

股骨有限元素分析之整合介面開發

陳精一, 伍紹均

機械工程學系

工學院

meching@chu.edu.tw

摘要

This study integrated previous developed programs to develop all-in-one user friendly windows based program. The analysis is based on the finite element analysis. This study presents a new method of using the integrated interface program to perform an automated three-dimensional finite element meshing for femur by using the ANSYS® Software alone. This new methodology could provide a smooth boundary around the contour of femur as well as avoiding the ill-conditioned element. Therefore, the integrated software developed in this study, the finite element stress analysis is performed to compare with stress distribution of the intact femur, the femur created through previous software. Hopefully, it can effectively shorten time for creating the finite element model as well as cost. At the same time, this new methodology developed here could be applied to other similar bone structure in the field of biomechanics research.

關鍵字：Finite Element Analysis, ANSYS® Software, Femur, Biomechanics