Development of a 3D Finite Element Model of the Chinese 50th Male for the Analysis of Automotive Impact

Hui-min Hu[1], Li Ding[2(✉)], Xianxue Li[2], Chaoyi Zhao[1], and Yan Yin[1]

[1] China National Institute of Standardization, Beijing, China
{huhm,zhaochy,yiny}@cnis.gov.cn
[2] Key Laboratory for Biomechanics and Mechanobiology of Ministry of Education, School of Biological Science and Medical Engineering, Beihang University, Beijing, China
{Ding1971316,li36101120}@buaa.edu.cn

Abstract. Occupant thoracic and abdominal injury during automotive crashes accounts for the biggest portion of all automotive injuries which is about 45 percent. So it's important to improve the vehicle's protective performance which leads to the high demand for crash test dummy. At present, crash test dummies are used in automotive impact in order to design and assess new vehicle safety performance. Hybird III I is widely used in the world and it's same in China. However Hybird III doesn't meet the Chinese anthropology which the Hybird III is bigger than Chinese. So it's in urgent need to establish the crash test dummy for Chinese. In this study, a finite element dummy with thorax and abdomen which is consistent with the 50th percentile Chinese male in order to predict the mechanism response of Chinese occupant during automotive impact and improve the impact automotive safety specifically for Chinese is developed.

Keywords: Finite element · Chest · Abdomen · Impact · Automobile

1 Introduction

Car has been essential for many people because of its convenience, however the problem of vehicle safety such as crash impacts is becoming more and more notable. In China, there were more than 100 thousands of people died because of traffic accidents which accounts 20 % of all the deaths derived from traffic accidents in the world [1]. According to the US National Accident Sampling System data which showed the damage parts of the body during the crash accidents, the head injuries accounted for 35.3 %, the chest injuries accounted for 26.7 %, the abdominal injuries accounted for 18.2 %, the lower limb injuries accounted for 4.6 % and then the upper limb and facial injuries [2]. From the above survey, we can know that the head, chest and abdomen are the most vulnerable parts to get injured during automobile crash accidents.

During the automobile crash accidents, the crash situations can be classified into frontal impact, side impact, rear impact and roll accident. Based on the survey, the frontal impact accounts for 57 %, and the side impact accounts for 27 % [3]. To improve the vehicle safety in the event of impacts, China has implemented the car

V.G. Duffy (Ed.): DHM 2015, Part II, LNCS 9185, pp. 258–265, 2015.
DOI: 10.1007/978-3-319-21070-4_26

design rules for occupant protection in frontal impact in 2003 and for the side impact in 2007 which the car manufacturers have to satisfy with the rules. In order to alleviate the cost and limitations of the automotive safety research through experiment, researchers have pursued the development of finite element models and the finite element modelling technology has been enabled with the advent of high-speed computing environment.

Understanding the mechanism behind the injuries during automotive impact is an important step in improving the impact crash safety of vehicles. Finite element modelling can represent the human body with accurate geometric or material properties and can simulate various complex situations [4–9].

The purpose of this study is to establish a finite element dummy with thorax and abdomen which is consistent with the 50th percentile Chinese male in order to predict the mechanism response of Chinese occupant during automotive impact and improve the impact automotive safety specifically for Chinese.

2 Methods and Model Description

2.1 Modeling Procedure

The finite element model consists of spine, ribs, heart, lungs and relative soft tissue. The spine, organs and soft tissue are established through three different approach. The geometry models of thorax and abdomen were finished in Solidworks of which the data was based on China national standard of human dimensions. The models of spine and ribs were developed in previous studies which were also based on the 50th percentile male dimensions. The organs like heart, lungs were established in Mimics from MRI data of a 50th percentile Chinese male. After the geometry model above were finished, the Hypermesh was used to mesh them into finite element model and then the finite element models were imported into Abaqus to analyze. The following will introduce each part in detail.

2.2 Finite Element Model of the Thorax and Abdomen

The geometry of the thorax and abdomen are based on China's National Standard GB 10000-88. The age group is 26–35 years old and all the samples are male. The 50 percentile data is used. The waist measurement, waist depth, waist width, chest measurement, chest width, chest depth, across shoulder width, neck width are used to develop the geometry model in Solidworks. The data is shown in Table 1. Firstly, the cross section curves of different parts, such as waist, chest, shoulder, are defined in Solidworks, and then they are lofted to form the 3D solid model just as shown in Fig. 1.

Then the thorax and abdomen model was meshed using Hypermesh and the finite element model contains 32338 nodes, 31244 elements which are mixed with C3D8R and C3D6. The finite element model of the thorax and abdomen is shown in Fig. 2.

Table 1. Measurement used in developing thorax and abdomen

Items	Waist measurement	Waist depth	Waist width	Chest measurement
Size (mm)	734	180	264	869
Items	Chest depth	Chest width	Shoulder width	Neck width
Size (mm)	212	281	376	120

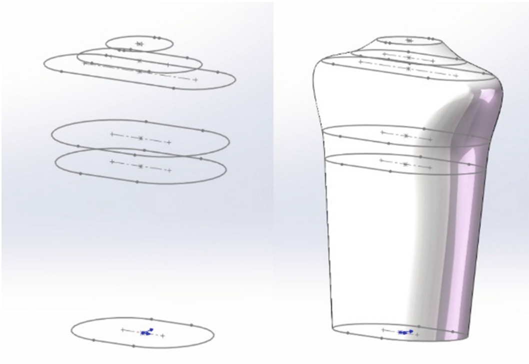

Fig. 1. Cross section curves and 3D solid model

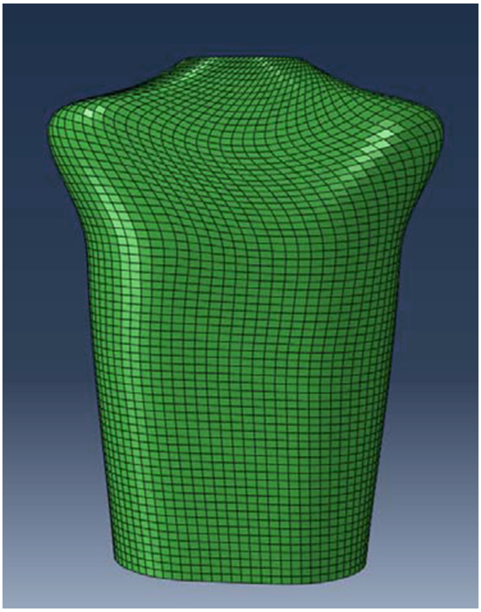

Fig. 2. Finite model of the thorax and abdomen

2.3 Finite Element Model of the Spine

The thoraco-lumbar spine has been developed in detail as shown in Fig. 3. The geometry of spine is based on the tomography data which was modeled in our previous study and now is scaled to represent the 50th percentile Chinese male spine. The spine contains 12 thoracic and 5 lumbar which is connected by 17 disks. The vertebral bodies and disks are all simulated using solid elements, however with different moduli. The whole spine element model contains 147655 nodes and 605943 elements which the type is C3D4.

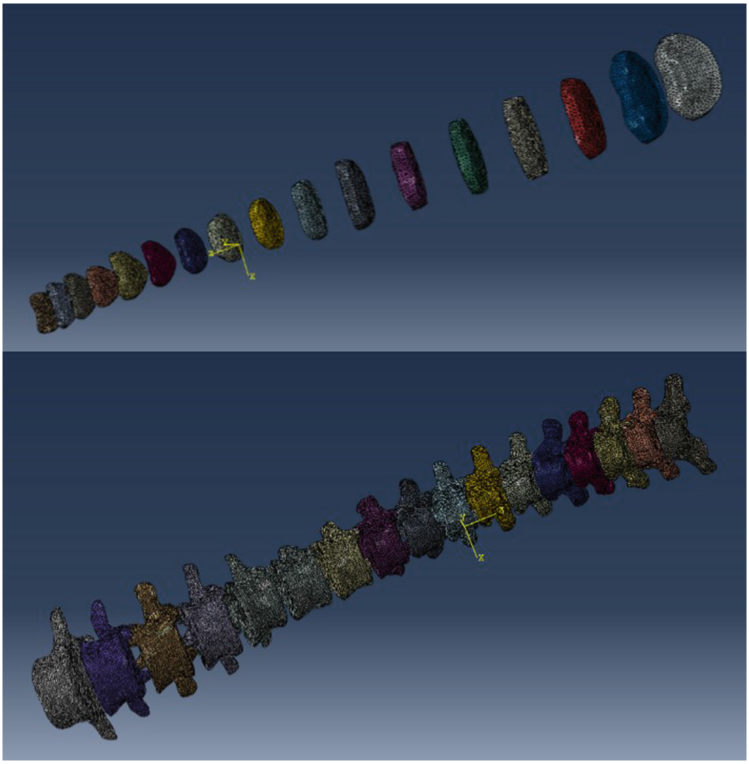

Fig. 3. Finite model of the spine

2.4 Finite Model of the Ribs

The ribs and sternum were modeled as shown in Fig. 4. The geometry of ribs and sternum is also based on the tomography data and process them to construct the 3D model with the most accuracy by Mimics software. Then the 3D model is scaled to represent the 50th percentile Chinese male ribs and sternum. To simplify the calculation, the ribs and sternum are modeled as one model except for the 11th and 12th ribs which ignores the ligaments and synchondrosis.

The geometry model then is processed and edited in Hypermesh software and meshed with trihedron. They are simulated using solid elements and share the same moduli. The whole model contains 154019 nodes and 480241 elements which mainly using C3D4 type.

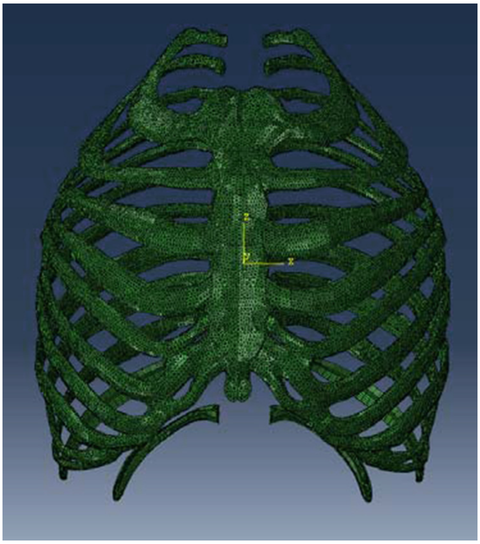

Fig. 4. Finite model of the ribs

2.5 Finite Element Model of the Heart and Lungs

The geometry of heart and lungs derives from previous study and is now modified in Hypermesh. Then they are meshed and scaled in Hypermesh to represent the 50th percentile male heart and lungs. They are simulated using solid elements and share the same material. The heart finite element model contains 3286 nodes and 15287 elements which are mixed with C3D8R and C3D4. The lungs finite element model contains 15660 nodes and 71266 elements which are all C3D4 (Fig. 5).

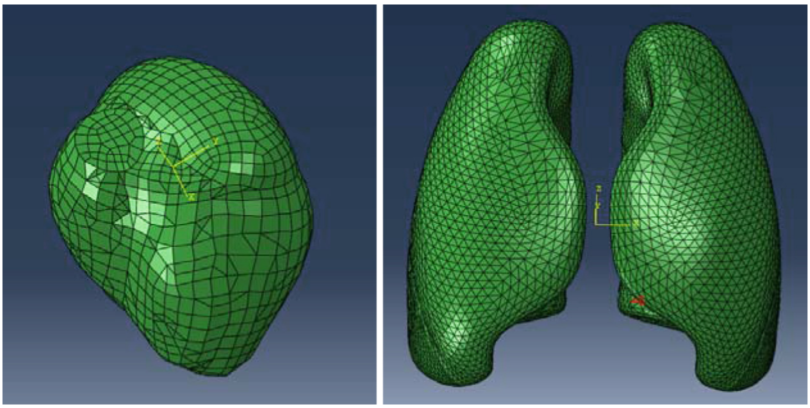

Fig. 5. Finite model of the heart and lungs

2.6 The Assembly Model

Since the different parts of the model are modeled in different coordinate systems, we should reassembly the models into one coordinate system. This work is done in the Abaqus Assembly module by rotation and translation. The assembled whole model is shown in Fig. 6.

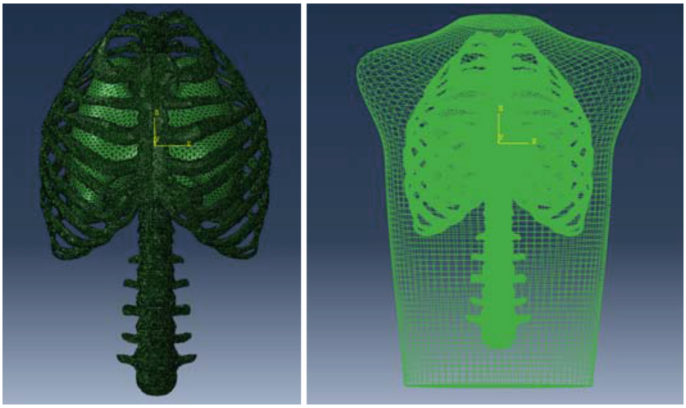

Fig. 6. The skeletal model with organs and the whole assembled model

All the anatomical components in the model are deformable and there is no rigid body. The articular joints between the ribs and the vertebrae, the facet joints between vertebrae, the joints between bones and outer soft tissue are defined by tied constraints. The interactions of the internal organs such as heart and lungs with outer soft tissue or bones are defined by contact interfaces.

3 Material Modeling

Since the thorax and abdomen consists of many different organs and each organ presents individual mechanical properties which makes it very complex to model, we simplified the whole model into two materials.

The constitutive laws for all the tissues in this model are either considered as linear elastic or linear viscoelastic. In this study, we refer the material settings in Ruan's paper [2, 10, 11]. Hard tissues, such as spine, ribs were modelled with liner elastic solid elements characterized by the stress-strain law:

$$\boldsymbol{\sigma} = \mathbf{E}\boldsymbol{\varepsilon} \tag{1}$$

with,
E = 4.2 × $\mathbf{10}^6$ Pa for spine and ribs.
Where **E** is the elastic modulus. The Poisson's ratio is set as 0.3.

Soft tissues, such as heart, lungs and other soft tissue were modelled with linear viscoelastic material characterized by:

$$\mathbf{Gt} = G_{\infty} + (G_0 - G_{\infty})e^{-\beta t} \tag{2}$$

with,

$G_0 = 2.24 \times 10^4 Pa, G_{\infty} = 7.5 \times 10^3 Pa, \beta = 0.25$ for the lungs and,
$G_0 = 4.42 \times 10^5 Pa, G_{\infty} = 1.74 \times 10^5 Pa, \beta = 0.25$ for the heart.

Where, G_0 and G_{∞} are short-term and long-term shear moduli, which govern the viscoelastic response. β is the decay factor. The density of the model tissue was maintained at 1000 $\mathbf{kg}/m^3$ to generate appropriate mass distribution throughout the model.

4 Results and Discussion

Based on the above work, finally, we established the 3D finite element model of the Chinese 50th percentile male thorax and abdomen, which consists of spine, ribs, heart, lungs and relative fresh. The mesh division and the elements of the model were 352958 nodes and 1203981 elements, the model were discriminated into 2 kind of material.

The results from this study suggested that the numerical finite element model established herein with intact structure and precise elements could be used as a powerful tool for biomechanics analysis and impact automotive safety research.

However, this finite element model was not been validated through the comparison of model predicted force-times and force-deflections with the experimental data from previous research. And also the large numbers of nodes and elements in order to simulate each detailed anatomical component will take much time to finish the calculation. The model built in this study has been simplified in material property, it may not represent the real mechanical response of human body. For real world occupant simulations, the model needs more improvement. And the large numbers of nodes and elements also need to be optimized.

Acknowledgements. Supported by the Fundamental Research Funds from Central Finance of China (282014Y-3353) and National Key Technology R&D Program (2014BAK01B05) and National Natural Science Foundation of China (51175021)

References

1. Xiao, K.: Traffic accident injuries in China (2006)
2. Ruan, J., et al.: Prediction and analysis of human thoracic impact responses and injuries in cadaver impacts using a full human body finite element model. Stapp Car Crash J. **47**, 299–321 (2003)
3. Li, F., Li, C.: Analysis on Application of Vehicle Side Impact Dummy (2007)

4. Deng, X., et al.: Finite element analysis of occupant head injuries: parametric effects of the side curtain airbag deployment interaction with a dummy head in a side impact crash. Accid. Anal. Prev. **55**, 232–241 (2013)
5. Gursel, K.T., Nane, S.N.: Non-linear finite element analyses of automobiles and their elements in crashes. Int. J. Crashworthiness **15**(6), 667–692 (2010)
6. Li, Z., et al.: Development, validation, and application of a parametric pediatric head finite element model for impact simulations. Ann. Biomed. Eng. **39**(12), 2984–2997 (2011)
7. Li, Z., et al.: Rib fractures under anterior–posterior dynamic loads: experimental and finite-element study. J. Biomech. **43**(2), 228–234 (2010)
8. Majumder, S., Roychowdhury, A., Pal, S.: Simulation of hip fracture in sideways fall using a 3D finite element model of pelvis–femur–soft tissue complex with simplified representation of whole body. Med. Eng. Phys. **29**(10), 1167–1178 (2007)
9. Moes, N.C.C.M., Horváth, I.: Using finite element models of the human body for shape optimization of seats: optimization material properties. In: Proceedings of the International Design Conference, Dubrovnik, Yugoslavia (2002)
10. Ruan, J.S., et al.: Biomechanical analysis of human abdominal impact responses and injuries through finite element simulations of a full human body model. Stapp Car Crash J. **49**, 343–366 (2005)
11. Ruan, J.S., et al.: Impact response and biomechanical analysis of the knee-thigh-hip complex in frontal impacts with a full human body finite element model. Stapp Car Crash J. **52**, 505–526 (2008)