ISSN 1846-6168 (Print), ISSN 1848-5588 (Online)
https://doi.org/10.31803/tg-20240426123331

Original scientific paper
Received: 2024-04-26, Accepted: 2025-01-02

A Comparative Structural Analysis of Total Hip Arthroplasty Designs

Elmedin Mešić*, Adela Zenkić, Adis J. Muminović, Nedim Pervan

Abstract: Structural analysis of two designs of total hip endoprosthesis was conducted with an emphasis on verifying stress conditions and displacement fields on the components and femur under loads typical for human gait. CAD (Computer Aided Design) models of a conventional endoprosthesis (Endoprosthesis 1) and a more modern solution (Endoprosthesis 2) were created using the CATIA (Computer Aided Three-dimensional Interactive Application) CAD/CAE (Computer Aided Engineering) system, based on which corresponding FEM (Finite Element Method) models were formed. By comparing the obtained results, it was found that a more uniform stress-strain pattern occurs on most components of Endoprosthesis 2 compared to Endoprosthesis 1. Additionally, the analysis of von Mises stresses and displacements at characteristic points of the femur showed a balanced distribution of stress and displacement. However, higher contact stresses between the stem and femur occur with Endoprosthesis 2, reducing the possibility of stress shielding due to the specifics of its design.

Keywords: CAD model; FEM model; stress shielding; structural analysis; total hip endoprosthesis; von Mises stress

1 INTRODUCTION

The occurrences of arthrosis, reduced mobility and hip joint fractures have led to the emergence and development of endoprostheses, which represent a clinical solution to these problems. Hip joint replacement is a surgical procedure aimed at improving the quality of life of individuals by removing damaged parts and replacing them with an endoprosthesis. Hip joint endoprostheses have evolved with advances in technology and science. From the first prostheses made from ivory (1891), through glass (1925) and the first metal-metal prostheses (1953), to prostheses similar to those used today, there has been significant progress. Indeed, Sir John Charnley, an orthopedic surgeon, is considered the father of modern total endoprostheses, having designed the first low friction arthroplasty in the early 1960s. Such prostheses typically involve femoral components made of metal and an acetabular component made of polyethylene or ceramics. Besides material changes, prostheses have also evolved in terms of design. There are one-piece (monolithic) or multi-piece (modular) prostheses, as well as partial prostheses that replace only the damaged part of the joint and total prostheses that involve replacing the entire joint body [1].

The development of design and selection of optimal materials for endoprostheses pose a challenge and serve as the foundation for new research. Carefully designed construction combined with the proper choice of materials creates an optimal combination for a hip endoprosthesis that will meet functional requirements. Balancing anthropometric requirements, biocompatibility, corrosion resistance, strength, toughness, material density, and other physical-mechanical properties is crucial in the construction of hip endoprostheses [1].

The modern era is characterized by the application of various CAD/CAE systems in many fields of science, including medical engineering. One of the many applications in this field is certainly in the development and testing of medical devices in orthopedics and orthodontics, such as external fixators [2 - 5] and mini-implants [6, 7].

The essential application of these systems is structural analysis of endoprostheses, where based on 3D CAD/FEM models, data on stress distribution and displacements can be obtained. In this way, the influence of the angle of the femoral component placement on the stresses generated on the femur and stem has been analyzed [8]. Also, an important parameter in the design of the endoprosthesis is considered to be the size of the femoral head component, so FEM analyses are conducted on its influence on the stress distribution of the prosthesis stem and femur [9]. Similarly, in the study, researchers analyze hybrid and conventional hip resurfacing implants using FEM analysis [10].

In addition to structural analysis of endoprostheses, tribological tests are conducted to reduce contact stresses and wear between components [11]. Thermoelastic stress analysis is used for obtaining and analyzing the complete stress distribution of the endoprosthesis and bone [12].

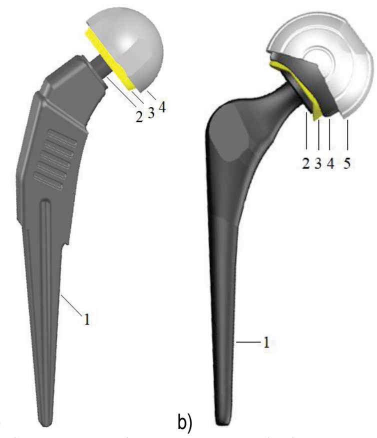
This research aims to conduct a comparative structural analysis of two designs of total modular hip endoprostheses, with a focus on verifying stress conditions and displacement fields on the components and femur under loading conditions typical for human gait.

Based on the obtained results, the influence of specific parameters of endoprosthesis design on the stress-strain picture has been determined.

2 THE DESIGN OF THE HIP ENDOPROSTHESES

In the first phase, CAD models of two analyzed designs of total hip endoprostheses were developed, where Endoprosthesis 1 represents a conventional modular solution typically consisting of a stem (1), head of femoral component (2), cup (3), and acetabular socket (4) (Fig. 1a). Endoprosthesis 2 represents a more contemporary modular solution for total hip endoprosthesis, which, in addition to the stem (1), includes components named according to the degrees of freedom they provide in relation to the acetabular socket (unidirectional cup) (5), namely abductor (2), rotator (3), and flexor (4) (Fig. 1b).

The fundamental difference of Endoprosthesis 2 compared to Endoprosthesis 1 lies in the design of the femoral head component and acetabular components, as well as the presence of an additional cup.

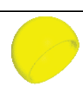
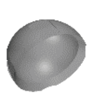


a) b)
Figure 1 CAD models: a) Endoprosthesis 1; b) Endoprosthesis 2

Endoprosthesis 1 belongs to the group of ball-and-socket implants. On the other hand, Endoprosthesis 2 utilizes three uni-directional sliding motions through three orthogonally positioned cylindrical joints (flexor, rotator, and abductor). This enables Endoprosthesis 2 to have three degrees of freedom, i.e. flexion/extension, internal/external rotation and abduction/adduction.

The differences in design are evident in the images in Tab. 1, which provide an overview of the design of all components of both endoprostheses.

Table 1 Comparative overview of hip endoprosthesis designs

Endoprosthesis 1		Endoprosthesis 2	
Femoral components			
Stem		Stem	
Femoral head		Abductor (Femoral head)	
Acetabular components			
/		Rotator (Cup 1)	
Cup		Flexor (Cup 2)	
Acetabular socket		Unidirectional cup (Acetabular socket)	

The femoral head component as well as the acetabular components of Endoprosthesis 2 are characterized by larger dimensions and a tendency towards a spherical-conical shape. Additionally, the cross-section of the stem of the femoral component of Endoprosthesis 2 is circular-oval in shape, unlike Endoprosthesis 1 where the cross-section is more rectangular. The materials of the analyzed hip endoprostheses need to possess excellent mechanical properties and wear resistance while satisfying biocompatibility requirements. This has resulted in the use of titanium, tantalum, ceramics and UHMWPE (Ultra-High Molecular Weight Polyethylene).

The Tab. 2 presents the material specifications and an overview of components for which these materials are applied. In addition to the endoprosthesis materials, the properties of cortical bone are also shown in the table [8 -11].

Table 2 Properties of endoprosthesis materials and cortical bone

Type of material	Application component	Elastic modulus, GPa	Tensile strength, MPa	Poisson's ratio	Density, kg/m^3
Titanium	Stem and heads of femoral components; Flexor	114	825	0.34	4460
Zirconium	Cup; Rotator	38	4000*	0.22	3960
Tantalum	Acetabular socket	186	705	0.35	16690
UHMWPE	Unidirectional cup	1	25	0.32	970
Cortical bone	Femur	17.4	115	0.33	1800

Note: * denotes compressive strength

3 STRUCTURAL ANALYSIS OF ENDOPROSTHESES

The FEM modeling of endoprosthesis components and the femur was performed using the CAD/CAE system CATIA. It is important to note that all input factors for the structural analysis of Endoprostheses 1 and 2 are identical, which is essential for obtaining relevant results.

All components of both hip endoprostheses and the femur model were discretized using finite elements of the parabolic tetrahedron type (TE10) with ten nodes and a total of thirty degrees of freedom (translations) [13-16]. The TE10 finite elements are isoparametric elements with second-order displacement interpolation functions. The need for more accurate results often leads to mesh refinement on characteristic components and locations. A finer mesh was applied to the femoral heads and acetabular components (Fig. 2a). The materials of the endoprostheses and femur were modeled as linearly isotropic (Tab. 2).

During the FEM analysis, the femur was sectioned below the endoprosthesis femoral component, and then a constraint in the form of clamp was applied to the lower surface resulting from the bone resection (Fig. 2b). This bone preparation for structural analysis is based on an analogy with experimental tests where the lower part of the bone is impinged, following ISO 7206-4 and researchers' work [9, 17, 18].

Figure 2 a) Finite element mesh and b) clamp constraint (Endoprosthesis 2)

In FEM analysis of the assembly, it is necessary to define interactions (connections) between components that share common boundaries. A rigid contact connection was used to define the connection between the stem of the femoral component and the femur, as well as between the stem and the head of the femoral component (Fig. 3a). The connection between the head of the femoral component and the acetabular cup, and between the cup and the acetabular bed (Endoprosthesis 1), as well as between the abductor and rotator, rotator and flexor, and flexor and unidirectional cup (Endoprosthesis 2), are defined as smooth contact connections to achieve appropriate rotations (Fig. 3b).

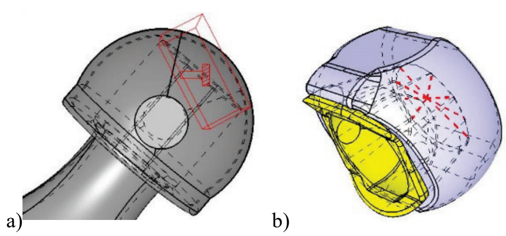


Figure 3 Contact connections (Endoprosthesis 2): a) rigid; b) smooth

The weight of the body above the hip joint and the abductor muscle force influence the generation of the resultant spatial force (R). The intensity and direction of the resultant force vary depending on the intensity of the acting forces and their lever arms, as well as the body's motion state. The structural analysis of the endoprostheses analyzed in this study was conducted for the case of normal walking of a person weighing 75 kg, where the resulting force amounts to 3 kN and acts at an angle of approximately 20° relative to the vertical axis [13, 15, 16]. The load from the pelvis is evenly applied to the upper outer surface of the acetabular bed, while the intensity, direction, and orientation are defined by the values of force components in the x, y, and z directions to achieve the required resultant force and angle of action (Fig. 4).

For the purpose of comparing the effects of two different endoprosthesis designs on the stress distribution and displacement fields on the femur, 12 characteristic points have been defined (Fig. 5).

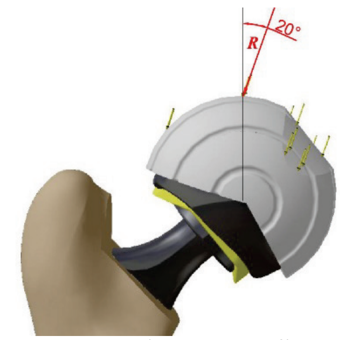


Figure 4 Load (Endoprosthesis 2)

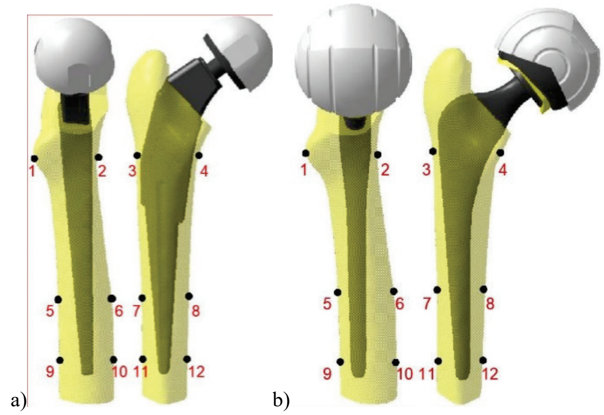


Figure 5 Characteristic points of the femur in conjunction with: a) Endoprosthesis 1; b) Endoprosthesis 2

3.1 Von Mises Stress Analysis

The display of von Mises stresses in the longitudinal section of Endoprosthesis 1 and Endoprosthesis 2 in conjunction with the femur is shown in Fig. 6. For the same loading and under the same boundary conditions, the stress pattern of Endoprosthesis 2 is more relaxed compared to the stress pattern of Endoprosthesis 1. The maximum von Mises stress in Endoprosthesis 1 occurred at the junction between the head and the stem (264 MPa), while in Endoprosthesis 2, it appears in the medial neck area (196 MPa), i.e., in the areas where the cross-sectional area of the endoprostheses is smallest which makes the specified locations critical points for the occurrence of fractures, especially with regard to fatigue [19-22].

In the case of Endoprosthesis 2, the results have shown that the von Mises stresses were higher at the interface between the stem and cancellous bone compared to Endoprosthesis 1 (Fig. 6). This indicates the potential of the Endoprosthesis 2 to reduce stress shielding. Additionally, studies [10, 16-18] associate contact stresses between the stem and femur with the occurrence of stress shielding.

A review of von Mises stresses for individual components of the endoprostheses is provided in Tab. 3. Due to the significant differences in stress levels across components, three stress scales have been introduced. The first scale pertains to femoral components, the second to acetabular components, and the third to the femur of both prostheses. Local maximum stresses occur at locations where there is a change in geometry and where the load from the previous component is significantly transferred to the observed component. The stress magnitude slightly varied from the acetabulum region to the femur region. Lower stresses are present on most components of Endoprosthesis 2. An exception is the flexator, which exhibits a higher stress by around 20 MPa compared to the cup of Endoprosthesis 1. A noteworthy finding is that the acetabulum socket of Endoprosthesis 2 has a stress over 100 MPa lower than the equivalent component of the conventional design (Tab. 3).

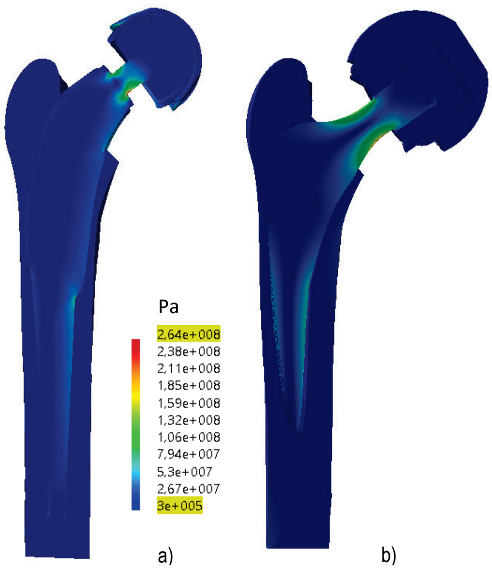
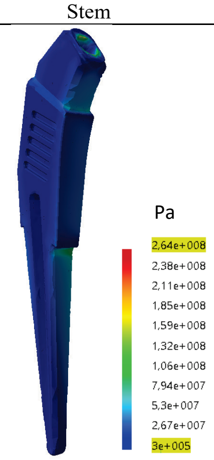
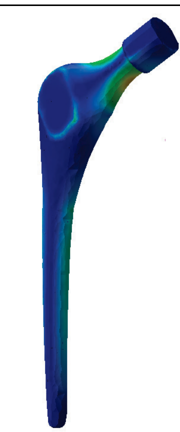
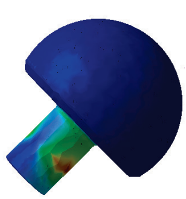
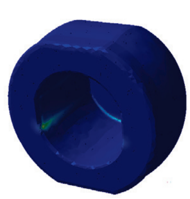
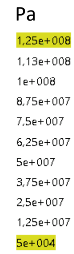
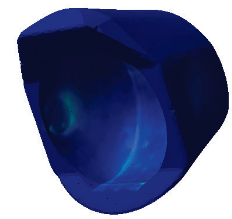
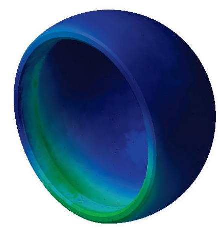
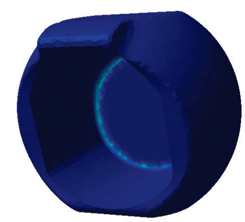
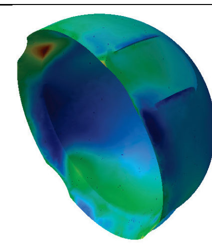
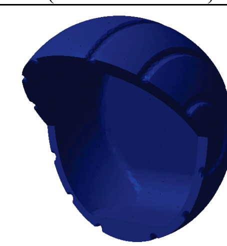


Figure 6 Von Mises stresses (longitudinal section) a) Endoprosthesis 1; b) Endoprosthesis 2

Table 3 Von Mises stresses of hip endoprosthesis components

Endoprosthesis 1	Endoprosthesis 2
Femoral components	
Stem	Stem
	
Femoral head	Abductor (Femoral head)
	
Acetabular components	
	Rotator (Cup 1) 
Cup	Flexor (Cup 2)
	
Acetabular socket	Unidirectional cup (Acetabular socket)
	
Femur	
	

It is evident that Endoprosthesis 2 is characterized by a more uniform stress distribution. The reason for this is certainly the design of Endoprosthesis 2, primarily the circular-oval cross-section of the stem with larger radii of curvature on the medial and lateral sides, larger dimensions of the femoral head component, and consequently larger dimensions of the acetabular components compared to Endoprosthesis 1. The minimum stress is observed at the upper part of the femur, where no load transfer occurs, resulting in the expected stress pattern.

A review of the results at 12 characteristic points on the femur is provided graphically in Fig. 7 and indicates a relatively uniform stress pattern of the femur for both endoprostheses. Stresses at points 2, 9, 11, and 12 for the case of using Endoprosthesis 2 are slightly more pronounced compared to the case of using Endoprosthesis 1. The reason for this is the position of the stem of Endoprosthesis 2 within the femur, which is closer to the end of the bone at these points. This is a consequence of the geometry of the stem of Endoprosthesis 2, especially its lateral part. Stresses at the other characteristic points of the femur in the case of applying Endoprosthesis 2 are lower compared to the case of applying Endoprosthesis 1 (Fig. 7).

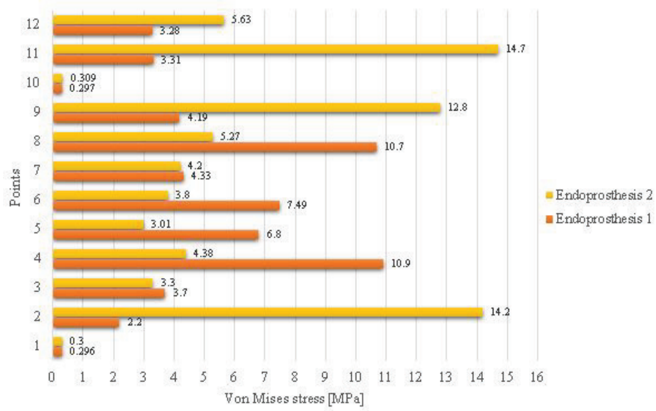


Figure 7 Von Mises stress at characteristic points of the femur

3.2 Displacement Analysis

The expected largest displacements are seen in the acetabular sockets (0.824 mm for the Endoprosthesis 1 and 0.479 mm for the Endoprosthesis 2), which represent the first link of the prosthesis-pelvis connection. The displacements of both hip endoprostheses are shown in Fig. 8. Tab. 4 provides an overview of the displacement results of individual components of both endoprostheses. The displacements of all components of both prostheses are shown using the same scale.

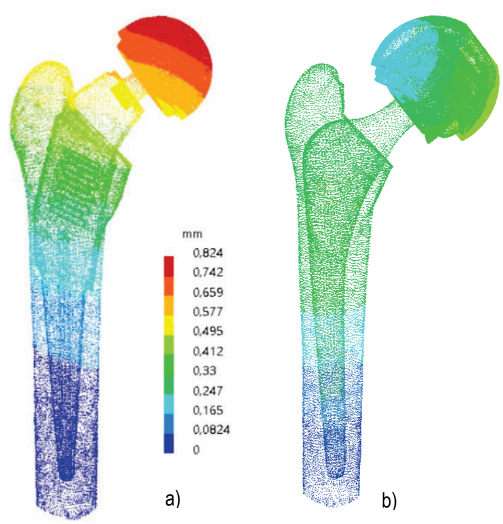


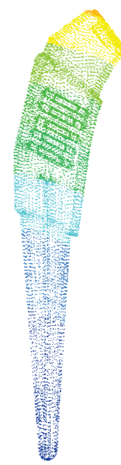
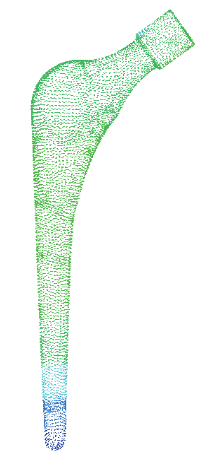
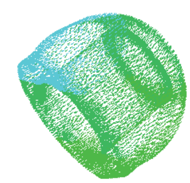
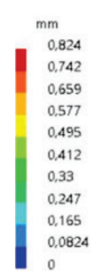
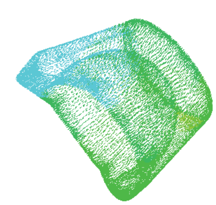
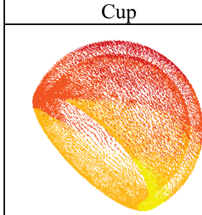
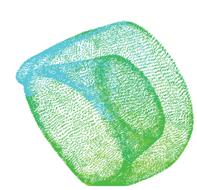
Figure 8 Displacements a) Endoprosthesis 1; b) Endoprosthesis 2

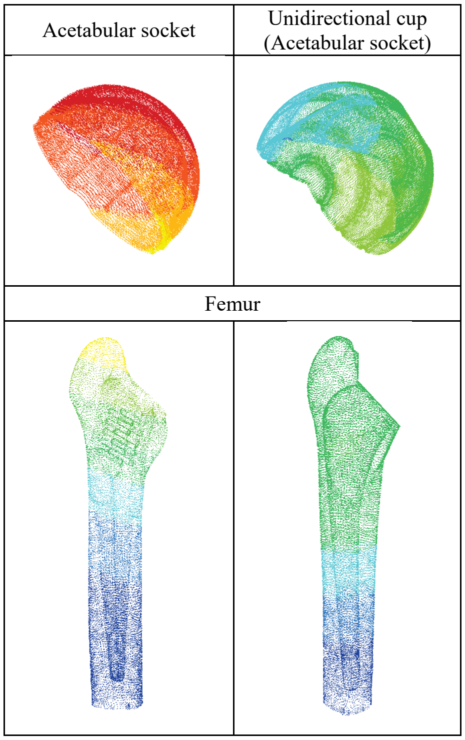
The overview of displacements of characteristic points of the femur shown in Fig. 9 indicates a consistent pattern of displacement in most observed points.

Displacements at points 1, 2, 4, 9, and 10 for the case of using Endoprosthesis 1 are slightly more pronounced than for the case of using Endoprosthesis 2. Displacements in other characteristic femoral points for the application of Endoprosthesis 1 are smaller compared to Endoprosthesis 2.

After conducting the analysis of von Mises stress and displacement, it is possible to identify a more relaxed stress-strain pattern with less pronounced extremes in the case of Endoprosthesis 2 compared to Endoprosthesis 1.

Table 4 Displacements of hip endoprosthesis components

Endoprosthesis 1	Endoprosthesis 2
Femoral components	
Stem	Stem
	
Femoral head	Abductor (Femoral head)
	
Acetabular components	
	Rotator (Cup 1) 
Cup	Flexor (Cup 2)
	



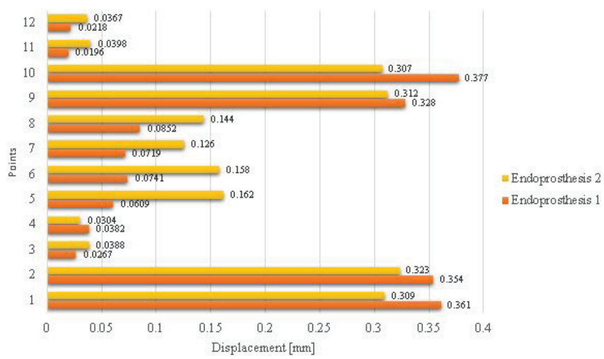


Figure 9 Displacement of characteristic points of the femur

3.3 Testing the Stability of the FEM Models

FEM solutions are essentially approximate and their accuracy, stability, and convergence can be questioned. Accuracy refers to the deviation of FEM solutions from the exact solution, while stability concerns the model's performance sensitivity to changes in significant factors. If a small change in a model parameter results in large changes in the solution, then the model is unstable. In the case where gradual increases in the number of finite elements do not lead to significant changes in displacement and stress results, convergence is achieved. If the solution is convergent, it is considered stable [23].

Stability and convergence check was performed by refining the finite element mesh in five iterations. Each iterative procedure was accompanied by a slight change in the number of finite elements. An overview of the number of nodes, number of finite elements, maximum displacement, and von Mises stress is provided in Tab. 5.

Table 5 Convergence of the FEM model of endoprosthesis.

Iteration	Number of nodes	Number of elements	Max. displacement, mm	Max. von Mises stress, MPa
Endoprosthesis 1				
I	161222	99900	0.827	278
II	164070	101637	0.827	272
III	169530	105486	0.825	266
IV	179013	111515	0.824	265
V	185831	116283	0.824	264
Endoprosthesis 2				
I	287128	180578	0.480	242
II	297516	186752	0.477	206
III	323385	204844	0.488	199
IV	338269	213779	0.484	196
V	344922	218321	0.479	196

Fig. 10 provides a graphical representation of the behavior of the maximum displacement values and maximum von Mises stresses of the hip endoprosthesis as a function of the number of finite elements in the FEM model.

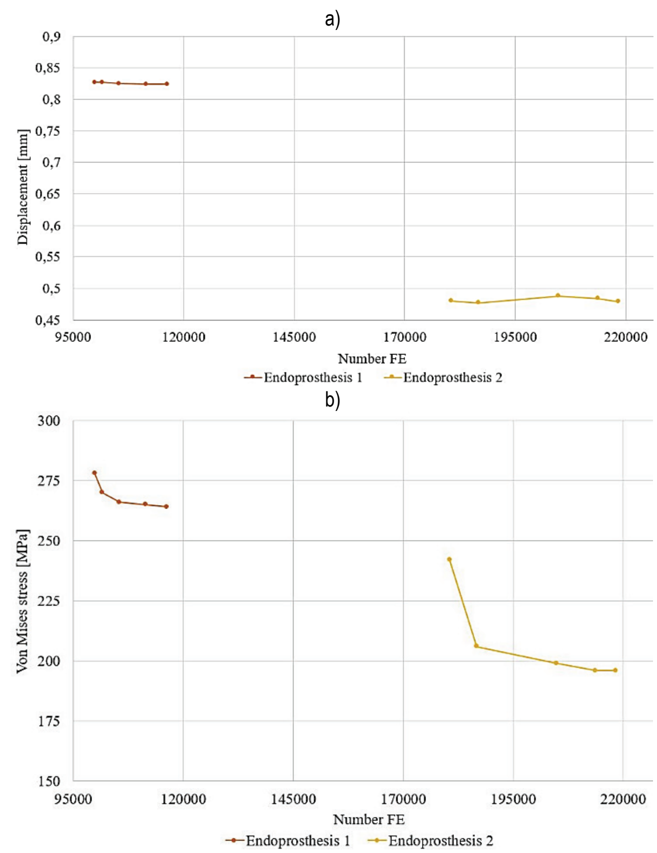


Figure 10 Stability diagram of the FEM model a) maximum displacement; b) maximum von Mises stress as a function of the number of elements

Through the displayed iterations, the size of the finite elements and the parameter 'sag' of all components of the FEM model were successively reduced. The 'sag' parameter represents the maximum deviation between the geometry of CAD and FEM model.

It is evident that with an increase in the number of finite elements, the values of maximum von Mises stresses exhibit a decreasing trend for both endoprostheses, with a clear convergence.

4 CONCLUSION

Improvement in the stability and the fixation of the stem in the bone are the main concerns for the surgeons during treatment. The design, material and application of the prosthesis are the crucial factors that influence the success of the treatment.

In this study, models of two total hip endoprostheses were created and analyzed with the aim of comparing the mechanical stability of conventional and modern solutions. The design and structural analysis were conducted in accordance with anthropometric-dimensional constraints, using a similar combination of materials.

The stability of the developed models was determined by examining the convergence of the FEM models of the endoprostheses.

When analyzing the obtained numerical results, it is evident that lower stresses and displacements occur on Endoprosthesis 2. However, the results showed that the von Mises stress value did not exceed the limit stresses of the femur material or prosthesis components. It was observed that with a longer radius on the lateral part of the stem of Endoprosthesis 2, a better distribution of the stresses was obtained, generating less displacement.

The significant difference in the distribution and magnitude of stress in the stem of Endoprosthesis 1 and the femur, with markedly lower contact stresses (44 MPa) compared to Endoprosthesis 2, is notable. On the other hand, the difference in the distribution and magnitude of stress in the stem of Endoprosthesis 2 and the femur is significantly smaller, with more pronounced contact stresses (93 MPa). Lower contact stresses between the stem of the endoprosthesis and the femur can lead to the so-called stress shielding phenomenon. The effect of stress shielding weakens the bond between the bone and the implant and is one of the serious problems in the long-term reliability of hip joint endoprostheses [16, 17]. Therefore, Endoprosthesis 2 indicates the potential to reduce stress shielding.

Also, the excessive displacement will drive implant loosening, which could affect the tissue healing response [17]. The maximum displacement is predicted at the proximal region of the endoprosthesis. The acetabulum loading applied to the prosthesis stem will lead to bending effects and further contribute to higher displacements at the proximal region of the endoprosthesis. Significantly larger displacements are present on all components and the femur in the case of Endoprosthesis 1, which can further lead to weakening of the bone-stem connection.

Based on the results of von Mises stress and displacement at characteristic points of the femur for both endoprosthesis solutions, it can be concluded that the position of the stem of the endoprosthesis plays an important role in the stress-strain pattern of the femur. Such observation provides a basis for further research in the field of optimizing the design and position of the endoprosthesis in the femur.

It can be concluded that Endoprosthesis 2 demonstrates its superiority by exhibiting a more relaxed stress-strain pattern primarily due to:

- Circular-oval cross-section of the stem of the femoral component
- Larger dimensions of the femoral head component
- Larger radii of curvature on the medial and lateral sides of the stem
- Larger dimensions of the acetabular components
- Spherical-conical shape of the acetabular components.

Additionally, thanks to the spherical-conical contact surfaces between the acetabular components, lower contact stresses are present.

The developed FEM models of endoprostheses have certain limitations in the sense that all materials are modeled as linear isotropic, which is certainly one direction for their improvement.

Finally, it can be stated that this study can serve as a basis for further research in the direction of redesign and material selection that performs an optimal function in the interaction of bone-implant. Also, the conducted research can serve as a basis for additional analysis and the development of new concepts of hip endoprostheses depending on the morphological needs and health issues that patients may have (such as osteoporosis or fractures) during surgery.

5 REFERENCES

[1] Knight, S. R., Aujla, R., & Biswas, S. P. (2011). Total Hip Arthroplasty - over 100 years of operative history. *Orthopedic Reviews, 3*(2), e16. https://doi.org/10.4081/or.2011.16

[2] Li, J., Zhao, X., Hu, X., Tao, C., & Ji, R. (2018). A theoretical analysis and finite element simulation of fixator–bone system stiffness on healing progression. *Journal of Applied Biomaterials & Functional Materials*, *16*(3), 115-125. https://doi.org/10.1177/2280800017750357

[3] Mešić, E., Pervan, N., Muminović, A. J., Muminović, A., & Čolić, M. (2021). Development of knowledge-based engineering system for structural size optimization of external fixation device. *Applied Sciences*, *11*, 10775. https://doi.org/10.3390/app112210775

[4] Amaro, A. M., Paulino, M. F., Roseiro, L. M., & Neto, M. A. (2020). The Efect of External Fixator Configurations on the Dynamic Compression Load: An Experimental and Numerical Study. *Applied Sciences*, *10*, 3. https://doi.org/10.3390/app10010003

[5] Mešić, E., Muminović, A., Čolić, M., Petrović, M., & Pervan, N. (2020). Development and Experimental Verification of a Generative CAD/FEM Model of an External Fixation Device. *Tehnički glasnik*, *14*(1), 1-6. https://doi.org/10.31803/tg-20191112161707

[6] Holberg, C., Winterhalder, P., Rudzki-Janson, I., & Wichelhaus, A. (2014). Finite element analysis of mono- and bicortical mini-implant stability. *European Journal of Orthodontics, 36*, 550-556. https://doi.org/10.1093/ejo/cjt023

[7] Mešić, E., Muratović, E., Redžepagić-Vražalica, L., Pervan, N., Muminović, A. J., Delić, M., & Glušac, M. (2021). Experimental & FEM Analysis of Orthodontic Mini-Implant Design on Primary Stability. *Applied Sciences*, *11*(12), 5461. https://doi.org/10.3390/app11125461

[8] Güvercin, Y., Yaylacı, M., Ölmez, H., Yaylacı, E. U., Özdemir, M. E., & Dizdar, A. (2022). Finite element analysis of the mechanical behavior of the different angle hip femoral stem. *Biomaterials and Biomedical Engineering, 6*(1), 29-46. https://doi.org/10.12989/bme.2022.6.1.029

[9] Chen, D. W., Lee, M. S., & Lin, C.-L. (2018). Finite Element Analysis of Stresses from Hip Implants with Different Head Sizes. *International Journal of Research Studies in Science, Engineering and Technology, 5*(5), 1-8.

[10] Bougherara, H., & Bureau, M. N., (2008). Biomimetic Composite-Metal Hip Resurfacing Implant. *Advances in Materials Science and Engineering*, vol. 2008, Article ID 368985, 4 pages. https://doi.org/10.1155/2008/368985

[11] Dalli, D., Buhagiar, J., Mollicone P., & Wismayer, P. S. (2022). A novel hip joint prosthesis with uni-directional articulations for reduced wear. *Journal of the mechanical behavior of biomedical materials*, *127*. https://doi.org/10.1016/j.jmbbm.2021.105072

[12] Zanetti, E. M., & Audenino, A. L. (2010). Differential thermography for experimental, full-field stress analysis of hip arthroplasty. *Journal of Mechanics in Medicine and Biology, 10*(3), 515-529. https://doi.org/10.1142/S0219519410003496

[13] Rezaei, F., Hassani, K., Solhjoei, N., & Karimi, A. (2015). Carbon/PEEK composite materials as an alternative for stainless steel/titanium hip prosthesis: A finite element study. *Australas. Phys. Eng. Sci. Med., 38*, 569-580.

[14] Guzmán, M., Durazo, E., Ortiz, A., Sauceda, I., Siqueiros, M., González, L., & Jiménez, D. (2022). Finite Element Assessment of a Hybrid Proposal for Hip Stem, from a Standardized Base and Different Activities. *Appl. Sci. 12*, 7963. https://doi.org/10.3390/app12167963

[15] Oshkour, A. A., Osman, N. A., Bayat, M., Afshar, R., & Berto, F. (2014). Three-dimensional finite element analyses of functionally graded femoral prostheses with different geometrical configurations. *Mater. Des., 56*, 998-1008. https://doi.org/10.1016/j.matdes.2013.12.054

[16] Bougherara, H., Zdero, R., Dubov, A., Shah, S., Khurshid, S. & Schemitsch, E. H. (2011). A preliminary biomechanical study of a novel carbon-fibre hip implant versus standard metallic hip implants. *Med. Eng. Phys*., 33, 121-128. https://doi.org/10.1016/j.medengphy.2010.09.011

[17] Konow, T., Glismann, K., Lampe, F., Ondruschka, B., Morlock, M. M., & Huber, G. (2023). Stem size and stemalignment affects periprosthetic fracture risk and primary stability in cementless total hip arthroplasty. *J Orthop Res.*, 1-8. https://doi.org/10.1002/jor.25729

[18] Ahmad, M. A., Zulkifli, N. N. M. E., Shuib, S., Sulaiman, S. H., & Abdullah, A. H. (2020). Finite Element Analysis of Proximal Cement Fixation in Total Hip Arthroplasty. *International Journal of Technology, 11*(5), 1046-1055. https://doi.org/10.14716/ijtech.v11i5.4318

[19] Sedmak, A., Čolić, K., Grbović, A., Balać, I., & Burzić, M. (2019). Numerical analysis of fatigue crack growth of hip implant, *Eng. Fract. Mech*. 216: 106492. https://doi: 10.1016/j.engfracmech.2019.106492

[20] Babić, M., Verić, O., Božić, Ž., & Sušić A. (2020). Finite element modelling and fatigue life assessment of a cemented total hip prosthesis based on 3D scanning. *Engineering Failure Analysis, 113*. https://doi.org/10.1016/j.engfailanal.2020.104536

[21] Reginald, J., Kalayarasan, M., Chethan, K. N., & P. Dhanbal, P. (2023). Static, dynamic, and fatigue life investigation of a hip prosthesis for walking gait using finite element analysis. *International Journal of Modelling and Simulation*, *43*(5), 797-811. https://doi.org/10.1080/02286203.2023.2212346

[22] van Doesburg, P. G., van Langelaan, E. J., Apachitei, I., Bénard, M. R., & Verdegaal, S. H. M. (2020). Femoral prosthesis neck fracture following total hip arthroplasty — a systematic review. *Arthroplasty* 2, 28. https://doi.org/10.1186/s42836-020-00047-3

[23] Zienkiewicz, O. C., Taylor, R. L., & Zhu, J. Z. (2005). *The Finite Element Method: Its Basis and Fundamentals*. 6th edition, Butterworth-Heinemann, Oxford, ISBN 0-7506-6320-0.

Authors' contacts:

Elmedin Mešić, PhD, Full Professor
(Corresponding author)
University of Sarajevo - Faculty of Mechanical Engineering,
Vilsonovo šetalište 9, 71000 Sarajevo, Bosnia and Herzegovina
+38733729800, mesic@mef.unsa.ba

Adela Zenkić, MSc in Mechanical Engineering, Designer
NewCold Metal Construction,
Žabljak bb, 74230 Usora, Bosnia and Herzegovina
adelazenkic@gmail.com

Adis J. Muminović, PhD, Full Professor
University of Sarajevo - Faculty of Mechanical Engineering,
Vilsonovo šetalište 9, 71000 Sarajevo, Bosnia and Herzegovina
+38733729800, adis.muminovic@mef.unsa.ba

Nedim Pervan, PhD, Full Professor
University of Sarajevo - Faculty of Mechanical Engineering,
Vilsonovo šetalište 9, 71000 Sarajevo, Bosnia and Herzegovina
+38733729800, pervan@mef.unsa.ba