Development of Detailed AF05%ile Hybrid III Dummy FE Model

Yoichiro Ohnishi   Mariko Mohri   Hayato Kaneko   Tatsuya Komamura   Tsuyoshi Yasuki

This paper describes a development of the Hybrid III AF05%ile dummy FE model to be used for frontal crash simulations. The precise geometries of the dummy were measured at a pitch of 1.0 mm using X-ray CT scans. The material properties and the mechanical responses of the components were measured in static and dynamic tests and were used for the model validation. The FE model results showed a good correlation with the kinematics and injury index values to those in the sled impact test.

Key Words: Safety, Occupant Protection, Finite Element Method / Frontal impact

1. INTRODUCTION

U.S. Federal Motor Vehicle Safety Standard and Regulation (FMVSS 208) specifies performance requirements for anthropomorphic test dummies. In this standard, occupant's injury risks are evaluated with the measured acceleration, force, moment, deformation in the attached sensors on the head, neck, thorax, tibia etc. of the dummy. One aim of developing vehicle safety performance is to reduce these injury risks by enhancing occupant restraint systems such as seat belts, airbags, etc.

In recent years, Finite Element Method (FEM) has played a significant role in the development of vehicle crash performance. FEM analysis is an effective approach for clarifying the phenomena and examining mechanisms in the crash test, since it is capable of estimating forces and deformation [1].

A dummy FE model for crash analysis has been developed by Moss et al [2]. In addition, Mohan et al. are currently developing a FE model for the AF05%ile Hybrid III crash test dummy [3][4]. In these developments, dummy geometry is modeled based on geometry data obtained from drawings or 3 dimensional (3D) measuring instruments.

This paper describes the development of an FE model of an AF05%ile Hybrid III crash test dummy (Figure 1). This dummy, which simulates an American small female, is one of the dummies used in frontal crash tests. In this development, X-ray computed tomography (CT) scans were used to precisely measure both the external geometry and internal structure of the dummy. The geometry of the dummy was then reproduced as fine grid. Subsequently, the mechanical properties of the dummy component parts and the stiffness of each joint were investigated and then applied these properties into the FE model. The force response of the FE model was then compared with the experimental data to validate

accuracy of the model. In addition, the FE dummy was placed on a sled FE model and used to analyze the dummy injury mechanisms in frontal impact tests.
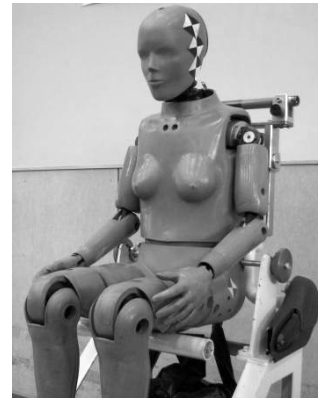

Figure 1: AF05%ile Hybrid-III Dummy

2. METHOD OF MODELING

2.1. Grid Generation

The development of the dummy FE model was carried out in 3 steps: (a) measurement of the 3D geometry of a physical dummy by X-ray, (b) generation of grid, and (c) input of material property and joint stiffness. The X-ray CT scan system used in this development [5] is capable of measuring the geometry of the dummy component parts as 3D data in an assembled state. The CT measuring resulted in a large series of 2 dimensional (2D) image slices, where the images were obtained with a pitch of 1.0 mm and a pixel size of 0.4 mm (Figure 2). Appropriate setting levels were selected for each target part from the obtained 1,340 sectional image groups, and 3D geometry data was created for each metal and non-metal material group. Then the dummy was modeled using shell and solid elements (Figure 3). The elements' size was 3 to 10 mm. The non-deformable metal parts of the dummy, such as parts of the skeleton, were

modeled as elastic element. The FE model consisted of 336 parts and 253,000 elements (Table 1). The mass and center of gravity of each component of the FE model was set based on the values referenced by the National Highway Traffic Safety Administration (NHTSA) in the U.S. (Table 2).
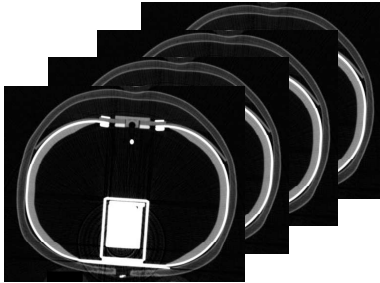

Figure 2: Thorax 2D image by CT with X-ray

(a) CT Scan Data   (b) FE Model

Figure 3: Comparison of Thorax Section

Table 1: The Number of Elements for FE Model

Part	336
Node	293712
Solid	253577
Beam	88
Shell	230282

Table 2: Comparison of Mass distribution

Assembly	Released Value	Model
	[kg]	[kg]
Head	3.37	3.74
Neck	0.91	0.89
Upper Torso	12.02	11.96
Lower Torso	13.24	13.32
Upper Arm	2.36	2.38
Lower Arm & Hand	2.36	2.32
Upper Legs	6.26	6.23
Lower Legs & Feet	8.12	8.13
Total	49	48.97

2.2. Material Properties

Dummy materials, such as the steel used for skeletal parts, the vinyl or urethane used for the dummy skin, and the rubber used for the neck, lumbar spine, deform in frontal crash test. The accurate material properties had to be integrated into the FE model to reproduce the deformation of each part of the dummy. In this development, the material properties of 49 materials used in the dummy were measured in static tension or compression (Table 3). Additionally, dynamic mechanical properties were measured for 7 materials identified as being strain rate dependent.

The measurement was performed using a Tensilon UCT-1T universal tester. The static tensile test was performed at 0.8 mm/s and the dynamic tensile test was performed at 10 m/s. The test specimens were prepared by analyzing a new physical dummy and cutting out material with the required length for measurement. Figure 4 shows an example of an extracted test specimen. The force and deformation properties from the measured data was recorded and used to extract the necessary parameters for the material model.

Table 3: The number of Test Specimens

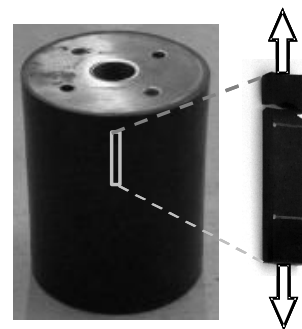| Material Type	Specimens
Steel	26
Aluminum	5
Dumping Material	2
Rubber	8
Vinyl	5
Ensolite	1
etc.	2

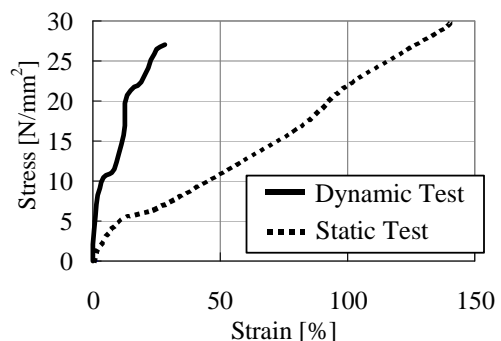**Figure 4:** Test Specimens (Lumber Spine)

Figure 5: Relation between Stress and Strain of Lumber Spine

2.3. Joint Stiffness

The dummy has 27 rotatable joints that simulate the rotation of joints in a human body. The torque of these joints is thought to influence dummy kinematics response and force transmission. In this development, the dummy joints' torque and rotation angle limits were measured. The joint torque was obtained using a push-pull gauge to measure the reaction force generated when the overall center of gravity of the movable portion was pulled up as shown in Figure 6. The joint torque value was then calculated by multiplying the measured reaction force by the distance from the joint to the center of gravity. At the same time, the rotation angle limits were calculated. The average of 15 data obtained from 5 dummies that were measured 3 times each was used in this development.
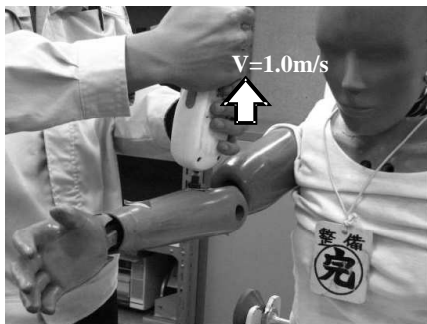

Figure 6: Measurement of Joint Stiffness

3. MODEL VALIDATION

3.1. Method of FE Model Validation

The FE model was validated to the standard certification tests recommended in the Code of Federal Regulations under 49 CFR Part 572[6]. To ensure consistency with the verification tests, each material model of the FE dummy and the minimum changes in material properties were applied. For example, the pelvic inner foam material was converted to a material model capable of reflecting strain rate dependence, and the properties beyond the strain region obtained in the material tensile tests were applied in consideration of calculation stability.

3.2. Head Drop Test

Figure 7 shows the verification FE model used in the head drop test. The aluminum skull was modeled as solid elements. The material type 77 (Ogden_rubber) is used to model the skin. In compliance with the Part 572 standard, the verification FE head model was dropped from a height of 376 mm onto a sufficiently wide 50.8 mm thick steel plate with nodes constrained in 6 degrees of freedom (DOF). The obtained deceleration waveform was run through an SAE class 1000 filter.

The time-history comparison for the drop test and simulation are shown in Figure 8. The deceleration results using the FE model are consistent with those from the test. Peak value and timing are consistent with the test. In addition, the maximum resultant deceleration fits the predefined response corridor.
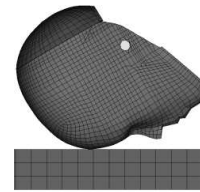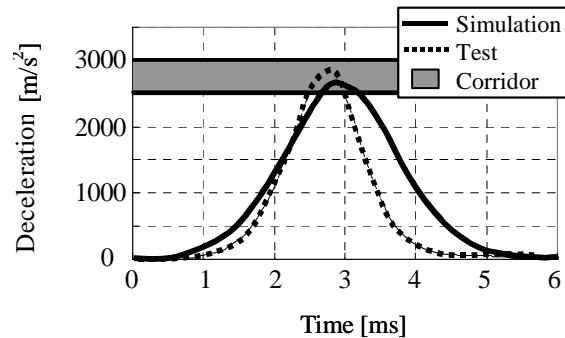

Figure 7: Head Drop Test

Figure 8: Comparison of Deceleration

3.3. Neck Extension and Flexion Test

Figure 9 shows the verification FE model used in the neck pendulum test. The 5 metal discs comprising the neck were modeled using shell elements designated as elastic element. The rubber between the discs was modeled as solid elements using the material type 77 (Ogden_rubber). The holes and slits of the neck in the dummy were also expressed. The pendulum with a length of 1,867 mm and a mass of 29.57 kg was modeled as elastic shell element. The pendulum side and dummy neck lower bracket were joined to ensure the same rotational movement. The obtained waveform was run through an SAE class 1000 filter.

The time-history of the rotation angle comparison for the test and simulation are shown in Figure 10. The simulation results show good correlation between the test and simulation result. It was confirmed that the maximum rotation angle and the time of intersection with the 10 Nm point fits the predefined response corridor. It was also confirmed that the resultant moment was within the corridor.
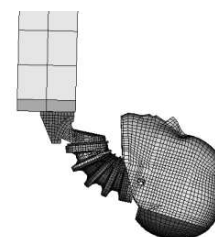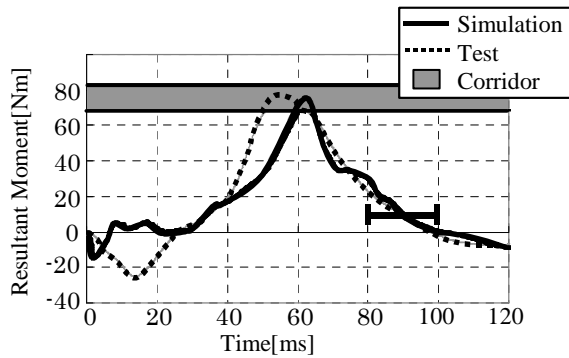

Figure 9: Neck Pendulum Test

Figure 10: Comparison of Rotation Angle

3.4. Rib Impact Test

Since ribs play a major role in frontal impact for generating reaction force among the parts comprising the chest, a verification test was conducted using a single rib from the thorax before carrying out the thorax impact test. In the FE rib model, the thin outer metal sheets of the rib were modeled as elastic shell elements, and the inner mass damper portion was modeled as solid elements. The spine box was also modeled as elastic solid elements. For the rib impact test, this development applied original static horizontal load test on a dummy rib and then compared with simulation. In the test, the spine box was constrained to the jig by a bolt. The rib impacted with the impactor at 20 mm/min. In the simulation, the spine box node was constrained in 6 DOF. A plate comprising a metal sheet and a buffer modeled from foam material were combined to the front of the rib. The rib impacted with the impactor at 0.1 m/s in the same way as in the test (Figure 11). Figure 12 compares the results from the simulation and test. The simulation results show good correlation to the test.
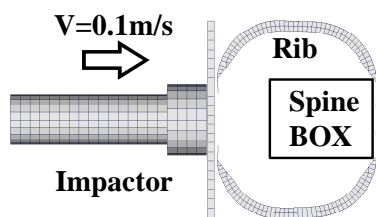

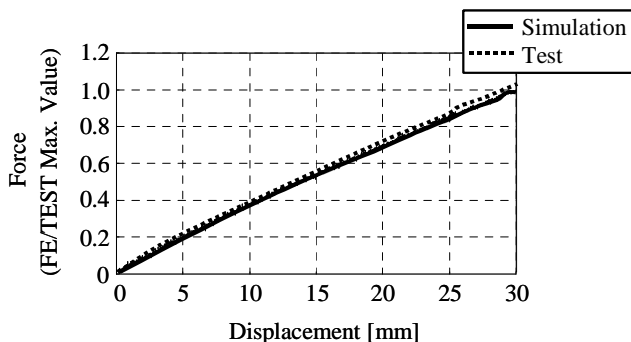**Figure 11:** Rib Impact Test

Figure 12: Comparison of Rib Force

3.5. Thorax Impact Test

Figure 13 shows the FE model used in the thorax impact test. In addition to the ribs and spine structured as described above, the thorax also consists of a jacket, bib, sternum, and other parts. The material type 77 was used to model the jacket skin. The material type 57 (Low density foam) was used to model the jacket inner urethane. A volumetric strain-dependent viscoelastic model was adopted for the bib to ensure accuracy under compressive force. The parts were joined by beam, rigid, and joint elements. The FE dummy was seated on a rigid surface and impacted by a pendulum with a mass of 13.97 kg at 6.7 m/s. Furthermore, original verification test was also carried out at 2.7 m/s, equivalent to the deformation rate of the thorax surface in a 56 km/h frontal crash. The obtained results were run through an SAE class 180 filter.

Figure 14 compares the chest deflection waveforms from the simulation and the test. Although the maximum value is within the corridor at initial speed of 6.7 m/s, the difference is observed in the transitional characteristics between the simulation and the test. However, both the maximum value and the transitional characteristics of the simulation results are good correlation to the test at 2.7 m/s. The results indicate that the FE model could be used for the impact analysis.
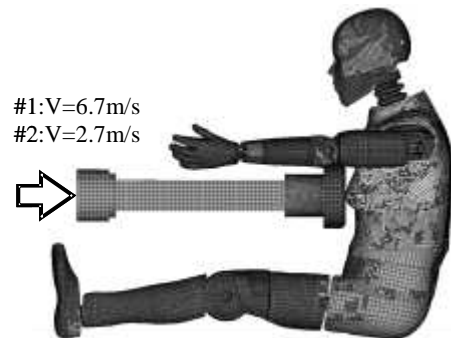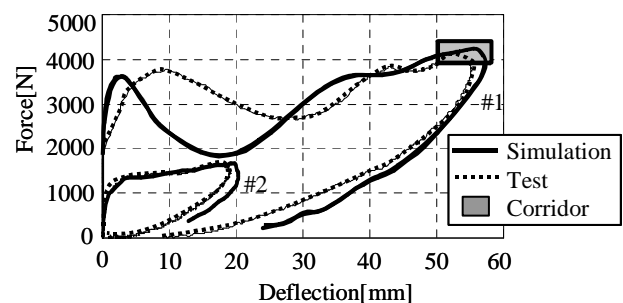

Figure 13: Thorax Impact Test (FE Model)

Figure 14: Comparison of Chest Deflection

3.6. Torso Flexion Test

Figure 15 shows the FE model used in the torso flexion test. The lower torso is the most complex assembly in the dummy. The main parts in the lower torso are as follows: lumbar spine, lumbar spine mount, aluminum girdle, outer vinyl, inner foam, and abdomen. The lumbar spine was modeled as solid elements, and assigned material type 77. The lumbar spine was connected to the spine box and the lumbar spine mount using the nodal rigid body. The wire rope used in the lumbar spine was modeled as beam elements. The pelvis, lumbar spine mount and inner foam were modeled as solid elements. The inner form was assigned material type of FU CHANG FOAM. The material type 77 was used to model the outer vinyl. Finally, the abdomen was modeled as solid elements. The material type 77 was used to the abdominal skin, and the internal abdomen was modeled as airbag defining stiffness. A rigid jig attached to the upper spine of the seated dummy on the rigid table was connected with cable to the pulley on the table. Apply a tension force in the midsagittal plane to the pull cable at 57.3rad/s. FE simulation was conducted based on the test procedure outlined in the 49 CFR Part 572. The obtained data were run through an SAE class 180 filter.

The torso flexion angle versus pulling force curves comparing between the test and simulation result are shown in Figure 16. The Figure shows good correlation between the test and simulation result. It was confirmed that the force at 45 degrees in the flexion was within the corridor.
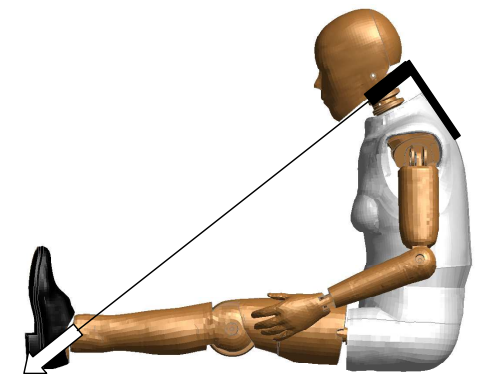
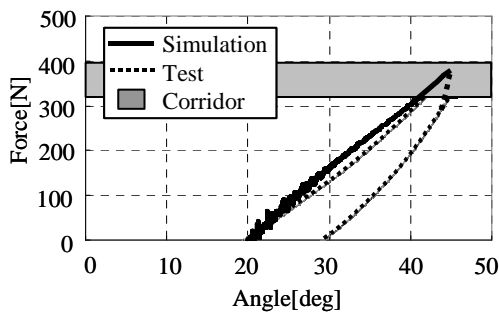

Figure 15: Torso Flexion Test (FE Model)

Figure 16: Comparison of Pulling Force

4. APPLICATION TO FRONTAL IMPACT SLED FE MODEL

4.1. Model Description

Figure 17 shows the sled FE model that was used in the validation calculation. The sled model was based on the model developed by Shirooka et al [1]. The components that affect the kinematics of the dummy, such as instrumental panel, airbag, seatbelt, seat foam, were modeled.

The dummy FE model was placed in the front passenger seat of the sled model and applied a 35 mph sled pulse. The first half of the simulation was used to seat the dummy in the target position. After reproducing the reaction force from the seat directly before the crash, the deceleration generated during a crash test was then applied to the sled body.
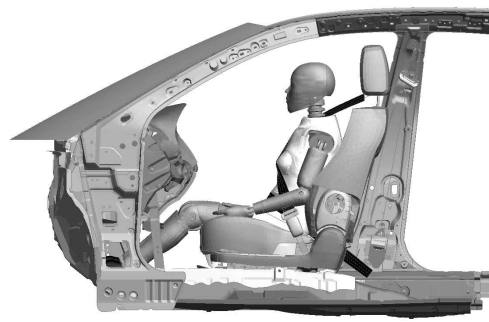

Figure 17: Sled FE Model

4.2. Model Validation

Comparison of the dummy kinematics between the sled test and the simulation at the timing of maximum chest deceleration (70 ms) is shown in Figure 18. As examples of injury criteria, the time-history of chest deceleration comparison for test and simulation are shown in Figure 19. The simulated dummy kinematics and maximum values of chest deceleration well coincide with the test results.
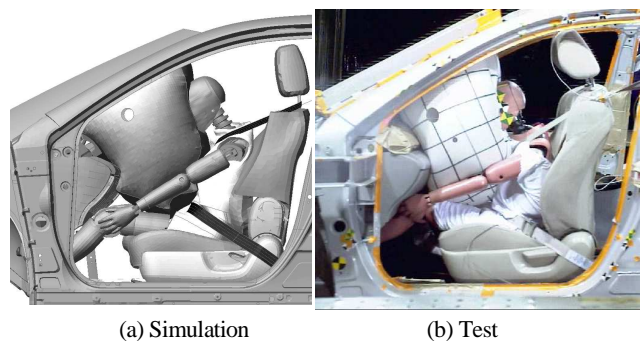

(a) Simulation   (b) Test

Figure 18: Comparison of Sled Test Kinematics (70 ms)
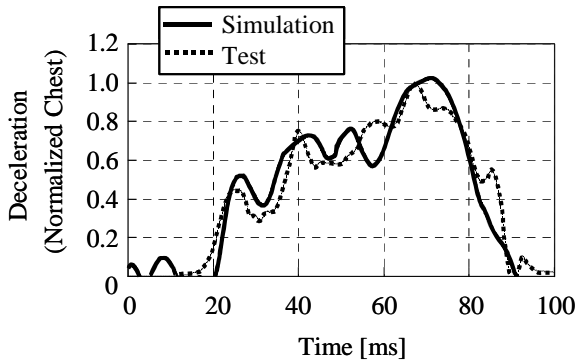
Figure 19: Comparison of Chest Deceleration at the Sled Test

Figure 21: Contour of Von Mises Stress (38ms)
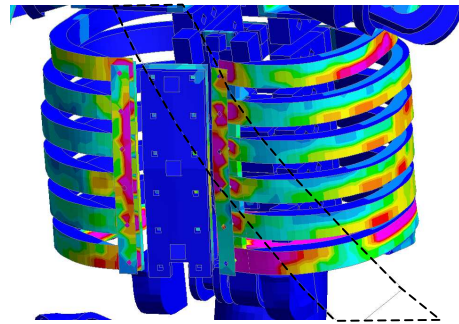
Figure 22: Contour of Von Mises Stress (70ms)

5. DISCUSSION

Figure 20 shows the time-history of chest deceleration and seatbelt shoulder force. Chest deceleration begins to rise up at 20ms with the seatbelt pretensioner on. Then, the chest deceleration starts to increase, while the shoulder seatbelt force keeps constant from 38ms with the load limiter. This phenomenon was analyzed using the FE model. Figure 21 and 22 show stress concentrates upon the upper thorax loaded by the seatbelt force at 38ms of the start timing of load limiter and at 70ms of the timing of maximum chest deceleration respectively. Comparing the stress contour between the two timings, the shoulder and the stress in the 1st rib to the 5th keeps low level, while the 6th rib is increasing. The result indicates that the force loaded on the shoulder and upper ribs are controlled by the load limiter, meanwhile, the force on the lower chest is rising up with the torso forward. The results show that the applied force to the lower chest transmitted to the spine box thorough the rib induced the increase of the chest deceleration.
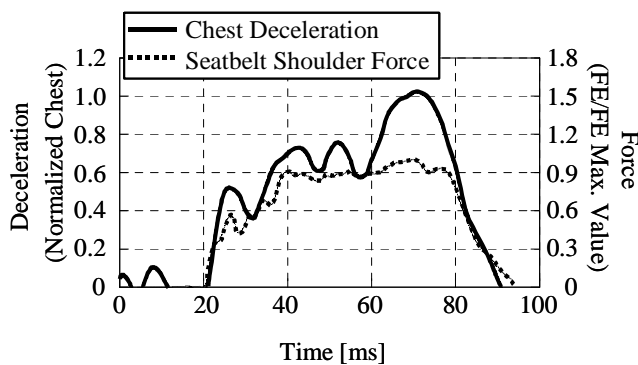

Figure 20: Comparison of Chest Deceleration and Seatbelt shoulder Force
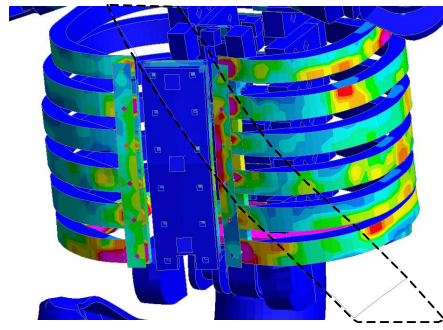
6. CONCLUSIONS

The detail geometry of the assembled dummy was measured using X-ray CT, and from the data, the FE model of the AF05%ile Hybrid III dummy was modeled.

Material properties were optimized by cutting out test specimens from dummy component parts and performing static and dynamic tests. The identified material properties were assigned to the FE model.

The developed detail FE model showed reasonable correlation for head drop, neck extension and flexion, thorax impact, torso flexion tests.

It was concluded that the FE model was effective for analyzing the internal deformation and load transfer during the crash test.

REFERENCES

1. Masakazu, S. et al. : Reduction of Unevenness in Occupant Injury Index on Passenger Side in Frontal Collision, JSAE, Vol.39, No.6, 20086031, (2008)

2. Moss, S. et al. : Development of an Advanced Finite Element Model Database of the Hybrid III Crash Test Dummy Family, SAE Paper No. 971042

3. Mohan, P. : Development and Validation of Hybrid III Crash Test Dummy, SAE Paper 2009-01-0473

4. Mohan, P. : LSTC / NCAC Dummy Model Development, 11th International LS-DYNA User Conference, (2010)

5. Okada, T. : Three-Dimensional Shape Measurement With High-Energy X-Ray CT-Scan, SAE Paper No. 2003-01-1033

6. http://www.access.gpo.gov/nara/cfr/waisidx_02/49cfr572_02