9

INJURY BIOMECHANICS RESEARCH
Proceedings of the Twenty Seventh International Workshop

Finite Element Model of the NHTSA Thor Dummy

J. Canha, F. DiMasi, P. Kwok,
Y. Tang, M. Haffner, T. Shams

ABSTRACT

This paper describes the development of a three-dimensional finite element model of the new advanced frontal crash test dummy, Thor (Test Device for Human Occupant Restraint). The physical dummy represents a fiftieth percentile male and incorporates improved biofidelic features and significantly expanded instrumentation. Development of both the physical and finite element dummies is sponsored and coordinated by the National Highway Traffic Safety Administration, with the finite element model being developed for the LS-Dyna solver at the Volpe Center in collaboration with the physical dummy hardware developer, GESAC, Inc. Each component of the dummy is modeled independently and is validated based on material, component and/or sub-assembly, and full-dummy experimental testing. The modeling methodology and specific test procedures used to create and validate the lumbar flex joint sub-assembly of the dummy model are illustrated. Specialized static and dynamic tests were performed to aid in defining and validating appropriate material models, joint characteristics, and dynamic response characteristics. Results of the material and component testing on the flex joint are presented.

INTRODUCTION

The National Highway Traffic Safety Administration's (NHTSA) has funded and directed the development of a new advanced frontal crash test dummy, named Thor (Test Device for Human Occupant Restraint). The primary goal of the new dummy is to be an effective tool for whole-body trauma assessment in a variety of automotive occupant restraint environments. The dummy, designed and built by GESAC, Inc., incorporates improved biofidelic features and significantly expanded instrumentation. Currently, the Thor dummy represents a fiftieth percentile male and is shown in FIG. 1.

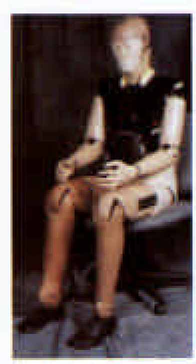

FIG. 1. Thor 50% Male

In addition to funding the design of the hardware, the NHTSA NTBRC is also funding the development of a detailed three-dimensional finite element model of the dummy. The model, being created in LS-Dyna format, will be an analytical design tool for crash safety research and may be used as a design tool for future Thor dummy designs, such as the fifth percentile female. The development process of the Thor fiftieth percentile male dummy model is described in this paper. The lumbar spine flex joint component development is used as an illustrative example of the modeling and validation methodology.

MODEL OVERVIEW

The finite element mesh of the Thor dummy model was built using the pre-processing programs Hypermesh and Patran. Part information was obtained from two-dimensional AutoCAD engineering drawings of the dummy hardware (see FIG. 2) as well as individual digitization of the geometrically complex molded parts, such as the skin foams.

FIG. 2. AutoCAD drawing and FE Model of Thor 50% Male

The major sub-assemblies of the dummy include the head and neck, shoulders, spine, ribcage, upper and lower abdomen, pelvis, femur, lower extremities, and arms. A description of the details of these components is being published in the 2000 SAE World Congress (Canha et al., to be published). Figure 3 reveals some of the detail of each component. Although the model pictured in this figure contains the arms and lower legs of the existing Hybrid III dummy, the lower extremities of the Thor dummy have been redesigned and are currently being modeled to reflect the new structure.

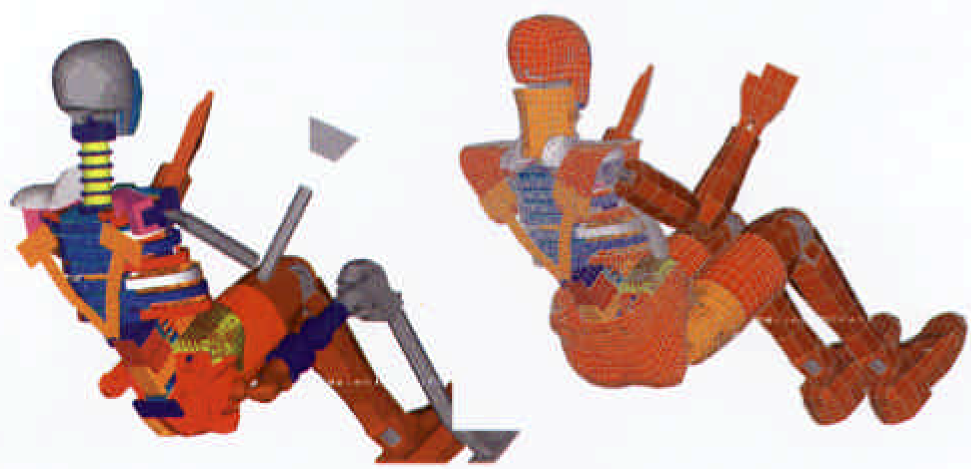

FIG. 3. Thor model detail

Each part of the dummy is modeled in detail to obtain the correct inertial properties as well as to achieve contact stability in the LS-Dyna contact algorithms. Each component is modeled independently and is validated based on material, component and/or sub-assembly, and full-dummy tests. The modeling methodology and specific test procedures used to create and validate the lumbar flex joint sub-assembly of the dummy model are illustrated next as an example.

LUMBAR FLEX JOINT SUB-ASSEMBLY DEVELOPMENT

The lumbar flex joint in the spine assembly, shown in FIG. 4, provides flexibility between the lower spine and pelvic regions and consists of a compliant molded urethane block sandwiched between two metal plates. Two stiff metal cables run between the metal plates to provide a fail-safe captivation device should the joint experience excessive loading. See FIG. 5 for an illustration of the lumbar spine flex joint mesh.

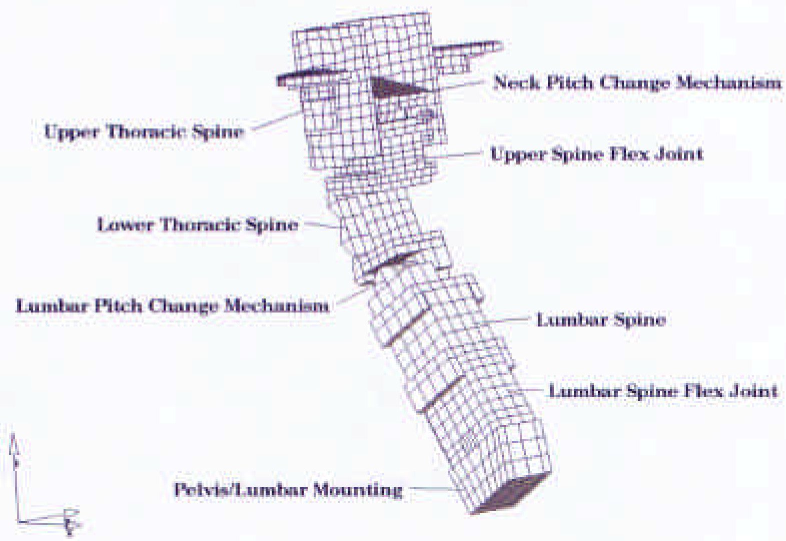

FIG. 4. Thor spine assembly model

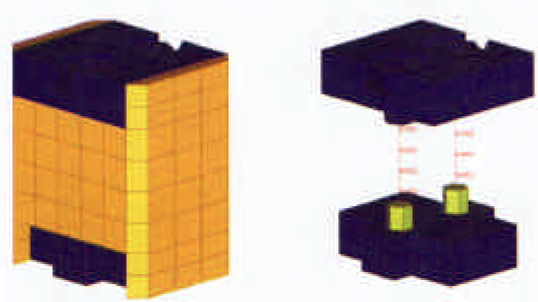

FIG. 5. Lumbar spine flex joint mesh

The top and bottom plates are modeled as rigid metal bodies connecting the flex joint between the lumbar pitch change mechanism of the upper spine and the pelvis/lumbar mounting block. These rigid bodies are modeled with brick elements using LS-Dyna material type 20, *MAT_RIGID, with a modulus of elasticity and a Poisson's Ratio of steel. The two metal cables are modeled as deformable elastic steel cables with rigid ends locked into the top and bottom plates. The cables are modeled with LS-Dyna material type 71, *MAT_CABLE_DISCRETE_BEAM, with the appropriate section properties of the steel cable.

Material Characterization and Validation.

The primary part that contributes to the structural response of the flex joint is the urethane mold. To obtain the appropriate material model and respective parameters to simulate the behavior of the urethane material, several tests were performed on a two-inch cube specimen of the urethane, specifically urethane CONAP TU-701. The cube specimens were tested (GESAC, 1999) in normal compression statically and dynamically at two speeds to capture any rate effects that might be present. The force and displacement time histories and force-deflection characteristics were obtained from each test. Also, static combined shear compression tests were performed on the specimens to observe material behavior in a combined loading condition.

From the static tests, an approximation of an elastic modulus was obtained and the response of the dynamic impacts exhibited a viscoelastic material behavior. LS-Dyna material model type 76, *MAT_GENERAL_VISCOELASTIC, was selected to represent the material. To obtain the parameter values of this general viscoelastic constitutive model, the experimental data from the dynamic impact tests were used. The method specifically consisted of using a numerical optimization program, named LS-OPT, together with LS-Dyna simulations to seek material parameter values that best matched the high and low speed experimental impact responses (Slavik, 1999). As with most optimization routines, an accurate initial set of parameters for the constitutive model was necessary to obtain the best solution. A nonlinear regression was performed on the test data to obtain the initial values of material parameters (Jeong at al., 1999). This analytical method, however, was performed on a simplified version of the viscoelastic model, namely a standard linear viscoelastic solid. Here, the shear relaxation behavior is given by

$$g(t) = G_\infty + (G_0 - G_\infty)e^{-\beta t}$$

where G_0 and G_∞ are the short and long time shear moduli, respectively, and β is the decay constant.

The parameters resulting from this analysis were used as starting values of the more general viscoelastic constitutive model in the optimization analysis, where the shear relaxation function is

$$g(t) = \sum_{i=1}^{6} G_i e^{-\beta_{G_i} t}$$

given by a Prony series of the form and the terms of this expression other than those from the nonlinear regression analysis are initially set to zero (i.e. $G_1 = G_\infty$, $G_2 = G_0 - G_\infty$ and $\beta_{G2} = \beta$). The parameters output from the optimization routine were then used in LS-Dyna simulations of the dynamic impact tests, shown in FIG. 6, to test the performance of the material model at the low and high impact speeds. Figure 7 and 8 are plots of the force-deflection response of the finite element model compared to the tests for the low and high speed impacts, respectively. The response of the material model starts to degrade at the higher impact velocity, but the response considering both speeds are very good.

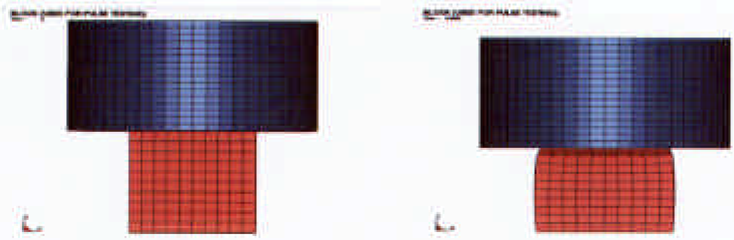

FIG. 6. LS-Dyna simulation of dynamic compression of urethane cube specimen

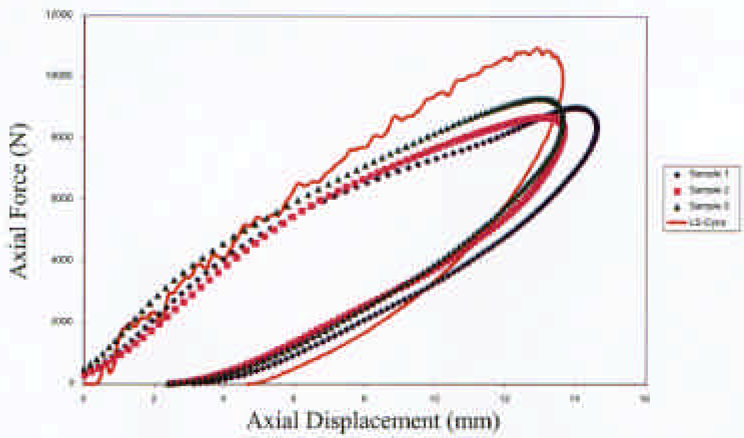

FIG. 7. Low-speed impact response of specimen tests and finite element simulation

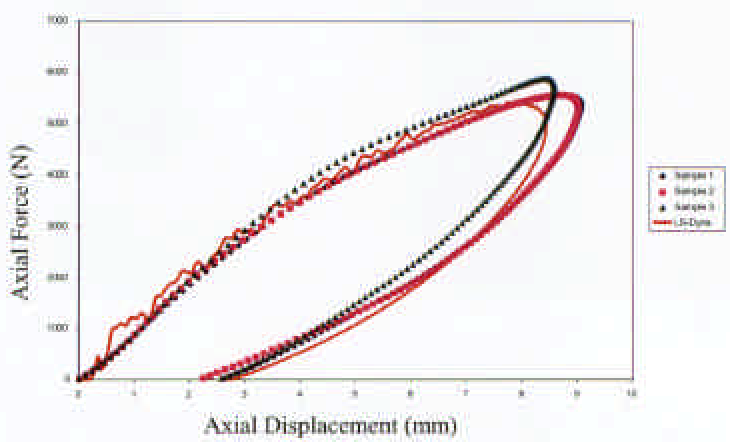

FIG. 8. High-speed impact response of specimen tests and finite element simulation

Component Validation

Once the material models and appropriate parameters were determined for each deformable part, simulations of the various lumbar flex joint component tests were performed. These tests include frontal and lateral bending of the joint both statically and dynamically impacted at two speeds. The experimental test setup, shown in FIG. 9, consists of the flex joint sub-assembly bolted to a fixture and a metal pole attached to its top plate. A rod shaped impactor with known mass and initial velocity strikes the pole and the location of two points on the pole are tracked to extract the bending angle. The moment is recorded with a five-axis load cell just below the flex joint bottom plate. Both the impacting rod and the pole on the flex joint are wrapped with a thin urethane material, also used on the dummy as a thoracic bib, to prevent a metal to metal contact.

The finite element simulations of the tests include the rigid flex joint top plate attached to a rigid cylindrical bar that is impacted with another rigid bar with appropriate mass and initial velocity conditions, see FIG. 10. The urethane material around each rod is modeled and the load cell is modeled as two rigid bodies with a locking joint between them that is used to extract the loads and moments at the appropriate location. The angle of the impacted pole was obtained in the model by tracking two nodes that were at the same two locations being tracked on the test structure.

FIG. 9. Lumbar flex joint flexion test setup

FIG. 10. Finite element simulation of lateral flexion impact test

The results of the low and high speed frontal flexion impacts of the lumbar flex joint (when the two cables are aligned perpendicular to the impact direction) are shown in FIG. 11 & 12. The lateral flexion results are in FIG. 13 & 14. The angle and moment time histories in these plots indicate a very good match between the finite element model and experimental tests at both impact speeds.

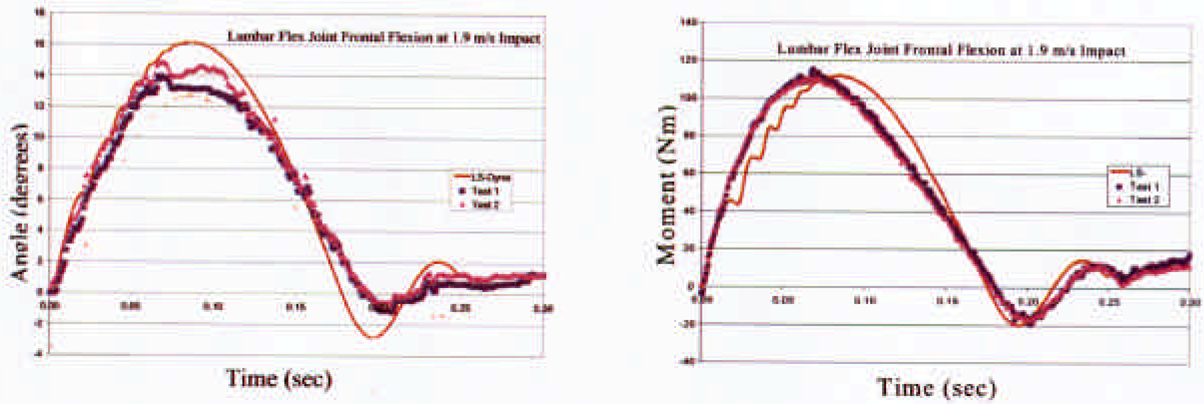

FIG. 11. Low speed lumbar flex joint frontal flexion response

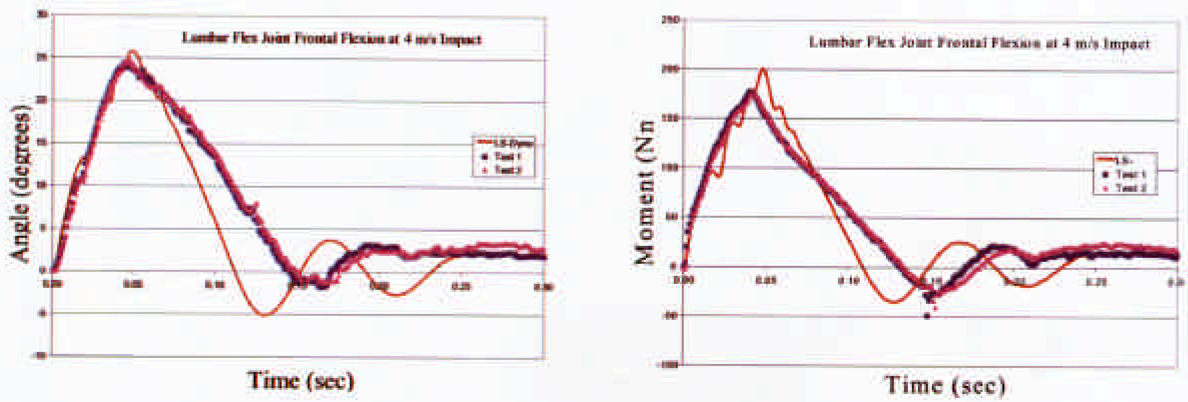

FIG. 12. High speed lumbar flex joint frontal flexion response

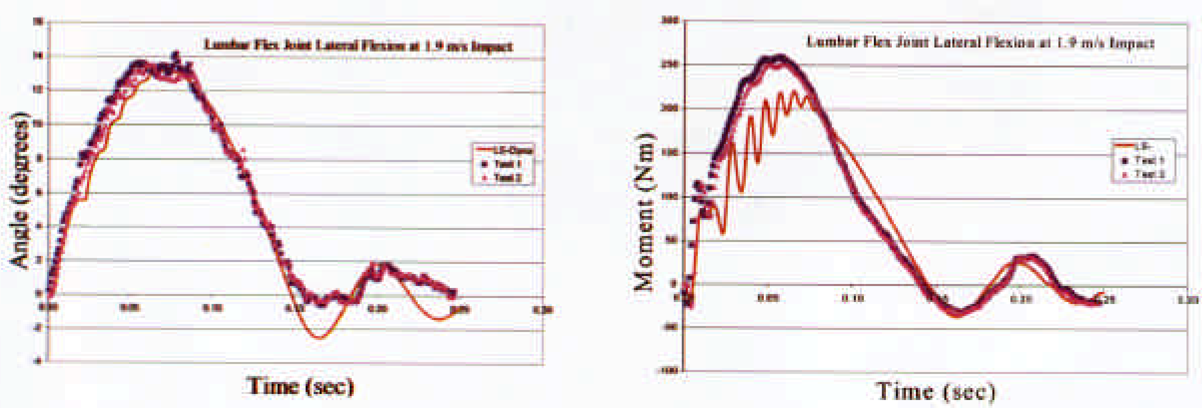

FIG. 13. Low speed lumbar flex joint lateral flexion response

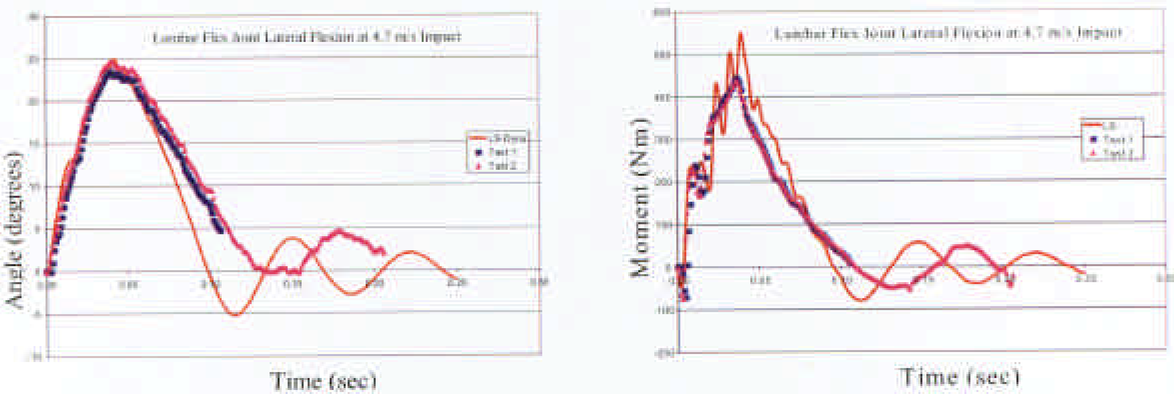

FIG. 14. High speed lumbar flex joint lateral flexion response

Full-Dummy Validation

Although it has not been completed to date, full-dummy validation tests and corresponding simulations are planned to evaluate the performance of the lumbar flex joint and all other components when incorporated into the full-dummy model. These tests will include the standard high and low speed thoracic (Kroell) pendulum tests (shown in FIG. 15), a sled test with a rigid seat and simple belts, and a test of an occupant compartment with simple belts and an air bag.

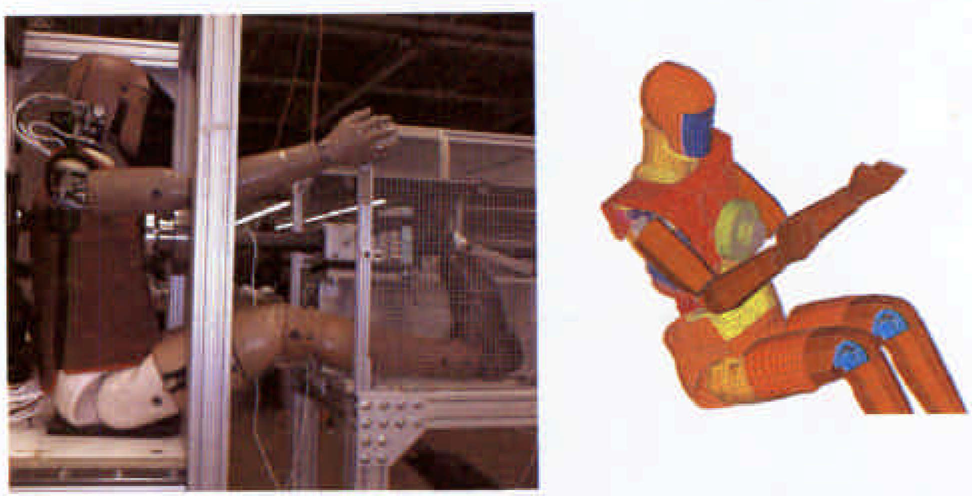

FIG. 15. Full dummy validation thoracic pendulum (Kroell) test and simulation

CONCLUSIONS

The procedure for developing a detailed finite element model of the new advanced crash test dummy, Thor, was presented with an example of the development of the lumbar flex joint component. Material and subassembly, or component, validation testing and corresponding simulations were described that are used to create and validate the lumbar flex joint component of the recently developed Thor fiftieth percentile male dummy. The results of the validation indicate very good agreement between the finite element model and various experimental tests at various impact velocities. The finite element modeling approach presented here is being used to develop the full model of the dummy and can also be used as a design tool for future Thor crash test dummies, such as the Thor fifth percentile female. The resulting model can also be an important crash safety tool useful for automotive safety systems design and development.

REFERENCES

CANHA, J., DIMASI, F., TANG, Y., HAFFNER, M., and SHAMS, T. (to be published). Development of A Finite Element Model of the Thor Crash Test Dummy. SAE 2000 World Congress, Paper No. 2000-01-0159, SP-1494.

GESAC, INC., "Test Support for Finite Element Modeling of Thor Crash Test Dummy", May 1999 Volpe Center report.

JEONG, D.Y., KWOK, P., and CANHA, J.V., Characterization of Viscoelastic Material Behavior from Dynamic Compression Test Data, Volpe Center Technical Information Exchange Document, November 1999.

SLAVIK, T.P., Using LS-OPT with LS-DYNA in a Material Model Parameter Identification Study, Volpe Center report from LSTC, September 1999.