

Chapter 11

Establishment of Craniomaxillofacial Model Including Temporomandibular Joint by Means of Three-Dimensional Finite Element Method

Zhang Jun, Zhang Wen-juan, Zhao Shu-ya, Li Na, Li Tao
and Wang Xu-xia

Abstract *Objective* The aim of this study was to establish a finite element model of the craniomaxilloface with higher biological and mechanical similarities for further biomechanical study on maxillary protraction. *Methods* The head was scanned by the spiral CT to acquire two-dimensional images of the craniomaxilloface. Then the original DICOM data from CT were processed by image processing softwares such as Mimics and Magics12.0 and disposed by remesh tool in Magics12.0. Finally the three-dimensional finite element model of the craniomaxilloface was constructed by the special finite element software MSC.Marc2005. *Results* A three-dimensional finite element model of the craniomaxilloface including the mandible and the temporomandibular joint was precisely established. *Conclusions* A three-dimensional finite element model of the craniomaxilloface including the mandible and the temporomandibular joint was precisely established. The model has a high accuracy, which will be a ideal model for further biomechanical study of the craniomaxilloface. Besides, the scientific and precise modeling method was achieved.

Keywords Craniomaxilloface · Three-dimensional finite element model · Maxillary protraction · Reacting force · Temporomandibular joint

Z. Jun · Z. Shu-ya · L. Tao · W. Xu-xia (✉)
School of Stomatology, Shandong University, 250012 Jinan, China
e-mail: wxx@sdu.edu.cn

Z. Jun · W. Xu-xia
Shandong Provincial Key Laboratory of Oral Biomedicine, 250012 Jinan, China

Z. Wen-juan
Department of Stomatology, Teaching Hospital of Shandong University of Traditional Chinese Medicine, 250011 Jinan, China

L. Na
Department of Orthodontics, Shandong Provincial Qianfoshan Hospital, 250014 Jinan, China

11.1 Introduction

The finite element method (FEM) is an advanced computer technique of structural stress analysis developed in engineering mechanics, which approximates a real geometry using a large number of smaller simple elements that are connected by points called nodes. Since it was introduced to stomatology by Thresherm and Farah in 1973, [1, 2] this method had been more widely used for stress analyses of jaw bones and teeth biomechanics to evaluate stresses in every field of stomatology. In recent years, the finite element analysis were applied to all kinds of complicated mechanical problems with the rapid development of computer technology and large analysis softwares, which is now well established as an effective tool for basic research and design analysis in oral orthodontic biomechanics [3]. Element types are decided, and each element is assigned material properties to represent the physical properties of the model. The forces and boundary conditions are defined to simulate applied loads and constraint of the structure. The accuracy of the finite element model is an decisive factor in the accuracy and scientific value of he finite element analysis results to a great degree. So the aim of this study was to establish a precise finite element model of the craniomaxilloface.

The last forty years has witnessed increasing use of FEM in orthodontic biomechanics. Miyasaka et al [4] formed three dimensional finite element model of craniofacial complex composed of 2,918 nodes and 1,776 units by using dry skull as modeling materials, and carving it into 14 layer along the horizontal direction mechanical for the first time in 1986. The modeling method was very complex, time-consuming and inefficient. Mechanical cutting caused loss of bone tissue and damaged specimen. However, it was the first time to establish a relatively reliable mathematical model of craniofacial complex. In 2000, Beek [5] established three dimensional finite element model of TMJ with magnetic track instrument measuring the geometry characteristics of the articular cartilage and joint plate surface. Basciftci et al. [6] modeled three-dimensional finite element models of the mandible and the temporomandibular joint with the mesh consisting of 1572 solid elements with 5432 nodes to evaluate stress and displacements in the mandible from various chincup force vectors on the mandible. Bodo Erdmann [7] constructed three dimension finite element model of the mandible in the adaptive finite element technique by CT which copied the mandibular geometry and separated the cortical bone and cancellous bone. It was accurate and efficient.

FEM in orthodontics is mainly used to study craniofacial morphology and biomechanics. Whether the nasomaxillary complex, mandible or TMJ, orthodontic scholars have established a large number of 3D finite element models for biomechanical study. Nevertheless, craniofacial structure is very complicated, so it is hard to establish a model which is the same as the anatomy of human. Therefore, the established finite element models in orthodontics before were single-structure models. In most cases, a single-structure model may be neither accurate nor comprehensive in biomechanics analysis. There is not yet a whole skull mole

including the mandible and the temporomandibular joint. In this study, we combined the nasomaxillary complex, the mandible and TMJ to build a relatively complete skull model for future further study of the skull biomechanics.

11.2 Materials and Methods

A health young man was selected and his head was scanned by CT in order to establish a three-dimensional finite element model. The volunteer took a supine position, keeping still. SOMATOM Balance spiral CT machine supporting DICOM 3.0 standard produced by the Siemens company, was vertical to his face vertical axis and scanned the skull from up to down (scanning layer thickness 1.0 mm, layer spacing 0.5 mm, window width 2000, window level 400). Then 1,245 images of CT were received. The image data were stored in the DICOM format and directly sent into the computer.

11.2.1 Image Preprocessing

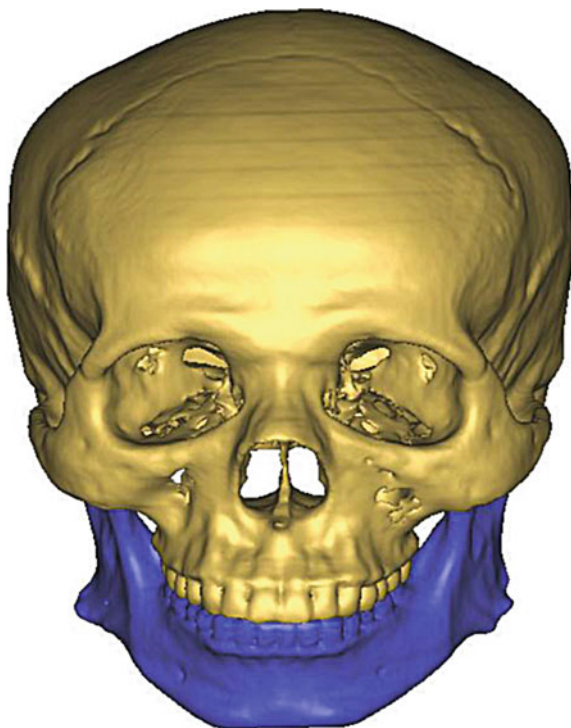
CT image is a gray image and gray value reflects its bone mineral density. In the process of 3D reconstruction we just cared about the internal and external sketches in each layer of bones, not the gray between them. A skull part without medullary cavity only had external sketch, while a part with medullary cavity had internal and external sketches. Open Mimics, select the File→Import, series of two-dimensional CT image Files were directly stored in DICOM3.0 and medical digital image communication standard were imported into this software.

Select Profile Line, and select the skull part from the upper left Front View (Front View), in whose medium we draw the buttocks, and then Profile Lines dialog popped. Click Start thresholding, select the default threshold, and go on growing. Front view spread to the whole interface and local region was enlarged.

Then image Segmentation and refining processing were achieved by the Segmentation module. The images were divided into different area characterized by different gray fields with slice tools, then the needed target area boundaries were extracted. After that, the internal and external sketches were very rough and serrated, so the dots in the sketch lines should be simplified in order that the image was more real image. Specific steps were as follows:

- Select Erase in the Edit Masks toolbar, then the joint between skull and vertebral bone would be rubbed off to separate each other.
- Use Draw to paint the edges of the skull, especially its top and bottom, and make its edges smooth. After this layer was finished, the other layers of the view were done in the same way. Then use the same method to handle the right vertical view. Figure 11.1 was a completed skull image.

Fig. 11.1 Skull CT image after segmentation and purification (The *yellow* part below is called the maxilla, while *blue* is called the mandible)



Then use 3D skull image generated by Calculate three-dimension toolbar to check the accuracy of the sketch line. According to the above the same steps the maxilla and the mandible were separated, and then 3D model of the mandible was set up. The 3D model was output by Binary STL format for further optimization.

11.2.2 Optimization of Three-Dimension Geometric Model

The 3D Binary STL format data were imported into Magics12.0 (Fig. 11.2). Because of the symmetry of the skull, half of the skull and mandible models were further optimized. Firstly, update information. There would be some useful information after selecting update, such as bad borders, bad sketches, holes, the number of shells, the covered surfaces and the crossing surfaces. Then some optimization were carried out to remove these influences in entity reconstruction. Use the corresponding tools in tool fix pages to correct errors automatically and inspect them repeatedly until shell was one and the other errors were eliminated.

In addition, we could delete something manually and create methods by triangle of the tool fix pages. When the automatic optimization could not eliminate errors

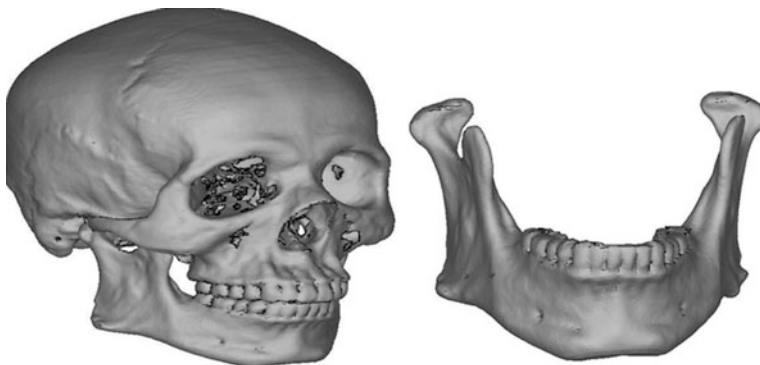


Fig. 11.2 The whole skull model and the mandible before the optimization

any more, it was still necessary to eliminate them manually. Switch display modes of the model which are shade mode and triangle model. There were still some edges in the shade mode, which meant that the triangle surface in the position was not smooth and had sharp edges. These triangle surfaces and the surrounding triangular surfaces were deleted and created to make the model more smooth. Finally the optimized model was achieved, as was shown in Fig. 11.3.

11.2.3 Establishment and Processing of 3D Finite Element Model

11.2.3.1 Generation of Finite Element Surface Mesh and Body Mesh

Select remesh in the magics to divide surface mesh in the mandible model and the whole skull model.

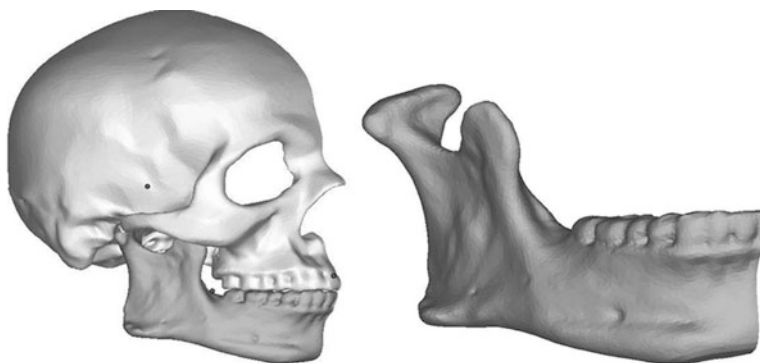


Fig. 11.3 The whole skull model and the mandible after the optimization

Select boolean to subtract the mandible model from the whole skull model after dividing surface mesh to get the maxilla model, which ensured that there were some contact area between the maxilla and the mandible and the grid node was coincident.

The finite element software MSC. Marc2005 was adopted and the geometric model was read from the CAD interface in Mentat. Do the geometry cleaning and repair before dividing mesh automatically because local modification and processing intersection or chamfering may lead to the small geometrical element when geometrically modeling in the CAD system. There were some unnecessary higher density units near these smaller geometrical elements when classifying units automatically was adopted. Besides, there were some defects in the geometric model, such as repeated dots, lines, faces, unclosed surfaces and the unmatched curves. Select geometric repair tools in Mentat, such as: CHECK GEOMETRY, ADD/REMOVE GEOMETRY, CLEAN LOOPS and so on. Clear unnecessary data above and repair the incomplete surface and the unmatched curve to ensure to generate high quality grids.

Select MESH GENERATION > AUTOMESH > SOLID MESH > SOLID TET MESH in Mentat, select all the skull and mandible surface grids and click ok to generate body grids automatically. The model after the formation of the grids was shown in Fig. 11.4.

There were node and unit numbers of the maxilla and mandible in Table 11.1 and the units are all tetrahedron entity units.

The Assumptions of Analysis Condition:

1. Materials of the model and organs were assumed to be continuous, homogeneous and isotropic elastic linear materials. Bone was a kind of anisotropic biological material. Due to too many anisotropic elastic constants, it was simplified as orthogonal anisotropy. Besides, because bone anisotropy is too weak at the same time, so it can be further simplified as isotropy.
2. The material deformation in stress was a small deformation.
3. Influences of masticatory muscle, ligament and gravity were ignored. Suppose in the relax state, the muscle force and gravity of the mandible was in equilibrium in horizontal, vertical, inside and outside directions. So remove influences of muscle,ligament and gravity to the mandible in the paper.

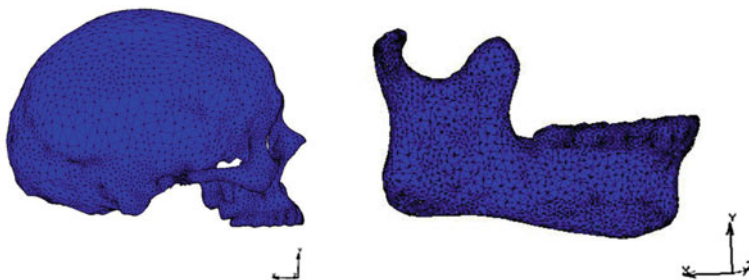


Fig. 11.4 Finite element models of the maxilla and mandible after dividing body grids

Table 11.1 The node number and unit number of three dimensional finite element model

Position	Node number	Unit number
Maxilla	2,0387	7,6892
Mandible	8,353	3,3878

11.2.3.2 The Boundary Conditions and Definition of Material Characteristics

Return to the main menu MAIN, go to the boundary condition submenu BOUNDARY CONDITIONS, and select the type of boundary condition to be the stress boundary condition MECHANICAL and the boundary condition to be a given displacement FIXED DISPLACEMENT. Constraint the displacement of Z direction to be zero and select NODES-ADD, then take all nodes on the symmetry plane of the skull to constraint and confirm. Select FACE FOUNDATION to boundary condition, fill in 1000 in stiffness STIFNESS and choose some mesh surface in the top of the skull to constraint and confirm. Then according to the experiment, something could be applied to the model in the load test (Fig. 11.5).

The material property parameters in the analysis were shown in Table 11.2.

From the main menu into material characteristics definition submenu MATERIAL PROPERTIES, select the type of isotropy ISOTROPIC, define the elastic modulus and poisson’s ratio of the maxilla and the mandible according to the parameter in Table 11.2 respectively and assign the MATERIAL PROPERTIES to the corresponding unit.

Fig. 11.5 The diagram of boundary condition (The angle between force and occlusal surface was 25 and the force was 3 N)

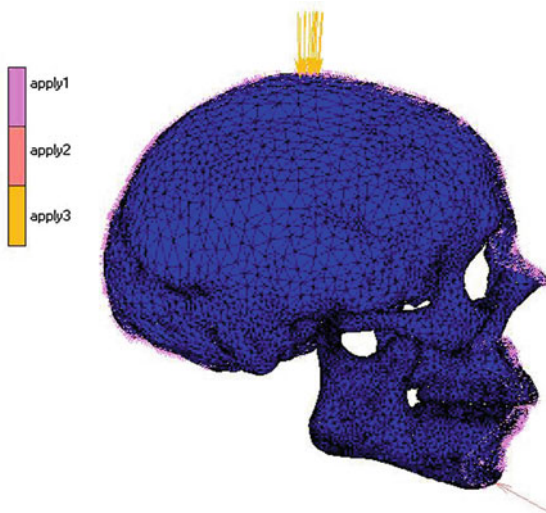


Table 11.2 Material property parameters

Materials	Elastic modulus(GPa)	Poisson's ratio μ
Maxilla	13.7	0.3
Mandible	13.7	0.3

11.3 Results

The model of human craniomaxilloface containing TMJ was successfully established, which had a high geometric similarity and complete structure. In modeling, the dots in the image contour lines were simplified to make it more real by Mimics. After being optimized by the Magics12.0, the finite element body grids would be generated by MSC. Marc2005. In this model, the maxilla was structured by 20387 nodes and 76892 units, while the mandible was structured by 8353 nodes 33878 units. The shape of the finite element model was very good, and the structure was very similar to the biological entity in geometry. The model materials and organs were assumed to be continuous, homogeneous and isotropic linear elastic and that the materials had a small deformation when stressed. Definite the elastic modulus and poisson's ratio of the mandibular according to the previous researches. Set the boundary constraint condition and the mechanical loading according to the need to mimic the stress and strain of the maxilla and the mandible when people were on the maxillary protraction in the clinical treatment.

11.4 Discussion

The finite element modeling technique is an alternative, indirect mathematical approach. Once the finite element model was established, the stress and strain of the object could be analyzed in different loading conditions, which was more comprehensive, and more accurate than other methods. [8] The method is now well established as a tool for basic research and design analysis in oral biomechanics.

There are about four modeling methods at present. ①Section/slice method. The objects was preprocessed and cut according to certain requirements, and cutting sections were scanned to obtain information for a finite element model. But this method is damage to the object and low accuracy. Therefore this method was less applied. ②3D measurement method. It acquired 3D information by contact or non-contact scanning or holography [9]. Although it can be used to study the object with a complicated morphology and has high accuracy, it need complex softwares and hardwares to support and cannot study interior structure of the object. Hence it is limited. ③Image processing method. The information from CT or MRI are processed by scanning or photography technology and put into the computer. Then they are treated by the related software to establish the model. Compared with the former two methods, this method is no damage to the object, suitable for the living,

simple and highly accurate. So it is usually used in the oral biomechanics research.④DICOM data modeling method. It is simpler and more precise and without some subjective factors. This method is used most in finite element modeling of oral biomechanical study, the nature of which is to process images. In this research, we adapted the last one to establish 3D model of the skull.

Class III malocclusion is one of commonly oral clinical disease. As for class III malocclusion caused by maxillary retrusion, one of the best treatments is maxillary protraction. The active force of maxillary protraction is on the maxilla, while reaction force is on the forehead and chin. So the craniofacial skeleton is subjected to complex loading and it would be difficult to assess the mechanical reaction to complex loading in 3D space by using conventional methods, such as strain gauge [10], photoelastic [11] or holographic techniques. FEM would be an effective way to study the biomechanics of craniofacial skeleton in 3D. However, the construction of 3D finite element models of craniofacial complex is quite complicated and still in a confused state. On the early days, some scholars had established finite element models of mandible or TMJ in the section method or CT [12]. Palomar et al. [13] established a finite element model of TMJ by CT and MRI. However, these models are all single-structure. In this research, we established a relatively complete and precise 3D model, of which TMJ and the mandible were included. The model with 20,387 nodes and 76,892 units in the maxilla, and 8,353 nodes and 33,878 units in the mandible helps to go on the further biomechanical study of maxillary protraction. It is more complete than the single-structure model, which will evaluate the stress and strain of the active force and reaction force in the maxillary protraction.

Acknowledgments It was Supported by Shandong Province Natural Science Foundation(Grant No. ZR2011HM036), Science and Technology Development Program of Medicine and Health of Shandong Province(Grant No. 2011QZ023) and Universities and Institutes Indigenous Innovation Program of Jinan(Grant No. 201202032).

References

1. Thresher RW, Saito GE (1973) The stress analysis of human teeth. *J Biomech* 6(5):443–449
2. Farah JW (1973) Photo-elastic and finite element stress analysis of a restored axisymmetric first molar. *J Biomech* 6(5):551—520
3. Fernandez JR, Gallas M, Burguera JM, Viano A (2003) A three-dimensional numerical simulation of mandible fracture reduction with screwed miniplates. *J Biomech* 36:329–337
4. Miyasaka J, Tanne K, Tsutsumi Set al (1986) Finite element analysis of the biomechanical effects of orthopedic forces on the craniofacial skeleton: construction of a three-dimensional finite element model of the craniofacial skeleton. *Osaka Daigaku Shigaka Zasshi* 31(2):393–402
5. Beek M, Koolstra JH, van Ruijven LJ, van Eijden TM (2000) Three-dimensional finite element analysis of the human temporomandibular joint disc. *J Biomech* 33(3):307–316
6. Basciftci FA, Korkmaz HH ,Serdarüsümez et al (2008) Biomechanical evaluation of chin cup treatment with various force vectors. *Am J Orthod Dentofacial Orthop* 134(6):773–781

7. Erdmann B, Kober C, Lang J et al (2001) Efficient and reliable finite element methods for simulation of the human mandible. Berlin, pp 1–15
8. Rick-Williamson LJ, Fotos PG, Coel VK (1995) A three dimensional finite-element stress analysis of an endodontically prepared maxillary central incisor. *J Endod* 21:362–367
9. Pavlin D, Vukicevic D (1984) Mechanical reactions of facial skeleton to maxillary expansion determined by laser holography. *AM J ORTHOD* 85:498–507
10. Tanne K, Miyasaka J, Yamagata Y et al (1985) Biomechanical changes in the craniofacial skeleton by the rapid expansion appliance. *J Osaka Univ Dent Soc* 30:345–356
11. Chaconas SJ, Caputo AA (1982) Observation of orthopedic force distribution produced by maxillary orthodontic appliances. *Am J of Orthod* 82:492–501
12. Tanaka E, Tanne K, Sakudam A (1994) A three-dimensional finite element model of the mandible including the TMJ and its application to stress analysis in the TMJ during clenching. *Med Eng Phys* 16:316
13. Perez Del Palomar A, Doblare M (2006) Finite element analysis of the temporomandibular joint during lateral excursions of the mandible. *J Biomech* 39(12):2153–2163