the locations of nodes. This introduces some initial localized stress, but it is not a serious problem. Large initial penetrations, however, can cause the local stresses to exceed the material's yield stress. In these cases, the initial node positions must be readjusted manually. Often this situation can be detected by running sub-models of each part in a static, load-free situation to see if the part breaks apart or exhibits large, spontaneous deformations.

2.3. Model validation

The third stage of reverse engineering is model validation. Initially, the model was divided into several major components. These included the engine cradle, the front bulkhead, the A pillar, the B pillar, the firewall, the floor, and the rear end. This separation into components allowed a smaller model to be used during the assembly phase, since each component required significantly less computer resources than the entire automobile. This parsing also allowed the entire model to be meshed and assembled on personal computers. To aid in the verification process, each subcomponent was subjected to a simulated 35 m.p.h. impact into a rigid wall. These simulations allowed the stability of each component to be tested and modified if necessary.

2.3.1. Component tests

Two components, the front driver's side door and the steel door beams inside were selected for component testing. A large (100 ton) tension-compression test machine was selected to load these components in three point bending.

High strength steel reinforcing beams (approximately 32 mm in diameter with a wall thickness of 3 mm) are built horizontally inside each door to satisfy the FMVSS 214 Side Impact Protection requirements [8]. Each beam is anchored at the door latch and door hinge assemblies. Quasi-static three point bending tests were conducted on several beams. A $-45°/0°/+45°$ strain gage rosette was placed on the tensile side of the beam. These tests were then used to estimate the elastic modulus for the high strength steel as 208 GPa. The load versus displacement data for this test are shown in Fig. 11.

Two complete front doors were also loaded in three point bending, Fig. 12. The indenter used was a 50 mm diameter, schedule 40 steel pipe. The force–displacement comparison for two sample doors and the model are seen in Fig. 13.

2.3.2. Full-vehicle tests

Results of several full-vehicle crash tests were available to the authors. Insight into the physical crash behavior of the car is helpful in making a comparison and interpretation of the FE analysis

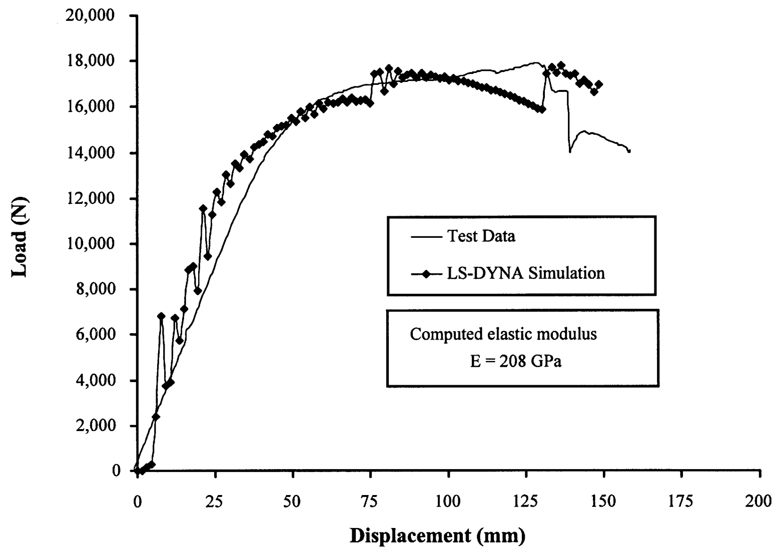

Fig. 11. Load vs. displacement comparison for high strength door beam.

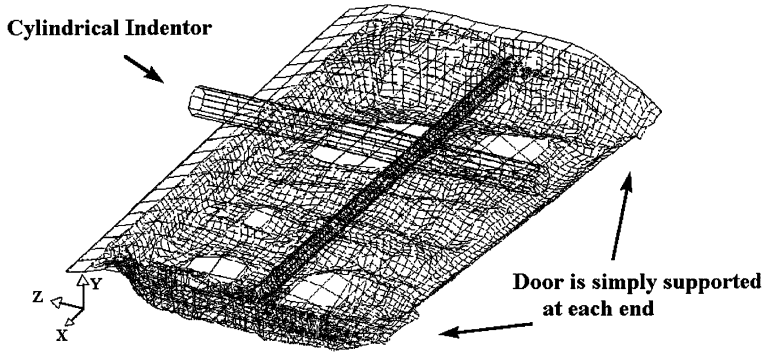

Fig. 12. Three point bending of door.

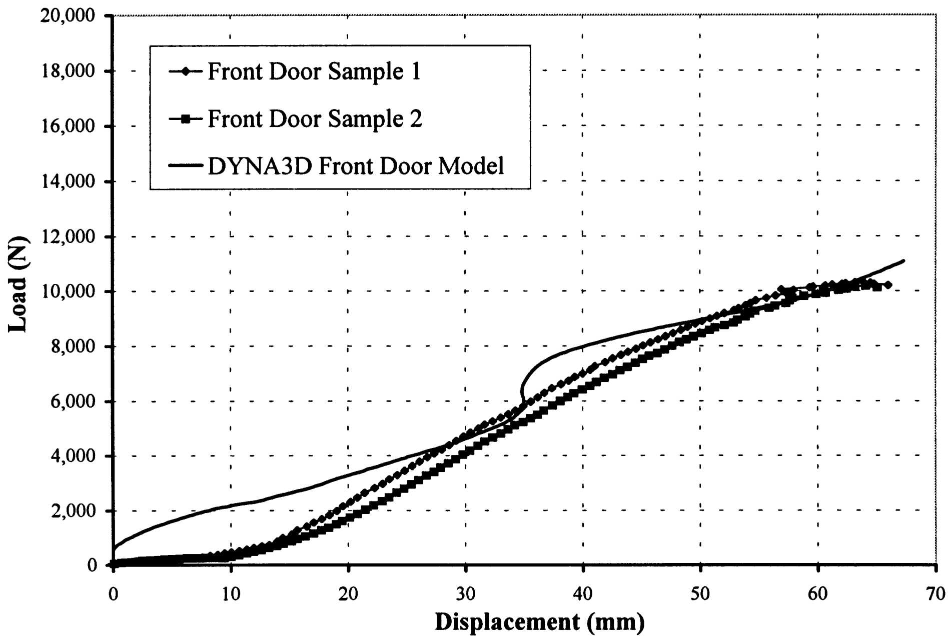

Fig. 13. Comparison of door model to component tests.

results. For a comparison, a primary quantity of interest is the velocity of certain locations in the structure. Although elemental strain and stress are of interest, they are of questionable reliability since in the FE analysis they are derived from severely under-integrated elements. As an illustration of one type of model validation, the full-vehicle test results [9] are compared relative to the FE analysis calculations for the velocity of the center of gravity (Fig. 14). Some FE simulations for a 40 m.p.h. frontal-offset crash are shown in Figs. 15–17. In order to conduct the simulated impact, the model was supplemented with a ground plane and a frontal rigid barrier overlapping the driver's side 41% of the car's width.

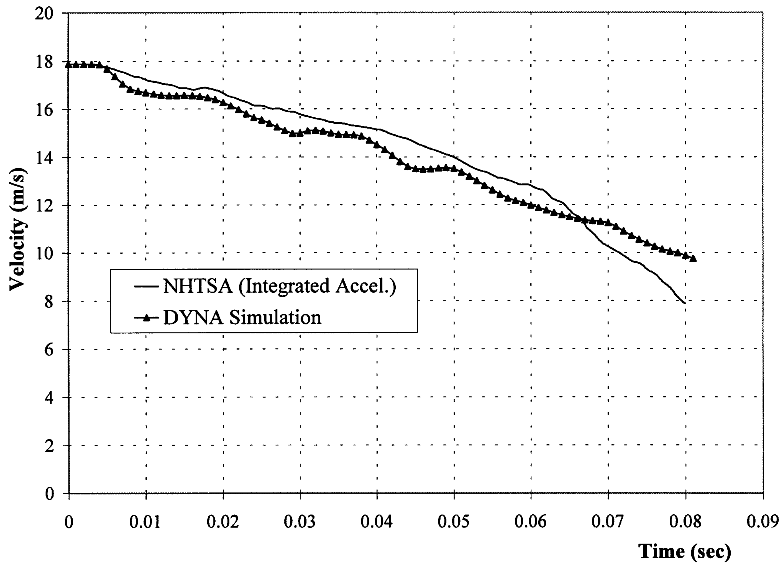

Fig. 14. Automobile center of gravity velocity comparison for a 40 m.p.h. frontal-offset crash.

Fig. 15. Isometric view of 40 m.p.h. frontal-offset crash simulation at $t = 0$ ms.

2.4. Model implementation

2.4.1. PNGV crash partners

As described in the Introduction, the Accord model developed during this project is a part of NHTSA's PNGV initiative. Altogether, about eight FE models of representative vehicles from the

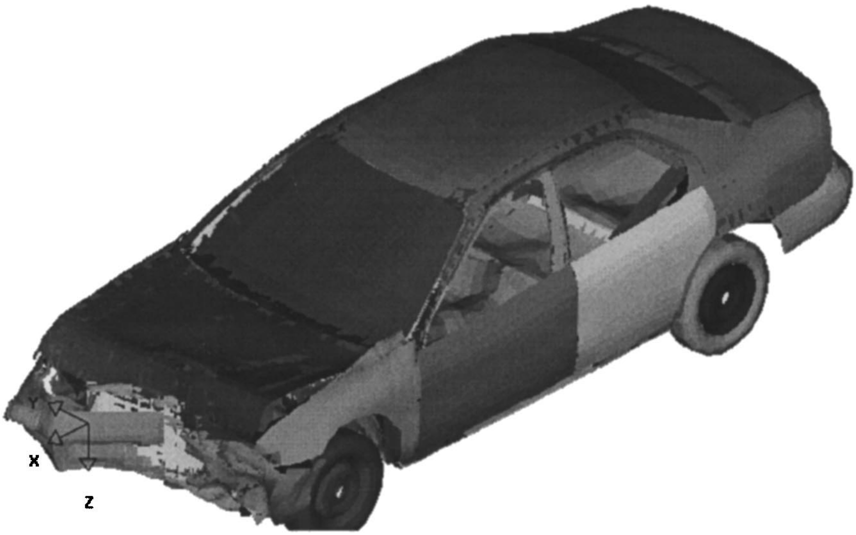

Fig. 16. Isometric view of 40 m.p.h. frontal-offset crash simulation at $t = 25$ ms.

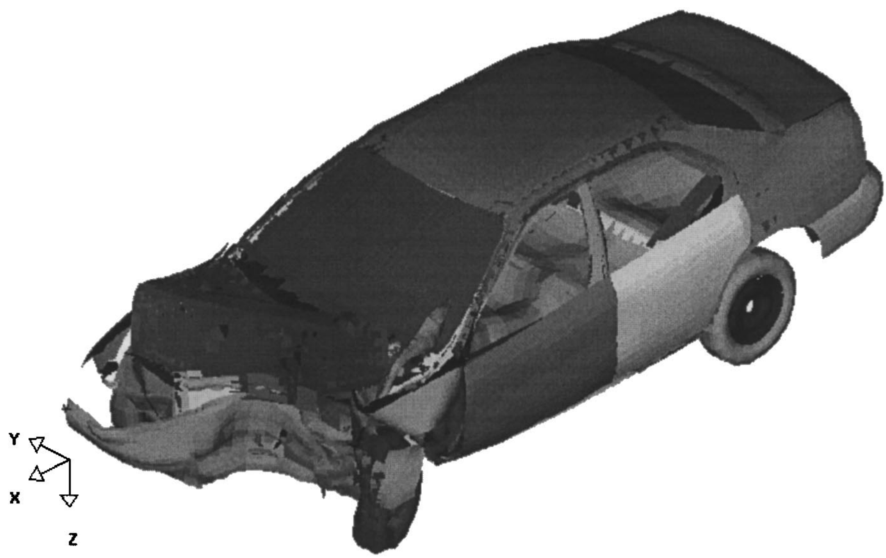

Fig. 17. Isometric view of 40 m.p.h. frontal-offset crash simulation at $t = 50$ ms.

current highway fleet are being built. These models will be provided to the U.S. auto industry so that their own FE models of PNGV vehicles can be crash-tested against them. The results from a safety point of view are uncertain. PNGV vehicles are typically much lighter than current vehicles; and light vehicles are normally less safe in crashes. However, PNGV vehicles are often much stiffer than conventional vehicles. This increase in stiffness can make the PNGV vehicles very aggressive crash partners. Creating a safe environment for passengers in both old and new vehicles is a major goal of the PNGV initiative.

2.4.2. From FE vehicles to multi-body occupants

Ultimately, the evaluation of how well a vehicle protects its occupants during a crash is based on several injury tolerance criteria that have been developed over the years. The primary measures involve head and chest acceleration, chest compression, and femur load. New cars must pass the required tests by having all of these measures fall below an acceptable threshold as measured by human-surrogate dummies called Anthropomorphic Test Devices (ATDs).

In principle, FE models of ATDs could be placed inside the FE vehicle models and an entire crash test event could be simulated. This, however, requires detailed occupant compartment geometry as well as a detailed dummy model. This could easily double the FE model's complexity and greatly increase the needed computer resources. A more efficient method uses the FE model to generate a vehicle compartment deceleration pulse which drives a multi-body occupant model. Since changes to compartment components, such as airbags, seat belts, and knee bolsters, have a negligible effect on the crash behavior of the vehicle itself, a single FE run can provide data to drive entire multi-body parameter studies. The savings in run-time are enormous. Present run-times on high-end workstations for LS-DYNA vehicle models are still measured in days, while multi-body run-times are typically less than 1 h, even for the most complex models.

3. Summary

A complete finite element structural model of a new vehicle has been created through reverse engineering. By carefully disassembling and digitizing the actual vehicle, an acceptably accurate computer model was constructed without the aid of mechanical drawing or computer-aided design information from the manufacturer. This model joins others from the PNGV project to form a fleet of finite element vehicles which can interact with the lighter and stiffer vehicles of the new generation. Occupant safety in both old and new vehicles will be maintained by refining new vehicle designs based on crash testing of the finite element models.

Acknowledgements

This work was supported by the Volpe Transportation Systems Center through the Partnership for a New Generation Vehicle program of the National Highway Traffic Safety Administration.

References

[1] J.O. Hallquist, LS-DYNA Keyword User's Manual, Livermore Software Technology Corporation, 1997.
[2] K.A. Ingle, Reverse Engineering, McGraw Hill Inc., New York, 1994.
[3] Faro Technologies, Inc., Lake Mary, Florida.
[4] AutoCAD Users Guide, Autodesk Inc., San Rafael, CA, 1996.

[5] R.D. Cook, Concepts and Applications of Finite Element Analysis, 3rd ed., Wiley, New York, 1989.
[6] J.C. Strikwerda, Finite Difference Schemes and Partial Differential Equations, Wadsworth Inc., Belmont, CA, 1989.
[7] J.O. Hallquist, LS-DYNA Theoretical Manual, Livermore Software Technology Corporation, 1997.
[8] Department of Transportation, National Highway Traffic Safety Administration, Federal Motor Vehicle Safety Standards, Side Impact Protection, Rule 49, CFR Part 571, 1990.
[9] Insurance Institute for Highway Safety, Crashworthiness Evaluation: Crash Test Report CF95009, 1995 Honda Accord LX, Arlington VA, 1995.