IOP Conference Series: Materials Science and Engineering*Open Access*Volume 991, Issue 122 December 2020 Article number 0121345th International Conference of Chemical Engineering and Industrial Biotechnology, ICCEIB 2020Kuala Lumpur9 August 2020through 11 August 2020Code 166005

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

Absract

For centuries, the finite element (FE) method has been extensively used to predict the fracture performance and various method has 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.