Metalurgi (2023) 3: 117 - 124

METALURGI
Majalah Ilmu dan Teknologi
p-ISSN 0126-3188 | e-ISSN 2443-3926
ejurnalmaterialmetalurgi.lipi.go.id

COMPARATIVE STUDIES SIMULATION SOFTWARE FOR BONE PLATE COMPRESSION

Dita Mayasari[a,*], Sirojuddin Muhammad[b], Joko Triwardono[c], Daniel Panghihutan Malau[c], Muhammad Satrio Utomo[d], Talitha Asmaria[c]

[a]Department of Biomedical Engineering, University of Dian Nuswantoro
Jl. Nakula, Semarang, Indonesia 50131
[b]Department of Biomedical Engineering, Airlangga University
Kampus C Mulyorejo, Surabaya, Indonesia 60115
[c]Research Center for Metallurgy, National Research and Innovation Agency
B.J. Habibie Sains and Technology Area, Banten, Indonesia 15314
[d]Department of Biomedical Engineering, University of Melbourne
Grattan Street, Parkville, Victoria, Australia, 3010
*E-mail: dita.mayasari@udinus.ac.id

Received: 06-11-2023, Revised: 24-12-2023, Accepted: 29-12-2023

Abstract

Medical applications occasionally require PSI (patient-specific implant) designs to match the implant bone's geometry. To verify and predict failures of the design as well as a treatment before the manufacturing process, FEA (finite element analysis) is employed to simulate when given a specific number of loads. Plenty of studies have done the FEA using a couple of types of software; however, to the best of our knowledge, there is no literature to compare those several FEA results with a comparable experiment. This study further analyzes material stress, particularly to compute the VMS (Von Misses stress) of the Ti6Al4V bone plate. Furthermore, this study proposes to examine and deliver a comprehensive understanding using the four most used software of COMSOL, Ansys, Abaqus, and Autodesk Inventor. The results of those four simulations are then compared with the stress test through the Hardness Vickers test. This study will contribute significantly as a novel comparison between VMS and hardness test as a stress prediction in an implant material.

Keywords: *Finite element analysis, titanium alloy, bone implant, Von Misses stress*

1. INTRODUCTION

Regular bone plate designs and geometry updates are crucial to meet the bone anatomy criteria. Plenty of patents and research reveal many challenges to improving the suitability and biocompatibility factors between the implant and the bone [1]-[5]. An implant aims to re-align and anchor the fragments into place and increase the recovery process. An implant with a configuration of bone plates is commonly made from metal to support the flexibility in following the bone structures. In a surgery procedure, it is inserting and fixating the bone plate in place. Among commercial implant products, there are occasions where a unique manufacturing process is required to fulfill a particular need called a PSI (patient-specific implant) [6]. It aims to tackle the problem when using commercial products where sometimes it may not fit properly with the patient and lead to a longer rehabilitation time [7]. The configuration of PSI will attempt to provide the patient's bone surface structures, has excellent accuracy, and expects to accelerate the recovery period. From a previous project, the PSI cost is also less than a massively produced implant product.

While making any implant geometries, it is essential to check the implant strength, both on design and material. Computational modeling on material analysis, particularly stress simulation,

DOI : 10.55981/metalurgi.2023.738

Metalurgi is Sinta 2 Journal (https://sinta.kemdikbud.go.id/journals/profile/3708) accredited by Ministry of Education, Culture, Research, and Technology, Republic Indonesia

is broadly used to investigate the ability of intended materials to hold specific loads [8]–[10]. Among many algorithms and techniques, a finite element method is popular for calculating the material capability for various purposes [10]. For example, FEA (finite element analysis) has been done in the medical field to understand the ability of a designed porous hip stem for children [11]. The stress simulation will be crucial since its prediction by scientific computation is accepted to be the actual condition in both the pre-treatment and after-manufacturing process [12]. By having the implant stress simulation, the researcher might be able to predict the material failure when applied.

Furthermore, in medical cases, particularly in implant device designs, the stress simulation using FEA software has successfully achieved the VMS (Von Misses stress) [13]. VMS is assumed as a value to understand the yield or fracture that might happen when the material is given a specific load number. Research of FEA to investigate fracture fixation, for instance, could produce von stress misses and bone strain in the lateral plate and anterior plate of the femur bone [12]. Moreover, VMS could predict the tension state of two new designs of dental screw implants [14]. Both previous studies summarize that the VMS through FEA simulation is a promising tool to clarify several upcoming conditions in post-implant installation, such as stress distribution, the obtained maximum stress, and the low and high tension.

Many software, for example, Ansys, Abaqus, Autodesk Inventor, MATLAB, COMSOL, and SOLIDWORKS, can simulate the FEA [8]-[9], [15]–[19]. Almost all authors will only choose one of those software as the FEA simulation tool for their designs. Based on the last research of FEA on a bone plate for a patient-specific implant, the authors try to simulate the FEA using Inventor and an additional comparison using ABAQUS [20]. The study's result shows a significant difference in the number for the VMS.

In this experiment, we are trying to understand the comparison from several simulation using four common FEA software that can provide a stress number and comparing it to the hardness Vicker's test. In this study, we also limit the studied material for the Ti6Al4V, as it the most usage implant material in actual. A small square was drawn using a CAD (computational aided design) software, that is assumed as a part of bone plate. In the hardness Vicker's test, we test the three-dimensional (3D) printed Ti6Al4V of a bone plate, cut it to the similar dimension to the CAD model. The results from the hardness Vicker's test would be converted into MPa unit as a representative of stress numbers [21]-[22].

Overall, an accurate implant design recommendation based on FEA software simulation is crucial for medical applications and patient life concerns. However, based on many literature reviews about FEA simulations, there is no literature that mentions if a chosen FEA software is better than others or if simulation using all of them could produce similar results. The FEA simulation using many software and setting the same parameters would give a beneficial understanding of which software can provide the correct strength number, or comparing at least two FEA software should be very important.

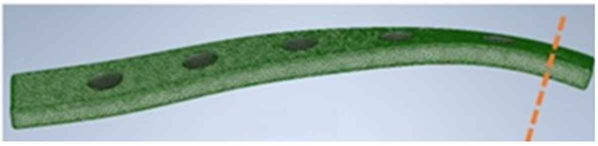
2. Materials And Methods

In previous study, a designed patient-specific implant for iliac bone has been simulated using two different software of Autodesk Inventor and ABAQUS [20]. It produced significant differences on the maximum number from FEA simulations when adjusted with a 307 N of loads [23]. This number is based on load transfer data that passes through the pelvis in an adult. In those two previous finite element simulations, the constraints and loads placements were also unchanging where the side that facing the iliac bone would be constrained and other sides would be given loads. In this study, the placement of constraints and loads are also referring the prior experiment where one side is constrained and other five sides are given a force load.

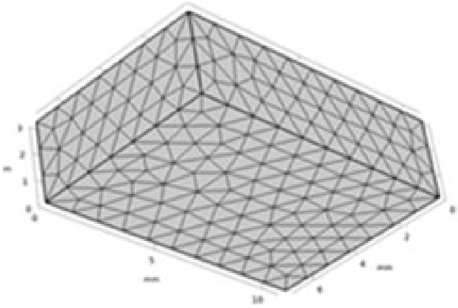
Furthermore, since it was questioned in last study that to prove the von Misses stress examination where in some cases results in huge different numbers, the simulation could be computed using plenty different FEA software and compared to each other. After that, the results could be furthered compared with an equal stress test such as using a hardness test that the results could converted into MPa. By this problem, this study proposed to design a small part of the intended iliac plate design and simulate it using FEA as well as compare to the hardness Vicker's Test. A square with a length of 10.8 mm, a width of 7 mm, and a height of 3 mm were drawn using a CAD software. The dimension of the prototype is following the dimension of a bone plate that were cut for this experiment. The used load for the small fragment is 50 N [24]. This number is considered as another load that is also used for bone plate configuration. The small number contributes to the less time consuming for the finite element simulations. Figure 1 shows the prototype design and its meshing scheme for the

FEA simulation. To arrange the boundary conditions, the elements used is a tetrahedron type with the element numbers that is as close as possible to all software, so that the calculation and the placement of the load and constraints of plate on all software is the same.

(a)



(b)



(c)

Figure 1. Patient specific implant design for iliac bone; (a) The intended placement of implant to the bone area, (b) Bone plate drawing scheme for implant, which the orange line indicated the cutting area for the simulated sample, (c) Discretization of the sample

In this study, the selected material is a library for Ti6Al4V annealed in the software of ANSYS (Ansys Student Version 2022 R2 (22.2), Pennsylvania-USA), Inventor (Inventor 2022 Student Version) and COMSOL (COMSOL Multiphysics 5.4). Unlike previous study, this work also compared to the simulation for ABAQUS (ABAQUS Free Learning Edition) and the setting material properties in ABAQUS is manually inputted. The material properties in the library of three software were similar to other literatures [11]. All FEA simulation were processed using one personal computer of Dell Vostro 3405 AMS Ryzen3SSD 254 GB. Table 1 explains the inserted Ti6Al4 material properties.

In this study, the hardness test is is only the test that is comparable to the FEA simulations.

All finite element computation results are then compared with the hardness test experiment using AFFRI EX 206 hardness Vicker's test (Wood Dale, USA). The Ti6Al4V bone plate fragment was printed by 3D Metalforge Pte Ltd, Singapore. Compared to other tests, the compressive test that have been done only confirm the material properties of Titanium alloy, which the strength is still on the range of the Ti6Al4V material properties in the Table 1. The bending test cannot be done since the sample is inadequate to do so.

The relationship between hardness and the strength has been confirmed by two previous studies [21]-[22]. The hardness test set to load 100 Kilogram force with five points of indentation. Hardness Vicker's test has been enormously an effective way to identify the mechanical properties of pure titanium and its modification [22]. Figure 2 shows the hardness Vickers test application from preparing the bone plate part that was cut from the whole bone plate (a) and the proof of bone plate part that has been given a load (b).

Tabel 1. Material properties of Ti6Al4V

Parameters	Minimum Value	Maximum Value
Density(kg/m^3)	4429	4512
Tensile strength (MPa)	900	950
Yield strength (MPa)	880	920

(a) (b)

Figure 2. Hardness Vicker's test on; (a) a small fragment of iliac bone plate titanium using and its indentation on (b) process

3. Result and Discussion

In the finite element simulations, the number of elements obtained in ANSYS, COMSOL, Autodesk Inventor and ABAQUS are 2707, 2701, 2759 and 2781 respectively. Figure 3 to Figure 6 illustrate the finite element simulation results in three different software of ANSYS, Inventor, COMSOL and Abaqus respectively.

In the VMS (Von Misses stress) results, it appears plenty of numbers regarding the colour and area of the plate. It could be interpreted that among three software of Ansys, Inventor and Comsol has quite similar ranges of VMS numbers, however Abaqus reveals greater numbers than others. If we look at the process of FEA (finite element analysis) simulation, Ansys, Inventor and Comsol already have libraries of intended materials. In this study, we employed those libraries of Ti6Al4V annealed for Ansys, Inventor and Comsol.

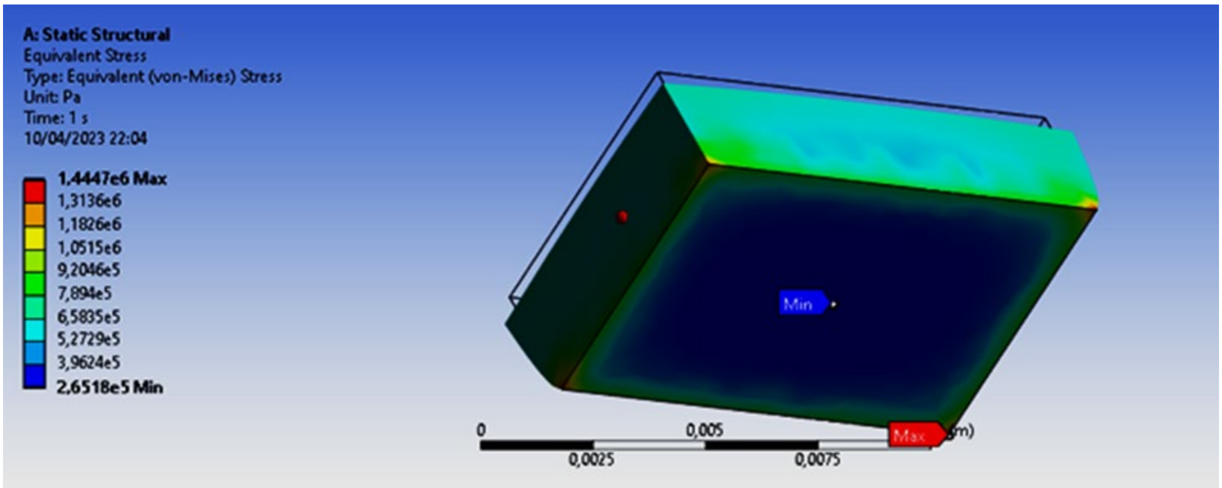


Figure 3. Von Misses stress from FEA simulation in ANSYS

Unlike other software, Abaqus must insert the material properties manually. Since the libraries on Ansys, Inventor and Comsol could not be opened, therefore we input material of Ti6Al4V that revealed in the Table 1.

The differentiation of colors appears based on the undergone simulation. The red or the maximum value appear as critical area/number after given a load in the simulation. Other colors appear as the range of distribution the stress results after given a load. Table 2 summarizes the results from all FEA simulations with limit the colour groups only into six categories.

Tabel 2. Von Misses stress results (MPa unit)

Colours	Ansys	Inventor	Comsol	Abaqus
Red	1.313-1.447	1.2-1.31	1.237-1.347	51.93-56.6
Orange	1.182-1.313	1-1.2	1.128-1.237	42.54-51.93
Yellow	1.051-1.182	0.8-1	0.908-1.128	37.84-42.54
Green	0.658-1.051	0.6-0.8	0.689-0.908	19.05-37.84
Sea Blue	0.396-0.658	0.4-0.6	0.469-0.689	9.651-19.05
Dark Blue	0.265-0.396	0.2-0.4	0.25-0.469	0.254-9.651

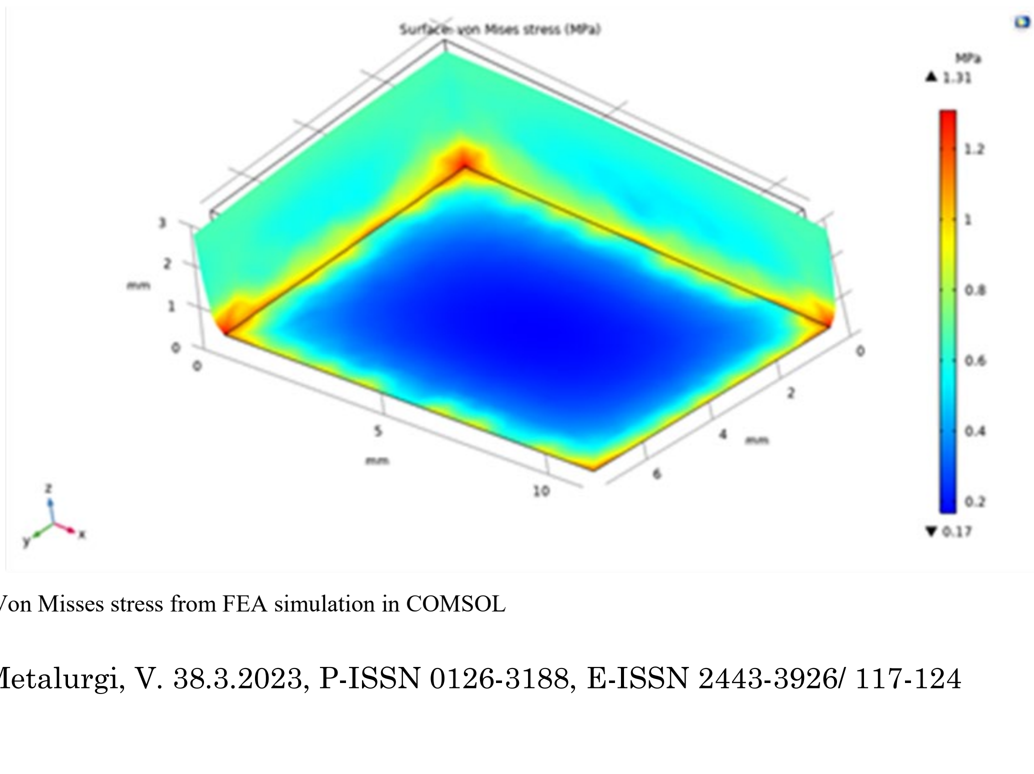


Figure 4. Von Misses stress from FEA simulation in COMSOL

In the hardness test, we use AFRI System for hardness Vicker's test. A load of 100 kilograms force was given to the surface of the bone plate part. The test was repeated five times in the middle of bone plate fragment and four times in the edge of the bone plate fragment. The results for five repeated hardness Vicker's test on the middle of the bone plate fragment are 12.5, 13, 13, 13.5 and 15.5. Then, at the four edges we achieved 46.8, 48.8, 39.4 and 45.2. The results are measured in hardness Vicker's (HV) unit. Figure 7 shows all indentation locations.

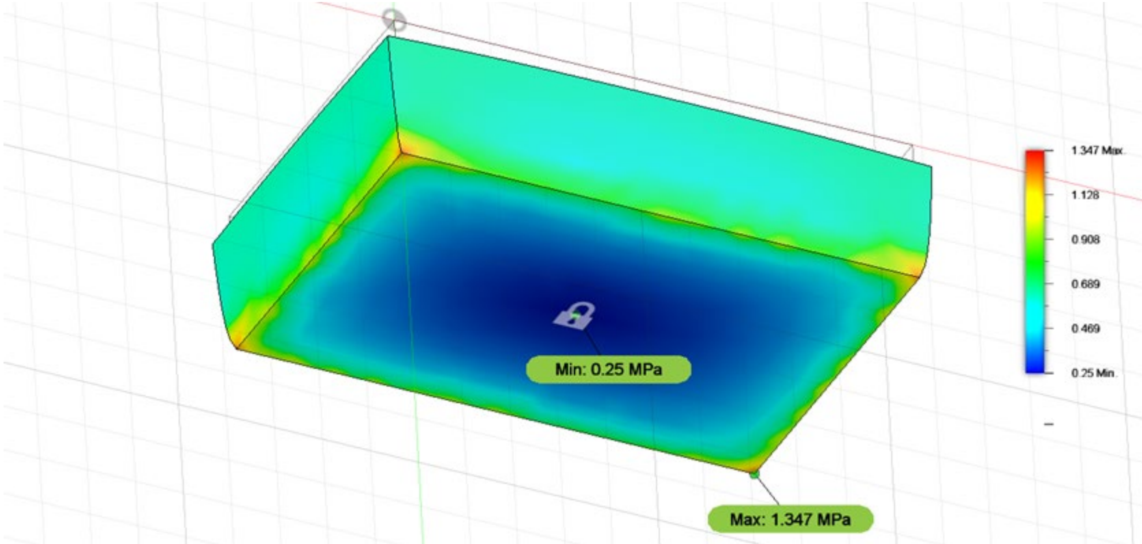


Figure 5. Von Misses stress from FEA simulation in INVENTOR

One kilogram force equals to 9.8 N or could be accumulated to 10 N. In the hardness Vicker's test, a load of 100 kilogram force (Kgf) is then equal to 1000N. This study computed 50 N as the load number in the FEA simulation. Table 3 summarizes the calculation of 50 N as the load number to the result of each indentation in HV units. The first column is the HRV results originally from the hardness apparatus, with the automatic set load of 100Kgf. Since the given load in the FEA simulation are only 50N, the second column convert the results from a load of 100Kgf to a load of 50N. Finally, the last column reveals the stress calculation from HRV results on 50N load to the MPa units. Based on [21]-[22], the relationship between hardness and yield stress can be estimated in the Equation 1.

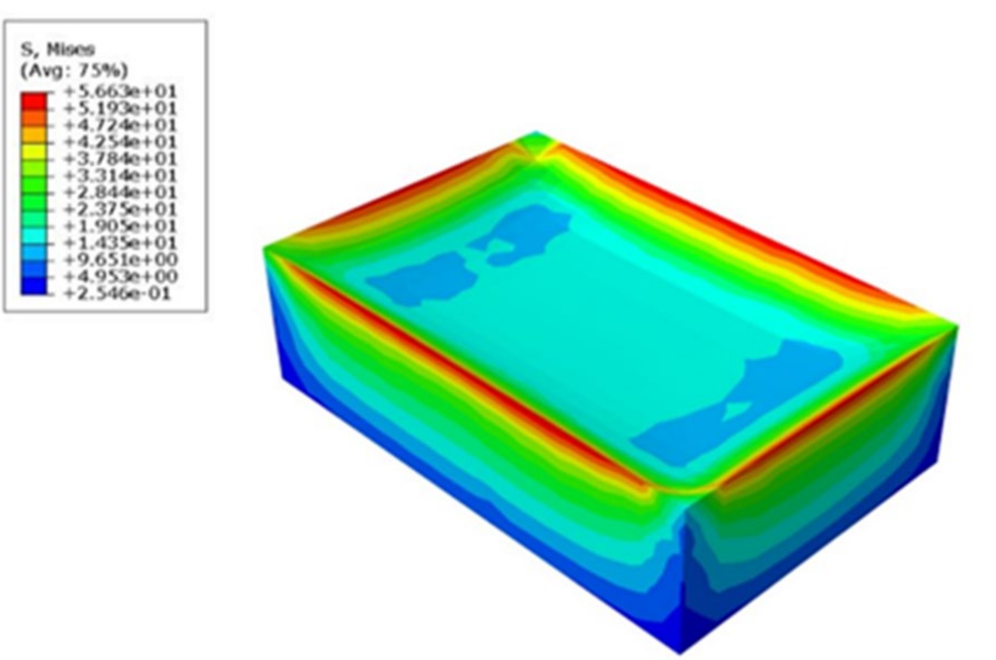


Figure 6. Von Misses stress from FEA simulation in ABAQUS

Figure 3 to Figure 6 are results from all FEA simulations and the VMS number already summarized in the Table 2. They indicate that the middle area that mostly appears as the dark blue zone having VMS number between 0.2 to 0.496 in three software of Ansys, Inventor, and Comsol, however it has 0.2 to 9.651 in Abaqus.

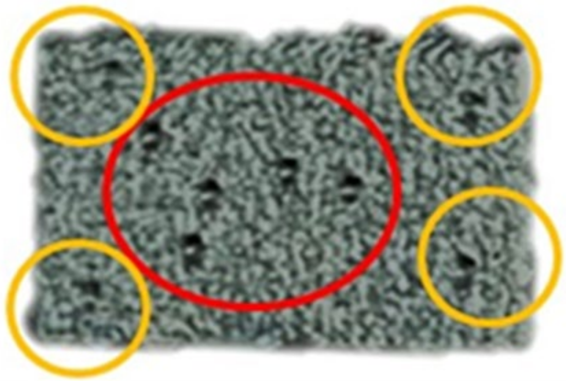


Figure 7. Nine indentation locations, five are in the middle of the bone plate fragment with a red circle and four are in the edge of the bone plate fragment with four yellow circles

$Hv \approx 3\sigma$ ……………………………………..(1)

1 Kgf = 10 N
100 Kgf = 1000N
100 Kgf = The HRV result
1000 N = The HRV result
50 N = The HRV result/20

Tabel 3. Conversion from HRV results to stress numbers

Area of Indentation	HRV Results on 100 Kgf AFRI (HV)	HRV Results on 50 N load (similar to FEA) (HV)	Stress Calculation using Equation (1) (MPa)
Edges	46.8	2.34	0.78
Edges	48.8	2.44	0.813
Edges	39.4	1.97	0.656
Edges	45.2	2.26	0.753
Middle	12.5	0.625	0.208
Middle	13	0.65	0.216
Middle	13	0.65	0.216
Middle	13.5	0.675	0.225
Middle	15.5	0.775	0.258

By this calculation using Equation 1, an indentation in the middle using HRV has results ranging from 12.5 to 15.5 HV and the conversion to the stress numbers shows a range from 0.208 to 0.258 MPa. It can be concluded that the stress number of indentation in the middle area is suitable with the results from FEA simulations, since it is still on their ranges.

Moreover, an indentation in the four edges has a range 1.97 HV to 2.44 HV and after conversion to the stress number, it has a range from 0.656 MPa to 0.813 MPa. If we look at the locations of indentation in four edges, they are in around green zones, which Ansys, Inventor, and Comsol show range from 0.6 to 1.051 MPa. By this number, it can conclude that the hardness Vicker's results is proved in that three software. In contrast, since the Abaqus input the material properties manually, it can be assumed that there are highly likely parameters that was not set as the other three software. Since the material properties in the library is all closed, we can only adjust based on general information of Ti6Al4V properties. In this study, Abaqus shows the result on green zone is extremely higher than other three software, which from 19.05 MPa to 37.84 MPa. The results from hardness Vicker's test in this study is not then comparable to the FEA simulation results using Abaqus.

The main objective of this study is to finding the VMS number using FEA simulation to prove if the intended load could withstand to the object. After that, since there are plenty of software are employed to run the FEA simulation, this study also tried to understand the correlation between one software to another one. By finishing the simulation and a comparative study, in this project, of hardness Vicker's test, it can be analyzed that simulation using Abaqus should be very careful to input the real material properties. One advantage is it could deliver a very precise result, particularly for having new development material properties. However, for general simulations, such as using commercial materials, in this study we use a well-known biocompatible material of Ti6Al4V, the results would be highly likely far different to other material-librabry-based software. Compared to other studies that also predict any further condition in medical application, the researchers also input manually the material properties [25]. In that case, the have two conditions of broken bones, so they differentiate the bone properties, particularly Young Modulus, mass density, and Poisson's ratio in those two conditions. By this comparison, Abaqus is actually more powerful to predict further condition based on very specific properties.

This study has several limitations regarding to the application of finite element method. First, the object of this study is only a small fragment of a personalized iliac bone implant, which in the future, the experiment could be extended for other implant objects. Second, the comparison method to the VMS is only based on Hardness Vicker's test results. Although a similar comparison study is hard to find, the VMS or other parameters on FEA simulation is actually could be explored through other methods, such as a tensile test or a pressure test using a sensor. Lastly, there is no post processing in the FEA simulation, since the results will directly appear as the stress number.

Overall, in the medical field, FEA is used to answer any further prediction after medical devices applied to the intended patient. This study proved that all finite element simulations using three software, particularly in Ansys, Inventor, and Comsol with the similar adjustment on the type of meshing process, close number of elements, unchanging placement of constraints and loads as well as material selection, do not result in significant differences. This study could conclude that all employed software in this study is equally powerful to provide stress number estimations that would be very impactful prior to the manufacturing process of any medical devices and designs. For further research, the finite element study could be functioned to predict other forthcoming physical conditions and prove it with associated experimental study.

4. Conclusion

This study proposes a comparison study between computational FEA (finite element analysis) and actual stress experiment with an object of a small fragment of a bone plate. In conclusion, the result from computational simulation and experiment are all similar in Ansys, Inventor, and Comsol. In the dark blue zone, the VMS (Von Misses stress) from FEA simulation reveals numbers with a range from 0.2 to 0.496 MPa. These ranges is comparable to the Hardness Vicker test result that after conversion shows a range between 0.208 and 0.258 MPa. The convenience using FEA software are more user-friendly in Ansys, Inventor, and Comsol since they already include material properties library and could directly apply for the constraint and load.

Acknowledgement

Authors greatly acknowledge National Research and Innovation Agency, Indonesia for resources support. The 3D titanium bone-plate was printed using funding from Health Research and Development Agency, Ministry of Health, Republic of Indonesia under scheme of IPTEKKES 2020. The usage of Ansys, Inventor, and Abaqus are free-student version, and Comsol license based on University of Melbourne.

References

[1] D. C. Ackland, D. Robinson, M. Redhead, P. V. S. Lee, A. Moskaljuk, and G. A. Dimitroulis, "Personalized 3D-printed prosthetic joint replacement for the human temporomandibular joint: From implant design to implantation," *Journal of Mechanical Behavior of Biomedical Materials*, vol. 69, pp. 404-411, 2017. Doi:10.1016/j.jmbbm.2017.01.048.

[2] A. A. Al-Tamimi, P. R. A. Fernandes, C. Peach, G. Cooper, C. Diver, and P. J. Bartolo, "Metallic bone fixation implants: a novel design approach for reducing the stress shielding phenomenon," *Virtual and Physical Prototyping*, vol. 12 no.2, pp.141-151, 2017. Doi:10.1080/17452759.2017.1307769.

[3] D. Wang, Y. Wang, S. Wu, H. Lin, Y. Yang, S. Fan, C. Gu, J. Wang, and C. Song, "Customized a Ti6Al4V bone plate for complex pelvic fracture by selective laser melting," *Materials*, vol. 10 no.1, pp. 35, 2017. Doi:10.3390/ma10010035.

[4] J. Rueber, R. Koehler, "Techniques for generating bone plate design" *United States Patent*: US10,595,942 B2, 2020.

[5] E. A. Lopez, K. Synder, "Variable angle bone plate" *United States Patent*: US10,624,686 B2, 2020.

[6] R. J. Mobbs, M. Coughlan, R. Thompson, C. E. Sutterlin, and K. Phan, "The utility of 3D printing for surgical planning and patient-specific implant design for complex spinal pathologies: Case report," *Journal of Neurosurgery*, vol. 26, no.4, pp. 513-518, 2017. Doi:10.3171/2016.9.SPINE16371.

[7] R. N. Maniar, and T. Singhi, "Patient specific implants: scope for the future," *Current Reviews Musculoskeletal Medicine*, vol. 7, no. 2, pp. 125-130, 2014. Doi:10.1007/s12178-014-9214-2.

[8] S. H. Abdullah, "Computational analysis for optimisation of baja SAE roll cage," *International Journal for Scientific Research and Development*, vol. 6 no. 4, pp. 1395-1399, 2018.

[9] S. Zandi, and M. Razaghi, "Finite element simulation of perovskite solar cell: A study on efficiency improvement based on structural and material modification,"

Solar Energy, vol. 179. pp. 298-306, 2019. Doi:10.1016/j.solener.2018.12.032.

[10] I. V. Antoniac, D. I. Stoia, B. Ghiban, C. Tecu, F. Miculescu, C. Vigaru, and V. Saceleanu, "Failure analysis of a humeral shaft locking compression plate-surface investigation and simulation by finite element method," *Materials*, vol. 12 no. 7, pp. 1128, 2019. Doi:10.3390/ma12071128.

[11] D. P. Malau, M. S. Utomo, D. Annur, T. Asmaria, Y. Prabowo, A. J. Rahyussalim, S. Supriadi, and M. I. Amal, "Finite element analysis of porous stemmed hip prosthesis for children," *AIP Conference Proceedings*, no. 2193, 2019. Doi:10.1063/1.5139393.

[12] G. S. Lewis, D. Mischler, H. Wee, J.S. Reid, and P. Varga, "Finite element analysis of fracture fixation," *Current osteoporosis reports*, vol. 19 no. 4, pp. 403-416, 2021. Doi:10.1007/s11914-021-00690-y.

[13] H. Wang, J. Liu, G. Wen, and Y. M. Xie, "The robust fail-safe topological designs based on the von Mises stress," *Finite Element in Analys and Design*, vol. 171, pp. 103376, 2020. Doi:10.1016/j.finel.2019.103376.

[14] F. A. Velázquez, R. C. Oyagüe, L. O. López, D. T. Lagares, A. M. González, A. P. Velasco, C. D. Lynch, J. G. Pérez, and M. S. Figallo, "Influence of bone quality on the mechanical interaction between implant and bone: A finite element analysis," *Journal of Dentistry*, vol. 88, pp. 103161, 2019. Doi:10.1016/j.jdent.2019.06.008.

[15] M. B. M. Salahuddin, A. F. Atikah, S. Rosnah, and M. N. M. Zuhair, "Conceptual design and finite element analysis of a high inclusion dough shaping machine using 3D-computer aided design (CAD) (solidworks)," *Materialwissenschaft undWerkstoftechnik*, vol. 50 no. 3, pp. 267-273, 2019. Doi:10.1002/mawe.201800205

[16] Y.W. Kwon, and H. Bang, "The finite element method using MATLAB," *CRC Press*, pp. 10-40, 2018.

[17] A. Muhammad, M. A. H. Ali, and I. H. Shanono, "Finite element analysis of a connecting rod in ANSYS: An overview," *IOP Conference Series: Materials Science and Engineering*, vol. 736 no. 2, pp. 022119, 2020. Doi:10.1088/1757-899X/736/2/022119.

[18] T. Jackson, A. Shenkin, A. Wellpott, K. Calders, N. Origo, M. Disney, A. Burt, P. Raumonen, B. Gardiner, M. Herold, T. Fourcaud, and Y. Malhi, "Finite element analysis of trees in the wind based on terrestrial laser scanning data," *Agricultural and Forest Meteorology*, vol. 265, pp. 137-144, 2019. Doi:10.1016/j.agrformet.2018.11.014

[19] D. Annur, M. S. Utomo, T. Asmaria, D. P. Malau, S. Supriadi, B. Suharno, A. J, Rahyussalim, Y. Prabowo, and M. I. Amal, "Material selection based on finite Element method in customized iliac implant," *Material Science Forum*, vol. 1000, pp. 82-89, 2020. Doi:10.4028/www.scientific.net/MSF.1000.82.

[20] T. Asmaria, D. A. Mayasari, S. Ramdhani, M. S. Utomo, D. P. Malau, D. Annur, M. I. Amal, and I. Kartika, "Finite element analysis of patient specific bone plate with Ti6Al4V material selection," *Jurnal Penelitian Fisika dan Aplikasinya*, vol. 11, no.1, pp. 83-93, 2021. Doi:10.26740/jpfa.v11n1.p83-93

[21] P. Zhang, S. X. Li, and Z. F. Zhang, "General relationship between strength and hardness," *Material Science and Engineering: A*, vol. 529, pp. 62-73, 2011. Doi:10.1016/j.msea.2011.08.061.

[22] J. Shen, T. Nagasaka, M. Tokitani, T. Muroga, R. Kasada, and S. Sakurai, "Effects of titanium concentration on microstructure and mechanical properties of high-purity vanadium alloys," *Materials & Design*, vol. 224, pp. 111390, 2022. Doi:10.1016/j.matdes.2022.111390

[23] M. Dalstra, and R. Huiskes, "Load transfer across the pelvic bone," *Journal of Biomechanics*, vol. 28, no. 6, pp. 715-724, 1995. Doi:10.1016/0021-9290(94)00125-N.

[24] A. R. Pramana, P. Marcián, L. Borák, N. Narra, T. Forouzanfar, and J. Wolff, "Structural and mechanical implications of PMMA implant shape and interface geometry in cranioplasty - A finite element study," *Journal of Cranio-Maxillofacial Surgery*, vol. 44, no. 1, pp. 34-44, 2016. Doi:10.1016/j.jcms.2015.10.014

[25] S. Mobasseri, B. Karami, M. Sadeghi, and A. Tounsi, "Bending and torsial rigidities of defected femur bone using finite element method," *Biomedical Engineering Advances*, vol. 3, pp. 100028, 2022. Doi:10.1016/j.bea.2022.100028.