Mesh refinement for cortical and trabecular bone finite element modeling: A review

N A M Zain1*, R Daud2, N S Kamarrudin2, K R Ahmad3 and N H A Aziz1

1Institute of Mathematics Engineering, Universiti Malaysia Perlis (UniMAP), Pauh Putra Campus, 02600, Arau Perlis, Malaysia.
2Fracture and Damage Mechanic Research Group, School of Mechatronic Engineering, Universiti Malaysia Perlis (UniMAP), Pauh Putra Campus, 02600, Arau Perlis, Malaysia.
3School of Material Engineering, Universiti Malaysia Perlis (UniMAP), Pauh Putra Campus, 02600, Arau Perlis, Malaysia.

*Corresponding email: aliazain@unimap.edu.my

Abstract. For centuries, the finite element (FE) method has been extensively used to predict the fracture performance and various methods have been implemented to yield accurate results especially in healthcare industries. Bone fracture has been a critical problem since it interrupts the strength and structure of human bone. Thus, this problem will lead to bone malfunction and cause excess bleeding of surround tissues. Human bone consists of cortical and trabecular bone which serve a different amount of load sustainability before the fracture occurred. One of the most vital problems arise is the inaccuracy of the stress intensity factor related to the bone fracture. Recent studies have proven that with the implementation of appropriate meshing element produce higher accuracy results especially with the implementation of mesh refinement in the finite element model. The singularity elements suggested by Barsoum (1976) has proven that the quarter-point triangular elements give highly accurate results. Several methods for stress intensity factor calculation has been implemented by various past researchers. Among all the methods used, J-integral has proven to be the most accurate method compared to the others. The first section in your paper.

1. Introduction

In recent years, there has been an increasing amount of literature on the finite element (FE) that have been widely used to estimate the behavior of fracture mechanics. The first serious discussions and analyses of FE emerged during the 1970s era where most researchers used the direct calculation of combined mode I and Mode II to calculate stress intensity factor (SIF) axisymmetric and planar structures of arbitrary geometry and loading [1]. Today, an increasingly popular way to solve the fracture mechanics problem is using a finite element model (FEM). In linear elastic fracture mechanic (LEFM), the main obstacle is to find the value of SIF at the crack tip singular stress. The development of singularity elements finite element method has shown that it is the best and efficient way to determine SIF [2] which provide an asymptotic solution in stress and displacement field at the crack tip.

In the past literature, FE methods widely used in fracture mechanic problems produced poor results in the FE solutions when they used the conventional elements because to the fact that conventional FE employs polynomials to interpolate field variables in the domain. Thus, these are not able to produce the singular crack tip fields [3] Then, a significant development of FE analysis of crack problems was introduced by “Henshell and Shaw” and Barsoum where it is independently showed the singularity at the crack tip is possible to be modeled by placing the mid-side node near the crack tip at the quarter-point position of the element. The linear-elastic singularity for stresses and strains is obtained by shifting a quarter to the crack tip midside nodes of all the surrounding elements. This is supported by recent literature done by Li [4] pointed out that with the implementation of the quarter-point element at the
crack tip increases the accuracy of SIF and displacements. Meshing scheme elements play an important role where the accuracy of the six values highly depend on the meshing elements. Different authors have measured the SIF in various ways. Recently, Liao [5] studied that with the adoption of mesh refinement, the accuracy of computed SIF increased. The size of the element effects the results where the smaller elements produced higher accuracy of the results. The stress-intensity concept is based on the parameter $K$, which quantifies the stresses at a crack tip. According to LEFM, the growth rate of a fatigue crack is determined by SIF by Paris Law [6]. Thus, the understanding of crack behaviour requires the knowledge of SIF expressed as the function of load and crack geometries.

There are various common methods in finding SIF value including displacement extrapolation method, stress extrapolation method [7,8], J-Integral [9], and node displacement method. A recent study conducted by [7] proven that the J-Integral method gives the highest accuracy compared to others. The most interesting finding is that the accurate results were obtained by the implementation of different meshing element types and sizes stated in [10] in previous studies. The finding is in agreement with Moteza (2015) findings which showed that meshing elements contribute the most accurate value of SIF and has less error compared to others [11]. Thus, this paper aims to serve as a base for future studies in the importance of mesh element in finding the most accurate result of SIF.

1.1. Bone modeling
Bone has a complex arrangement of structures at multiple scales which contributes to its ability to function as mechanical support to human body. Human bone consists of cortical and trabecular bone which each has serve a different functionality due to the micro structure of the bone. Cortical bone is a compact bone surround the outer layer of the bone which is vital in providing the strength and stiffness to the bone. Trabecular bone has less bone matrix appear to have porous structure in the inner space of the bone providing the functionality of weight bearing which help the human bone to resist bending and twisting. The main difference of compact and trabecular bone is, the compact bone consists of osteons while the trabecular bone is made up of trabeculae. In developing a complex geometry of biological human structure, it is necessary to have the accurate geometrical modeling as it would give a great impact to the computational results analysis. Thus, in designing the FEA model there are three important steps which includes, the material mechanical properties of the bone, the boundary condition and lastly the geometry of the developed bone.

1.2. Bone model meshing (2D)
For the past centuries, FEM has been widely used for medical imaging application and remarkably produced high-resolution images and allowed computational research being implemented to the interested bone model. However, only a few results are available in literature on applying the meshing techniques in bone modeling. The accuracy of FEM simulation depends on the quality input surface and volume of the mesh (sizes and shape of the meshing element). A good meshing technique would produce accurate simulation results and reduces the reliability in the FEM simulation. In bone analysis study, a little attention is paid on the meshing techniques implemented in the simulation study. The previous research done by Han et al. [7] has proven that a well-shaped meshing element produce the most accurate result compared to the others.

1.3. Global and local meshing
In finite element formulation, there are two coordinate system that are usually used which include global coordinate system and local coordinate system. The global coordinate system represents the entire designated body used to define the nodes of the entire body while local coordinate system resembles a particular element in the body, and the numbering is done to that particular element neglecting the entire body. The main idea of mesh refinement is dividing the element into half and as the result of the refinement, it doubles the line of the elements quadruple the number of areas in the elements and increase the number of volume elements by a factor of eight. The global refinement of the entire mesh are often unnecessary because the mesh is only needed to be refined with higher gradient at a particular area of
interest so that the mesh refinement can achieved as more effective tools as global mesh refinement [12]. In engineering world problems, the local meshing refinement are proved to be a better solution in solving fluid dynamic problems done by recent researcher which used quadrilaterals or triangles in two dimensions meshing geometry [13]. Table 1 presents the overview of meshing techniques used by previous researchers.

**Table 1. The overview of meshing techniques by past literatures.**

| Source | Meshing method used | Significant |
|--------|---------------------|-------------|
| [14]   | Extended finite element method and automatic adaptive mesh refinement | These two meshing methods prove to be highly efficient in the discontinuities for a variety of fracture analysis. |
| [15]   | Used moving mesh strategy | The meshing method used to predict crack growth on the basis of fracture mechanics variables by using the modified moving computational from a fixed referential coordinates system. |
| [16]   | Local mesh free numerical model (ILMF) | Used the ILMF technique to solve 2D in linear elastic fracture mechanic to find stress intensity factors (SIF). |
| [17]   | Varying the meshing size (mesh refinement) | The accuracy of crack propagation prediction becomes weak as the meshing size increase. |
| [18]   | Mesh superposition method | The meshing method has been verified that it can be used in the dynamic crack propagation. |
| [19]   | Local adaptive mesh rezoning for five-node crack-tip elements | The method used proved to minimize the energy error during successive remeshing operations |
| [20]   | Poisson equation and adaptive mesh refinement. | Both 2D and 3D crack propagation has successfully validated by using the method proposed. |
| [21]   | Mesh refinement and mesh convergence analysis | Mesh refinement method was done until the changes in node displacements and maximum von Mises stresses in two consecutive models were less than 5%. |
| [22]   | Mesh refinement | The 2D model developed with mesh refinement of four-noded and eight-noded quadrilateral elements and proves that mesh refinement plays an important role. |
| [23]   | Mesh density | Mesh was refined in the region of the prosthesis where stress concentration was expected, and the value of SIF was predicted. |
| [24]   | Mesh orientation sensitivity | Three different mesh structured was developed and the model which is more sensitive to the mesh orientation is discovered. |

In the past literature, FE methods widely used in fracture mechanic problems produced poor results in the FE solutions when they used the conventional elements because to the fact that conventional FE employ polynomials to interpolate field variables in the domain. Thus, these are not able to produce the singular crack tip fields [11]. Then, a significant development of FE analysis of crack problems was introduced by Henshell and Shaw (1975) and Barsoum (1976) where it is independently showed the singularity at the crack tip is possible to be modelled by placing the mid-side node near the crack tip at
the quarter-point position of the element. The linear-elastic singularity for stresses and strains is obtained by shifting a quarter to the crack tip the midside nodes of all the surrounding elements. A considerable amount of literature has been published on singularity elements used in finite element to compute the SIF results. Morteza et al. [11] points out that in determining the SIF values, the adaptation of correct singularity elements is far more accurate which yield a very low percentage of error.

1.4. Fracture parameter

In the early of 2000, the interest of fracture parameters has emerged as stated by previous researcher, Shi [25] stated that there are two parameters usually used to represents fracture behaviour which is stress intensity factor ($K$) and energy release rate ($G$). The stress-intensity concept is based on the parameter $K$, which quantifies the stresses at a crack tip. The stress-intensity concept is based on the parameter, which quantifies the stresses at a crack tip. According to LEFM, the growth rate of a fatigue crack is determined by SIF by Paris Law [6]. Thus, the understanding of crack behaviour requires the knowledge of SIF expressed as the function of load and crack geometries. There are various of common methods in finding SIF value including displacement extrapolation method (DEM), stress extrapolation method [7,8], J-Integral [9], and node displacement method. Another parameter that must be taken is the energy release rate, $G$ which indicates the decreasing of potential energy of each unit area of the crack. The relationship between these parameters and the SIF value is given by,

$$K = J = \frac{K^2_t}{E'} + \frac{K^2_u}{E'} + \frac{K^2_m}{2\mu}$$  \hspace{1cm} (1)

where $E$ is the Young’s modulus of the material which $E'=E$, for plane stress while This method used for SIF calculation was proposed by Rice [26] in LEFM given that,

$$J = \int_{\Gamma} \left[ W d\gamma - T \frac{\delta u}{\delta x} ds \right]$$  \hspace{1cm} (2)

where $W$ is the strain energy density, $T$ is the traction vector on a plane defined by the outward normal, $u$ is the displacement vector, $\Gamma$ is an arbitrary contour surrounding the crack tip with an initial points lying on the two crack surfaces as shown in Fig. 1 in the previous part and $ds$ is the element of arc along the contour $\Gamma$. $J$ – Integral is conservative and path independent where different integral paths effect in the same value. Rice [26] claims that the $J$ – Integral path is independent. The value of SIF can be calculated by using this method under single mode condition according the equation of,

$$K = \sqrt{JE'}$$  \hspace{1cm} (3)

$E' = \frac{\nu}{1-\nu}$ for plane strain. $\mu$ and $\nu$ indicates the shear modulus and Poisson’s ratio of the material.

There is a large volume of published studies describing the role of $J$ – Integral in determining SIF values showing the method is has high accuracy. Xin Cui [27] and Giancarlo [28] make the valid point and strongly agrees that $J$ – integral method proves to be the most efficient and give accurate values of SIF in finite element analysis results. Table 2 represents the overview of fracture parameter and methods used in fracture analysis of bone done by previous researchers.
Table 2. The overview of fracture parameter and method used in bone fracture analysis.

| Source | Method | Bone | Significant |
|--------|--------|------|-------------|
| [29]   | Analyse the fracture toughness ($K$) of the cortical bone simulation model that was developed in MATLAB | Human cortical bone | The crack propagation paths show crack deflections due to osteons existence which improve in fracture resistance |
| [30]   | Simulation of crack growth using X-FEM | Cortical bone | Prediction of crack path with the availability of osteons in the cortical bone |
| [31]   | 2D microscale of crack propagation using X-FEM | Cortical bone | Weak cement line in cortical bone proves to reorient the crack propagation |
| [25]   | Stress intensity factor ($K$) and energy release rate ($G$) | NA | Elastic theories at crack tip under linear and non-linear elastic mechanic problems were discussed |
| [32]   | Finite element with a non-homogeneous mesh elements analyzing stress in the trabecular bone | Trabecular bone (femur) | Mesh with a non-homogeneous distribution of elasticity modulus were constructed to analyze principal stresses of the bone |
| [33]   | Fracture load and damage analysis of the non-homogeneous finite element model | Trabecular bone (femur) | The model developed stated that mesh sensitivity remains concerns in the model. The fracture load has high relationship with FE predicted |
| [34]   | Finite element with three-point bending based on experimental-numerical approach | Porsine rib bone | The FE model developed proves to have high efficiency in forecasting both force-displacement curve and the fracture patterns of the model |
| [35]   | Finite element model developed to test the load effect and failure pattern data were gathered | Proximal femur bone | FE model formulation able to predict femur bone’s strength and stiffness for quasi-static and impact conditions separately |

The DEM method has been widely used in FE analysis due to its simplicity and the method requires lessen memory and computing time requirement. However, this approach has some limitation where the accuracy of the SIF is strongly dependable on the mesh elements refinement. This is supported by the previous literature as mentioned in [5], DEM method has become obsolete because it gives inaccurate results of SIF compared to energy-based methods. Nevertheless, the method would produce accurate results with the adopting of singularity elements and mesh density in the model. The accuracy of the SIF is greatly dependent on the mesh densities introduced. The singularity elements suggested from Barsoum stated that there are two type of singularity elements which are triangular and rectangular singular quarter-point elements as seen in Figure 1.
Figure 1. Rectangular quarter-point elements (a) triangular quarter-point elements (b) and linear quadrilateral element (c).

2. Recommendation enhancement
The accuracy and efficiency of the data simulated is highly validated by previous studies, where the mesh refinement has proven to give the best result in the simulation data. Based on the previous study on meshing element for bone fracture related problem done by the researcher Solanki et al. [22], there are evidence showing the relationship of mesh elements and the accuracy of simulation results. However, previous studies on meshing elements have not dealt with meshing formulation for bone fracture that able to fit the harversian canal exist in the osteon of cortical bone. Thus, the suggestion for future enhancement can be summarized as:

a) The FE model developed represent the bone fracture behavior should enhance and focus on the mesh refinement to achieve more accurate results.

b) Validation of the developed model should be done to see the relationship between the mesh refinement elements and the accuracy of the simulation data gathered.

The proposed FE model of fracture mechanism is performed under specific biomechanics loading condition where all the fracture parameter is controllable. Hence, the fracture mechanic is favored to be accepted in clinical application.

3. Conclusions
This paper has given an account the importance of mesh elements in determine the most accurate computational analysis results in cortical and trabecular bone fracture study. From the past literatures, there is a lack amount of study involving the importance of implementing the mesh elements refinement in the finite element analysis study involving the bone fracture behaviour. Hence, a further study on the mesh element should be done in future studies so that the finite element models involving orthopaedics studies can be validated and verified. The authors of this paper attempted to highlight the importance of mesh refinement criteria to enhance the accuracy of the simulation model developed in human fracture bone applications. On the other hand, in finding the related SIF of the simulation model, several methods the SIF calculation has been implemented by various past researchers. Among all the methods used, J-integral has proven to be the most accurate method compared to the others.

Acknowledgments
The authors would like to acknowledge the support funded from Universiti Malaysia Perlis (UniMAP).

References
[1] Nash Gifford L and Hilton P D 1978 Eng. Fract. Mech. 10 485–96
[2] Zhu W X and Smith D J 1995 Eng. Fract. Mech. 51 391-400
[3] Nejati M, Paluszny A and Zimmerman R W 2015 *Int. J. Solids Struct.* **69-70** 230-251
[4] Li H, Guo R and Cheng H 2019 *Theor. Appl. Fract. Mech.* **101** 269-278
[5] Liao M and Zhang P 2019 An improved approach for computation of stress intensity factors using the finite element method *Theor. Appl. Fract. Mech.* **101** 185–90
[6] Paris P and Erdogan F 1963 *J. Basic Eng.* **85(4)** 528-533
[7] Han Q, Wang Y, Yin Y and Wang D 2015 *Eng. Fract. Mech.* **138** 118-126
[8] Guinea G V., Planas J and Elices M 2000 *Eng. Fract. Mech.* **66** 243-255
[9] Jia L-J, Shi M X, Zhao Y and Liu B 2013 *J. Appl. Mech.* **80(5)** 051014
[10] Cai Z, Kim S and Lee H-C 2018 *Comput. Math. with Appl.* **76** 2402–8
[11] Nejati M, Paluszny A and Zimmerman R W 2015 *Eng. Fract. Mech.* **101** 269–78