THE CONTACT MECHANICS OF THUMB CARPOMETACARPAL JOINT

1,3Mei-Ling Chou, 2Tsung-Chih Yu, 1Chih-Han Chang and 1Fong-Chin Su
1Institute of Biomedical Engineering, National Cheng Kung University, Tainan, Taiwan
2Metal Industries Research & Development Centre, Kaohsiung, Taiwan
3Email: dixmg@hotmail.com

SUMMARY
The osteoarthritis (OA) of thumb carpometacarpal (CMC) joint is a critical disease in upper extremity. Several studies suggested that high contact stress induced by unstable contact would result in osteoarthritis. However, only limited articles had investigated the relationship between contact modes and contact stress. This research examined the contact mechanics of the first CMC joint by finite element analysis and emphasized on the posture effect, including: resting and neutral in order to link the contact characteristics with the association with OA. The result showed the stress were concentrated on contact regions especially dorsalradial. The bending force would result in higher peak stress than compression force. The contact characteristics of finite element analysis were coinciding with previous experimental studies.

INTRODUCTION
The thumb plays an important role in various hand activities. The damage of thumb was associated with 50% loss of the hand function. Based on saddle-shaped CMC joint; thumb has a wide range of motion and can perform more complex function in comparison to four fingers. Moreover, the CMC joint affords high compression force so it is also a primary location for OA in the upper extremity. Several studies pointed out high stress induced OA. The CMC unstable factor including ligament laxity and joint incongruous would result in higher stress concentration. However, the contact mode of CMC joint is unclear. The evaluation of contact mechanics in thumb CMC joint is benefit for the understanding of OA.

To perform cadaveric experiments with the application of pressure sensitive film is one approach to measure the contact pressures and contact areas. Eaton and Littler pointed that CMC joint was less congruent during opposition and pinch; the peak contact stress was on dorsalradial of the trapezium [1, 2]. Koebbe found pronation of the metacarpal would contribute to the incongruous of joint and higher stress on volar-ulnar and dorsalradia [3]. Pieron noted that the joint contact on palmar side during basic postures, flexion/adduction [4]. Due to different postures and articular structure, the specimen would contact on various regions; several studies indicated that dorsalradial was the primary region.

Finite element analysis is a method to simulate load distribution and deformation patterns in the human body. It is widely applied on the contact mechanics analysis of knee, ankle, wrist, hip, elbow, and shoulder. Qilin et al. used the technology to research the pathogenesis of osteoarthritis in the CMC joint [5]. In the study, two-dimensional finite element method was used to simulate the CMC joint stress distribution. The study suggested that CMC joint instability resulting in minimal displacement cause higher stress concentration. Nevertheless, the two-dimensional model could not display the contact stress of CMC joint completely.

The objective of this research is to create three-dimensional finite element model in order to investigate the contact mechanics and characteristics of the first CMC joint under two thumb postures: resting and neutral. Because all thumb activity is governed by the composition of basic movements, the contact mechanism of the joint could be identified from the analysis of basic postures.

METHODS
The subject of this study was a male’s right hand. The subject was scanned by four-dimensional CT in two static postures: resting and neutral. The scans were displayed from thumb distal phalange to carpal bones. DICOM images were obtained for bone model reconstruction (pixel size: 0.3mm and slice distance: 0.6 mm).

One cadaver was dissected and only retained thumb metacarpal and trapezium bone with cartilage. The carpometacarpal joint was scanned to investigate the cartilage’s boundary and thickness by Micro CT (pixel size: 0.0344mm and slice distance: 0.0344 mm) and the cartilage model was reconstructed. Another model was manually acquired from the bone patch thicken 0.8mm. The manual model was compared with cartilage model.

The three-dimensional finite element model was imported into ANSYS 12 which was used to simulate the contact mechanics. This model included thumb metacarpal, trapezium bone and manual cartilage on two bones contact surface in two postures (Figure 1). The material properties were shown in Table 1. To simulate the compression and bending moment were 100N and 2.5Nm acting on distal head of metacarpal bone respectively. The trapezium in proximal and ulna side was fixed as the boundary condition.
RESULTS AND DISCUSSION
The cartilage model didn’t have significant difference with manual model. Therefore, the manual model was used to replace cartilage model and it could simplify cartilage geometry and reduce node and element number. Higher stresses concentration was located on contact area. The maximum Von Mises stresses was concentrated on dorsal-radial side (Figure 2, 3, 4, 5). In different postures, the sliding of joint space would result in the change of stress concentration region. The bending moment applied on the metacarpal would contribute to much higher peak stress on the CMC joint than compression pressure.

CONCLUSIONS
The study presented high contact stresses regions some as contact sites. The contact characteristics of finite element analysis were coinciding with previous studies.

REFERENCES

Table 1: Material properties

<table>
<thead>
<tr>
<th></th>
<th>Young’s modulus (MPa)</th>
<th>Poisson’s ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bone</td>
<td>12000</td>
<td>0.3</td>
</tr>
<tr>
<td>Cartilage</td>
<td>35</td>
<td>0.45</td>
</tr>
</tbody>
</table>