UKACC International Conference on Control 2012
Cardiff, UK, 3-5 September 2012

Designing and Finite Element Analysis of customized titanium plate for Mandible

Tianbiao Yu, Guoqiang Ma
Tianhua Wei, Wanshan Wang
School of Mechanical Engineering & Automation
Northeastern University
Shenyang, China
tbyu@mail.neu.edu.cn

Xingjun Qin
Department of Oral and Maxillofacial Surgery
China Medical University
Shenyang, China

Abstract—**The Rigid Internal Fixation is one of the commonly used methods to reconstruct the mandible in bone repair surgery. This research proposed a complete flow of the designing and Finite Element Analysis (FEA).The customized titanium plate and screws model were imported in Mimics, then fitted and simulated with the shape of mandible 3D model. The simulation model would be imported into Magic's and the screw holes were located and punched. In order to ensure the validity of model for surgical guidance, the FEA was employed by FEA software Abaqus in this study. Different angles were selected and different loads were loaded. Comparing with the nomal mandible mechanical data, the von mises stress and deformation which was from the reconstruction of mandible were in the regular range and the RP model which would be used in the medical guidence was manufactured according to the 3D model. The research for this processing has important guide significance and good business prospects.**

Keywords-customized ti-plate, fixation and simulation, FEA in Abaqus, model buliding

I. INTRODUCTION

The reconstruction of mandible is a great challenge in oral and maxillofacial surgery. The loss of mandibular bone leads to severe deformation and dysfunctions such as swallowing, chewing and speaking. Usually bone defects in mandible region are caused by tumors and bone infections [1]. The traditional reconstruction plates are difficult to form into the mandible anatomical shape. Plate failures always occur at the plate bending area and usually lead to unsatisfactory functional results to patients. Because of the complexity of human anatomy and individual difference in general, the construction of true anatomical model based on the individual patient has great significance. Combining with image processing and visualization techniques, the customized model as well as the finite element biomechanical simulation model has possibility to be established by medical images which have no trauma. The customized biomechanical modeling based on CT images provides accurate and simple way to establish the finite element model of organisms which try its best to be faithful to the customized anatomy structure and simulate the biomechanical behavior of living tissue with high precision.

In this study, the mandible reconstruction was conducted by using the segmental fibulas, which were similar to the defect of mandible in morphology and fixed by titanium plate and screws. This method involves Rigid Internal Fixation (RIF), which takes bony cortex of lateral law as basis of fixation. Michelet et al. (1973) proposed the notion of simple cortex of bone osteosynthesis. RIF achieves rigid fixing in 3D and has the advantages of fast healing, no external callus and direct cicatrization.

In order to fix the reconstruction mandible with the fibular, a customized titanium plate will be designed. The reconstructive mandible FE model will be established and prepare to be analysed in Abaqus. After the mandible reconstructed, the mandible model will be disposed by a Finite Element Analysis. Comparing with the normal mandible mechanical data, if the FEA data is regular, the RP model will be manufactured according to the 3D model. The whole process is represented by the Fig.1.

As in engineering, the initial work with FEA in medicine was the analysis of the macroscopic solid structures [2].The FEA was also used some years ago to predict dental implant loading [3], [4] and for micro structural analysis [5].

II. METHODS

A. Mandible Fixation and Simulation.

The reconstructive mandible 3D virtual model was established from the patient's CT data in Mimics [6].According to the mandible shape, the customized titanium plate was modeling in Pro/E condition. The titanium screw model also was built and assembled with the titanium plate. In order to accurately conduct FEA, the thread should be omitted to prevent irregular triangles and other errors appearing. It was the titanium screw model in Fig.2 and the assembly of titanium plate and titanium screws was in Fig.3.

The assembled titanium plate and titanium screws were imported Mimics with STL format, then fitted and simulated with the shape of mandible 3D model as shown in Fig. 4.

After the simulation, the 3D model would be input Magics software (Materialise, Inc., Leuven, Belgium). Depending on the location of titanium screws, the screw holes would be located and punched as shown in Fig. 5 and Fig.6. In Magics, the STL file was edited and simplified so that it was only one shell and contained a regular and reduced number of triangles.

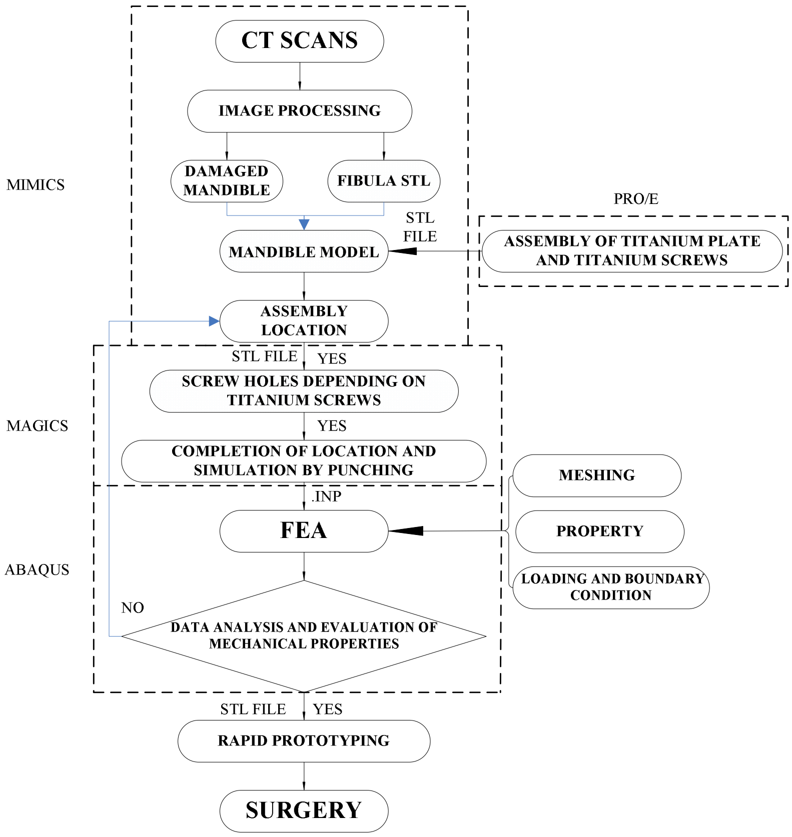


Figure 1. Whole process of designing and force analysis

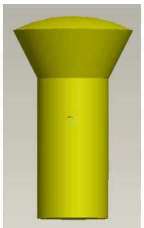


Figure 2. Titanium screw model

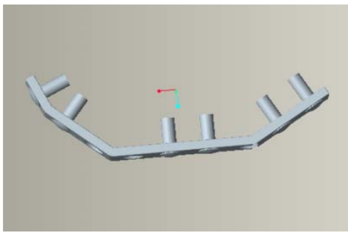


Figure 3. Assembly of titanium plate and titanium screws

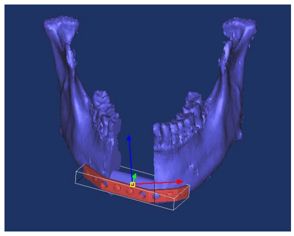


Figure 4. Fitting of titanium and mandible 3D model

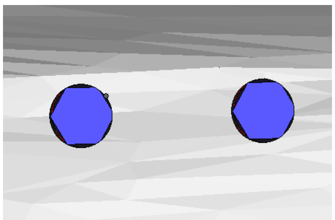


Figure 5. Screw holes

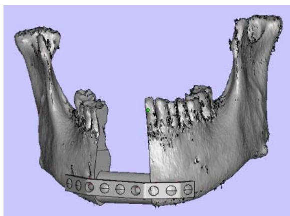


Figure 6. Reconstructive mandible and titanium plate

This research has its advantage of mandible fixation and simulation. The screw holes were punched firstly in traditional CAD model, then the titanium screws were located depending on the screw holes. In this research, the mandible shape was irregular and the titanium plate and titanium screws were modeling in Pro/E condition. If one screw hole was located and punched at first, the location of the next screw hole must be wrong because the titanium plate is limited. Contrary to the traditional way, the assembly of titanium plate and titanium screws was located at first, then the screw holes would be fixed and punched depending on the titanium screws. This method ensures the accuracy of screw hole's location in this non-traditional CAD model.

B. FE Model Establishment

In Magics, the repaired model was imported to the remeshing module which could divide the model into triangle meshes and finish the repair and optimization, and then the preprocessing documents of finite element method were exported. In this paper, a nonlinear finite element analysis software named Abaqus was adopted, so the exported document should be stored as .inp. Then the .inp model would be imported into Abaqus and converted from triangular meshes to tetrahedral meshes through the mesh module of Abaqus. The mesh model is shown in Fig.7. The tetrahedral meshes totally had 138866 tetrahedral elements and 30586 nodes as shown in Table I.

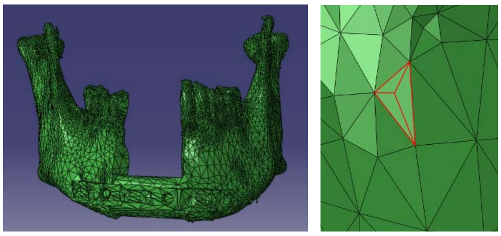


Figure. 7. Reconstruction mandible after remeshed and its grid cell

TABLE I. PARAMETER OF MANDIBLE FE MODEL

Number of nodes	Number of elements	Element types
30586	138866	C3D4

C. Property

In Abaqus, a new job in FE model would be created and exported to Mimics with .inp file. The property was given by module which the Mimics came with. The following is the formula for calculating the property parameters.

$$\begin{cases} \text{Density} = -13.4 + 1017 \times \text{Grayvalue} \\ \text{E - Modulus} = -388.8 + 5925 \times \text{Density} \end{cases} \quad (1)$$

After the property was given completely, the model should be exported from Mimics to Abaqus with .inp file. Examining the property module, the model's color had turned light blue, which proved that the property was given successfully. The model with property is shown in Fig.8.

This is the essence: CT images of bone were calculated according to the pixel and the CT value would be obtained. The density of each part would be got from CT value and the property parameters of each skeletal part could be calculated successfully relying on the formula (1).The mandible consists of many composite biomaterials,so endowing property artificially would caused unexpected inaccuracy and errors. Comparing with the artificial way, it is a good choice to endow property with Mimics software.

D. Loading and Boundary Condition

Before the FE model was done an analysis, the loading and boundary condition need to be set.

Boundary constraint is a very important issue related to the accuracy of FE model. The exactness or not of boundary constraint will directly affect the accuracy of the result. The perfect instance is that all boundary conditions are the same with actual instance, but it is almost impossible owing to the complex configuration. Therefore, the boundary conditions need to be predigested. As to the definition of mandibular boundary conditions, there was no uniform conclusion. Li ling et.al [7] achieved reasonable result with such boundary conditions: the mandibular lower edge as the underside boundary, acetabulum plane as the upside boundary and back of mandibular rise-offset as definition. This paper set the boundary conditions as shown in Fig. 9.

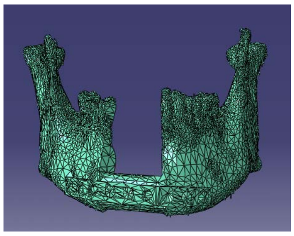


Fig.ure 8. Model with property

Figure 9. Loading and boundary conditions

The left tooth and the fibular surface were loaded in this analysis. The following are the boundary conditions.

$$\begin{cases} \text{acetabulum plane:} \\ ZASYMM(U1 = U2 = UR3 = 0) \\ \text{The reconstructive mandible middle line:} \\ XSYMM(U1 = UR2 = UR3 = 0) \\ \text{The mandible trailing edge:} \\ ENCASTRE(U1 = U2 = U3 = UR1 = UR2 = UR3 = 0) \end{cases}$$

E. Finite Element Analysis in Abaqus

According to the actual mandibular occlusal force condition, the elastic mechanics theory was used to analyze the mandibular biomechanics. Spatial problem has 15 unknown numbers: six stress components σ_x 、 σ_y 、 σ_z 、 $\tau_{yz} = \tau_{zy}$ 、 $\tau_{zx} = \tau_{xz}$ 、 $\tau_{xy} = \tau_{yx}$; six deformation components ξ_x 、 ξ_y 、 ξ_z 、 γ_{yz} 、 γ_{zx} 、 γ_{xy} ; three displacement components u 、 v 、 w . These 15 unknown functions should meet the 15 basic equations.

The space problem of equilibrium differential equation:

$$\left.\begin{aligned} \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + f_x = 0 \\ \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \frac{\partial \tau_{xy}}{\partial x} + f_y = 0 \\ \frac{\partial \sigma_z}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + f_z = 0 \end{aligned}\right\} \quad (2)$$

According to the elastic mechanics theory, the elastomer stress boundary condition:

$$\left.\begin{aligned} l(\sigma_x)_s + m(\tau_{yx})_s + n(\tau_{zx})_s = \overline{f_x} \\ m(\sigma_y)_s + n(\tau_{zy})_s + l(\tau_{xy})_s = \overline{f_y} \\ n(\sigma_z)_s + l(\tau_{xz})_s + m(\tau_{yz})_s = \overline{f_z} \end{aligned}\right\} \quad (3)$$

It demonstrated the relationship between the stress components of the boundary value and the plane force components [8].

Considering the above points, the steps were set for Finite Element Analysis after importing mandibular model into Abaqus. Two steps were set separately to raise the simulation reality. In order to raise the load regularly, the first step load is set low, while the second normal. This research mainly analyzed the force of chewing, as the force of not chewing was too low to consider; thus, we adopt Static-General analysis.

Calculation formula of Von Mises (σ_e) is：

$$\sigma_e = \left\{ \frac{1}{2} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right] \right\}^{\frac{1}{2}} \quad (4)$$

σ_1、σ_2、σ_3 is the first, second and third principal stress [8].

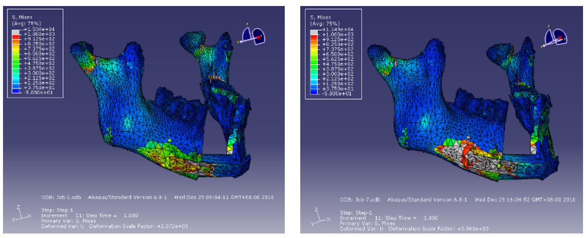
F. Results and Discussion

When occluding, chewing muscles drive teeth to chew the food so as to make the mandible suffer load; and when not occluding, mandible hardly support load. Therefore, the analysis of mandibular force is mainly the force analysis of occluding. Bai shizhu, Li dichen et.al [9] applied force on the occluding plane to FEA the mandible, and got relatively accurate result.

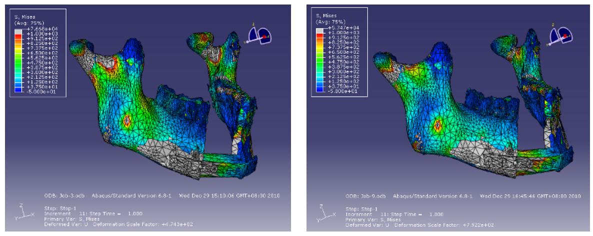
In this study, the force was applied separately in the direction of vertical, 45° with the occluding plane (teeth force range when occluding) to analyze the force. The load magnitudes are 100N and 500N.

Depending on the magnitude of the load and the analysis of different loading directions, we got reconstruction mandible stress simulation results as shown in the reconstruction of mandibular force after the simulation process. The maximum stress was at mandibular condylar neck.

According to the Von Mises stress diagram of reconstructive mandible, we can get the results
with the magnitude of load increasing, right side of the mandible and fibula junction, and the mandible condyle the stress concentration region increased. The maximum stress of the Von Mises stress diagram of reconstructive mandible increased as the load increased gradually. The Von Mises stress is shown in Fig.10.



（A）100N-0°　　（B）100N-45°



（C）500N-0°　　（D）500N-45°

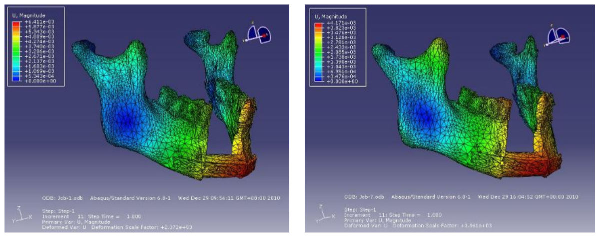
Figure 10. Result of reconstructive mandible Von Mises stress on different conditions

Reference to the classification criteria from Urken et al [10], this model belongs to B-type damage. Based on the results from Abaqus, the maximum stress of the Von Mises is shown in Table II.

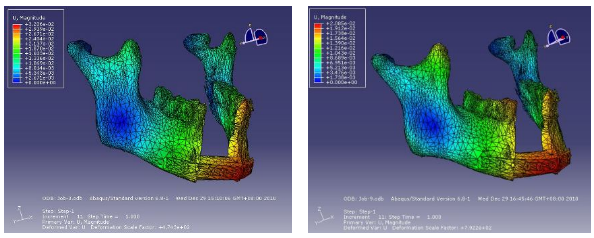
TABLE II. MAXIMUM STRESS OF THE VON MISES

Angles and loads	Maximum stress of the Von Mises (Mpa)
100N-0°	0.01534
500N-0°	0.07668
100N-45°	0.01149
500N-45°	0.05747

According to the deformation results, with the magnitude of load increasing, the maximum deformation increased at the mandible midline. The mandible maximum deformation was increasing as the load was increasing gradually. The mandible deformation is shown in Fig.11.



（A）100N-0°　　（B）100N-45°



（C）500N-0°　　（D）500N-45°

Figure 11. Result of reconstructive mandible deformation on different conditions

The maximum deformation was at left side of the mandible and fibula junction and it is shown in Table III.

TABLE III. MAXIMUM DEFORMATION

Angles and loads	Maximum deformation (mm)
100N-0°	0.06411
500N-0°	0.32060
100N-45°	0.04171
500N-45°	0.20850

III. MODEL BUILDING

Refer to the reference [11], the von mises stress and deformation were in the regular range and the RP model can be manufactured with 3DP rapid prototyping which is from Israel. The model with support materials is shown in Fig.12 and the model removing support materials is shown in Fig.13. The doctor could measure kinds of datum and practice operation to enhance the efficiency and success rate of surgery by 3D model which had been manufactured.

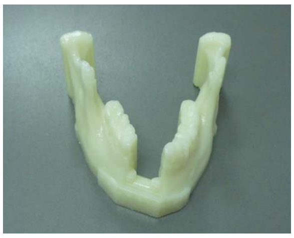


Figure 12. Model with support materials

Figure 13. Model without support materials

IV. CONCLUSIONS

Sekou Singare and Dichen Li [12] had established the mandible model successfully with CT data and curve fitting. The RP model had been manufactured at first, then the mold would be reproduced. After that, the titanium implant had been casted and implanted into the human body completely as shown in Fig.14.Comparing with the method which Sekou Singare had proposed, this set of methods described in this study have four advantages.

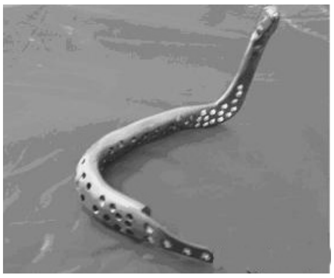


Figure 14. Titanium imported part

1) This set of methods were made up of kinds of professional software and largely reduced uncertain factors and errors which came from researcher's action. The researcher could build accurate outer contour and precise FEA, so the accuracy of model and the success rate of operation could be enhanced. This research is applicable to most of bone repair and large-scale promotion, so they have good business prospects.

2) Because the titanium implant was in the human body for a long time, the metal ions may leak and have a bad effect on human health [13]. Long-term use would also lead to increasing the internal stress and the rate of fatigue fracture. When the RIF was used, the titanium plate could be removed by the second operation after the mandible grew together with the fibula and the repaired mandibular could stand kinds of normal bite force.

3) After the FE model was established, the mechanical data would be analyzed to judge whether the model was qualified. It could decrease the risk of surgery, optimize the surgery program and alleviate the patient's suffering.

4) Autogenous bone graft has the capacity of becoming its marked osteogenic bone and resisting against infections.

This study represents a new approach for surgical planning and simulation, and both the designer and surgeon appreciate the meaning for comprehending the model. The mandible FE model has the ability to do the biomechanical analysis for the mandible model, and increase surgical precision over surgeries performed without the aid of mandible FE model. It should be recognized that there are a few shortcomings to improve such as: the intermediate links which were prone to lose some datum, higher operating skills which need higher knowledge to master, so it remains a big challenge and deserves the need for more attention and further research.

V. REFERENCES

[1] S. Singare, D. Li, B. Lu, Y. Liu, Z. Gong and Y. Liu, "Design and fabrication of custom mandible titanium tray based on rapid prototyping", Medical Engineering & Physics, Vol. 26, p. 671-676, 2004.

[2] Kavanagh EP, Frawley C, Kearns G, McGloughlin T and Jarvis J, "Use of finite element analysis in presurgical planning: treatment of mandibular fractures", Ir J Med Sci, Vol.177, p. 325-31, 2008.

[3] O'Mahony AM, Williams JL and Spencer P, "Anisotropic elasticity of cortical and cancellous bone in the posterior mandible increases peri-implant stress and strain under oblique loading", Clin Oral Implants Res, Vol.12, p. 648-57, 2001.

[4] Huang HL, Lin CL, Ko CC, Chang CH, Hsu JT and Huang JS, "Stress analysis of implant-supported partial prostheses in anisotropic mandibular bone: in-line versus offset placements of implants", J Oral Rehabil, Vol.33, p.501-8, 2006.

[5] Korioth TW, Romilly DP and Hannam AG, "Three dimensional finite element stress analysis of the dentate human mandible", Am J Phys Anthropol, Vol.88, p. 69-96, 1992.

[6] W.Z. Wu, X.J. Qin, Y. Zhang and W.S. Wang, "Mandibular Virtual Reconstruction Surgery Guided by Mimics", Applied Mechanics and Materials, Vol. 16-19, p. 842-846, 2009.

[7] L. Li, S. Xue and F. Q. Zhang, "The establishment of three-dimension finite element model for Upper and lower jaws and dentition", Dental Materials and Devices, Vol.12, p.117-121, 2003. (in Chinese)

[8] Z. L. Xu, Theory of elasticity(volume 1)[M], China: Higher Education Press, 2008.

[9] S. Z. Bai, D. C. Li, Y. M. Zhao and X. Li, "Study on the finite element analysis boundary conditions of exognathion", J Oral Maxillofac Surg, Vol.22, p. 720-723, 2006. (in Chinese)

[10] Urken ML, Buchbinder D, Weinberg H, et al., "Functional evaluation following microvascular oromandibular reconstruction of the oral cancer patient : A comparative study of reconstructed and nonreconstructed patients", Laryngoscope, Vol.101, p. 935-950, 1991.

[11] T. Ji, Y. Tie, D. M. Wang and C. P. Zhang, "Three -dimensional finite element analysis of the mandible reconstruction with fibula", West China Journal of Stomatology, Vol.27, p.143-146, 2009. (in Chinese)

[12] S. Singare, Liu Yaxiong, Li Dichen, et al., "Fabrication of customised maxillo-facial prosthesis using computer-aided design and rapid prototyping techniques", Rapid Prototyping Journal, Vol.12, p. 206–213, 2006.

[13] Ajit K M, James A D, Paul K and RobertA P, "In:Eylon D, Boyer R R, Koss D A eds", Beta Titanium Alloys in 1990's, TMS, Warrendale, PA, p. 61, 1993.