A finite element study to assess fracture risk in humans with low bone density

RECKNELL, Conner and SAATCHI, Reza <http://orcid.org/0000-0002-2266-0187>

Published version

RECKNELL, Conner and SAATCHI, Reza (2020). A finite element study to assess fracture risk in humans with low bone density. In: Advances in Asset Management and Condition Monitoring: COMADEM 2019. Smart Innovation, Systems and Technologies (166). Springer, 1143-1150.

A FINITE ELEMENT STUDY TO ASSESS FRACTURE RISK IN HUMANS WITH LOW BONE DENSITY

Connor Recknell[1*] (0000-0003-4703-4411) ·
Dr. Reza Saatchi[1]

Received: 03/06/2019 / Accepted: 30/06/2019

Abstract Osteoporosis is a bone-related illness which causes a reduction in bone density, where affected individuals have a higher risk of fracture.

This research uses current Finite Element Analysis (FEA) techniques such as; geometric modelling, meshing, application of materials, loading and boundary conditions, and the capture of time-dependent simulation data. To study physical properties the Human Clavicle bone. There focus is on transverse fractures in compression loading.

Previous research is detailed showing the impact of pathological fractures and its effect on the bone, this comprises of theoretical and experimental results. The study concludes on the importance of the gathered data and its uses in future applications of which encompass design, diagnostics and research.

Keywords Finite Element Analysis · Bone Mechanics · Clavicle Bone · Transverse Fracture · Fracture Risk · ANSYS · Osteoporosis · Bone Density

---

C. Recknell
Sheffield Hallam University
Tel.: +44-943-847-289
E-mail: connorrecknell@hotmail.co.uk

R. Saatchi
Sheffield Hallam University
E-mail: engrs@exchange.shu.ac.uk

1 Introduction

Osteoporosis is a bone-related illness in which the bone density decreases. This makes the bone more fragile increasing the risk of fractures. Osteoporosis alone is the cause of over 9 million fractures per year, worldwide [1].

This paper shows the investigation into a realistic model to evaluate the effects and risks of bone-related illnesses on the Human skeleton. The model is produced using ANSYS, a simulation tool that conducts Finite Element Analysis (FEA). The paper will focus on the Clavicle bone, as it is one of the most commonly fractured bones in the body. In addition, it will study the elastic and strength properties of bone defined by the background research.

2 Literature Review

Advancements in both computers and mathematical modelling have allowed FEA to be more widely used than ever before. Medical applications of FEA alone have advanced significantly in the past ten years. Its main focal points being a fundamental understanding of bone implants, assisting the design of pre-clinical testing of new implants, and comparison of pre-existing devices performance [2]. FEA is used to analyse problems of a mechanically complex structure. Where typical hand calculations are insufficient in representing the overall structure of the problem. Screening devices like FRAX (Fracture Risk Assessment Tool) can only be used for diagnostics [3]. This makes the use of FEA as an important tool that doctors and engineers could use post-diagnostic, to not only analyse a patients bones but potentially implement protection or help guide some form of clinical therapy.

The Human body is comprised of 206 bones, which account for approximately 7 percent of the bodies total mass. Cortical and Cancellous are the two main bone tissues that give the bone material properties. Cancellous honeycomb shapes are arranged in order to fit particular bones needs. Thicker beams are shown in bones that deal with high compressive stress such as the femur [4]. Reilly and Burstien showed that bone is more susceptible to tension than compression loading [5]. Several studies indicate that wet and dry bone display a different Young's modulus. The main difference being Cancellous has a lower Young's modulus to that of Compact bone [6]. Density and strength share a close linear relationship. Gibson's work indicates that Cancellous bone with a density of 1800kgm^{-3} will have an approximate strength of 205MPa [7].

3 Anatomy and Pathology

The Clavicle bone or more commonly known as the collarbone resides between the shoulder blade (Acromion) and the breastbone (Sternum). The Clavicle bone lies close to the surface of the skin.

Three ligaments connect the bone to the rest of the body. At the chest, the Sternoclavicular and at the shoulder blade, the Acromioclavicular and Coracoacromial ligaments. The clavicle, in essence, is a long tube that is mechanically designed to support the lumber of the arm.

Typical positions of injuries on the Human Clavicle happen in the middle third of the bone. This is located in the bone model as the distance between 47mm – 93mm [8].

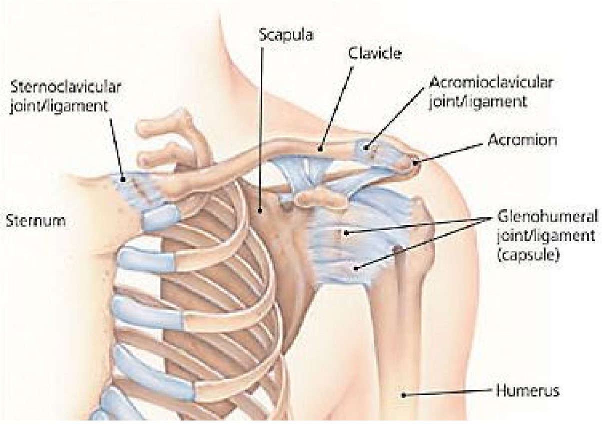

Fig. 1 Shoulder blade

Density within a patient diagnosed with Osteoporosis are shown to have a Bone Material Density (BMD) of between -1 and -2.5 standard deviations. Compared with a control group of 20-year-old population [10].

A fracture is defined as a disruption in a bones normal architecture, where the bone has undertaken disruption from an external source known as trauma. Fractures that have been acquired from bone reaching its maximum yield stress is considered a stress fracture. Pathological fractures are a result of certain medical conditions that weaken the bone prior to injury, such as Osteoporosis [11]. A transverse fracture is generated from high point load impacts, fatigue or compression of the bone. Transverse fractures occur perpendicular to the bones longest axis.

4 Methodology

The Finite Element Method (FEM) is a numerical procedure that can be applied to solve difficult geometric problems. The shape of the bone is created and its volume is known as a domain. Mesh discretisation divides its complex shape into smaller elements, similar to how individual pixels create an image. Calculations can be run across each element to simulate the problem.

Conducting an FEA study takes three phases. The initial phase is to insert the Parasolid model of the bone into the finite element software and check the geometry is correct. For the second phase is where the model is solved by the ANSYS/FE

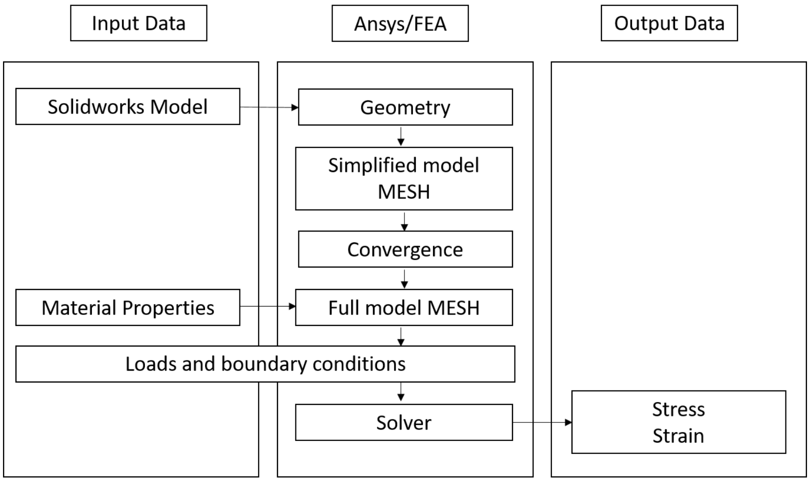

Fig. 2 FEA process

software. Lastly, the post-process is where the solved data is acquired and displayed.

The model is based on 3D medical scans supplied by Zygote, which is accurate to the 50th percentile.

4.1 Mesh

The complex shape of the Clavicle bone means that it is almost impossible to implement a hexahedral mesh, therefore, a tetrahedron mesh is used. The shape of the tetrahedron allows for the split-lines and main body to be followed at a much better shape and displays less jagged lines. The model converges at approximately 2147990 elements. An increase in the number of elements after this displays negligible results.

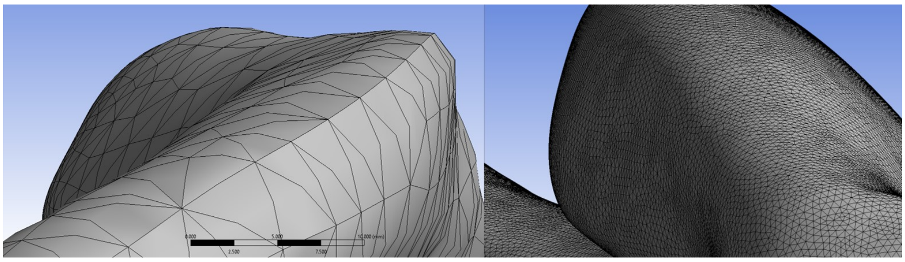

Fig. 3 Mesh discretisation, before refinement (left) and after (right)

4.2 Boundary Conditions

The Clavicle bone is approximately 140mm in length. These joints are represented as fixed points, as the structure (bone), is not designed to move away from them. Given a great enough force applied to the bone ligaments could rip or tare along with the bone fracturing. However, for the purpose of this simulation, it is assumed that applying a force that will fracture the bone will not cause the ligaments to yield or strain in anyway and will be treated as fixed points.

The Acromioclavicular and Sternoclavicular ligaments are treated as covering the 'flat' surface at each end of the Clavicle. A small area has been assumed for the Coracoclavicular ligament and has been estimated 24mm across from the Acromioclavicular and is $10x10mm^2$.

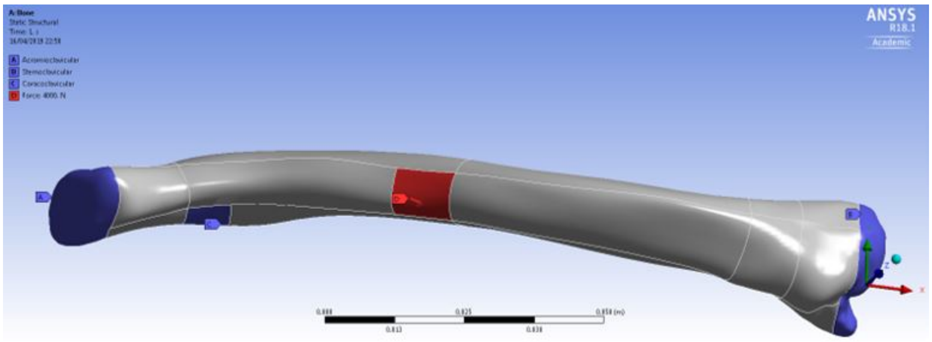

Fig. 4 Fixed points (blue) and load point (red)

4.3 Loading Conditions

Loading points for the bone cannot simply be designated as singular point, in real life the load would be spread across an entire area for. This has been modelled as a small rectangular area. The area and has dimensions of $20x10mm^2$.

Table 1 Loading conditions

	Condition 1	Condition 2	Condition 3
Distance along x (mm)	50	50	50
Force (N)	4000	4000	4000
Density (kgm^{-3})	1800	1750	1700
Strength (MPa)	205	180	155

Note: Tests were also conducted at different values along the X-Axis/Distance from Acromioclavicular at 50, 80 and 100mm.

5 Results and Discussion

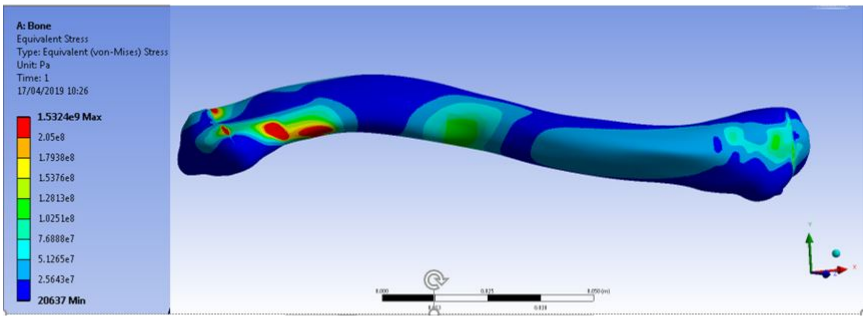

Fig. 5 Load Condition 1, density 1800kgm^{-3} strength 205MPa

Figure 5 shows the Clavicle under a force of 4kN and giving a strength of 205MPa. This is is 50mm inward of the Acromioclavicular ligament. The load point has shown to have the largest amount of elastic strain. The centre of the bone is deflected from its original point by 3.765mm. The highest stresses at this density are located towards the ends of the bone and the middle section.

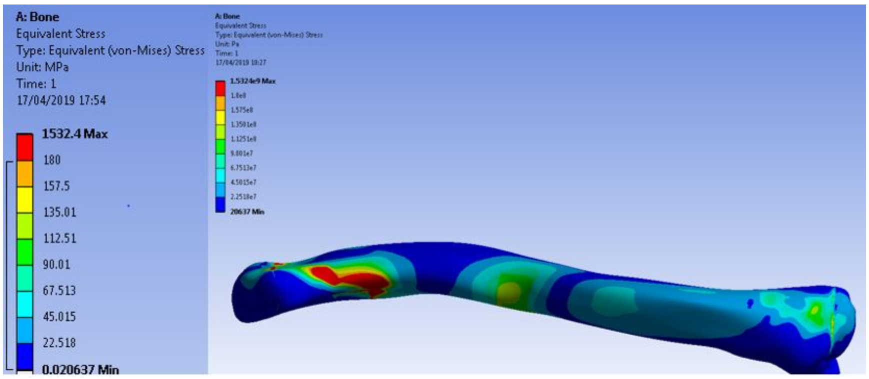

Fig. 6 Load Condition 2, density 1750kgm^{-3} strength 180MPa

Within all simulations of the clavicle bone, high levels of stress are located towards the centre of the model (the middle third). All deflection data sits within the majority of this region.

High levels of stress are displayed around the connection of the joints/fixed point in 7. Furthermore, other figures have demonstrated this characteristic. This is both mechanically correct but in-coherent as forces would build up around the joints due to moments. However, the way in which the model displaces the stress is not true to life nor can be assumed to be accurate. This particular area of high stress is due to the complexity of the shape and the bones cross-sectional area.

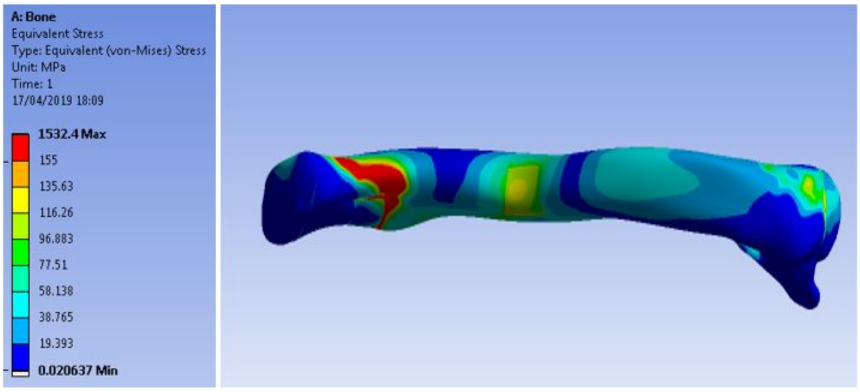

Fig. 7 Load Condition 3, density 1600kgm^{-3} strength 155MPa

Reducing the density generates larger regions of stress around the centre and edges of the bone. Although the bone still has the same yield strength (point of maximum stress energy) the point in which the bone reaches maximum plastic stress (strength) is lowered. This lower value means that a force of 4kN, when applied to the healthy bone, may not fracture fully or not fracture at all. Compared with that of a lower density indicating that the bone is at a lower peak stress and will fracture. Full certainly cannot be given when assessing if a bone will break every time given a load or location, as Humans will still vary in bone shape, size and density. Scanning techniques could indicate the likelihood of fracture more accurately for an individual.

6 Conclusion

The project has yielded some justified results for the relationship in bone density and fracture risk. There is a correlation between density, strength and an increase in fracture risk with lower density bones. This should supply some confidence with the association of the materials. The study has also highlighted the key characteristics of the clavicle bone.

The FEA models have shown valuable results and set the stage for further testing to be conducted. Given the several types of pathologies related to fracture risk, such as Osteoporosis. The use of FEA could help design not only protective equipment for the shoulder, but the whole skeletal system. Reducing expense on the health service and increasing the survivability of patients with bone related illness.

7 Future work

Cancellous and Cortical bone could be modelled individually with their own sets of material properties to increase accuracy. The materials within the Cancellous bone could be simulated using mixing theory to allow for more data to be represented within the model. The ligaments and surrounding muscle, bone and tissues could be applied to generate more realistic fracture data. Statistics from vehicles accidents could be used in setting up representative test scenarios for the bones to be compared to. Finally, further testing of the bone materials themselves should be conducted as research into the mechanics of bones is still ongoing.

Acknowledgements I would like to thank Dr Green and Dr Madan at Sheffield's Children's hospital for taking the time to help with the project, their incite and knowledge proved incredibly valuable. Dr John D. Currey who's work in the area proved not only hugely valuable to the project but is an inspiration in the field of bone mechanics. A special thanks to Kingsley Hanlon and Samin Hasnat for their guidance. Finally, my parents, for all their support.

Conflict of interest

The authors declare that they have no conflict of interest.

References

1. Johnell O., Kanis J.: An estimate of the worldwide prevalence and disability associated with osteoporotic fractures, Osteoporosis Int, vol 17, pp. 1726 - 1733 (2006). doi:10.1007/s00198-006-0172-4
2. Taylor, M., Prendergast, P.J.: Four decades of finite element analysis of orthopaedic devices: Where are we now and what are the opportunities, Journal of Biomechanics, vol 48, pp. 767-778. (2014). doi:10.1016/j.jbiomech.2014.12.019
3. Imai, K.: Computed tomography-based finite element analysis to assess fracture risk and Osteoporosis treatment, World Journal of Experimental Medicine, pp. 182-187. (2015). doi:10.5493/wjem.v5.i3.182
4. Solomon, L., Warwick, D., Nayagam, S.: Apley and Solomon's Concise System of Orthopeadics and Trauma. CRC Press, New York (2014)
5. Reilly, D.T., Burstien, A.H.: The elastic and ultimate properties of Compact bone tissue, Journal of Biomechanics, pp. 393-405. (1975). doi:10.1016/0021-9290(75)90075-5
6. Currey,J.D.: Bones, Princeton University Press, Princeton (2002)
7. Gibson, L.J.: The mechanical behaviour of Cancellous bone, Journal of Biomechanics, vol 18, pp. 317-328. (1985). doi:10.1016/0021-9290(85)90287-8
8. Bangash, H.M., Al-Obaid, F.Y., Bangash, N.F., Bangash, T.: Trauma: An Engineering Analysis, pp. 91. Springer, Berlin (1982)
9. Krum, S.A., Brown, M.: Unraveling Estrogen action in Osteoporosis, Cell Cycle, pp. 1348-1352. (2008). doi:10.4161/cc.7.10.5892
10. Kanis, J.A., McClosky, E.V., Harveny, N.C., Johansson, H., Leslie, W.D.: Intervention Thresholds and the Diagnosis of Osteoporosis, Journal of Bone Material Research, vol 30, pp. 1747-1753. (2015). doi:10.1002/jbmr.2531
11. Townsend, C., Beauchamp, D.R., Evers, B.M., Mattox, K.L.: Sabiston textbook of surgery (18th ed). Saunders, Online (2007)