BUMPER DESIGN USING COMPUTER SIMULATION

HWA - WON LEE
Senior Research Engineer, M.Sc.

SUNG - KUK JANG
Senior Research Engineer, Ph. D

Vehicle R&D Department
SsangYong Motor Company
448 - 2, Togok - 2dong, Kangnam - gu
Seoul, KOREA

ABSTRACT

In order to meet the current safety standards, it is necessary that a series of destructive tests for new vehicles including automotive bumpers to be performed. These tests are very expensive and time consuming. Therefore, the necessity of economical design and analysis using finite element method is increasing day by day.

This paper attempts to present such a design and analysis method, using relatively simple beam model and fine meshed shell model. The analysis has been performed for center - center pendulum and barrier hits of a bumper system of XENOY^{TM} 1102 newly developed by GEP (General Electric Plastics US) under the process of injection molding.

Finally, the results predicted from finite element method are compared with those of experimental tests to evaluate the analysis procedure.

INTRODUCTION

Automotive bumper system plays a very important role not only in absorbing impact energy (original purpose of safety) but also in a styling stand point. A great deal of attention within the automotive industry has been focused upon light weight and sufficient safety in recent years. Therefore, the bumper system equipped with thermoplastic and energy absorbing element is a new world trend in the market.

The major point for the design of bumper system is summarized as a degree of absorption of impact energy in a limited clearance between back face of bumper and body parts of vehicle. While experimental test is rather costly and time consuming, finite element analysis helps engineers to study design concept at an early design stage when prototypes are not available.

This paper attempts to show a method using computer simulation which has been broadly adapted in the various design stages of vehicle development. The analysis based on the international safety standards [1] is divided into two categories, the one is a pendulum type impact and the other is a barrier type impact using linear beam and / or thin shell element in implicit and/or explicit crash codes such as ABAQUS [2] and MSC / DYNA [3].

The analysis and experimental tests results are compared under FMVSS (Federal Motor Vehicle Safety Standard) Part 581 which is to provide protection against damage affecting the performance of external lamps and other components.

DESIGN AND ANALYSIS PROCEDURE

The objective of a bumper system design and analysis procedure is to develop a system which satisfies the safety requirements with the minimum weight. Commercial finite element packages, such as ABAQUS and MSC/DYNA, are used to evaluate displacements, strains, decelerations, and forces of the bumper system.

The design and analysis procedure is divided into two principal stages ; a concept phase analysis of the bumper system and a detailed phase which covers a dynamic aspect of the bumper system with highly geometric and material non - linearity.

Figure 1 shows a design and analysis procedure for the bumper system.

FINITE ELEMENT MODEL

Figure 2 shows a sectional view of bumper system for the analyses. The configuration of fascia is determined in vehicle styling view point, but is not designed to absorb the impact energy, meanwhile the back beam plays an important role in maintaining stiffness during impact.

PENDULUM TYPE IMPACT - The simple beam model utilizes the beam and gap elements to simulate the structural interactions of bumper system and pendulum. Figure 3 shows a simple beam model of bumper system and only the left half of the system has been modeled to reduce the calculation time.

Instead of modeling the right side, the symmetric boundary condition is applied on the center line of the vehicle. This technique does not effect the accuracy of the simulation since the deformation mode of the bumper system is symmetric for the analysis.

The rigid beam element is used for the pendulum. Gap elements are used for the interaction between bumper and pendulum. Linear beam elements are used for the back beam and supporting brackets.

This model has 24 elements and takes 148 CPU seconds on VAX 6410 VP upto 70 msec using ABAQUS. The maximum deformed shape at 34.0 msec is displayed in Figure 4.

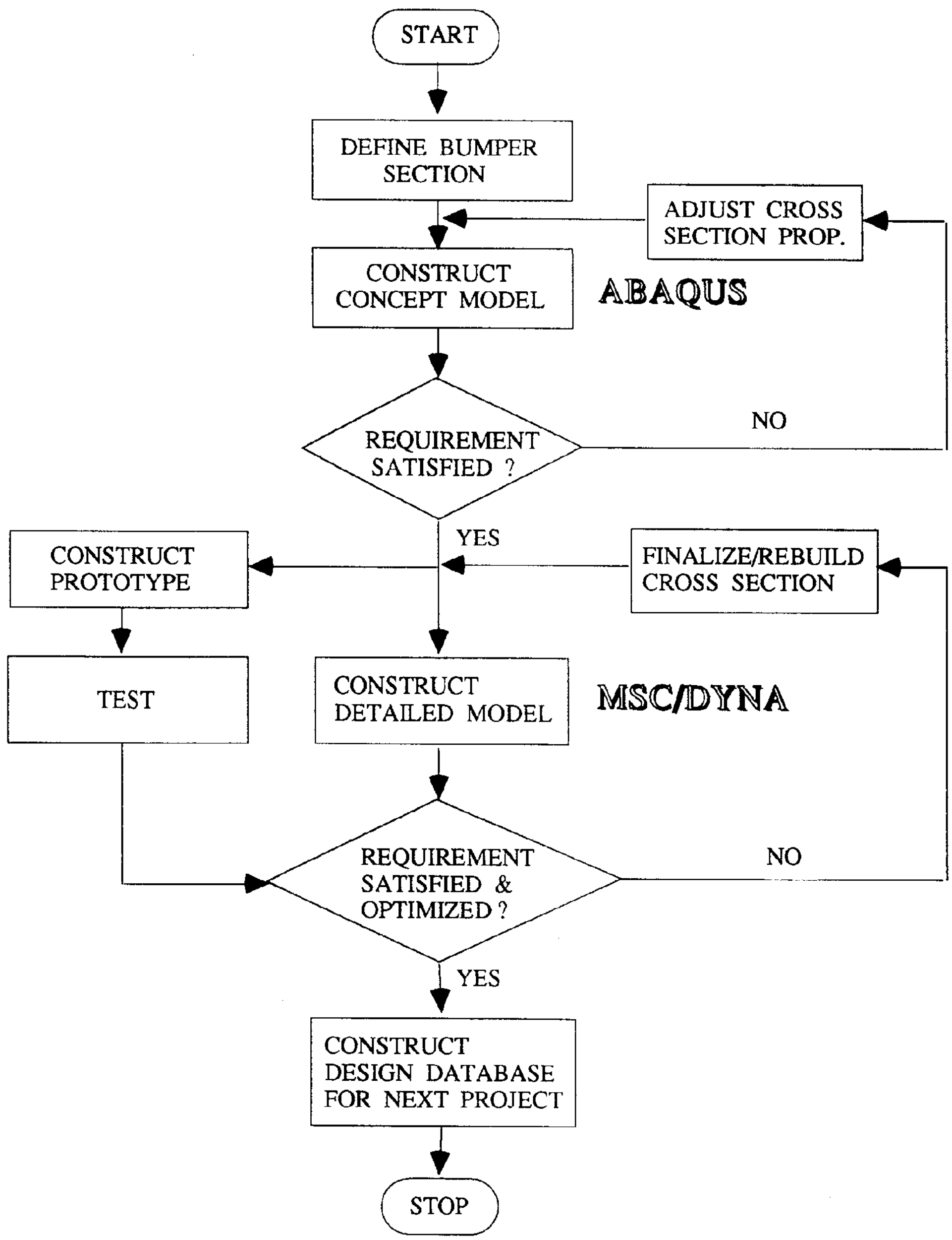

Figure 1 Design and analysis procedure of bumper system

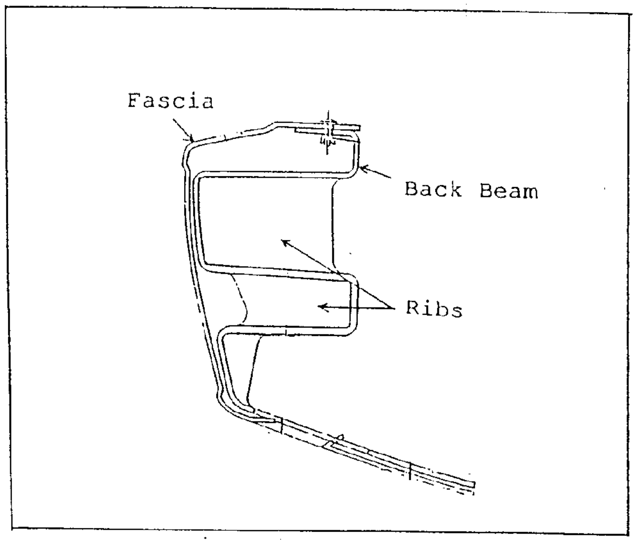

Figure 2 Sectional view of bumper beam

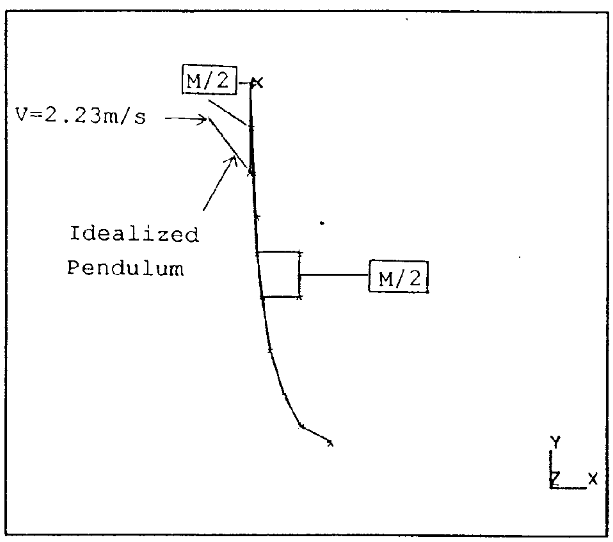

Figure 3 Simple beam model of bumper system (pendulum hit)

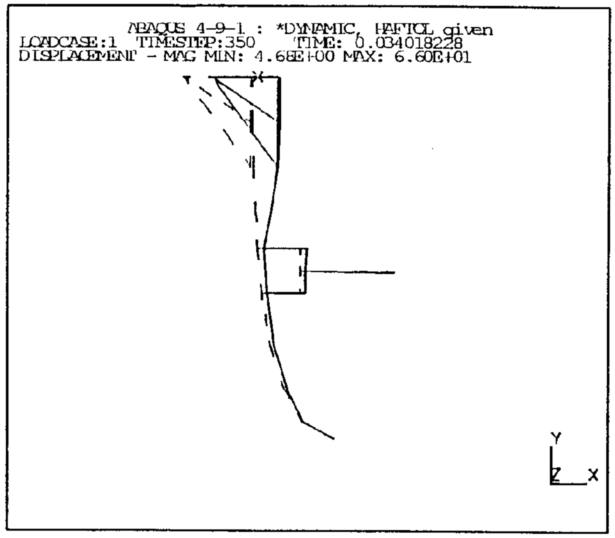

Figure 4 Maximum deformed shape - simple beam model (pendulum hit)

Figure 5 shows a detailed model for the bumper system in I - DEAS [4]. The back beam of XENOY™ 1102 consists of shell elements with 4 mm of average gage thickness. The steel side member is also modeled by quadrilateral thin shell element. The pendulum consists of 380 solid elements. In order to prevent the penetration between the pendulum and bumper system, contact surfaces for back beam and pendulum have been defined.

The solution time takes 241 592 CPU seconds on VAX 6410 VP using MSC/ DYNA. The maximum deformed shape at 39.5 msec is displayed in Figure 6.

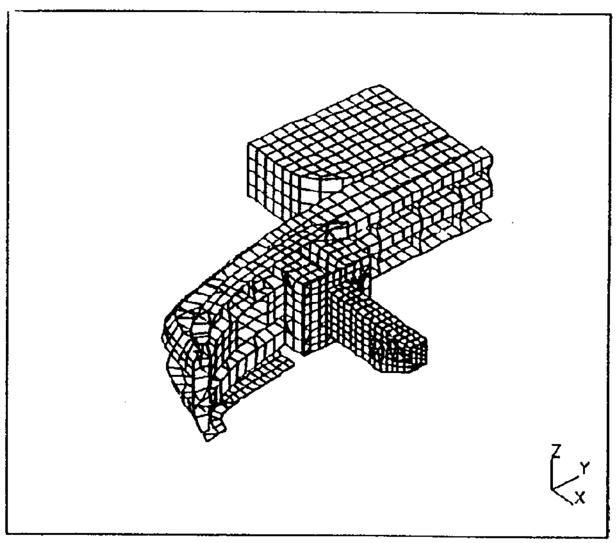

Figure 5 Detailed model of bumper system (pendulum hit)

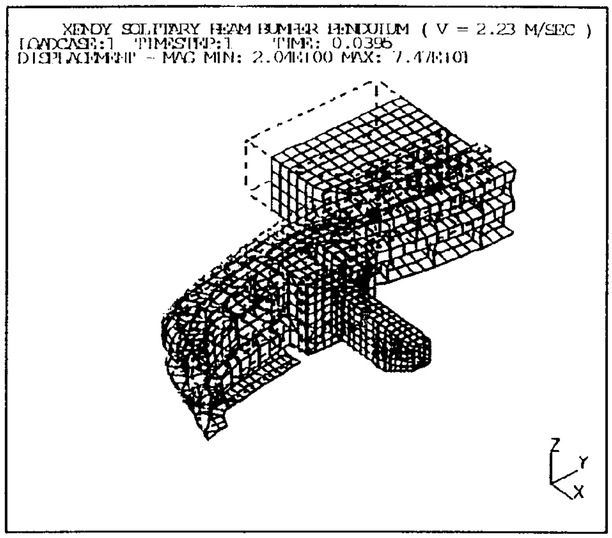

Figure 6 Maximum deformed shape - detailed model (pendulum hit)

BARRIER TYPE IMPACT - A detailed model for the barrier type impact with energy absorbing back beam is shown in Figure 7. This model is composed of 1831 elements. The element size around the back beam is approximately 7 × 7 mm. A rigid wall has been defined for the fixed barrier and the simple beam model also utilizes the beam and gap elements as shown in Figure 8.

The solution time took 189753 CPU seconds upto 60 msec on VAX 6410 VP using MSC / DYNA. On the other hand, the solution time for the simple beam model took 134 CPU seconds.

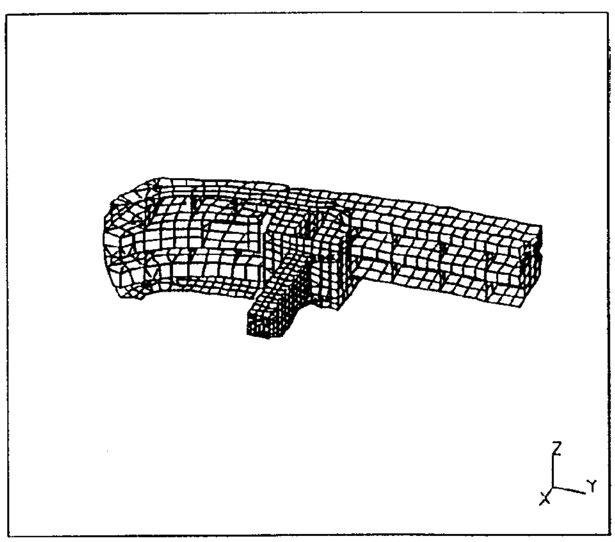

Figure 7 Detailed model of bumper system (barrier hit)

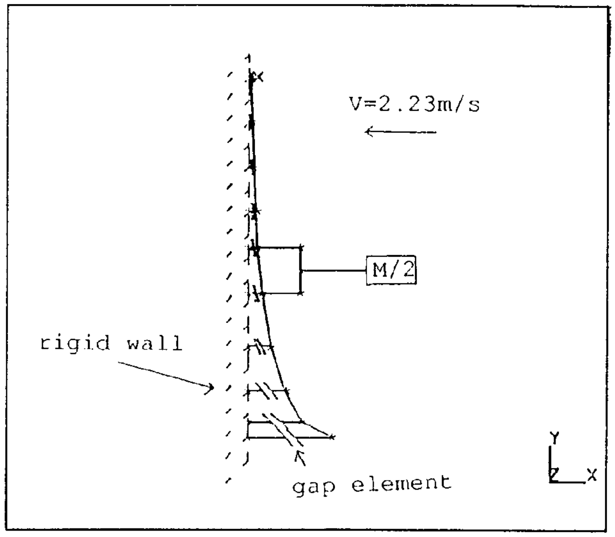

Figure 8 Simple beam model of bumper system (barrier hit)

RESULTS AND DISCUSSION

This paper has been focused on effective modeling for the bumper system at the initial design stage by comparing simple beam model results and detailed shell model results with experimental test.

Figures 9 through 12 show the time histories for vehicle deceleration, displacement of the simple and detailed models, and experimental test. The maximum values for displacement, deceleration, and force are listed in Table 1.

Table 1 Maximum displacement, deceleration and force

Description		Displacement (mm)	Deceleration (g)	Force (kN)
Pendulum	Simple	49.6	5.6	72
	Detail	56.8	4.4	57
	Test	51.6	4.4	57
Barrier	Simple	69.8	9.2	118
	Detail	65.1	7.5	96
	Test	64.5	8.2	106

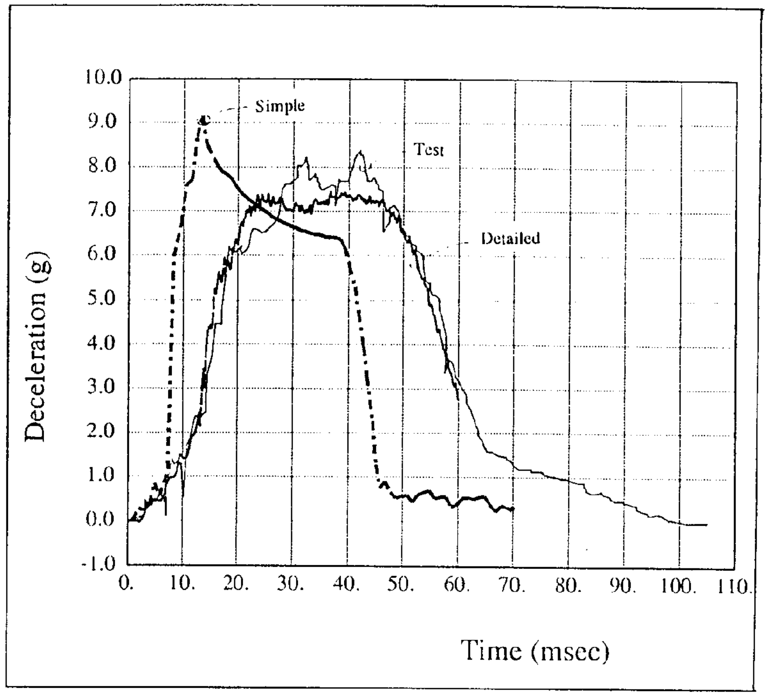

Figure 9 Deceleration vs. time (barrier hit)

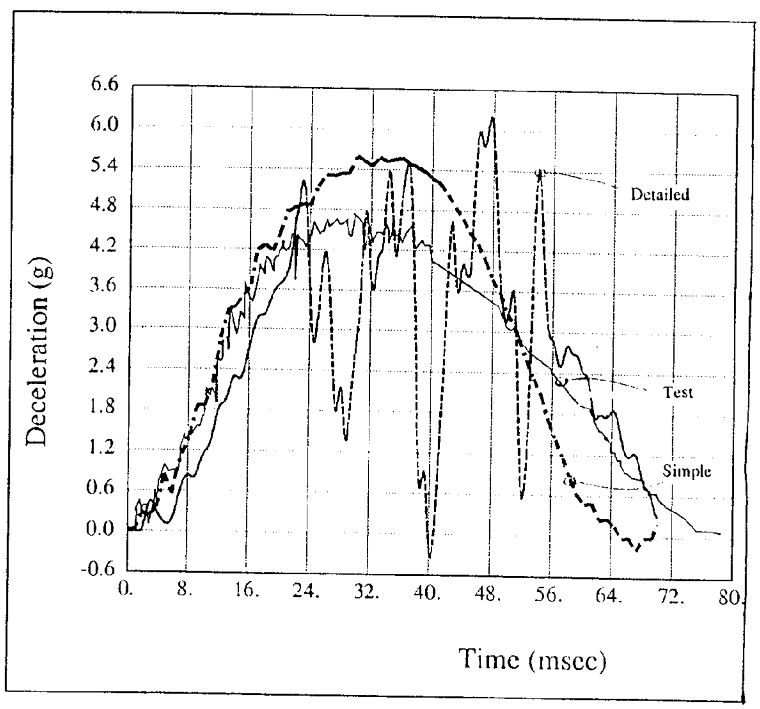

Figure 10 Deceleration vs. time (pendulum hit)

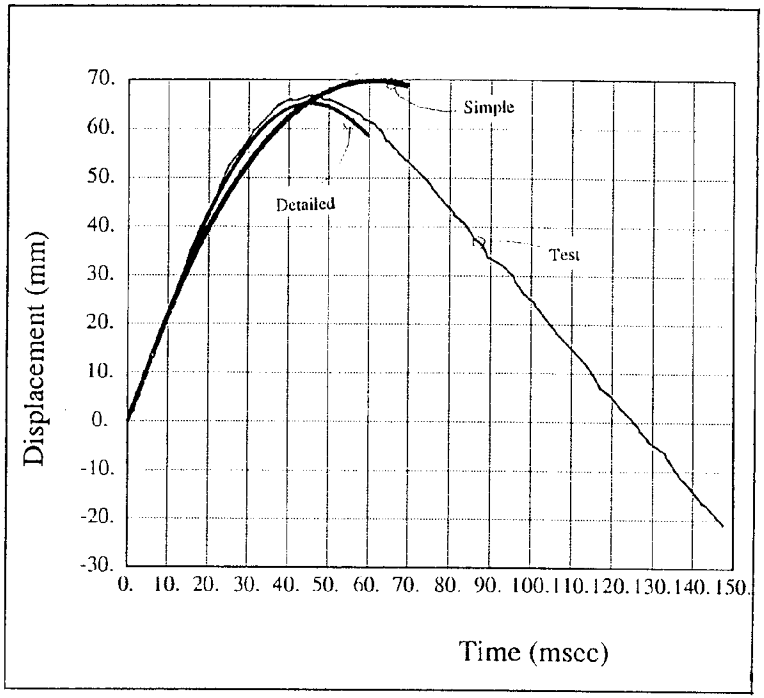

Figure 11 Displacement vs. time (barrier hit)

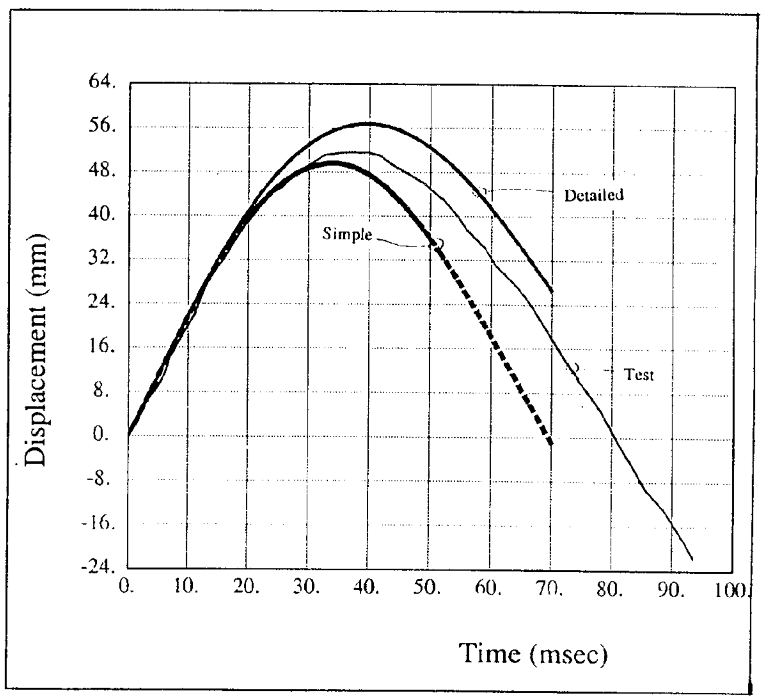

Figure 12 Displacement vs. time (pendulum hit)

The deviation between simple model and test for the barrier impact is 8.2 % for maximum displacement, 6.4 % for maximun deceleration, and detailed model is 0.9 % and 9.3 %.

Figures 13 and 14 show the absorbed energy for simple, detailed models, and experimental test. For the analysis of a simple beam model, a special consideration has been accounted for. That is, a separate fine shell model for the tower has been analyzed to obtain the axial force vs. displacement characteristic of the tower. The axial force vs. displacement of the tower has been applied in the beam model.

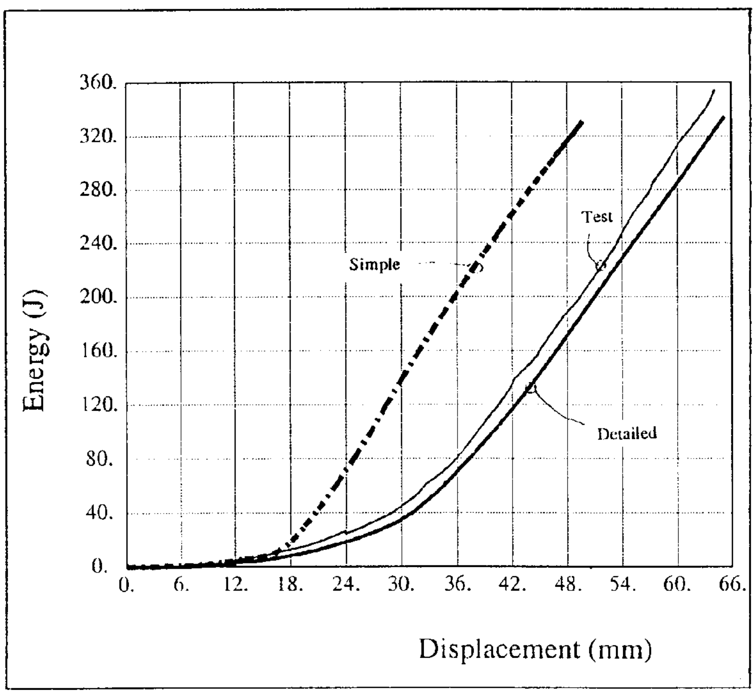

Figure 13 Absorbed energy (barrier hit)

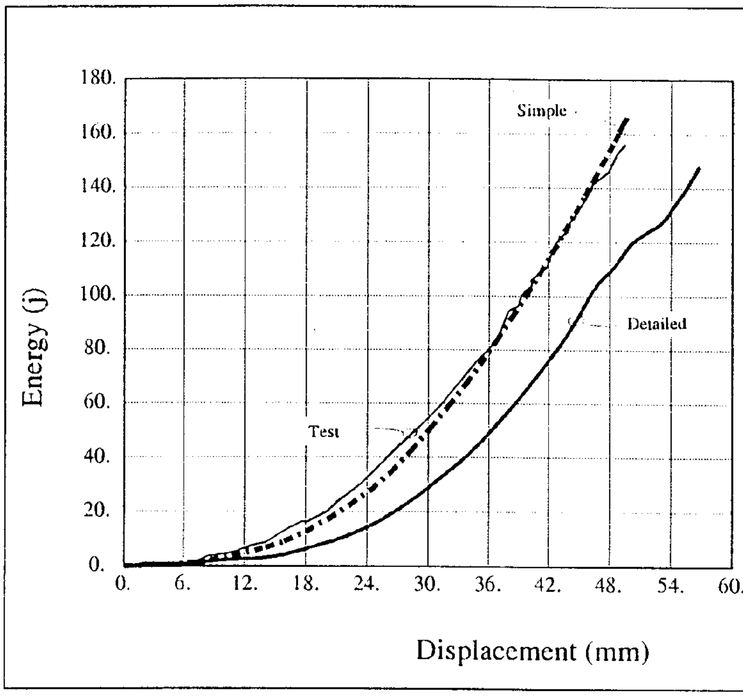

Figure 14 Absorbed energy (pendulum hit)

CONCLUSION

Two types of impact simulation method for the bumper system has been discussed and results presented in this paper were experimentally verified.

1) The finite element method using simple beam element which is more economical than using shell element can be successfully applied to the bumper impact analysis at an early design stage.

2) In order to obtain useful results using simple beam element, prior calculation of the axial force vs. displacement characteristic for the supporting tower should be provided.

3) The analyses for barrier and pendulum type hits using fine shell element in MSC/DYNA give excellent corelation with the experimental tests. Accordingly, it is possible to reduce the number of experimental bumper tests at the design stage.

REFERENCES

[1] Bumper Evaluation Test Procedure - Passenger Cars
Federal Motor Vehicle Safety Standard, Part 581 Safety Regulation, US

[2] ABAQUS User's Manual , Version 4 . 7, Hibbit, Karlsson & Sorensen Inc, 1988

[3] MSC / DYNA User's Manual , DEC (VAX / VMS) Edition Version 2B, The MacNeal - Schwendler Corporation , CA , 1991

[4] I - DEAS User's Manual , Version 4 . 1, Structural Dynamics Research Corporation, 1989