國立陽明交通大學
NATIONAL YANG MING CHIAO TUNG UNIVERSITY

電腦輔助工程分析

ANSYS WORKBENCH

國立陽明交通大學 生物醫學工程系
林峻立 特聘教授

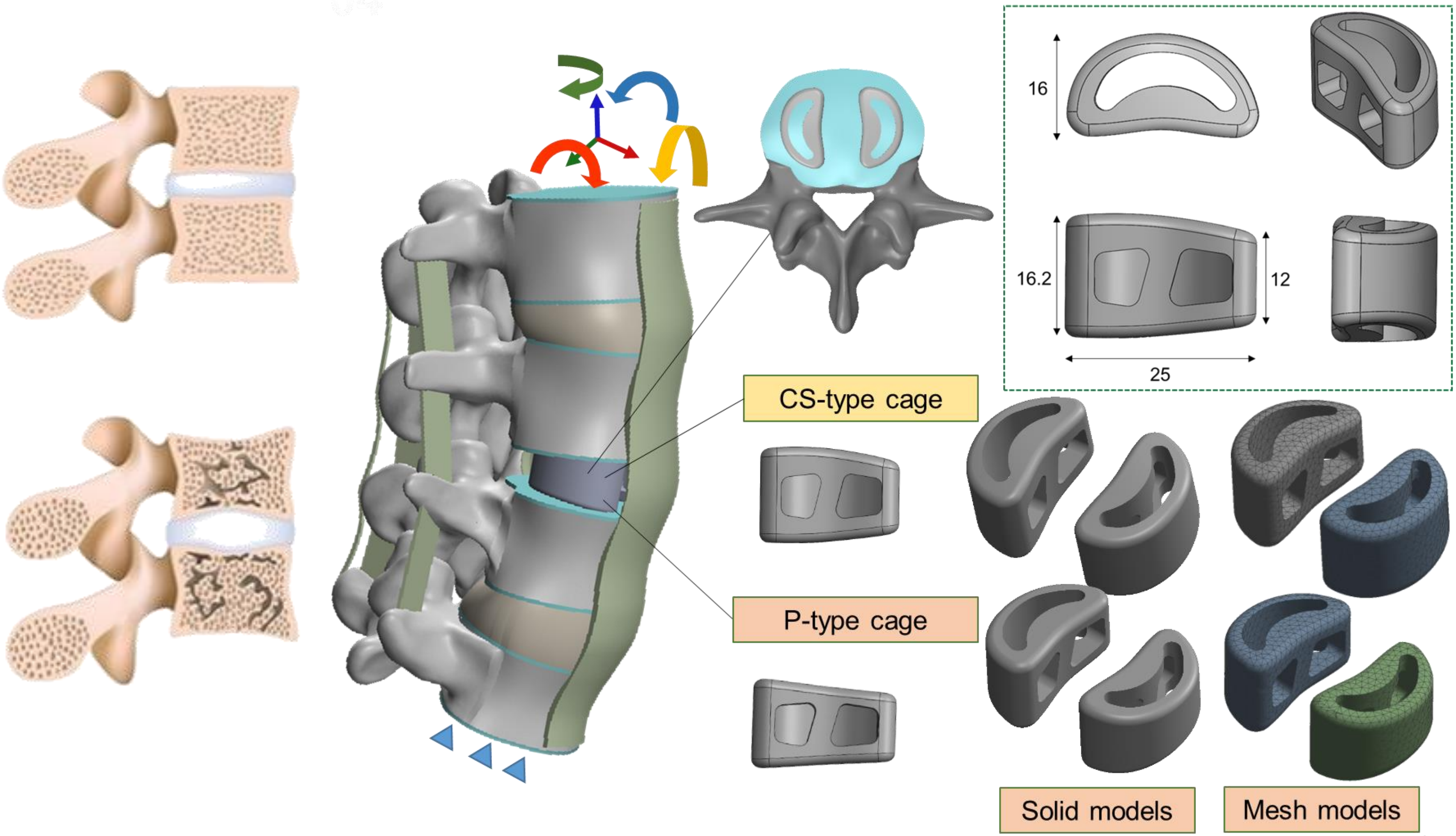
2023/02

OUTLINE

- **00** **Class Introduction**
- **01** **Concept Introduction**
- **02** **Workbench**
- **03** **Design Modeler**
- **04** **Static Structural**
- **05** **Advanced Analysis**

CAE/FEM Applications

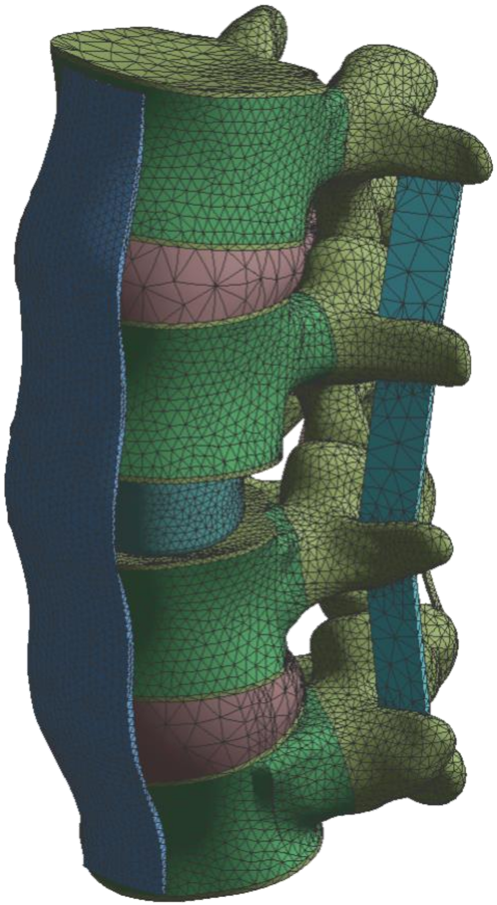
■ 新型骨鬆用椎籠(Cage)設計與分析

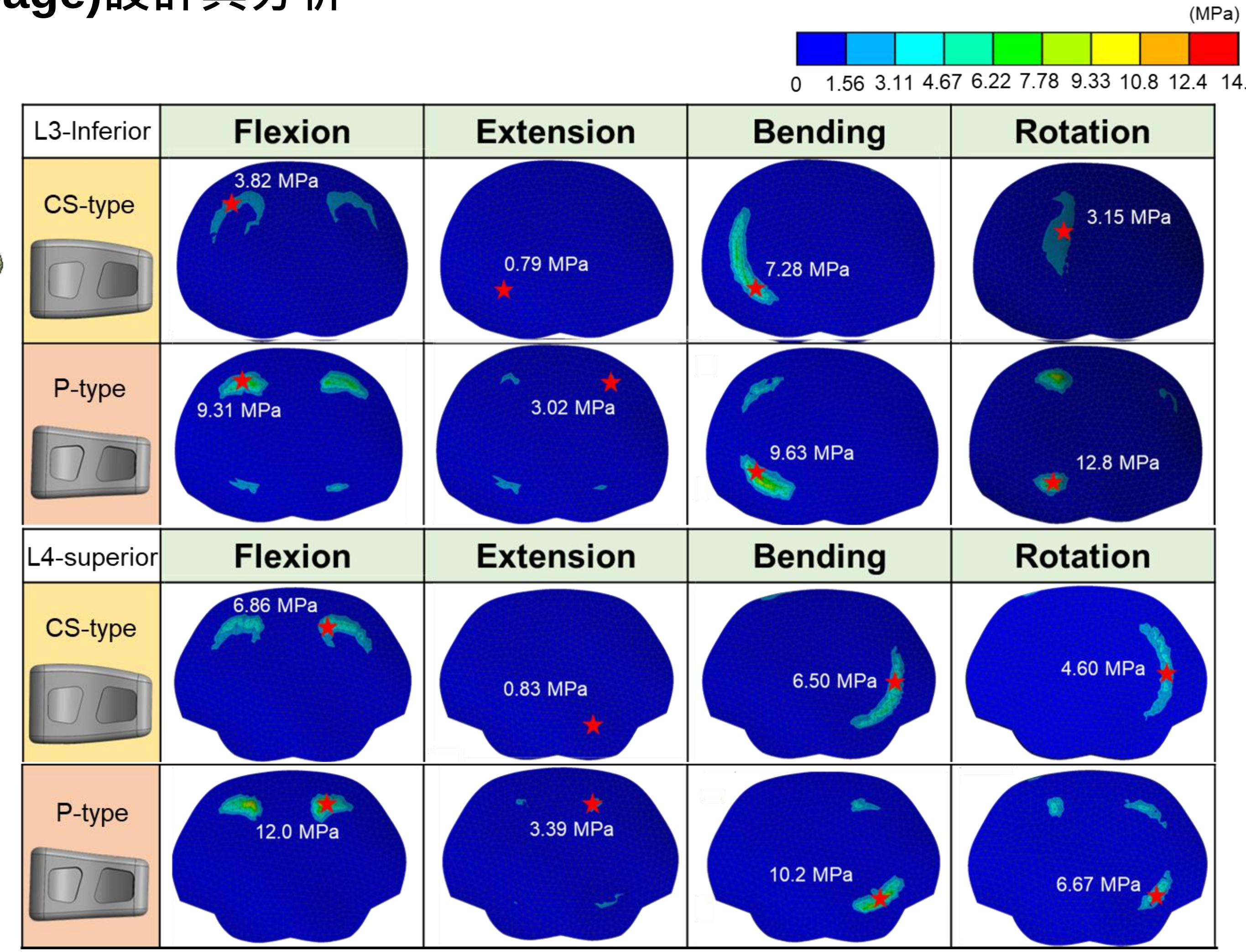


詹益庭（2019）。適合骨質疏鬆患者使用之新型椎籠設計及生物力學分析。

CAE/FEM Applications

■ 新型骨鬆用椎籠(Cage)設計與分析

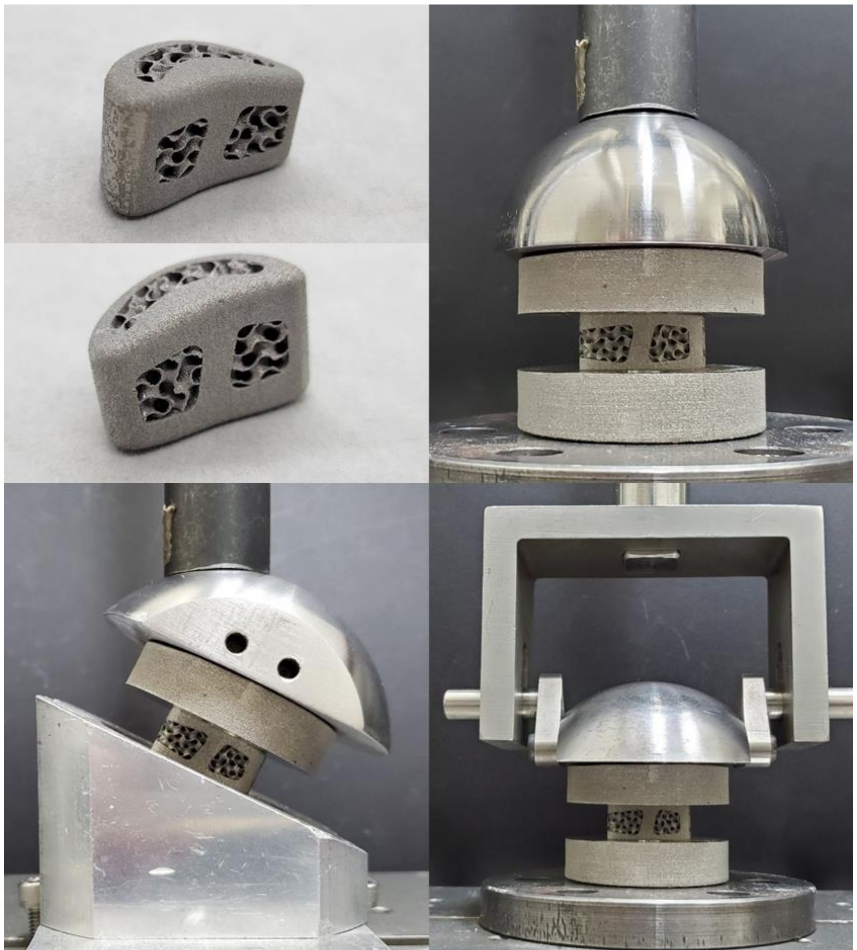




詹益庭（2019）。適合骨質疏鬆患者使用之新型椎籠設計及生物力學分析。

CAE/FEM Applications

■ 新型骨鬆用椎籠(Cage)設計與



WHIOCE PUBLISHING PTE. LTD.

International Journal of Bioprinting

RESEARCH ARTICLE

Biomechanical evaluation of an osteoporotic anatomical 3D printed posterior lumbar interbody fusion cage with internal lattice design based on weighted topology optimization

Shao-Fu Huang[1,2], Chun-Ming Chang[3], Chi-Yang Liao[1,4,5], Yi-Ting Chan[1], Zi-Yi Li[1], Chun-Li Lin[1,2]*

[1]Department of Biomedical Engineering, National Yang Ming Chiao Tung University, Hsinchu, Taiwan

[2]Innovation and Translation Center of Medical Device, Department of Biomedical Engineering, National Yang Ming Chiao Tung University, Hsinchu, Taiwan

[3]National Applied Research Laboratories, Taiwan Instrument Research Institute, Hsinchu, Taiwan

[4]Department of Orthopedics, Tri-Service General Hospital Songshan Branch, National Defense Medical Center, Taipei, Taiwan

[5]Department of Surgery, Tri-Service General Hospital Songshan Branch, National Defense Medical Center, Taipei, Taiwan

***Corresponding author:** Chun-Li Lin (cllin2@ym.edu.tw)

Citation: Huang S, Chang C, Liao C, *et al.*, 2023, Biomechanical evaluation of an osteoporotic anatomical 3D printed posterior lumbar interbody fusion cage with internal lattice design based on weighted topology optimization. *Int J Bioprint*, 9(3): 0212. https://doi.org/10.18063/ijb.v9i3.0212

Received: September 27, 2022

Accepted: November 27, 2022

Published Online: XXX

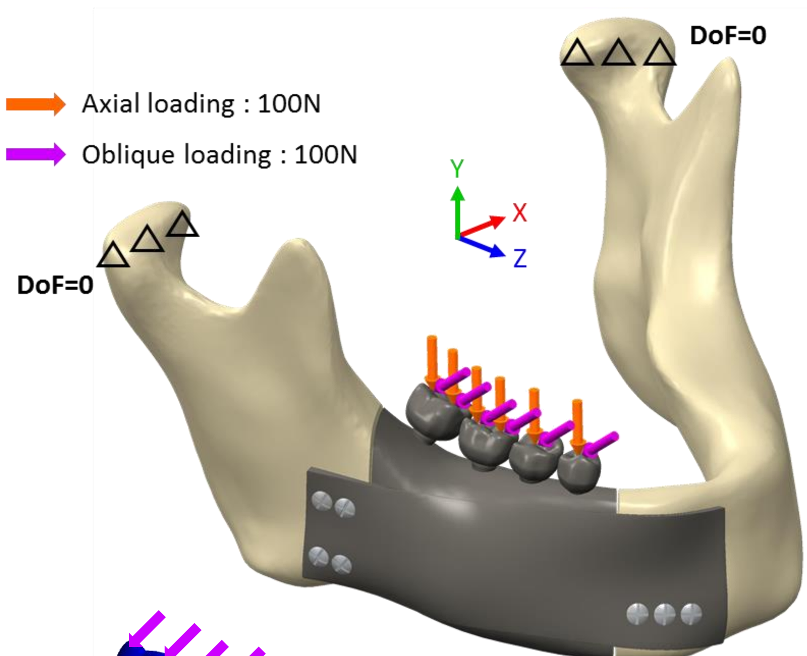
Abstract

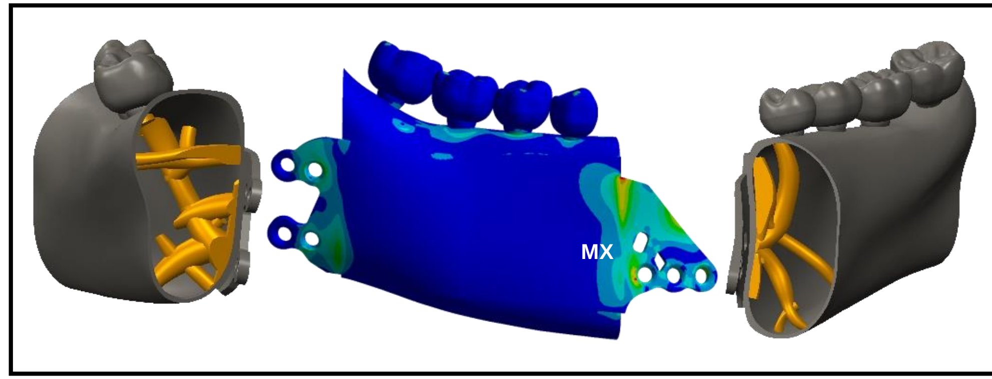
In this study, we designed and manufactured a posterior lumbar interbody fusion cage for osteoporosis patients using 3D-printing. The cage structure conforms to the anatomical endplate's curved surface for stress transmission and internal lattice design for bone growth. Finite element (FE) analysis and weight topology optimization under different lumbar spine activity ratios were integrated to design the curved surface (CS-type) cage using the endplate surface morphology statistical results from the osteoporosis patients. The CS-type and plate (P-type) cage biomechanical behaviors under different daily activities were compared by performing non-linear FE analysis. A gyroid lattice with 0.25 spiral wall thickness was then designed in the internal cavity of the CS-type cage. The CS-cage was manufactured using metal 3D printing to conduct *in vitro* biomechanical tests. The FE analysis result showed that the maximum stress values at the inferior L3 and superior L4 endplates under all daily activities for the P-type cage implantation model were all higher than those for the CS-type cage. Fracture might occur in the P-type cage because the maximum stresses found in the endplates exceeded its ultimate strength (about 10 MPa) under flexion, torsion and bending loads. The yield load and stiffness of our designed CS-type cage

詹益庭（2019）。適合骨質疏鬆患者使用之新型椎籠設計及生物力學分析。

CAE/FEM Applications

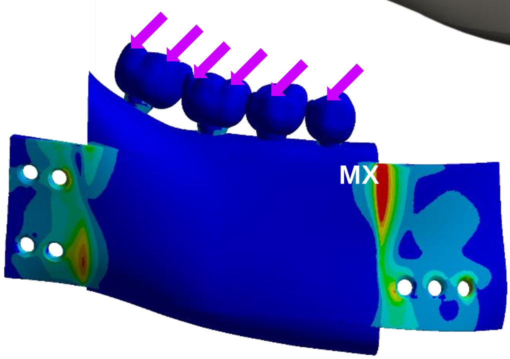
- 下顎骨植入物最佳化與力學分析





結構最佳化與力學分析結果

金屬3D列印利用分析開發出之新型植入物



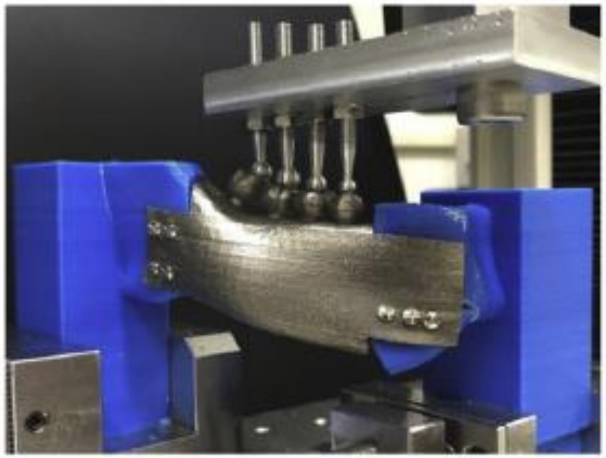
結構固定處設計分析

Li, C. H., Wu, C. H., & Lin, C. L. (2020). Design of a patient-specific mandible reconstruction implant with dental prosthesis for metal 3D printing using integrated weighted topology optimization and finite element analysis. *Journal of the mechanical behavior of biomedical materials*, *105*, 103700.

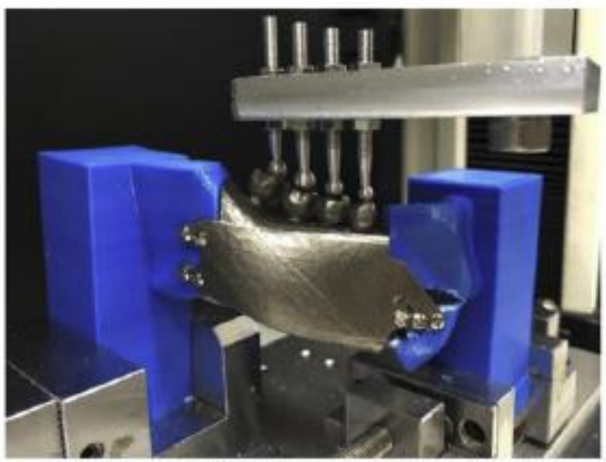
CAE/FEM Applications

■ 下顎骨植入物最佳化與力學分析

Fracture patterns



OM model



WBOS model

journal of the mechanical behavior of biomedical materials 105 (2020) 103700

Contents lists available at ScienceDirect

ELSEVIER

Journal of the Mechanical Behavior of Biomedical Materials

journal homepage: http://www.elsevier.com/locate/jmbbm

Check for updates

Design of a patient-specific mandible reconstruction implant with dental prosthesis for metal 3D printing using integrated weighted topology optimization and finite element analysis

Chia-Hsuan Li [a], Cheng-Hsien Wu [b], Chun-Li Lin [a,*]

[a] *Department of Biomedical Engineering, National Yang-Ming University, 2 No.155, Sec.2, Linong Street, Taipei, 112, Taiwan*

[b] *Oral & Maxillofacial Surgery, Taipei Veterans General Hospital, School of Dentistry, National Yang-Ming University, 2 No.155, Sec.2, Linong Street, Taipei, 112, Taiwan*

A R T I C L E I N F O

Keywords:
Patient-specific implant
Mandibular reconstruction
Dental prosthesis
3D printing
Topology optimization
Finite element analysis

A B S T R A C T

The aim of this study was used a weighted topology optimization method to design a patient-specific mandibular implant for reconstruction and restoration of appearance in patients with severe mandibular defects. A finite element (FE) model was constructed and the defect region was defined from the unilateral first premolar to the second molar. The reconstruction implant included main body, fixation wing and dental prosthesis. Standard topology optimization was performed using stress constraint to identify optimal fixation wing structure (denoted as WOS) with solid core main body. Two independent optimal main body with internal beam supporting structures defined as WOSA and WOSO optimized from the WOS model under axial and oblique conditions were then obtained, respectively. Final optimal model (WBOS) was generated using a weighted topology optimization that considered 60% and 40% contributions of WOSA and WOSO models, respectively. The WBOS model was fabricated using metal 3D printing and fixed on the resting acrylonitrile butadiene styrene (ABS) bone to perform fracture testing. Stress concentration were found in the upper area connected to the main body of the mesial wing and corresponding maximum values under axial/oblique loads were reduced from 778/925 MPa of the WOS model to 764/720 MPa of the WBOS model. The reduction in percentage variations of weight between original (91.1 g) and final optimal (24.5 g) models was 73.14% for fabricated 3D printing models. The WBOS model also exhibited a higher resistant force (2163 N) when compared with the original model (1678 N). This study developed a design strategy with weighted topology optimization and fabrication for producing patient-specific implants using metal 3D printing. The obtained reconstruction implant can provide good biomechanical performance and recovery of appearance for oral rehabilitation.

1. Introduction

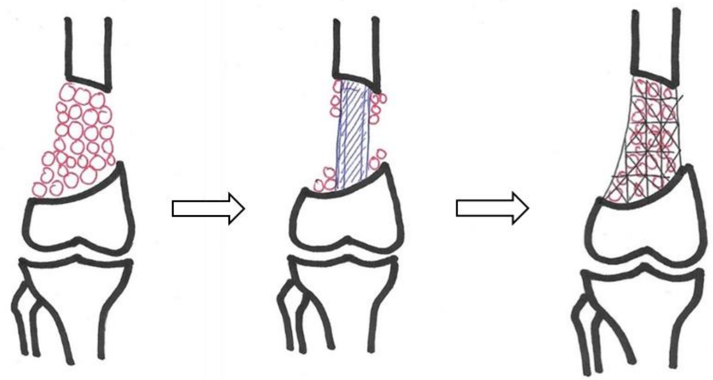
The main objective of reconstruction for severe mandibular defects is to restore functional components of the facial skeleton and contribute to individual facial identity, mastication, speech, swallowing, and

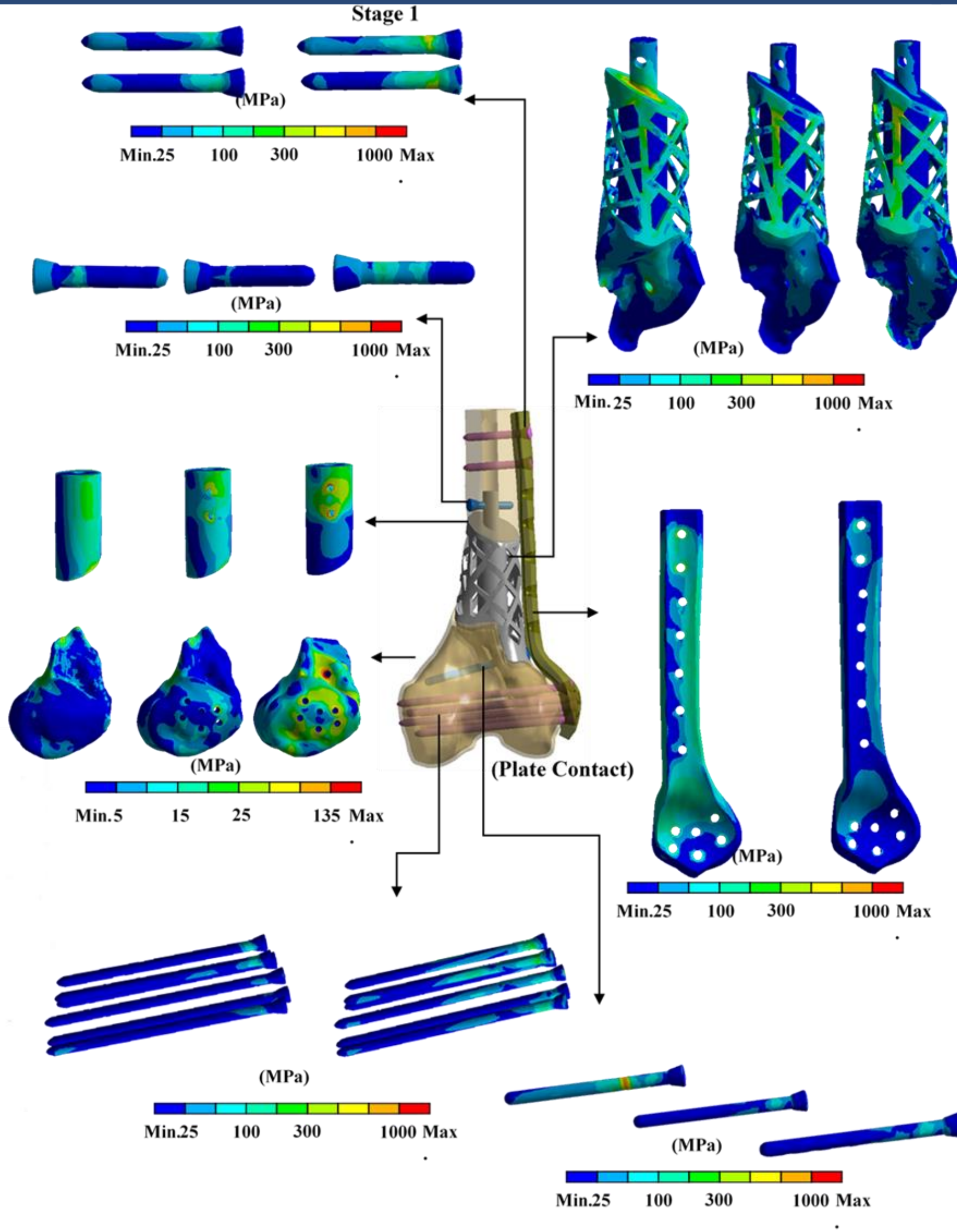
of the mandible, and to restore facial contours and masticatory function (Pinheiro and Alves, 2015; Stoor et al., 2017; Yusa et al., 2017; Lee et al., 2018; Cheng et al., 2019). These considerations are particularly important for patients who need complex postoperative dental prostheses that ensure quality of life.

Li, C. H., Wu, C. H., & Lin, C. L. (2020). Design of a patient-specific mandible reconstruction implant with dental prosthesis for metal 3D printing using integrated weighted topology optimization and finite element analysis. *Journal of the mechanical behavior of biomedical materials*, *105*, 103700.

CAE/FEM Applications

- 股骨大範圍缺損力學分析





資料來源https://www.cybernet-ap.com.tw/?m=688&appcenter_catid=24&appcenter_storyid=174

CAE/FEM Applications

■ 股骨大範圍缺損力學分析

applied sciences

MDPI

Article

Patient-Specific 3-Dimensional Printing Titanium Implant Biomechanical Evaluation for Complex Distal Femoral Open Fracture Reconstruction with Segmental Large Bone Defect: A Nonlinear Finite Element Analysis

Kin Weng Wong [1,2], Chung Da Wu [2], Chi-Sheng Chien [2,3], Cheng-Wei Lee [4], Tai-Hua Yang [1,5,6,*] and Chun-Li Lin [4,*]

1. Department of Biomedical Engineering, National Cheng Kung University, Tainan 601, Taiwan; P88071046@ncku.edu.tw
2. Department of Orthopedic Surgery, Chi-mei Medical Center, Tainan 601, Taiwan; wcd@mail.chimei.org.tw (C.D.W.); cschien@stust.edu.tw (C.-S.C.)
3. Department of Electrical Engineering, Southern Taiwan University and Technology, Tainan 112, Taiwan
4. Department of Biomedical Engineering, National Yang-Ming University, Taipei 11221, Taiwan; justinlee102185@ym.edu.tw
5. Department of Orthopedic Surgery, National Cheng Kung University Hospital, Tainan 601, Taiwan
6. Medical Device Innovation Center, College of Medicine, National Cheng Kung University, Tainan 601, Taiwan

* Correspondence: yangtaihua@mail.ncku.edu.tw (T.-H.Y.); cllin2@ym.edu.tw (C.-L.L.)

Received: 14 May 2020; Accepted: 10 June 2020; Published: 14 June 2020

check for updates

Abstract: This study proposes a novel titanium 3D printing patient-specific implant: a lightweight structure with enough biomechanical strength for a distal femur fracture with segmental large defect using nonlinear finite element (FE) analysis. CT scanning images were processed to identify the size and shape of a large bone defect in the right distal femur of a young patient. A novel titanium implant was designed with a proximal cylinder tube for increasing mechanical stability, proximal/distal shells for increasing bone ingrowth contact areas, and lattice mesh at the outer surface to provide space for morselized cancellous bone grafting. The implant was fixed by transverse screws at the proximal/distal host bone. A pre-contoured locking plate was applied at the lateral site to secure the whole construct. A FE model with nonlinear contact element implant-bone interfaces was constructed to perform simulations for three clinical stages under single leg standing load conditions. The three stages were the initial postoperative period, fracture healing, and post fracture healing and locking plate removal. The results showed that the maximum implant von Mises stress reached 1318 MPa at

資料來源https://www.cybernet-ap.com.tw/?m=688&appcenter_catid=24&appcenter_storyid=174