CONSIDERATIONS UPON CONTACT STRESS MODELLING IN DENTAL ARTICULATOR PAIRS

Lecturer Florina Carmen **CIORNEI**, Suceava University, *florina@fim.usv.ro*
Associate Professor Stelian **ALACI**, Suceava University, *stalaci@yahoo.com*
Professor Gheorghe **FRUNZĂ**, Suceava University, *frunza@fim.usv.ro*
Lecturer Cătălina **CIORNEI**, "Carol Davila" University of Medicine and Pharmacy, Bucureşti
Assistant Ovidiu **RUSU**, Suceava University, *rusu.o@fim.usv.ro*
Associate Professor, Delia-Aurora **CERLINCĂ**, Suceava University, *delia@fim.usv.ro*

Abstract:*A dental articulator is a mechanism which simulates the temporo-mandibular joint. The articulator is essential as it replicates the basic motions of the upper and lower mandibles, both revolve and translational motions. In the present paper the stresses from an articulator TMJ modelled as a bronze sphere into a cylindrical steel cavity are analyzed by two methods, first applying the Hertzian contact theory and then numerically, by means of finite element analysis using the simulation module in CATIA.*

Keywords: stress, dental articulator, prosthodontic;

1. INTRODUCTION

An articulator is a mechanical device which simulates the temporo-mandibular joint *(TMJ)* and jaw members to which maxillary and mandibular casts may be attached to represent jaw movements. The articulator is important because it replicates the basic pivot action of the upper and lower mandibles, as well as translational motions, Fig.1. Casts of the maxillary and mandibular teeth are fixed to an articulatorand reproduces recorded positions of the mandible in relation to the maxilla. The entire assembly attempts to reproduce the movements of the mandible and the various intercuspidian relationships that accompany those movements, [1].

A dental articulator assists in the fabrication of removable prosthodontic appliances, fixed prosthodontic restorations and orthodontic appliances. In order to reproduce the individual parameters of the patient the articulator must be adjustable, Fig. 2. The settings are measured on the patient and using a face bow, the relative location of the occlusal plane is transferred from the patient to the mechanical dental articulator, Fig.1, [1], [2].

Dental articulators are used for more than 150 years, [2], [3], [4], Fig. 3., and different types of articulators were constructed, such as: semi-adjustable articulator, anatomical articulator, Adjustable articulator, Disposable articulators.

The articulator structures contain upper and lower bodies, and the TMJ-s, which are the most important part of the articulators, present a great variety of configurations. In the present paper the stresses from an articulator TMJ modelled as a sphere into a spherical/cylindrical cavity are analyzed by two methods, first applying the Hertzian contact theory and secondly using a FEA simulation.

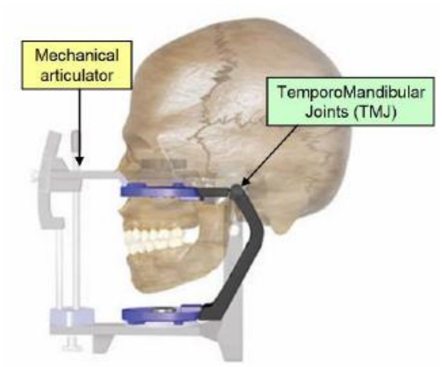


Fig.1. Temporo-Mandibular Joint and principle of dental articulator, [2]

Fig.2. Yamahachi OHTAK 1dental articulator, [5]

S. S. White Articulator, Henry Coy, 1862

"Independent Motion" Articulator,T.G. Lewis, late 1860's - 1890's

Fig.3. Early dental mechanical articulators, [3]

2. STUDIED *TMJ* MODEL

The studied model, from a Yamahachi dental articulator, [5], Fig.2., that is a standard arcon type articulator with a ball slot type path, designed in CATIA, is presented in Fig. 4 and consists of a sphere (1), guided by an unfigured pin, placed into a spherical/cylindrical groove, (2) and pressed onto it in the necessary position by a screw (3). The mandible does not act like a simple pivot but it rotates around three reciprocal orthogonal axes. Recent researches studied and modelled the human jaw and chewing behaviour, [6], [7], [8], [9].

A model should achieve, in terms of kinematics, the human-like chewing movements. For a spatial mechanism where at least one of the elements presents a general spatial motion, the mobility can be obtained using Kutzbach's criterion, [10]:

$$M = 6(n-1) - \sum_{k=1}^{5} k \cdot c_k \quad (1)$$

where*n* is the total number of elements (including the ground), c_k is the number of joints restrictioning*k* elementary motions and *k* is the number of c_k joints.

For the studied TMJ model, $n = 4$, namely: the ground, the spheres and the mobile jaw; $c_2 = 2$ between the spheres and channel and $c_4 = 2$ between spheres hole and pins an it results:

$$M = 6(4-1) - (2 \cdot 2) - (2 \cdot 4) = 6 \quad (2)$$

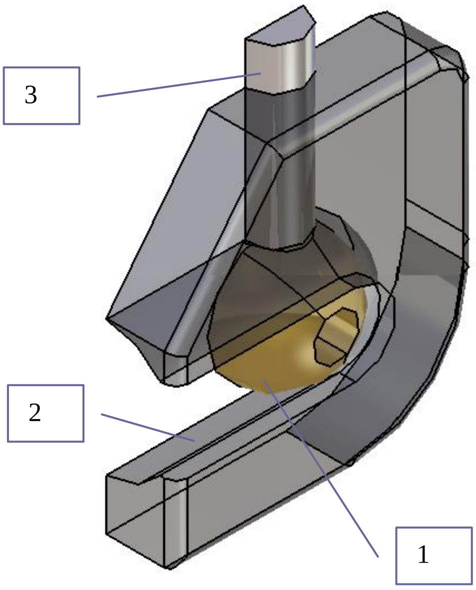


Fig.4. Model of a dental articulator TMJ

Therefore, the model presents 6 degrees of freedom, likewise the real *TMJ*, ensuring a complete positioning of the cast from the mobile jaw.

Assuming that the sphere and the groove satisfy the hypothesis shown by Johnson, [12], the contacts from *TMJ* model can be described by Hertzian theory and are identified as:

- Sphere and spherical cavity, for a symmetric anatomical position;
- Sphere-cylindrical cavity, for asymmetric lower jaw anatomy;
- Sphere-flat screw end;
- Cylindrical shaft-inner ball hole.

In the present work, the case of asymmetrical position of the mandible is studied and the following simplifying assumptions were made, in order to avoid too complex computations in FEA:

- the mandible is in a asymmetrical position, therefore the contact between the sphere and the groove occurs in the cylindrical cavity;
- the screw end has a concave surface conforming the sphere and the normal force is applied on the flat top of the screw, as a distributed pressure;
- the linear contact between the pin and the sphere was not considered;

the finite element analysis was performed only for one-half of the model with the purpose to reduce the computing time and better allocate the computer resources as the symmetrical loading and designing geometry allows it.

3. RESULTS

The contact of main interest is the sphere-groove contact and it is studied both by analytical method and numerical methods. First, a Mathcad program was used for computing contact parameters of sphere-cylindrical cavity with different elastic characteristics in normal contact, using the relations from classical Hertzian theory. The geometry of contact is determined by the radii of curvature for sphere, $R_{1x} = R_{1y}$ and radii of curvature for the groove,

$R_{2x} = \infty, R_{2y} < 0$ for cylinder. The contact rigidity depends on the elastic characteristics of contacting materials. The contact geometry in the studied *TMJ* model is given by the radius of sphere and the radius of the cylindrical cavity: $R_s = 5.7\,mm;\ R_c = 5.78\,mm$, respectively. The Young moduli of the materials for bronze sphere and steel channel are $E_s = 1.1 \cdot 10^{11}\,N/m^2;\ E_c = 2 \cdot 10^{11}\,N/m^2$ and the Poisson coefficients are $\nu_s = 0.341;\ \nu_c = 0.266$ respectively. In the human chewing process, the maximum forces vary, but can attain even 652N. Here, a regular load was considered, $Q = 9.8\,N$. The results obtained applying the relations from theory of elastic contact, [11] are given in ***Table 1***.

A strong eccentricity of elliptical contact area is obtained.

Table 1. *Results obtained applying analytical relations from Herzian theory*

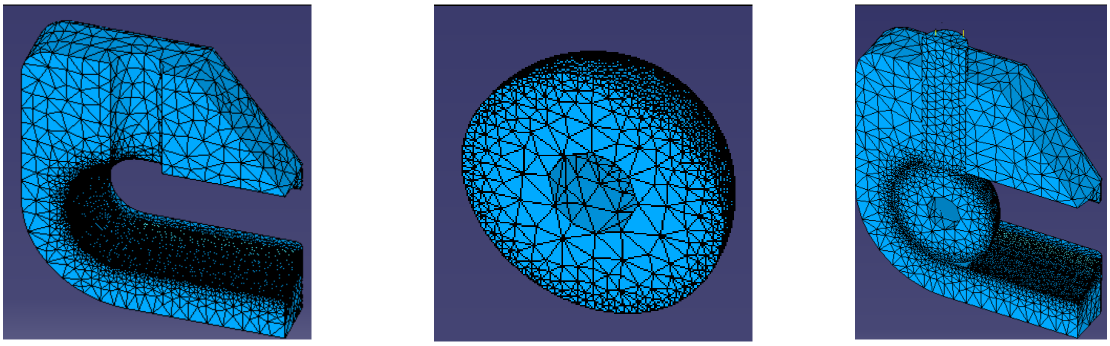
Contact parameter	***Symbol and relation***	***Sphere in cylindrical groove***
maximum contact pressure	$p_0 = n_p \left[\frac{3}{2} \left(\frac{k}{\eta} \right)^2 Q \right]^{1/3}$	247.2 *MPa*
contact area major one-half axis	$a = n_a \left(\frac{3}{2} \eta \frac{Q}{k} \right)^{1/3}$	$0.534\,mm$
contact area minor one-half axis	$b = n_b \left(\frac{3}{2} \eta \frac{Q}{k} \right)^{1/3}$	$0.036\,mm$
normal approach	$\delta = \frac{n_\delta}{2} \left(\frac{9}{4} \eta^2 Q^2 k \right)^{1/3}$	$4.565 \cdot 10^{-4}\,mm$
contact area eccentricity, *e*	$F(e) = (1-e^2)\frac{D(e)}{K(e)-D(e)} - \frac{A}{B} = 0$	0.998

The results obtained by finite element analysis were found using Generative Structural Analysis module from CATIA software package developed by Dassault Systems and they are presented in the next figures. The refined meshing can be observed in the vicinity of contacting surfaces for components of *TMJ* and for the assembly, in Fig. 5.

The elements of the mesh are parabolic tetrahedrons. The contact pressure obtained by FEA is shown in Fig. 6 for sphere with mesh visualisation and in Fig. 7 the equivalent von Mises stresses in sphere are seen. The finite element analysis performed in CATIA gives the tensor components for both stress and strain and for exemplifying, the strains ε_z for ball and seat are presented in Fig. 8 and Fig. 9, respectively.

In Fig. 10.a, the contact pressure in the seat is shown and, as at a first estimation, the maximum values were different for sphere and groove, a detail was taken, Fig. 10.b. It revealed strong variation of pressure values on the same mesh element in the vicinity of initial contact, due to coarse meshing. Detail of FEA contact pressures in assembly is presented in Fig. 11, and it is obvious that the meshing element size is comparable to the contact area major one-half axis but much greater than the minor one-half axis.

The equivalent von Mises stresses are presented for the two parts in Fig. 12, in a section made with a plane normal to the channel axis, for a more complete visualisation of stresses and trying to highlight the other strongly stressed regions from the seat.



Seat *Sphere* *Assembly*

Fig.5. Refined meshing on contacting surfaces from TMJ model

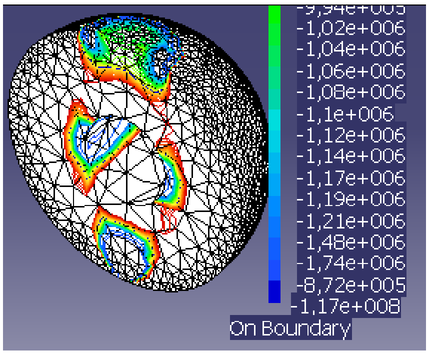


Fig.6. FEA contact pressures in sphere with mesh visualisation

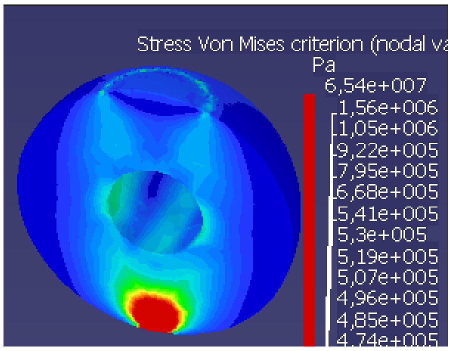


Fig.7. FEA von Mises equivalent stresses in sphere

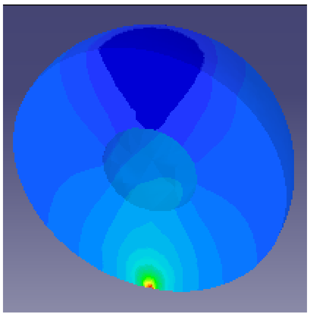


Fig.8. FEA ε_z strains in sphere

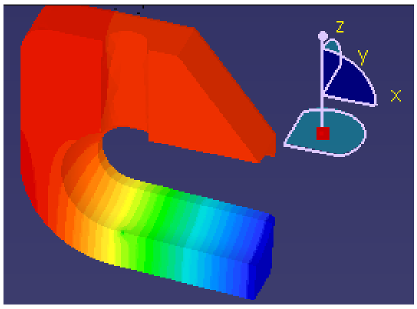


Fig.9. FEA ε_z strains in seat

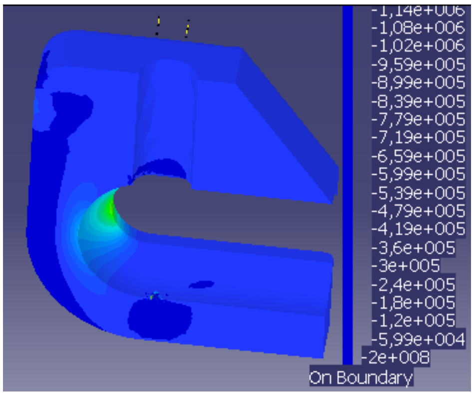


Fig.10a. FEA contact pressures in seat

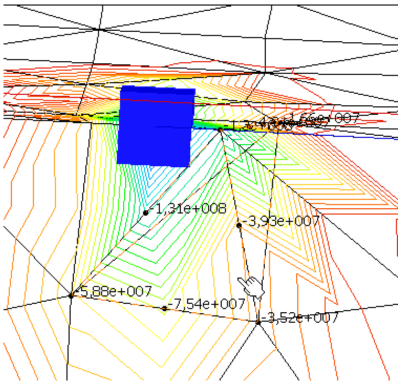


Fig.10.b. Detail revealing strong variation of pressure values in the vicinity of initial contact

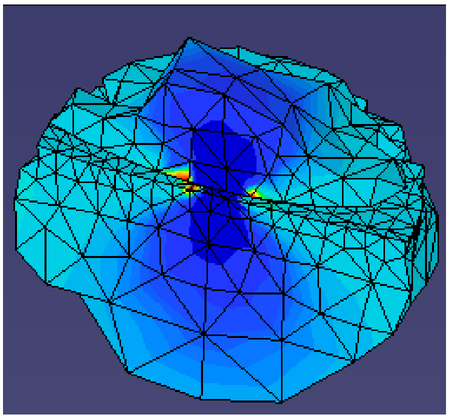


Fig.11. Detail of FEA contact pressures in assembly

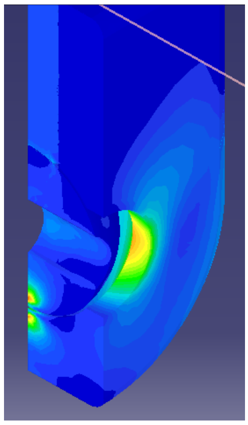


Fig.12. Von Mises stresses in assembly sections

From the FEA analysis, the maximum Hertzian pressure was $p_{0FEA} = 211 MPa$ and the error computed for the maximum contact pressure, compared to Hertzian $p_0 = 247.2\ MPa$ was found of 14.57%. This is considered a good error value, allowing for the rather coarse meshing performed. The element was chosen parabolic and the dimension of the local mesh size is *0.3mm*, having thus the same magnitude order as the contact area. A parabolic meshing element and a refined meshing offer more precise results, [12], [4]. The maximum contact pressure from the cavity must be very carefully analyzed. The FEA program shows the extreme stress, but this value is obtained at a detailed analysis and view. The maximum contact pressures have in fact the same values in sphere and in the channel. The major advantage provided by finite element analysis is highlighted, as it provides the stress and strain tensor components and the equivalent von Mises stress, for both contacting bodies, from a unique analysis.

4. CONCLUSIONS

Contact stresses from a modelled articulator *TMJ* were studied by classical Hertzian theory and by finite element analysis. In a model *TMJ* occur several Hertzian contacts but in the present paper only the sphere in cylindrical cavity was solved. A precise analysis assumes considering only the neighbourhood around contact, in order to allow a fine meshing.

The values for contact parameters obtained applying Hertzian theory were quickly found, using Mathcad. The FEA maximum contact pressure from sphere-cylindrical channel contact validates the Hertzian contact pressure found applying analytical relations. Once the analysis made with finite element method in CATIA, the stress and strain fields from the contacting bodies are visualised rapidly, for stress tensor and strain tensor components or for equivalent von Mises stresses and can provide a general image. The FEA method is useful for a preliminary step in design for establishing the most stressed regions of the parts from the assembly. An additional analysis is required in the local region, for contact stress analysis as the manner to diminish the size of the mesh element is not a recommendable technique.

REFERENCES

[1] **Burlui V.**,s.a. *"Clinica si terapia edentatiei partiale reduse"* tipografia Alfa Centaur, Apollonia 2001;

[2] **Solaberrieta E., Etxaniz, o., Minguez., R., Muniozguren, J., Arias, A.**, *Design of a virtual articulator for the simulation and analysis of mandibular movements in dental CAD/CAM*, Proc. 19th CIRP Design Conference-Competitive Design, Cranfield University, 2009, 323

[3] ***http://www.db.uth.tmc.edu/Articulators/Lewis_independant_motion/index.htm

[4] **Ciornei F., S. Alaci, I. Muscă, Gh. Frunză, L. Irimescu, D. Cerlincă**, *Some aspects concerning contact stresses from a dental articulator joint*, Varehd 15 Conference, Suceava University, 6-8 May 2010;

[5] ***http://www.yamahachi-dental.co.jp/en/products/08articulator/01correct_articulator/

[6] **Daumas, B., Xu., W.L., Bronlund, J**., *Jaw mechanism modeling and simulation*, Mech. Mach. Theory, 40, 2005, 821-823;

[7] **Koolstra J.H., van Eijden T.M.G.J**., *Combined finite element and rigid body analysis of human jaw joint dynamics*, Journ. Biomechanics, 38, 2005, 2431-2439;

[8] **Hannam, A.G., Stavness I., Lloyd J.E., Fels S**., *A dynamic model of jaw and hyoid biomechanics during chewing*, J. Biomechanics, 41, 2008, 1069-1076;

[9] **Xu, WL, Lewis D, Bronlund JE, Morgenstern MP,***Mechanism, design and motion control of a linkage chewing device for food evaluation*, Mech. Mach. Theory, 43, 2008, 376-389;

[10] **Uiker J.J. jr., Pennock G.R., Shigley J.E**., *Theory of machines and mechanisms*, Oxford University Press, New York 2003;

[11] **Johnson, K.L**., *Contact Mechanics*, CambridgeUniversity Press, 1985;

[12] **Zienkiewicz O.C., Taylor, R.L.**, *The Finite Element Method for Solid and Structural Mechanics*, Butterworth-Heinemann, 2005;