Finite element analysis of pedestrian lower limb fractures by direct force: The result of being run over or impact?

Zhengdong Li a,1, Donghua Zou a,1, Ningguo Liu a, Liangwei Zhong b, Yu Shao a, Lei Wan a, Ping Huang a,* Yijiu Chen a,*

a Shanghai Key Laboratory of Forensic Medicine, Institute of Forensic Science, Ministry of Justice, 1347# West Guangfu Road, Shanghai 200063, China
b College of Mechanical Engineering, University of Shanghai for Science and Technology, 516# J ing Road, Shanghai 200093, China

A R T I C L E   I N F O
Article history:
Received 30 December 2011
Received in revised form 7 December 2012
Accepted 4 March 2013
Available online

Keywords:
Lower limb injury
Finite element model
Forensic medicine
Biomechanics
Accident reconstruction

A B S T R A C T
The elucidation and prediction of the biomechanics of lower limb fractures could serve as a useful tool in forensic practices. Finite element (FE) analysis could potentially help in the understanding of the fracture mechanisms of lower limb fractures frequently caused by car–pedestrian accidents. Our aim was (1) to develop and validate a FE model of the human lower limb, (2) to assess the biomechanics of specific injuries concerning run-over and impact loading conditions, and (3) to reconstruct one real car–pedestrian collision case using the model created in this study. We developed a novel lower limb FE model and simulated three different loading scenarios. The geometry of the model was reconstructed using Mimics 13.0 based on computed tomography (CT) scans from an actual traffic accident. The material properties were based upon a synthesis of data found in published literature. The FE model validation and injury reconstruction were conducted using the LS-DYNA code. The FE model was validated by a comparison of the simulation results of three-point bending, overall lateral impact tests and published postmortem human surrogate (PMHS) results. Simulated loading scenarios of running-over the thigh with a wheel, the impact on the upper leg, and impact on the lower thigh were conducted with velocities of 10 m/s, 20 m/s, and 40 m/s, respectively. We compared the injuries resulting from one actual case with the simulated results in order to explore the possible fracture bio-mechanism. The peak fracture forces, maximum bending moments, and energy lost ratio exhibited no significant differences between the FE simulations and the literature data. Under simulated run-over conditions, the segmental fracture pattern was formed and the femur fracture patterns and mechanisms were consistent with the actual injury features of the case. Our study demonstrated that this simulation method could potentially be effective in identifying forensic cases and exploring the injury mechanisms of lower limb fractures encountered due to inflicted lesions. This model can also help to distinguish between possible and impossible scenarios.

1. Introduction

More than 1.17 million people die in traffic accidents around the world every year, and 65 percent of deaths involve pedestrians [1]. Meanwhile, 85 percent of pedestrian casualties include lower limb injuries, a much higher level compared to motorized vehicle occupants [2]. Lower limb injuries are the most common in pedestrian casualty forensic practices. Legal representatives rely on the expertise of forensic pathologists to determine the most probable type of accident scenario based upon examination findings. In general, impact and run-over injuries by direct force can be distinguished according to characteristic injured skin, subcutaneous hemorrhage, and fractures [3,4]. However, the elucidation of injury mechanisms is difficult when soft tissue injuries are mild. One potential approach to assess the injury mechanisms by biomechanical evaluation is the application of the FE model [5].

In recent years, several lower limb FE models [6–9] have been developed to improve vehicular design parameters and evaluate potential human protection in car–pedestrian crashes. However, due to the difficulties of rebuilding the complex anatomical geometry model, as well as the reproducing accurate injury patterns, the FE models have not been extensively researched and adopted in forensic science. FE models and head injury models are currently employed to assess the mechanisms of head injury during shaking, impact, fall, and gunshot scenarios in forensic...
practices [10–19], thus they can potentially serve as an effective tool for forensic investigation to distinguish various scenarios and explore possible lower limb injury mechanisms.

In the present study, a complete subject-specific lower limb model was developed to evaluate the bio-mechanism of bone tissue under the conditions of being run-over and impacted. Injuries resulting from one accident case were compared with various FE simulation results under different scenarios in order to explore lower limb injury bio-mechanism concerning Von Mises stress, maximal principle strain, and fracture patterns.

2. Materials and methods

2.1. FE model

The lower limbs of a postmortem female corpse was scanned using 40-Slice Computed Tomography (MSCT, Siemens Ltd., Germany) with xy-resolution of 512 × 512 pixels. There were 994 total scan images imported into specialized medical processing software, Mimics (Mimics 13.1, Materialize Inc., Belgium).

The image series was oriented anterior to posterior, and upon selecting an appropriate threshold value between 1250 and 2753 HU (Hounsfield Unit) with the “Thresholding” function of Mimics, all of the bones were isolated from the soft tissue. The femur, tibia, fibula, and patella were then separated with the “Region Growing” feature to form a series of masks. Small cavities in the masks were eliminated manually using the “Edit Masks” feature. After completion of the manual operations, all bone and soft tissue masks with high-quality were reconstructed into a three dimensional model with the anatomical features of the human lower limb. Because the original geometrical surface was rough and discontinued, we applied the “Smooth” and “Wrap” functions to make the model smoother. The smooth model was converted into STL (Standard Triangulated Language) format using the “Remesh” function. By utilizing several remesh algorithms and user-specified parameters, STL triangular numbers were reduced and reshaped to create an even mesh. Next, the lower limb surface model was imported to ICEM CFD software (ICEM CFD 12.1, ANSYS Inc., USA) in order to generate a lower limb FE model.

To obtain higher speed and precision computation, the femur, tibia, fibula, and patella were meshed and refined manually with 8-node hexahedral elements. For the remaining surrounding soft tissue zone, tetrahedral FE mesh was automatically generated based on the geometrical shape created by the Mimics. The major knee ligaments, including the patellar ligaments, the medial collateral ligament (MCL), the anterior cruciate ligament (ACL), the posterior cruciate ligament (PCL), and the lateral collateral ligament (LCL), were created and meshed using quadrilateral shell elements. The menisci was constructed and meshed with hexahedral elements manually. In total, the lower limb model consisted of 375,919 solid elements and 402 shell elements. The FE model of a 300 mm × 70 mm × 50 mm impact hammer was meshed with 11,940 8-node hexahedral elements.

The quality of the FE model for computational reliability and stability was verified by checking the distribution of multiple element geometric parameters such as aspect ratio and element quality. We limited the worst value of the mesh quality parameters to values that would not cause model failure, taking aspect ratio ≤ 5 and the element quality ≥ 0.3 for example [6].

2.1.1. Material constitutive model and properties assignment

All the material models were chosen from the existing LS-DYNA material library. Material mechanical properties were selected for published literatures in which several experiments were conducted on the lower limb bones and soft tissue. The cortical and trabecular bones were defined as simplified isotropic elasto-viscoplastic material [20]. To characterize the passive compressive properties of flesh and skin under dynamic loading conditions, viscoelastic material models were employed. Other soft tissues, such as ligament and cartilage, were defined as elastic models. All soft tissue densities were assumed to be identical. The mechanical properties for the lower limb are summarized in Table 1.

2.1.2. Lower limb bone fracture tolerance limits

In order to accurately explore bone fracture mechanisms due to direct force, we employed the element elimination approach to simulate the bone fracture based on the failure strain and the ultimate stress. As found in the literatures, the stress-based and strain-based tolerance limits were used commonly to predict the bone fracture. Hence, the combination of ultimate stress and maximum principle strain were employed to predict the potential bone fracture in the LS-DYNA code. All stress and strain thresholds are summarized in Table 1. In the table, \( \sigma_{\text{fract}} \) refers to the maximum tensile strain and \( \sigma_{\text{fract}} \) refers to the maximum compressive strain. \( \sigma_{\text{fract}} \) is the equivalent stress at failure. The stress and strain failure criteria were applied to the model elements independently. Once any of the defined criteria is satisfied, the element is deleted automatically from the computation.

2.1.3. Model assembly

The developed lower extremity model consisted of the main anatomical features in which all parts of the model utilized one integration point element. An hourglass control with hourglass coefficients ranging from 0.03 to 0.05 was used to prevent zero energy modes. The TIED_NODES_TO_SURFACE contact algorithms were utilized to simulate the attachment between ligaments and bones. The left contacts used surface-to-surface contact algorithms with a friction coefficient of 0.1. Elements of the middle femur region were constrained by the CONSTRAINED_TIED_NODES_FAILURE algorithms to simulate the fracture initiation and crack propagation. The mechanical properties of the model were able to identify the potential risk of fracture and the fracture onset location in the simulation test.

2.2. Model validation

The developed model was verified by comparison between simulated responses and PMHS test data found in published reports. We chose the experimental tests based on the details regarding impact conditions, model postures, and PMHS test results. Three point bending tests of the bone tissue and overall lower limb, as well as lateral impact to the limb, were executed to evaluate the model. References for the chosen PMHS tests are summarized in Table 2.

2.2.1. Three-point bending tests of femur, tibia, and fibula

The bone model was subjected to a quasi-static and dynamic three-point bending test at the mid-shaft to corroborate the mechanical behavior of the hard tissues. The force-displacement curve, fracture force and bending moment generated were compared with the PMHS test data. Two ends of the long bone were potted in the roller supports and were free to rotate on the base plane. The tibia and fibula were validated using quasi-static tests. The bone was constrained to

### Table 1

<table>
<thead>
<tr>
<th>Material models and properties used in lower limb FE model.</th>
<th>LS-DYNA material type</th>
<th>Material properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Femur/patella [8,41]</td>
<td>Elasto-viscoplastic</td>
<td>( \rho = 1990 \text{kg/m}^3 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( E = 14635 \text{MPa}, v = 0.3435, \sigma_{\text{fract}} = 133 \text{MPa} )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( E_t = 1024 \text{MPa}, \sigma_{\text{fract}} = 0.0104 )</td>
</tr>
<tr>
<td>Tibia/fibula [42]</td>
<td>Elasto-viscoplastic</td>
<td>( \rho = 1990 \text{kg/m}^3 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( E = 18,500 \text{MPa}, v = 0.3, \sigma_{\text{fract}} = 146 \text{MPa} )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( C = 360.7, P = 4.605 )</td>
</tr>
<tr>
<td>Flesh/skin [6]</td>
<td>Quasi-linear viscoelastic</td>
<td>( \rho = 1000 \text{kg/m}^3 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( C_1 = 0.12 \text{kJ}, C_2 = 0.25 \text{kPa} )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( S_1 = 1.2, S_2 = 0.8, T_1 = 23, T_2 = 63 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( k = 20 \text{kPa} )</td>
</tr>
<tr>
<td>Patellar ligaments [43]</td>
<td>Elastic</td>
<td>( \rho = 1000 \text{kg/m}^3 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( E = 459.3 \text{MPa}, v = 0.3 )</td>
</tr>
<tr>
<td>Other ligaments [43]</td>
<td>Elastic</td>
<td>( \rho = 1000 \text{kg/m}^3 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( E = 303.9 \text{MPa}, v = 0.3 )</td>
</tr>
<tr>
<td>Menisci [44]</td>
<td>Elastic</td>
<td>( \rho = 2000 \text{kg/m}^3 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( E = 110 \text{MPa}, v = 0.2 )</td>
</tr>
</tbody>
</table>
rotate in the lateral–medial direction. The impactor was defined as a 25-mm diameter cylindrical rigid body moving in the vertical direction with a constant velocity of 1 m/s. The quasi-static test was conducted using implicit methods to avoid the strain-rate influence and inertia effect. The fracture force and bending moment were generated to corroborate the effectiveness and sensitivity of the bone model.

In the dynamic test, a 12-mm diameter cylindrical impactor was used to apply the dynamic load at the mid shaft of the femur in the anterior–posterior direction moving vertically at a constant speed of 1.5 m/s. The impact contact was created based on the penalty method. The force–deflection curve, peak force and bending moment were calculated to verify the mechanical properties of the femur model. Contact between bone and roller supports was defined as a shared node with no sliding or friction.

2.2.2. Dynamic three-point bending test of the lower limb

To validate the dynamic bending load response of the overall limb FE model, we simulated the reported three-point bending tests [6,21,22] with surrounding soft tissue. The ends of the model were inserted into the supported rollers that were placed onto flat supports with frictionless contact. The loading conditions were identical to the bending tests in literature; the load was exerted in the lateral-medial direction in the middle of the model with a speed in the range of 2.1–4.2 m/s. The rollers and impactor were modeled as rigid materials. The peak force and bending moment-history deflections were used to validate the model.

2.2.3. Lateral impact to the lower limb

This test was conducted to characterize the compressive response of the soft tissue upon lateral impact. The soft tissue surrounding the bones was compressed when the crash occurred, and absorbed the kinetic energy to alleviate lower limb injury. The loading condition was close to the lateral impact experiment performed by Dhaliwal [23], which was applied to the FE model during the simulation. The proximal femur, distalibia and fibula nodes were rigidly constrained. A 1.84 kg impactor with dimensions of 45 mm × 142 mm impacted the leg model at a speed of 2.5 m/s. The maximum force, maximum deflection, and ratio of lost kinetic energy were obtained from the simulation. The maximum force was derived from the contact between the impactor and soft tissue. The ratio of energy loss was directly calculated by LS-DYNA in the following equation:

\[ E = 1 - \frac{E_{\text{final}}}{E_{\text{initial}}} \]

*E* refers to the ratio of the absorbed proportion of initial kinetic energy divided by the initial kinetic energy. \( E_{\text{final}} \) and \( E_{\text{initial}} \) represent the final kinetic energy and initial kinetic energy, respectively, of the impactor as calculated by LS-DYNA.

2.2.4. Accident reconstruction

An adult woman was impacted by a Santana taxi when she was walking across the road. She lay on the road next to the Santana when the driver and other passersby were waiting for the police and ambulance. When this happened, another car passed over the dying woman and ran away from the scene unexpectedly. The second car driver was found next day while he insisted that he had nothing to do with the accident. Since no significant damages and traces were found on the second car, there was a doubt concerning whether the victim died from being run over by the second car or impacted by the Santana.

Upon the forensic examination, there were contusions on the right tempus and right pars zygomatica, bruised lacerations on the right chin, basioccipital fractures and hemorrhage in both external ear canals, abrasions on the right elbow, slight contusion on the lateral tibial plateau with 35 cm height from the plantar. Besides, segmental fracture of the right femur was revealed by the inspection of the autopsy and CT scan. The maximum length between two ends of the wedge-like fracture segment was 16.5 cm and the minimum was 8.5 cm. There were neither apparently visible tire marks on the outer clothes and skins nor avulsion injuries, but slight skin bruises and subcutaneous hemorrhage in the right thigh.

In classical expertise, the serious cranio-cerebral injuries tended to be formed by falling onto a blunt plane (such as road surface and car hood). Additionally, the segmental fracture and skin contusion on the lower limb might be formed by running-over or collision. Therefore, the fracture mechanism of the lower limb was the bone of contention about who should be responsible for the death of the victim.

Based on the inferences, we compared forensic findings to different FE simulated results in order to validate and determine which accident scenario most likely occurred.

2.2.5. FE simulations of three different loading scenarios

We assessed the consequences of possible crash scenarios using the FE lower extremity model we developed. Three types of loading conditions were applied on the FE model. Three sketches in Fig. 1 illustrate the three loading conditions separately, helping to understand the possible scenarios.

1. The FE model was located on a rigid plane with a lumped mass of 30 kg equally distributed to the femoral head nodes to simulate an impact scenario (Fig. 1a). A simplified deformable bumper model like the Santana bumper, with a height of 35 cm impacted the proximal tibia with a speed of 10 m/s, 20 m/s, and 40 m/s. To characterize the mechanical behavior of the simple bumper we used a density of 7890 kg/m³, Young modulus of 210 GPa, Poisson ratio of 0.3 and yield stress of 222 MPa.

2. The FE model was placed on the rigid plane horizontally to simulate a run-over scenario (Fig. 1b). A simplified wheel FE model of 370 kg (a quarter of the total Santana mass) performed a run-over of the model at a speed of 10 m/s, 20 m/s, and 40 m/s. The base width of the wheel was 185 mm. To characterize the mechanical behavior of wheel with Young modulus of 2.461 GPa, and Poisson ratio of 0.323, the mass of 370 kg was uniformly distributed into every nodes of the model.

3. The loading conditions were similar to the initial loading scenario aside from the fact that the bumper impactor model impacted the lower third of the thigh (Fig. 1c). This simulation was used to explore whether the direct impact could form the segmental fracture patterns.

Various loading scenarios were calculated. The fracture locations and patterns were simulated by the element elimination approach. We drew effective stresses from the elements at both of the two ends and middle of the segmental fracture, respectively, under the second loading scenario. Strains were obtained from elements at both proximal tibia and distal femur under the first and third loading scenarios. We calculated averages of strain and stress at certain model parts in

---

**Table 2**

<table>
<thead>
<tr>
<th>Loading type</th>
<th>FE model</th>
<th>Direction</th>
<th>Load type</th>
<th>References</th>
</tr>
</thead>
<tbody>
<tr>
<td>Quasi-static</td>
<td>Tibia</td>
<td>L-M</td>
<td>Three-point bending</td>
<td>Rabl [28], Laurence [29]</td>
</tr>
<tr>
<td></td>
<td>Fibula</td>
<td>L-M</td>
<td></td>
<td>Kress [27]</td>
</tr>
<tr>
<td>Dynamic</td>
<td>Soft tissue</td>
<td>L-M</td>
<td>Compression</td>
<td>Dhaliwal [23]</td>
</tr>
<tr>
<td></td>
<td>Femur</td>
<td>P-A</td>
<td>Three-point bending</td>
<td>Untaroudi [26], Funk [25]</td>
</tr>
<tr>
<td></td>
<td>Lower limb</td>
<td>L-M</td>
<td></td>
<td>Kerrigan [21], Ivarsson [22], Kennedy [30]</td>
</tr>
</tbody>
</table>


---

**Fig. 1.** Scenario sketches of different loading conditions: (a) bumper collision with the tibial plateau as in loading condition 1; (b) wheel running over the thigh as in loading condition 2; (c) bumper collision with the mid femur as in loading condition 3.
different loading conditions to draw strain and stress versus time curves. These curves and the fracture patterns were used to analyze the mechanisms of the injury.

2.3. Data analysis

The data derived from the simulation were expressed in terms of mean ± SD. The analyses were performed using Stata 7.0. The analytical parameters consisted of Von Mises stress, strain and bending moment [24].

3. Results

3.1. The FE model generated

The lower limb FE model was created from the CT scans as shown in Fig. 2. The bone tissues modeled in the 8-node hexahedral solid elements include the femur, tibia, fibula, and menisci components. All the femur–patella and femur–tibia joint ligaments were modeled as 4-node shell elements connected to the bone tissues via tie-break contact. The surrounding flesh and skin were meshed with a hex-dominant methodology because of the complicated interior and exterior surfaces. The overall FE model consisted of 375,919 solid and 402 shell elements. This subject-specific FE model exhibits geometric similarity.

3.2. Three-point bending tests of femur, tibia, and fibula

We compared the predicted force-deflection curve produced by the femur model to the published PMHS test results (Fig. 3a) [25,26]. The fracture occurred in proximity to the crash region, as determined by clinical practice and experiments at the peak impact force of 3380 N, which was within the scope of the reference data. The maximum bending moment that we found using the model was 521 Nm, in concordance with the value obtained by Funk (458 ± 95 Nm) [25]. The simulation results demonstrated that the FE model was a biofidelic model. The model was able to fully reproduce the global femur bending behavior in terms of force-deflection history as well as local deformation near the crash region.

Comparisons of fracture impact force and bending moment between the calculated results and published literature data for tibia and fibula are shown in Table 3. The fracture impact force was drawn from the contact interface between the impactor and bones. The maximum bending moment was measured directly at the fracture location. All of the data regarding force and bending moment were within the range of previously reported data [27–29]. The validation simulation indicated the FE model of the tibia and fibula was effectively able to reproduce bending response behaviors and bone deformations.

3.3. Dynamic three-point bending test of the leg

The bending moment–deflection curve generated by the model was compared with the PHMS experimental test data (Fig. 3b). The results of the simulation corresponded with the experimental data [21,22,30]. The curve indicated that the fibula was typically loaded first and fractured first under lateral impact. The maximum bending moment of the fibula was 63 Nm and that of the tibia was

![Fig. 2. Details of the femur FE model.](image)

![Fig. 3. Three point bending validation results: (a) comparison between the FE predicted force vs. deflection curves and experimental results for dynamic three point bending of femur; (b) comparison between the FE predicted bending moment vs. deflection curves and experimental results for three point bending of lower leg at the mid shaft.](image)
Table 3

Comparison between the three point bending test data and FE simulation of tibia and fibula.

<table>
<thead>
<tr>
<th>Part</th>
<th>Test</th>
<th>Fracture force (N)</th>
<th>Bending moment (Nm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tibia</td>
<td>PMHS</td>
<td>5803 ± 1986</td>
<td>152 ± 63</td>
</tr>
<tr>
<td></td>
<td>FE simulation</td>
<td>4707</td>
<td>188</td>
</tr>
<tr>
<td>Fibula</td>
<td>PMHS</td>
<td>570 ± 280</td>
<td>–</td>
</tr>
<tr>
<td></td>
<td>FE simulation</td>
<td>602</td>
<td>–</td>
</tr>
</tbody>
</table>

176 Nm, values within the range of previously reported experimental data. The bending moment of the thigh was 260 Nm, which was slightly below the previously reported test data (352 ± 83 Nm). The peak force of thigh was 4383 N, compared to the reported data (4780 ± 792 N). The peak force of leg was 3720 N, close to the average literature test data value of 3708 N. The simulation results within the previously reported experimental values demonstrated the validity of the leg model with lateral impact.

3.4. Lateral impact of the lower limb

A comparison between PMHS test data and FE simulation results is summarized in Table 4. The peak force was 561 N for the FE lower limb model, with no statistically significant difference from the previously reported test data. The peak deflection of 28 mm was higher than the published test data. The kinetic energy loss ratio of 60% was lower than the PMHS and relaxed volunteer test results, but similar to the dummy test.

Table 4

Comparison between the PMHS data and the FE simulation results for lateral impact of lower limb.

<table>
<thead>
<tr>
<th>Test</th>
<th>Peak force (N)</th>
<th>Peak deflection (mm)</th>
<th>Energy lost ratio (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cadaver</td>
<td>596 ± 168</td>
<td>22 ± 3</td>
<td>82 ± 2</td>
</tr>
<tr>
<td>Dummy</td>
<td>515 ± 8</td>
<td>24 ± 0.1</td>
<td>60 ± 1</td>
</tr>
<tr>
<td>Relaxed volunteer</td>
<td>498 ± 52</td>
<td>21 ± 3</td>
<td>82 ± 4</td>
</tr>
<tr>
<td>FE simulation</td>
<td>561</td>
<td>28</td>
<td>60</td>
</tr>
</tbody>
</table>

Fig. 6a. Resulting maximally stressed areas were primarily located in the contact regions between sharp wheel margins and lower limb; the stress concentrations led to fracture initiation. Fig. 6b and c shows the femur, tibia, and fibula fracture processes under impact scenarios. Before fracture initiation, the maximal Von Mises stress was located on the opposite side of the impact region, and the stress distribution was exhibited in a transverse pattern. Fig. 7 reveals the average stress and strain trends in different locations under various loading scenarios. Fig. 7a demonstrates that the contact regions between the sharp margins of the wheel and thigh withstood stress concentrations, which led to bone fracture under loading conditions. Elements of the middle fracture segment underwent high stress distribution with few elements removed, and exhibited a stable stress curve. Fig. 7b demonstrates femur and tibia injury response under bumper impact. The femur and tibia exhibited similar responses during their direct impact with the bumper model. The strain of the bone was high in the contact area near the impactor, causing the bone to break rapidly. The remaining portions of the model withstood much lower strain and no injuries would have occurred.

Other simulations at the speed of 10 m/s and 40 m/s in three types of loading conditions were conducted. Bone tissues broke in a similar manner as that at the speed of 20 m/s, aside from the injury timing and severity of injuries. Under lower velocity conditions, the lower limb had more time to respond to the impact; the contrary was found under high speed impact.

4. Discussion

Pedestrians are the most vulnerable road user. In the car-pedestrian accidents, many pedestrian victims suffer significant lower extremity injuries as a result of being run-over or impacted [2]. In several previous reports skin abrasions, lacerations, and bone fractures characteristic of an accident were considered the routine tools used to distinguish injury mechanisms for accident reconstruction in forensic practices [4,31]. However, for cases in which these specific findings did not occur, or for untypical or complicated injuries with overlapping causes, sufficient details of the accident were not available. Furthermore, it may be difficult to explain the injury mechanisms to non-specialists [12]. Biomechanical studies have been conducted for many years in the investigation of lower injury mechanisms. Currently, researchers are applying biomechanics to elucidate the mechanism of a given injury, to explore body response to forces, and to determine injury tolerance limits [32].

The FE model of a lower limb did not simply present the dynamic response between a pedestrian and vehicle, but also realistically simulated local deformation in the model. The model can serve as an investigative tool to shed light on the mechanical response of the lower limb under loading conditions. Yet, to date, no FE model has been employed to assess lower limb injuries in forensic cases, not to mention the application of a FE model with the failure criteria.

Recently, several detailed lower limb FE models have been developed and employed for crash simulation. Takahashi [33] developed a lower limb FE model to explore injury mechanisms of both ligament and bone. The model was based on a commercial model and validated against test data in published literature. Untaroiu [6] used data from several PHMS tests to validate a biodfelic FE model of the lower limb based on CT scan slices. Even though the model demonstrated the capacity to simulate bone fractures in a given scenario, further improvement was warranted. Snedeker [34] produced a validated lower limb model to optimize automobile hood shape to reduce pelvis/upper leg injury risk. Kim [8] developed and validated a complex knee–thigh–hip (KTH) model to study the interactions between KTH and vehicular
interior design parameters that was used to reduce injury potential. Yasuki [35] created a lumped mass-spring lower leg model to predict the relationships among vehicle stiffness, knee-bending angle, and upper tibia acceleration. The results demonstrated that upper bumper stiffness exhibits a significant impact on the knee-bending angle and the lower bumper stiffness exhibits a significant impact on upper tibia acceleration. Teng [36] developed a highly simplified pedestrian model to assess the injuries affecting body segments that was used to redesign the pedestrian-friendly cars. These models were focused on vehicle design improvement, human safety protection, and exploration of lower limb tolerance limits, without reconstructions of the lower limb bone fractures to determine the injury manners; however, the methods could help to elucidate injury mechanisms in forensic science.

In this study, we developed a subject-specific lower limb FE model based on the CT scans of an accident victim. The model was validated against previously reported test data.

4.1. Model validation

In the model we created, the geometry of the lower limb was based on CT-scans that reflected the exact contours of parts of lower limb. The material constitutive models were filtered from previous reports that were the current, advanced models. To ensure the reliability of the newly created model, the model was tested under three-point bending and lateral impact tests. The validation tests of the lower limb exhibited a good correlation in the prediction of results with previously reported test data. In the component and overall model level, the model produced biofidelic responses under loading conditions. The element elimination approach demonstrated the ability to reproduce fracture initiation and crack propagation. We demonstrated that the bending moment of the thigh and the energy loss ratio under the lateral impact test were both underestimated in the model. Additionally, the maximum deflection under lateral impact to the leg was higher than that of experimental tests. Imperfectly applied loading conditions and simplified constitutive models are potential factors that contribute to these differences. These findings can be used for further model adjustments and improvements.

4.2. Case simulation and exploration

Our study involved a real accident and the victim exhibited complicated fracture patterns; an autopsy and CT scans were performed. The simulated wound patterns correlated with the segmental fractures of the case. In car–pedestrian accidents, the bumper directly strikes the proximal tibia and in most instances subsequently forms oblique or “butterfly” fracture patterns [37]. The femur is also sometimes broken by direct impact due to the position of the victim and the impact angle. If the victim is thrown in front of the vehicle, part of the body is usually run-over by a wheel. There are frequently hazardous injuries such as crush fractures, abrasions, and avulsion injury [4,38]. However, the skin and muscle injuries in our case were mild. This may be due to the condition of the worn-out tires, a non-heavy car weight, or the soft pavement.

In our real case simulation study, the comparison of simulation results between direct impact and run-over scenarios indicated that segmental fracture patterns generated by a wheel passing-over exhibited a good correlation with the forensic examination results. The stress trends and concentration of the model demonstrated that the femur fractures began near the sharp tire edges and the shape of crack segment was preserved, giving rise to segmental patterns.

The femur and tibia experienced a bending crack during the direct bumper impact simulation. The fracture began opposite to the area of impact and radiated back through the bone. The rest of the model experienced much lower strain and generated no fractures. The generated fracture locations and patterns under simulation scenarios indicated the wheel passing over the lower limb very likely caused the femur segmental patterns. According to the data generated by the model, the comparison potentially excluded the possibility of segmental fracture formed by direct impact. Based on the FE simulation and forensic results, we concluded that the second suspicious car had run over the victim while the impact caused by the Santana could not likely form this kind of fracture. Besides, the primary cause of death was the head injuries due to the first Santana impact. We provided the results to the police and eventually both parties accepted the conclusion and shared the compensation liability. There was a clear correlation between the forensic examination findings and the simulation results. The simulation results could be used as an indicator of cases in which further investigation of the accident is warranted. The use of FE models should be restricted until an adequate scope
**Fig. 6.** Von Mises stress distribution and injury patterns of lower limb at different time step in: (a) thigh under running over scenario; (b) upper leg under direct impact scenario; (c) lower thigh under direct impact scenario.

**Fig. 7.** Stress and strain trends under three loading scenarios: (a) running over injury by wheel; (b) injuries by bumper impacts.
of validation tests have been performed [6], and our conclusion is drawn from this particular case alone.

Although the only way to reconstruct a pedestrian accident is to analyze all the traces, concerning the pedestrian, the car and the accident site, the FE model can be a compensatory tool for the injury analysis. The developed FE model can predict the occurrence of bone fracture and the fracture site under the external force, thus we can conclude the injury manner from the initial loadings. FE simulations make the classical experiential judgments more reliable and intuitive. The first damaged FE model in forensic science shows that it can effectively produce the injury patterns and locations. That is potentially beneficial for forensic examinations.

4.3. Limitations and potential improvements

Although the FE model was well validated and the simulation results were in accordance with the autopsy findings, there are some limitations to this study [24]. Firstly, the simplification of material constitutive models and biotic tissue structures play a major role in the difference between experimental tests and FE simulations. Human bone is a non-linear, viscoelastic, anisotropic, heterogeneous, and discontinuous material [39]. In the model development, the human bone tissue was assumed to be a homogeneous and isotropic material, which was limited by the software. The trabecular bone was not included in the model since it contributes little to overall stiffness or strength [40]. The porous structure in the biotic tissue was ignored in order to generate a robust model and to obtain stable computation. Additional biofidelic constitutive models and geometry will be available for future model improvement. Secondly, the element elimination approach was used to predict fracture initiation and crack propagation. The method is mesh dependent and element elimination causes a loss of mass, which in turn affects the accuracy of finite element analysis. Therefore, a more advanced failure method that is mesh independent with no elements deleted should be developed in future studies. Thirdly, in the simulation studies, only a single lower limb FE model was employed, and this may have slightly affected the accuracy of the simulated results. Thus, developing a full human FE model is the ultimate goal of future investigations of injury mechanisms in automobile–pedestrian accidents.

In our study, only the fractures from direct force were simulated and distinguished. The fractures created by indirect force are more complex and are often formed by angulation, rotation, and compression. A full exploration of possible fracture mechanisms can help elucidate injury formation more clearly and increase the accuracy of simulated accident scenarios, thereby providing biomechanical evidence for legal representatives. In our future research, we will create a more biofidelic lower limb model, and fractures caused by indirect force will be explored in order to provide increase the amount of reliable biomechanical evidence available to forensic experts.

In conclusion, we developed a detailed lower limb FE model to explore the mechanisms of fracture caused by direct force, and tested our model with a real forensic case. The model reconstructed lower limb injury mechanisms and allowed for the distinction between possible accident scenarios by means of simulated injury patterns and stress/strain-time curves. Further improvement of the model is warranted. The lower limb model could potentially serve as a complementary tool for forensic evaluation and investigation.

Acknowledgments

The author would like to thank Dr. Liangwei Zhong (College of Mechanical Engineering: University of Shanghai for Science and Technology) for the kind instruction in finite element analysis technique. The author would also like to thank Dr. Lu Liang from Journal of Forensic Medicine Editorial Office for language editing and Jianjun Wang from the Institute of Forensic Science, China. This study was funded by Key Projects in the National Science & Technology Pillar Program during the Twelfth Five-Year Plan Period (No. 2012BAK16B02), the Council of National Science Foundation China (No. 811102300, 81273338), the Science and Technology Commission of Shanghai Municipality (No. 12DZ2271500, 10ZR1431200) and the Central Research Institute Public Project (No. GY0906, GY11105, GY2013Z-3).

References


