SIMULATION OF CRASH-TEST USING THE ACCELERATION AS A PARAMETER TO CHECK THEORETICAL AND EXPERIMENTAL RESULTS

LUIZ CARLOS H. RICARDO

AUTOLATINA BRASIL S.A.
SÃO BERNARDO DO CAMPO - SP

OSCAR BRITO AUGUSTO

UNIVERSIDADE SÃO PAULO
SÃO PAULO - SP

ABSTRACT

A compact car was modeled using MSC/NASTRAN. The car model is composed by 6 beam elements and 2 rigid elements, also representing the engine and body masses, besides 2 other gap elements for mathematical reasons.

The hyphothesis was that a vehicle would make a crash-test at 14 m/s, with accelerometers placed in the tunnel. We have obtained, with this test, the curve acceleration versus time. These data were used for comparison with the MSC/NASTRAN model. Solution 129, transient non-linear response from version 67, was used.

Looking at the graphs and comparing the analysis, the MSC/NASTRAN versus experimental results are very close.

I - INTRODUCTION

Nowadays we already have tools able to simulate the impact of vehicles through finite elements with elasto-plastic behavior capacity and large displacements. As a matter of fact, we find in technical literature several problems where those elements have been utilized (vide refs. 1,3), nevertheless, if we adopt a similar procedure in the starting development stages of this research, we would lose the sensibility of the influence of each dynamic structural component in the final results of the simulation.

To facilitate the obtaining of preliminary crash acceleration figures, we will use a simplified model, in spite of the results doubtful precision. The global characteristics of the phenomenon obtained by this way would not damage and/or modify the sensibility stated before.

II - MODEL DESCRIPTION

The model elaboration with its respective elements is based in turning a real complex component into a beam model, determining the stiffness and the elasticity modulus of the same; this beam will work only with traction and compression. This way to a new project the methodology to be used at the drawing phases will be:

1º) Verify if the new project has, in general lines, the same characteristics of the previous one, such as: weight of vehicle, distance between axles, classified considering the type of vehicle, whether compact, standard, sedan or van and kind of engine, whether longitudinal or transversal.

2º) Select the similar characteristics of the previous project:

- verify with Design Department the first conception of front crossmember.
- estimate transversal section and component length.
- simulate through finite element method utilizing an element beam, the front sidemember, restricting in every direction an extremity, thus obtaining a displacement and with it estimate the stiffness and the elasticity modulus in the following way:

K = stiffness
Fu = force unit
Lc = length components
Atrc = transverse area of the component

$$K = \frac{Fu}{D} \; ; \; E = \frac{K.Lc}{Atrc}$$

Sensibility of Structure

- To the engine mount, transmission mount and dash panel, in case there isn't information in this phase yet, the data from the previous project can be used in a first moment to feel the structure behavior. Receiving the complementary information during the project we will improve the model with the missing information, assuring a better quality of results which will be compared with the previous project, to be verified if there are differences of one project related to the preceding one.

3º) If the new project doesn't have similar characteristics to the former one, then:

- Verify with the Design Department the first conception of the front sidemember and estimate the stiffness and the elasticity modulus using the already mentioned procedure.

Within these considerations the adopted hypothesis to the model creation are listed below:

- the structure will be represented by beam elements and gaps
- the engine and the body are not deformed and so will be represented by mass;
- only the frontal part of the vehicle will be modeled;
- for mathematics reasons a gap was placed, linking the front upper panel to the engine, to simulate the movement of the engine rigid body until it beats the impact barrier;
- the impact barrier is represented by a gap of # 6 which is linked to the node #1;
- the modulus of elasticity and the yield stress utilized here are not related to the material but to the individual components;
- the components' length, the mount of engine and the dash panel were adapted to the model;
- the frontal structure was modeled considering the frontal area that starts at the bumper until the windshield, including the hood, fenders and bumper (see fig. # 1).

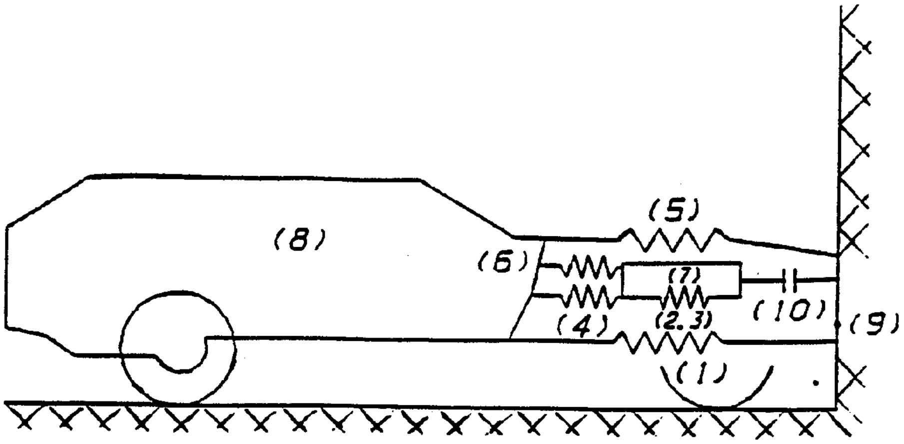


Fig. # 1 Mathematical model of vehicle to frontal impact analysis

Lettering:

(1)	Sidemember
(2,3)	Engine mount
(4)	Transmission mount
(5)	Frontal structure (hood, fenders and bumper)
(6)	Dash panel
(7)	Engine mass
(8)	Passenger compartment mass
(9)	Impact barrier
(10)	Upper front panel

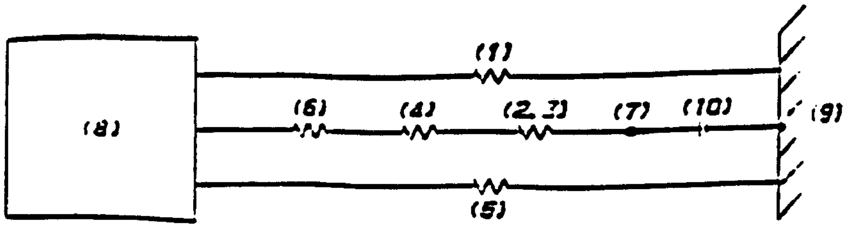


Fig. 2 Mathematical model of the vehicle

The simplification of the calculation of the model, if in one hand makes easier the understanding of the phenomenon, on the other hand it complicates an estimate of the elasto-plastic and geometric characteristics of the components.

These characteristics, in the present study, were experimentally chosen for each component. The obtained results will be presented below.

Chart 1.1 Stiffness data:

Component	Force (kN)			Displacement (1E-3 m)		
	Fi	Fint.	Ff	Di	Dint.	Df
engine mount	0	20	40	0	15	25
transmission mount	0	10	18	0	11	15
dash panel	0	15	18	0	10	15
sidemember	0	170	500	0	150	400
frontal structure	0	160	550	0	120	450

Obs.: The data above will be utilized to obtain the curve of force x displacement to the structural component that later will be transformed in a curve of σ vs. ε to the material utilized in the element of the model Conrod (MSC/NASTRAN).

Chart 1.2 Characteristics of Deformable Elements:

component	EL #	area (m^2)	length (m)	elast. model (N/m^2)	yield S. (N/m^2)
sidemember	1	5.55E-4	1.2	1.04E9	4.92E6
engine mount transmission	2,3	2.0E-4	0.5	1.67E8	4.92E6
mount frontal	4	4.0E-3	0.05	1.51E7	2.52E6
structure	5	1.50E-3	1.2	1.33E9	8.56E7
dash panel	6	4.0E-4	0.5	1.88E8	3.76E7

Not deformable Elements

element # 7 = engine mass
MASS = 100 kg
element # 8 = passenger compartment mass
mass = 900 kg
element # 9 = gap simulating impact barrier
stiffness = 1.0E12 N/m^2
element # 10 = gap linking front upper panel to engine
length = 0.2 m
stiffness = 6.4E4 N/m^2

Loading and Boundaries Conditions

We will consider that the model only will be able to dislocate in the direction x, restricting all other degrees of freedom. The model will be submitted to a initial velocity of 14 m/s and it will calculate displacement, velocity and acceleration during and after the vehicle impact at the barrier.

Thus we will obtain the curve acceleration vs. time, which will be compared to the results of curves obtained experimentally.

III - Solution Method

MSC/NASTRAN version 67, with solution 99, was used. It utilized the element CONROD to the model creation, this element has 6 degrees of freedom and works only with compression and traction. In our analysis we won't consider the buckling effect, torsion and flexion. It will only be considered the in x direction; as the frontal structure is projected using the theory of plasticity, it is necessary to adopt an interactive process where we can follow the development of the plastic deformation when it comes out the yield stress region. Our analysis utilized the Newton-Raphson Modified interactive method, which allows us to give small "time-steps" which were of 1.0E-3 during 200 ms, that is the total time of duration of crash test measured, thus obtaining an improvement in the quality of results.

IV - EXPERIMENTAL AND THEORETICAL RESULTS COMPARISON

We will describe below a crash-test summary, as well as its steps related to the chronological participation of each component in the event. Besides, the test results will be compared to the results of theoretical models and, in the end, it will be commented the relevance of the theoretical results facing the experimental ones.

The vehicle is pulled by a traction cable until it develops a constant velocity, then the traction cable is interrupted and the vehicle moves through inertia to the impact barrier.

Immediately after the impact of the vehicle to the barrier, the front structure starts to dislocate followed by the sidemember and keeps dislocating until the engine is met. From that moment the displacement continues increasing but more slowly, achieving a maximum value. This moment characterized by the rebound of the vehicle from the wall until stopping.

With the objective of comparing experimental and theoretical results, two analysis with the same model were made, with which theoretical accelerations were obtained and compared to the real ones, as demonstrated in the figures # 5 and 6.

The first analysis was done considering the elasto-plastic model or, in the beginning the structure suffers deformation in the plastic region and after achieving the proportionality limit, the material comes into the plastic regimen thus obtaining an increasing and constant deformation (see fig. # 3).

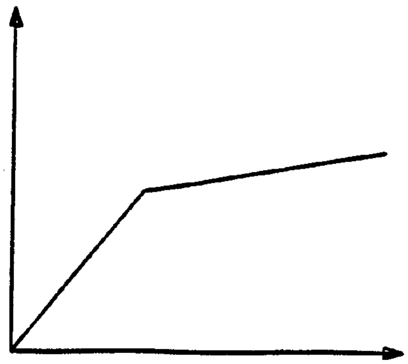


Fig. 3 Typical curve of elasto-plastic material

The other analysis was made considering material with perfectly elasto-plastic behavior. In this case the material suffers deformation in the elastic region and when it achieves the proportionality limit it presents constant tension (see fig. # 4).

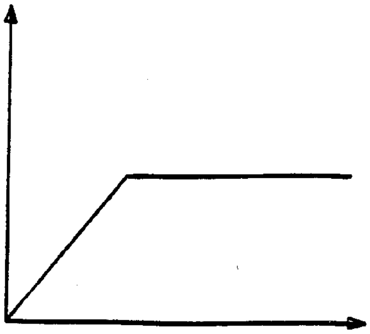


Fig. 4 Typical curve of perfectly elasto-plastic material

In the two analysis the Newton-Raphson modified method was used (as stated before) to verify which of the models has a better representation in front of the results of the experiment. The results to both cases are present with the experimental results in the figures 5 and 6, respectively.

The experimental and theoretical accelerations were compared in the tunnel, because, as already stated, this is the region of the vehicle that doesn't suffer deformations that could damage the qualities of the measured vehicles in the test.

Besides this, the tunnel is placed within the passenger compartment, which accelerations should be controlled.

Observing the theoretical models (fig. 5 and fig. 6), we note a reasonable adherence between experimental and theoretical results.

We also verify that the models with the proposed discretization, filtered the high frequency accelerations derived, probably, from the local dynamic characteristics of the structure where the accelerometer was fixed (tunnel). Despite the high frequency components, present at the experimental results, both the elasto-plastic and perfectly elasto-plastic models have a reasonable adherence related to the experimental curve, mainly in the rate of 0 to 60 ms the results of the models are very close to the experimental curve. Nevertheless in the rate of 80 to 200 ms both theoretical curves deviate from each other. We believe that one of the reasons to this deviation is related to the rebound of the vehicle from 100 ms approximately. After the vehicle crash at the barrier, the tires are arrested to the wheel house, making the vehicle absorb another parcel of energy, increasing the deceleration.

The divergence obtained in the theoretical and experimental results is related to factors that we aren't able to reproduce in the theoretical model, among them we can mention the rebound of the vehicle after the crash at the impact barrier. Even so we believe that the proposed methodology is a valuable tool to the structural simulation of the vehicles crash-test.

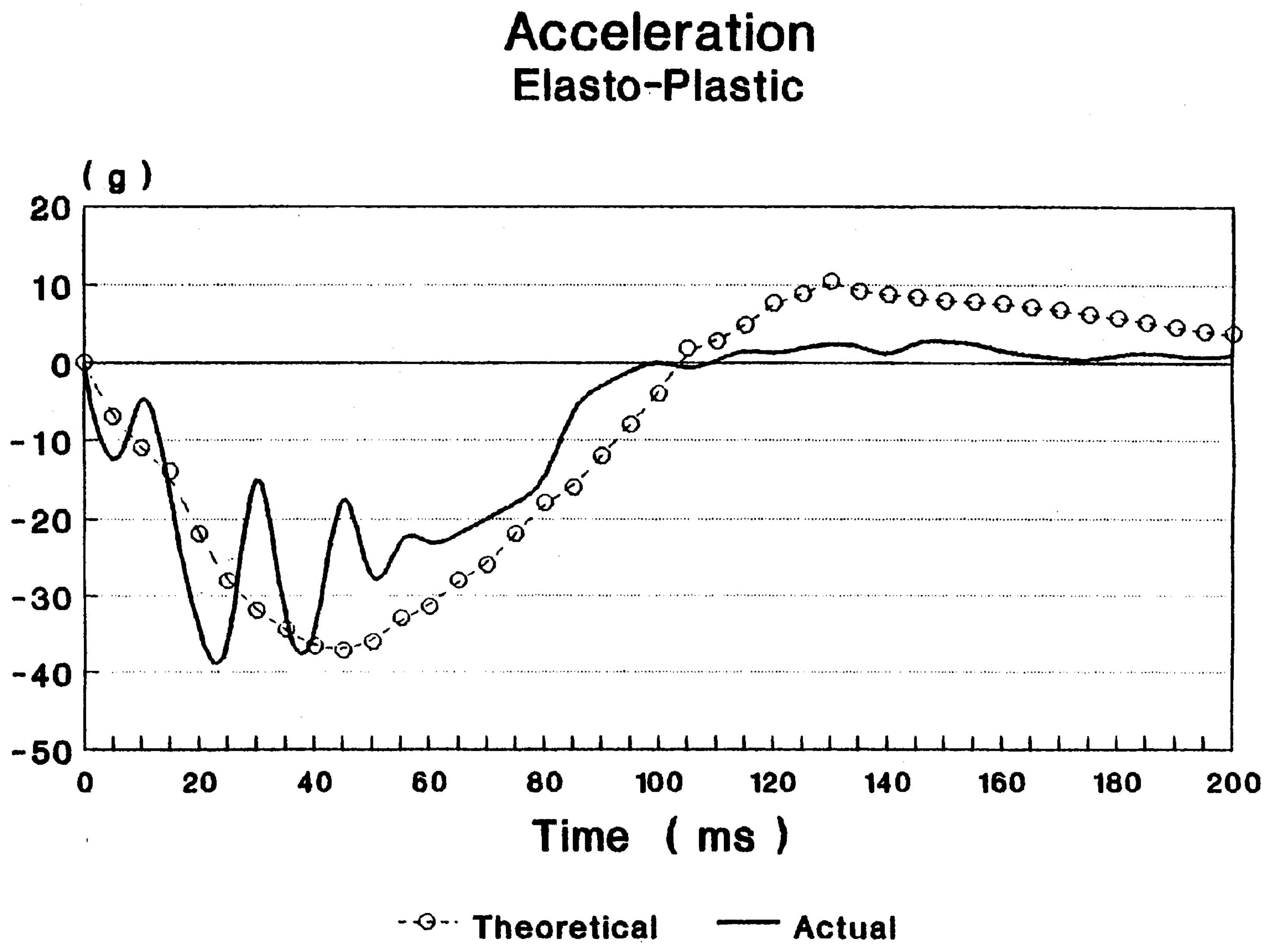
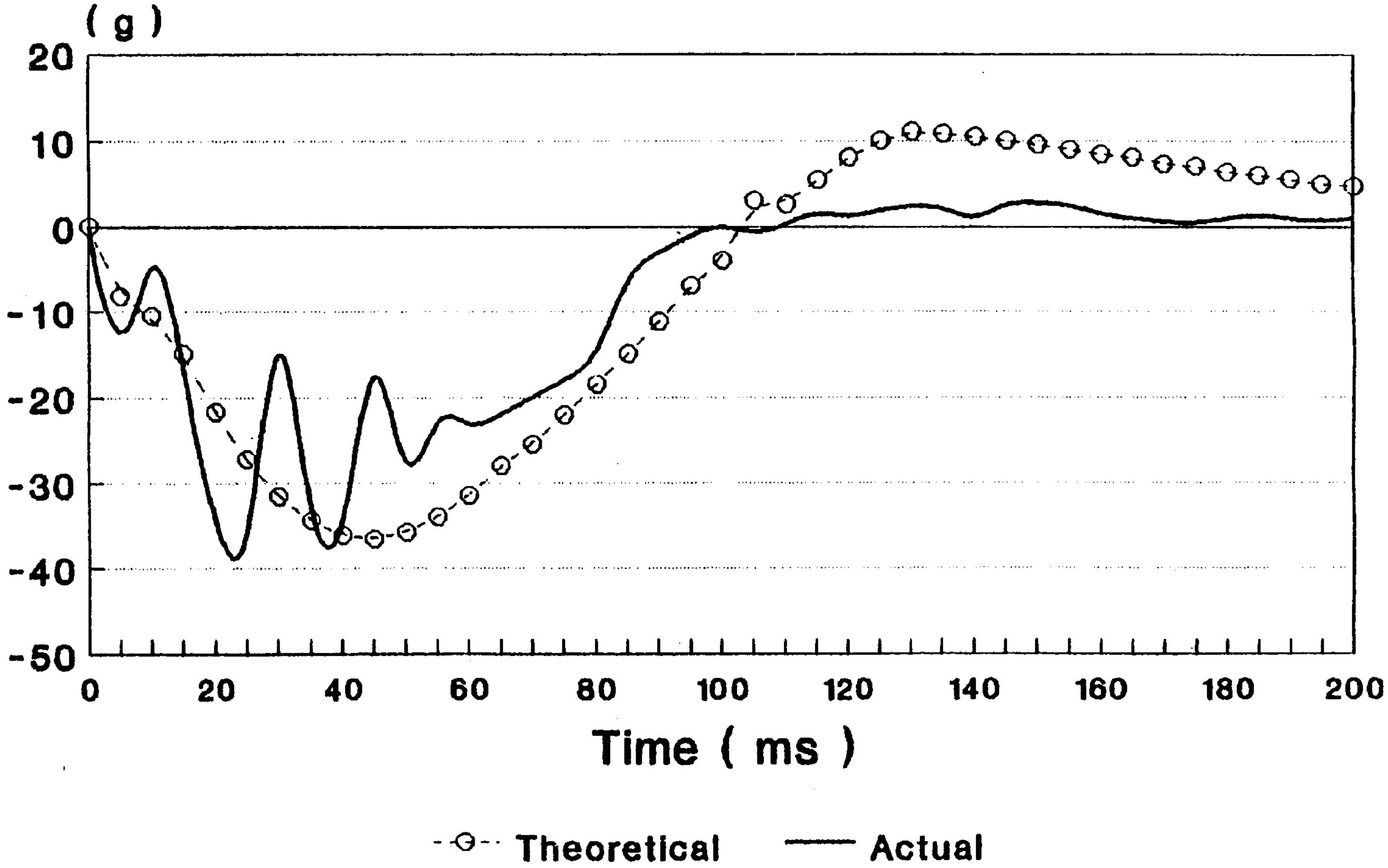

Acceleration
Elasto-Plastic
(g)
20
10
0
-10
-20
-30
-40
-50
0
20
40
60
80
100
120
140
160
180
200
Time (ms)
Theoretical
Actual

Figure 5



Acceleration
Elasto-perfectily Plastic

Figure 6

V - Conclusion

Considering that it is a simplified model, we can say that the obtained results are qualitatively good, with relatively little calculation, not to mention the costs of a prototype test, results can be obtained which may indicate, still at the board, undesirable characteristics in the behavior of the vehicle at an impact situation.

With this work we had the intention of developing a project methodology where in general lines it is possible to evaluate the structural behavior of a new vehicle project, at an impact situation, before doing a prototype crash-test.

VI - REFERENCES

[1] Kamal, M.M. "Analysis and Simulation to Barrier Impact", SAE Paper nº 700414, 1970

[2] Lin, K. H. "A Rear-End Barrier Impact Simulation Model for Unobody Passenger Cars", SAE Paper nº 730157, 1973

[3] Thompson, J.E. "Vehicle Crash Prediction Using Finite Element Techniques", SAE Paper n 730157, 1973

[4] Butkunas, A. A. "Analysis of Vehicle Structure", I. J. Vehicle Design, Vol. 5, nºs 1/2, PP 3-25, 1984

[5] Wilson, R.A. "A Review of Vehicle Impact Testing: How it Began and What is Being Done", SAE Paper nº 700403, 1970

[6] Chang, D. C. "A Design-Analysis Method for the Frontal Crush Strength of Body Structures", SAE Paper nº 770593, 1977

[7] Owen, D. R. J. and Hinton, E. "Finite Elements in Plasticity", Pineridge Press Limited, Swansea, V.K., 1980

[8] Shieh, R.C. "Some Analytical Crashworthiness Studies and Automobile Front Structures", SAE Paper nº 730612, 1973

[9] Anderson, W.J. and Mclvor, I.K. "Dynamic Validation of a Computer Simulation for Vehicle Crash", SAE Paper nº 770591, 1977

[10] Grosh, L. Kanser, H.P. and Schmid, W. "Mathematical Movement and Load Simulation for Persons Involved in an Automobile Accident", I.J. Vehicle Design, Special Issue on Vehicle Safety, pp 41-48, 1986

[11] Henson, S.G. "Computer Modeling of Inrusion Effects on Occupant Dynamics in Very Severe Frontal Crashs", SAE Paper nº 830613, 1983

[12] Ricardo, L.C.H. "Simulação de Crash-Test pelo Método de Elementos Finitos" (in Portuguese), Second Brazil User's Conference in São Paulo, March 2-3, 1993

[13] Zienkiewicz, O.C. "The Finite Element Method", Migraw Hill, London, 1985

[14] MSC/NASTRAN, Theoretical Manual, version 67