A Finite Element Model of the Foot and Ankle for Prediction of Injury in Under-Body Blast

Grigoris Grigoriadis, Diagarajen Carpanen, Anthony M J Bull and Spyros D Masouros*

I. INTRODUCTION

The usage of improvised explosive devices (IEDs) in modern warfare has been the leading cause of casualties [1]. In these incidents, the most prevalent region of injury of the survivors has been the lower extremity [2]. Lower limb injuries occurring to occupants of vehicles attacked by anti-vehicular (AV) IEDs have been reported to be severe, difficult to treat and, associated with high rates of amputation [3]. These injuries are mostly located in the foot and ankle, and are caused by the axial loading transmitted to the lower limb by the rapidly deforming floor of the vehicle above the explosion [3]. Platforms that replicate the physical incident and the respective loading environment offer a repeatable means for assessing injury to design new mitigation strategies. However, experiments replicating mounted blast conditions are complex, not very repeatable, expensive, and labour intensive. Validated computational models are a cost-efficient and repeatable alternative. As previous computational attempts to simulate under-body blast (UBB) [4-5] are limited, there is potential in developing and using a biofidelic finite element (FE) model of the foot and ankle to understand load transmission and design mitigation for UBB. The aim of this study was to develop a validated finite element model of the foot and ankle, which can be used as a tool to understand and predict injury in under-vehicle explosions.

II. METHODS

A subject-specific FE model of the foot and ankle (Figure 1) was developed from Magnetic Resonance Imaging (MRI) and Computed Tomography (CT) scans of a cadaveric lower limb (male, 48 y.o., 1.75 m, 93 kg) using Mimics (v15.0, Materialise HQ, Leuven, Belgium) and MSC Marc (v2013, MSC.Software, CA, USA). The response of the model was compared against experimental data obtained from static and dynamic tests performed in three different traumatic injury simulators [6-8].

The bones and cartilage of the ankle joint complex were represented by tetrahedral finite elements. The forefoot was modelled as one rigid geometry and the ligaments were represented by non-linear springs able to withstand tension only. Cortical and trabecular bone, and cartilage were assigned linearly elastic material properties while hyper-viscoelastic material properties were used for the heel fat pad [9] (Table I).

TABLE I

THE MATERIAL PROPERTIES IMPLEMENTED IN THE FE MODEL OF THE FOOT AND ANKLE FOR THE DIFFERENT SIMULATIONS. E – YOUNG'S MODULUS, ν – POISSON'S RATIO, C_{ij} – HYPERELASTIC MATERIAL CONSTANTS (YEOH MODEL), A_i, τ_i – VISCOELASTIC MATERIAL CONSTANTS (PRONY SERIES).

Structure	Static load cases	Dynamic load cases
Cortical Bone	E=14 GPa, ν= 0.3	E=17.5 GPa, ν= 0.3
Cartilage	E=10 MPa, ν= 0.3	E=200 MPa, ν= 0.3
Trabecular Bone*	E=0.5 GPa, ν= 0.4	
Heel Fat Pad*	C_{10}=0.1 MPa, C_{30}=6 MPa, A_1=0.06 for, A_2=0.8, A_3=0.02, τ_1=1 ms,τ_2=10 ms,τ_3=10 s	

**Trabecular bone and heel fat pad were assigned with the same properties for both dynamic and static loading scenarios.*

The geometry of the loading plate was modelled as a flat rigid surface and the input to the simulation was the displacement or the initial velocity of this surface, depending on the simulated load case. The proximal ends of the tibia and fibula, potted with polymethylmethylacrylate and mounted on a load-cell in all experimental setups, were 'glued' in the model to a rigid geometry fixed in space or able to move only in the direction of the loading for the static and dynamic simulations, respectively.

---

*Grigoris Grigoriadis is a PhD student at Imperial College London, Diagarajen Carpanen is a Postdoctoral Researcher at Imperial College London (+44 (0)20 7594 2646, gg2111@imperial.ac.uk), Anthony MJ Bull is Professor of Musculoskeletal Mechanics at Imperial College London, and Spyros D Masouros is a Lecturer in Trauma Biomechanics at Imperial College London

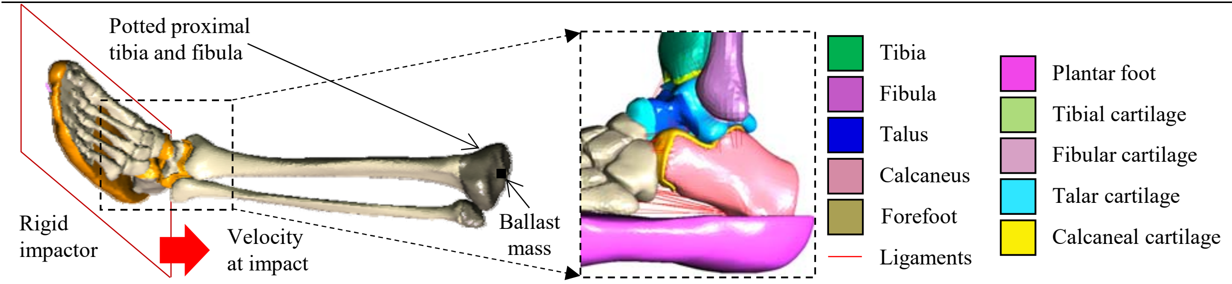

Figure 1 – The FE model of the foot and ankle able to simulate UBB. The configuration and the boundary conditions shown are for the simulation of a pendulum experiment.

III. INITIAL FINDINGS

The response of the FE model of the foot and ankle was compared against static compressive (Figure 2a), pendulum (Figure 2b), and drop tests (Figure 2c) on cadaveric limbs. The outcome verified the ability of the model to simulate accurately various axial loading scenarios.

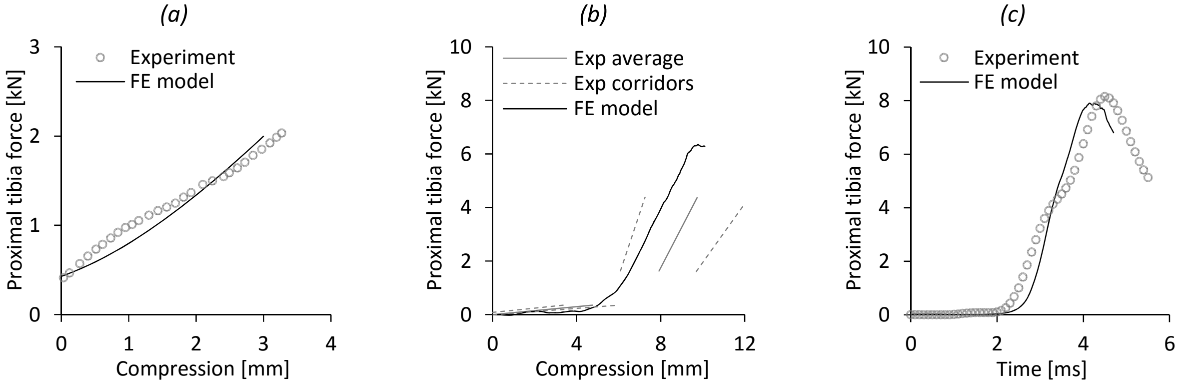

Figure 2 – (a) Comparison between the stiffness of the foot and ankle recorded in a static compression test on a cadaveric specimen [6] and predicted by the FE model. (b) The response of proximal tibia force against ankle joint compression predicted by the model for a pendulum test (5.7 kg pendulum weight and 4.5 m/s velocity at impact) is within the experimental corridors derived for pendulum strikes of various masses (3.3-12.3 kg) and velocities at impact (4-5 m/s) on cadaveric lower limbs [7]. (c) Comparison between the force-time response of the model and that of a cadaveric specimen on which a 34.2 kg mass was dropped from a height of 1.4 m [8].

IV. DISCUSSION

The response of a subject-specific FE model of the foot and ankle that was developed to simulate UBB compares well with data from experiments replicating axial loading scenarios of various severities. After examining further the validity of the numerical response by performing sensitivity analyses, the model can be used to examine the load pathway from the plantar foot to the proximal tibia and identify areas that are prone to injury in case of UBB but also test the efficacy of existing and new mitigation strategies.

V. REFERENCES

[1] Ramasamy et al, Injury, 2009.
[2] Ramasamy et al, J R Soc Interface, 2010
[3] Ramasamy et al, J Trauma Inj Infect, 2011
[4] Dong et al, J Mech Behav Biomed, 2013
[5] Gabler et al, IRCOBI, 2014
[6] Crandall et al, Stapp Car Crash J, 1996
[7] Gallenberger et al, Ann Adv Automot Med, 2013
[8] Henderson et al, IRCOBI, 2013
[9] Grigoriadis et al, IRCOBI, 2014