David R. Lemmon
Ming-yi Wu
Ronald L. Huston
Department of Mechanical, Industrial, and Nuclear Engineering
University of Cincinnati
Cincinnati, OH 45221-0072

Head Injury Reduction in Automobile Pedestrian Impact

This article presents and discusses automobile hood/fender rail design to reduce head injury of pedestrians struck by the front of the vehicle. Fender seam designs are presented that reduce the head injury criterion values by over 50%. The procedures and analysis are conducted using a nonlinear dynamic finite element program for an Oldsmobile Ciera hood and a Ford Taurus hood/fender. © *1994 John Wiley & Sons, Inc.*

INTRODUCTION

Head injuries in pedestrian/automobile impacts are among the most common and most harmful of all pedestrian impact inquiries. Field studies, experimental studies, and analytical simulations all show that when pedestrians are struck by the front of an automobile, they are generally thrust onto the hood or fender, and sometimes onto the windshield. Pedestrian impacts onto the hood, fender, or windshield generally result in a head impact injury.

Upon impact the softness of the hood or fender greatly affects the severity of the head injury. If the impact is to a relatively flat region of the hood, sufficiently high above the motor and frame understructure, the deformation of the hood will absorb the energy of the impact and thus reduce the injury severity. Alternatively, if the impact is to a "hard" spot, the injury severity will increase. Especially hard regions are the seams between the hood and the fender and between the hood and the windshield or cowl.

In this article we develop and present procedures for evaluating hood/fender/cowl impact response and design. Specific designs are presented for softening the fender seam while maintaining the overall integrity of the fender. The designs are evaluated by calculating head injury criteria (HIC) values.

The head impact phenomenon and the associated structural deformation is modeled using the finite element method (FEM) using the ANSYS and DYNA3D codes (Hallquist and Benson, 1989), processed on a Cray YMP/832 supercomputer. The analyses are validated with experimental data recorded through the Advanced Pedestrian Protection Program (APPP) (Millaris, Hitchcock, and Hansen, 1985) of the National Highway Traffic Safety Administration (NHTSA). The experiments were conducted at the Vehicle Research and Test Center (VRTC)

Received October 18, 1993; Accepted June 1, 1994.
This research was partially supported by grants from the Ohio Supercomputer Center and from the Transportation Research Center. Each is gratefully acknowledged.
Shock and Vibration, Vol. 1, No. 6, pp. 559–568 (1994)

CCC 1070-9622/94/060559-10

(MacLaughlin and Kessler, 1990) at the Transportation Research Center (TRC) in Ohio.

The balance of the article is divided into four parts with the first providing information on the methods of analysis. The second presents results of the analysis together with comparisons with experimental data. The third section presents a parameter study and the resulting improved fender seam designs followed by final discussion and concluding remarks.

METHODS OF ANALYSIS

Experimental Work

In both the analytic and experimental procedures, pedestrian head impact is simulated by thrusting a head form onto a hood, hood/fender, or hood/cowl structure. The head form is attached to the end of a pneumatic ram and is accelerated to a speed of 23 MPH. Details of the experimental work, including test equipment and methodology, are described in MacLaughlin et al. (1990). The 23 MPH speed was chosen because a 30–35 MPH vehicle/pedestrian impact is believed to produce a typical head impact speed onto the hood of approximately 20–25 MPH (MacLaughlin, Hoyt, and Chu, 1987). Thus a 23 MPH head impact speed would represent some urban intersection accidents.

The head form used in both the theoretical and experimental analysis was a rigid or nearly rigid spherical segment. The head form is not intended to be a biodynamic model of the head but instead it is intended to model the impact to the automobile structure.

HIC

A widely used measure of the head injury occurring from direct head impact is the HIC (Newman, 1980). Although there are recognized problems with the HIC (ibid.) it has nevertheless been shown to roughly correlate with the severity of the injury for pedestrian accidents (MacLaughlin, Wiechel, and Guenther, 1993). The HIC is calculated from acceleration/time data according to the formula:

$$\mathrm{HIC} = \left[(t_2 - t_1) \left(\int_{t_1}^{t_2} a(t)dt/(t_2 - t_1) \right)^{2.5} \right]_{\max}$$

where a is the acceleration of the head mass center expressed in g (gravity acceleration), t is time, and t_1 and t_2 ($t_2 > t_1$) are two points in the impact time interval chosen to maximize the resulting HIC value. An HIC value of 1000 has been identified as a threshold value for severe head injury (Newman, 1980).

FE Modeling

The FEM is an attractive procedure for studying such impact phenomena. Indeed, the FEM and auxiliary procedures have now been developed to the point where nonlinear dynamics in the form of large rapid deformation due to impulsive forces can be simulated. In particular, the recently developed nonlinear dynamic FE code, DYNA3D, with computations performed on a supercomputer together with the graphical/animation procedures of I-DEAS software, provide a means for simulating the large, rapid deformation of the sheet metal structure of automobile hoods and fenders when struck by a head form.

Unlike DYNA3D, the FE code ANSYS was not primarily developed for nonlinear dynamic analyses. However, ANSYS is well established and it has nonlinear analysis capabilities. Thus, ANSYS is an excellent code for validating the newer DYNA3D.

DYNA3D employs an explicit (as opposed to the ANSYS implicit) scheme in the time domain. As a result, with DYNA3D time-history data is more finely recorded and more accurate. Computation time is reduced from hours to minutes on a supercomputer. In addition, DYNA3D provides an easy interface between contacting surfaces as opposed to the use of the gap elements of ANSYS. Specifically, with DYNA3D the impact between the head form and the hood structure is obtained by selecting elements on both the head form and the hood structure that may contact one another.

To ease input file preparation for the FE analysis and for interpretation of the results, I-DEAS was used, a computer software procedure with advanced computer graphics.

TEST RESULTS AND VALIDATION

Hood Impact

Experiments conducted at the VRTC (MacLaughlin and Kessler, 1990) show that the 1985 Oldsmobile Ciera hood provides good perfor-

mance in reducing the head form deceleration. Also, the understructure of the Ciera hood is relatively simple so that FE modeling is more exact. Hence, the Ciera hood was chosen as a basis for the study.

The Ciera hood is composed of an upper surface panel and an underside panel of support ribs. The two panels are attached by welds and folds around the perimeter of the hood.

The FE model for an ANSYS analysis employed quadrilateral and trilateral shell elements. The head form was also modeled using FEs with nodes in a spherical arrangement. The contact was then modeled through the closing of gap elements. Figure 1 shows the half mesh of the symmetric model.

The elastic/plastic behavior of the hood structure was simulated by a two-line (bilinear) stress–strain curve, characterized by elastic and tangent moduli and yield strength of steel. The ANSYS transient analysis was divided into 20 time steps, each 1 ms in duration. Twenty iterations were specified for each time step to obtain convergence. The nodes representing the head form were moved at 23 MPH (37 km/h) into the hood model. Displacement and gap element force data was obtained for each of the 20 time steps. The resulting deformation time history is depicted in Fig. 2.

This FE model was slightly modified for the DYNA3D analysis. The head form was modeled as a separate body composed of 8-node brick elements with material properties of aluminum. A rigid mass was added to the head form to bring the total weight of the head form to 10 lb. (4.54 kg). (A ¼ in. thick rubber head form covering used in the experimental work was not included in the FE model, but will be added in future work.) Instead of gap elements, sliding contact was specified between the hood and the head form as represented in Fig. 3.

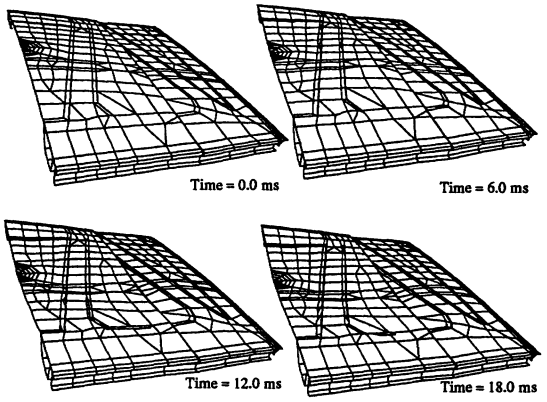

FIGURE 2 ANSYS Ciera hood model: deformations.

In the analysis, the head form is given an initial velocity of 23 MPH (37 km/h) and impacted onto the hood surface. DYNA3D's explicit algorithm automatically solves the dynamic impact response at fine time intervals.

The resulting deformation time history is shown in Fig. 4. Figure 5 presents a comparison between ANSYS, DYNA3D, and experimental results.

Hood/Fender Seam Impact

Most automobile hoods are soft and compliant in the center. However, impacts at the edges can produce severe injury to the head of a struck pedestrian. The edges are more rigid because of structural members that support the hood. HIC values of 2000 or more have been measured from impacts at areas where the hood and fender meet. It is therefore reasonable that a compliant,

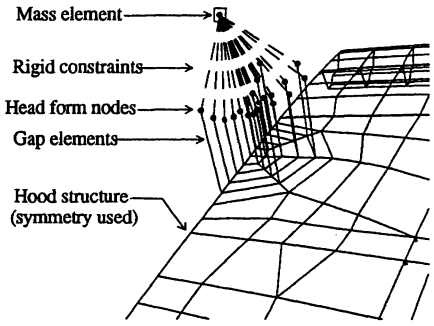

FIGURE 1 ANSYS Ciera hood model: head form representation.

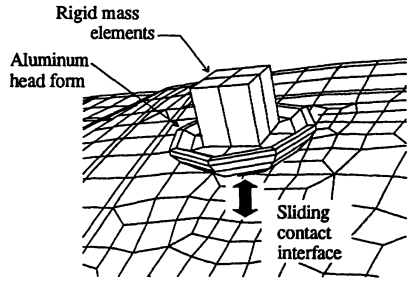

FIGURE 3 DYNA3D Ciera hood: head form representation.

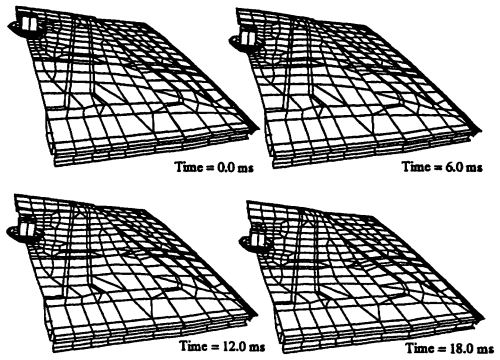

FIGURE 4 DYNA3D Ciera hood model: deformations.

energy-absorbing design can greatly reduce the risk of injury at the hood–fender interface.

A model of a 1988 Ford Taurus fender was the first step in finding such a design. An inboard flange at the hood–fender interface provides most of the rigidity in the fender during a top impact as depicted in Fig. 6. Upon impact, the flange buckles and dissipates energy across its entire height. By empirically altering the way this flange buckles under impact we can determine the manner in which rapid head deceleration can be reduced.

A source of stiffness or "hardness" is the apron rail or "shotgun rail." The fender is attached to the top of the apron rail by screws. The apron rail, which resembles a shotgun, is common in modern unit body automobile frames. Its primary purpose is to provide rigidity between the front of the automobile and the central body. In the case of the 1988 Taurus, the apron rail has a vertically protruding flange used to weld the two halves of the rail (Fig. 6). Head form impacts to the fender strike this flange, increasing the force to the head form, and hence, increasing the HIC values.

A baseline fender model composed of quadrilateral shell elements incorporates these factors. The inboard flange mentioned earlier is 1.5 in. (3.81 cm) high and contain 0.25 × 0.25 in. (0.64 × 0.64 cm) shell elements in the area of impact. Such a mesh is required to model the collapse of the wall during impact. The apron rail top surface and flange were also modeled. All edges of the fender model except for the edge that contacts the apron rail were completely fixed by boundary conditions.

For DYNA3D analysis, sliding contact was specified between the head form and the fender top surface. It was also necessary to specify "single surface" sliding contact on the vertical wall. A single surface sliding contact interface prevents a surface from passing through itself as it collapses. Finally sliding contact was specified between the underside of the fender and the apron rail as in Fig. 7.

In addition to an analysis of the baseline model the fender rail was modified to decrease acceleration peaks and thus lower the HIC values. Specifically, the height of the inboard flange was increased from 1.5 to 2.25 in. (5.72 cm). Next, holes were placed in the inboard flange to weaken it for downward directed impact and to thus control the collapse. Third, the positioning and the size of the holes were varied.

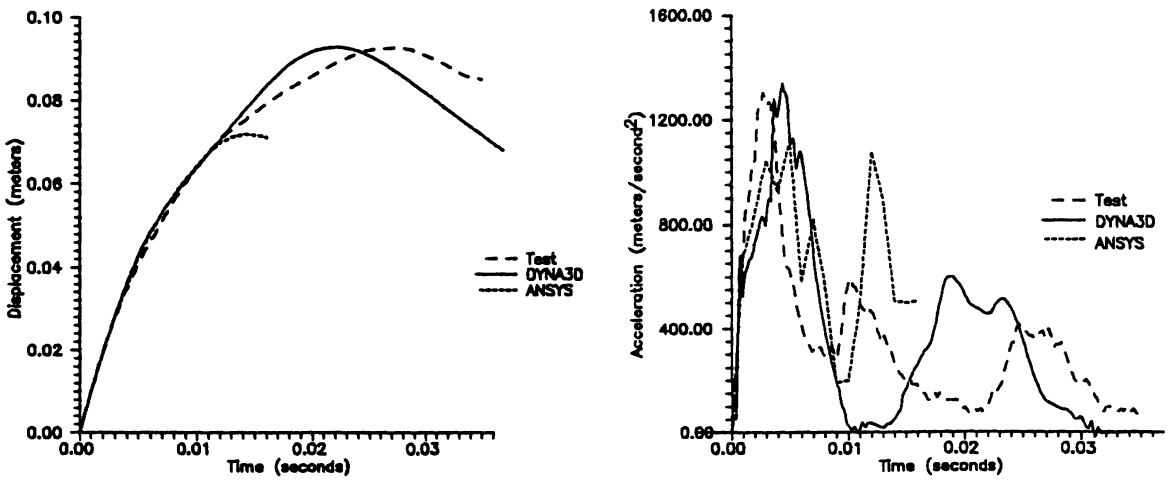

FIGURE 5 Ciera central hood: head form displacement and acceleration.

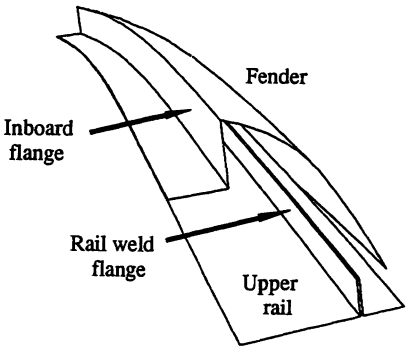

FIGURE 6 Taurus fender model: design features.

An FE analysis of this modified design with vertical holes 0.5 in. (1.17 cm) wide × 1.75 in. (4.45 cm) high and spaced 1.0 in. (2.54 cm) center-to-center was compared with experimental tests. Figure 8 depicts the modified design and its response to head form impact.

To study the effects of the seam and the adjacent structures, combined hood/fender and hood/fire wall models were developed. These models and their deformations are depicted in Figs. 9–11.

Two impact locations were used to simulate baseline configuration response. The first was located 4.375 in. (11.11 cm) forward and 3.175 in. (8.06 cm) to the right of the fender origin, defined as the rearmost point of the hood along its centerline. The second impact location is 2.0 in. (5.08 cm) forward and 14.5 in. (36.83 cm) to the left of the origin. The analysis was conducted for a fire wall with material properties of aluminum, with the impact location being 2.0 in. (5.08 cm) forward and 14.5 in. (36.83 cm) to the left of the origin.

Figures 12–18 show comparisons between the numerical simulations and experimental results. Acceleration comparisons show higher frequency components of acceleration in the

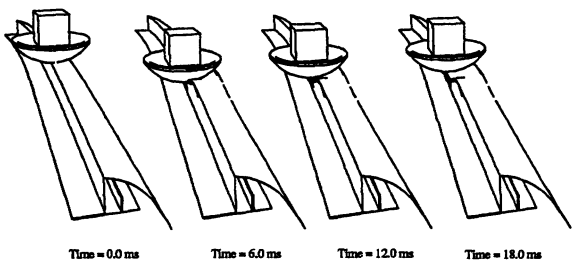

FIGURE 7 Baseline Taurus fender model: deformations.

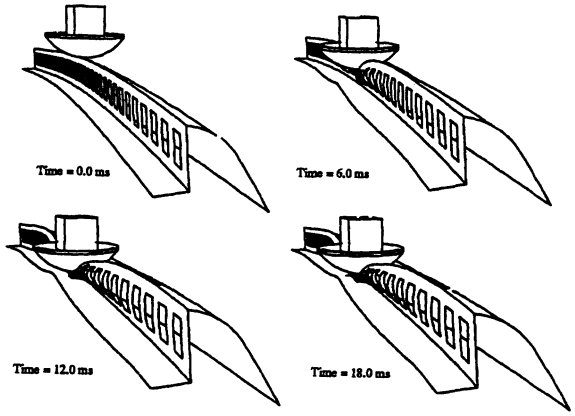

FIGURE 8 Modified Taurus fender model: deformations.

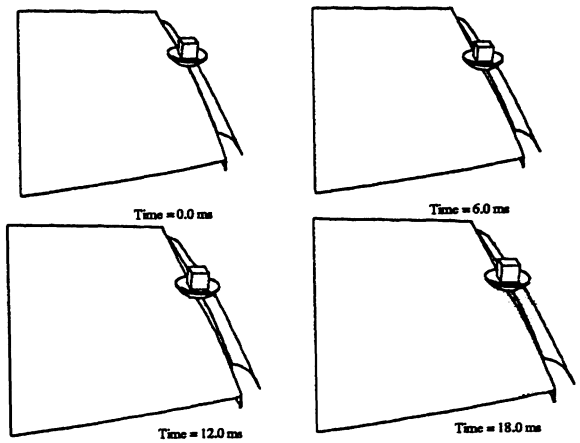

FIGURE 9 Baseline Taurus hood/fender model: deformations.

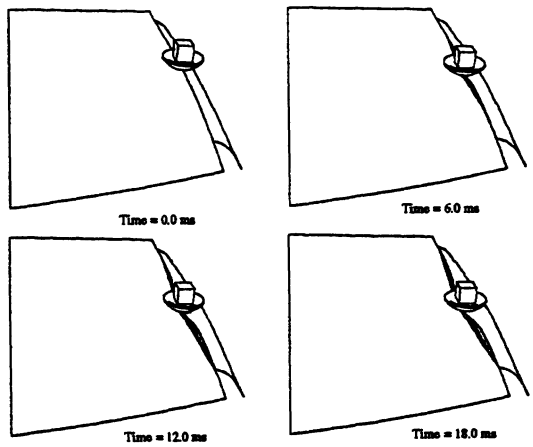

FIGURE 10 Modified Taurus hood/fender model: deformations.

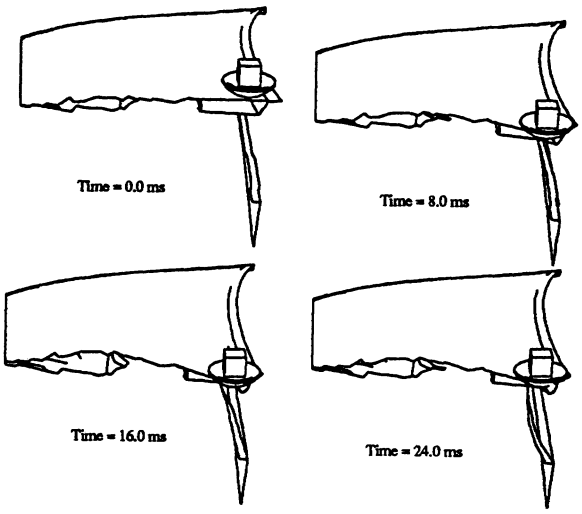

FIGURE 11 Baseline Taurus hood model: deformations. (Hood is cut away to reveal fire wall.)

DYNA3D results than in the experimental results. This is mainly due to numerical noise introduced by FE discretization and by the contact–impact interface, but may also be due to the exclusion of the rubber head form cover in the current FE model.

PARAMETER STUDIES

These results show that minor changes in the design of the fender flange can significantly reduce HIC values without adversely compromising the structural integrity of the fender. However, this raises two questions: How should the fender flange holes be shaped and placed to optimally reduce HIC values? And how does the fender flange thickness affect the HIC values?

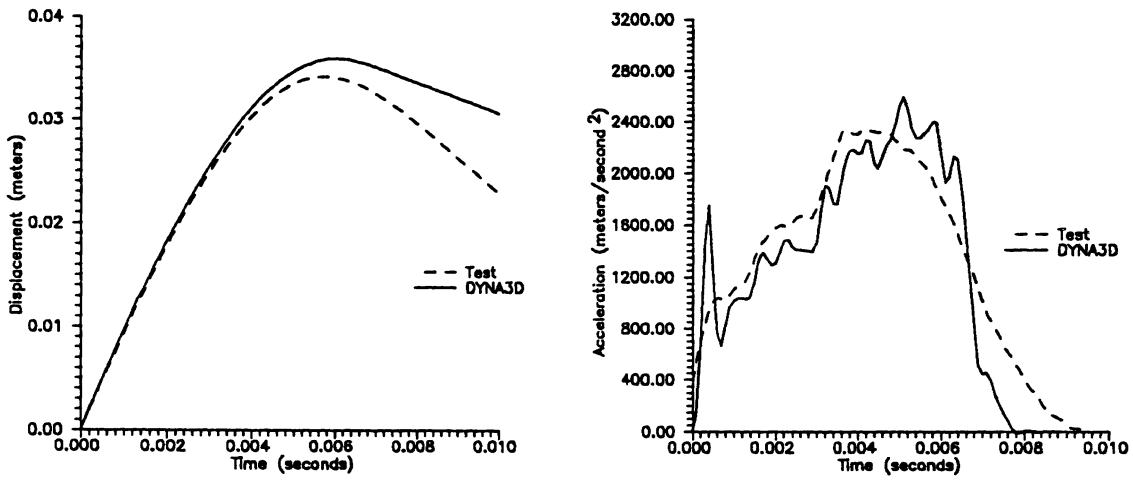

FIGURE 12 Baseline Taurus fender: head form displacement and acceleration.

Table 1. HIC from Test and DYNA3D Results

Description	HIC from Test Results	HIC from DYNA3D Results	Error (%)
Ciera, central hood, baseline configuration	442 (0.875–5.125)	537 (3.600–10.400)	30.1
Taurus fender			
Baseline configuration	2816 (1.000–6.875)	2739 (1.291–6.591)	2.8
Modified (holes)	1236 (0.375–11.875)	1831 (8.500–11.400)	48.1
Taurus hood–fender			
Baseline configuration	3351 (0.000–5.625)	2856 (1.000–7.700)	14.8
Modified (holes)	1182 (0.125–11.625)	1518 (1.100–10.400)	28.4
Taurus hood			
Loc. 1, baseline configuration	1391 (0.625–10.375)	1461 (1.900–11.400)	5.0
Loc. 2, baseline configuration	1898 (0.625–9.000)	2002 (9.100–10.400)	5.5
Modified (aluminum fire wall)	844 (0.375–14.875)	1061 (3.100–16.200)	25.7

Interval of maximum HIC in milliseconds shown in parentheses.

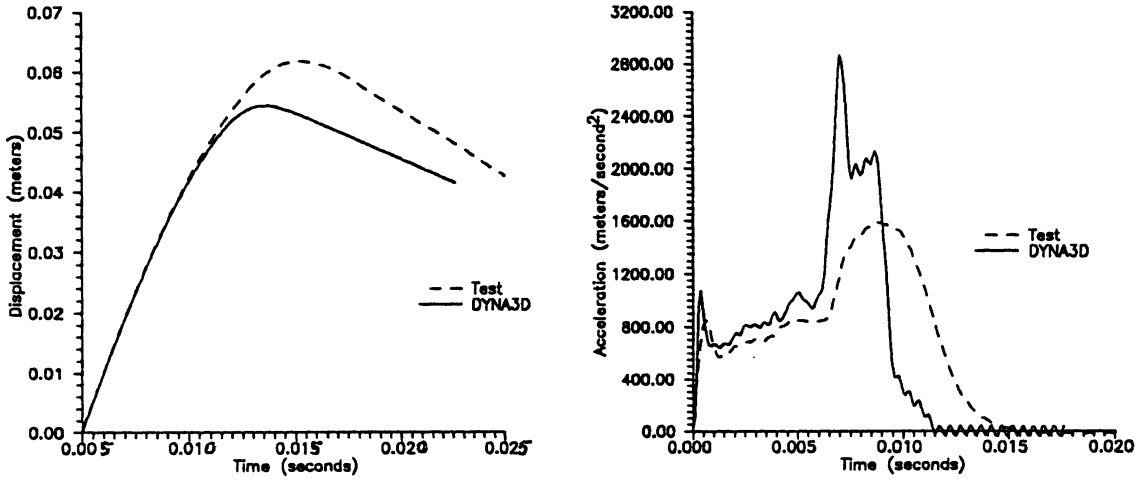

FIGURE 13 Modified Taurus fender: head form displacement and acceleration.

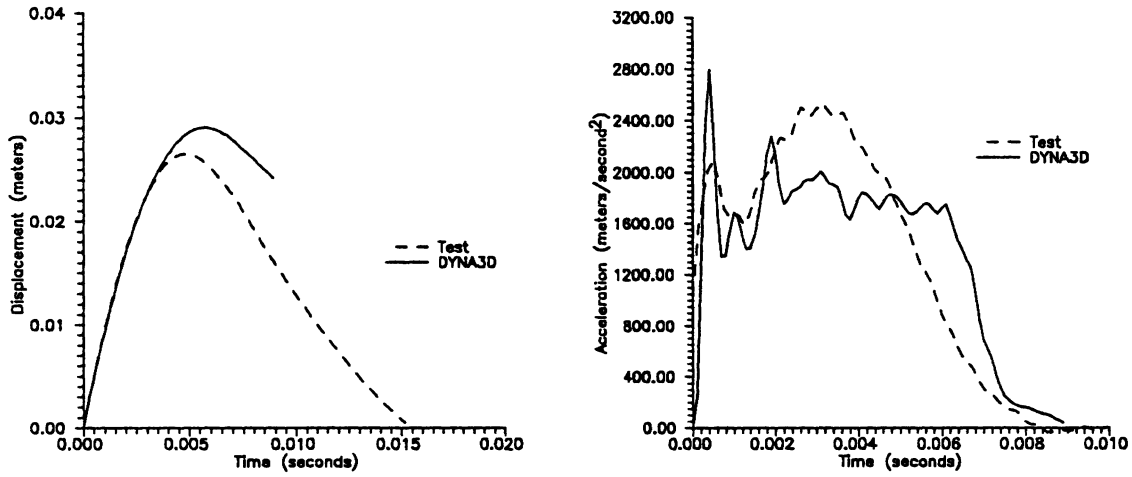

FIGURE 14 Baseline Taurus hood/fender: head form displacement and acceleration.

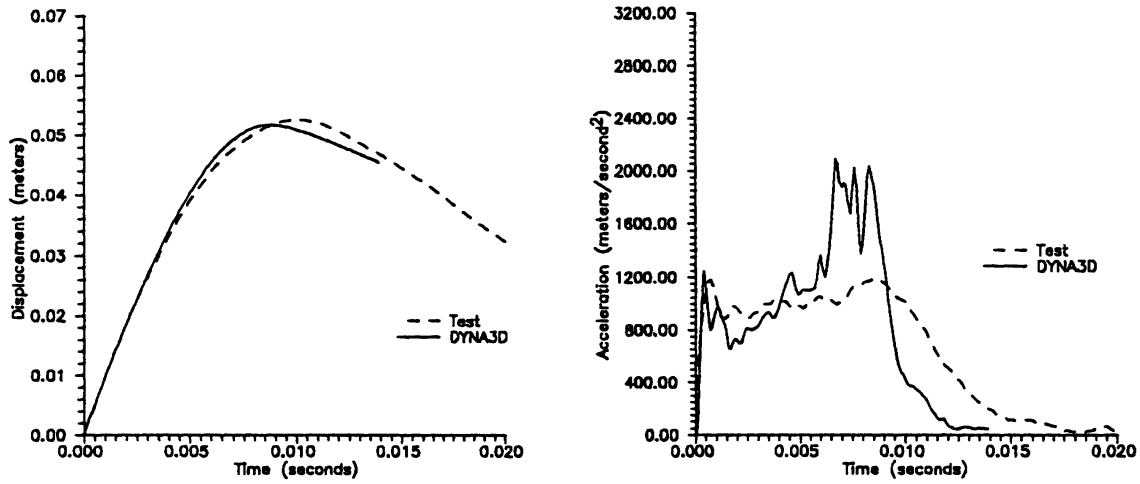

FIGURE 15 Modified Taurus hood/fender: head form displacement and acceleration.

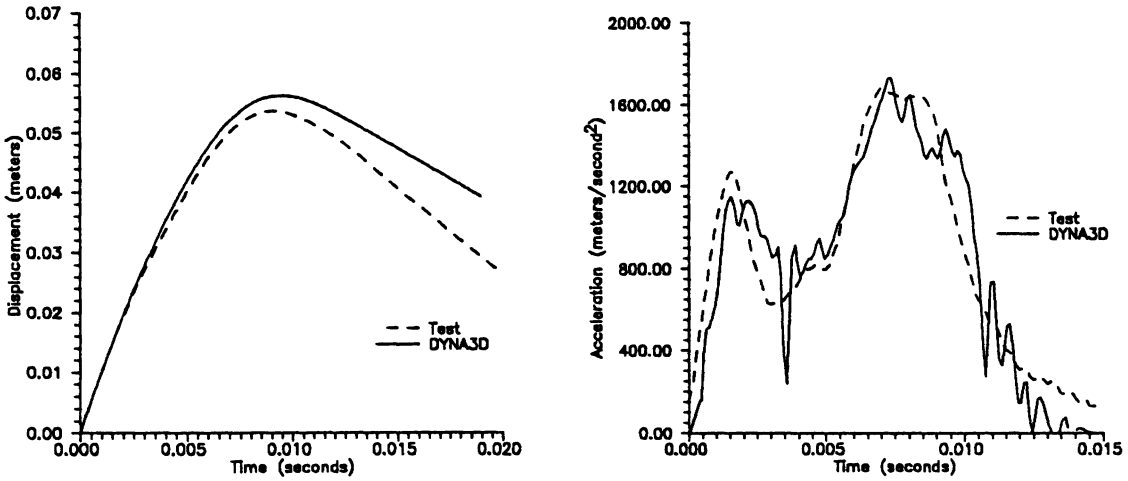

FIGURE 16 Taurus rear hood, location 1: head form displacement and acceleration.

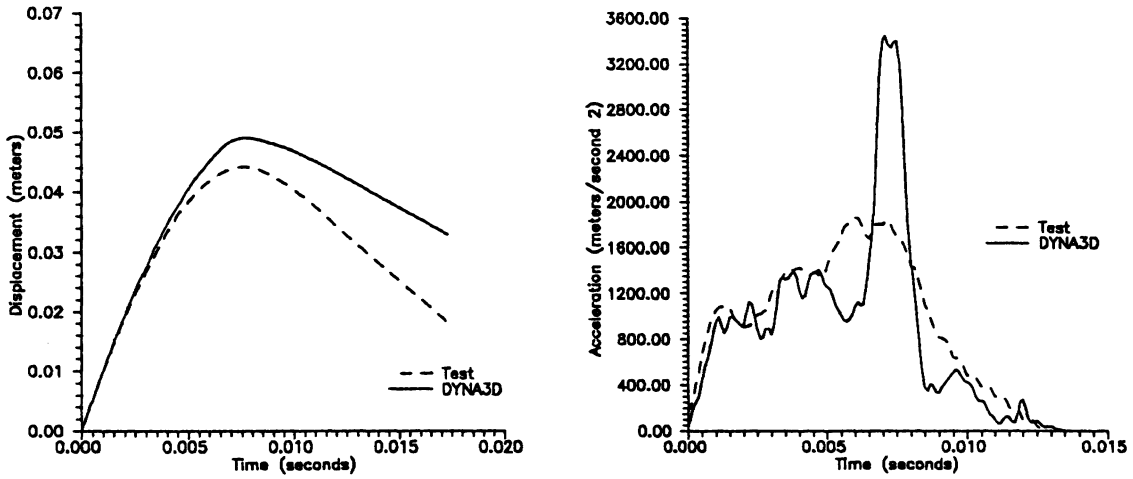

FIGURE 17 Taurus rear hood, location 2: head form displacement and acceleration.

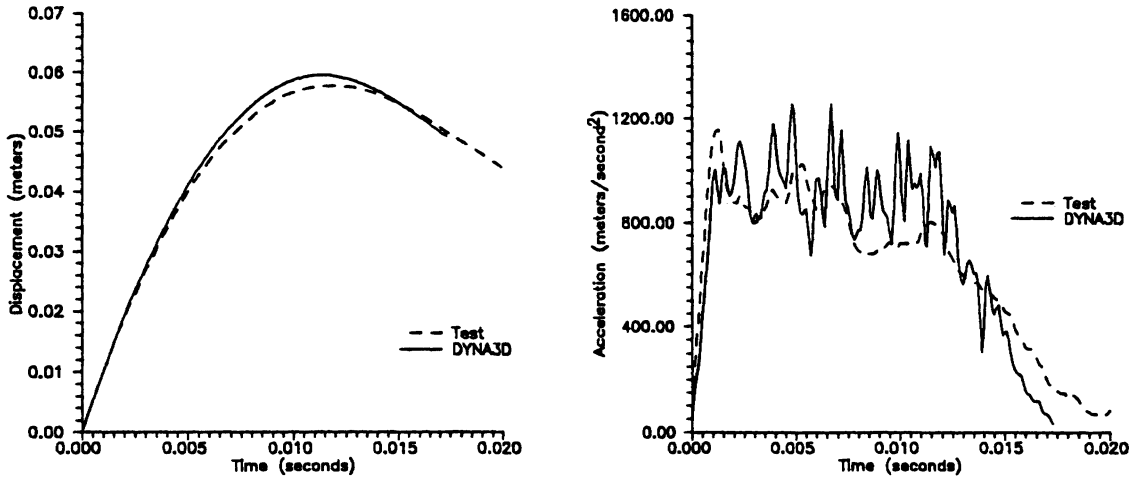

FIGURE 18 Taurus rear hood, modified fire wall: head form displacement and acceleration

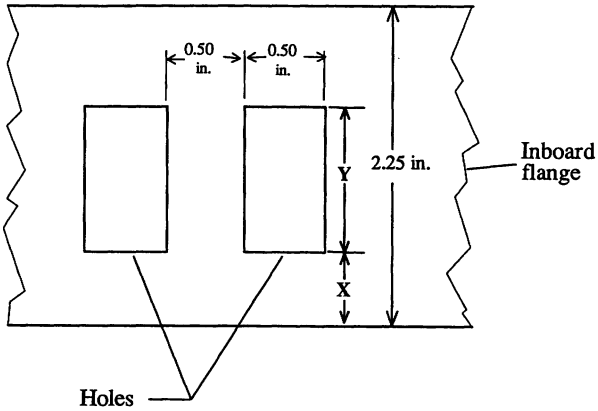

FIGURE 19 Two parameters for hole pattern design study.

To answer these questions two parameter studies were conducted. In the first study the flange hole shape was adjusted by changing the distances X and Y as in Fig. 19. In the second study the fender thickness was changed. In both studies the HIC was used as the evaluation criterion.

The first study shows how holes in the inboard flange affect the acceleration while the flange is collapsing. Preliminary analyses showed that holes wider than 0.5 in. weakened the flange so that the head impacts the apron rail. A fender with holes spaced farther apart than 1.0 in. center-to-center was analyzed with impact at a slightly different position along the hood–fender seam, and HIC values were found to vary with position. To avoid these ill effects, holes 0.5 in. wide with centers 1.0 in. apart were chosen for the study. Figure 20 shows a contour plot of HIC versus the parameters X and Y, obtained from DYNA3D results. Holes 0.90 in. high placed 1.35 in. above the base of the inboard flange produced the lowest HIC value.

Finally, the metal thickness of the flange (without holes) was varied for a fender without holes and resulting HIC values were determined. Figure 21 shows the results. A 0.030 in. (0.76 mm) thickness produced the lowest HIC value.

DISCUSSION AND CONCLUSIONS

Several conclusions can be reached from these findings. First, the analysis and results show that

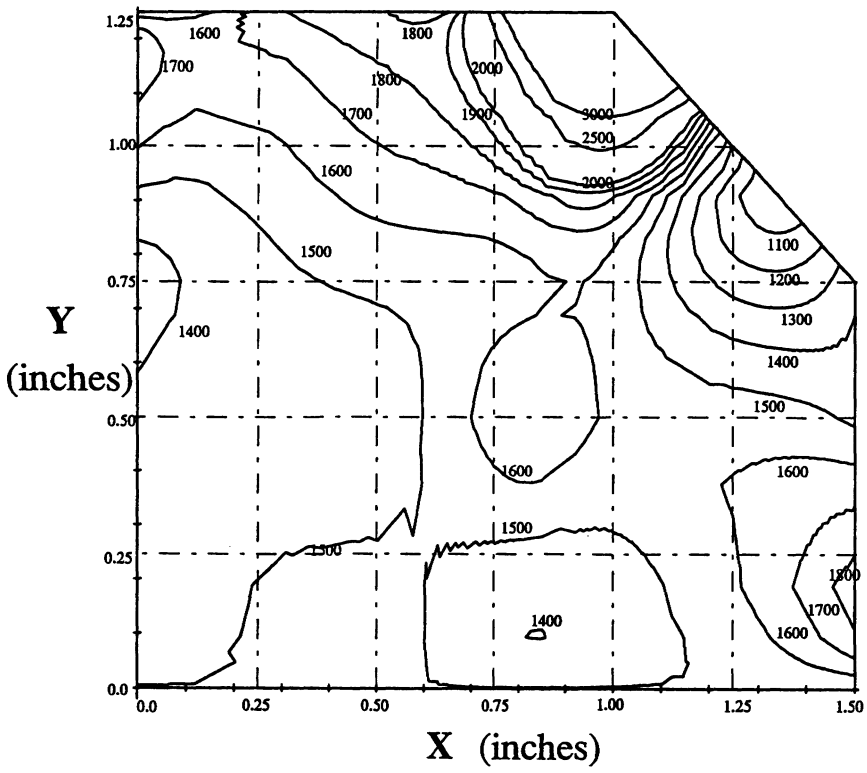

FIGURE 20 Contour plot, curves of constant HIC while varying two parameters: X (distance from base of inboard flange to the bottom of the hole) and Y (height of the hole). The parametric values for the optimal design are X = 1.35 in., Y = 0.09 in.

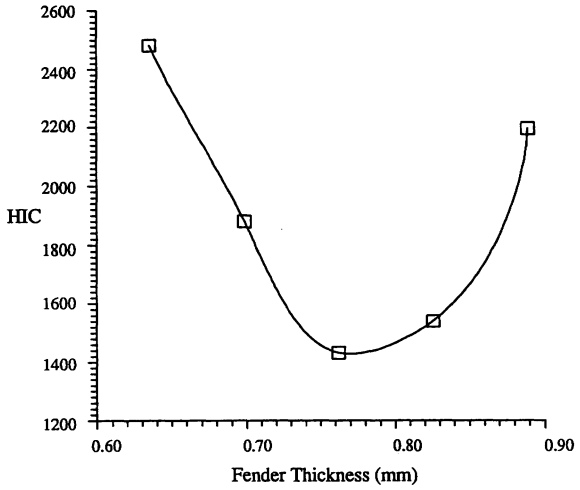

FIGURE 21 Spline curve fit of HIC versus fender material thickness.

hood/fender design can greatly affect HIC values. Applying the already established technique of placing holes in the fender flange can greatly reduce HIC values without significantly altering the overall structural integrity of the fender.

Second, the fender flange thickness has a critical or optimal thickness for minimal HIC values. If the flange is too thin, the head form strikes the apron nail. If the flange is too thick it does not collapse effectively.

Third, the fender flange hole shape and positioning leading to the lowest HIC values are such that the holes are relatively tall, close together, and high above the apron rail. This gives the most effective deceleration reduction for the head form.

Fourth, the studies show relatively close agreement between the experimental data and the numerical analysis (see Fig. 5). The validity of the DYNA3D analysis is also established by the ANSYS data. The slightly higher displacement and acceleration values for the numerical simulation (as opposed to the experimental values) in Figs. 12–18 may be due to the absence of the protective head form cover in the FE model. Experiments suggest however that the absence of the protective cover produces little change in the response of the automobile structure.

Fifth, the optimal design was obtained for the hood–fender interface, which contains an inboard flange 2.25 in. high with holes 0.5 in. wide, spaced at 1.0 in. intervals, and 0.9 in. high, located near the top of the flange. Other design requirements such as manufacturability, durability, and cost still must be addressed before the concept can be incorporated into a given vehicle design. However, the benefits in reduced head injury are expected to be very significant.

REFERENCES

Hallquist, J. O., and Benson, D. J., 1989, *DYNA3D—An Explicit Finite Element Program for Impact Calculations, Crashworthiness and Occupant Protection in Transportation Systems,* AMD-Vol. 106, BED-Vol. 13, American Society of Mechanical Engineers, New York.

MacLaughlin, T. F., Wiechel, J. F., and Guenther, D. A., 1993, *Head Impact Reconstruction—HIC Validation and Pedestrian Injury Risk,* SAE International Congress and Exposition, Society of Automotive Engineers Special Publication 946, pp. 175–183.

MacLaughlin, T. F., and Kessler, J. W., 1990, *Pedestrian Head Impact Against the Central Hood of Motor Vehicles—Test Procedure and Results,* SAE Transactions, Vol. 99, Section 6, pp. 1729–1737.

MacLaughlin, T. F., Hoyt, T. A., and Chu, S. M., 1987, *NHTSA's Advanced Pedestrian Protection Program,* Proceedings of the 13th International Technical Conference on Experimental Safety Vehicles (ESV).

Millaris, A. C., Hitchcock, R., and Hausen, M., 1985, *Harm Causation and Ranking in Car Crashes,* SAE Paper No. 850090.

Newman, J. A., 1980, *Head Injury Criteria in Automotive Crash Testing,* SAE Paper 8801317, Proceedings of the Twenty-Fourth Stapp Car Crash Conference, pp. 703–747.