

Finite Element Modeling of Rollover Crash Tests with Hybrid III Dummies

Keith Friedman, John Hutchinson and Dennis Mihora
Friedman Research Corporation

Copyright © 2008 SAE International

ABSTRACT

The objective of this study was to demonstrate the ability to reproduce the impact environment occurring in rollover crash tests. There are over 26,000 fatalities and serious injuries annually occurring in rollover accidents in the United States [1]. Many of these are to restrained occupants and their head and spinal injuries have been associated with contact with the roof structure. Finite element models of the Hybrid III dummy and vehicles were used to model the rollover crash tests conducted for Ford. The rollover crash tests involved a production vehicle in a baseline form and one with a roll cage added to it. The impact conditions were incorporated and the results compared with the published test results. The results show that finite element modeling can reproduce the results from rollover crash tests.

INTRODUCTION

There are large numbers of fatalities and serious injuries annually occurring in rollover accidents in the United States. Many of these are to restrained occupants and their head and spinal injuries have been associated with contact with the roof structure.

On an overview basis, it can be expected that evaluation of a vehicle design's ability to meet corporate performance goals under rollover impact conditions will be accomplished utilizing finite element methods. In addition, to ensure the vehicle design meets government requirements, finite element methods are utilized first. As hardware test devices like the Jordan Rollover System (JRS) and the Controlled Rollover Impact System (CRIS) have become available to support repeatable dynamic rollover impact tests, the ability to model these dynamic tests will be needed. Ultimately, it can be expected that the evaluation of corporate goals, will lead to finite element model evaluations of rollover occupant protection designs utilizing finite element models of humans under a variety of impact conditions. Here, we report on the ability to model the results of

physical tests from an example of a dynamic rollover impact test device. The paper describes the results of LS-DYNA finite element simulations of rollover crash tests that had been conducted using a production and roll caged passenger vehicle with a restrained Hybrid III dummy.

BACKGROUND

A percentage of automotive rollover accidents lead to substantial roof deformation or intrusion into the automobile. Most of the serious injuries or fatalities suffered during rollovers are related to ejection control, padding, restraint and/or structure. The typical rollover neck injury mechanism is flexion-compression[2]. The Hybrid III test dummy has a neck that is too stiff in the vertical direction [3]. However, they have been and continue to be used by the automotive industry to evaluate the benefits of alternative designs under rollover conditions [1, 4, 5].

Computer simulations have been used for engineering analyses for over half a century. The Finite Element Method (FEM) for solving complex structural problems was first introduced in the 1950's.[6] Some of the earliest work reported using modern numerical methods in the automobile industry occurred in the late 1950's and early 1960's.[7] Finite element modeling of vehicle crush during impacts was reported by Chrysler by 1973.[8] Terex described using finite element analysis to evaluate the rollover protective cab for its large vehicles in 1985.[9] Also in 1985, Kecman described the use of finite element modeling in the optimization of a bus superstructure from a rollover safety viewpoint and the use of computer simulation as the basis for self certifying vehicles. A capability to predictively simulate passenger dynamics during automobile rollover accidents was developed and demonstrated using a complex rollover test as validation was reported by Kaleps in 1986.[10] Also in 1986 Obergerfell demonstrated that the motion of a Hybrid III dummy was predicted dynamically using a computer

simulation. The accuracy of the predicted motion was established by its favorable comparison to that recorded on high speed film during a 60 mph rollover in which the vehicle rolled over four times.[11] Other examples of rollover related computer modeling in 1986 were from Robbins and Almusallam [12, 13]. In 1989, Rizer demonstrated the capability to predictively simulate vehicle rollover dynamics using the Articulated Total Body (ATB) model using the results of two controlled automobile rollover crash tests.[14] In 1991, Friedman demonstrated substantial improvement in occupant protection through improved roof strength, padding, laminated and fixed glazing, and pretensioned seat belts utilizing computer simulation.[15] Also in 1991, Mitsubishi reported studying rollover impacts with Hybrid III dummies to analyze roof crush and head and neck injury prediction utilizing finite element modeling.[16] NHTSA reported, in 1991, results of finite element analysis results showing the effects of design modifications to a pickup under rollover impact conditions; this was followed by further extension of the work in 1993.[17] In 1994 Kumagi reported on the use of finite element modeling using LS-DYNA to evaluate the rollover strength of bus structures.[18] In 1995 computer simulations using ATB were reported by Cheng to predict passenger ejection and minimal driver impacts for a rollover accident.[19, 20] In 1998 Renfro used MADYMO to model a rollover crash test with dummies reporting good agreement between simulation and test.[21] Also in 1998, Friedman reported using finite element modeling with optimization to demonstrate improved roof strength.[22] In 1999 Yaniv reported research including MADYMO simulations to demonstrate rollover ejection mitigation under rollover conditions.[23] In 2000, engineers at Siemens describe using ADAMS and MADYMO for rollover simulation and analysis including both vehicles and occupant motions.[24] In 2001, Friedman reported finite element analyses using LS-DYNA to study roof intrusion velocities during rollover impacts, as well as effects of roof strength and restraints on injury potential during rollovers.[25, 26] In 2001, engineers at Delphi and Saab reported using computer simulations and MADYMO to characterize rollovers, occupant kinematics and injury measures in rollovers.[27] In 2001, Ward utilized MADYMO to evaluate occupant motions and injury measures under inverted impact conditions.[28] In 2004

Mao described finite element modeling of static and dynamic roof crush analyses.[29]

METHOD

The methodology consisted of utilizing FE analysis to study rollover impact tests of 1998-2000 Crown Victoria vehicles that were conducted for Ford [5]. These tests for Ford involved the production vehicle and the same vehicle fitted with a rollcage. In the tests for Ford, Hybrid III dummies were positioned in the vehicle. In the present FE analysis study, a model of the vehicle in the rollover tests was utilized. In the present study the FE models of the vehicle and dummy were validated and utilized under the defined impact conditions. The results were then compared with the published dummy injury results from the crash tests. The sections below describe various aspects of the method.

HYBRID III DUMMY MODEL VALIDATION

A 50th percentile Hybrid III finite element model originally obtained from Livermore Software Technology Corporation (LSTC) was validated using neck force data obtained from drop tests [30, 31]. In these tests, an inverted dummy, shown in Figure 2, was dropped from varying heights onto a load table, and the forces in the upper neck were recorded using a force transducer. Finite element simulations of the same impacts were conducted, and the resulting peak upper neck forces show good agreement with the experimental data (Figure 3).

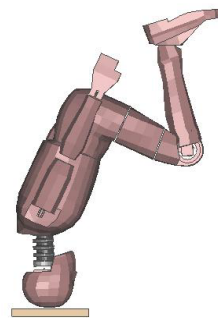


Figure 1 Dummy drop test in finite element simulation.

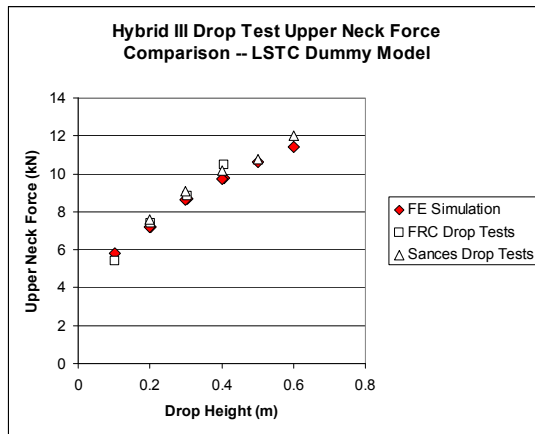


Figure 2. Peak axial compression neck forces in Hybrid III dummy during drop tests

A drop test [5] in which a Hybrid III head was dropped from 653 mm (25.7 in) onto the production car roof used in the crash test without a headliner was simulated with an FE model of the Hybrid III head. The accelerations at the head CG are compared in Figure 3.

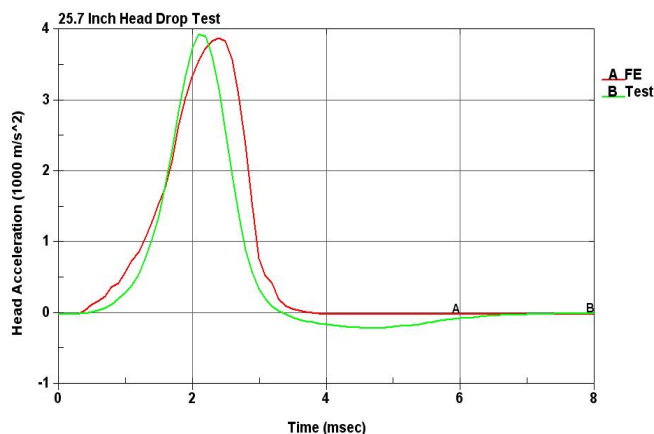


Figure 3. Head CG accelerations for 653 mm drop test.

The dummy model utilized approximately 6000 solid elements and 1900 shell elements.

PRODUCTION VEHICLE FINITE ELEMENT MODEL

A finite element model of the applicable production vehicle from the CRIS testing (Tests 51502 and 61102) [6] was obtained from NHTSA and adjusted to more accurately reflect the structure. A finite element simulation of a 216 test resulted in platen loads which are in agreement with published results from physical 216 tests (Figure 4). The finite element model roof structure is therefore representative of roof structure involved in the rollover tests.

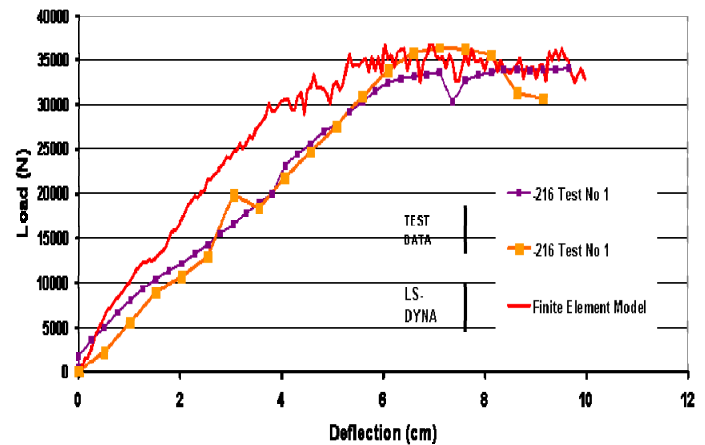


Figure 4. Platen load history during 216 tests and simulation

The vehicle model consisted of about 47000 shell elements and 3300 solids elements with masses and inertial properties modeled to approximate the vehicle used in the tests. A roll cage reflecting that used in the tests conducted for Ford was modeled and incorporated in the vehicle model (Figure 5). The model for the rollcage consisted of about 6500 shell elements. The material constitutive model for the steel utilized the Johnson-Cook representation.

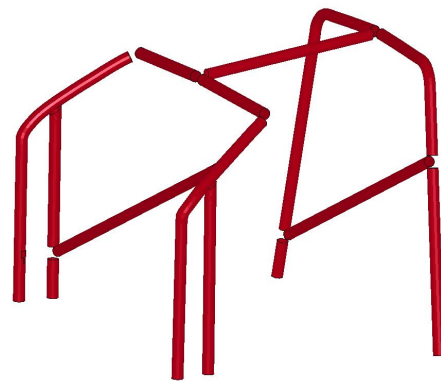


Figure 5 Roll cage model

ANALYSIS SOFTWARE

LS-DYNA version 970 was utilized for this study. [32]

IMPACT CONDITIONS

The impact conditions from the reference crash tests (CRIS 51502 & 61102) [5] were a lateral velocity of about 13 kph, a vertical CG drop velocity of about 8.5 kph, and a lateral roll rate of about 225 deg/sec.