

## PARAMETRIC FINITE ELEMENT MODEL OF A SPORTS UTILITY VEHICLE - DEVELOPMENT AND VALIDATION

Gustavo A. Aramayo  
Computation Materials Science Group  
Computer Sciences and Mathematics Division  
Oak Ridge National Laboratory  
Oak Ridge, Tennessee 37831  
aramayoga@ornl.gov

Matthew H. Koebbe  
XYZ Scientific Applications, Inc.  
1324 Concannon Blvd.  
Livermore, California 94550  
(925) 373-0628  
koebbe@truegrid.com

Abbreviations used within this paper:

2WD – 2 wheel drive  
4WD – 4 wheel drive  
SUV – Sports Utility Vehicle  
IIHS – Insurance Institute for Highway Safety  
NCAP – New Car Assessment Program

Keywords: crashworthiness, LS-DYNA, model development, parametric mesh development, TrueGrid

### ABSTRACT

As part of the NCAP (New Car Assessment Program) a finite element model of a Ford Explorer SUV has been developed using a formulation that results in a model with arbitrary element size and element size distribution. This model enables the developer and analyst to choose a finite element model most applicable to the specific crash scenario under study. The element size and element distribution are parametrically defined at the time of the model generation.

**Comment:** Make sure that this is the correct Abstract.

The general model is verified against several specific models with different element size densities and distribution, and compared with experimental results of crash tests. In the analytical study, models are generated for several crash scenarios and for several degrees of vehicle engagement with the stationary obstacle.

## INTRODUCTION

As computers continue to become faster, with more RAM and larger hard drives, finite element vehicle models provide an increasingly effective alternative to experimentally testing the integrity of passenger vehicles. Mathematical models of passenger vehicles have evolved both in complexity and in size. The number of elements in a typical, single vehicle impact event model ranges from less than 50,000 elements to 500,000 elements. The advent of massively parallel computers provides a justification for the use of larger, more detailed vehicle models.

But computer resources are still limited as we tackle larger problems such as modeling multiple full vehicle crashes in an effort to better understand the effect on the passengers inside. A finite element mesh created for such a simulation needs to be refined in the region of impact and can transition to be coarser elsewhere. The need to study impact scenarios where the vehicle impacts targets with different orientations and degrees of engagement necessitates the development of models that specifically address these scenarios. Typically, there have been only 2 alternatives: 1 large, highly refined model or a different finite element model for every crash scenario. Neither option is attractive.

Multiple vehicle impacts with either full or partial engagement result in coupled models that may have a total size of 500,000 to 700,000 elements or more. Including the occupants in a vehicle model analysis requires detailed modeling of the interior regions of the car in order to obtain realistic results. A highly refined model of 2 vehicles, while still representing the passengers as rigid bodies, can easily range in size to 1,500,000 elements and beyond. Few people have access to a computer that can analyze such a mesh.

It is clear then that there is a need to develop individual models that address specific crash scenarios. However, there are an ever-increasing number of scenarios that need to be modeled as we wish to know the effects of frontal, side and rear impacts with other vehicles, poles and barriers with varying offsets and incoming angles. Creating an autonomous finite element mesh for each and every scenario is a daunting task.

This paper presents the methodology used in the development of a parametric finite element model of an SUV (Ford Explorer). This model is sufficiently flexible so that, with minor changes in the input at the model generation level, any degree of detail and complexity can be incorporated into each predefined region of the model. This enables us to address very specific crash scenarios with a single input file and a suitable set of parameters.

The finite element model generation is performed using the preprocessor program *TrueGrid* (XYZSA, 2000), the analysis package LS-DYNA (LSTC, 1995), and the post processing package LS-POST (LSTC, 1999b). These softwares are readily available commercially and require no customization. The pre- and post-processing hardware consists of an O2 Silicon Graphics workstation. The actual computation is performed on an SP2 IBM-6000 computer at the Oak Ridge National Laboratories.

The results of our experimental and simulation effort are laid out herein, the basic concepts are discussed, and the concept of parameterization as used in this effort is presented. Different frontal impact results are presented for varying degrees of complexity in the model. These results are briefly compared against experimental tests and, finally, models developed for other than frontal impact cases are presented.

## METHODOLOGY

The development of the finite element model was performed using the program *TrueGrid*, version 2.0.6. *TrueGrid* is an interactive and batch pre-processor which utilizes a top-down modeling approach appropriate for predominately quad and hex mesh generation. It generates multiple block structured meshes. Each block is bounded by partitions, which correspond to a set of reduced indices in the computational space. The arrangement of the multiple blocks is known as the mesh topology. Hexahedral elements may be degenerated into wedges or tetrahedrons. Quadrilateral shells may be degenerated into triangles.

Our model surfaces are obtained from the measurement of the actual part geometry. The vehicle is completely disassembled so that it is physically possible to access each component for correct measurement. The data is subsequently converted into polygonal surfaces. The polygonal surface data is composed of two files, one identifies nodal coordinates and the other defines the nodal connectivities that form polygonal surfaces. The data format is

similar in character to a typical finite element data set. Once these surfaces are read into *TrueGrid*, the software employs the projection method to mold a block mesh to the desired shape. In particular, the program constrains mesh facial nodes to lie on surfaces, guaranteeing that the mesh represents the actual part geometry.

Finite element models that are used in the analysis of passenger vehicles primarily use shell elements, so we will illustrate the concepts described above with a quadrilateral shell example. In this case, the target surface is a polygonal surface dataset representing a recognizable component of a well-known passenger vehicle. Only one entered *TrueGrid* command and a few clicks of the mouse are needed to define the initial multi-block structure and to move each of the corners to the corner of the geometry. Figure 1 shows the original surface, this mesh first positioned near the target surface and then after it has been projected onto that surface. The tessellation of the surface that appears in red is not a finite element mesh and is used here for visualization purposes.

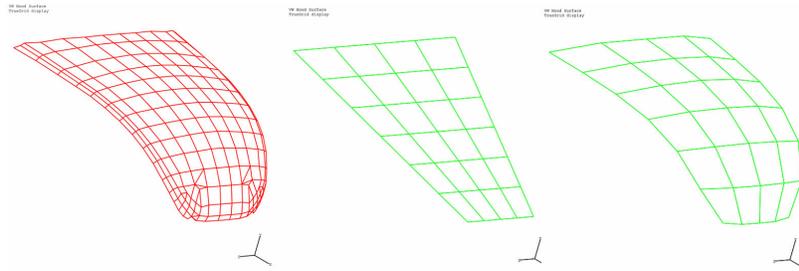


Figure 1. Meshing a Simple Model Surface

Notice that *TrueGrid* projects the mesh (in green) across multiple polygonal surfaces (in red), in effect smoothing them while still requiring that each node lies on one of the original component polygons.

The mesh density can now be parameterized in different regions by modifying the command that defines the initial multi-block structure. As a simple example, we define parameters to determine the number of elements in different regions of the mesh. On the left side of Figure 2, the parameter values have been chosen to provide a fine mesh in one half of the front of the object and a coarse mesh elsewhere. *TrueGrid* automatically generates the transition elements that are used to keep the refinement from spreading throughout the model. On the right side of the same figure, different parameter values are used to refine the mesh in a different corner while leaving a coarse mesh elsewhere. In this last figure, notice that we modified the mesh outside the region of particular interest in order to bound the global maximum aspect ratio from above.

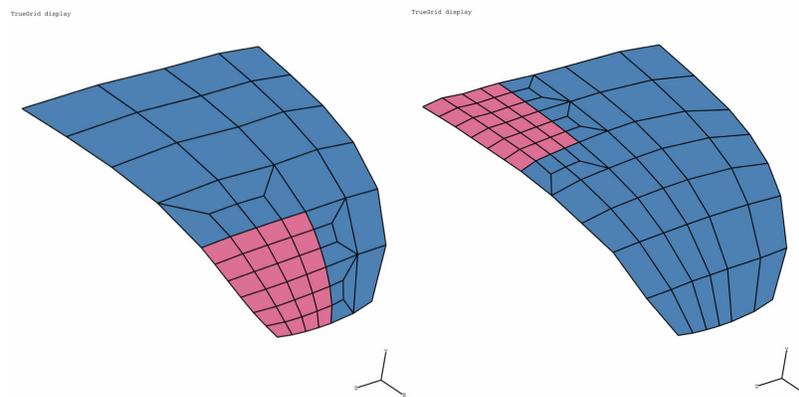


Figure 2. Parametric Localized Mesh Refinement in a Simple Model

## FINITE ELEMENT MODEL DEVELOPMENT

The finite element model of a 1996 Explorer 2WD vehicle has been developed using the concepts outlined in the previous section. The model has been developed in two independent parts. The frame structure (chassis) is composed of telescopic steel components and the cab is formed from sheet metal.

*Frame Model Development*

The frame is an assembly of formed (stamped or forged) components that are basically either a closed section formed by the weldment of two U-shaped parts, followed by open section U-shaped parts, or the cross members that connect the outer and inner assemblies. The rear and forward suspension assemblies are attached to the rail and cross member components. The engine and transmission are attached to the forward rail sections and the forward cross members.

The model has been generated so that the finite elements of the frame rail assemblies are generated in a parametric form. Specifically, we are able to vary the density of the elements in specific regions of the thick sheet metal representing the U-shaped components.

These parameters define the boundaries of regions of equal dimension. The parameters are defined for the longitudinal direction of each rail (both rails, left and right have the same topology), and around the perimeter of the rail at specific constant axial locations on each rail. The thickness of the rail components along the longitudinal direction is also specified in parametric form. The cross members that connect left and right rails are not parameterized. In order to preserve reasonable aspect ratios, the interface of these cross members represents a region in which the rails must maintain a constant mesh density.

*Cab Model Development*

The parametric input of model parameters for the sheet-metal components of the cab assembly is typically given in the form presented in Table 1.

Table 1. Parametric Input for Sheet-Metal Assembly Definition

```

c CONTROL PARAMETERS--- GRID FOR SHEET METAL-- EXPLORER II
c t=0.5 unless specified different for cab floor
c see file      merge_floor_cab
c -----
c LEFT (driver side)  RIGHT (passenger side)
c -----
c surface 421
para i4211x 3          i4214x 3
      i4212x 2          i4215x 2
      i4213x 2          i4216x 2 ;
c
c surface 249-- door frame structure --outer part
para i24911 3 i24912 2 i24913 2 ; c z axis flange lips
para k249x3 3 k248x3 3 ; c low rocker panel/with b pillar
if (%k249x4.gt.10) then
  para k249x4 10 ;
endif
c
para k249y1 1          k248y1 1 ; c new rib along y -b pillar
if (%k249y1.le.3) then
  para k249y1 3 ;
elseif (%k249y1.ge.3) then
  para k249y1 9 ;
endif

```

This table shows the implementation of logical commands that bound the size of the elements from below. The upper bound is not limited and is constrained only by the visual analysis of the generated part. The aspect ratio of the generated elements is simply addressed by observing the generated model and properly adjusting the parameters that generate the elements with unacceptable ratios. Notice, in the previous table, that the conditional statements at the end of the table limit the density of the mesh both from below and above. In turn, these statements effectively confine the aspect ratio to lie within a pre-determined range of values.

An example of the implementation of the parametric information associated with the generation of sheet metal components is shown in Table 2.

Table 2. Block Representation of Typical Sheet-Metal Command

```

c $$$$$$$$$$$$$$$$$$$$$$$$$$$$
c 249.m731-Ah 5/7/98
c A-pillar section left side
c $$$$$$$$$$$$$$$$$$$$$$$$$$$$
c
c 1
para ibigx [3*%j24911] ;
block 1 [1+%ibigx] [1+%ibigx+%i24911] [1+%ibigx+%i24911+%i24912] ;
1 4 [4+%i210y1] [7+%i210y1] [7+%i210y1+%i210y2]
[7+%i210y1+%i210y2+%i210y3] [7+%i210y1+%i210y2+%i210y3+%i210y4]
[10+%i210y1+%i210y2+%i210y3+%i210y4]
[10+%i210y1+%i210y2+%i210y3+%i210y4+%i210y5]
[12+%i210y1+%i210y2+%i210y3+%i210y4+%i210y5] ; -1;
0 0 0 0 0 0 0 0 0 0 0 0 0
c
pb 1 1 1 1 1 1 xyz -1.0354000E+03 -4.5493002E+02 7.4788196E+02
pb 1 2 1 1 2 1 xyz -1.0353085E+03 -4.2595297E+02 7.4725775E+02
.
.
pb 4 9 1 4 9 1 xyz -8.6397302E+02 -1.0895200E+01 7.1484698E+02
pb 4 10 1 4 10 1 xyz -8.6190607E+02 1.3969900E+01 7.1235101E+02
c
bb 1 10 1 2 10 1 2491 ;
mate 73
thic 1.955 c in 151199
endpart

```

This table illustrates the typical block representation used to generate the finite element mesh on a section of the model. A 3:1 transition is imposed between this region and the next and the proper element ratio is established by the first parameter definition.

The figures that follow illustrate the finite element models associated with different impact scenarios. Figure 3 illustrates the side impact case where the finite element mesh in the doors and supporting components has been refined. Notice the relatively coarse level of detail that appears at the rear of the vehicle in this figure. In contrast, Figure 4 illustrates the rear impact scenario; in this case all the rear part of the vehicle is refined in greater detail while the side remains coarser.

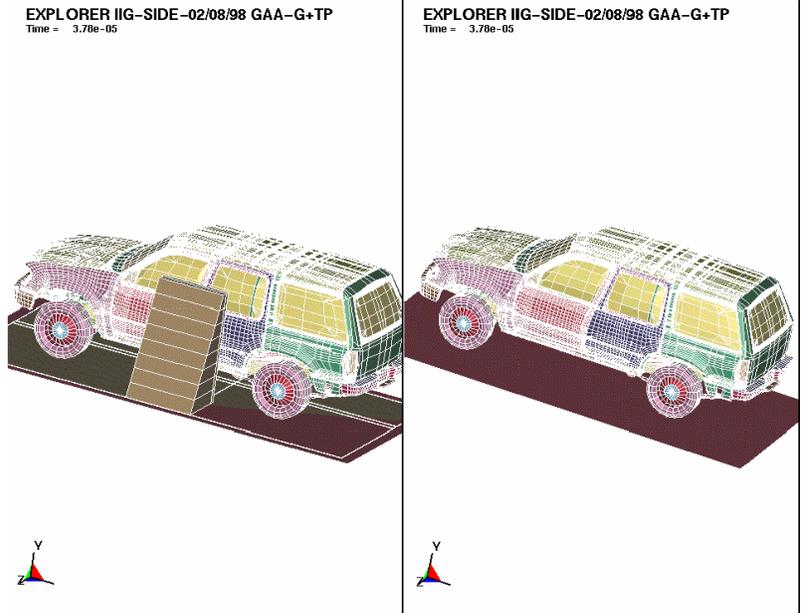


Figure 3. Side Impact Finite Element Model

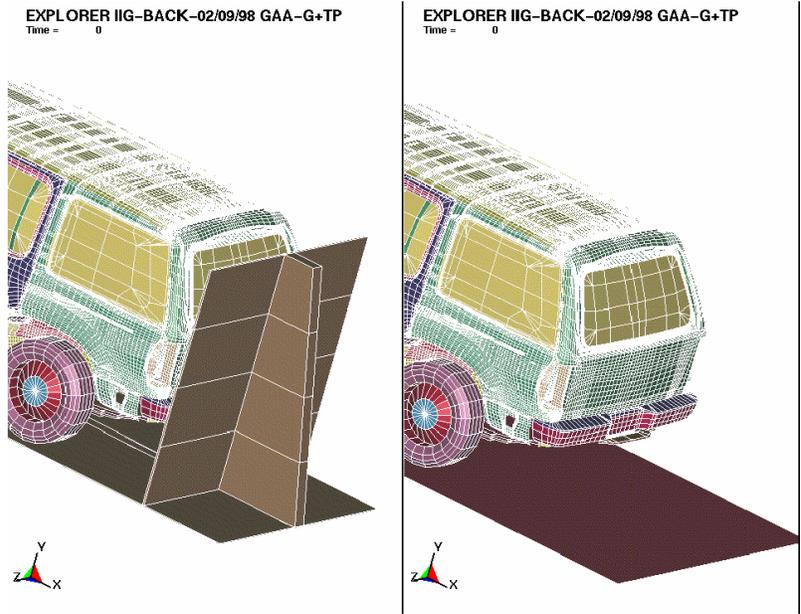


Figure 4. Rear Impact Finite Element Model

**MODEL VERIFICATION**

Two specific scenarios have been selected for verification of the model developed under the methodology specified in the previous section. The first scenario addresses the frontal impact of the vehicle against a rigid barrier under full

engagement with an initial velocity of 35 mph. A full-scale test of this case (NCAP, 1995) on a very similar vehicle has been used to assess the quality of the developed model. There are some discrepancies between the test specimen and the model used in the analysis. The test specimen is a 4WD Explorer and is a 1995 model. The analysis model is a 2WD vehicle model 1996. Because of the minor variation in vehicle weight and stiffness in the impact area, some of the accelerations present minor variation and further testing continues today. The analysis model does not include spotwelds as the implementation of this joining technique in a parametric model presented significant challenges to the analyst at the time that the model was created. Since that time, features have been added to TrueGrid that would permit the non-interactive parametric generation of spotwelds. Figures 5 through 9 illustrate the analysis model before and after impact.

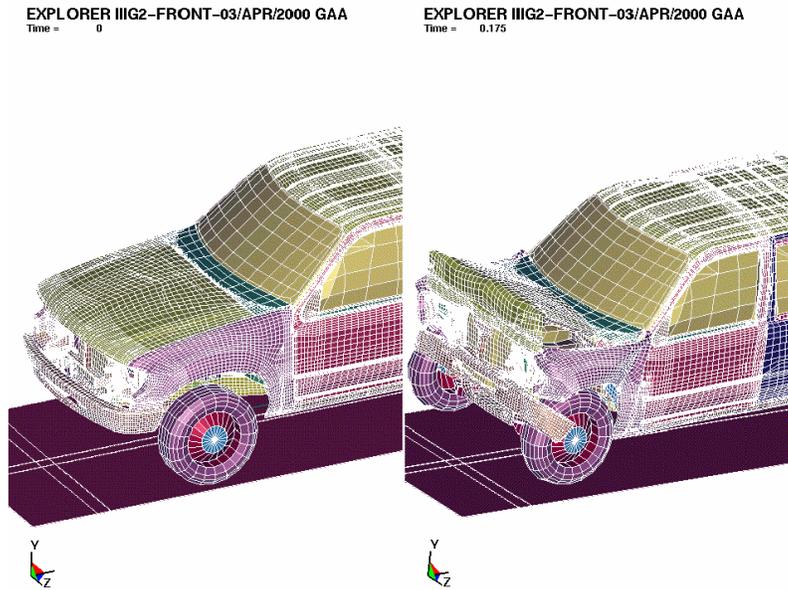


Figure 5. Finite Element Model Simulation of Frontal Impact

There is good correlation between the experimental and analytical results for this crash scenario as seen in Figure 6. The reduction of length in the analytical model is about 95% of the reduction that occurs in the full-scale test. Considering the discrepancy in weight associated with the models compared, this correlation is considered to be very good. There is abundant analytical and experimental data, the comparison needs further study and it will be presented at a later date.

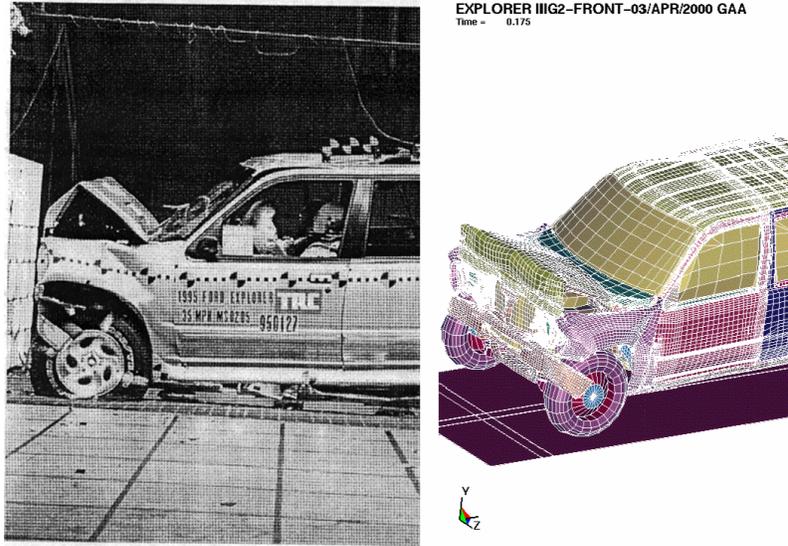


Figure 6. Full Frontal Impact – Test Results and Simulation

There is another test scenario that has been used to verify the model. This full-scale test (IIHS, 1996) was conducted in a 1996 4WD Explorer. Some test details differ from the analysis case. The velocity at impact in the full-scale test is 40 mph and the test is performed with 40% engagement.

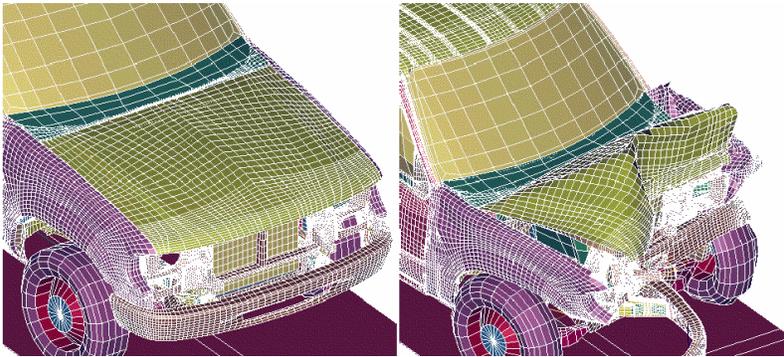


Figure 7. 50% Offset Front Impact

The test report does not contain much engineering data. Only qualitative verification is possible. Figure 8 shows the results of the analytical and test cases. Comparison between test and analysis is made only relative to external features of the SUV. There is excellent correlation in the mode and magnitude of deformation of the forward bumper, left front fender and driver door.

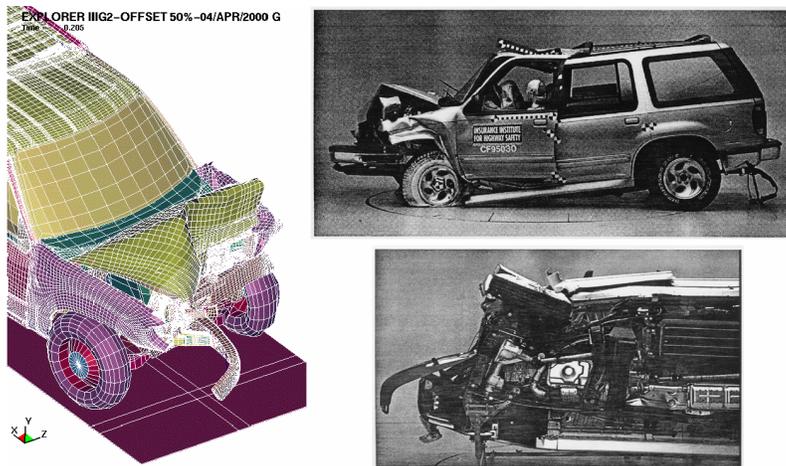


Figure 8. Simulation 50% Engagement Frontal Impact  
Full Scale 40% Engagement Test Results

### CONCLUSIONS

Project results have shown the feasibility and value of implementing a parametric approach to the generation of a generalized model that, with minor modifications in the parametric input definition, can be used for many postulated impact scenarios. This approach is a cost effective tool that eliminates the need to generate either a single large model that can be used for multiple impact scenarios, or several very specific models that address individual impact cases.

The use of parametrically defined mesh density, and the parametric definition of some structural features (e.g. material thickness) provide the analyst with a practical tool for the optimization of vehicle components that meet specific design objectives. In combination with existing optimization tools such as LS-OPT (LSTC, 1999b), it is possible to couple a pre-processor (*TrueGrid*) with an analysis package (LS-DYNA) and a numerical optimizer (LS-OPT) to implement the parametric scheme discussed in this paper and use this total package as an efficient design tool.

### ACKNOWLEDGEMENT

Research sponsored by the National Highway Traffic Safety Administration, U.S. Department of Transportation, under Inter-agency agreement DOE No. 2117-IO27-AI, NHTSA No. DTNH22-95-X-07229, under contract DE-AC05-00OR22725 with UT-Battelle, LLC.

**REFERENCES**

ARAMAYO, G.A. (2001). Design Optimization of an SUV for Crashworthiness with a Small Passenger Vehicle. Prepared by ORNL-Computation Material Science Group for DOT 2001.

IIHS (1996). Crashworthiness Evaluation. Crash Test Report, Ford Explorer – CF95030.

LSTC (1995). LS-DYNA, An Explicit Finite Element Non-Linear Analysis Code for Structures in 3 Dimensions, version 942.

LSTC (1999). LS-POST, Program for processing LS-DYNA output, version 14,.

LSTC (1999b). LS-OPT, Design Optimization for Engineering Analysis.

NCAP (1995). Frontal Barrier Impact Test, Ford Explorer. Report Number TRC-95-N10. Prepared by Transportation Research Center for DOT.

XYZSA (1996). *TrueGrid*, Preprocessor Version 2.0.6. XYZ Scientific Applications, Inc., 1996.