Development of a 50th Percentile Hybrid III Dummy Model

Authors:

Cing-Dao (Steve) Kan,
Dhafer Marzougui,
and Nabih E. Bedewi

FHWA/NHTSA National Crash Analysis Center
The George Washington University

Correspondence:

Cing-Dao (Steve) Kan
FHWA/NHTSA National Crash Analysis Center
The George Washington University, VA Campus
20101 Academic Way, Ashburn, VA 20147 USA

Tel: 703-726-8365
Fax: 703-726-8358
e-mail: cdkan@ncac.gwu.edu

Keywords:

automotive safety, occupant, hybrid III, dummy modeling,
finite element modeling, crashworthiness analysis

ABSTRACT

Due to significant improvements in computer technology and finite element (FE) code capabilities, it has become more feasible and effective to incorporate occupant models in the analysis and evaluation of vehicle crashworthiness and safety. Using Detailed FE models that incorporate the vehicle, restrains systems, and occupants, in automotive safety analyses have shown advantages over the traditional methods where the vehicle and occupants are uncoupled.

This paper describes a finite element model of a 50th percentile Hybrid III dummy. The model was developed with several fundamentals in mind. First the resulting FE model must accurately represent the actual dummy. This is accomplished by incorporating correct geometry, material properties, and connectivity for all components. Second, the dummy model should be easily positioned and incorporated in vehicle and vehicle compartment models. This is achieved by using a systematic scheme for numbering and organizing the parts and joints in the model. The model should also be efficient, robust, and reliable. For this, special modeling techniques, such as using a uniform mesh for all important components and using one “automatic single surface interface” to treat the contact between all dummy components are incorporated in the model.

INTRODUCTION

It is estimated that over 40,000 people are killed yearly in vehicle related accidents in the United States. In an effort to eliminate this costly loss of life and resources due to automotive related accidents, the National Highway Transportation Administration (NHTSA) of the US Department of Transportation (DOT) mandates safety requirements for all new vehicles sold in the US [1]. The automotive manufacturers are required to build cars, which meet or exceed these minimum safety requirements. To this end, crash test dummies, such as the Hybrid III, are placed in the vehicles to simulate a human.

Hybrid III dummies are the most commonly used test devices for assessing “injury” in crash testing. They are currently the federally mandated and regulated industry standard for frontal crash testing [1]. When designing these dummies, the developers utilized different materials including rubber, foam, vinyl, steel, and aluminum, in an attempt to create a biofidelic occupant for measuring crash severity. The focus when designing the Hybrid III dummies was only on front impacts and all measurements and injury assessments are calibrated for these types of impact. The fundamental measurements which these dummies are designed to provide include: head and neck acceleration, chest acceleration, chest deflection, and femur load [2]. These measurements are used to assess occupant injury level and hence the vehicle crashworthiness and safety. The 50th Percentile Hybrid III, representing an average 50th percentile male occupant in mass and inertia, was the first developed and is the most fequenty used dummy in the Hybrid III family.

In this study a finite element model of a 50th percentile Hybrid III dummy is developed. This paper presents the methodology used in developing the model. A general description of the model and its subcomponents is also presented. The model was developed for the LS-DYNA [3,4] finite element code.

METHODOLOGY

A methodology for creating highly accurate occupant finite element models has been developed and used to create a computer model of a 50th percentile Hybrid III dummy [5]. When developing this methodoly, several key modeling features were thoroughly examined and special modeling techniques were adapted to insure that the resulting model is accurate, efficient, robust, reliable, and easy to use. Some of these techniques included:

1- Using uniform size mesh for all deformable components of the dummy. The size of the mesh is optimized for accuracy and computation speed. This makes the model more robust and reliable since the contact algorithm works best with uniform mesh. This also improves the accuracy and efficiency since an optimum mesh size is used.

2- Using the correct geometry and material properties for all components in the dummy that appear to function as deformable. In other words, every effort should be made to incorporate the true physical properties of the dummy into the model without rough estimation and assumptions. This not only ensures accurate response but also increase its functionality since no limitations are imposed on the model

3- Attaching null or rigid beam elements to the joint nodes. This feature makes it easy to visualize the location, orientation, and motion of each joint. It also makes positioning the dummy and its parts easier by using these null beams as axes of rotation.

4- Using one Automatic Single Surface Contact for all dummy components. This coupled with uniform mesh size, is the most efficient and robust contact setup in LS-DYNA leading to improved computation speed and accuracy. Also, this simplifies the task of combining the dummy model with other vehicles, restraint systems, and occupant interiors models.

5- Another example is the use of a systematic numbering for parts, nodes, and elements. This organization makes it easier to manipulate and position the model.

The methodology for creating occupant models consists of three main phases: 1- Model Development, 2- Component Level Validation, and 3- System Level Validation. This paper focuses on the first phase, Model Development. Phases 2 and 3 will be documented in future publications.

In the first Phase, all necessary information related the dummy are gathered and incorporated in the model. First, general information is collected. This includes components weight and inertia, design drawing of the dummy, photographs, publications relating to the device, etc. Next, the three dimensional geometry of all the dummy components is extracted and meshed. The geometry can be obtained from CAD data or digitized from an actual dummy. Material and sectional properties are then obtained and added to the model. This data can be obtained from the literature or through coupon testing. Next, the dummy components are connected and the contact interfaces are defined. Some of the steps involved in this first phase of creating the model are described in the next sections. This includes the geometry, material properties, connections, and contact interfaces.

Geometry

Geometry was considered the most important aspect of the physical dummy that needed to be gathered in order to create an accurate FE model. Due to the inertia, and kinematic properties of the parts of a dummy; shape, orientation, and location have significant effect upon its overall responce. To extract the three dimensional geometric data of the dummy, a passive six-degree of freedom digitizing arm was used. A grid is placed on the surfaces to be digitized and the coordinated of the intersecting points in the grid are digitized with the arm (Figure 1). The digitized data is automatically stored in a desktop computer to which the arm is connected. The surface patches generated from specified digitized data were stored in IGES format. These IGES files are then imported into the preprocessor for mesh generation and model assembly.

To ensure accurate geometry of the dummy, a global coordinate system was establised (by defining three points) around the dummy all components could be "digitized" in the assembled location. Also, each component was given its own set of three reference points for ease of location and geometry cleanup later in the process. Subsequently, each component of the dummy was digitized and these geometry files containing points, curves, and surface patches were imported into a pre-processor for FE creation. The process of gathering geometry is entirely non-destructive, and although the dummy was disassembled during the digitizing all parts were in proper working order upon completion of this process. Through this process each component was uncovered and examined further enhancing the understanding of the form, function, and importance of each dummy part.

The pictures in Figure 1 show the entire dummy with the digitizing grid in place and coordinate reference frames located upon various segments. Also, the two pictures of the dummy abdomen show the actual vinyl foam insert and the corresponding resulting digitized geometry.

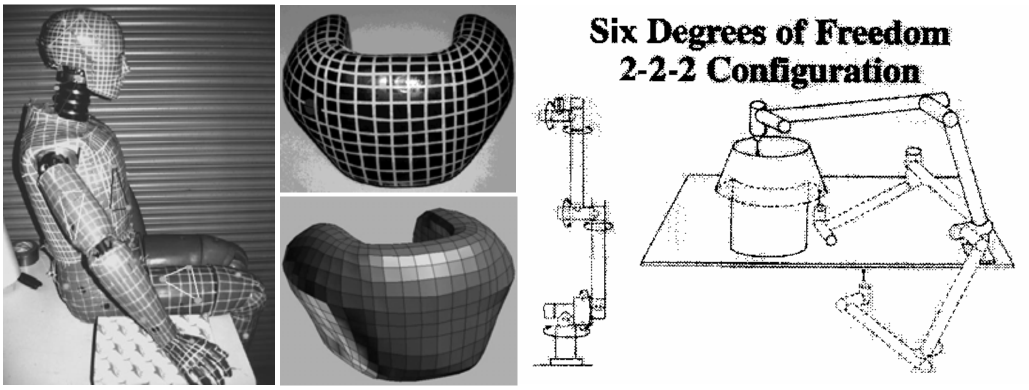

Figure 1: Geometry discretization process.

Material Properties

Another characteristic of vital importance in developing a robust finite element model is accurate material properties. The materials used to construct the hybrid III ATD cover a wide range of material types. The “bone like” load bearing structures of the human body are replicated with the use of steel and aluminum. Parts such as the human tibia, femur, pelvis, shoulder, spine, ribs, and skull are formed out of these sturdy materials. Where as vinyl and foam are used to replicate human soft tissue. These visco-elastic materials provide a human like look and feel to the dummy while also acting in a similar energy absorbing manner as part of skin and muscle tissue in a human. Also, complex geometric and kinematic structures such as the human neck and lumbar spine incorporate rubber, aluminum and steel cable.

The Steel and Aluminum used in the Hybrid-III are commonly used materials. For the purposes of this study it was assumed that these materials are well enough understood that they did not need to be tested. Some of the Steel and Aluminum components in the dummy do not undergo any significant deformations and were made rigid in the model. The correct material density and geometry were given to these of the component to ensure the proper mass and moment of inertia these rigid bodies. The remaining Steel and Aluminum components were assumed elastic. The material properties used for the Steel and Aluminum components were extracted from the literature.

All non-structural materials in the dummy, such as foam and vinyl, needed to be tested. These materials are considered to display time/rate dependent behavior and modeled using visco-elastic constitutive models. A universal testing machine, shown in Figure 2, was used to perform all the tests. The tested specimens were cut from actual dummy components supplied by the manufacturer. The specimens were tested in compression and tensile and at different strain level using different strain rates to determine their effects on the material parameters. Each test was repeated to ensure the consistency and repeatability of the tests. Load and displacement curves were recorded and used to extract the necessary parameters for the material model.

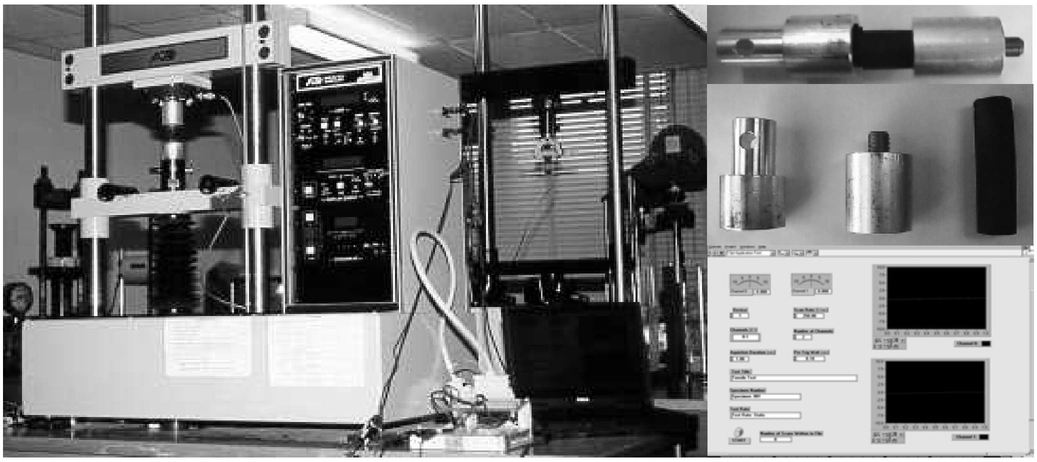

Figure 2: Testing machine (left), specimen and fixture (top-right), and computer interface (bottom-right).

Connections

One of the most important aspects of assembling an occupant model is the manner in which the parts are interconnected. These connections can range from joints, to spotwelds or bolts, and even various types of contact interfaces. Therefore, it is extremely important to capture the locations of these connections when gathering geometry so that their position and orientation can be properly incorporated in the model.

“Joints” are a form of kinematic constraint for a finite element model. They are placed between two bodies to constrain one or more of the six relative motion between them. Several types of joints are implemented in the LS-DYNA finite element code. Three different joint types are used in the H3 model: Revolute, Spherical, and Translational. The Revolute joint constrain two bodies in all but one rotational degree of freedom. The Translational joints allow for only one rotational degree of freedom between the two bodies. The Spherical joint constrains all translational motion between the two bodies allowing only for the three rotational degrees of freedom. Joints allow for the incorporation of both joint stiffness’ and joint stops. Joint stiffness throughout the hybrid III is considered to be 1g, all revolute joints on the physical dummy are tightened to a point where the limb will barely rotate due to its own weight. Furthermore, several joints have built in stops that are critical in the specialized *Soft-Stop* Spherical Ankle joint, and spherical hip joint. Joint stiffness is usually defined in the form of a load curve plotting joint angle vs. stiffness moment.

To facilitate the manipulation and modification of these joints, null beam elements are added between the nodes defining the joint axes. These elements do not affect the outcome of the simulation but they act as visual representation of the joint during the pre- and post-processing phases. This reduces the chances of making a mistake when defining the joint and makes it easier to monitor the joint behavior. These beams also simplify the dummy repositioning since they can be used as a frame of reference for translating and rotating the different components of the dummy model.

In addition to discrete joints, three types of nodal connections were used in the Hybrid III model, "Nodal Rigid Body Constraints", "extra nodes added to rigid body", and "rigid body merge". "Nodal Rigid Body Constraints" create a rigid body link between two or more nodes. They are were primarily used in the model to replicate the bolts in the physical dummy. "Extra nodes added to rigid bodies" is a means of tying a flexible body to rigid structure. This options is used in the model to connect some interior nodes of a soft pad to its neighboring rigid “bone” structure without having to sacrifice geometry. The "Rigid body merge" to connect two rigid bodys and treat them as one rigid body. This option is used in the model to connect two or more of the stiff components of the dummy. .

Another option that is used to for connecting the dummy components is the “Nodes Tied to Surface“ contact. Each node from the slave side of the contact is constrained to the closest segment in the master side and remains in the same position relative to that segment throughout the simulation. This option is used in the dummy model to connect the head vinyl to the inner head form.

Contact Interfaces

The contact interface, also known as sliding interface, is a mean of preventing the different components of the finite element model from penetrating each other. If two parts come in contact with each other, i.e. their outer surfaces touch each other, equal and opposite forces are applied to the nodes of the two surfaces until the penetration between them is eliminated. The magnitude if these forces are determined from the material properties of the interacting parts and amount of intrusion between them. The most efficient way of defining the contact between the different components of the dummy is to include all relevant parts in one single automatic contact. In doing so, all the components are checked against each other as well as within themselves. This eliminates speculation and guessing as to which parts might touch during the course of the simulation. Further more, by including all components in one contact, the code can more efficiently sort the nodes and segments hence reducing computational costs. Because "shell" is the most commonly used element type in the code, the contact for shell element is very robust. Most solid components in the dummy are covered with vinyl material, which is modeled with shell elements. The rest of the solid components in the model were covered with null shell elements for contact purposes only. This method removes all the solid elements from the automatic contact. Also, special care was taken to make sure that initial penetrations are eliminated from the model. Initial penetrations can lead to distorted geometry as well as high non-physical stress concentrations, both of which result in inaccurate simulation results.

MODEL DESCRIPTION

This section describes what was resulted from the implementation of the modeling methodology. Model summay information is shown in Table 1. As it can be seen from this table the model consist of approximately 40,000 elements. This level of detail was applied specifically to ensure the accuracy, reliability, and multi-functionality of the model. The model consists of six main assemblies, head, neck, torso, pelvis, arm, and leg. These assemblies are shown in Figure 3. Each of these components will be described in more detail in the next several sections. These sections are interconnected as follows; the head is connected to the neck through a revolute joint at the occipital condyle. The neck is connected to the torso through a rigid constraint (extra node added to rigid body). Each arm is attached to the torso through two revolute joints located at the shoulder/yoke position, thus allowing longitudinal adduction & abduction, and lateral adduction & abduction (abduction is motion away from the body, adduction is moving toward the body). The pelvis and torso are connected using a rigid constrain applied at the base of the lumbar spine mount. Lastly, both legs are attached to the pelvic girdle through spherical (ball & socket) joints. The stiffness of the dummy joints is represented by a load curves of moment vs. angle. These load curves were determined such that they would represent the 1g requirement defined in the dummy user's manual. The Hybrid III FE model is shown in Figure 3.

Table 1: Hybrid III FE model statistics.

Number of Parts	152
Number of Nodes	38,521
Number of Elements	39,974
Number of Materials	19
Number of Shell elements	21,880
Number of Solid elements	16,453
Number of Beam elements	68
Number of Joints	17

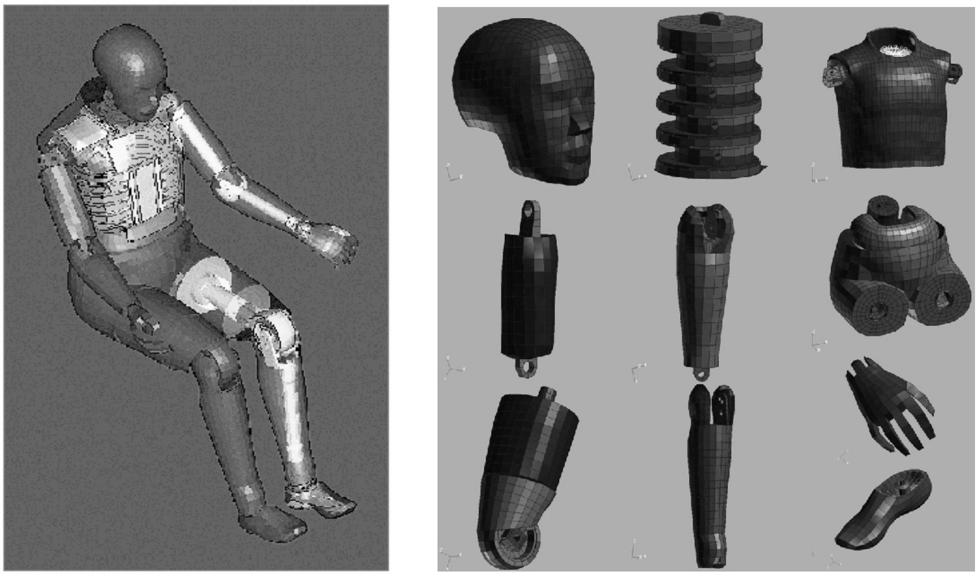

Figure 3: Hybrid III finite element model.

Head model

The head consists of six parts: head vinyl skin, head aluminum skull top and bottom, skull aluminum chin, inner steel plates, and neck connection bracket. The skull top, bottom, and chin were modeled as elastic material type 1 in LS-DYNA. These three sections were connected by merging their intersecting nodes. All sections are composed of shell elements and assigned material properties of flexible aluminum. The outer vinyl skin, meshed as solids and assigned material properties from the testing, was connected to the skull using the “nodes tied to surface” option. The visco-elastic material type 6 in LS-DYNA was used to model the vinyl head skin. Finally, an accelerometer was placed at the center of gravity of the head to measure the accelerations during impact.

Neck Model

The neck model is composed of six parts: head neck connection bracket, metal neck discs, rubber neck discs top and bottom, chest bib simulator, neck spine mount. The neck assembly is a combination of aluminum and rubber discs, with steel cabling running through the interior. The rubber components are modeled as solid elements and have the holes and slits from the physical dummy incorporated in their geometry. The rubber discs are assigned material properties from the rubber tested in the experimental testing. Furthermore, the rubber is modeled as visco-elastic type 6 in LS-DYNA. The contact between the slits is modeled as surface to surface type 3. The rubber and aluminum discs are connected using nodes tied to surface type 6 interface. The aluminum discs are meshed as a single layer of fully integrated solid elements, and assigned linear elastic type 1 material model, and standard aluminum material properties. The neck spine mount is attached to the thoracic spine box using the extra node added to rigid body option in LS-DYNA.

Torso Model

The torso is the most complex assembly in the dummy and consists of 27 parts. The main parts in the torso are as follows: ribs, jacket, bib, and thoracic spine box. These parts interconnect to form the load bearing structure of the dummy chest. They are composed of both shell and solid elements. Their material definitions include rigid, elastic, and visco-elastic material models from LS-DYNA. The ribs were modeled with elastic material incorporating the properties of steel. They are modeled with three layers of shell elements across their width in order to capture the correct bending behavior. The damping material is attached to the ribs by merging intersecting nodes. The damping material is modeled with solid elements assigned visco-elastic material properties. Each of the rivets connecting the bib and sternal plate to the rib cage was modeled with a rigid body constraint. All of the components are included in the automatic contact therefore all the interactions between the parts were considered in the model. The jacket was modeled with one layer of fully integrated solid elements across the thickness to ensure its bending resistance. The Thoracic spine box is modeled with a rigid material because it sees very little deformations that can be neglected. Both the ribs and the rib stiffening plates were attached to the thoracic spine box using the extra node added to rigid body option.

Pelvis Model

The pelvis is composed of seven parts: the lumbar spine, lumbar spine mount, abdomen, outer vinyl, inner foam, aluminum girdle, and hip. The lumbar spine was modeled with solid elements, and assigned visco-elastic material properties. The lumbar was attached to the bottom of the thoracic spine box and the top of the lumbar spine mount using the "nodes tied to surface" option. The lumbar spine mount is a rigid block of solid steel elements, which are merged to the pelvis foam and vinyl. The soft outer surfaces of the pelvis are modeled as visco-elastic shell elements. These shells cover and inner structure of solid foam elements which surrounds the rigid aluminum pelvic girdle modeled with solid elements. Finally, the abdomen is modeled as a visco-elastic material. The abdomen is meshed with shell elements and its stiffness is defined by a "control volume".

Arm Assembly model

The arm assembly is composed of the upper arm, lower arm, and hand. The upper arm is made of a metallic inner piece modeled as rigid shell elements, an inner solid vinyl pad, and an outer vinyl shell made specifically for the automatic contact. The lower arm is constructed in the same manner as the upper arm and the two are connected through a revolute joint at the elbow. The hand is made of a single layer of shell elements and is attached to the lower arm by means of a revolute joint at the wrist.

Leg Assembly model

The leg is constructed of four sections: upper leg, knee, lower leg, and foot. It is attached to the pelvis at the hip through a spherical joint. The upper leg is composed of a load cell and a vinyl pad. Both components are modeled using solid elements and their outer surfaces are covered with shell element for contact purposes. The load cell is assigned rigid material properties and the vinyl is assigned visco-elastic properties. The knee assembly consists of an aluminum insert and a vinyl cover. Once again the interior structure is modeled with solid elements and encapsulated in shells. The knee is connected to lower leg through two slider blocks. The slider blocks allow for both rotational and translational motions between the knee and lower leg. These motions are represented with planar and revolute joints in the model. The lower leg consists a load cell and a foam pad. It is modeled in a similar manner as the upper leg. The lower leg is connected to the foot through a “soft-stop” ankle assembly. A spherical joint is defined between the ankle components to replicate the motion between the lower leg and the foot. The foot is made up of an inner steel plate and an outer soft vinyl. The steel plate is modeled with shell elements and the vinyl is modeled with solid elements. The foot model incorporates the heel insert, which is modeled with solid elements and assigned foam material properties.

CONCLUSIONS

This project successfully developed and implemented a process for creating occupant finite element models. When developing the method, key issues were investigated and special modeling techniques were adopted to create an accurate, efficient reliable, robust, and easy to use occupant model. These techniques included: detailed geometry and material characterizations of all important components, uniform mesh size for all flexible components, one single surface automatic sliding interface to treat the contact, rigid or null beam elements at the joint locations, and systematic part and joint numbering. This method was used to develop a detailed model of the 50th percentile Hybrid III dummy. In the future, the same methodology will be used to incorporate any modifications made to the actual Hybrid III dummy as well as creating other dummy models.

REFERENCES

1. “Traffic Safety Facts 1996,” Publication DOT HS 808 649, (1996) United States Department of Transportation, National Highway Traffic Safety Administration, Washington, D.C.

2. Backaitits,S. H., Mertz, H.J., “Hybrid III: The First Human-Like Crash Test Dummy” pp. 49-55, SAE PT-44, Warrendale PA, 150906.

3. Hallquist, J. O. (1994). “LS-DYNA Theoretical Manual,” Livermore Software Technology Corporation, Livermore, California.

4. Hallquist, J. O. (2001). “LS-DYNA User Manual,” Livermore Software Technology Corporation, Livermore, California.

5. Ennis, J.B. (1999). "The Methodology and Development of a 50th Percentile Anthropomorphic Crash Test Device Finite Element Model" Master's Thesis, The George Washington University, Washington, DC.