Jurnal Polimesin
Disseminating information on the research of mechanical engineering
Index: ICI, SINTA 2, Google Scholar ISSN Online: 2549-1199, ISSN Print: 1693-5462
Department of Mechanical Engineering, Lhokseumawe State Polytechnic, Lhokseumawe 24301, Indonesia



# #2961 Summary

SUMMARY REVIEW EDITING

## Submission

| | |
|---|---|
| Authors | Irza Sukmana |
| Title | Finite Element Analysis of Magnesium AZ31B Materials for Biodegradable Bone Screw Application |
| Original file | 2961-7513-2-SM.DOC 2022-06-04 |
| Supp. files | None |
| Submitter | Dr. Irza Sukmana |
| Date submitted | June 4, 2022 - 11:38 AM |
| Section | |
| Editor | Amir Mubarak |
| Abstract Views | 595 |

## Status

| | |
|---|---|
| Status | Published Vol 20, No 2 (2022): August |
| Initiated | 2022-08-31 |
| Last modified | 2022-09-07 |

## Submission Metadata

### Authors

| | |
|---|---|
| Name | Irza Sukmana |
| URL | https://www.researchgate.net/profile/Irza-Sukmana |
| Affiliation | Department of Mechanical Engineering, Faculty of Engineering, University of Lampung |
| Country | Indonesia |
| Bio Statement | |

**Assoc. Prof. Irza Sukmana, Ph.D.**

**Curriculum Vitae**

Doctor of Philosophy.

Department of Mechanical EngineeringFaculty of Engineering, Universitas Lampung

**Publikasi:**

**GOOGLE SCHOLAR, SINTA, SCOPUS**

Principal contact for editorial correspondence.

### Title and Abstract

Title: Finite Element Analysis of Magnesium AZ31B Materials for Biodegradable Bone Screw Application

Abstract: The bone implant functions for the load bearing and aims to support the integration of fractured bone. It may increase the strength of the broken bones and also at the same time support bone regeneration and integration. Bone screws are usually attached to the implant plate and bonded to the surface of the fractured bone by screwing the bone screws through the bone structure. In its implementation, a non-degradable implant needs a second operation for the patient to take out the implant. Currently, biomedical researchers are trying to produce bone implants that are degradable or bioresorbable materials. Magnesium (Mg) alloy is a potential biomaterial for bone implants, as Mg is a degradable material. Mg is one of the elements needed and harmless to the human body. This study focuses on finite element analysis (FEA) for the bone screw design of Magnesium alloy that has been well known as a potential candidate for biodegradable bone screw and plate application. The three dimensions (3D) design was done by using Solidworks software, and finite element analysis was performed using ANSYS by calculating the moment, pullout, and force bending received by bone screws. The validatation results of the design carried out with several analytical tests before the production of bone screws is proposed. The FEA simulation of bone screws pullout has a total deformation of 0.028 mm and a von Mises stress of 134.25 MPa for a pullout load of 1100 N. The bone screws torsion with a torque of 883 N.mm, the total deformation is 0.988 mm and for bone screws bending with a total deformation of 5,4352 mm has a von Mises stress of 25.706 MPa. AZ31B bone screws, based on the design, are safe and capable of handling the maximum load and deformation during the implantation. In vitro biocompatibility and in vivo studies is needed for further assessment of the design.

## Indexing

| | |
|---|---|
| Keywords | Magnesium AZ31B, Bone Screws, FEA, Pullout, Torsion, Bending. |
| Language | en |

## Supporting Agencies

| | |
|---|---|
| Agencies | — |

## References

References

[1] Stahel, P.F., Alfonso, N.A., Henderson, C., Baldini, T. Introducing The “ Bone-Screw-Fastener ” For Improved Screw Fixation In Orthopedic Surgery: A Revolutionary Paradigm Shift, Patient Saf Surg. 11 1–8. Doi:10.1186/S13037-017-0121-5. November 2017.
[2] Tilton, M., Armstrong, A.D, Wee, H., Hast, M.W., Manogharan, G., Lewis Gs. Finite element-predicted effects of screw configuration in proximal humerus fracture fixation. Journal of Biomechanical Engineering [Internet]. 2020 [cited 2021 Mar 25];142. Available from: https://doi.org/10.1115/1.4045907. March 2021.
[3] Chandra, G., & Pandey, A. (2021). Design and analysis of biodegradable buttress threaded screws for fracture fixation in orthopedics: a finite element analysis. Biomedical Physics & Engineering Express, 7(4), 045010.
[4] Wang, S., Wang, D., Lin, Y., Li, X. Biomechanical Analysis Of Femoral Fixation With A New Compression Plate Using Finite Element Method During Stair Climbing. Key Engineering Materials Submitted: 2015-11-24 Issn: 1662-9795, Vol. 693, Pp 297-305 Accepted: 2015-12-31.Doi:10.4028/Www.Scientific.Net/Kem.693.297 Online: 2016-05-20 2016 Trans Tech Publications, Switzerland. 2016.
[5] Ibrahim, F., & Sukmana, I. (2021). Application of magnesium alloys in orthopedic implant. Jurnal Polimesin, 19(2), 182-187.
[6] Catalog Depuy Synthes Instrument And Implants. 2011 http://synthes.com accessed on Tuesday, June 15, 2021 at 16.00 WIB.
[7] Emzain, Z. F., Huang, S. C., Yang, Y. S., & Qosim, N. Design and Analysis of a Dynamic Splint Based on Pulley Rotation for Post-Stroke Finger Extension Rehabilitation Device. Jurnal Rekayasa Mesin, 11(3), 477-485. 2020.
[8] Emzain, Z. F., Amrullah, U. S., & Mufarrih, A. M. (2020). Analisis elemen hingga untuk siklus berjalan pada model prostetik lentur pergelangan kaki. Jurnal Polimesin, 18(2), 91-98.
[9] Suhendrianto, S., Zuhri, S., Tarigan, H. A. N., & Asbar, A. (2021). Konsep desain dan perbaikan produk Ankle Foot Orthosis menggunakan metode kansei engineering. Jurnal Polimesin, 19(2), 140-146.
[10] Lee, E. S., Goh, T. S., Heo, J. Y., Kim, Y. J., Lee, S. E., Kim, Y. H., & Lee, C. S. Experimental evaluation of screw pullout force and adjacent bone damage according to pedicle screw design parameters in normal and osteoporotic bones. Applied Sciences, 9(3), 586. 2019.
[11] Witte, F. The history of biodegradable magnesium implants: A review. Acta Biomater. 6, 1680–1692. 2008.
[12] Biswas, J. K., Sahu, T. P., Rana, M., Roy, S., Karmakar, S.K., Majumder, S., & Roychowdhury, A. Design factors of lumbar pedicle screws under bending load: A finite element analysis. Biocybernetics and Biomedical Engineering. doi:10.1016/j.bbe.2018.10.003. January 2019.
[13] Tetteh, E., Mccullough, M.B. Impact Of Screw Thread Shape On Stress Transfer In Bone: A Finite Element Study, Computer Methods In Biomechanics And Biomedical Engineering, Pp. 1-6, 2020.
[14] Zain, N., Daud, R., Aziz, N., Ahmad, K., Ismail, A., Izzawati, B. Stress Analysis Prediction On Screw Orthopedic Implant In Trabecular Bone, Materials Today: Proceedings, Vol. 16, Pp. 1838-1845, 2019.
[15] Wang, W., Baran, G.R., Garg, H., Betz, R.R., Moumene, M., Cahill, P.J. The benefits of cement augmentation of pedicle screw fixation are increased in osteoporotic bone: a finite element analysis. Spine Deform;2(4):248–59. 2017.
[16] Mau, J. R., Hawkins, K. M., Woo, S. L. Y., Kim, K. E., & Mccullough, M. B. Design of a new magnesium-based anterior cruciate ligament interference screw using finite element analysis. Journal of Orthopaedic Translation, 20, 25-30. 2020.
[17] Tejszerska, D., Świtoński, E., & Gzik, M . (2011) Biomechanics of the human locomotor system. Publishing house.
[18] Wilkie, J., Docherty, P. D., & Moller, K. Stripping Torque Model for Bone Screws. 11th IFAC Symposium on Biological and Medical Systems BMS 2021 Ghent, Belgium, 19-22 September 2021. Volume 54, Issue 15, 2021, Pages 442-447. 2021.
[19] Silva, A. S., Mendes, J. M., Araujo, T., Aroso, C., & Barreiros, P. Comparison of Mechanical Resistance to Maximal Torsion Stress in Original and Nonoriginal or Compatible Prosthetic Implant Screws: An In Vitro Study. International Journal of Dentistry, 2021.
[20] Keshtiban, P.M., Regbat, M., & Keshtiban, M.M. An investigation of tensile strength of Ti6Al4V titanium screw inside femur bone using finite element and experimental tests. Journal of Computational Applied Mechanics, 51(1), 91-97. 2020.
[21] Xu, M., Yang, J., Lieberman, I. H., & Haddas, R. Finite element method-based study of pedicle screw–bone connection in pullout test and physiological spinal loads. Medical engineering & physics, 67, 11-21. 2019.
[22] Lee, E. S., Goh, T. S., Heo, J. Y., Kim, Y. J., Lee, S. E., Kim, Y. H., & Lee, C. S. Experimental evaluation of screw pullout force and adjacent bone damage according to pedicle screw design parameters in normal and osteoporotic bones. Applied Sciences, 9(3), 586. 2019.
[23] Moldovan, F., & Bataga, T. Torque Control during Bone Insertion of Cortical Screws. Procedia Manufacturing, 46, 484-490. 2020.

ABSTRACTING & INDEXING BY:

INFORMATION

» For Readers
» For Authors
» For Librarians

KEYWORDS

: Oil Treatment, Motor bensin, Viskositas, Aditif. Air cooling, fin lapping, thermal performance, photovoltaic, and simulation CFD CAD/CAE CD304SS, mechanical strength, transient deformation, high temperature, dislocation. Carbide cutting tool, TiAlN/Al2O3 & TiN, tool life, side wear (VB), Taylor Equation Characteristic, Filament 3D, 3D Printer, Simulation, and Experiment DOE Direct solar dryer, Indirect solar dryer FDM Handcycle, adjustable chair, stress analysis, disabilities Impact strength, composite, sugar palm fiber (SPF), polyester, fiber volume fraction Magnesium AZ31B, Bone Screws, FEA, Pullout, Torsion, Bending. Small axial pico-propeller turbine, runner blades, design, experiment test Solar drying Solar module, Rooftop solar power plants, On-grid, AMR kWh meter, kWh export-import Turpentine oil, Torque, Power, and Specific Fuel Consumption. coconut fiber human hair mechanical properties polyester volume fraction

Web Traffic Analysis

Visitors See
68,873 635 29
11,105 597 28
4,387 593 22
2,217 545 21
1,601 471 20
1,551 468 17
1,356 416 15
1,124 410 14
1,124 357 13
921 354 12
FLAG CO
00357836
Web Traffic Analysis

WRITING TOOLS

turnitin
MENDELEY
Mendeley Citation.pdf
grammarly
IEEE

JOURNAL HELP

Ciptaan disebarluaskan di bawah Lisensi Creative Commons Atribusi-BerbagiSerupa 4.0 Internasional.

**Mailing Address:**

**Politeknik Negeri Lhokseumawe**

Jl. Banda Aceh-Medan
Km. 280,3, Buketrata, Mesjid Punteut, Blang Mangat,
Kota Lhokseumawe, 24301

Propinsi Aceh,
Indonesia

# Review Form Response

Abstract (Abstrak) *

- ok

Introduction (Pendahuluan) *

- what is the problem with Bone Crews Biodegradable with Magnesium AZ31B so this needs to be optimized?
- the purpose of the research
- add international journal citations about this case
- can there be bio-corrosion in the use of Magnesium AZ31B material when implanted in the body? and affect its mechanical properties

Method (Metode) *

............ok

Result (Hasil) *

ok

Discussion (Pembahasan/ Diskusi) *

can there be bio-corrosion in the use of Magnesium AZ31B material when implanted in the body? and affect its mechanical properties

Conclusion (Kesimpulan) *

Conclusions must include all aspects of the research objectives. review the purpose of the research

Literature Cited (Kepustakaan) *

adjust to the number of citations

Notes/ Reason (Catatan) For Author: *

revision

# Optimization Design of Bone Crews Biodegradable with Magnesium AZ31B Based on Finite Element Analysis

…

Commented [M1]: good article, collaboration with foreign researchers is better


**Abstrack**

*Bone implant has a function for the load bearing and aims to support the integration of fractured bone. It may increases the strength of the broken bones and also in the same time support bone regeneration and integration. Bone bolts are usually attached to the implant plate and bonded to the surface of the fractured bone by screwing the bolt through the bone structure. In its implementation, a non-degradable implant that need the second operation for the patient to take out the implant. Currently, biomedical researchers are trying to produce bone implants that are degradable or bioresorbable materials. Magnesium (Mg) alloy is a potential biomaterial for bone implants, as Mg is a degradable material. Mg is one of the element that needed and harmless to the human body, light weight, and has good mechanical strength. This study aims to simulate the load on biodegradable bone screws of Magnesium AZ31B. The three dimensions (3D) design was done by using Solidworks software, and then the finite element analysis was perform using ANSYS by calculating the moment, pullout and force bending received by bone screws. The FEA simulation of bone screw pullout has a total deformation of 0.028 mm and a von misses stress of 134.25 MPa for a pullout load of 1100 N, then for bone screw torsion with a torque of 883 N.mm, the total deformation is 0.988 mm and for bone screw bending with total deformation of 5,4352 mm has a von misses stress of 25,706 MPa. The use of AZ31B bone screws based on the designed is safe and capable to handle the maximum load and deformation during the implantation, and it recommends for further in vitro biocompatibility and clinical testing*.



***Keywords****: Magnesium AZ31B, Bone Screw, FEA, Pullout, Torsion, Bending*.


## 1. Introduction

Bone bolts are used to increase strength and resistance to failure in the healing process of bone cracks. To optimize the design, which reduces the destructive effect and increases the efficiency of the implant, many studies use numerical simulations in combination with algorithm optimization. One of the main objectives of implantable components is to strengthen bones and reduce damage by using the taguchi method and the finite element method so as to optimize the design and engineering of bone screws [1]. In research conducted previously by tilton [2] using mechanical testing and simulation of elements to increase the bending strength of tibia bone screws, currently the use of titanium and stainless steel materials that are still often used but have shortcomings that is, the second operation was performed. ….

Commented [M2]: add international journal citations about this case

Magnesium material is one of the recommended materials for biomaterial base materials. The mechanical properties of magnesium and alloys have a low density (1.74-2.0 g/cm3) and a modulus of elasticity (41-45 GPa), Approaching the bone modulus [3], The biodegradebility of magnesium and its alloys in the human body environment avoids a second operation for temporary implants. The design of the bone bolts to be used refers to the Depuy Synthes Instruments and implants catalog [4], with specifications, sizes and types of materials. The use of biomaterials is not only used in the body but something that helps or is in contact with the human body aims to cure a malfunction of the body in humans such as cracked bone healing aids (bone bolts-plates), stroke disease therapy tools [15] and such glasses used for the therapy of eye diseases (cornea).

Commented [M3]: can there be bio-corrosion in the use of Magnesium AZ31B material when implanted in the body? and affect its mechanical properties

…..

Commented [M4]: what is the problem with Bone Crews Biodegradable with Magnesium AZ31B so this needs to be optimized?

…..

Commented [M5]: the purpose of the research

## 2. Materials and Methods

The research was conducted in March - August 2022, the design process and simulation of bone bolts were carried out at the Structural Mechanics Laboratory of the Department of Mechanical Engineering, University of Lampung in accordance with the catalog of Depuy Synthes Instruments and implants, then using magnesium AZ31B material in the form of a pejal cylinder with an initial diameter of 10 mm and a length of 130 mm. After preparing the tools and materials first carry out a 3-dimensional design process using solidworks then the FEA analysis process is carried out. The analysis process is carried out using ansys software with procedures such as input data engineering, import geometry, meshing, boundary conditions, pullout bone screw, torsion and bending force testing, so as to get FEA results in the form of total deformation and von misses stress.

### 2.1. Materials

Magnesium material is a light metal with a density of about 1.77 g/cm$^3$. The use of material in the FEA simulation of cortical bones is assumed to be normal cortical bones with a density of 1.61 g/cm$^3$ – 1.77 g/cm$^3$ [13]. Magnesium-based biomaterials have greater toughness compared to ceramic and polymer-based biomaterials because their elastic modulus is about 45 GPa. The modulus of elasticity of magnesium metal material has characteristics that are almost close to the nature of the modulus of elasticity of human bones, which is about 7-25 GPa [5] while bone bolts are made of Ti-6Al-4V material with modulus young and poisson ratios of 113 GPa and 0.3 [6]. This study used magnesium material AZ31B extrude rod with a diameter of 10 mm with a length of 130 mm. The tensile test graph and mechanical properties of magnesium material AZ31B with cortical bone are as follows figure 1.

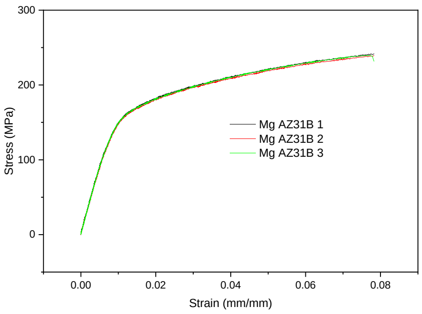



**Figure 1. Tensile test graph of stress vs strain**

**Table 1.** Mechanical properties Mg AZ31B and cortical bone

| No | Materials | Parameter | |
|---|---|---|---|
| 1 | Mg AZ31B | Yield Strength | 153,15 MPa |
| | | Modulus young | 44,28 GPa |
| | | Density | 1,77 g/cm$^3$ |
| | | Poisson Ratio | 0,35 |
| | | Ultimate | 258,44 MPa |
| 2 | Cortical bone | Modulus Young | 20 GPa |
| | | Density | 1,7 g/cm$^3$ |
| | | Poisson Ratio | 0,25 |

### 2.2. Analysis Methods

Development and production in bone bolt engineering where this research is expected to be in accordance with standard procedures and can be commercialized. In the method carried out the analysis process, conduct tensile testing (ASTM B557) to determine the ultimate tensile strength and modulus young, enter engineering data from the results of tensile testing of magnesium material AZ31B then simulate FEA bone screws torsion, bone screws pullout and bone screws bending then carry out mesh convergence. In the analysis with FEA, the results of total deformation, von misses stress and linearized von misses stress were obtained. The method used in conducting the analysis using ansys software is like figure 2.


**Figure 2. Flowchart of simulation methods**

### 3. Results and Discussion.

The results and discussions in the research that have been carried out are through several stages such as CAD design (solidworks), input engineering data (ultimate tensile strength, poisson ratio, density), determination of meshing, boundary conditions (bone screws torsion, bone screws pullout, bone screws bending testing force), solve and result (total deformation and von misses stress) are as follows:

### 3.1 Geometry CAD

Bone screws with a diameter of 5 mm, length of 40 mm, and pitch of 1 mm are the geometric dimensions of the cortical bone screws to be used. The bone screws in previous studies were modified to provide variations in design factors such as pitch length, main diameter, thread profile, and geometry. In biswas research [6], a total of 84 FE models were developed with seven pitch lengths (1, 1.5, 2, 2.5 ,3, 3.5 and 4 mm). Tetteh and McCullough [17] focused on the thread profile and thread shape effect on the transfer rate of stress distribution in the bone using finite element analysis. It was concluded that bone screws with a trapezoidal threaded shape are able to transfer higher stress to surface contact in cortical bones. Zain's research [18] studied the orthopedic stress distribution of screws implants in trabecular bones,

Commented [M6]: can there be bio-corrosion in the use of Magnesium AZ31B material when implanted in the body? and affect its mechanical properties which are very porous with biological tissues. In this study the shape of the screws, the profile of the thread, and its details are presented in figure 3.

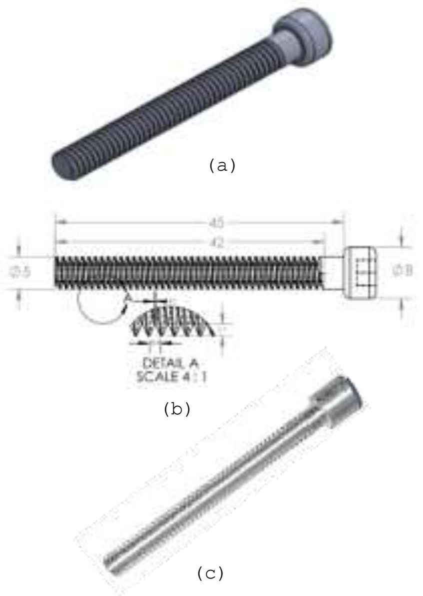



**Figure 3. (a) 3D geometry of bone screws, (b) Geometry of 2-D bone screws, (c) manufacturing of bone screws**

The geometric dimensions of bones are adjusted to human bones in general. For the step to be worked out, first make a hole with a hole diameter of 5 mm produced by drilling in the middle part of the bone prepared in such a way that the upper and lower parts of the bone can be drilled in the same direction. For the process of combining the bolts and bone plates to be attached can be described in figure 4.

**Figure 4. Assembly bolts and bone plates**

While in the geometry assembly of bone bolts, bone plates and cortical bones, there will be contact with surfaces such as plate contact with cortical bone, bone plate with bone bolts and bone bolts with cortical bone. For bone plate assembly, bone bolts and cortical bone as figure 5.


**Figure 5. Assembly of bone screws, bone plates and cortical bones**

### 3.2 Meshing

Meshing is performed to divide the model parts into small element parts. Meshing is useful for determining the distribution of a given stress. For the type of meshing used using tetrahedral elements because it is a type of meshing that is quite accurate for fairly complex geometry details. In the study conducted there were 3 parts, namely bone bolts and bone bolt assembly, bone plates with cortical bones as figured 6.

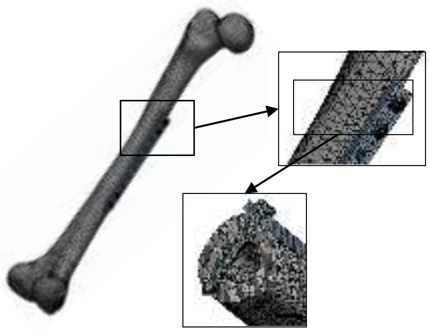


**Figure 6. Assembly element meshing**

**Table 2.** Mesh Convergence

| Simulasi FEA | Element | Total Deformasi (mm) | Von Misses Stress (MPa) |
|---|---|---|---|
| Bone Screw Torsion | 230247 | 0,988 | 16.984 |
| Bone Screw Pullout | 401119 | 0,028 | 134,25 |
| Bone Screw Bending Force | 230247 | 5,4352 | 25.706 |

For meshing, a 4-noded tetrahedron element is used and convergence analysis is performed by resetting the element size until an error of less than 3% [7]. Meanwhile, the research conducted mau [14] carried out meshing with tetrahedral elements as many as 140298 elements with an element size of 0.8 mm with almost the same design dimensions geometry.

### 3.3 Boundary Conditions

Boundary conditions were performed on FEA simulations of bone screws aimed at determining the limit conditions to be accepted by the predetermined bone bolt design. Boundary conditions are performed on bone screws with torsion conditions, pullout bone screws and bending bone screws. In the boundary conditions of bone screws pullout, FEA simulations use variations in the load that is assumed to be received by the human body, namely 700 N, 800 N, 900 N, 1000 N and 1100 N. With FEA simulation, it is able to find out the location of the maximum stress and deformation on the bone bolts. There is a series in carrying out boundary conditions, which is like figure 7 below.

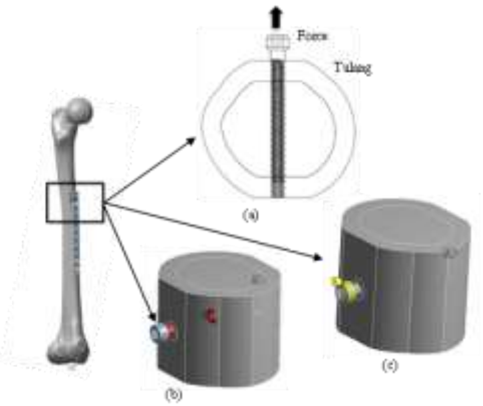



**Figure 7. Boundary Conditions Assembly Bone Plate Bolts, (a) Bone Screw Pullouts, (b) Bone Screw Torsions, (c) Bone Screw Bending**

In the FEA simulation, boundary conditions have been carried out with fix support on the bone while the force is given in the direction of the bone screws axis with tensile load conditions. While the Boundary condition is carried out with bone assembly and screws with engineering data according to the actual condition of both bone material and Mg AZ31B material. The torsion FEA simulation test was given moments of 883 N.mm [8], 706. 9 N.mm, 651.1 N.mm, 628.9 N.mm [9] clockwise and in bone screws bending boundary fixed condition support was given to the bone then loading by diplacement against the axis perpendicular to the bone screws. In the FEA simulation research that has been carried out, namely the x-axis: 5 mm, the y-axis: free and the z-axis: free.

### 3.3 Bone Screws Pullout

Bone screws pullout testing is a test with FEA simulation to determine the deformation and maximum load that a bolt can receive on a load in the direction of the bone screws. The bone screws pulllout image is like figure 8 as follows.

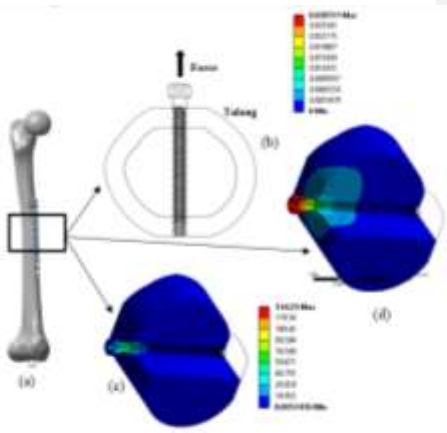



**Figure 8. (a). Cortical Bone (b). Pullout schematic, (c) Von Misses Stress, (d) Total Deformation**

Then after FEA simulations were carried out on bone screws pullouts total deformation of 0.028 mm and von misses stress of 134.25 MPa against pullout loads of 1100 N in FEA simulations, in the test the greater the load given by pullouts, the greater the deformation produced. The research conducted by Keshtiban [16] was obtained with a cortical screw FEA von misses stress of 449 MPa and a total deformation of 0.0624 mm with

Ti6Al4V material. The bone screw pullout chart is like figure 9 as follows.

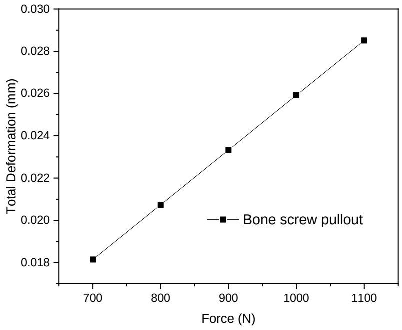



**Figure 9. Graph of total defromation to force variations**

In FEA simulations that the greater the pullout tensile load, the greater the deformation that the bone screws produce against the cortical bone. The pullout load on the normal cortical bone is 1450 N [10] while at a load of 1100 N with a total deformation of 0.2 mm it is made of stainless steel [11]. In addition to knowing the load given to the total deformation and von misses stress obtained, it is also necessary to know the stress distribution of bone screws in pullout conditions. This condition is carried out in order to be able to find out the stress distribution that occurs on the length of the bone screws. The linearized von misses stress condition against the length of the screws is as figured 10-11 as follows.

**Figure 10. Linearized equivalent stress bone screws pullout**

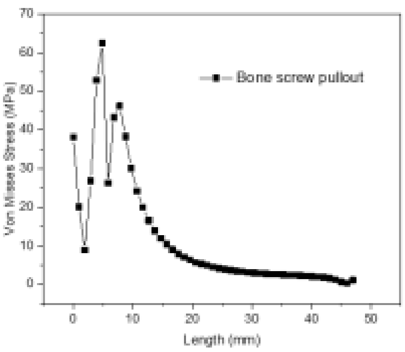



**Figure 11. Linearized von misses stress bone screws pullout graph**

In FEA testing, a graph of stress distribution over bone screws length was obtained. The stress distribution that occurs, namely on the screws head of 2.9376 mm, the stress occurs at 26,852 MPa then on the screws rod 4.896 mm experiences a maximum stress of 62,538 MPa, then the farther from the pullout condition, the stress drops more and lower at a distance of 46.023 mm has the lowest stress of 0.26643 MPa.

### 3.3 Bone Screws Torsion

After the boundary condition is carried out, the results of the FEA simulation of total deformation will be obtained and it appears that there is damage to the bone which resulted in the moment given. FEA moment simulation testing is carried out in order to find out the maximum moment given by the bolt until there is damage to the bone. From the research of Moldovan and Bataga [12] to determine the optimal torque, it refers to the graph of the torque variation curve such as threshold torque, peak clamping torque, peak failure torque. The chart is like figures 12-13.

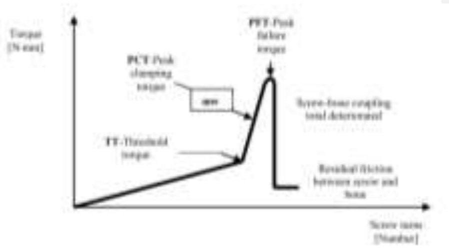


**Figure 12. Graph of variations in torque conditions**

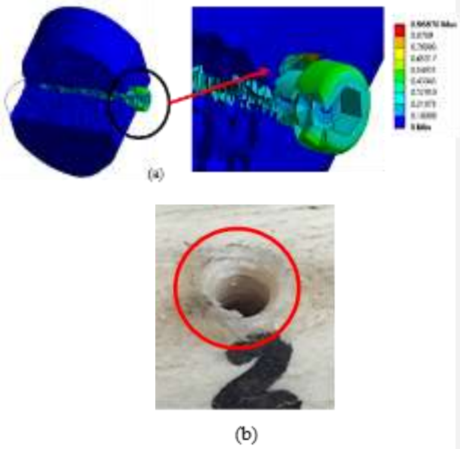



**Figure 13. Total Deformation (a). FEA, (b). Ekperimental [12]**

### 3.4 Bone Screws Bending

In bone screws bending testing aims to determine the maximum load that can be received by the bolt assembly and bone with a longitudinal loading perpendicular to the bone screws. From the boundary conditions given, then a FEA simulation is carried out, it will find out the deformation of the stress. The results of the FEA simulation are like figure 14 as follows.

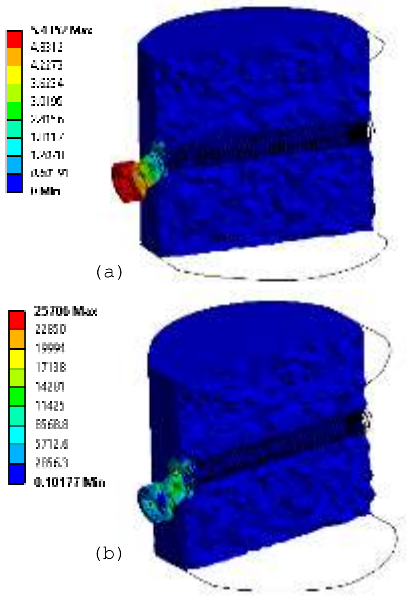



**Figure 14. (a). Total Deformation, (b). Von Misses Stress**

Then bone screws bending is given a conditions on the path along the screws to determine the stress distribution to the length of the bone screws. the results of linearized equivalent srtess bone screws bending are as figure 15 as follows.


**Figure 15. Linearized equivalent stress bone screws bending**

In a FEA simulation that has been done that the bone screws will deform when receiving the maximum load. The part of the bone screws that receives tension in the bone screws display is known to be linearized von misses stress. The graph of the stress that occurs along the bone screws is as figure 16 as follows.

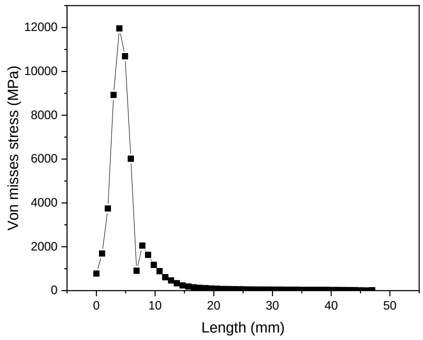



**Figure 16. Stress graph along bone screws**

From a linearized graph of von misses stress along bone screws that receive the load. The stress distribution that occurs, namely in the screws head of 2.9376 mm, the stress occurs at 8924.8 MPa (8.9248 GPa) then in the screws rod 4.896 mm it experiences a maximum stress of 11960 MPa (11,960 GPa), then the farther from the bending condition, the stress drops further at a distance of 46,023 mm has the lowest stress of 8.8482 MPa.

## 4. Conclusions.

In the FEA simulation test, the greater the tensile load given, the greater the deformation produced, then for the torsion bone screws with a torque of 883 N.mm the maximum deformation of 0.988 mm, and the bone screws bending the greater the deformation, the greater the deformation, it will produce a greater force reaction. From the research, it can be seen that by simulating the FEA method, it can minimize failures in designing before manufacturing, especially for the application of bone screws made from magnesium AZ31B.

Commented [M7]: Conclusions must include all aspects of the research objectives. review the purpose of the research

## References.


[1] Stahel, P.F., Alfonso, N.A., Henderson, C., Baldini, T. Introducing The " Bone-Screw-Fastener " For Improved Screw Fixation In Orthopedic Surgery : A Revolutionary Paradigm Shift, Patient Saf Surg. 11 1–8. Doi:10.1186/S13037-017-0121-5. November 2017.

[2] Tilton, M., Armstrong, A.D, Wee, H., Hast,

M.W., Manogharan, G., Lewis Gs. Finite element-predicted effects of screw configuration in proximal humerus fracture fixation. Journal of Biomechanical Engineering [Internet]. 2020 [cited 2021 Mar 25];142. Available from: https://doi.org/10.1115/1.4045907. March 2021.

[3] Wang, S., Wang, D., Lin, Y., Li, X. Biomechanical Analysis Of Femoral Fixation With A New Compression Plate Using Finite Element Method During Stair Climbing. Key Engineering Materials Submitted: 2015-11-24 Issn: 1662-9795, Vol. 693, Pp 297-305 Accepted: 2015-12-31.Doi:10.4028/Www.Scientific.Net/Kem.693.297 Online: 2016-05-20 2016 Trans Tech Publications, Switzerland. 2016.

[4] Catalog Depuy Synthes Instrument And Implants. 2011 http://synthes.com accessed on Tuesday, June 15, 2021 at 16.00 WIB.

[5] Witte, F. The history of biodegradable magnesium implants: A review. Acta Biomater. 6, 1680–1692. 2008.

[6] Biswas, J. K., Sahu, T. P., Rana, M., Roy, S., Karmakar, S.K., Majumder, S., & Roychowdhury, A. Design factors of lumbar pedicle screws under bending load: A finite element analysis. Biocybernetics and Biomedical Engineering. doi:10.1016/j.bbe.2018.10.003. January 2019.

[7] Wang, W., Baran, G.R., Garg, H., Betz, R.R., Moumene, M., Cahill, P.J. The benefits of cement augmentation of pedicle screw fixation are increased in osteoporotic bone: a finite element analysis. Spine Deform;2(4):248–59. 2017.

[8] Wilkie, J., Docherty, P. D., & Moller, K. Stripping Torque Model for Bone Screws. 11th IFAC Symposium on Biological and Medical Systems BMS 2021 Ghent, Belgium, 19-22 September 2021. Volume 54, Issue 15, 2021, Pages 442-447. 2021.

[9] Silva, A. S., Mendes, J. M., Araujo, T., Aroso, C., & Barreiros, P. Comparison of Mechanical Resistance to Maximal Torsion Stress in Original and Nonoriginal or Compatible Prosthetic Implant Screws: An In Vitro Study. International Journal of Dentistry, 2021.

[10] Xu, M., Yang, J., Lieberman, I. H., & Haddas, R. Finite element method-based study of pedicle screw–bone connection in pullout test and physiological spinal loads. Medical engineering & physics, 67, 11-21. 2019.

[11] Lee, E. S., Goh, T. S., Heo, J. Y., Kim, Y. J., Lee, S. E., Kim, Y. H., & Lee, C. S. Experimental evaluation of screw pullout force and adjacent bone damage according to pedicle screw design parameters in normal and osteoporotic bones. Applied Sciences, 9(3), 586. 2019.

[12] Moldovan, F., & Bataga, T. Torque Control during Bone Insertion of Cortical Screws. Procedia Manufacturing, 46, 484-490. 2020.

[13] Lee, E. S., Goh, T. S., Heo, J. Y., Kim, Y. J., Lee, S. E., Kim, Y. H., & Lee, C. S. Experimental evaluation of screw pullout force and adjacent bone damage according to pedicle screw design parameters in normal and osteoporotic bones. Applied Sciences, 9(3), 586. 2019.

[14] Mau, J. R., Hawkins, K. M., Woo, S. L. Y., Kim, K. E., & Mccullough, M. B. Design of a new magnesium-based anterior cruciate ligament interference screw using finite element analysis. Journal of Orthopaedic Translation, 20, 25-30. 2020.

[15] Emzain, Z. F., Huang, S. C., Yang, Y. S., & Qosim, N. Design and Analysis of a Dynamic Splint Based on Pulley Rotation for Post-Stroke Finger Extension Rehabilitation Device. Jurnal Rekayasa Mesin, 11(3), 477-485. 2020.

[16] Keshtiban, P.M., Regbat, M., & Keshtiban, M.M. An investigation of tensile strength of Ti6Al4V titanium screw inside femur bone using finite element and experimental tests. Journal of Computational Applied Mechanics, 51(1), 91-97. 2020.

[17] Tetteh, E., Mccullough, M.B. Impact Of Screw Thread Shape On Stress Transfer In Bone: A Finite Element Study, Computer Methods In Biomechanics And Biomedical Engineering, Pp. 1-6, 2020.

[18] Zain, N., Daud, R., Aziz, N., Ahmad, K., Ismail, A., Izzawati, B. Stress Analysis Prediction On Screw Orthopedic Implant In Trabecular Bone, Materials Today: Proceedings, Vol. 16, Pp. 1838-1845, 2019.

# Review Form Response

Abstract (Abstrak) *

- In the paper title mentions optimization, what optimization method is used?
- What parameter to describe the degradable material? And how does it relate to the research results obtained, namely deformation and stress?
- It is necessary to explain why only bone screws is analyzed, even though there are three components involved where the strength is lower than bone screws, especially cortical bone.

Introduction (Pendahuluan) *

How is the correlation between the literature cited in the introduction with the research conducted so that the research contribution can be shown.

Method (Metode) *

- The modeling used is static or dynamic method?
- What modeling material is used?, Elastic, elastic-plastic, viscoelastic or something else? Do Bone Screw, Cortical Bone and plate use the same material model?
- The stated convergence is written from the literature. What is the actual convergence result of the modeling? Convergence based on number of elements or size of elements?
- What type of contact between bone screw – plate, plate-cortical bone and bone screw-cortical plate?
- Models are given various loads. The variation of the load describes the load according to age, weight of people or something else?

Result (Hasil) *

Considering that the research conducted did not use experiments, convergence used to verificate the modeling result. The claimed convergence of less than 3 percent is obtained from what number of elements? Is there a convergence graph?
Acctually, verification of modeling results using convergence is not sufficient for 3D elements. The accuracy of 3D modeling depends not only on the number of elements but also on the shape of the elements.

Discussion (Pembahasan/ Diskusi) *

Discussion of the results does not only comment on the magnitude of deformations and stresses but it is necessary to discuss the type and position of failure for each type loading.

Conclusion (Kesimpulan) *

In this section it is necessary to show what is difference from previous work on this area. Provide relevant references

Literature Cited (Kepustakaan) *

- At point 2$^{nd}$. There is still a word in Indonesian "pejal"
- In 7$^{th}$ and 8$^{th}$ pictures, the description of the picture still has the Indonesian word "tulang"
- Literature citation writing based on bibliographic number. In the Introduction, after 4$^{th}$ citation number, next citation jumps to the 15$^{th}$ citation. It should be written in order.

Notes/ Reason (Catatan) For Author: *

# Optimization Design of Bone Crews Biodegradable with Magnesium AZ31B Based on Finite Element Analysis

**Abstract**

*Bone implant has a function for the load bearing and aims to support the integration of fractured bone. It may increases the strength of the broken bones and also in the same time support bone regeneration and integration. Bone bolts are usually attached to the implant plate and bonded to the surface of the fractured bone by screwing the bolt through the bone structure. In its implementation, a non-degradable implant that need the second operation for the patient to take out the implant. Currently, biomedical researchers are trying to produce bone implants that are degradable or bioresorbable materials. Magnesium (Mg) alloy is a potential biomaterial for bone implants, as Mg is a degradable material. Mg is one of the element that needed and harmless to the human body, light weight, and has good mechanical strength. This study aims to simulate the load on biodegradable bone screws of Magnesium AZ31B. The three dimensions (3D) design was done by using Solidworks software, and then the finite element analysis was perform using ANSYS by calculating the moment, pullout and force bending received by bone screws. The FEA simulation of bone screw pullout has a total deformation of 0.028 mm and a von misses stress of 134.25 MPa for a pullout load of 1100 N, then for bone screw torsion with a torque of 883 N.mm, the total deformation is 0.988 mm and for bone screw bending with total deformation of 5,4352 mm has a von misses stress of 25,706 MPa. The use of AZ31B bone screws based on the designed is safe and capable to handle the maximum load and deformation during the implantation, and it recommends for further in vitro biocompatibility and clinical testing.*

***Keywords**: Magnesium AZ31B, Bone Screw, FEA, Pullout, Torsion, Bending.*

## 1. Introduction

Bone bolts are used to increase strength and resistance to failure in the healing process of bone cracks. To optimize the design, which reduces the destructive effect and increases the efficiency of the implant, many studies use numerical simulations in combination with algorithm optimization. One of the main objectives of implantable components is to strengthen bones and reduce damage by using the taguchi method and the finite element method so as to optimize the design and engineering of bone screws [1]. In research conducted previously by tilton [2] using mechanical testing and simulation of elements to increase the bending strength of tibia bone screws, currently the use of titanium and stainless steel materials that are still often used but have shortcomings that is, the second operation was performed.

Magnesium material is one of the recommended materials for biomaterial base materials. The mechanical properties of magnesium and alloys have a low density (1.74-2.0 g/cm3) and a modulus of elasticity (41-45 GPa), Approaching the bone modulus [3], The biodegradebility of magnesium and its alloys in the human body environment avoids a second operation for temporary implants. The design of the bone bolts to be used refers to the Depuy Synthes Instruments and implants catalog [4], with specifications, sizes and types of materials. The use of biomaterials is not only used in the body but something that helps or is in contact with the human body aims to cure a malfunction of the body in humans such as cracked bone healing aids (bone bolts-plates), stroke disease therapy tools [15] and such glasses used for the therapy of eye diseases (cornea).

## 2. Materials and Methods

The research was conducted in March - August 2022, the design process and simulation of bone bolts were carried out at the Structural Mechanics Laboratory of the Department of Mechanical Engineering, University of Lampung in accordance with the catalog of Depuy Synthes Instruments and implants, then using magnesium AZ31B material in the form of a pejal cylinder with an initial diameter of 10 mm and a length of 130 mm. After preparing the tools and materials first carry out a 3-dimensional design process using solidworks then the FEA analysis process is carried out. The analysis process is carried out using ansys software with procedures such as input data engineering, import geometry, meshing, boundary conditions, pullout bone screw, torsion and bending force testing, so as to get FEA results in the form of total deformation and von misses stress.

### 2.1. Materials

Magnesium material is a light metal with a density of about 1.77 g/cm$^3$. The use of material in the FEA simulation of cortical bones is assumed to be normal cortical bones with a density of 1.61 g/cm$^3$ – 1.77 g/cm$^3$ [13]. Magnesium-based biomaterials have greater toughness compared to ceramic and polymer-based biomaterials because their elastic modulus is about 45 GPa. The modulus of elasticity of magnesium metal material has characteristics that are almost close to the nature of the modulus of elasticity of human bones, which is about 7-25 GPa [5] while bone bolts are made of Ti-6Al-4V material with modulus young and poisson ratios of 113 GPa and 0.3 [6]. This study used magnesium material AZ31B extrude rod with a diameter of 10 mm with a length of 130 mm. The tensile test graph and mechanical properties of magnesium material AZ31B with cortical bone are as follows figure 1.

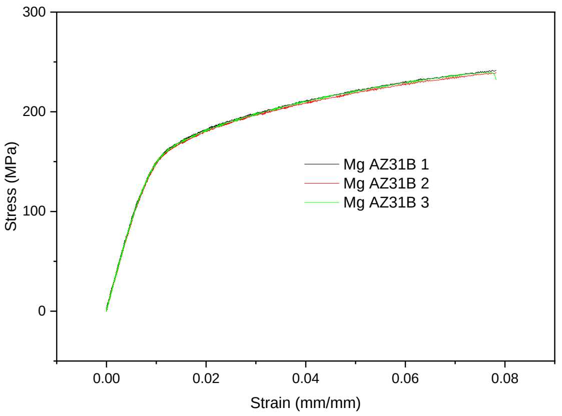



**Figure 1. Tensile test graph of stress vs strain**

**Table 1.** Mechanical properties Mg AZ31B and cortical bone

| No | Materials | Parameter | |
|---|---|---|---|
| 1 | Mg AZ31B | Yield Strength | 153,15 MPa |
| | | Modulus young | 44,28 GPa |
| | | Density | 1,77 g/cm$^3$ |
| | | Poisson Ratio | 0,35 |
| | | Ultimate | 258,44 MPa |
| 2 | Cortical bone | Modulus Young | 20 GPa |
| | | Density | 1,7 g/cm$^3$ |
| | | Poisson Ratio | 0,25 |

### 2.2. Analysis Methods

Development and production in bone bolt engineering where this research is expected to be in accordance with standard procedures and can be commercialized. In the method carried out the analysis process, conduct tensile testing (ASTM B557) to determine the ultimate tensile strength and modulus young, enter engineering data from the results of tensile testing of magnesium material AZ31B then simulate FEA bone screws torsion, bone screws pullout and bone screws bending then carry out mesh convergence. In the analysis with FEA, the results of total deformation, von misses stress and linearized von misses stress were obtained. The method used in conducting the analysis using ansys software is like figure 2.

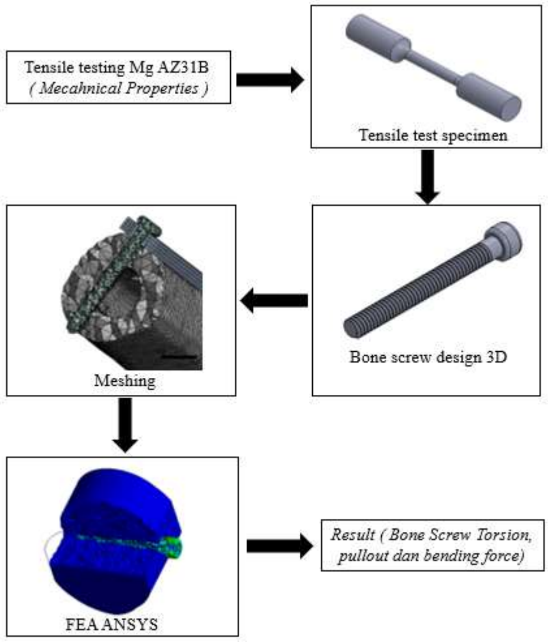



**Figure 2. Flowchart of simulation methods**

## 3. Results and Discussion.

The results and discussions in the research that have been carried out are through several stages such as CAD design (solidworks), input engineering data (ultimate tensile strength, poisson ratio, density), determination of meshing, boundary conditions (bone screws torsion, bone screws pullout, bone screws bending testing force), solve and result (total deformation and von misses stress) are as follows:

### 3.1 Geometry CAD

Bone screws with a diameter of 5 mm, length of 40 mm, and pitch of 1 mm are the geometric dimensions of the cortical bone screws to be used. The bone screws in previous studies were modified to provide variations in design factors such as pitch length, main diameter, thread profile, and geometry. In biswas research [6], a total of 84 FE models were developed with seven pitch lengths (1, 1.5, 2, 2.5 ,3, 3.5 and 4 mm). Tetteh and McCullough [17] focused on the thread profile and thread shape effect on the transfer rate of stress distribution in the bone using finite element analysis. It was concluded that bone screws with a trapezoidal threaded shape are able to transfer higher stress to surface contact in cortical bones. Zain's research [18] studied the orthopedic stress distribution of screws implants in trabecular bones, which are very porous with biological tissues. In this study the shape of the screws, the profile of the thread, and its details are presented in figure 3.

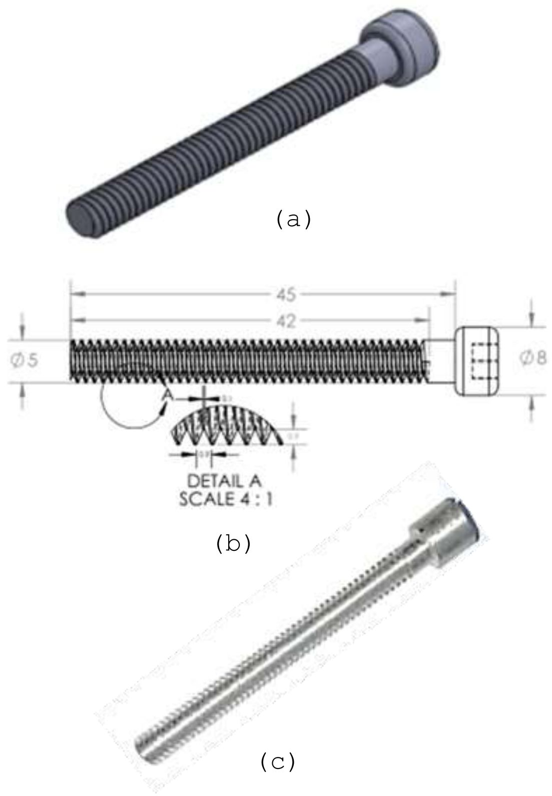



**Figure 3. (a) 3D geometry of bone screws, (b) Geometry of 2-D bone screws, (c) manufacturing of bone screws**

The geometric dimensions of bones are adjusted to human bones in general. For the step to be worked out, first make a hole with a hole diameter of 5 mm produced by drilling in the middle part of the bone prepared in such a way that the upper and lower parts of the bone can be drilled in the same direction. For the process of combining the bolts and bone plates to be attached can be described in figure 4.

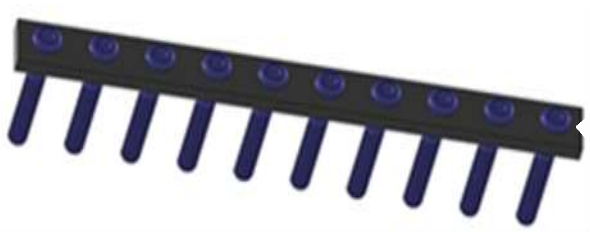


**Figure 4. Assembly bolts and bone plates**

While in the geometry assembly of bone bolts, bone plates and cortical bones, there will be contact with surfaces such as plate contact with cortical bone, bone plate with bone bolts and bone bolts with cortical bone. For bone plate assembly, bone bolts and cortical bone as figure 5.

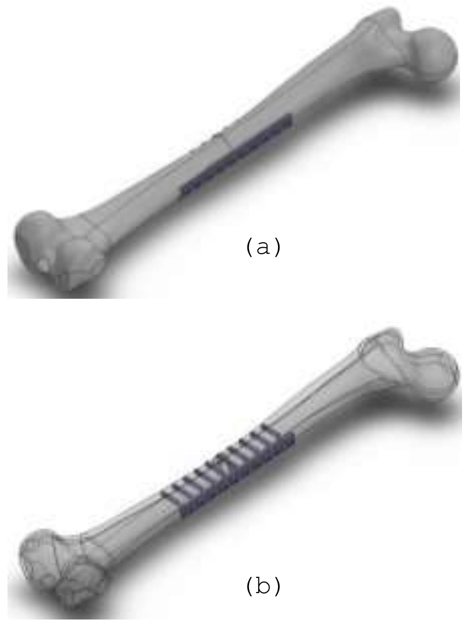



**Figure 5. Assembly of bone screws, bone plates and cortical bones**

### 3.2 Meshing

Meshing is performed to divide the model parts into small element parts. Meshing is useful for determining the distribution of a given stress. For the type of meshing used using tetrahedral elements because it is a type of meshing that is quite accurate for fairly complex geometry details. In the study conducted there were 3 parts, namely bone bolts and bone bolt assembly, bone plates with cortical bones as figured 6.

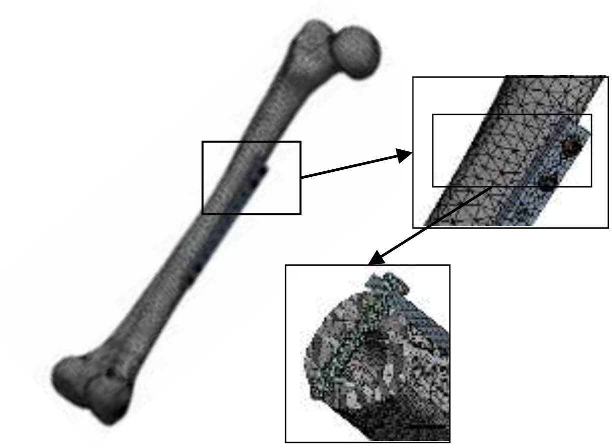


**Figure 6. Assembly element meshing**

**Table 2.** Mesh Convergence

| Simulasi FEA | Element | Total Deformasi (mm) | Von Misses Stress (MPa) |
|---|---|---|---|
| Bone Screw Torsion | 230247 | 0,988 | 16.984 |
| Bone Screw Pullout | 401119 | 0,028 | 134,25 |
| Bone Screw Bending Force | 230247 | 5,4352 | 25.706 |

For meshing, a 4-noded tetrahedron element is used and convergence analysis is performed by resetting the element size until an error of less than 3% [7]. Meanwhile, the research conducted mau [14] carried out meshing with tetrahedral elements as many as 140298 elements with an element size of 0.8 mm with almost the same design dimensions geometry.

**3.3 Boundary Conditions**

Boundary conditions were performed on FEA simulations of bone screws aimed at determining the limit conditions to be accepted by the predetermined bone bolt design. Boundary conditions are performed on bone screws with torsion conditions, pullout bone screws and bending bone screws. In the boundary conditions of bone screws pullout, FEA simulations use variations in the load that is assumed to be received by the human body, namely 700 N, 800 N, 900 N, 1000 N and 1100 N. With FEA simulation, it is able to find out the location of the maximum stress and deformation on the bone bolts. There is a series in carrying out boundary conditions, which is like figure 7 below.

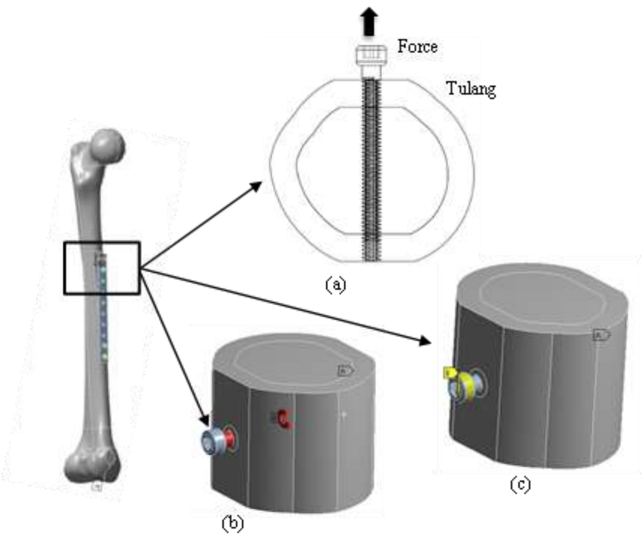



**Figure 7. Boundary Conditions Assembly Bone Plate Bolts, (a) Bone Screw Pullouts, (b) Bone Screw Torsions, (c) Bone Screw Bending**

In the FEA simulation, boundary conditions have been carried out with fix support on the bone while the force is given in the direction of the bone screws axis with tensile load conditions. While the Boundary condition is carried out with bone assembly and screws with engineering data according to the actual condition of both bone material and Mg AZ31B material. The torsion FEA simulation test was given moments of 883 N.mm [8], 706. 9 N.mm, 651.1 N.mm, 628.9 N.mm [9] clockwise and in bone screws bending boundary fixed condition support was given to the bone then loading by displacement against the axis perpendicular to the bone screws. In the FEA simulation research that has been carried out, namely the x-axis: 5 mm, the y-axis: free and the z-axis: free.

**3.3 Bone Screws Pullout**

Bone screws pullout testing is a test with FEA simulation to determine the deformation and maximum load that a bolt can receive on a load in the direction of the bone screws. The bone screws pulllout image is like figure 8 as follows.

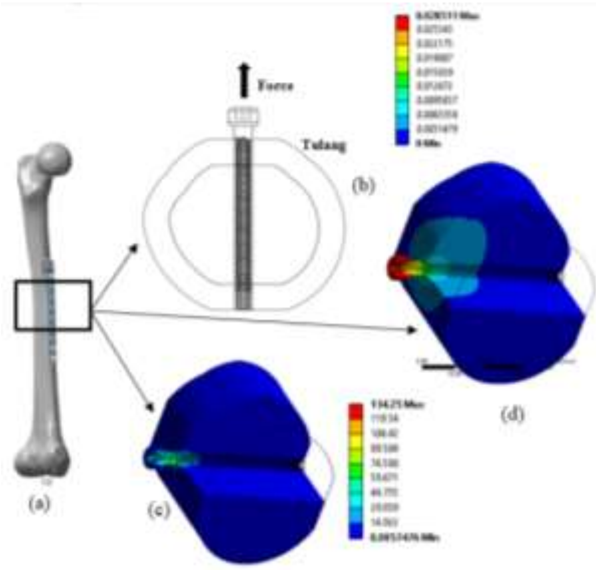



**Figure 8. (a). Cortical Bone (b). Pullout schematic, (c) Von Misses Stress, (d) Total Deformation**

Then after FEA simulations were carried out on bone screws pullouts total deformation of 0.028 mm and von misses stress of 134.25 MPa against pullout loads of 1100 N in FEA simulations, in the test the greater the load given by pullouts, the greater the deformation produced. The research conducted by Keshtiban [16] was obtained with a cortical screw FEA von misses stress of 449 MPa and a total deformation of 0.0624 mm with

Ti6Al4V material. The bone screw pullout chart is like figure 9 as follows.

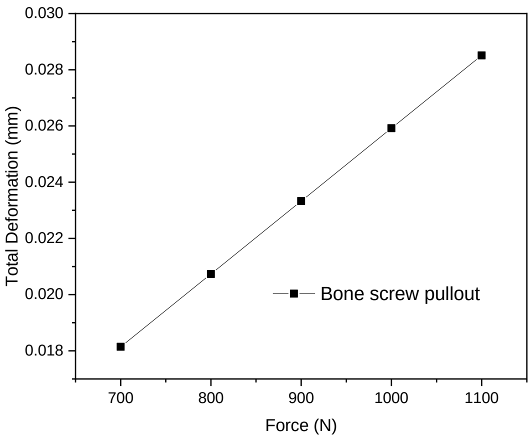



**Figure 9. Graph of total defromation to force variations**

In FEA simulations that the greater the pullout tensile load, the greater the deformation that the bone screws produce against the cortical bone. The pullout load on the normal cortical bone is 1450 N [10] while at a load of 1100 N with a total deformation of 0.2 mm it is made of stainless steel [11]. In addition to knowing the load given to the total deformation and von misses stress obtained, it is also necessary to know the stress distribution of bone screws in pullout conditions. This condition is carried out in order to be able to find out the stress distribution that occurs on the length of the bone screws. The linearized von misses stress condition against the length of the screws is as figured 10-11 as follows.

**Figure 10. Linearized equivalent stress bone screws pullout**

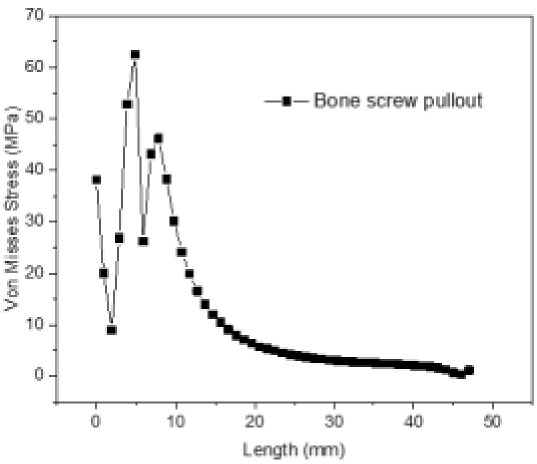



**Figure 11. Linearized von misses stress bone screws pullout graph**

In FEA testing, a graph of stress distribution over bone screws length was obtained. The stress distribution that occurs, namely on the screws head of 2.9376 mm, the stress occurs at 26,852 MPa then on the screws rod 4.896 mm experiences a maximum stress of 62,538 MPa, then the farther from the pullout condition, the stress drops more and lower at a distance of 46.023 mm has the lowest stress of 0.26643 MPa.

### 3.3 Bone Screws Torsion

After the boundary condition is carried out, the results of the FEA simulation of total deformation will be obtained and it appears that there is damage to the bone which resulted in the moment given. FEA moment simulation testing is carried out in order to find out the maximum moment given by the bolt until there is damage to the bone. From the research of Moldovan and Bataga [12] to determine the optimal torque, it refers to the graph of the torque variation curve such as threshold torque, peak clamping torque, peak failure torque. The chart is like figures 12-13.

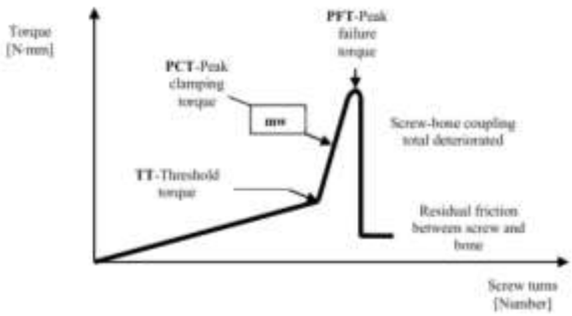



**Figure 12. Graph of variations in torque conditions**

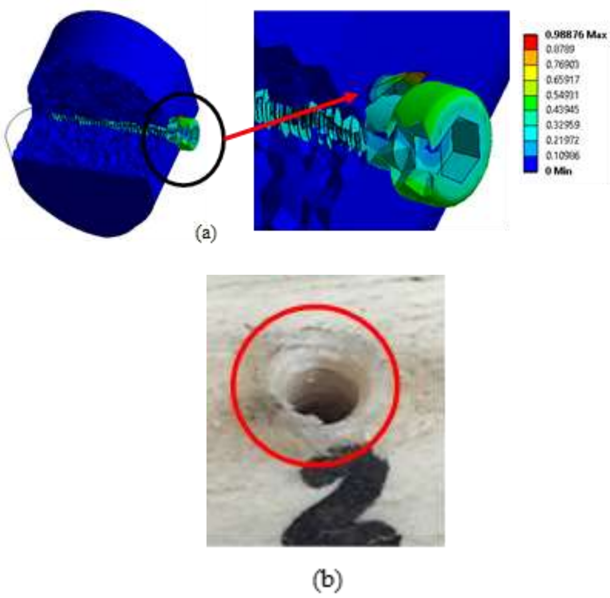



**Figure 13. Total Deformation (a). FEA, (b). Ekperimental [12]**

### 3.4 Bone Screws Bending

In bone screws bending testing aims to determine the maximum load that can be received by the bolt assembly and bone with a longitudinal loading perpendicular to the bone screws. From the boundary conditions given, then a FEA simulation is carried out, it will find out the deformation of the stress. The results of the FEA simulation are like figure 14 as follows.

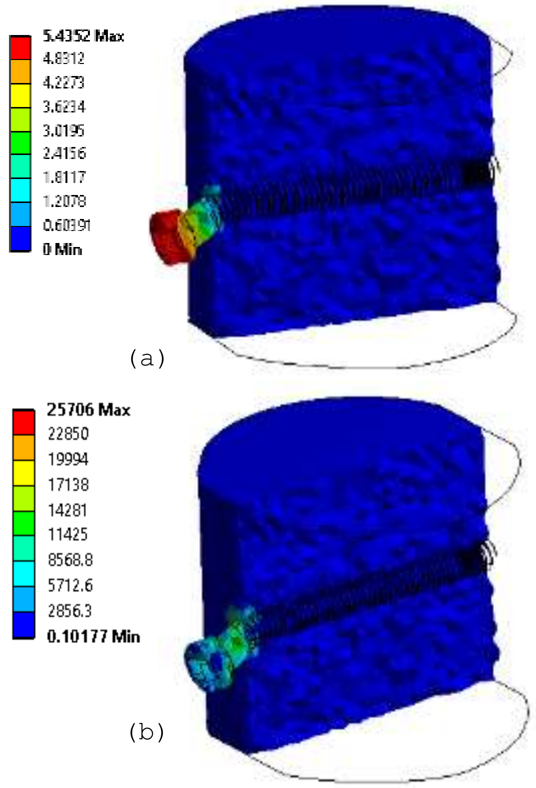



**Figure 14. (a). Total Deformation, (b). Von Misses Stress**

Then bone screws bending is given a conditions on the path along the screws to determine the stress distribution to the length of the bone screws. the results of linearized equivalent srtess bone screws bending are as figure 15 as follows.


**Figure 15. Linearized equivalent stress bone screws bending**

In a FEA simulation that has been done that the bone screws will deform when receiving the maximum load. The part of the bone screws that receives tension in the bone screws display is known to be linearized von misses stress. The graph of the stress that occurs along the bone screws is as figure 16 as follows.

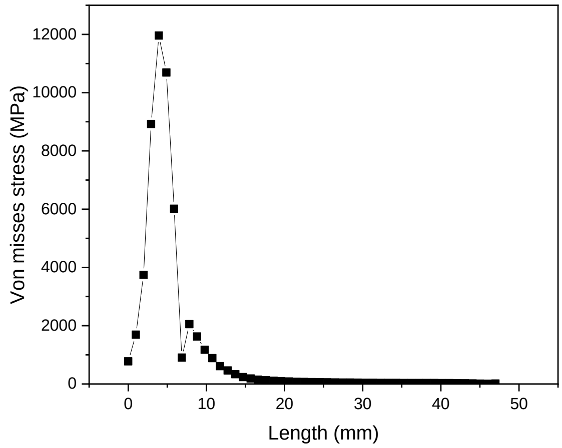



**Figure 16. Stress graph along bone screws**

From a linearized graph of von misses stress along bone screws that receive the load. The stress distribution that occurs, namely in the screws head of 2.9376 mm, the stress occurs at 8924.8 MPa (8.9248 GPa) then in the screws rod 4.896 mm it experiences a maximum stress of 11960 MPa (11,960 GPa), then the farther from the bending condition, the stress drops further at a distance of 46,023 mm has the lowest stress of 8.8482 MPa.

### 4. Conclusions.

In the FEA simulation test, the greater the tensile load given, the greater the deformation produced, then for the torsion bone screws with a torque of 883 N.mm the maximum deformation of 0.988 mm, and the bone screws bending the greater the deformation, the greater the deformation, it will produce a greater force reaction. From the research, it can be seen that by simulating the FEA method, it can minimize failures in designing before manufacturing, especially for the application of bone screws made from magnesium AZ31B.

### References.


[1] Stahel, P.F., Alfonso, N.A., Henderson, C., Baldini, T. Introducing The “ Bone-Screw-Fastener ” For Improved Screw Fixation In Orthopedic Surgery : A Revolutionary Paradigm Shift, Patient Saf Surg. 11 1–8. Doi:10.1186/S13037-017-0121-5. November 2017.

[2] Tilton, M., Armstrong, A.D, Wee, H., Hast,

M.W., Manogharan, G., Lewis Gs. Finite element-predicted effects of screw configuration in proximal humerus fracture fixation. Journal of Biomechanical Engineering [Internet]. 2020 [cited 2021 Mar 25];142. Available from: https://doi.org/10.1115/1.4045907. March 2021.

[3] Wang, S., Wang, D., Lin, Y., Li, X. Biomechanical Analysis Of Femoral Fixation With A New Compression Plate Using Finite Element Method During Stair Climbing. Key Engineering Materials Submitted: 2015-11-24 Issn: 1662-9795, Vol. 693, Pp 297-305 Accepted: 2015-12-31.Doi:10.4028/Www.Scientific.Net/Kem.693.297 Online: 2016-05-20 2016 Trans Tech Publications, Switzerland. 2016.

[4] Catalog Depuy Synthes Instrument And Implants. 2011 http://synthes.com accessed on Tuesday, June 15, 2021 at 16.00 WIB.

[5] Witte, F. The history of biodegradable magnesium implants: A review. Acta Biomater. 6, 1680–1692. 2008.

[6] Biswas, J. K., Sahu, T. P., Rana, M., Roy, S., Karmakar, S.K., Majumder, S., & Roychowdhury, A. Design factors of lumbar pedicle screws under bending load: A finite element analysis. Biocybernetics and Biomedical Engineering. doi:10.1016/j.bbe.2018.10.003. January 2019.

[7] Wang, W., Baran, G.R., Garg, H., Betz, R.R., Moumene, M., Cahill, P.J. The benefits of cement augmentation of pedicle screw fixation are increased in osteoporotic bone: a finite element analysis. Spine Deform;2(4):248–59. 2017.

[8] Wilkie, J., Docherty, P. D., & Moller, K. Stripping Torque Model for Bone Screws. 11th IFAC Symposium on Biological and Medical Systems BMS 2021 Ghent, Belgium, 19-22 September 2021. Volume 54, Issue 15, 2021, Pages 442-447. 2021.

[9] Silva, A. S., Mendes, J. M., Araujo, T., Aroso, C., & Barreiros, P. Comparison of Mechanical Resistance to Maximal Torsion Stress in Original and Nonoriginal or Compatible Prosthetic Implant Screws: An In Vitro Study. International Journal of Dentistry, 2021.

[10] Xu, M., Yang, J., Lieberman, I. H., & Haddas, R. Finite element method-based study of pedicle screw–bone connection in pullout test and physiological spinal loads. Medical engineering & physics, 67, 11-21. 2019.

[11] Lee, E. S., Goh, T. S., Heo, J. Y., Kim, Y. J., Lee, S. E., Kim, Y. H., & Lee, C. S. Experimental evaluation of screw pullout force and adjacent bone damage according to pedicle screw design parameters in normal and osteoporotic bones. Applied Sciences, 9(3), 586. 2019.

[12] Moldovan, F., & Bataga, T. Torque Control during Bone Insertion of Cortical Screws. Procedia Manufacturing, 46, 484-490. 2020.

[13] Lee, E. S., Goh, T. S., Heo, J. Y., Kim, Y. J., Lee, S. E., Kim, Y. H., & Lee, C. S. Experimental evaluation of screw pullout force and adjacent bone damage according to pedicle screw design parameters in normal and osteoporotic bones. Applied Sciences, 9(3), 586. 2019.

[14] Mau, J. R., Hawkins, K. M., Woo, S. L. Y., Kim, K. E., & Mccullough, M. B. Design of a new magnesium-based anterior cruciate ligament interference screw using finite element analysis. Journal of Orthopaedic Translation, 20, 25-30. 2020.

[15] Emzain, Z. F., Huang, S. C., Yang, Y. S., & Qosim, N. Design and Analysis of a Dynamic Splint Based on Pulley Rotation for Post-Stroke Finger Extension Rehabilitation Device. Jurnal Rekayasa Mesin, 11(3), 477-485. 2020.

[16] Keshtiban, P.M., Regbat, M., & Keshtiban, M.M. An investigation of tensile strength of Ti6Al4V titanium screw inside femur bone using finite element and experimental tests. Journal of Computational Applied Mechanics, 51(1), 91-97. 2020.

[17] Tetteh, E., Mccullough, M.B. Impact Of Screw Thread Shape On Stress Transfer In Bone: A Finite Element Study, Computer Methods In Biomechanics And Biomedical Engineering, Pp. 1-6, 2020.

[18] Zain, N., Daud, R., Aziz, N., Ahmad, K., Ismail, A., Izzawati, B. Stress Analysis Prediction On Screw Orthopedic Implant In Trabecular Bone, Materials Today: Proceedings, Vol. 16, Pp. 1838-1845, 2019.

Gmail

Irza Sukmana <irza.sukmana@gmail.com>

# Optimization Design of Bone Crews Biodegradable with Magnesium AZ31B Based on Finite Element Analysis

1 message

**Dr. Irza Sukmana** <ejurnal@pnl.ac.id> Thu, Jul 21, 2022 at 12:25 PM
To: Amir Zaki Mubarak <amir_zm@unsyiah.ac.id>

Dear Sir,

We have modified and revised our manuscript according to the reviewers comment as on the attached file name "clean" and also attached the modified section in file "marked" one.

For you consideration, we attached also the responses to reviewer letter that listed all comments and changes in the manuscript as suggested. Also, we have run the proofreading to our revised manuscript above.

With this submitted requirement, we hope now the Editorial Board may accept our manuscript for publication in Polimesin Journal.

Thank you.
Regards,

Irza
Dr. Irza Sukmana
Jurusan Teknik Mesin, FT Unila

---

Editor-in-Chief
Jurnal POLIMESIN
http://e-jurnal.pnl.ac.id/index.php/polimesin

---

**3 attachments**

**M A Wicaksono_I Sukmana_Jurnal Polimesin_revised_clean.doc**
3557K

**M A Wicaksono_I Sukmana_Jurnal Polimesin_revised_marked.doc**
3565K

**Response_letter_Polimesin_Wicaksono and Sukmana 21072022.docx**
44K

**Responses to reviewer's comment and the list of changes**

**Manuscript ID#: #2961, Jurnal Polimesin**
Author(s): Mahruri Arif Wicaksono and Irza Sukmana
Paper Title: Optimization Design of Bone Screws Biodegradable with Magnesium AZ31B Based on Finite Element Analysis

*Response and Revision made by Author(s)*
**Reviewer #1:**

<table>
<tr><th>No</th><th>Comments</th><th>Revision/Changes</th></tr>
<tr><td>1</td><td><ul><li>In the paper title mentions optimization, what optimization method is used?</li><li>What parameter to describe the degradable material? And how does it relate to the research results obtained, namely deformation and stress?</li><li>It is necessary to explain why only bone screws is analyzed, even though there are three components involved where the strength is lower than bone screws, especially cortical bone.</li></ul></td><td><ul><li>We have changed the title accordingly to, "Finite Element Analysis of Magnesium AZ31B Materials for Biodegradable Bone Screw Application".</li><li>In this research we concern on finite element analysis or the bone screw design for Mg that has been well known as the best candidate for biodegradable bone screw and plate. We have mentionded the condition and focus of this study in the abstract as, "In vitro biocompatibility and in vivo studies is needed for further assessment of the design."</li><li>It can be analyzed separately or in an assembly one, however in this manuscript we focus on the bone screw separately. We have inform the condition in the abstract as foloow, "This study focuses on finite element analysis (FEA) for the bone screw design of Magnesium alloy that has been well known as a potential candidate for biodegradable bone screw and plate application.</li></ul></td></tr>
<tr><td>2</td><td>How is the correlation between the literature cited in the introduction with the research conducted so that the research contribution can be shown</td><td>We have added other two international references to explain the correlation and contribution of our study.<ol><li>Chandra, G., & Pandey, A. (2021). Design and analysis of biodegradable buttress threaded screws for fracture fixation in orthopedics: a finite element analysis. Biomedical Physics & Engineering Express, 7(4), 045010.</li><li>Mau, J. R., Hawkins, K. M., Woo, S. L. Y., Kim, K. E., & Mccullough, M. B. Design of a new magnesium-based anterior cruciate ligament interference screw using finite element analysis. Journal of Orthopaedic Translation, 20, 25-30. 2020.</li></ol></td></tr>
<tr><td>3</td><td><ul><li>The modeling used is static or dynamic method?</li></ul></td><td><ul><li>We have added information of the study is for static structural in material and method section as follow, "The analysis</li></ul></td></tr>
</table>

<table>
<tr>
<td></td>
<td>

- What modeling material is used? Elastic, elastic-plastic, viscoelastic or something else? Do Bone Screw, Cortical Bone and plate use the same material model?
- The stated convergence is written from the literature. What is the actual convergence result of the modeling? Convergence based on number of elements or size of elements?
- What type of contact between bone screw – plate, plate-cortical bone and bone screw-cortical plate?
- Models are given various loads. The variation of the load describes the load according to age, weight of people or something else?

</td>
<td>

process is carried out using Ansys software with procedures as follow: input data engineering, import geometry, static structural, meshing, boundary conditions, pullout bone screw, torsion and bending testing, to get FEA results in the form of total deformation and von misses stress.

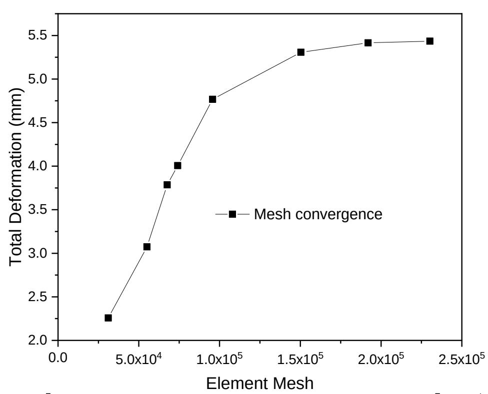
- The modeling material used is elastic-platic as the bone plate and cortical bone also do the same material model. We added the information accordingly, “The use of the material in the FEA simulation of cortical bones is assumed to be normal cortical bones with a density of 1.61 g/cm3 – 1.77 g/cm3 [10], and the modelling is for material elastic-platic.”.
- Contact surface used “Frictional” one as explained in the boundary condition.
- FEA simulations use the variations in a oad that is assumed to be the weight body of humans with various ages, as follow: 700 N (71,38 kg), 800 N (81.57 kg), 900 N (91.77 kg), 1000 N (101.97 kg) and 1100 N (112.16 kg). The information is mentioned in the sub section of Boundary Condition.

</td>
</tr>
<tr>
<td>4</td>
<td>

- Considering that the research conducted did not use experiments, convergence used to verificate the modeling result. The claimed convergence of less than 3 percent is obtained from what number of elements? Is there a convergence graph?
- Acctually, verification of modeling results using convergence is not sufficient for 3D elements. The accuracy of 3D modeling depends not only on the number of elements but also on the shape of the elements.

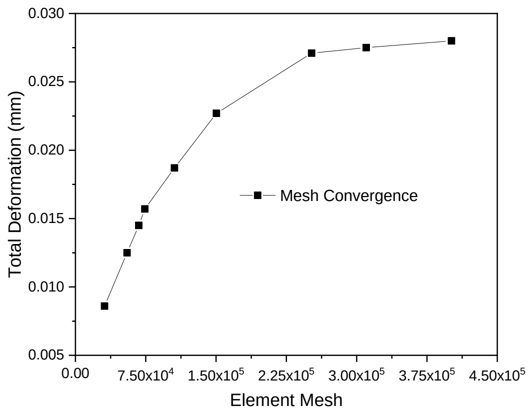
</td>
<td>

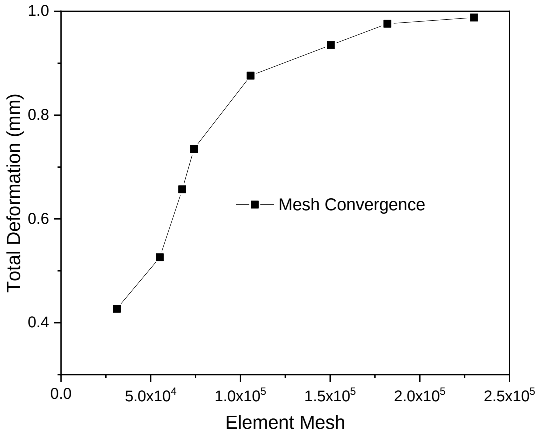
- 3 percent is meant is the level of error that occurs between the experiment and the simulation. in the research conducted not doing experimental but doing validation with research and approach parameters. such as the moldovan study (Moldovan, F., & Bataga, T. Torque Control during Bone Insertion of Cortical Screws. Procedia Manufacturing, 46, 484-490. 2020) can be validated to have the same fault location between the experimental and simulated FEA.




(Mesh Convergence Bone Screw Bending)

</td>
</tr>
</table>

<table>
<tr><td></td><td></td><td>


(Mesh Convergence Bone Screw Pullout)


(Mesh Convergence Bone Screw Torsion)

- Meshing is performed to divide the model parts into small element parts. Meshing helps determine the distribution of a given stress. There are several types of elements such as hexahedron (simple components), tetrahedron and polyhedron (complex geometry details). The type of mesh used uses tetrahedron elements because it is a type of meshing that is entirely accurate for fairly complex geometry details. In the study, there were three parts, namely bone screws, bone screws assembly, and bone plates with cortical bones.
</td></tr>
<tr><td>5</td><td>Discussion of the results does not only comment on the magnitude of deformations and stresses but it is necessary to discuss the type and position of failure for each type loading.</td><td>We have included that issue as, “It can be concluded that in the bone screw test that with the design of the screw that at a load of 1100 N the pullout deformation only occurs at 0.028 mm then the torsion bone screw at 883 N.mm is damaged in the bone screw hole and is declared unsafe or has already been damaged.”</td></tr>
</table>

| | | |
|---|---|---|
| 6 | In this section it is necessary to show what is difference from previous work on this area. Provide relevant references | The difference between the research that has been done previously and what is currently being done is that there are differences in the materials used such as (Ti-6Al-4V and Stainless steel) while the material used in this study is Magnesium AZ31B with a purity of 99.9%. At this time the use of magnesium AZ31B is only carried out as a basic material for biomaterials and has not yet reached the design and analysis stage. There are design differences such as pitch, bone screw length and bone screw diameter. |
| 7 | • At point 2nd. There is still a word in Indonesian "pejal"<br>• In 7th and 8th pictures, the description of the picture still has the Indonesian word "tulang"<br>• Literature citation writing based on bibliographic number. In the Introduction, after 4th citation number, next citation jumps to the 15th citation. It should be written in order. | • We have revise the un aprropriate word in Bahasa accordingly and also proofread the manuscript secondly to improve the english quality.<br>• We have modified the figures<br>• Thank you, we have adjusted the number of citation as suggested. |

**Reviewer #2:**

| No | Comments | Revision/Changes |
|---|---|---|
| 1. | • what is the problem with Bone Crews Biodegradable with Magnesium AZ31B so this needs to be optimized?<br>• the purpose of the research<br>• add international journal citations about this case<br>• can there be bio-corrosion in the use of Magnesium AZ31B material when implanted in the body? and affect its mechanical properties | • We have changed the title accordingly to, “Finite Element Analysis of Magnesium AZ31B Materials for Biodegradable Bone Screw Application”.<br>• We have inform the purpose of this research as follow, “This research is conducted to design and analyze with the finite element analysis method on bone screws based on material Magnesium AZ31B.<br>• We have add the citation [3] Chandra, G., & Pandey, A. (2021). Design and analysis of biodegradable buttress threaded screws for fracture fixation in orthopedics: a finite element analysis. Biomedical Physics & Engineering Express, 7(4), 045010.<br>• We have explain the condition as follow, “Approaching the bone modulus [4], the biodegradability of magnesium and its alloys in the human body avoids a second operation for temporary implants. The mechanical properties of AZ31B magnesium material will decrease because it’s degradation process. Therefore the degradation rate of the magnesium implant should fit with the bone fracture recovery. The percentage of elements in magnesium alloys greatly affects the mechanical properties and biocompatibility magnesium [5]. |
| 2. | can there be bio-corrosion in the use of Magnesium AZ31B material when implanted in the body? and affect its mechanical properties | • Yes, we haven mentioned the condition in the text accordingly as follow, “The mechanical properties of AZ31B magnesium material will decrease because it’s degradation process. Therefore the degradation rate of the magnesium implant should fit with the bone fracture recovery”. |
| 3. | Conclusions must include all aspects of the research objectives. review the purpose of the research | We have included that issue as, “It can be concluded that in the bone screw test that with the design of the screw that at a load of 1100 N the pullout deformation only occurs at 0.028 mm then the torsion bone screw at 883 N.mm is damaged in the bone screw hole and is declared unsafe or has already been damaged.” |
| 4. | adjust to the number of citations | Thank you, we have adjusted the number of citation as suggested. |

# Finite Element Analysis of Magnesium AZ31B Materials for Biodegradable Bone Screw Application

Mahruri Arif Wicaksono, and Irza Sukmana*)
Department of Mechanical Engineering, Faculty of Engineering, Universitas Lampung, Gedung H Lt. 2, Jl. Prof. Soemantri Brojonegoro No. 1, Bandar Lampung, 35143, Indonesia
*) Corresponding author: irza.sukmana@gmail.com

**Abstract**

*The bone implant functions for the load bearing and aims to support the integration of fractured bone. It may increase the strength of the broken bones and also at the same time support bone regeneration and integration. Bone screws are usually attached to the implant plate and bonded to the surface of the fractured bone by screwing the bone screws through the bone structure. In its implementation, a non-degradable implant needs a second operation for the patient to take out the implant. Currently, biomedical researchers are trying to produce bone implants that are degradable or bioresorbable materials. Magnesium (Mg) alloy is a potential biomaterial for bone implants, as Mg is a degradable material. Mg is one of the elements needed and harmless to the human body. This study focuses on finite element analysis (FEA) for the bone screw design of Magnesium alloy that has been well known as a potential candidate for biodegradable bone screw and plate application. The three dimensions (3D) design was done by using Solidworks software, and finite element analysis was performed using ANSYS by calculating the moment, pullout, and force bending received by bone screws. The validatation results of the design carried out with several analytical tests before the production of bone screws is proposed. The FEA simulation of bone screws pullout has a total deformation of 0.028 mm and a von Mises stress of 134.25 MPa for a pullout load of 1100 N. The bone screws torsion with a torque of 883 N.mm, the total deformation is 0.988 mm and for bone screws bending with a total deformation of 5,4352 mm has a von Mises stress of 25.706 MPa. AZ31B bone screws, based on the design, are safe and capable of handling the maximum load and deformation during the implantation. In vitro biocompatibility and in vivo studies is needed for further assessment of the design.*

***Keywords****: Magnesium AZ31B, Bone Screws, FEA, Pullout, Torsion, Bending.*

## 1. Introduction

Bone screws are used to increase strength and resistance to failure in the healing process of bone cracks. Many studies use numerical simulation in combination with optimization algorithms to optimize the design, which reduces the destructive effect and increases the efficiency of the implant. One of the primary purposes of implant components is to strengthen bones and reduce damage using the Taguchi and finite element method to optimize the design and engineering of bone screws [1]. In research conducted previously by Tilton [2] mechanical testing and simulation of elements to increase the bending strength of tibia bone screws, titanium, and stainless steel, which are still often used but have the disadvantage of doing a second operation.

Magnesium is one of the potential candicate materials for bioodegradable implant. The mechanical properties of magnesium and alloys have a low density (1.74-2.0 g/cm$^3$) and a modulus of elasticity (41-45 GPa). Finite element analysis (FEA) is on of the technique that has been used to test and validate the desired characteristics for biodegradable Mg-based support threads. The characteristics of interest include the maximum tensile resistance possible to hold the bone segment firmly, the torsional ability to tighten [3].

Approaching the bone modulus, the biodegradability of magnesium and its alloys in the human body avoids a second operation for temporary implants [4]. The mechanical properties of AZ31B magnesium material will decrease because it’s degradation process. Therefore the degradation rate of the magnesium implant should fit with the bone fracture recovery. The percentage of elements in magnesium alloys greatly affects the mechanical properties and biocompatibility magnesium [5]. The design of the bone screws refers to the Depuy Synthes Instruments and implants catalog, with specifications, sizes, and materials [6].

The biomaterials are not only used in the body but something that helps or is in contact with the human body aims to cure a malfunction of the body in humans such as cracked bone healing aids (bone screws-plates), stroke disease therapy tools [7], prosthetic models compliant anklefoot [8] and ankle foot orthosis (AFO) is an orthopedic device that is attached to the foot to improve the structure of the foot and help the user to return to normal running [9].

Optimization is done to get an effective bone screws design according to the application. The design is carried out from the specific thread, bone screws length, bone screws diameters and bone screws material. This research is conducted to design and analyze with the finite element analysis method on bone screws based on material Magnesium AZ31B. Study of the corrosion and biodegradation study is not included in this paper.

## 2. Materials and Methods

The research was conducted in March - August 2022, the design process and simulation of bone screws were carried out at the Structural Mechanics Laboratory of the Department of Mechanical Engineering, the University of Lampung under the catalog of Depuy Synthes Instruments and implants, then using magnesium AZ31B material in the form of a rigid cylinder with an initial diameter of 10 mm and a length of 130 mm. After preparing the tools and materials first carry out a 3-dimensional design process using Solidworks then the FEA analysis process is carried out. The analysis process is carried out using Ansys software with procedures as follow: input data engineering, import geometry, static structural, meshing, boundary conditions, pullout bone screw, torsion and bending testing, to get FEA results in the form of total deformation and von Mises stress.

### 2.1. Materials

Magnesium material is a light metal with a density of about 1.77 g/cm$^3$. The use of the material in the FEA simulation of cortical bones is assumed to be normal cortical bones with a density of 1.61 g/cm$^3$ – 1.77 g/cm$^3$ [10], and the modelling is for material elastic-platic. Mg-based biomaterials have more excellent toughness then ceramic and polymer-based biomaterials because their elastic modulus is about 45 GPa and The tensile test graph and mechanical properties of magnesium material AZ31B with the cortical bone are as follows in figure 1.

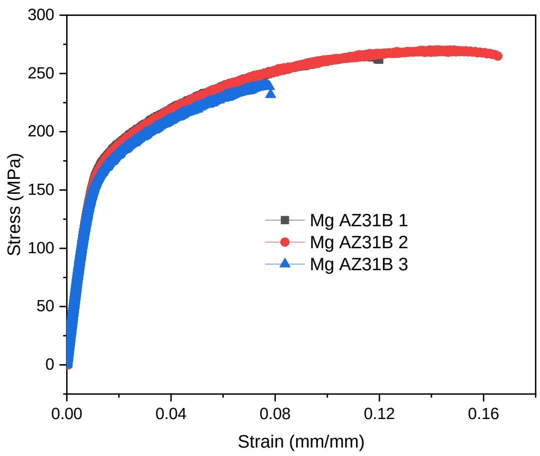



**Figure 1. Tensile test graph of stress vs strain**

The modulus of elasticity of magnesium metal material has characteristics that are almost close to the nature of the modulus of elasticity of human bones, which is about 7-25 GPa [11] while bone screws are made of Ti-6Al-4V material with modulus young and poisson ratios of 113 GPa and 0.3 [12]. This study used magnesium material AZ31B extrude rod with a diameter of 10 mm with a length of 130 mm. Magnesium AZ31B was purchased at the manufacturing company Xi'an Yuechen Metal Products Co., Ltd, Using production technology in accordance with international standards and product specifications SGS BV model number AZ31B-H24 specifically for production using ASTM B107/B107M-13 standards.

**Table 1.** Mechanical properties Mg AZ31B and cortical bone

| No | Materials | Parameter | |
|---|---|---|---|
| 1 | Mg AZ31B | Yield Strength | 153,15 MPa |
| | | Modulus young | 44,28 GPa |
| | | Density | 1,77 g/cm$^3$ |
| | | Poisson Ratio | 0,35 |
| | | Ultimate | 258,44 MPa |
| 2 | Cortical bone | Modulus Young | 20 GPa |
| | | Density | 1,7 g/cm$^3$ |
| | | Poisson Ratio | 0,25 |

### 2.2. Analysis Methods

Development and production in bone screws engineering where this research is expected to be in accordance with standard procedures and can be commercialized. The method used to analyze the Ansys software is like figure 2.

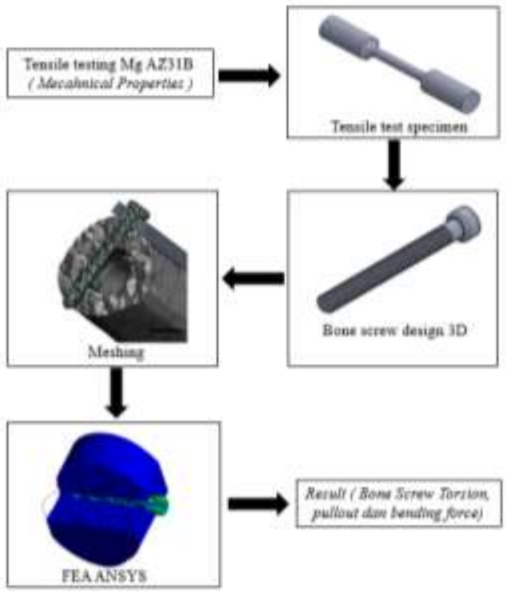



**Figure 2. Flowchart of simulation methods**

In the method carried out the analysis process, conduct tensile testing (ASTM B557) to determine the ultimate tensile strength and modulus young, enter engineering data from the results of tensile testing of magnesium material AZ31B then simulate FEA static structural bone screws torsion, bone screws pullout and bone screws bending then carry out mesh convergence. In the analysis with FEA, the results of total deformation, von Mises stress and linearized von Mises stress were obtained.

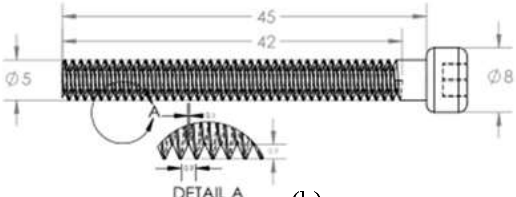
## 3. Results and Discussion.

The results and discussions in the research that has been carried out are through several stages such as CAD design (Solidworks), input engineering data (ultimate tensile strength, Poisson ratio, density), determination of meshing, boundary conditions (bone screws torsion, bone screws pullout, bone screws bending testing force), solve and result (total deformation and von Mises stress) are as follows:

### 3.1 Geometry CAD

Bone screws with a diameter of 5 mm, length of 40 mm, and pitch of 1 mm is the geometric dimensions of the cortical bone screws to be used. The bone screws in previous studies were modified to provide design factors such as pitch length, primary diameter, thread profile, and geometry variations. In Biswas's research [12], 84 FE models were developed with seven pitch lengths (1, 1.5, 2, 2.5 ,3, 3.5 and 4 mm). Tetteh and McCullough [13] focused on the thread profile and thread shape effect on the transfer rate of stress distribution in the bone using finite element analysis. It was concluded that bone screws with a trapezoidal threaded shape could transfer higher stress to surface contact in cortical bones. Zain's research [14] studied the orthopedic stress distribution of screws implants in trabecular bones, which are very porous with biological tissues. In this study, the shape of the screws the profile of the thread, and its details are presented in figure 3.

(a)




(b)

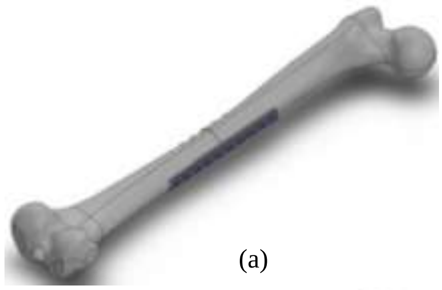
(c)

**Figure 3. (a) 3D geometry of bone screws, (b) Geometry of 2-D bone screws, (c) Manufacturing of bone screws**

The geometric dimensions of bones are adjusted to human bones in general. For the step to be carried out, first make a hole with a hole diameter of 5 mm produced by drilling in the middle part of the bone prepared in so that the upper and lower parts of the bone can be drilled in the same direction. The process of combining the bone screws and bone plates with being attached can be described in figure 4.

**Figure 4. Assembly screws and bone plates**

While in the geometry assembly of bone screws, bone plates, and cortical bones, there will be contact with surfaces such as plate contact with cortical bone, bone plate with bone screws, and bone screws with the cortical bone. For bone plate assembly, bone screws and cortical bone, as shown in figure 5.



(a)

(b)

**Figure 5. Assembly of bone screws, bone plates, and cortical bones**

### 3.2 Meshing

Meshing is performed to divide the model parts into small element parts. Meshing helps determine the distribution of a given stress. There are several types of elements such as hexahedron (simple components), tetrahedron and polyhedron (complex geometry details). The type of mesh used uses tetrahedron elements because it is a type of meshing that is entirely accurate for fairly complex geometry details. In the study, there were three parts, namely bone screws, bone screws assembly, and bone plates with cortical bones, as shown in table 2 and figure 6.

**Table 2.** Mesh Convergence

| Simulasi FEA | Element Size (mm) | Element | Total Deformasi (mm) | Von Mises Stress (MPa) |
|---|---|---|---|---|
| Bone Screws Torsion | 1.5 | 230.247 | 0,988 | 16.984 |
| Bone Screws Pullout | 1.25 | 401.119 | 0,028 | 134,25 |
| Bone Screws Bending | 1.5 | 230.247 | 5,4352 | 25.706 |

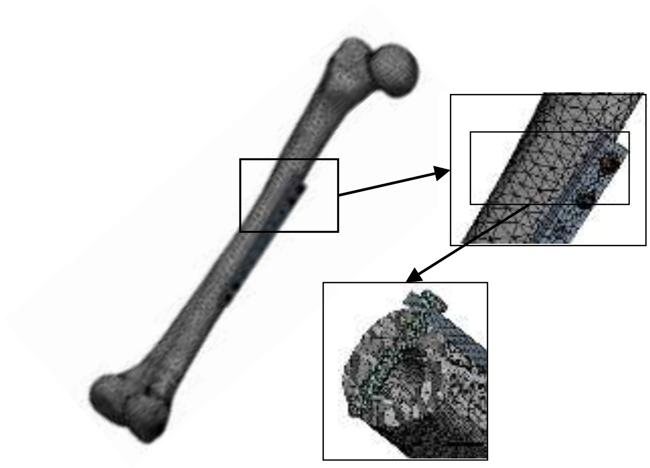


**Figure 6. Assembly element meshing**

For meshing, a 4-noded tetrahedron element is used and convergence analysis is performed by resetting the element size until an error of less than 3% from eksperiment [15]. Meanwhile, the research conducted by Mau et al. [16] carried out meshing with tetrahedral elements as many as 140298 elements with an element size of 0.8 mm with almost the same design dimensions geometry.

### 3.3 Boundary Conditions

Boundary conditions were performed on FEA simulations of bone screws to determine the limit conditions to be accepted by the predetermined bone screws design. Contact surface used "frictional" because the bone screw will cause friction between the bone plate with bone screw, bone screw with cortical bone then bone screw receive tensile and bending loads so that it will cause widening of the bone screw with bone screw holes. (gap open/close, sliding and friction). Boundary conditions are performed on bone screws with torsion conditions, pullout bone screws and bending bone screws. In the boundary conditions of bone screws pullout, FEA simulations use variations in the load that is assumed to be the weight body of humans of various ages, namely 700 N (71,38 kg), 800 N (81.57 kg), 900 N (91.77 kg), 1000 N (101.97 kg) and 1100 N (112.16 kg) [17]. With FEA simulation, it can find out the location of the maximum stress and deformation on the bone screws. In the FEA simulation, boundary conditions have been carried out with fixed support on the bone while the force is given in the direction of the bone screws axis with tensile load conditions. While the Boundary conditions is carried out with bone assembly and screws with engineering data according to the actual condition of both bone material and Mg AZ31B material.

The torsion FEA simulation test was given moments of 883 N.mm [18], 706. 9 N.mm, 651.1 N.mm, 628.9 N.mm [19] clockwise and in bone screws bending boundary. fixed condition support was given to the bone then loading by diplacement against the axis perpendicular to the bone screws. In the FEA simulation research that has been carried out, namely the x-axis: 5 mm, the y-axis: free and the z-axis: free.There is a series in carrying out boundary conditions, which is like figure 7 below.

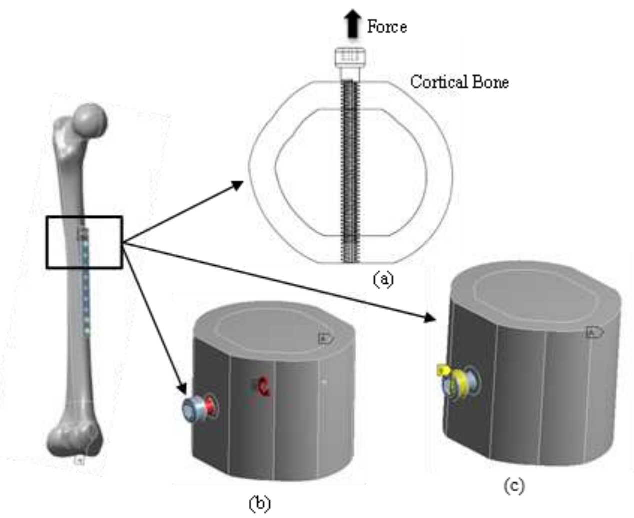



**Figure 7. Boundary conditions assembly bone plate screws, (a) Bone screws pullout, (b) Bone screws torsion, (c) Bone screws bending**

### 3.4 Bone Screws Pullout

Bone screws pullout testing is a test with FEA simulation to determine the deformation and maximum load that a screw can receive on a load in the direction of the bone screws. Then after FEA simulations were carried out on bone screws pullouts total deformation of 0.028 mm and von Mises stress of 134.25 MPa against pullout loads of 1100 N in FEA simulations, In the test the greater the load given by pullouts, the greater the deformation produced.

The research conducted by Keshtiban [20] was obtained with a cortical screw FEA von Mises stress of 449 MPa and total deformation of 0.0624 mm with Ti6Al4V material. In FEA simulations the greater the pullout tensile load, the greater the deformation the bone screws produce against the cortical bone. The pullout load on the normal cortical bone is 1450 N [21] while at a load of 1100 N with a total deformation of 0.2 mm it is made of stainless steel [22]. The bone screws pullout image is like figure 8 as follows.

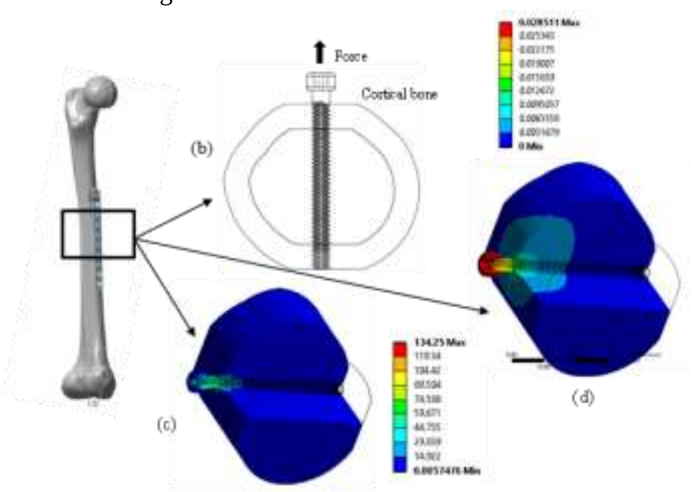



**Figure 8. (a). Cortical bone (b). Pullout schematic, (c) Von Mises stress, (d) Total deformation**

The bone screws pullout chart is like figure 9 as follows.

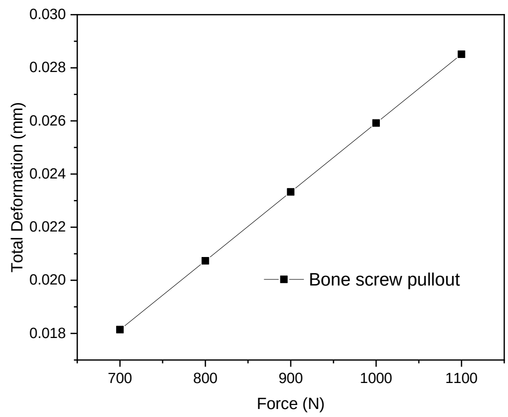



**Figure 9. Graph of total deformation to force variations**

In addition, to know the load given to the total deformation and von Mises stress obtained, it is also necessary to know the stress distribution of bone screws in pullout conditions. This condition is carried out in order to be able to find out the stress distribution that occurs on the length of the bone screws. The linearized von Mises stress condition against the length of the screws is as figured 10-11 as follows.

**Figure 10. Linearized equivalent stress bone screws pullout**

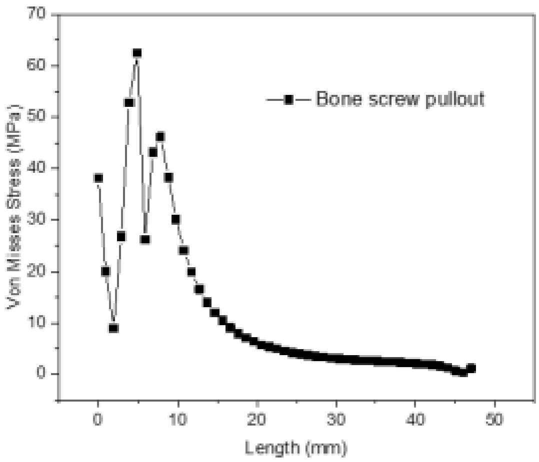



**Figure 11. Linearized von Mises stress bone screws pullout graph**

A graph of stress distribution over bone screws length was then obtained (fig. 11). The stress distribution that occurs, namely on the screws head of 2.9376 mm, the stress occurs at 26,852 MPa then on the screws rod 4.896 mm experiences maximum stress of 62,538 MPa, then the farther from the pullout condition, the stress drops more and lower at a distance of 46.023 mm has the lowest stress of 0.26643 MPa.

### 3.5 Bone Screws Torsion

After the boundary conditions are carried out, the results of the FEA simulation of total deformation will be obtained. FEA moment simulation testing is carried out to determine the maximum moment given by the screws until bone damage occurs. The research of Moldovan and Bataga [23] determined the optimal torque, refers to the graph of the torque variation curve such as threshold torque, peak clamping torque, and peak failure torque. There appears to be damage to the bone resulting at the moment given. The chart is like figures 12-13.

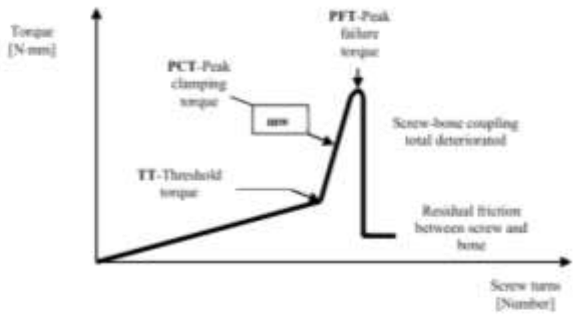
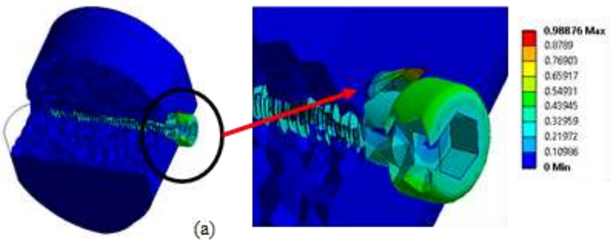



**Figure 12. Variations in the torque conditions [23]**



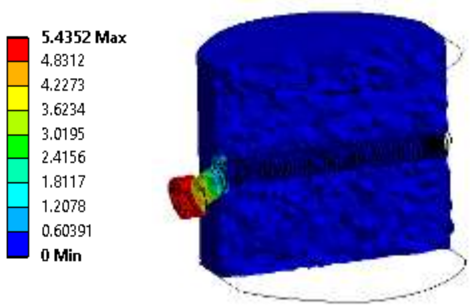

(a)

(b)

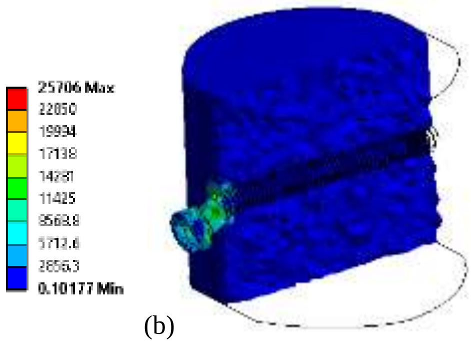
**Figure 13. Total deformation (a). FEA, (b). Experimental [23]**

### 3.6 Bone Screws Bending

In bone screws bending testing aims to determine the maximum load that the screws assembly and bone can receive with a longitudinal loading perpendicular to the bone screws. From the boundary conditions given, then a FEA simulation is carried out, it will find out the deformation of the stress. Then bone screws bending is given a conditions on the path along the screws to determine the stress distribution to the length of the bone screws.The results of the FEA simulation are like figure 14 as follows.




(a)




(b)

**Figure 14. (a). Total deformation, (b). Von Mises stress**

The results of linearized equivalent stress bone screws bending are as figure 15 as follows.


**Figure 15. Linearized equivalent stress bone screws bending**

In a FEA simulation, the bone screws deform when receiving the maximum load. The part of the bone screws that receives tension in the bone screws display is known to be linearized von Mises stress. The graph of the stress that occurs along the bone screws is in figure 16 as follows.

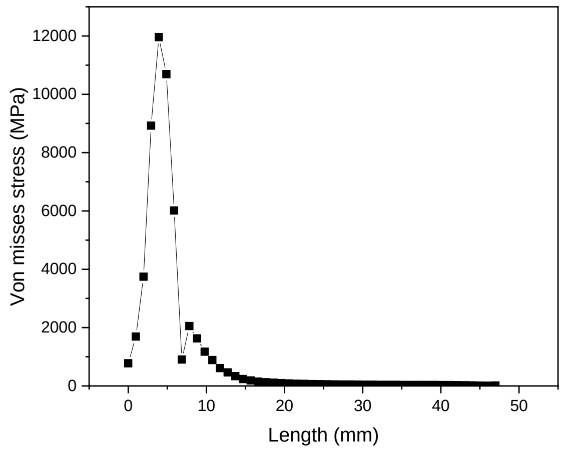



**Figure 16. Stress graph along with bone screws**

From a linearized graph of von Mises stress along with bone screws that receive the load. The stress distribution that occurs, namely in the screws head of 2.9376 mm, the stress occurs at 8924.8 MPa (8.9248 GPa) then in the screws rod 4.896 mm it experiences maximum stress of 11960 MPa (11,960 GPa), The farther from the bending condition, the stress drops further at a distance of 46,023 mm and has the lowest stress of 8.8482 MPa.

**4. Conclusions**

This research design and analysis of bone screw, simulation results were obtained using the finite element analysis (FEA) method in the form of pullout bone screw, torsion bone screw and bending bone screw have been done. It can be concluded that in the bone screw test that with the design of the screw that at a load of 1100 N the pullout deformation only occurs at 0.028 mm then the torsion bone screw at 883 N.mm is damaged in the bone screw hole and is declared unsafe or has already been damaged.

**References.**


[1] Stahel, P.F., Alfonso, N.A., Henderson, C., Baldini, T. Introducing The " Bone-Screw-Fastener " For Improved Screw Fixation In Orthopedic Surgery : A Revolutionary Paradigm Shift, Patient Saf Surg. 11 1–8. Doi:10.1186/S13037-017-0121-5. November 2017.

[2] Tilton, M., Armstrong, A.D, Wee, H., Hast, M.W., Manogharan, G., Lewis Gs. Finite element-predicted effects of screw configuration in proximal humerus fracture fixation. Journal of Biomechanical Engineering [Internet]. 2020 [cited 2021 Mar 25];142. Available from: https://doi.org/10.1115/1.4045907. March 2021.

[3] Chandra, G., & Pandey, A. (2021). Design and analysis of biodegradable buttress threaded screws for fracture fixation in orthopedics: a finite element analysis. Biomedical Physics & Engineering Express, 7(4), 045010.

[4] Wang, S., Wang, D., Lin, Y., Li, X. Biomechanical Analysis Of Femoral Fixation With A New Compression Plate Using Finite Element Method During Stair Climbing. Key Engineering Materials Submitted: 2015-11-24 Issn: 1662-9795, Vol. 693, Pp 297-305 Accepted: 2015-12-31.Doi:10.4028/Www.Scientific.Net/Kem. 693.297 Online: 2016-05-20 2016 Trans Tech Publications, Switzerland. 2016.

[5] Ibrahim, F., & Sukmana, I. (2021). Application of magnesium alloys in orthopedic implant. Jurnal Polimesin, 19(2), 182-187.

[6] Catalog Depuy Synthes Instrument And Implants. 2011 http://synthes.com accessed on Tuesday, June 15, 2021 at 16.00 WIB.

[7] Emzain, Z. F., Huang, S. C., Yang, Y. S., & Qosim, N. Design and Analysis of a Dynamic Splint Based on Pulley Rotation for Post-Stroke Finger Extension Rehabilitation Device. Jurnal Rekayasa Mesin, 11(3), 477-485. 2020.

[8] Emzain, Z. F., Amrullah, U. S., & Mufarrih, A. M. (2020). Analisis elemen hingga untuk siklus berjalan pada model prostetik lentur pergelangan kaki. Jurnal Polimesin, 18(2), 91-98.

[9] Suhendrianto, S., Zuhri, S., Tarigan, H. A. N., & Asbar, A. (2021). Konsep desain dan perbaikan produk Ankle Foot Orthosis menggunakan metode kansei engineering. Jurnal Polimesin, 19(2), 140-146.

[10] Lee, E. S., Goh, T. S., Heo, J. Y., Kim, Y. J., Lee, S. E., Kim, Y. H., & Lee, C. S. Experimental evaluation of screw pullout force and adjacent bone damage according to pedicle screw design parameters in normal and osteoporotic bones. Applied Sciences, 9(3), 586. 2019.

[11] Witte, F. The history of biodegradable magnesium implants: A review. Acta Biomater. 6, 1680–1692. 2008.

[12] Biswas, J. K., Sahu, T. P., Rana, M., Roy, S., Karmakar, S.K., Majumder, S., & Roychowdhury, A. Design factors of lumbar pedicle screws under bending load: A finite element analysis. Biocybernetics and Biomedical Engineering. doi:10.1016/j.bbe.2018.10.003. January 2019.

[13] Tetteh, E., Mccullough, M.B. Impact Of Screw Thread Shape On Stress Transfer In Bone: A Finite Element Study, Computer Methods In Biomechanics And Biomedical Engineering, Pp. 1-6, 2020.

[14] Zain, N., Daud, R., Aziz, N., Ahmad, K., Ismail, A., Izzawati, B. Stress Analysis Prediction On Screw Orthopedic Implant In Trabecular Bone, Materials Today: Proceedings, Vol. 16, Pp. 1838-1845, 2019.

[15] Wang, W., Baran, G.R., Garg, H., Betz, R.R., Moumene, M., Cahill, P.J. The benefits of cement augmentation of pedicle screw fixation are increased in osteoporotic bone: a finite element analysis. Spine Deform;2(4):248–59. 2017.

[16] Mau, J. R., Hawkins, K. M., Woo, S. L. Y., Kim, K. E., & Mccullough, M. B. Design of a new magnesium-based anterior cruciate ligament interference screw using finite element analysis. Journal of Orthopaedic Translation, 20, 25-30. 2020.

[17] Tejszerska, D., Świtoński, E., & Gzik, M . (2011) Biomechanics of the human locomotor system. Publishing house.

[18] Wilkie, J., Docherty, P. D., & Moller, K.

Stripping Torque Model for Bone Screws. 11th IFAC Symposium on Biological and Medical Systems BMS 2021 Ghent, Belgium, 19-22 September 2021. Volume 54, Issue 15, 2021, Pages 442-447. 2021.

[19] Silva, A. S., Mendes, J. M., Araujo, T., Aroso, C., & Barreiros, P. Comparison of Mechanical Resistance to Maximal Torsion Stress in Original and Nonoriginal or Compatible Prosthetic Implant Screws: An In Vitro Study. International Journal of Dentistry, 2021.

[20] Keshtiban, P.M., Regbat, M., & Keshtiban, M.M. An investigation of tensile strength of Ti6Al4V titanium screw inside femur bone using finite element and experimental tests. Journal of Computational Applied Mechanics, 51(1), 91-97. 2020.

[21] Xu, M., Yang, J., Lieberman, I. H., & Haddas, R. Finite element method-based study of pedicle screw–bone connection in pullout test and physiological spinal loads. Medical engineering & physics, 67, 11-21. 2019.

[22] Lee, E. S., Goh, T. S., Heo, J. Y., Kim, Y. J., Lee, S. E., Kim, Y. H., & Lee, C. S. Experimental evaluation of screw pullout force and adjacent bone damage according to pedicle screw design parameters in normal and osteoporotic bones. Applied Sciences, 9(3), 586. 2019.

[23] Moldovan, F., & Bataga, T. Torque Control during Bone Insertion of Cortical Screws. Procedia Manufacturing, 46, 484-490. 2020.

Gmail Irza Sukmana <irza.sukmana@gmail.com>

# [JPL] Editor Decision

2 messages

**Amir Zaki Mubarak** <ejurnal@pnl.ac.id> Thu, Jul 21, 2022 at 9:49 PM
To: "Dr. Irza Sukmana" <irza.sukmana@gmail.com>
Cc: Irza Sukmana <irza.sukmana@eng.unila.ac.id>

Dr. Irza Sukmana:

We have reached a decision regarding your submission to Jurnal Polimesin, "Optimization Design of Bone Crews Biodegradable with Magnesium AZ31B Based on Finite Element Analysis".

Our decision is to: Accept Submission

Amir Zaki Mubarak
Syiah Kuala University, Scopus Author ID: 57190937549
amir_zm@unsyiah.ac.id

---

Editor-in-Chief
Jurnal POLIMESIN
http://e-jurnal.pnl.ac.id/index.php/polimesin

---

**26. Bapak Irza Sukmana.pdf**
220K

---

**Irza Sukmana** <irza.sukmana@gmail.com> Mon, Jul 25, 2022 at 5:48 AM
To: Amir Zaki Mubarak <ejurnal@pnl.ac.id>

Kepada Yth. Bpk Amir Zaki Mubarak,

Terima kasih atas acceptance for our article to be published at Jurnal Polimesin.

Terilampir adalah bukti transfer biaya penerbitan artikel kami tersebut.

Terima kasih banyak.

Regards,
Irza

----
**Assoc. Prof. Irza Sukmana, Ph.D.**
Department of Mechanical Engineering
Faculty of Engineering, Universitas Lampung
Jl. Professor Soemantri Brojonegoro No. 1
Bandar Lampung 35145, Indonesia
phone: +62-812 94836432 | facsimile: +62-721 704947
email: irza.sukmana@gmail.com | http://www.eng.unila.ac.id/

profile: //www.researchgate.net/profile/Irza_Sukmana2

[Quoted text hidden]

**WhatsApp Image 2022-07-24 at 5.37.06 PM.jpeg**
73K