Fatigue Analysis of Cannulated Pedicle Screw

Rosdi Daud[1], Y. Nurliyana[2], H. Mas Ayu[3] and A. Shah[4]

[1,2,3]Faculty of Mechanical & Automotive Engineering Technology , Universiti Malaysia Pahang, Pekan, Malaysia
[4]Faculty of Technical and Vocational ,Universiti Pendidikan Sultan Idris, Tanjung Malim, Malaysia

ABSTRACT
A cannulated pedicle screw (CPS) is a typical type of bone screw used to implant into a vertebral in the medical field. The purpose of these screws is to treat lumbar (lumbosacral) spine trauma. The screw is used to form spinal fusion transpedicle screw devices. Although the CPS is made by high strength material, the fatigue failure is still happened by time. Nevertheless the detail investigation on fatigue life cycle of screw also is lacking. This is maybe due to difficulty to investigate it by experimental (in vivo) since it is involved with human life. However, this paper focus on investigation of CPS fatigue life cycle using finite element method (FEM) since it is considered as acceptable method for biomechanics. By using the Ansys software as finite element method software, we can properly estimate the life span of the CPS. Based on the FE simulation results obtained, we found that our FE model is capable to predict fatigue life of CPS since the FE von mises stress result of our model only 7.1% difference with previous research result. Based on the prediction by FEM, the CPS life cycle is up to 3.1 years if the continuous load 11000 N is applied on the CPS by the time. Although our FE model is proven has potential in assisting CPS design, however in the future fabrication of the CPS and further testing needs to be conducted in order to evaluate this finding experimentally.

Keywords : *Cannulated pedicle screw, finite element method, fatigue analysis, life cycle.*

I. INTRODUCTION

Cannulated pedicle screw (CPS) is the instrumentation that has been introduced in medical field in order to stabilize the spine structure. Hollow screws are used for filling the wound with cement. Since the bone during drilling fulfills blood and other impurities, which could be the cause internal inflammation [1]. The main advantage of the CPS is the prevention of the pullout for osteoporotic cases. Two type of cannulated pedicle screw have been introduced which is cannulated pedicle screw with cement augmented and without cement augmented. Christodoulou et al. studied on parameters such type of cement, position and numbers of radial holes. They also investigated the effect of cement injection technique to the pullout performance [2]. Cement augmentation, or percutaneous vertebroplasty, is a minimally invasive treatment option that is reported to achieve pain relief and stabilization of the fracture site. According to Kiyak et al. expandable screws with cement augmentation had two or three times higher pullout strength than only solid cored screws for osteoporotic cases [3]. The cannulated pedicle screw can be considered as replace design of previous pedicle screw [4]. Although cannulated pedicle screw is considered as a goal standard for vertebral treatment, friction is still be re-ported happen at the screw and vertebral contact surface and also screw failure or fracture still happened due to the fatigue. Thus, the aim of this paper is to investigate the fatigue life cycle of CPS using finite element method.

II. METHODOLOGY

The pedicle screw was generated using Solidworks software. The current design dimension is based on previous study on the pedicle screw in order to validate the simulation result with the previous work result. Pedicle screw modelled in solid works consider diameter 6.0 mm and length 45 mm with titanium material shows in Fig. 1.

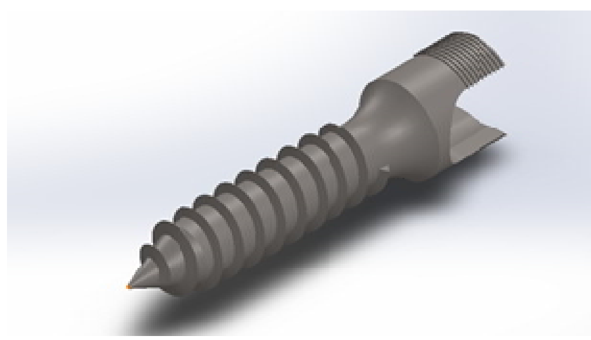

Fig. 1. 3D model of Cannulated Pedicle Screw

The material used in this study is titanium alloy (Grade 23) for pedicle screw. Titanium was once considered rare metal, but nowadays, it was used in varieties of medical application up to date [5,6]. Interest in the Titanium Alloy for total joint prostheses grew sharply in the United States toward the late 70s because due to its high strength, low elastic modulus and high corrosion resistance [6]. Titanium based alloys one of the ma-terial used perfectly as an implant due to the compatibility that it offers compare to other material such as the stainless steel. Table 1 shows the properties of titanium alloy.

Table 1. Properties of titanium alloy (Grade 23)

Properties	Magnitude
Young's modulus, E (Mpa)	110000
Tensile strength, σ_x (Mpa)	860.0
Yield stress, σ_y (Mpa)	820.0
Poisson's ratios, ν	0.342
Density (gcm^{-3})	4.43

Since this study was to analyze the fatigue lifetime of the screw, the analysis choice should be correct. As for the fatigue analysis, the selection of study in ANSYS was Static Structural. A static structural analysis that performed in ANSYS determines the displacements, stresses, strains, and forces in structures or parts. The path function under construction geometry tab was established after the material was assigned. Using the path function is intended to identify the results in only a certain path or lines. The path function can be chosen by right-clicking the construction geometry tab, insert then select path. A path can be defined in two principal ways, by start point and end point. Fig. 2 shows the path selection from point 1 to 2. A path is classified as a form of geometry of construction and is represented as a spatial curve to which the results of the path can be extended. The results are evaluated along this curve at dis-crete points. After selecting the path, the path type can be chosen either two points, edge or x-axis intersection, each having their own ways to select the pathways.

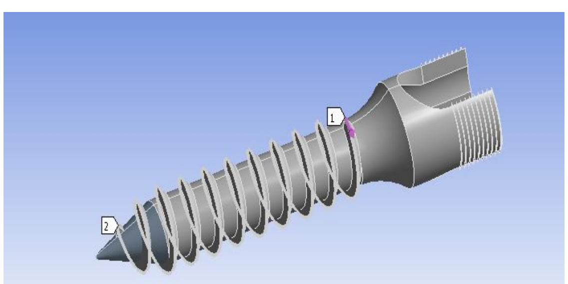

Fig. 2. Path selection from point 1 to 2

A. Mesh Properties

The sizing of the meshing is very important because by meshing process, the comput-er will be able to solve the mathematical matrix and accurately give answers which are the results. The whole-size control method was used for grid partition. Meshing is a process in which the structure / part is made up of many small pieces. In this paper, three-dimensional meshes generated at the pedicle screw using tetrahedral element for simulation. The element size has been taken 0.5 mm on the pedicle screw. As we know, The finer the meshing size the precise the value of the Equivalent Stress (Von Mises Stress) also the longer the duration of the analysis to finished. The meshing result was shown in Fig. 3.

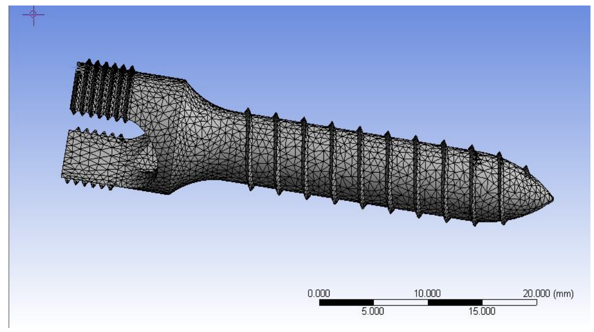

Fig. 3. Meshing result using element quality

According to the boundary condition of pedicle screw in real condition, the screw tip was completely fixed in all direction. Refer Fig. 4 for the fixed support that being place on the CPS model. The cannulated pedicle screw is loaded with a force parallel to the screw axis, locate at the top of the screw. During the simulations, the samples were loaded by this force 588.6 N for validation method. The pedicle screw then exerted with a force 490.5 N, 690 N and 790 N to see the stress distribution of screw. Then a new forces which are 7000 N, 8000 N, 9000 N, 10 000 N and 11 000 N were applied to predict the fatigue failure of pedicle screw. For the environment condition, the temperature is set 37°C following the normal body temperature and loading frequency was f1 = 24 Hz respectively f2 = 12 Hz. The result then being validated to the previous research result with the condition present of error between FEA and published journal should be lower than 15%. While, Fig. 5 shown the force applied to the screw.

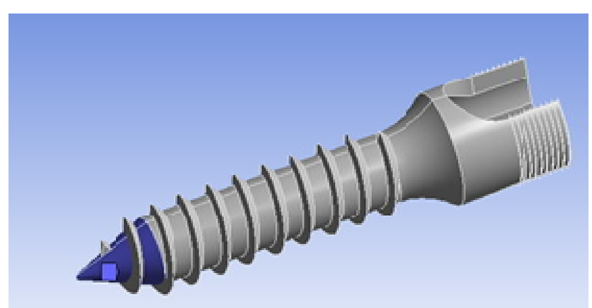

Fig. 4. Fixed support being place on the screw tip of the model

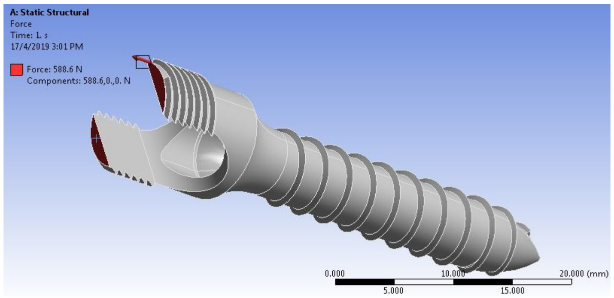

Fig. 5. Force applied on the pedicle screw

B. Finite Element Model Validation

Validation method is a technique for assessing how the result of a structural analy-sis (model) is correct. It is mainly used to estimate how accurately a predictive FE model will perform in practice. FE models must be validated and updated to assess the quality of FE models and increase confidence in the results as did by many previous researchers [7-11]. Screw diameter (D) and length (L) is 6.0 mm and 45 mm respectively. The tip of the CPS is fix and load was applied on the top surface of pedicle screw as done by previous researcher [12]. The load applied to the model is 588.6 N and 490.5 N as shown in Fig. 5.

We have considered magnitude of the forces 588.6 and 490.5 newton load for analysis based on previous research [12]. As for the material, the mechanical properties of titanium alloy grade 23 is used in pedicle screw which listed in Table 1 as has been carried out by Rishikant Sahani et. al [12].

C. Fatigue Failure Analysis

Screw breakage is mainly attributable to metal fatigue and cause screw breakage. Fatigue failure analysis is the crucial part in this study to determine the fatigue life of the cannulated pedicle screw. The fatigue life numbers will be predicted and calculated through S-N curve after FEM analyses under the given load. This test results in data presented as a plot of stress (S) against the number of cycles to failure (N), which is known as an S-N curve. Fig. 6 illustrates the inserting of fatigue tool in Ansys software. From the fatigue tools, life can be choose indicating the cycle of model after begin subjected to stress. The fatigue life was then calculated using relevant mathematical formula of S-N Curve. Lastly, user can set up the path that was mention in Fig. 2 onto the stress or life results that was desired to form the Stress over Life cycle graph.

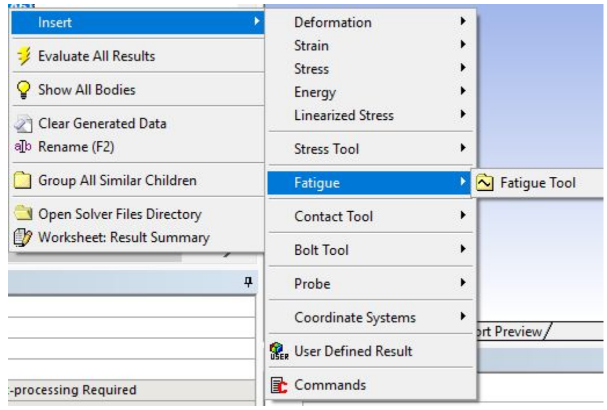

Fig. 6. Inserting of fatigue tool

III. RESULT AND DISCUSSION

Based on FE simulation results, the FE model and setup is considered valid since our FE simulation result different is less than 10% when compared to the FE simulation results obtained by previous researchers [12]. Table 2 shows the different between our FE simulation results and previous research results obtained. Our FE detail simulation results under 588.6 N and 490.5 N loads are shown in Fig. 7 and 8.

Load	588.60 N		490.5 N	
	Previous Research [12]	Our Research	Previous Research [12]	Our Research
Stress (Mpa)	354.43	330.66	295.36	275.55
Percentage of Error	7.1 %		7.1 %	

Table 2. FE Model validation

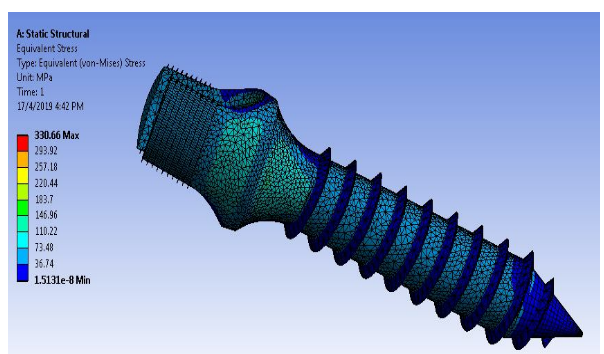

Fig. 7. Von misses stress of commercial design (588.6 N)

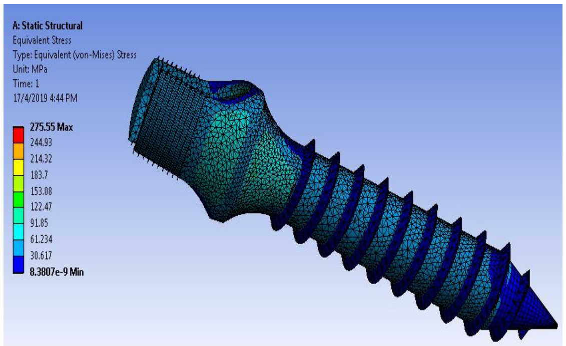

Fig. 8. Von mises stress of commercial design (490.5 N)

Regarding on fatigue analysis, our FE simulation which using the maximum load that cannulated pedicle screw (CPS) can stand, 11000N to investigate the fatigue life cycle of the CPS. Fig. 9 and 10 shows the FE simulation results in term of von mises stress and fatigue life of the CPS.

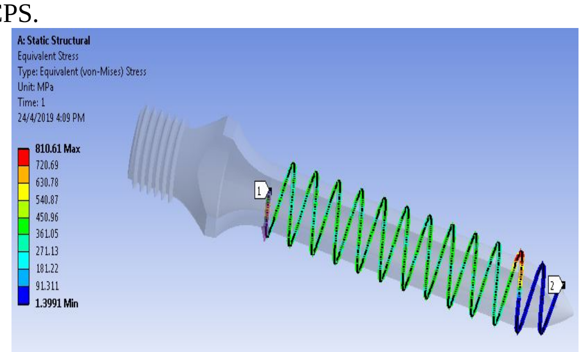

Fig. 9. Von Misses Stress on path for max load that the CPS can stand (11000 N)

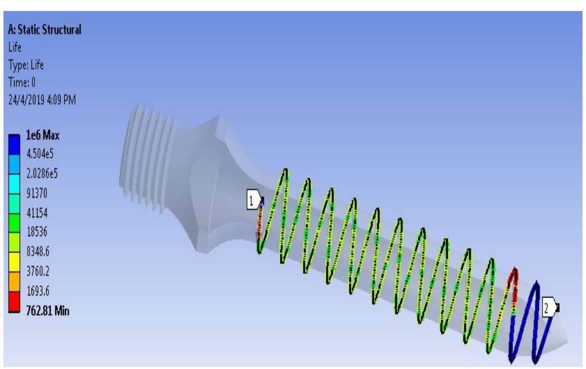

Fig. 10. Life Cycle on path for max load that the CPS can stand (11000 N)

Based on the S-N curve for cannulated pedicle screw (CPS) as shown in Fig. 11, the expected life cycle will be approximately 1×10^6cycles. This means that the current design of CPS will have a million cycles. If each movement use 100 seconds per cycle, a million cycles would be equivalent to 100 million seconds which is equal to 3.1 years

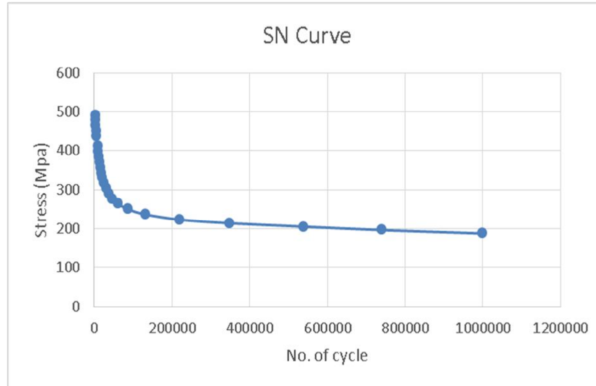

Fig. 11. S-N Curve for CPS life cycle

IV. CONCLUSION

We can conclude that our FE model is capable to predict fatigue life of CPS since the FE von mises stress result of our model only 7.1% difference with previous research result. Based on the prediction, the CPS life cycle is up to 3.1 years if the continuous load 11000 N is applied on the CPS by the time. Based on the FE result also, same as reported by previous researchers, we realized that the von misses stress give a signifi-cant contribution to the fatigue life cycle which is the higher the von misses stress, the lower the life cycle of CPS. However, in the future fabrication of the CPS and further testing needs to be conducted in order to evaluate this finding.

ACKNOWLEDGEMENT

We would like to thank Universiti Malaysia Pahang through research grant RDU190327 for fully support the facilities and resources for this research.

REFERENCES

[1] S. Major, V. Kocour and P. Cyrus. "*Fatigue life prediction of pedicle screw for spinal surgery*". Frattura ed Integrità Strutturale 35, 379 – 388, (2016).

[2] E. Christodoulou, S. Chinthakunta, D. Reddy, S. Khalil, T. Apostolou, P. Drees and K. Kafchitsas. "*Axial pullout strength comparison of different screw designs: fenestrated screw, dual outer diameter screw and standard pedicle screw*". Scoliosis 10(15), (2015)

[3] G. Kiyak, T. Balikci, A. M. Heydar and M. Bezer. "*Comparison of the Pullout Strength of Different Pedicle Screw Designs and Augmentation Techniques in an Osteoporotic Bone Model*". Asian Spine Journal 12(1), pp. 3 – 11, (2018).

[4] T. Demir.: "*Possible usage of cannulated pedicle screws without cement augmentation*". TOBB University of Economics and Technology, Ankara, Turkey. 149–155, (2014)

[5] Liu, X., Chu, P. K., & Ding, C. "*Surface modification of titanium, titanium alloys, and related materials for biomedical applications*". Materials Science and Engineering: R: Reports 47(3), 49-121. (2004).

[6] Wang, K. "*The use of titanium for medical applications in the USA*". Materials Science and Engineering: A 213(1), 134-137(1996).

[7] Anna G., Seungwon B., Brian P. Kelly, Neil R. Crawford. "*Effect of screw position on load transfer in lumbar pedicle screws: a non-idealized finite element analysis*". Computer Methods in Biomechanics and Biomedical Engineering 20(2), pp.182-192, (2016)

[8] F. A. Zakaria, R. Daud, H. Mas Ayu, S. H. Tomadi, M. S. Salwani, Mohammed Rafiq Abdul Kadir. "*The effect of position and different size of radial hole on performance of cannulated pedicle screw*". MATEC Web of Conferences 108, EDP Sciences, (2017).

[9] FT Pawi, Rosdi Daud, H Mas Ayu, Tedi Kurniawan, SH Tomadi, MS Salwani, A Shah. "*Design and analysis of lightweight polyetheretherketone (PEEK) front lower control arm*". AIP Conference Proceedings 2059(1), AIP Publishing, (2019).

[10] EL Sin Jian, Rosdi Daud, HM Ayu, MH bin Yusoff, J Jamiluddin, A Shah.: "*Application of a finite element method to predict fatigue life of the knee mobile bearing*". IOP Conference Series: Materials Science and Engineering 469(1), IOP Publishing, (2019)

[11] NMA Azam, Rosdi Daud, H Mas Ayu, J Ramli, MFB Hassan, A Shah, MAHM Adib. "*The Effect of Knee Flexion Angle on Contact Stress of Total Knee Arthroplasty*". MATEC Web of Conferences 225, EDP Sciences, (2018)

[12] R. Sahani. "*Finite element analysis of human lumbar vertebrae in pedicle screw fixation*". Department of Industrial Design National Institute of Technology Rourkela, India, (2015).