applied sciences

MDPI

Case Report

Accident Reconstruction of Damaged Human Body Using MDCT and Computer Numerical Analysis

Eui Soo Kim

Department of Safety Engineering, Korea National University of Transportation, Chungju-si, Chungbuk 27469, Korea; es92kim@ut.ac.kr

Received: 26 March 2020; Accepted: 27 April 2020; Published: 28 April 2020

check for updates

Abstract: Techniques to analyze damage to a human body provide an important foundation to investigate the human body's dynamics during accidents. However, the systematic investigation and analysis of accidents' causes are limited due to a lack of suitable technology, personnel, and equipment. Recently, 3-D technologies and engineering verification through the finite element method have become widespread in forensics to investigate accidents' causes and dynamic environments. Bone fracture analyses can provide important information on how victims may have died and injured. In this study, 3-D images obtained from multi-detector computed tomography of personal injuries and closed-circuit television, as well as image analyses based on forensic investigations, are used in a finite element program to analyze how ribs are broken during an accident and the possibility of further body damage. Technologies that deduce stress states and mechanisms are also developed in this study using FE analyses of the reconstructed model.

Keywords: MDCT; accident analysis; body damage; bone modeling; finite element method

1. Introduction

Techniques for analyzing the damaged human body due to crashes, falling and traffic accidents provide important information in the investigation of the dynamics of the human body during these events. However, the systematic investigation and analysis of accidents' causes have been limited because the procedure, method and equipment for identifying the cause are not specified in terms of standards or regulations and there is no engineering verification in the field of normal work and household accidents. Human body damage is mainly the result of dropping and crashes. In the case of the falling accident, the human body falls free from a building and collides with the ground. This is very different from the case in which an external object is actively impacting the body in terms of the damage mechanism. In the case of the falling accident, the human body has different impact areas on the ground depending on the height of the fall. In general, the ribs are mainly fractured among damage to the human body. The rib cage is comprised of a cylindrical connection of ribs and has a certain degree of elasticity. When a force is applied on the rib cage, not only is the part on which the force acts directly susceptible to damage, but other parts on which a force is not directly applied may also break. Furthermore, as is often the case, the other side of the forced part of the rib cage can be broken when its structure is twisted. This is the reason why the distribution of rib fractures is hardly estimated in the case of crashes and accidents. Therefore, an estimation of the distribution of a rib fracture through a finite element (FE) model of the rib cage has become an indispensable part of rib fracture research. The finite element method (FEM) is widely used in the mechanical engineering field in order to divide an object into elements of virtual finite size for structure analysis. In recent years, it has been used frequently with 3D scanning technology to identify causes for engineering-based accidents such as dropping and collision [1,2]. In particular, bone fracture analyses are able to provide important information regarding the damage a victim may have sustained during an accident. Consequently, verification through actual dynamic simulations of bone fracture analyses helps increase forensic reliability [3,4]. Images from multi-detector computed tomography (MDCT) have great pathological potential in the visible representation of forensic evidence, tools, and materials. To check their relevance to crime, the kinds and directions of forces applied to a body should be analyzed for their injury mechanism, so that 3-D images obtained from MDCT can be used for the verification of injury mechanisms as well as the damage reconstruction.

2. Background of Damaged Human Body

In this study, using three-dimensional images obtained from MDCT for personal injury and closed-circuit television (CCTV) as well as image analysis based on the results of forensic investigations, FE analysis was performed utilizing a commercially available finite element program to evaluate how ribs are broken and the possibility of further body damage due to an accident. The event applied in this study is that of a man who weighs 58 kg, who crashed onto a concrete floor from a height of 21.3 m. The scene of the fall accident was recorded as shown in Figure 1. As a result of the autopsy, it was determined that his back crashed first against the floor. In this study, fracture behavior is reconstructed as a three-dimensional model using an estimation of the mechanism at the time of the crash. Technologies that are able to deduce the state of stress and the state of the mechanism will also be developed in this study by performing FE analysis of the reconstructed model.

3. Reconstruction of the Damaged Human Model

Reconstructing the damaged human model involves three steps: (1) The reconstruction of the damaged human bone from computer tomography (CT) images; (2) The conversion of CT image files to digital human bone; and (3) The reconstruction of the digital bone model. The software "MIMICS," which was developed by "Materialise NV" was used to reconstruct 3-D images from the 2-D CT scan images obtained in this study as shown in Figure 2. This software has been used mainly in the fields of biomedical and biomechanical engineering and has been certified as reliable software by ISO9001, the United States Food and Drug Administration (FDA), the European Economic Area (EEA), and European Standards (EN). This software is able to extract 3-D models from the medical CT and magnetic resonance imaging (MRI) image data using the value of a particular area of Hounsfield units (HU) or gray value of a region of interest (ROI). To convert CT image data of the damaged human bone to digital human bone, the software "MIMICS" performs 3-D object construction for the region of vertebra, sternum, rib, or costal cartilage. The relevant region is classified by setting the HU value including the relevant pixel. The imported CT image data in the "MIMICS" program is shown in Figure 2 and the converted digital human bone model is shown in Figures 3 and 4. As shown in the converted digital human model, rib and costal cartilage in the front and rear are the most damaged; the shape and orientation of these fractures and breakage can be clearly seen. Therefore, easier and more detailed analyses are possible by converting the CT image data to a 3-D digital human bone model than by using conventional 2-D images. Lastly, the damaged bone model is reconstructed to its original state to obtain the state of stress and mechanism at the time of the crash. The reconstruction can be performed by adjusting and bonding the processes of the fracture area of the damaged model. Figure 5 shows the digital bone model before and after its reconstruction.

Figure 1. The recorded information at the time of the falling accident.

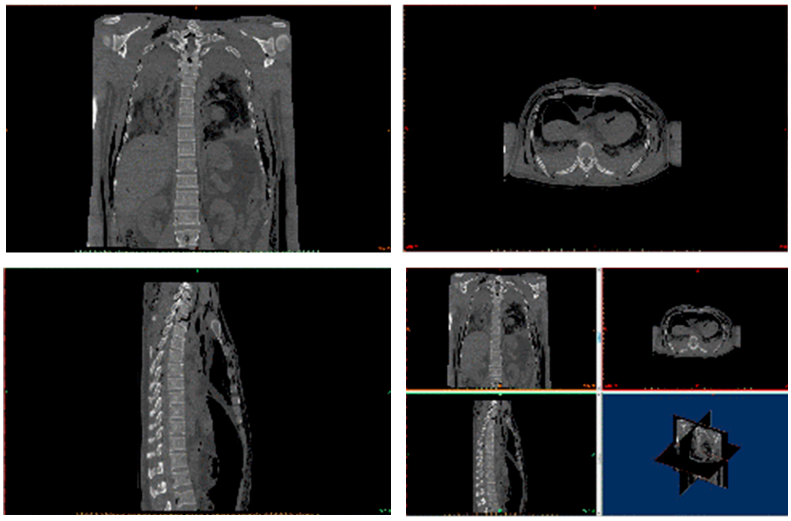


Figure 2. Imported computer tomography (CT) image data in "MIMICS" program.

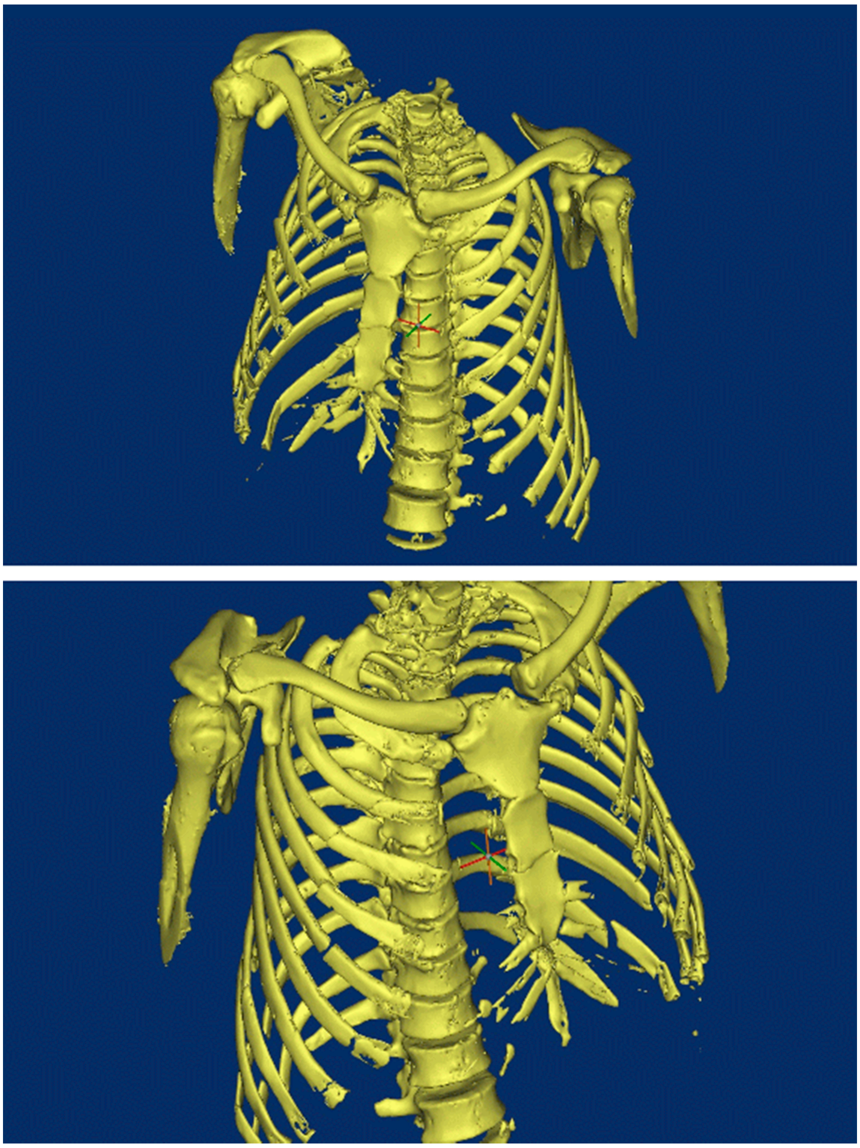


Figure 3. Transferred digital bone model from CT image data, front view.

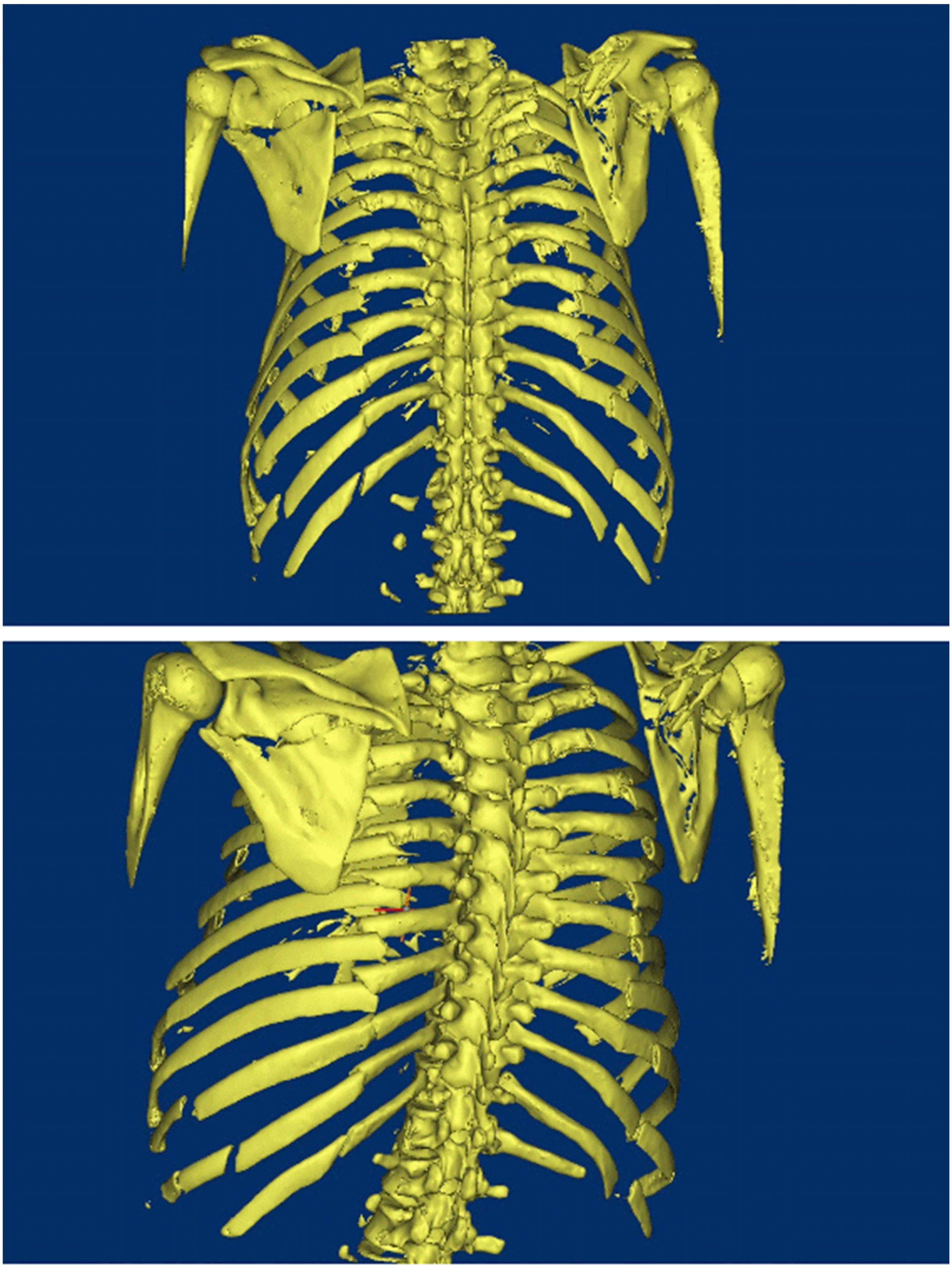
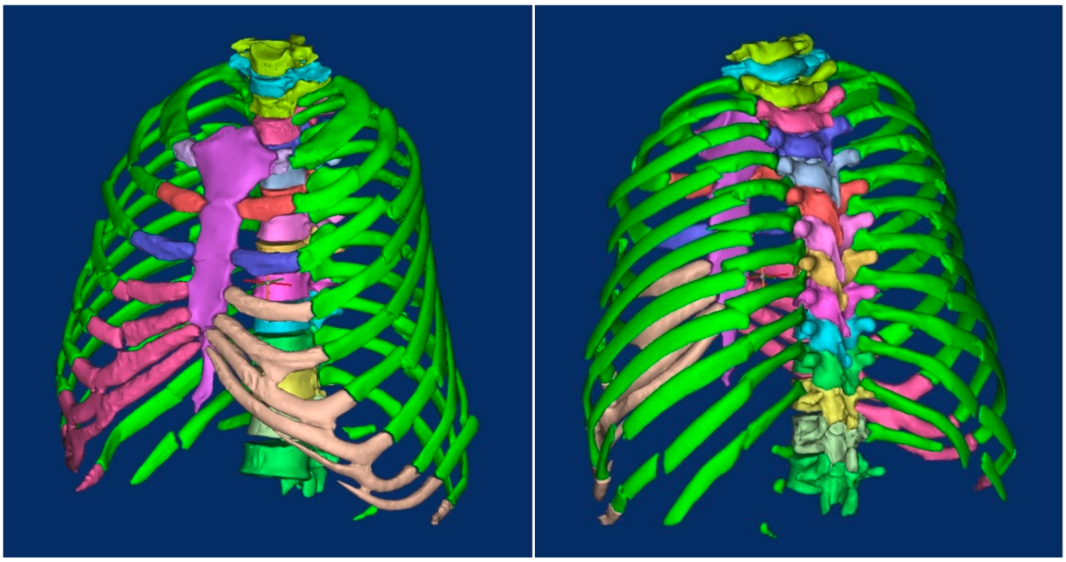
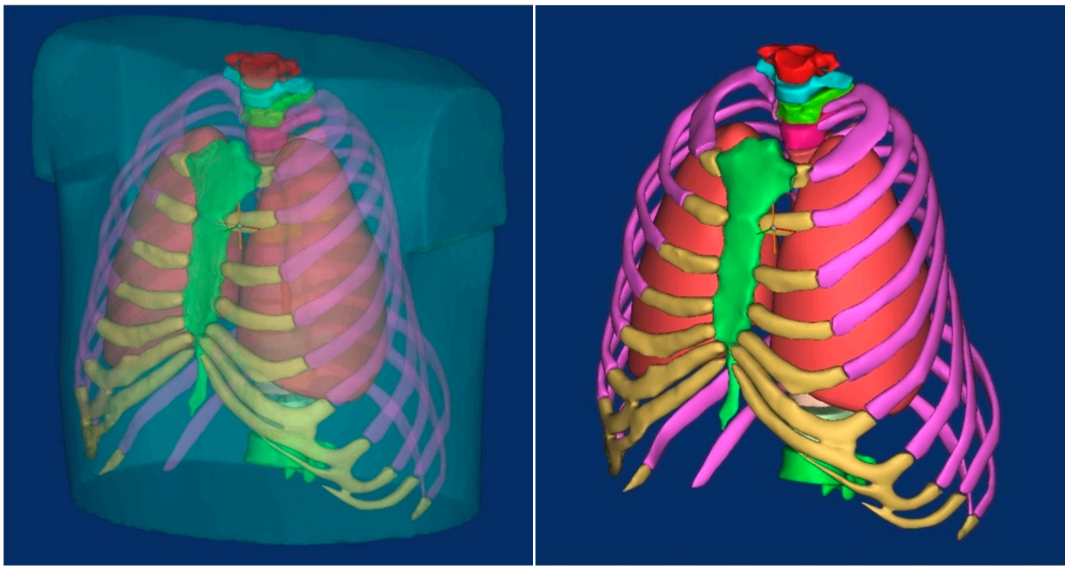


Figure 4. Transferred digital bone model from CT image data, back view.



(a) Pre-reconstruction of the digital bone model.



(b) Post-reconstruction of the digital bone model.

Figure 5. Reconstruction of the digital bone model's damaged parts.

4. Analysis of Damage Mechanism Using ADINA

In this study, post fracture analysis and accident reconstruction were carried out for the finite element model of the previously obtained digital human bone using the ADINA structure simulation, which is a commercially available finite element simulation program. ADINA enabled the coupled field analysis with FEM and FVM (finite volume method) within one solver. The goal of this study is to find out the state of stress and mechanism at the point of crash by performing finite element analysis of the human bone.

4.1. Pre-Processing of Computational Numerical Analysis

The digital human bone model of this study is composed of ribs, sternum, spine, and rib cartilage; Table 1 shows the mechanical properties of these components. There are discs between vertebrae and Table 2 shows their elastic modulus. In the case of costal cartilage, it is impossible to obtain its mechanical properties by conventional test methods such as tensile tests due to difficulties in using test specimens as costal cartilage is limited in size. Therefore, an instrumented indentation technique that uses AIS3000 and a specific jig for this study was applied to obtain the mechanical properties of costal cartilage [5,6]. Using the specimens collected from the body, the elastic modulus, yield strength, and tensile strength were obtained and their average values calculated from the results of 15 tests were used and are shown in Table 1. Figure 6 shows a photograph of the specimen of costal cartilage and the results of the tests. The conversion of human bone to a finite element model is necessary before performing a finite element analysis of the reconstructed digital human bone model. Figures 7 and 8 show a finite element model of the digital bone model. Generating a finite element model, the connection of nodes between the ribs and cartilage was constructed using a rigid link element and the connection of nodes between the spines was constructed using a spring element. The mechanical properties of the connections are designated as those of the disc. Finite element analysis was performed by applying the appropriate mechanical property to each model of the converted digital human bone model. To obtain compatibility and reliability between 2-D CT images and a 3-D human bone model, skin and lung tissue are excluded and analysis was performed only for the pure human bone. The 3-D model of the human bone and the number of elements was 398,174. To obtain the value of impact force, a mechanical environment is assumed and estimated using three-dimensional images obtained from MDCT for personal injury and CCTV image analysis based on the results of the investigation of the forensic point of view. Potential energy from 21.3 m high is fully converted to kinetic energy by the principle of the conservation of energy. The speed of the falling body at the ground is calculated as Equation (1).

$$v_{final} = \sqrt{2gh_{initial}} \tag{1}$$

Here, v_{final} is the speed of the falling body at the ground, g is acceleration of gravity and $h_{initial}$ is height of the falling. From the image analysis of the CCTV information of the event, delay time was measured as 3.56 ms and finally, impact force is calculated by Equation (2).

$$\text{Impact Force} = \frac{mv_{final}}{\Delta t} \tag{2}$$

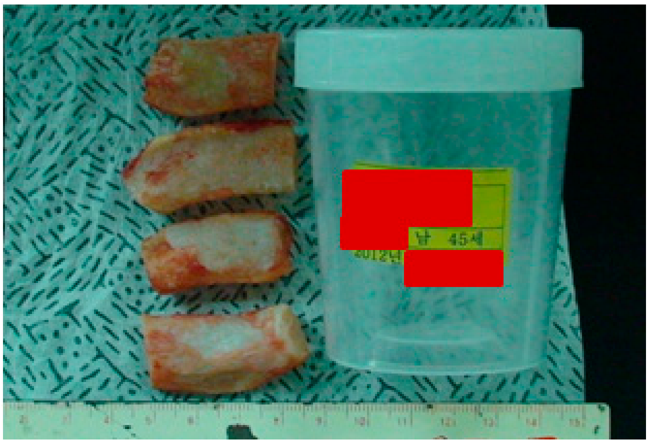
Here, m is the weight of the body and Δt is the delay time. Boundary conditions and load conditions are shown in Figure 9 considering the calculated impact force, 5750 N. Each 50 N was applied to 115 nodes which are estimated to contact the ground as shown in Figure 9. In the case that the body falls freely from 21.3 m high, finite element analysis is performed. The analysis starts at 2 mm away from the rigid wall and the initial speed is calculated as 20.4417 m/s by Equation (1). Figure 10 shows the finite element model and the boundary condition for the analysis of the falling body.

Table 1. Material properties for each part of bone.

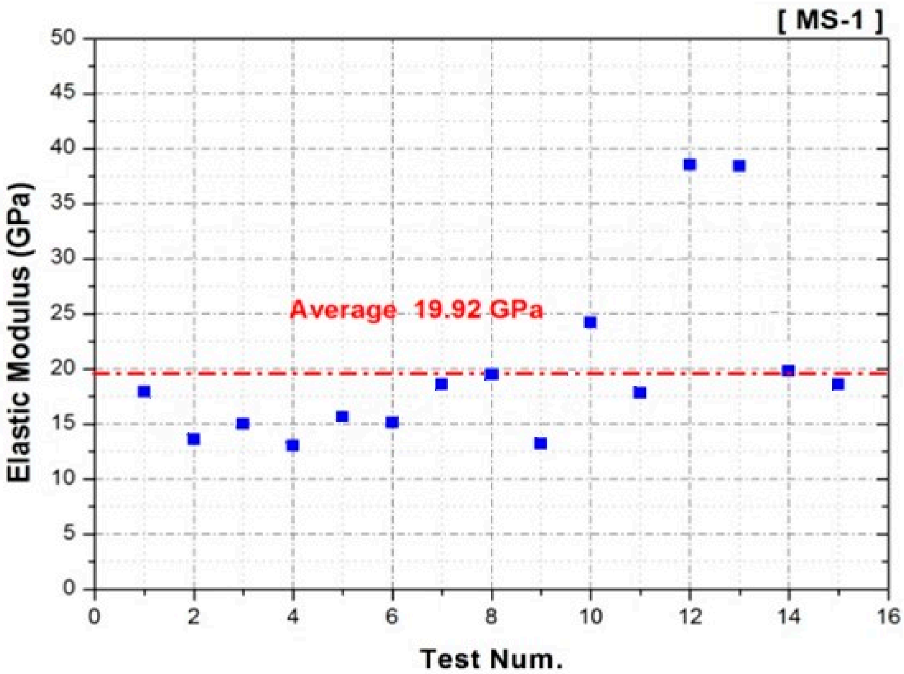
	Young's Modulus	Density (kg/m^3)	Poisson's Ratio	Yield Strength (MPa)	Tension Strength (MPa)
Rib	13.9 GPa	1561	0.3	93.9	124.2
Sternum	3.51 GPa	1354	0.387	34.48	48.27
Vertebra	4.67 GPa	1500	0.3	52.62	69.06
Costal Cartilage	19.92 MPa	1203	0.4	141.32	285.94

Table 2. Elastic modulus of disc according to direction.

Direction	Value(N/mm)
Tensile direction	486
Compressive direction	3300

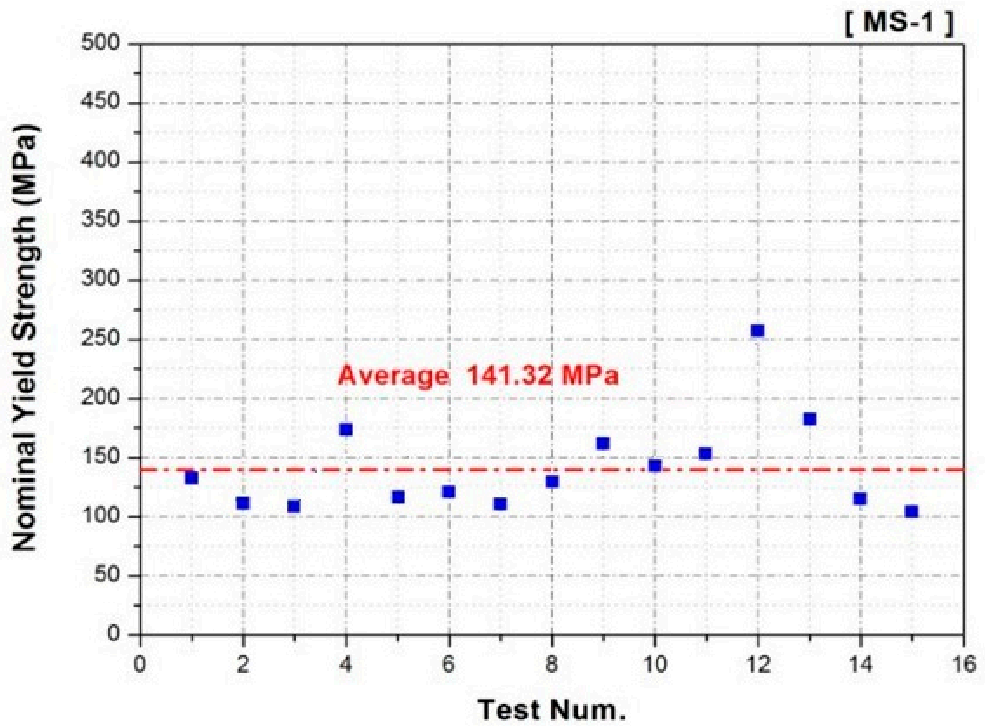


(a) Photograph of the specimen.

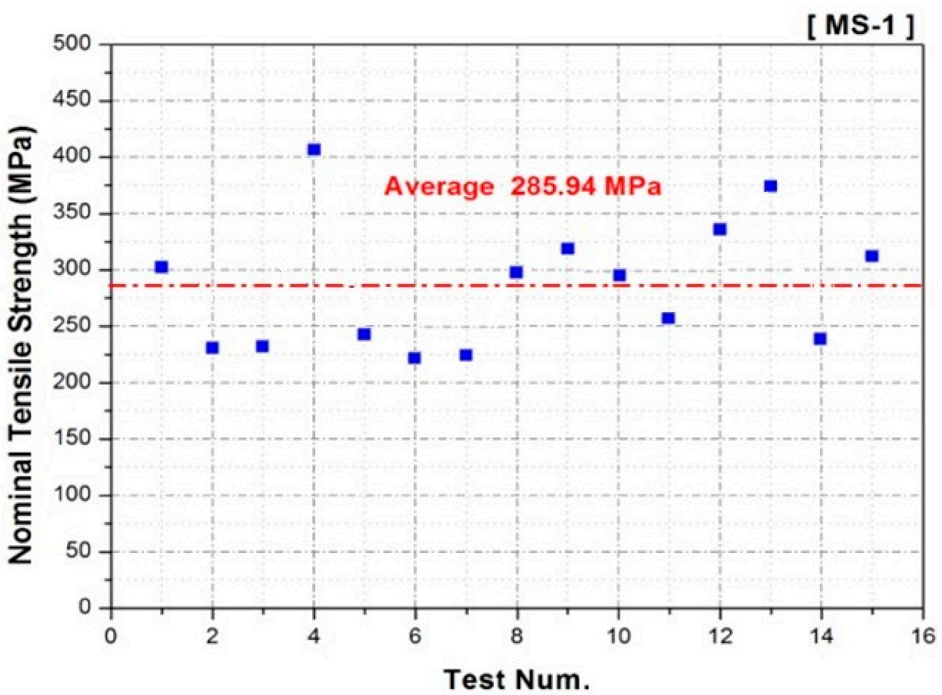


(b) Elastic modulus obtained from the tests.

Figure 6. *Cont.*



(c) Nominal yield strength from the tests.



(d) Nominal tensile strength from the tests.

Figure 6. Photograph of test specimen and the results of mechanical properties.

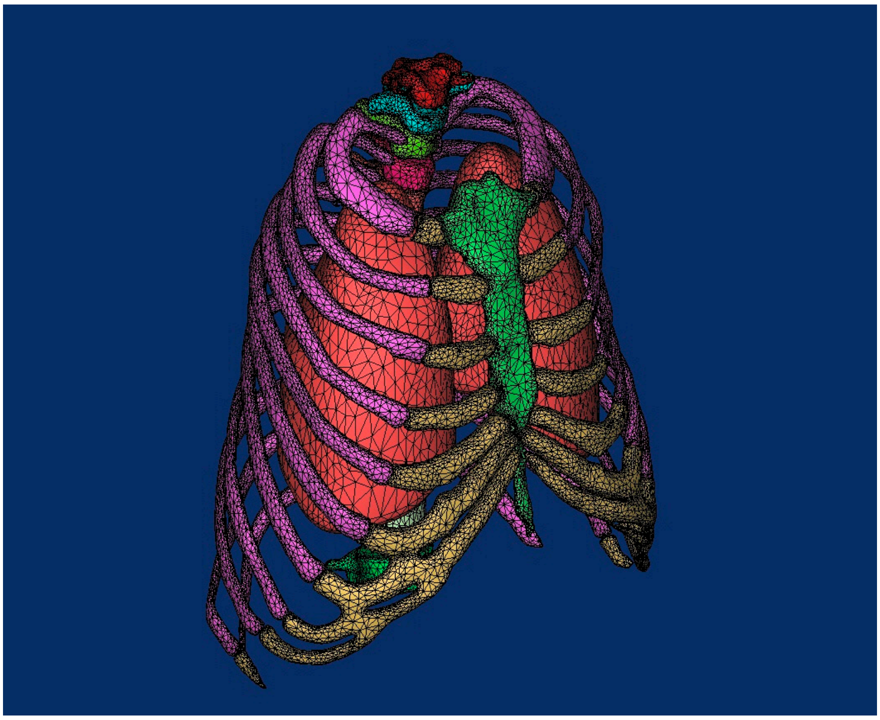


Figure 7. Finite element model of the digital bone model.

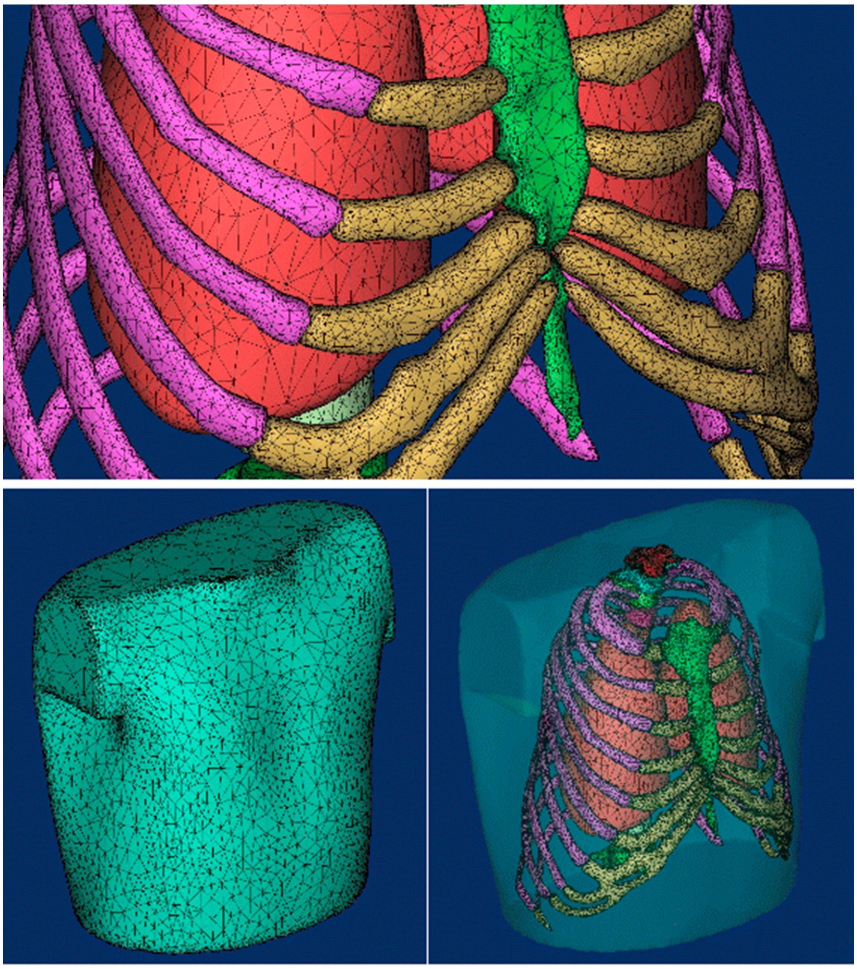


Figure 8. Finite element model of the digital bone model.

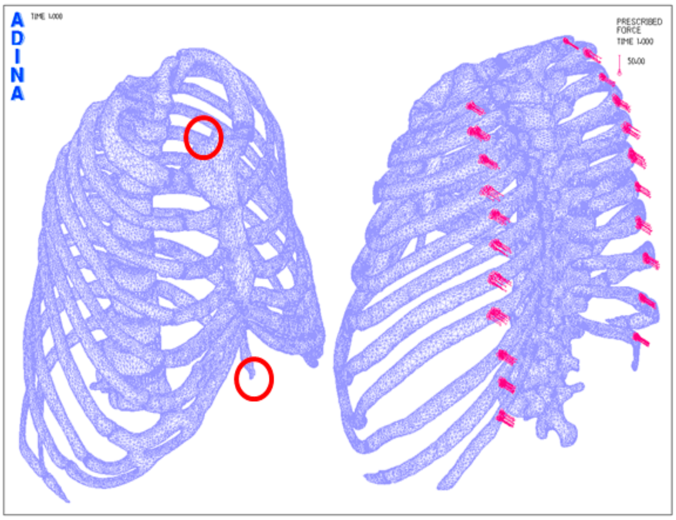


Figure 9. The finite element model and the boundary condition.

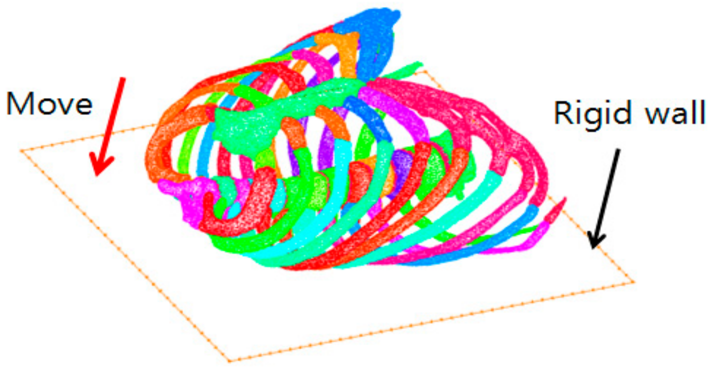
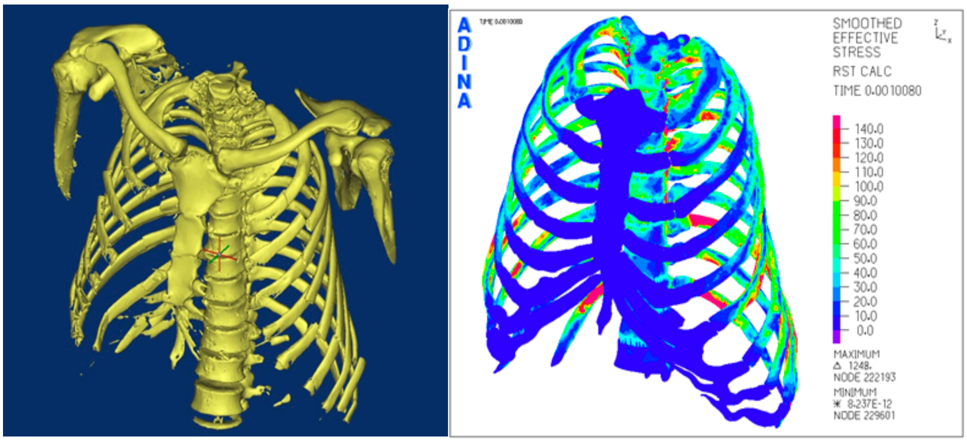


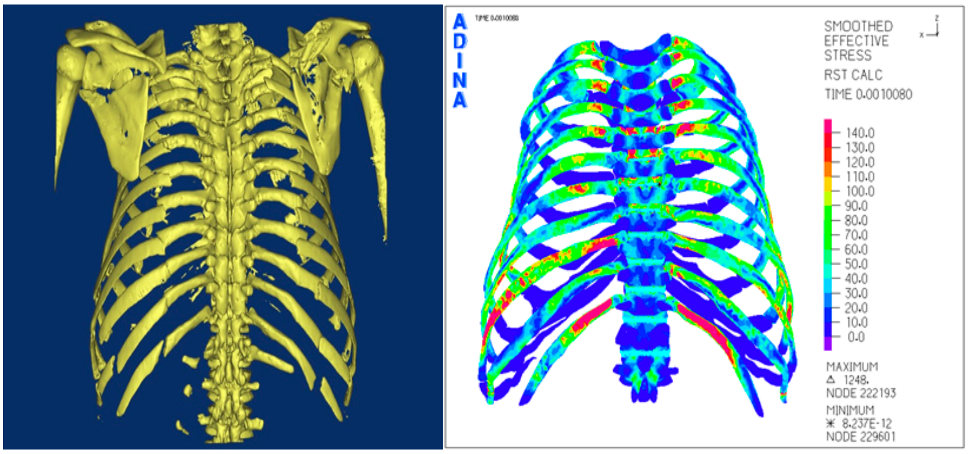
Figure 10. The schematic boundary condition for the analysis of the falling body.

4.2. Post-Processing of Computational Numerical Analysis

As a result of finite element analysis using this model and the aforementioned conditions, the maximum stress was distributed around rib(1), costal cartilage(1), vertebra(1), and the sternum on which the force was directly applied, as shown in Figure 11. Furthermore, the load was delivered to a rib below the sternum, so that a smaller distribution of stress could be found. As there is no stress concentration around the rigid beam element, it was concluded that load could be easily delivered. Figure 12 shows stress variation as a function of time for stress analysis. The maximum stress was found at point 1, which is the top part of the rib; point 4 had the lowest stress value. In the case of costal cartilage, the maximum stress was found at point 5. In the case of the vertebrae, point 7 had a higher value of stress than point 8, which means that higher stress was applied to the vertebrae located in higher positions. In particular, in the rib and the costal cartilage in the front and rear of the upper part, the stress value is over 140 MPa. This value exceeds the yield strength of the rib, 93.9 MPa and the tensile strength, 124.2 MPa as well, which means a fracture of the rib on the same part. In the front of the sternum, the stress value ranges up to 140 MPa from which it is inferred that the impact from the back has a great effect on the front. These results are in good agreement with the damaged bone model in Section 2.



(a) Isotropic view.



(b) Front view.

Figure 11. Result of stress distribution for finite element analysis.

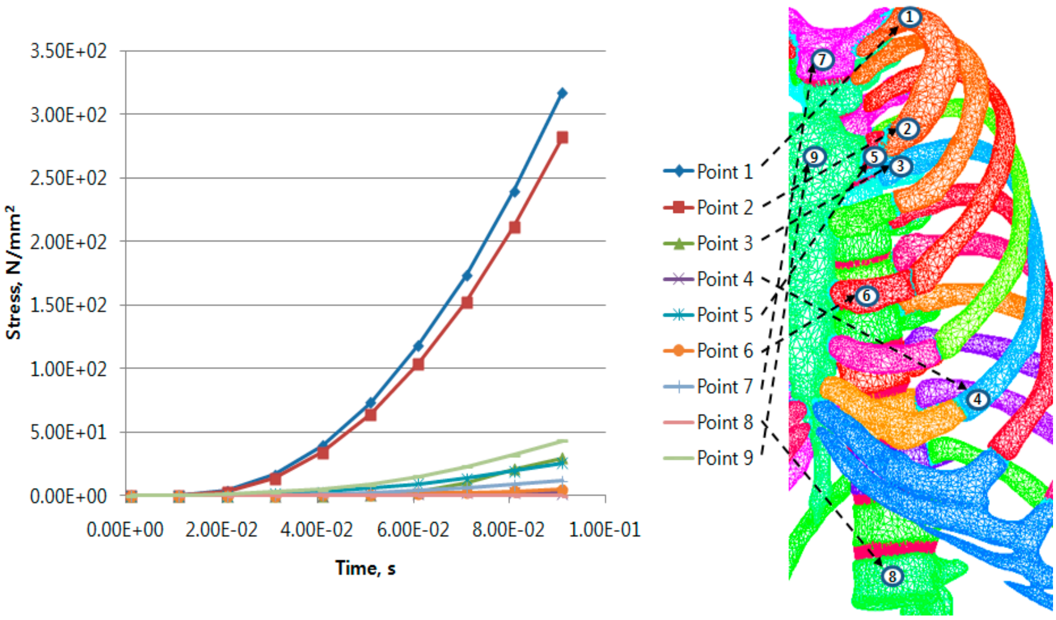


Figure 12. Stress according to time variance on each section.

5. Conclusions

The goals of this study were to find the stress states and mechanisms at the point of a crash by performing finite element analyses of the human bone and to develop technologies that can deduce the state of stress and its mechanisms by performing FE analysis of a reconstructed model. In this study, a 3-D digital human model was reconstructed using CT of the damaged human bone. The finite element model was obtained by a meshing operation using "MIMICS" software and finite element analysis was performed using the mechanical property as shown in the previous table. By performing finite element analysis using a transformation model, the mechanism regarding fracture of the ribs due to the falling accident was verified in terms of engineering by deriving stress-time curves for the stress generated in ribs, costal cartilage and vertebrae. As a result of the analysis in this study, it was found that the location of the maximum stress obtained from the analysis is in good agreement with that of the actual fracture, which proves that our analysis methodology has adequate reliability in forensic fields. As a future study, comparison between analysis data and the actual accident data considering the impact force should be performed for reliability. Furthermore, finite element analysis will be able to be performed for digital bone models including skin tissue and lung tissue, based upon the previous analyses of digital bone models.

Funding: This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education (2017R1D1A1B03028163).

Conflicts of Interest: The author declares no conflict of interest.

References

1. Greer, A. Numerical Modeling for the Prediction of Primary Blast Injury to the Lung. Master's Thesis, University of Waterloo, Waterloo, ON, Canada, January 2007.
2. Bass, C.R.; Salzar, R.; Davis, M.; Lucas, S.; Donnellan, L.; Folk, B. Injury risk in behind armor blunt thoracic trauma. *Int. J. Occup. Saf. Ergon.* **2006**, *12*, 429–442. [CrossRef] [PubMed]
3. Dupre, S. Modelisation en Elements Finis du Complexe de L'epaule et Simulation de Sa reponse a un Choc Lateral. Ph.D. Thesis, L'institut National des Sciences Applique's de Lyon, Villeurbanne, France, 2007.
4. Stemper, B.D.; Board, D.; Yoganandan, N.; Wolfla, C.E. Biomechanical properties of human thoracic spine disc segments. *J. Craniovertebral Junction Spine* **2010**, *1*, 18–22.
5. Lau, A.G.; Oyen, M.L.; Kent, R.W.; Murakami, D.; Torigaki, T. Indentation stiffness of aging human costal cartilage. *Acta Biomater.* **2008**, *4*, 97–103. [CrossRef] [PubMed]
6. Kemper, A.R.; McNally, C.; Kennedy, E.A.; Manoogian, S.J.; Rath, A.L.; Nq, T.P.; Stitzel, J.D.; Smith, E.P.; Duma, S.M.; Matsuoka, F. Material properties of human rib cortical bone from dynamic tension coupon testing. *Stapp. Car. Crash J.* **2005**, *49*, 199–230. [PubMed]