Acta Polytechnica **58**(4):259–263, 2018
doi:10.14311/AP.2018.58.0259

© Czech Technical University in Prague, 2018
available online at http://ojs.cvut.cz/ojs/index.php/ap

NUMERICAL SIMULATIONS OF STATIC STRENGTH OF NOVEL DENTAL IMPLANTS

Luboš Řehounek*, Aleš Jíra, František Denk

Czech Technical University in Prague, Faculty of Civil Engineering, Thákurova 7 166 29, Prague, Czech Republic

* corresponding author: lubos.rehounek@fsv.cvut.cz

Abstract. It is important to know the mechanical and micromechanical characteristics of conventional dental implants in order to design and develop novel dental implants and surface treatments that ensure a good biomechanical stability and almost fully replace the biological tissue. A successful integration of the implant depends on its chemical, mechanical and physical properties and the quality of the bone tissue. Considering these facts, we developed two novel variants of dental implant stems with an anti-rotational geometrical solution. On these variants, we have then performed numerical analyses of static strength. Outcomes of simulations vary depending on the type of the implant and load mode.

Keywords: dental implant; numerical; static strength; osseointegration; FEM.

1. Introduction

Every successfully implanted prosthetic has to be accepted by the body of the patient. This is a process we call osseointegration – the full functional and structural connection between the living tissue and an implant [1]. Many implants fail to achieve osseointegration because of an early failure [2–4]. The extent of osseointegration depends on many factors, such as the chemical, mechanical and physical properties of the implant, bone-implant bonding, cytotoxicity or the implant geometry. This article will focus on the implant geometry and simulations of static strength of two novel dental implants that were patented in the Czech Republic [5, 6].

These novel dental implants are shown in Fig. 1. The first implant is the 0001 "Four leaf clover" variant. It has a system of stabilizing ribs that resembles a four leaf clover in its cross-section. These ribs connect into a half-spherical root ending and serve to increase the torsional stability of the implant. The second variant is the 0002 "ribbed" variant. Its cylindrical shape transforms into a conical part, which is equipped with a system of vertical stabilizing ribs. There is a small hollow chamber situated at the intersection of the beams that serves as a free space for human bone to grow into. The vertical ribs coupled together with the transversally-oriented chamber provide a good vertical and torsional stability. The vertical ribs run parallel to each other and they form a 90° angle. The lower intraosseous part of the implant is comprised of a tapering conical surface with the stabilizing ribs. This part is especially important in a replacement of front teeth as there is usually not enough space to introduce a cylindrical implant.

Both implants offer greater torsional stability and much greater bonding surface for osseointegration than conventional implants, which is beneficial in the case of front teeth or in an applications where there is generally not enough space (e.g., in between two roots of adjacent teeth).

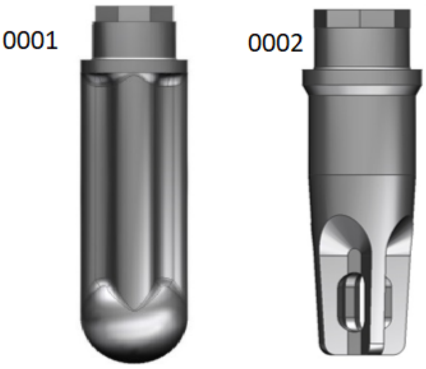

Figure 1. Two novel dental implant types, the "four leaf clover" variant (left) and the "ribbed" variant (right). The four leaf clover implant variant has a half-spherical ending and four vertical ribs running through the whole body of the implant. The ribbed implant has four vertical ribs situated in the conical intraosseous part of the stem as well as a small hollow chamber. Both implants belong to the "push-in" category of dental implants.

Another two variants of the 0002 implant have been developed, both using a half-spherical ending. One of them has a parallel system of vertical ribs and the second has a system of slant ribs (Fig. 2).

2. Numerical Analyses

The main goal of the presented numerical analyses is the evaluation of the intraosseous parts of dental implants in regard to their ability to withstand a mechanical load. For these analyses, we chose two novel patented dental implant types (type 0001, the "four leaf clover" variant and type 0002, the "ribbed" variant). For the sake of comparison, we also analysed a

FIGURE 2. Other versions of the ribbed dental implant. One with a system of slant stabilizing ribs (left) and one with a system of parallel stabilizing ribs (right). These implant variants were not considered in the numerical analysis at this point in time.

one conventionally manufactured dental implant, type ProSpon ZV14112-3200-00008. All analysed implants are from the "push-in" category of dental implants.

The analyses focus on evaluating the static strength of the implants. This simulation and evaluation was performed using the ČSN EN ISO 14801 – "Dentistry – Implants – Dynamic loading test for endosseous dental implants" standard [7]. The goal of this analysis was to determine the stress and strain response of the novel dental implants and compare them with a real approved implant (ProSpon ZV14112-3200-00008), which conforms the given standard. The analyses will provide the determination of critical locations where stresses concentrate along with viable design changes that could potentially benefit the stress distributions.

The main observed attribute of the analysed intraosseous parts of novel dental implant stems is their distribution of stress and strain. The environment used for the simulations was ANSYS APDL.

3. CREATING FEM MODELS

3.1. DENTAL IMPLANTS GEOMETRY

The geometrical models of dental implants 0001, 0002 (novel implants) and implant ZV14112-3200-00008 (reference implant) were created using the ANSYS APDL environment. The foundation for these geometries was a precise project documentation provided in PDF and DXF formats by ProSpon spol. s r.o. The outer surfaces of the implants are created in detail, whereas the inner parts are created without a great attention to detail (e.g., without the screw-thread).

3.2. MESH

We used the SOLID 187 quadratic tetrahedrons generated in the "smart size" mode as mesh elements. Every single element has 10 nodes situated in its peaks and in half of every edge. The numerical analysis assumes a linear elastic behaviour of all materials. The structure of all materials was defined as homogeneous and isotropic. Individual material properties were specified for each material, but left out of this paper for the sake of brevity.

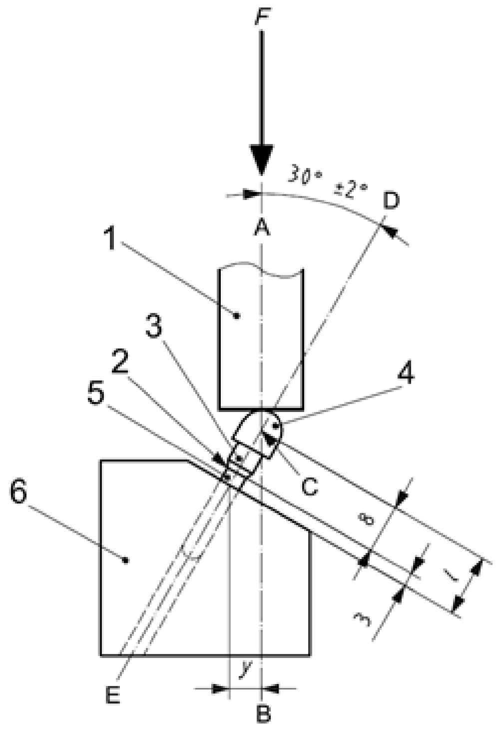

FIGURE 3. A schematic of the testing assembly according to the used standard. 1 – loading head, 2 – nominal surface level of bone tissue, 3 – connecting part, 4 – hemispherical loading part, 5 – dental implant stem, 6 – fixation device.

3.3. NUMERICAL SIMULATION AND LOADS

The loads described in this subsection are specified in the ČSN EN ISO 14801 – "Dentistry – Implants – Dynamic loading test for endosseous dental implants" standard [7]. The test set comprises of the dental implant stem, a pillar with a loading head, connecting screw M 2.2 with an inner hexagon and a device for sample anchoring (Fig. 3)

The ANSYS APDL simulation represented testing the anchored specimen with a static load. The stem was equipped with a proper superstructure. The anchoring device is represented by a hollow cylindrical body of an outer diameter of 8.0 mm and a height of 18.0 mm. The outer surface of the cylindrical part has a defined boundary condition that eliminates its displacement in all directions ($u_x = u_y = u_z = 0$). The superstructure is set on the upper hexagon of the specimen. It is represented by a full cylinder with a diameter of 3.8 mm and 4.0 mm and a height of 4.0 mm. The superstructure's surface is defined by the shape of the implant. The interconnection of these bodies is attained by using rigid constraints generated by volumetric "vglue" operations.

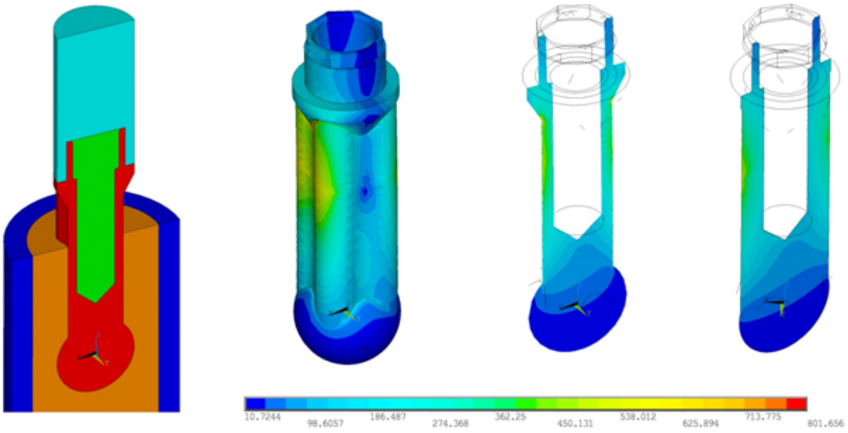

Figure 4. Equivalent von Mises stress distribution (MPa) for the 0001 implant variant. Whole body and 2 different load modes.

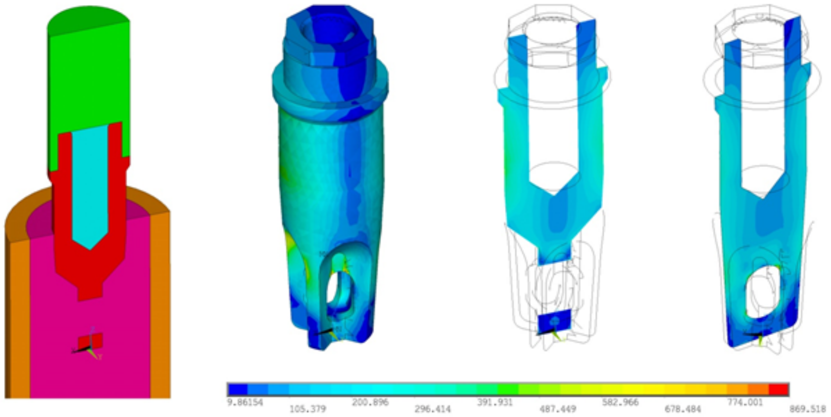

Figure 5. Equivalent von Mises stress distribution (MPa) for the 0002 implant variant. Whole body and 2 different load modes.

The load is applied by a single static force located in the centrepoint of the upper base of the cylindrical superstructure. The load magnitude was set as $F = 350\,\mathrm{N}$. This force is applied with a 30° deviation. This magnitude of force was chosen because it corresponds with the static failure of the implant during loading in the mode of controlled deformation. The whole assembly was investigated in two load modes – in the plane of the centreline of the ribs and in the vertical plane running in between them. The outcomes of the analysis are shown in Tab. 1.

4. Results and Discussion

In the following figures, we show the stress distributions among different dental implants. We chose to use Von Mises stress as it is the most widely used criterion for analysing ductile materials, such as metal.

4.1. The 0001 Variant Results

The analysis was carried out in two different planes in regard to the orientation of the system of stabilizing ribs and the applied load. It was found that the stem loaded in the plane of the ribs exhibits a better mechanical response. The extreme values of stress are approximately 12 % lower while the displacement values are almost the same for both load modes. From this fact, we can deduce that a dental implant stem with more than four ribs would have a superior mechanical response.

Concentrations of stresses occur mainly at places of sudden changes of a curvature, geometrical inhomogeneities and at the area of the transition of the implant into the fixation device (Fig. 4). This shortcoming can be accounted for by rounding all sharp edges in the geometry by a radius of at least 0.5 mm.

The comparison showed that the extreme values of stress are approximately 35 % lower than those of the ProSpon ZV14112-3200-00008 reference implant.

The tested implant variant 0001 is able to withstand load values up to 350 N without any plasticity in the body of the stem. Maximum values of stress are 801.656 MPa, which is approximately 95 % of the yield strength ($f_k = 850.000\,\mathrm{MPa}$). Further attention will be directed towards determining the ultimate bearing capacity with an elastoplastic model. Also, a numerical model of the abutment and the connecting screw will be created to realistically simulate the connection to the stem.

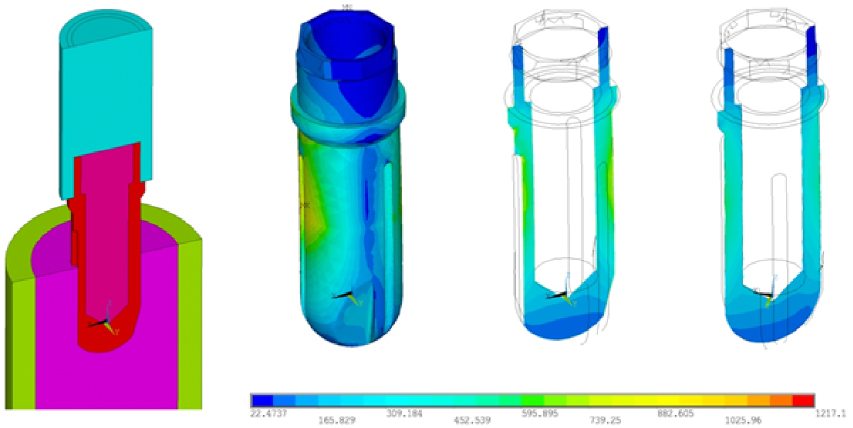

Figure 6. Equivalent von Mises stress distribution (MPa) for the ZV14112-3200-00008 implant variant. Whole body and 2 different load modes.

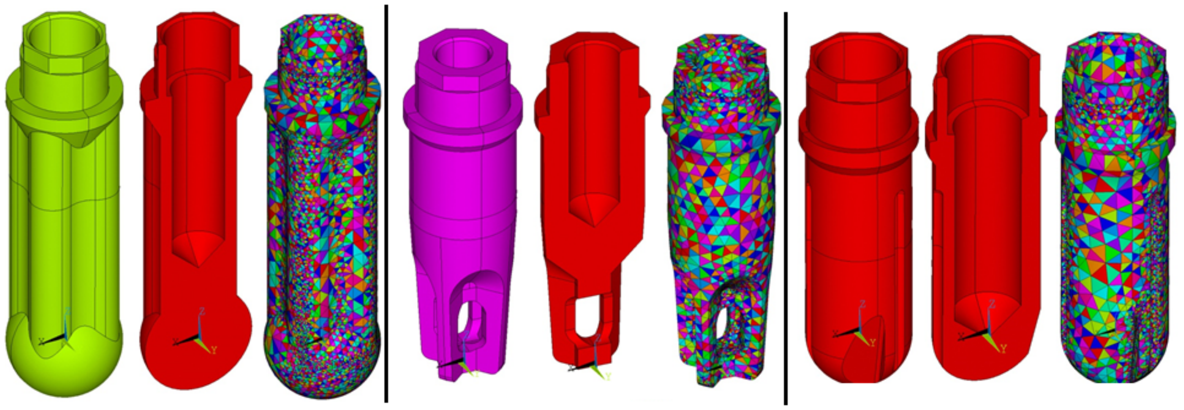

Figure 7. Three trios of models of individual dental implants. Left – the "four leaf clover" variant, middle – the "ribbed" variant, right – the reference implant. Images in trios from left to right – whole 3D model, longitudinal section, FEM mesh.

4.2. The 0002 Variant Results

The analysis was carried out in two different load modes with regard to the placement of the stabilizing ribs and the applied force. Similarly to implant 0001, the stem of the implant variant 0002 exhibits lower values of stress (approximately 11 % lower) when tested in the plane of the stabilizing ribs. It is, therefore, assumed that the stress distribution will be more favourable for the direction of the force in the plane of the stabilizing ribs.

The analysed stem is not thin, but rather bulky. Despite this fact, the analysis showed that greater values of stress occur at the upper part of the stem in comparison to the variant 0001 (8 % greater values of maximum stress). The displacement values are approximately 8 % lower.

When compared to the ProSpon ZV14112-3200-00008 reference implant, the implant 0002 has lower extreme values of stress (approximately by 29 %).

Concentrations of stresses in the intraosseous parts of the stem occur at the root areas of the stem, at the stabilizing ribs and at the location of their anchoring into the cylindrical part. The stress concentrations occur also at the walls of the transversal chambers (Fig. 5). This is a fact that could potentially negatively affect osseointegration and bone ingrowth. The next development will be directed towards an optimization of the shape and length of the stabilizing ribs, rounding the sharp edges by a radius of at least 0.5 mm and adjusting the root with regard to an optimal distribution of vertical forces into the tissue of a cancellous bone.

The tested implant variant 0002 is not able to withstand a load of 350 N without any plasticity. Plasticity is expected to be present at the area of anchoring of the stabilizing ribs into the cylindrical part of the intraosseous stem. Maximum stress in this area reaches values of 869.518 MPa, which is approximately 102 % of the yield strength ($f_k = 850.000$ MPa) and 91 % of the ultimate strength ($f_p = 950.000$ MPa). The next development in verifying the mechanical behaviour of this implant will be directed towards determining the values of the ultimate bearing capacity with an assumption of the elastoplastic behaviour in the body of the implant stem. As for the implant variant 0001, a numerical model of the abutment and the connecting screw will be created to realistically simulate the connection to the stem.

Figure 8. All manufactured implant variants. Variant with a conical shape and ribs (left), variant with a system of slant ribs (middle) and variant with a parallel system of ribs (right). All implants have been manufactured from the Ti6Al4V ELI alloy and have a diameter of 3.8mm and length of 8 mm (the most conventionally used size).

Tested implant	Deformation of the implant head u_{imp} [µm]	Deformation of the superstructure head u_{c} [µm]	Equivalent von-Mises stress σ_{eqv} [MPa] min σ_{eqv}	max σ_{eqv}
Implant 0001	104	215	10.724	801.656
	104	213	10.964	703.792
Implant 0002	95	190	9.862	869.518
	95	188	9.121	777.285
Reference implant	180	361	22.474	1217.100
ZV14112-3200-00008	180	357	22.249	1166.960

Table 1. An overview of the results of the numerical analysis for all tested dental implants.

5. Conclusions

The outcomes of the numerical analysis provide an insight into the stress distribution of two novel dental implants. Another conventionally manufactured reference dental implant was tested to compare the obtained results. The main investigated properties are the stress distribution, strain distribution and locations of their concentration. All values mentioned in the subsections of this section are listed in Tab. 1.

It was found that the implant variant 0001 has a better overall mechanical response (1). Although the differences between variants 0001 and 0002 are small, it was proven to be a better variant of the two. By modifying the geometry of the implant, we were able to reduce maximum values of stress by up to 35 % compared to the reference conventional implant.

Acknowledgements

The financial support by the Technology Agency of Czech Republic (TAČR project No. TA03010886) is gratefully acknowledged.

References

[1] T. Albrektsson, P. I. Branemark, H. A. Hansson, J. Linderstrom. Osseointegrated titanium implants.Requirements for ensuring a long lasting, direct bone-to-implant anchorage in man. *Acta Orthop Scan* **52**(2):155–70, 1981.

[2] B. Friberg, T. Jemt, U. Lekholm. Early failures in 4,641 consecutively placed Brånemark dental implants: a study from stage 1 surgery to the connection of completed prostheses. *The International journal of oral & maxillofacial implants* **6**(2):142–6, 1991.

[3] R. Osman, M. Swain. A Critical Review of Dental Implant Materials with an Emphasis on Titanium versus Zirconia. *Materials* **8**(3):932–958, 2015. doi:10.3390/ma8030932.

[4] C. Palma-Carrió, L. Maestre-Ferrín, D. Peñarrocha-Oltra, et al. Risk factors associated with early failure of dental implants. A literature review. *Medicina Oral, Patologia Oral y Cirugia Bucal* **16**(4):2–5, 2011. doi:10.4317/medoral.16.e514.

[5] F. Denk, F. Denk, A. Jíra. A dental implant shaft, Czech Republic, 306457, 2017.

[6] F. Denk, F. Denk, A. Jíra. A dental implant shaft, Czech Republic, 306456, 2017.

[7] ČSN EN ISO 14801: Dentistry – Implants – Dynamic loading test for endosseous dental implants.