MATERIALS RESEARCH COMMUNICATIONS
Vol. 2, No. 1 (2021) (7 pages)
@ Pusat Publikasi Ilmiah ITS

FINITE ELEMENT ANALYSIS FOR DESIGN OF PLATE FOR FRACTURED FEMUR

MAS IRFAN P. HIDAYAT*

Department of Materials and Metallurgical Engineering,
Institut Teknologi Sepuluh Nopember,
Kampus ITS Keputih Sukolilo, 60111, Surabaya, East Java, Indonesia
irfan@mat-eng.its.ac.id

YULI SETIYORINI

Department of Materials and Metallurgical Engineering,
Institut Teknologi Sepuluh Nopember,
Kampus ITS Keputih Sukolilo, 60111, Surabaya, East Java, Indonesia
yuli_setyo@mat-eng.its.ac.id

BELLADINA CLARA SHINTA

Graduate Study Program Alumna, Department of Materials and Metallurgical Engineering,
Institut Teknologi Sepuluh Nopember,
Kampus ITS Keputih Sukolilo, 60111, Surabaya, East Java, Indonesia
bella_clara92@gmail.com

*** Corresponding author**

Received 31 August 2021; **Published** 30 November 2021

Abstract – In this paper, finite element analysis for design of plate for fractured femur is presented. The plates with different hole types are simulated. The types of hole are duo, flower and sloted forms and paired with a screw of 4.5 mm diameter. The plate and screw is made of SUS316L. A maximum load of 550 N is applied for each variation of plate design. The results are presented in terms of deformation and stress for both plate and screw. Numerical results show that the sloted plate is the best among the considered designs, from which the produced stress is below the material yield stress for almost all configurations.

Keywords: FE analysis; plate and screw; holes; fractured femur; bone fixation.

1. Introduction

Medical treatment and rehabilitation in the form of surgery is commonly performed to connect fractured bone by using plates and screws. It is important to rightly estimate the plate length and screws size in accordance with the dimensions of the bones. Too long plates will damage the bones, while the strength of the plates will be affected if they are too short. Hence, it is necessary to do simulation tasks in order to support the inquiry for optimum design for the fixation plate. To this end, numerical simulation has been proven to be a reliable approach in design and optimization tasks (Kaman et al., 2014; Hidayat et al., 2017), in particular finite element (FE) method (Syahroni and Hidayat, 2011; Maharaj et al., 2013; Waas et al., 2019).

2. Finite Element Analysis for Structural Problem

The structural static finite element equations can be written as follows:

$$[\mathrm{K}]\{U\} = \{F\} \tag{1}$$

Here, $\{U\}$ is the vector of nodal displacements, [K] is the stiffness matrix defined for an element by:

$$[k] = V^e [\mathrm{B}]^T [\mathrm{D}][\mathrm{B}] \tag{2}$$

[B] is the strain-displacement matrix, which is the matrix of derivatives of shape function, [D] is the matrix of material property and V^e is the volume of element.

In addition, $\{F\}$ is the vector of force values, which can be accumulative contribution from point, pressure and body loads.

In the FE analysis, the vector of nodal displacements and stress values at every step of analysis are updated as follows:

$$\{U\}^{i+1} = \{U\}^{i} + \{\Delta U\} \tag{3}$$

$$\{\sigma\}^{i+1} = \{\sigma\}^{i} + \{\Delta \sigma\} \tag{4}$$

from which deformation and stress of a structure can be acquired accordingly by using proper material constitutive relationship. Further aspects of stress analysis on a bone structure may be also referred in Al-Jassir et al. (2013) and Haase and Rouhi (2008).

3. Materials and Methods

Data of the femur bone used in this research is obtained from Oshkour et al. (2013). The data is used as a reference geometry to draw the femur CAD and subsequently imported into into ANSYS software. In addition, the input material is stainless steel 316L used for plate and screw. Elastic stress analysis is considered in this study. The yield strength (σ_{YS}) of 316L is 170 MPa.

Three types of hole to be drilled in the plate are examined in this study, designed as duo holes, flower holes and sloted holes, respectively. The hole types are in accordance with the standard instrument and implant by the AO Foundation (Colton and Orson, 2020). In this research, the hole diameter is varied between 3 and 5 mm. The number of holes is fixed to be eight holes, but the hole position filled by screw is varied, as shown in Fig. 1.

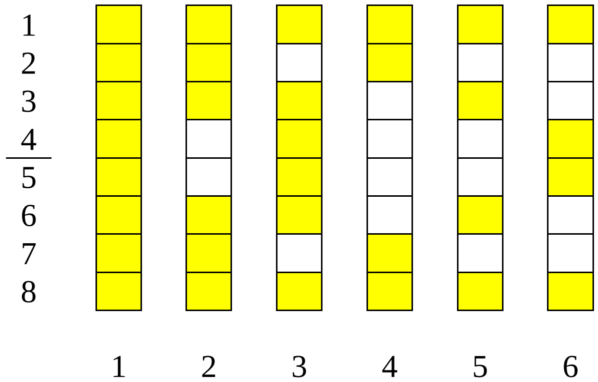

Fig. 1. Holes position filled by a screw in a plate. Screw position in white color.

Static simulation is carried out by applying a unidirectional load for the configuration of combined plate, screw and fractured bone. Fine mesh is used for each variation of the combined structures. The load is taken as F_y = -550 N. It is assumed from a total body weight of 110 kg, distributed equally to the right and left pelvis. Hence, this design can be used for a range of weights from 55 kg to 110 kg. The equivalent or von Misses stress is used to determine corresponding critical areas in the structure according to a certain safety factor N, given by:

$$\sigma_{eq} = \sqrt{\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2\right]} \tag{5a}$$

$$\sigma_{eq} \leq \frac{\sigma_{YS}}{N} \tag{5b}$$

An FE model consisting of the combined plate, screw and fractured bone is shown in Fig. 2.

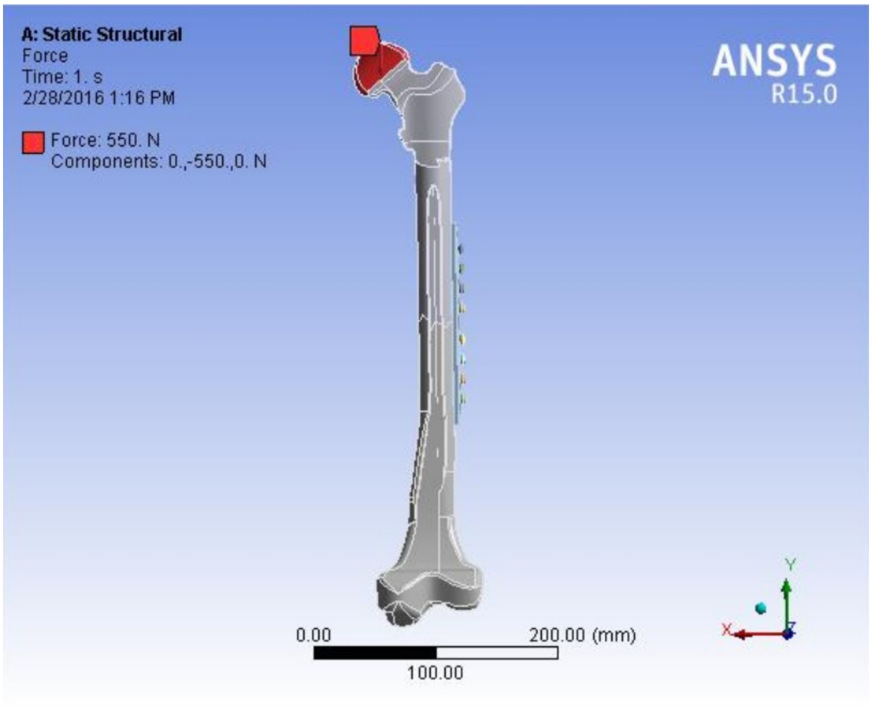

Fig. 2. Combined structure of plate, screw and fractured bone under a vertical load (weight load).

4. Results and Discussion

After many simulations, it has been observed that the screw diameter of 4.5 mm appears to be a suitable diameter value for the holes giving less stress concentration in the plate. Hence, only the corresponding results related to this diameter value are presented and discussed here.

Table 1 shows the produced stresses in each member of the integrated structure by referring to configurations shown in Fig. 1, where the duo holes are employed. As expected, a plate without screws (just in contact with the fractured bone) experienced less stress. The plate is only used for checking the consistency of simulation results and has no practical application value. On the other hand, the integrated structure of configuration 6 produces highest stress values at the plate, screw and bone, respectively. It is evident that as the holes in the plate are only partially filled with the screws, stress concentrations may arise at the screw locations.

Table 1. Produced maximum stresses in the plate with duo holes, screw and fracture bone, respectively

Configuration	Max. stress of plate (MPa)	Max. stress of screw (MPa)	Max. stress of bone (MPa)
1	54.477	183.380	41.187
2	57.618	185.910	45.186
3	63.535	212.380	50.635
4	65.834	209.890	48.974
5	70.314	224.400	53.100
6	70.715	240.520	56.269

In addition, body weight supposed to be supported by the plate and screw is not distributed evenly along the plate. It may also happen the possibility that the screw head does not attach perfectly on the plate section with hole so a high contact stress occurs (Chao et al., 2013).

For clarity of presentation, Fig. 3 shows the stress distribution in the plate with duo holes. It is clear that while the plate is safe, the screw section is not safe for all configurations as the produced maximum stress is above the yield strength of material. Clearly, the plate with duo holes is unlikely to be a preferred choice as it is not an optimum design for the bone fixation.

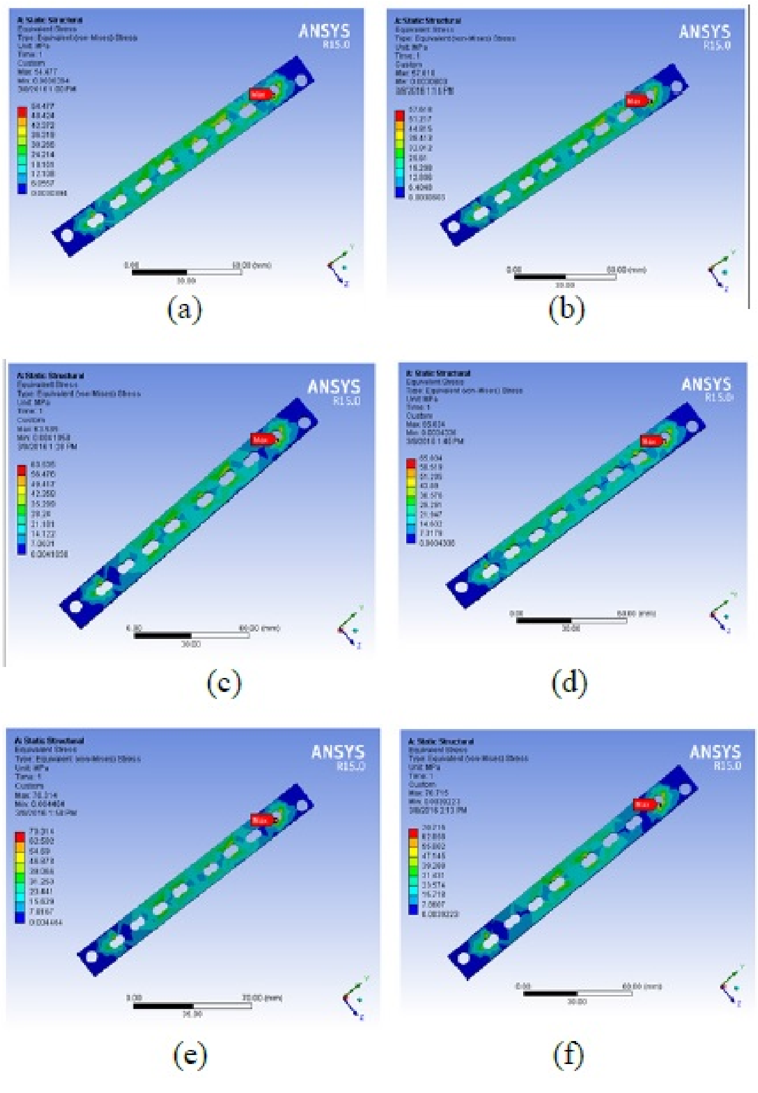

Fig. 3. Stress distribution in the plate with duo holes. Letters a-f are related to Configuration 1-6, respectively.

Subsequently, the plate with flower holes is examined as the next design. Table 2 presents the value of maximum stresses in the integrated structure. Also, Fig. 4 depicts stress distribution in the plate with flower holes. It is interesting to note that the simulation results are now in contrast to those of the plate with duo holes. It can be seen that the plate and screw sections are not safe for all configurations as the produced maximum stresses are all above the yield strength of material. Therefore, this design is also not an optimum design.

Table 2. Produced maximum stresses in the plate with flower holes, screw and fracture bone, respectively

Configuration	Max. stress of plate (MPa)	Max. stress of screw (MPa)	Max. stress of bone (MPa)
1	544.20	163.870	39.119
2	570.54	160.950	45.025
3	547.64	166.670	44.855
4	667.63	185.290	54.132
5	730.13	209.530	52.418
6	796.54	214.740	63.998

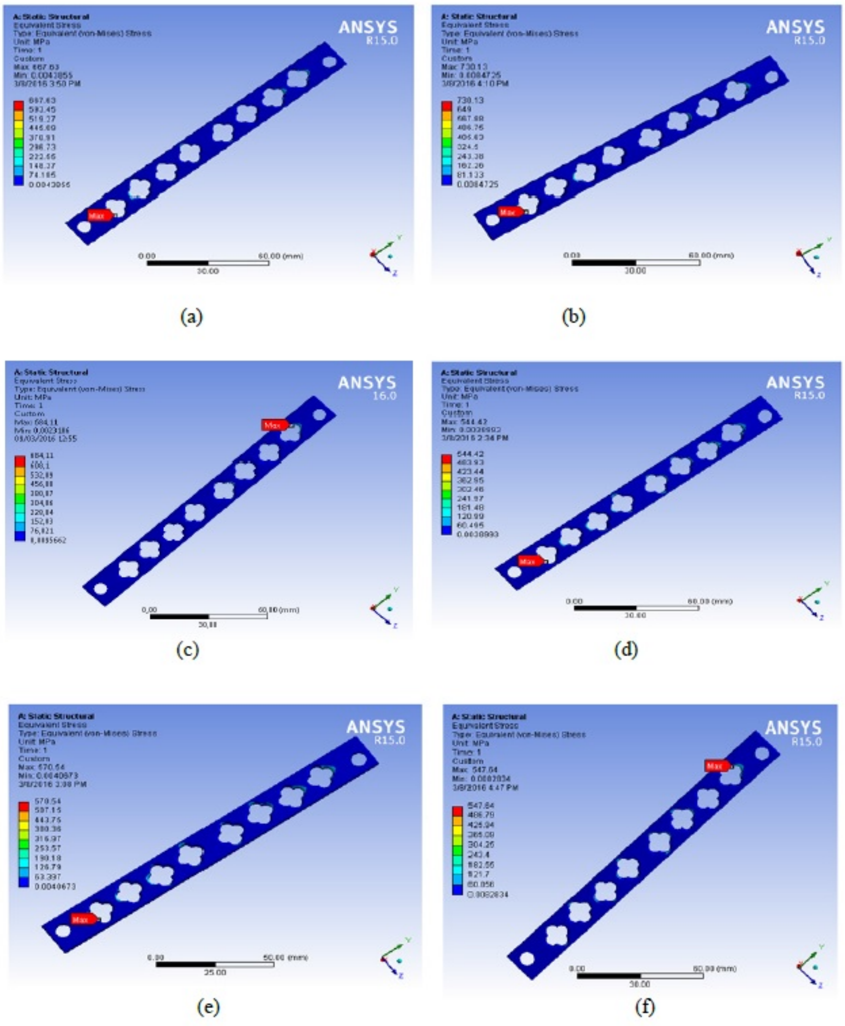

Fig. 4. Stress distribution in the plate with flower holes. Letters a-f are related to Configuration 1-6, respectively.

It appears that the plate with flower holes produces sites of high stress concentrations to the plate and screw, making it fails to be a suitable bone plate (Hohman et al., 2012).

Lastly, the plate with sloted holes is examined and compared with the previous designs. Table 3 shows the value of maximum stresses in the integrated structure. Further, Fig. 5 depicts stress distribution in the plate.

Table 3. Produced maximum stresses in the plate with sloted holes, screw and fracture bone, respectively

Configuration	Max. stress of plate (MPa)	Max. stress of screw (MPa)	Max. stress of bone (MPa)
1	124.870	126.360	45.374
2	129.150	131.890	48.308
3	146.050	156.120	54.189
4	141.970	145.790	53.748
5	153.020	165.220	58.551
6	155.350	184.890	63.603

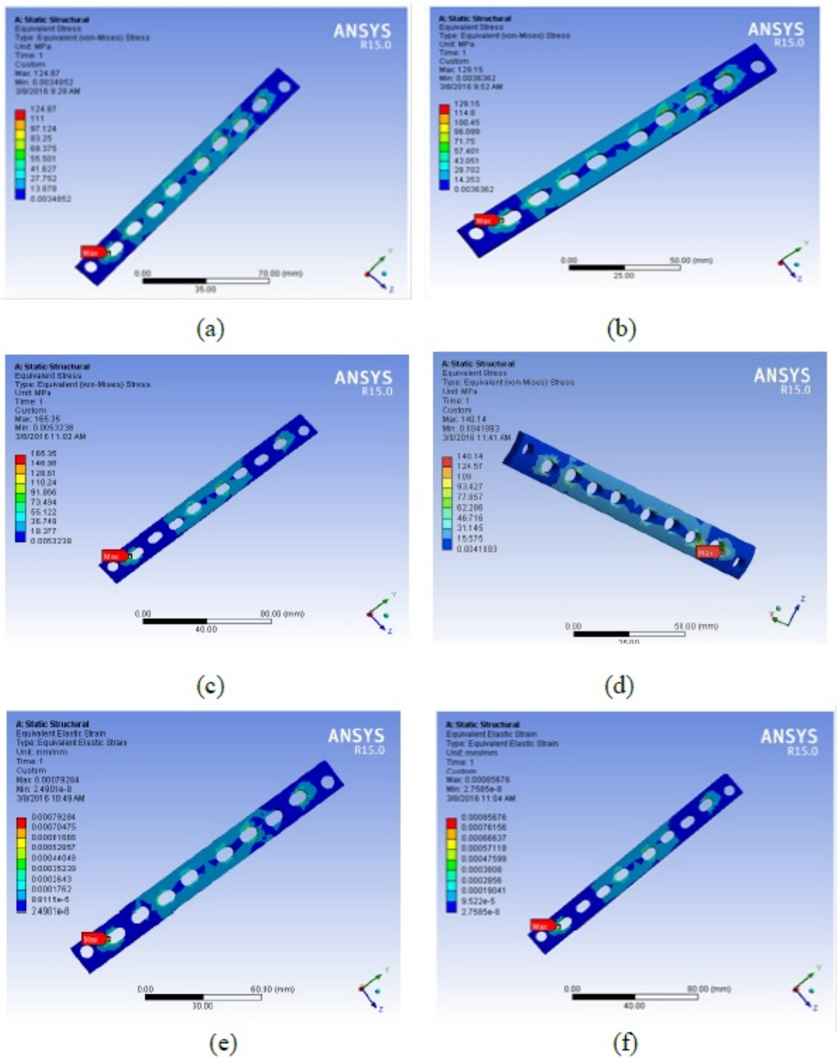

Fig. 5. Stress distribution in the plate with sloted holes. Letters a-f are related to Configuration 1-6, respectively.

It can be observed that the plate and screw sections are safe for all configurations with the produced maximum stresses are all below the yield strength of material, unless for the screw of Configuration 6. The results also show that the plate with sloted holes is best when used in an assembly with the screw and fractured bone. From the geometrical aspect, it appears that the plate with sloted holes give less sites of stress concentration as it has two sides with slightly enlarged section on one side. As the screw is placed at the smaller section side, the design thus allows the screw head to shift to the larger section side. Such design also allows the assembly to be well fitted/attached together, allowing the bone fractures or cracks to be better connected and closed, thus producing better healing (Weinans et al., 1992).

5. Conclusions

Finite element analysis for design of plate for fractured femur has been presented in this study. To achieve better assembly or integrated connection between the plate, screw and fractured bone without inducing high stress or pain, various designs of holes are simulated and examined i.e. duo, flower and sloted holes. Numerical results show that the plate with sloted holes is the best among considered designs in the present study, from which the produced stress is below the material yield stress for almost all configurations. The static stress analysis is therefore meaningful and helpful in designing plate, screw and types of hole for bone fixation of fractured femur. Optimization between material selection, geometry/design and safety factor would be the next research subject.

Acknowledgments

The present authors gratefully acknowledge the support from ITS Surabaya.

References

A.A. Oshkour, N.A. Osman, Y.H. Yau, F. Tarlochan and W.A.B. Wan Abas, Design of new generation femoral prostheses using functionally graded materials: a finite element analysis. Proceedings of the institution of mechanical engineers, part H. Journal of Engineering in Medicine 227(1) (2013) 3-17.

C. Colton and J. Orson, Plates─Form and function, AOT Trauma ORP Handout (2020) 1-12.

D.W. Hohman, J. Affonso, J. Budny and M.J. Anders, Calcaneal Plate Fixation of Distal Femoral Fractures, American Journal of Orthopedics 41(3) (2012) 140-143.

F.F. Al-Jassir, H. Fouad and O.Y. Alothman, In vitro assessment of Function Graded (FG) artificial Hip joint stem in terms of bone/cement stresses: 3D Finite Element (FE) study, BioMedical Engineering OnLine 12(5) 2013 1-17.

H. Weinans, R. Huiskes and H.J. Grootenboer, Effect of Material Properties of Femoral Hip Components on Bone Remodelling, Journal of Orthopaedic Research 10 (1992) 845-853.

K. Haase and G. Rouhi (2008). Finite Element Analysis of a Fracture Fixation Plate. *In Canadian Society for Mechanical Engineering Forum 2008*. Ottawa, Canada, 2008.

M. O. Kaman, N. Celik and S. Karakuzu, Numerical Stress Analysis of the Plates Used to Treat the Tibia Bone Fracture, Journal of Applied Mathematics and Physics 2 (2014) 304-309.

M.I.P. Hidayat, B. Ariwahjoedi, S. Parman and T.V.V.L.N. Rao, Meshless local B-spline collocation method for two-dimensional heat conduction problems with nonhomogenous and time-dependent heat sources, Journal of Heat Transfer 139(7) 2017 071302-1 - 071302-11.

N. Syahroni and M.I.P. Hidayat, Numerical Simulation of Welding Sequence Effect on Temperature Distribution, Residual Stresses and Distortions of T-Joint Fillet Welds, Advanced Materials Research 264-265 (2011) 254-259.

P. Chao, B.P. Conrad, D.D. Lewis, M. Horodyskiand A. Pozzi, Effect of plate working length on plate stiffness and cyclic fatigue life in a cadaveric femoral fracture gap model stabilized with a 12-hole 2.4 mm locking compression plate, BMC Veterinary Research 9(125) 2013 1-7.

P.S.R.S. Maharaj, R. Maheswaran and A. Vasanthanathana, Numerical Analysis of Fractured Femur Bone with Prosthetic Bone Plates, Procedia Engineering 64 (2013) 1242 – 1251.

V.D. Waas, M.I.P. Hidayat and L. Noerochim, Finite Element Simulation of Delamination in Carbon Fiber/Epoxy Laminate Using Cohesive Zone Model: Effect of Meshing Variation, Materials Science Forum 964 (2019) 257-262.