Direct Research Journal of Engineering and Information Technology
Vol.5 (1), pp. 10-15, April 2018
ISSN 2354-4155
DOI: https://doi.org/10.26765/DRJEIT.2018.3904
Article Number: DRJA108263904

Direct Research Journal of Engineering and Information Technology
http://directresearchpublisher.org/journal/drjeit/

Research Paper

Numerical life prediction of lumbar pedicle screw based Construct: Finite element approach

Mosbah Moustafa[1]* and Bendoukha Mohamed[2]

[1]Laboratory of Numerical and Experimental Modeling of Mechanical Phenomena Faculty of Sciences and Technology FST.
[2]Department of Mechanical Engineering, University Abd El Hamid Ibn Badis. Mostaganem .Algeria.
Corresponding author E-mail: scienza87@hotmail.fr

Received 3 March 2018; Accepted 4 April, 2018

This project intended to analyze the fatigue life of a rigid posterior lumbar spine fusion structure, the finite element analysis (FEA) was used to explore the fatigue behavior of the stabilization system. The conventional rigid stabilization is widely being used in orthopedic surgery and presumed to be the gold solution of degenerative disorders (Sengupta et al., 2001). We study the fatigue, deformation and safety factors. It was observed that the simulation results are in good agreement with experimental cases. To acquire a more realistic simulation, boundary conditions were applied according to standards specified in the ASTM Standard F1717-04 (ASTM, 2013). Three-dimensional Model of the stabilization structure with two rods were built by the Solid Works 2013, and imported into ANSYS software for static analysis using ncode design life for fatigue analysis. The maximum and minimum stresses of risk nodes under different loads and moments were located. The fatigue life was then assessed using relevant mathematical formula of S-N curve and Goodman curve. It was found that the stress at the middle of the crossbeam between the two bars is larger than the surroundings and is liable to suffer from fatigue. Three different types of materials were selected during analysis. Two materials were used: Ti-6Al-4V and 316L and UHMWPE. Minimum fatigue cycles, critical fatigue areas, stresses and safety factor values have been identified. The predicted results showed that this system was safe enough in terms of fatigue life. As a result of fatigue analysis, and found to be successful. The current study has also demonstrated that analyzing spinal implants with the FE method and applied with confidence to support standard fatigue testing and used as an alternative.

Keywords: Posterior; lumbar; spine; fatigue; The Finite Element analysis

INTRODUCTION

Pedicle based Stabilization system is the successfully applied implantation to the patients affected by spinal disorders and fractures, degeneration, deformity correction and reconstruction and many other disease treatments (Maserati et al., 2010). Although many studies have been done concentrating on the biomechanical aspects of pedicle screws (ASTM, 2013; Oktenoglu et al., 2014). Instrumented spinal fusion constructs are aimed to provide intermediate stability and assure successful fusion. The natural rate of bone healing and remodeling exposes the metallic construct to duty cycles for up to 12 months postoperative term (Sears et al., 2011; Maserati et al., 2010). The forces applied to the implant during human activity generate dynamic stresses varying gradually and may result in fatigue failure of an implant. Therefore, it is important to ensure that spinal implant endure against fatigue failure and damage. Fatigue testing of the fusion system is a significant part of the design approval of implant. In order to confirm this, the construct should be tested according to international testing standards (Kashkoush et al., 2016) in which the implant should survive for minimum cycle of 5 million. In this work, the FE was used to simulate fatigue damage of implants under static loads (Lai et al., 2009; Teoh, 2000). The results have been compared to the material fatigue strength (ASTM, 2013, Teoh, 2000). The analysis of stabilization construct was implemented into ANSYS ncode design life. The force was applied on the upper UHMWPE block gradually from 100-220 N as defined in the ASTM Standard F1717-04 test standard (Figure 1). In this study, in order to assess the fatigue life changes and the fatigue behavior, we have used two types of materials: Ti-6Al-4V alloy, and stainless steel 316 L.

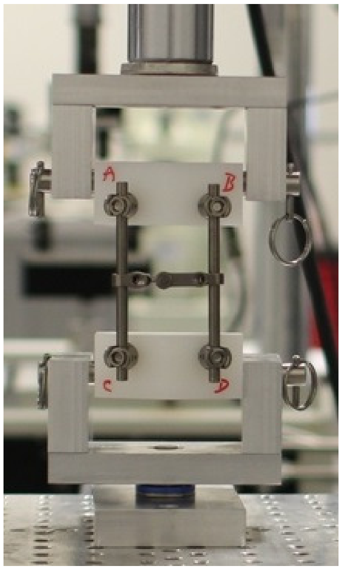

Figure 1. ASTM standard ISO F1717

Finite element analysis

The finite element criterions were used to quantify the fatigue life, stress distribution and critical location of the life factors. The material properties, loading history and geometry of the stabilization construct were input data to the analysis (Figure 2). The model used in this study is composed of four pedicle screws, two rigid rods and a rigid trans-connector according to the ISO 1717 test conditions. The pedicle screws were embedded in block at a specific angle as shown in (Table 1) in frontal plane. The block and the screws were meshed using a higher order three dimensional solid element (solid 187) which is suitable for modeling the complex geometry. The rods were meshed using solid 185. Contact and sliding between screws and block, interfaces modeled with contact (CONTA174 and TARGE 170) elements. The contact elements themselves overlay the solid elements describing the boundary of a deformable body and that are in contact with the target surface. The average number of elements is 1430 hexahedral element and 81000 tetrahedral elements (Figures 1 and 2).

Static model

The stabilization structure is pierced into lumbar spine through pedicle screws. The force is shared through the fixed structure to shield the treated level. The fusion rate of the patients is higher if the spinal internal fixation is used (Tsuang et al., 2009; Sears et al., 2011). The pedicle screws and their host structure are characterized as an integral part in this model (Figure 2).

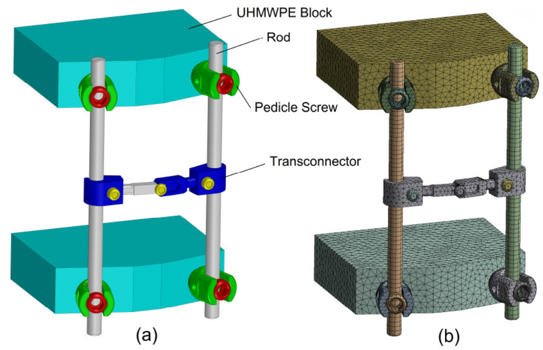

Figure 2. (a) 3-D model (b) Finite element model.

Model formulation and boundary conditions

The detailed 3D model was created using the 3D commercial solid-building software Solid Works 2013. Then was imported into Workbench statics analysis module of ANSYS 15.0 (Figure 3). We applied the tetrahedron method to generate the grid partition. The rods were meshed with 1.5 mm unit size using Brick 8 nodes element. Solid185 unit element was applied on two UHMWPE holding blocks, screws, and trans-connector. The meshed model was shown in (Figure 2). According to the workflow of fatigue test ASTM standard ISO F 1717 and based on the behavior of vertebral body bending motions, a compressive load, a tensile load, a bending moment and a lateral load were applied gradually for this assessment (Table 1 and Figures 3-5). Compressive- tensile loads of 100-220 N, bending moments of 200-500 Nmm, and lateral loads of 20-80 N were incrementally used.

Fatigue calculation

The analysis by ANSYS ncode design life module was carried out for the lumbar stabilization system with four corresponding loads. The fatigue lives of lateral load type were presented in (Figure 7).

RESULTS AND DISCUSSION

Pairs of compressive-tensile loads, bending moments

Table1. Mechanical properties of materials used in analysis.

Part	Material	Elastic Modulus (GPa)	Poisson's Ratio	Yield Strength (MPa)	Ultimate Strength (MPa)
Rod	Ti-6Al-4V [Teoh, 2000]	113.8	0.33	880	950
Screw Tans-connector	Stainless Steel (316L) (SpecSearch, 2011; Teoh, 2000)	195.0	0.30	170	480
Block	UHMWPE (Mamdouhv et al., 2012)	1.2	0.4		

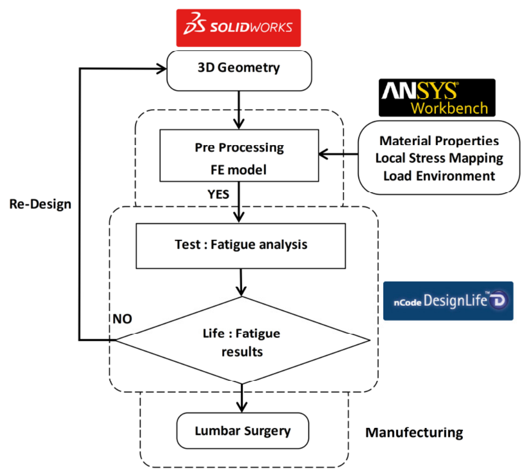

Figure 3. Whole process of FE-Model Formulation and Fatigue analysis prediction strategy.

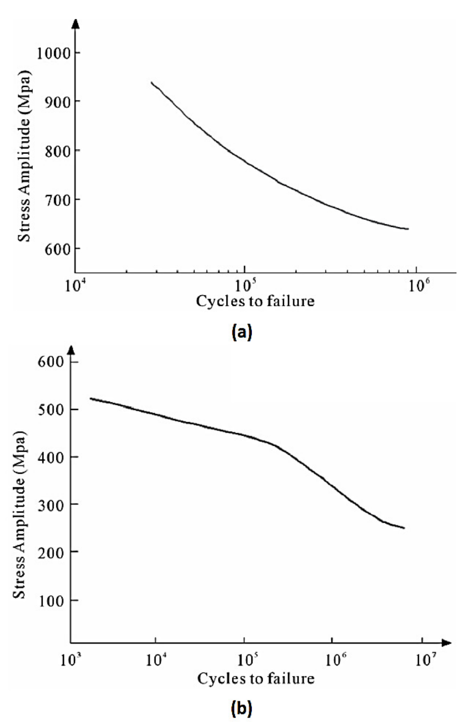

Figure 4. S-N curves used for (a) 316L (SpecSearch, 2011); (b) Ti4Al6V (Teoh, 2000).

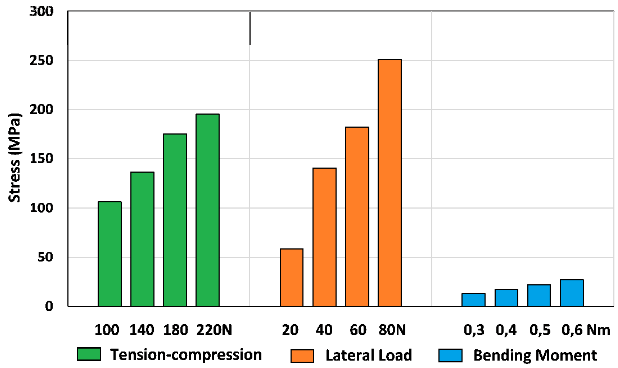

Figure 5. Stress in the rods for different loading cases.

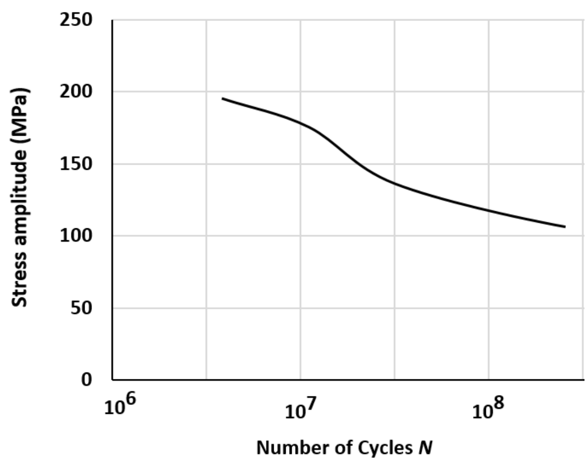

Figure 6 S-N curve of the rod.

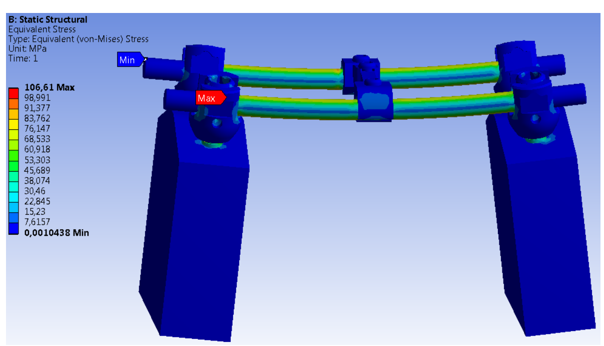

Figure 7. Stress distribution for 100N (a) compression.

and a single lateral load were added on holding blocks. The stress clouds, safety factor, biaxial indication, life, damage were shown in (Figures 7-11). Maximum stress reached a maximum during Lateral load comparing to Bending moment. Figure 6 shows the good agreement of SN curve of the rod made of Ti4Al6V comparing to experimental S-N curve of Ti4Al6V material (SpecSearch, 2011). Figures 7 and 8 show that higher stress is distributed in the connection area of the screw head and the rod when the structure is loaded with compressive, tensile and lateral force. When the structure is loaded with bending moment, higher stress locates in the area of the middle of the rod, as shown in (Figures 7 and 8). Majority of the structure is under a pure uniaxial

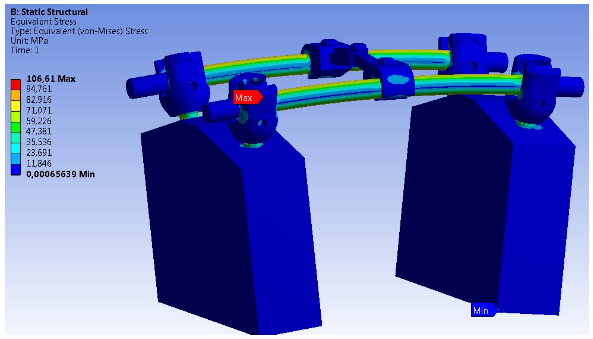

Figure 8. Stress distribution for 100N (a) compression; (b) Tension.

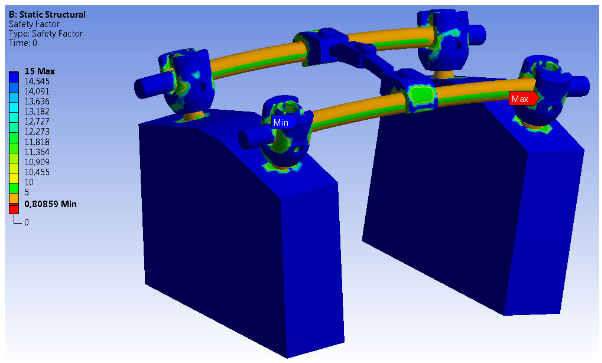

Figure 9. Safety factor distribution for compression.

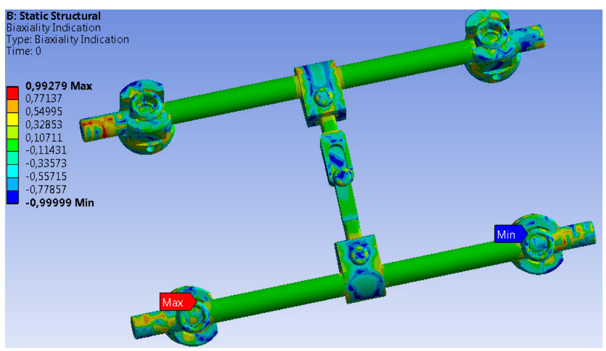

Figure. 10 Max Stress Biaxiality Indication for bending moment.

stress, with region exhibiting both pure shear and nearly pure biaxiality, because the most damaged point located at the middle region of the rod which arises at a point of closely uniaxial stress. The stress was drastically increased for lateral load compared with the Tension-Compression condition (Figure 4).There were significant differences observed within the life distribution and safety factor between different regions of the construct tested. There were no life differences at the level of screws and trans-connector connection area, the life was increased by greater than middle region in all modes of loading. (Figures 9 and 11).

Fatigue analysis

The finite element method was used to evaluate the implant in terms of the fatigue life safety factor. Fatigue lives of pedicle constructs particularly the rods were calculated based on the Goodman mean-stress fatigue

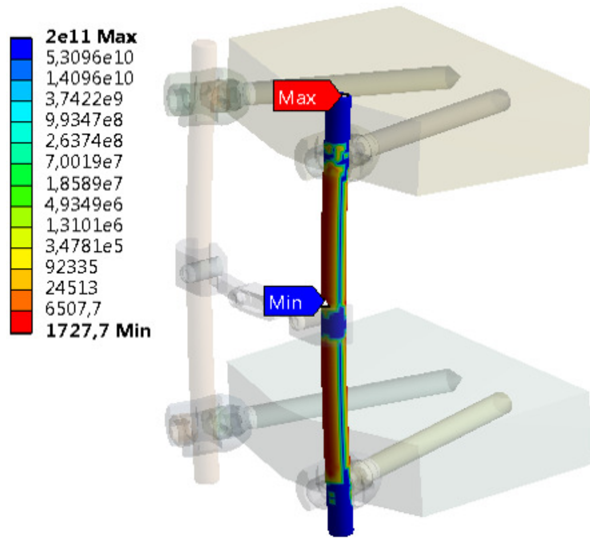

Figure 11 Life distribution.

theory. According to the modified Goodman theory fatigue life, the relation between alternation stress σ_a and the mean stress σ_m is:

$$\sigma_a = \frac{\sigma_{\max} + \sigma_{\min}}{2}$$

$$\sigma_m = \frac{\sigma_{\max} - \sigma_{\min}}{2} \qquad \text{(Lai et al., 2009)}$$

Conclusion

The life distribution and the stress distribution of the structure was studied under four different types of loads using Ansys Ncode design life module. The most loaded part under compression moment is the central part of the rod. When the fixed structure was loaded with compressive-tensile load, lateral load, the most fragile part is the connection area between screw head and the rods. The fatigue life numbers could be calculated through S-N curve and Goodman curve under different types and different values of loads. The results show that the fatigue life decreases rapidly as the value of the load increases by contrary of the stress magnitude.

Significance

Our anatomical FE-model describe the mechanical influence of the experimental loading cases on the lumbar spine fixation construct and it seems to play a relevant knowledge of the construct life and damage before using to treat spinal disorders as well. As discussed, it could mechanically justify the presence and identify the influence of some components choosing the optimized material and design to explore the mechanical role of each components of the construct and improve life factors of the spinal structures. Using rigid fixator to fusion implant system provides additional rate of fusion success however it induces stress shielding. Further studies can expand the simulations to the clinical relevance due to complex physical relevance using dynamic constructs.

REFERENCES

ASTM Standard F1717 (2013). “Standard Test Methods for Spinal Implant Constructs in a Vertebrectomy Model,” ASTM International, West Conshohocken, http://dx.doi.org/10.1520/F1717.

Oktenoglu T, Erbulut D.U, Kiapour A, Ozer AF, Lazoglu I, Kaner T, Goel VK (2014). Pedicle screw-based posterior dynamic stabilisation of the lumbar spine: in vitro cadaver investigation and a finite element study. Computer Methods in Biomechanics and Biomedical Engineering. doi:10.1080/10255842.2014.890187

Kashkoush A, Agarwal N, Paschel E, Goldschmidt E, Gerszten PC(2016). Evaluation of a Hybrid Dynamic Stabilization and Fusion System in the Lumbar Spine: A 10 Year Experience. Cureus. 10;8(6):e637. doi:10.7759/cureus.637

Tsuang YH, Chiang YF, Hung CY, Wei HW, Huang CH, Cheng CK (2009). Comparison of cage application modality in posterior lumbar interbody fusion with posterior instrumentation—A finite element study. Medical Engineering & Physics 31:565–570.

Sears WR, Sergides IG, Kazemi N, Smith M, White GJ, Osburg B(. 2011). Incidence and prevalence of surgery at segments adjacent to a previous posterior lumbar arthrodesis. Spine J.11(1):11–20. doi:10.1016/j.spinee.2010.09.026

Lai HS, Li YJ, Xuan FZ, Tu SD (2009). “High Cycle Fatigue in Ti-6Al-4V Titanium Alloy,” *Journal of Nanjing University of Technology*, Vol. 31(5):15-19.

SpecSearch (2011). AISI type 316L. http://www.efunda.com/materials/alloys/stainless_steels/show_stainless.cfm?ID=AISI_Type_316L&show_prop=all&Page_Title=AISI%20Type%20316L

Teoh SH (2000). Fatigue of biomaterials: A review. *International Journal of Fatigue*, **22**, 825-837. doi:10.1016/S0142-1123(00)00052-9

Mamdouh M. Monif Teoh, S.H (2012). Finite element study on the predicted equivalent stresses in the artificial hip joint. J. Biomedical Science and Engineering, 5:43-51. http://dx.doi.org/10.4236/jbise.2012.52007

Maserati MB, Tormenti MJ, Panczykowski DM, Bonfield CM, Gerszten PC (2010). The use of a hybrid dynamic stabilization and fusion system in the lumbar spine: preliminary experience. Neurosurgical focus. 28(6):2.

Sengupta DK, Webb JK, Mulholland RC (2001). Can soft stabilization in the lumbar spine unload the disc and retain mobility? A biomechanical study with fulcrum assisted soft stabilization on cadaver spine. ISSLS Annual Meeting; 2001 Jun 19-22; Edinburgh, Scotland.