Original Article

Imaging and Finite Element Analysis Application on Biomechanical Evaluation of Bone Fracture Case

Maya Genisa[1], Solehuddin bin Shuib[2], Zainul Ahmad Rajion[3], Amran Ahmed Shokri[4], Aryenti[5]

[1] YARSI University.

[2] Universiti Teknologi MARA.

[3] International Islamic University Malaysia.

[4] Universiti Sains Malaysia.

[5] YARSI University.

Email: maya.genisa@yarsi.ac.id[1], solehuddin2455@uitm.edu.my[2], zainulrajion@iium.edu.my[3], amrankb @usm.my[4], aryenti@yarsi.ac.id[5]

DOI: 10.47750/pnr.2022.13.S01.200

Abstract

Bone fracture injury is a common problem as of various accidents or other causes. The presence of a bone fracture can easily be identified by imaging such as using a CT scan. However, the dynamic behavior of bones experiencing fractures, especially during the healing period, is rarely well known. The bone itself has the ability to regenerate itself in the context of a healing fracture. However, if the growth of the broken bone is not in the right position, it will result in inhibition of healing or the formation of imperfect bones. Various factors can cause bone disposition caused by loading factors. This paper aims to discuss how the dynamic behavior of fractures is evaluated using imaging techniques and finite element analysis. The biomechanical system evaluation is very important to be able to simulate the condition of the bones due to static loading. Several scenarios of static loading type are performed to simulate the loading propagation and its effect on the spread of stress in the fracture area. Modeling was carried out on a sample of patients who had fractures in the ischium bone due to traffic accidents. The 3-D image data from the patient were taken using a CT scan, which was then processed for the segmentation to separate the study target area from the rest of the body. The segmentation process is carried out to separate the soft tissue and leave only the bone for the simulation process. Other pre-processing such as smoothing and meshing is done to ensure that the FEA can be applied to the object. Simulations on objects are carried out which include simulations of various types of load forces and rotational forces or torque. The simulation result is a stress distribution that covers the area around the fracture and several other parts of the pelvis. From the simulation results, it can be shown that there has been a disposition and accumulation of stress around the fracture area, this can cause an influence on the position of the fracture bone junction and new bone growth in the area. However, to determine how much influence stress has on the mechanism of bone growth, further research is still needed.

Keywords: fracture, stress, CT scan, bone, Finite element analysis.

INTRODUCTION

Bones are the main part of the body that supports the body. The bone formation consists of the main parts, namely cells and bone matrix. The bone matrix itself consists of inorganic components which include about 69% consisting of hydroxyapatite and 22% of organic components consisting of collagen as the main constituent (Kini & Nandeesh, 2012). From a physiological point of view, bone consists of supporting cells which are osteoblasts and osteocytes, and remodeling cells which are osteoclasts. These components are responsible for the dynamic processes that comprise the modeling and remodeling processes in bone.

Bone is a unique organ in the body because of its ability to regenerate itself completely. Therefore, in cases of bone fracture, the healing condition can be carried out independently over time just by keeping the bones in their original position. If the position of the fractured bone is not in the correct position, the healing process might be late and the shape of the bone is not properly built. For example, in the case of a patient who has a fracture in the hip, if the position of the bone during the healing stage is not correct it may result in a curved shape of the bone. So that the size between the left and right legs is not the same, which can cause the patient to limp.

Bone fractures are common at any time and to anyone, whether accidentally or patients who have a high risk, such as people with osteoporosis. Patients with osteoporosis where bone density becomes reduced and brittle(Qaseem, Forciea, McLean, & Denberg, 2017; Garach, Fontana & Torres,2020). Imaging using X-ray energy is often used to identify fractures ranging from low-dose to high-dose. The cortical bone was identified as translucent white on the CT-scan image. Meanwhile, bone fractures are identified as black lines with a shape that follows the fracture(Doblaré, García, & Gómez, 2004). If the fracture is very strong it will be easy to identify, but if the fracture is smooth and the image quality is poor, the fracture will be difficult to be identified.

Shifting in the position of the bone from what should have been possible during healing is the result of overloading the fractured bone. Therefore, evaluation of the biomechanical system is very important to be able to simulate the condition of the bones due to overloading. Through this biomechanical evaluation, the types of load forces, and their effects on the distribution of stress distribution and changes in bone position due to loading can be simulated. So that it can be advised to patients to avoid burdens that can interfere with the bone healing process.

The mechanism of loading and its effect on bone healing is very important, especially in the rehabilitation period and it is a challenge to be able to evaluate the behavior of the bone system in the area around the fracture, including evaluation of the distribution of stress and its displacement are still not clearly explained clinically. Therefore, this mechanism will be tried to solve through a simulation study using the Finite Element Analysis (FEA) method.

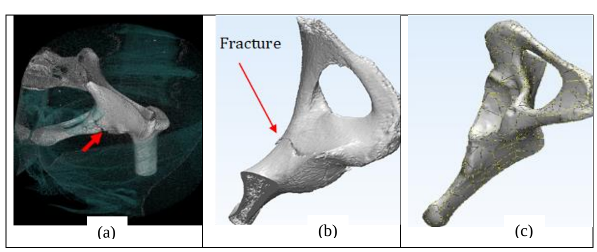
Finite Element Analysis (FEA) is a numerical method that can break down complex object problems into small elements (Geng et al., 2008; C. L. Lin, Kuo, & Lin, 2005; Genisa et.al.,2017). This method has been widely accepted to solve biomechanical problems related to human body systems, such as to determine the distribution of stress due to loading(Lin, Li, Li, Ichim, & Swain, 2007; Papavasiliou, Kamposiora, Baynet, & Felton, 1997;Izmin et.al.,2020), which until now there is no single clinical tool that can measure the spread of this stress.

This study intends to integrate the simulation methods and object imaging systems using imaging techniques such as CT scans and numerical simulations to study the behavior of stress spreading and bone displacement caused by certain loads.

MATERIAL AND METHOD

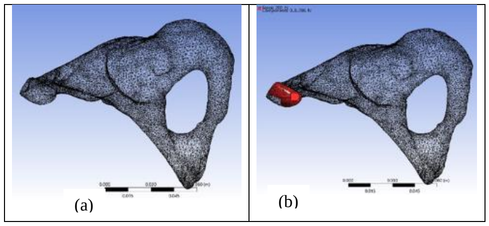
A three-dimensional(3-D) finite element model of pelvic has been created from Computed Tomography (CT) data of bone fracture of the patient. For this case, the targeted area bone fracture is located at ischium bone which was segmented from the whole hip on the CT data. The illustration of the segmentation process is shown in Fig 1. In the first stage of segmentation, the hip was separated from soft tissue based on a threshold of HU on CT data. The Region of Interest (ROI) is cropped to get the target of the only element. The ROI is segmented to get only targeted bone within the fracture area using Boolean operation. In the final stage, wrapping and smoothing are performed to get the smooth surface and closed volume of the object.

Fig 2. Process of segmentation from CT data to construct 3-D objects. a) bone separated from soft tissue, b) selected bone with a fracture located in the ischium, and c) smoothed and wrapped object.



After all pre-processing stages are completed, the model is transferred to build the meshing for FEA preparation. The number of mesh was controlled by the quality of elements of the mesh including the angle and size of the mesh element. Meshing is refined to get more detail, especially in the edge and fracture area. The meshing of the model which is resulted from this stage is shown in Fig 2.

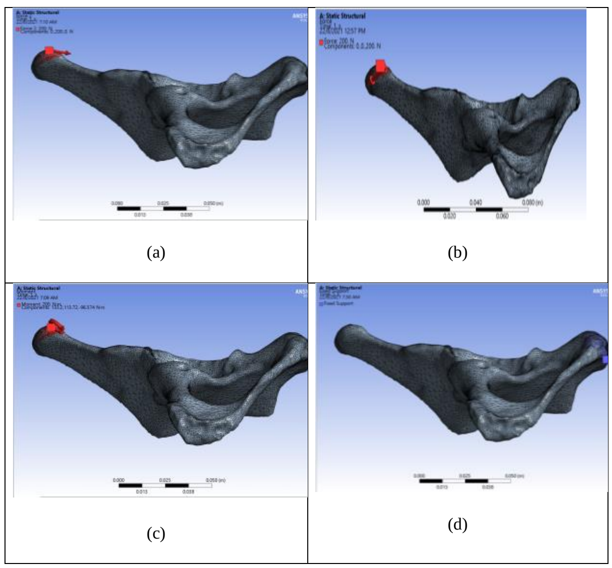
Fig 2. a) volume meshing of the object, and b) loading point position.



Preparation of material assignment is quite simple because the object only consists of one component only with the homogenous properties of bone are applied(Chirchir, 2016). Those material assignments are including the properties of bone density (2.17 g/cc), Poisson's ratio (0.3), and Young's modulus (13.7 GPa). The behavior of fractured bone is assumed has a contact in between with friction is allowed. There are three different simulations (Normal force, Shear force, and rotational force or torque) were performed. The stress distribution due to each different loading simulation on implant has been analyzed using ANSYS software.

Three different types of loading have been tested: Normal force, shear force, and torque. All values of the loading are set to 200 N for the force and 200 N.m for torque simulations. Fig 3. Shows a position of loading and fixed support.

Fig 3. location of loading and fixed support: a) Normal Force, b) Shear Force, c) Torque, and d) Fixed support.



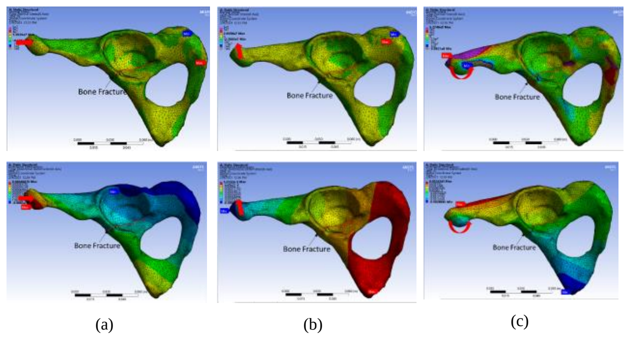
RESULTS AND DISCUSSION

The ischium is the bone below the ilium and next to the pubis. This bone is thick because it is formed from two bones that are fused and circular. The ischial bone plays a role in the delivery process when the fetal head begins to move through the birth canal.

This study is to simulate the case of ischium fracture how to respond to the different loading due to the force either coming during delivery or from external activities.

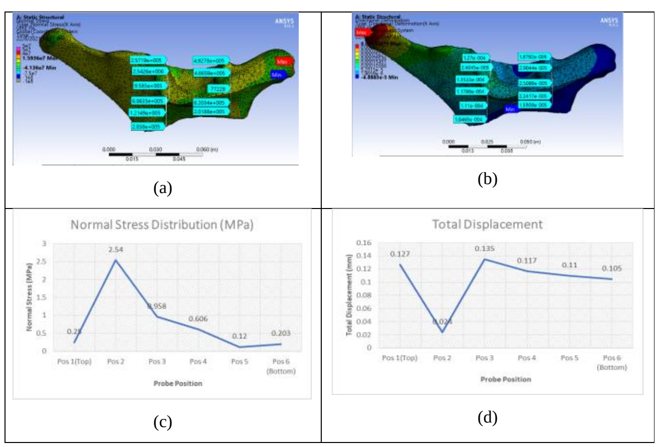
There are three different types of loading that have been tested: Normal force, shear force, and torque which possibly unintentionally convey to the bone fracture location. All values of the loading are set to 200 N for the force and 200 N.m for torque. FEA simulations are performed to evaluate biomechanically the distribution of stress and displacement around the fracture location. Figure 3 shows the distribution of normal stress and deformation or total displacement for the whole pelvic bone of different loading. The patterns of normal stress distribution due to both normal and shear/tangential forces are almost similar. However, the stress generated by torque loading is quite significantly different. The distribution of normal stress due to normal and tangential force is most likely distributed to the whole body of the pelvic bone. Meanwhile the normal stress distribution due to torque most concentrated in the area around the location of loading, and less distributed on the fracture location (red line). The position of fractures is protected by acetabulum from suffering by loading. meanwhile, the maximum deformation is also located quite far from the fracture location which is mostly concentrated around the loading area (Figure 3c). High deformations occurred significantly due to tangential force at the location of fractured bone and another side of the pelvic (Figure 3b). this loading force is most suffering the fractured bone by generating the high deformation around the fracture.

Fig 3. Stress (upper) and deformation (lower) distribution of: (a) 200 N Normal forces, (b) 200 N shear forces, and (c) 200 N.m Torque.



To understand the stress distribution and deformation vertically due to loading, a slicing view is showed in Figure 4. The slicings are taken along the fracture line, both normal stress and deformations are quantified using a probe placed from top to bottom of ischium bone. The distribution vertically of normal stress and displacement are plotted in the graph for each loading simulation (Figure 4-6). The probe positions are placed from top to bottom position which measures both normal stress distribution and deformation/displacement. Vertically the stress distribution due to normal force is most likely distributed in the surface of ischium bone, less penetrated inside the bone. The reduction of stress distribution in the vertical direction from top to bottom of ischium bone is followed by a reduction of displacement/deformation. The displacement in the surface of the top part of ischium bone has a high value than in the bottom part. This possibly impacts the healing process of bone union, in the upper part of fracture is possible will late to recover compared with the bottom part due to this high deformation in this area.

Fig 4. Slicing of Stress distribution (a & c), and total displacement (b &d) around fracture due to 200 N Normal force.



The effect of shear force on the area surrounding bone fracture vertically can be seen in Figure 5. The normal stress is distributed gradually decrease from the upper part to the bottom part of the ischium bone. The maximum stress is located in the top part and the minimum normal stress is located in the bottom part. This decreasing vertically of normal stress distribution is followed by the increasing negatively the trend of displacement/deformation measured on the same probe measurement. The minimum displacement occurred just about 2-4 mm from the top of the ischium bone. It indicated that the bone fracture in the upper part of the ischium bone will have less displacement compare to the bottom part. In this case, the shear force will impact the healing fracture especially in the lower part of the ischium bone. Fortunately, the total displacement due to shear force is still lower impact compared with normal force. The impact of shear force on the fracture in terms of displacements is about 700-800 times lower than due to normal force.

Fig 5. Slicing of Stress distribution (a & c), and total displacement (b &d) around fracture due to 200 N Shear Force.

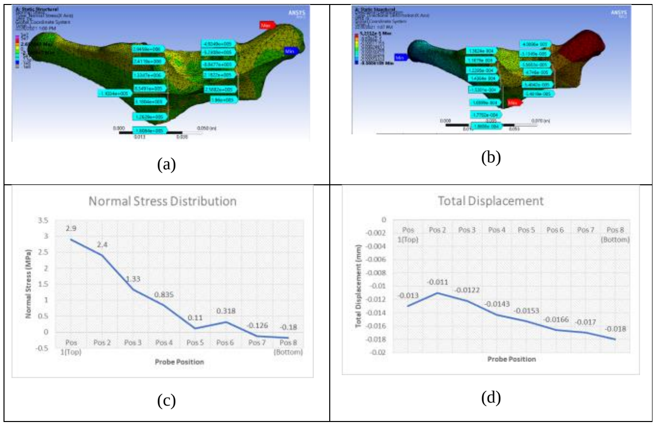
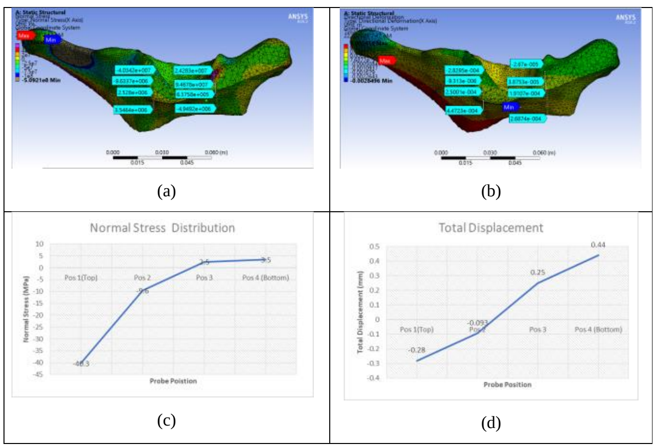


Figure 6. Slicing of Stress distribution (a & c), and total displacement (b &d) around fracture due to 200 N.m Torque.



Meanwhile, the impact of torque on the ischium bone fracture gives a more significant impact. The increasing of normal stress distribution from the upper to lower part of ischium was followed by increasing of displacement on the same probe location. On the lower part of the ischium, the maximum displacement has occurred. The value of displacement on this location is about 300-400 times of normal force and 2000-2500 times of shear force. The upper part of a fractured bone in this location will get more difficult to be recovered due to the instability of the bone position due to this loading type.

Ischium bone can be fractured due to accidents like traffic accidents with the possible minimum force required to cause the ischium fractures is 10.28 kN at the impulse of 154.2 N/s (impact speed 44.6 km/h)(Arkusz, Klekiel, Niezgoda, & Będziński, 2018). The simulations above show the impact of force with the force is 200 N which is expected not to cause the ischium fracture, just only to simulate the possibility of force generated by some light activity during healing of the existing bone fracture in this region. The light activity can be related to daily activity like sport, carriage, or some shock of shift belt during riding the car. However, this simulation was assumed the bone density of the patient was normal, it would have a different result if the patient with osteoporosis or osteopenia where a density of bone becomes lower than the normal one.

Even the fractures of the ischium are uncommon and can be ranged widely from mild to severe(major broken). The healing process can also range from a few weeks to months for mild cases with or without surgery, and can be life-threatening and may involve other organs damages which can be followed by some symptoms pain in the groin, hip, or lower back, other symptoms like abdominal pain, bleeding and difficulty in walking or standing. This study of stress simulation using FEA, in this case, able to be used as an early warning or basic information that can be used for further medical treatment like a fixation. Together with the fixation model, the FEA study also can be extended to investigate the biomechanical behavior post-fixation.

CONCLUSION

The stress distribution and deformation in ischium bone fracture are investigated using FEA under three different types of loading; 200 N of Normal force, 200 N of shear force, and 200 N of torque. The region of interest has been prepared from the patient who has bone fractured due to a traffic accident.

The simulation result showed a significant impact of torque on the instability of existing bone fracture compared to normal force and shear force. The sheer force producing the smallest deformation compared to the other two types of simulation. The generated stress is mostly distributed on the surface compared to the body of ischium bone where the lower part of ischium bone received a lower portion of stress. This instability position on the region of fracture will possibly impact the healing process if the stress and deformation persistently exist.

Further study on ischium fracture cases can be extended with more scenarios by taking more variables into account to simulate different situations including post-fixation and different types of bone diseases like osteoporosis or osteopenia. the result can be used as an early warning or additional information for further medical treatment of avoiding the unwanted risk to minimize the damage.

ACKNOWLEDGMENT

We would like to thank Universiti Sains Malaysia (USM) and Universiti Teknologi Mara for supporting this research.

REFERENCES

1. Arkusz, K., Klekiel, T., Niezgoda, T. M., & Będziński, R. (2018). The influence of osteoporotic bone structures of the pelvic-hip complex on stress distribution under impact load. Acta of Bioengineering and Biomechanics, 20(1), 29–38. https://doi.org/10.5277/ABB-00882-2017-02
2. Chirchir, H. (2016). Limited Trabecular Bone Density Heterogeneity in the Human Skeleton. Anatomy Research International, 2016, 1–7. https://doi.org/10.1155/2016/9295383
3. Doblaré, M., García, J. M., & Gómez, M. J. (2004). Modelling bone tissue fracture and healing: A review. Engineering Fracture Mechanics, 71(13–14), 1809–1840. https://doi.org/10.1016/j.engfracmech.2003.08.003
4. Garach A.M., Fontana B.G., & Torres M.M., “Nutrients and Dietary Patterns Related to Osteoporosis”, Nutrients, Jul 3;12(7), 2020.
5. Geng, J., Yan, W., Xu, W., Tan, K. B. C., Lee, S., Xu, H., … Chen, J. (2008). Finite Element Modelling in Implant Dentistry. Advanced Topics in Science and Technology in China.
6. Genisa M., Rajion Z.A., Shuib S., Dasmawati M., Pohchi A.,”Evaluation Of Stress Distribution And Micromotion Of Dental Implant: In Vivo Case Study”, ARPN Journal of Engineering and Applied Sciences, VOL. 12, NO. 14, JULY 2017.
7. Izmin,N.A Nor, Hazwani F., Todo M., and Abdullah A.A., Risk of Bone Fracture in Resurfacing Hip Arthroplasty at Varus and Valgus Implant Placements,” International Journal of Technology, vol. 11, No. 5, pp. 1025-1035, 2020.
8. Lin, C. L., Kuo, Y. C., & Lin, T. S. (2005). Effects of dental implant length and bone quality on biomechanical responses in bone around implants: A 3-D non-linear finite element analysis. Biomedical Engineering - Applications, Basis and Communications, 17(1), 44–49. https://doi.org/10.4015/S1016237205000081
9. Lin, D., Li, Q., Li, W., Ichim, I., & Swain, M. (2007). Biomechanical Evaluation of the Effect of Bone Remodeling on Dental Implantation using Finite Element Analysis. 5th Austalasian Congress on Applied Mechanics, ACAM 2007, (December).
10. Papavasiliou, G., Kamposiora, P., Baynet, S. C., & Felton, D. A. (1997). 3D-FEA of osseointegration percentages and patterns on implant-bone interfacial stresses. Journal of Dentistry, 25(6), 485–491.
Qaseem, A., Forciea, M. A., McLean, R. M., & Denberg, T. D. (2017). Treatment of low bone density or osteoporosis to prevent fractures in men and women: A clinical practice guideline update from the American college of physicians. Annals of Internal Medicine, 166(11), 818–839. https://doi.org/10.7326/M15-136.