Validation of Finite Element Analysis for a New External Finger Fixator to Correct Flexion Deformity – A Preliminary Result

H.S. Abd Rahman[1], N.A. Abu Osman[1], W.A.B. Wan Abas[1],
T.S. Tunku Ahmad[2] and E.S. Ng[2]

[1] Department of Biomedical Engineering, Faculty of Engineering, University of Malaya, Kuala Lumpur, Malaysia
[2] Department of Orthopaedic Surgery, Faculty of Medicine, Kuala Lumpur, Malaysia

Introduction

Nowadays finite element analysis have been applied comprehensively with much success in the analysis of orthopaedic devices [1,2,3,4,5]. This kind of analysis will show how models can be constructed for the optimal solution of problems in contact mechanics. New development in this analysis is of supreme importance in the design for a new external finger fixator. Basically, this device relates generally to the treatment of contractures of a skeletal joint and more particularly to an external fixation device for allowing the fingers joint to be flexed or extended by the patient actively or passively and maintain its alignment in the natural axis of rotation for managing contractures or fractures of the fingers. A simple external finger fixator has been developed using small threaded pins which are inserted on both side of the respective bone, and which are connected by a fixation device comprising an articulated joint permitting, under medical supervision, a natural movement of the articulation. In order to affirm that the new fixator can actually withstand the subjected load (occurs from the stiffness joint), finite element analysis was produced to validate the performance of each component rather than on physical testing.

Methodology

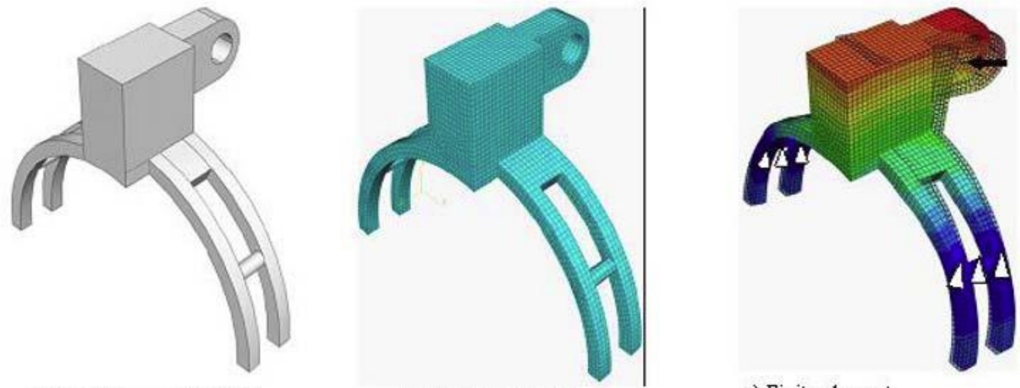
A. Geometry of the new external fixator

The new external finger fixator consist of proximal, middle, and distal bracing segment which are adapted to stay in site relative to one another, for example as shown in Fig. 1. The design of the new external finger fixator was made through some revision to improve from the previous external finger fixator by Siow *et al* [6]. The geometry of each part was modeled by using Computer Aided Design (Pro-Engineer® version 3.0, PTC, Needham, MA) software and all parts for each component were imported into the solid modeller, ABAQUS/CAE 6.7 software enabling the finite element mesh to be generated for each component.

Fig. 1 The new external finger fixator consists of distal, middle, and proximal bracing segment which constructed on the skeleton model

B. FE Mesh creation

Compared with physical testing, FEA offers a number of significant advantages in parameter studies, such as in internal stress analysis [7]. Using currently available software, the analyst has great flexibility in meshing, but must set element sizes and fine-tuning throughout the domain. All assumptions need to be simple to keep the FE models controllable, not only from the viewpoint of the complex geometries, but also in view of the computational working environment. From Fig. 2, distal part was applied to an automatic procedure of FE meshing with a quadratic mesh element base from the geometry model build in CAD drawing.



a) Modelling part in CAD b) Mesh generator c) Finite element

Fig. 2 Procedure from CAD modeling of distal part to finite elements

C. Loading and Material Properties

To facilitate the convergence tests, simplified loading, constraints and material properties were applied. Each of the components was modeled as a linear geometry model and were represented by 4-noded quadrilateral elements which defined internally by ABAQUS 6.7. All the components were made from medical grade stainless steel (316 L) and their properties were included into the calculation (Table 1). Table 2 shows the loading, loading direction and where the constraint applied during analysis. In terms of the finger force, many researchers have studied tendon forces of the finger in various actions such as gripping and pinching [8,9,10] using 2D or 3D biomechanical finger models. A uniformly distributed load of total 50N is initially applied at each section which results at a maximum deflection. The 50N load is actually the maximum grip force for each finger joint as described in several literatures [11,12,13]

Full text is available at :

http://link.springer.com/chapter/10.1007/978-3-540-69139-6_118