Proceedings of the 9th World Congress on Mechanical, Chemical, and Material Engineering (MCM'23)
Brunel University, London, United Kingdom - August 06-08, 2023
Paper No. ICMIE 149
DOI: 10.11159/icmie23.149

L-Brackets for Heavy Duty Shelves: Stress Analysis

Ahmad Eshaiyan
Supervisor: **Dr. Murat Demiral**
Department of Mechanical Engineering, American University of the Middle East
250 Street, Eqaila, Kuwait
47292@aum.edu.kw

Abstract – L-brackets are a **s**imple rigid mechanical structure utilized for fixing two components where one component which is usually the smaller component is horizontal (shelf), and the other component which is usually the larger component is vertical (wall). L-brackets are acknowledged to be used for several distinct applications and in countless industries; however, they are commonly used for fixing static structures for housing and construction applications. In engineering, as much as it is of high importance that a product must be functional, it is also crucial that the product must be safe whilst using as minimal material as possible to acquire an optimal design. The purpose of this paper is to conduct static analysis and illustrate the influence and effect of certain parameters on the safety and functionality of the design using finite element analysis (FEA), then optimizing the structure of our proposed design using SOLIDWORKS software in order to obtain a minimum satisfactory factor of safety equivalent to two according to Von Mises criterion.

Keywords: L-bracket, Shelves, Finite Element Analysis (FEA), Factor of safety, Von Mises, Displacement.

1. Introduction

L-brackets used for shelves possess a simple but effectual design and structure that issues support for shelving units. The design of the L-bracket contains a horizontal arm which is the longer arm to carry the shelf and the vertical arm which the shorter arm is fixed to the wall or another vertical surface of a structure. This design minimizes the presence and effect of structural issues by distributing the load evenly across the length of the bracket. The dimensions of the L bracket that are used for shelves vary depending on the application they are used for. For heavy duty applications, larger brackets with a higher thickness are utilized to provide support and ensure that the shelf will not fail or collapse. On the contrary, for bookshelves or decorative purposes, smaller brackets are considered. The materials that are used for the L-brackets are also dependent on the type of application they are used in; Metals and plastics are commonly used for the brackets in which metals are used for heavy duty applications due to their properties. Plastics are used for decorative purposes as they might seem more aesthetic than metal whilst also weighing less. In addition, there are several types of L brackets used for shelves in which some brackets may contain an arm to furnish a decorative spark instead of being a plain and simple design. Conversely, for heavier shelves, brackets contain an arm which is known as a gusset for supplemental support to ensure a static and stable shelf. L brackets used for shelves are designed for the user to be able to install them easily in which most brackets contain predrilled holes or slots to be screwed or fastened through. The simple but efficacious design of the L-bracket for shelves makes them a common and excellent alternative for home and business owners searching for a dependable solution for shelving units. Finite element analysis is a method used to represent a complex structure or object by dividing the structure into numerous smaller elements to ameliorate the accuracy of the representation, by simulating a physical object to understand its nature when put under physical conditions [1]. By utilizing the significance of FEA, we are able to acknowledge the behaviour of our L-bracket model when applying or changing certain parameters and optimizing the structure of our design to be optimal without needing to produce and test any physical prototypes.

2. Literature Review

M. Aisha and I. H. Shanono [2] performed strength analysis and structural optimization on an L-shaped bracket using finite element analysis (FEA) to determine the optimum shape and size in which the L-shaped bracket would be considered the safest whilst removing unneeded material without altering specifications within the design constraint; therefore, reducing cost and reducing the weight of the structure. They used structural analysis to assess the endurance of the bracket when put through a load and a disturbance stress, along with deformation characteristics. D. Srivastava et al. [3] conducted analysis on an L-shaped bracket used as a support for bookshelves with a uniform distributed load applied on the top surface and calculated the value of Von Mises stress, principal stress, and the total deformation of the structure on two dissimilar structures where one is a simple geometry and the other is a complex geometry using ANSYS to find out which structure is more superior than the other. A. B. Raghunandan et al. [4] analysed and made comparisons between fly-ash-epoxy composite structures with varying volume fractions of fly-ash and fly-ash content (5%, 10%, 15%) and alloys for bracket applications. They modelled three different prototypical brackets and conducted analysis on each bracket by the utilization of finite element analysis (FEA) in which the results they obtained from the analysis were compared to the results obtained from common alloys used in the manufacturing of L-shaped brackets. The main objective of their research was to model a fly-ash epoxy composite using finite element analysis and to compare the simulation results with the properties of alloys that are used mainly in the present market. S. Suryawanshi et al. [5] aimed to analyse a guide bracket using two different methods of finite element mesh (FEA) in which the methods are the tetrahedral meshing and the hexahedral meshing and made comparisons to find out which method is more superior in terms of accuracy. The guide bracket was designed in SOLIDWORKS software and then it was imported to ANSYS workbench for finite element analysis (FEA). In the FEA, the structural analysis is performed; constraints such as stress, deformation of different materials, toughness, elasticity etc. were applied. A.S. Adkine et al. [6] wrote this paper about modal analysis of engine brackets using finite element analysis. The Engine supporting bracket is used to reduce noise, vibration, and harshness. They used a CAD model of an engine supporting bracket created in CATIA software and analysed for stress and vibration analysis using ANSYS workbench. Modal analysis was used to check for output responses in terms of equivalent Von-Mises stress, deformation and strain energy absorbed. N.R.E. Domingues et al. [7] produced this research paper which discusses the experimental and numerical failure analysis of aluminium/composite single L-joints. They aimed to study single L-joints between aluminium components and carbon-epoxy composites under a peel loading. The numerical analysis included stress distributions, damage evolution, strength, and failure modes. A. R. Jaware & Dr. K.B. Kale [8] conducted research on a Compressor ascent bracket. Mostly done in clay software and analysed application ANSYS Software. the core of this study and its problem was the reduction in weight, vibration, and acoustics of the compressor-mounted automotive part. Dr. V. N. Rao et al. [9] found out that Topology optimization could be an exceptionally capable instrument in numerous ranges of plan such as optics, hardware, and structural mechanics. The field materialized from structural plan and so topology optimization connected in this setting is additionally known as structural optimization. In this work, a software-based approach for topology optimization of an L-shaped bracket to attain 60% weight lessening subjected to different stacking conditions is displayed through a commercially accessible finite element re-enactment computer program.

3. Problem Definition

When designing a structure, it is pivotal to be aware of what parameters have influence on the safety and reliability of the structure to efficiently produce a functioning and reliable product. We have decided to analyse the effect of each of the following parameters of the L-brackets: changing the length of the protruded arm (the arm holding the shelf), changing the thickness, changing the diameter of the clearance holes, the effect of applying a gusset, and the effect of applying a fillet edge at the critical location. In addition, after inspecting numerous models of L-Brackets in the market, we have come to acknowledge that each L-Bracket is designed to only withstand a certain amount of load, in which the L-Brackets that bear less significant loads usually acquired a smaller geometry or size; on the other hand, the L-Brackets that had a high load bearing capacity, were larger in size. Our secondary objective of this research is to design and optimize an optimal L-Bracket that could bear loads up to 50 kilograms with as minimal amount of material as possible whilst maintaining an acceptable factor of safety (no less than two) to be used for heavy duty shelves.

4. Numerical Modelling

A three-dimensional finite model of the reference L-bracket was developed via SOLIDWORKS software. After applying mesh convergence on the finite element model, we found out that the difference in the values obtained at a single element using medium sized mesh and finest mesh is negligible; consequently, we elected medium sized mesh for all our simulations. Additionally, the geometry was fixed at the clearance holes parallel to the wall and the applied load was a point load at the centre. When choosing the material, one must consider certain material properties depending on the application that is in consideration. In our case, regardless of the fact that all material properties must be observed, Young's modulus (the ratio of stress to strain), Poisson's ratio (the ratio of transverse strain to longitudinal strain), and yield strength are the most important material properties to contemplate. As a result, we chose AISI 4340 steel, normalized due to its high yield strength (710 MPa) to guarantee a safe design.

Table 1: Material Properties of AISI 4340 Steel, Normalized

Property	Value	Units
Young's Modulus	20500	MPa
Poisson's Ratio	0.32	N/A
Yield Strength	710	MPa
Mass Density	7850	Kg/m^3
Tensile Strength	1110	MPa
Shear Modulus	80000	MPa

5. Design Specifications

Before proceeding with simulation and analysis, we defined several factors before proceeding to be able to advance without hesitation by making our path clear and decisive without having to reconsider any external factor that could be affecting our results or influencing them; In addition, we set these factors as a reference to visualize and understand the effect if we applied any changes. For the dimensions of the reference L-bracket model, the dimensions of the bracket are: 150 millimetres length (protruded arm length), 100 millimetres height (vertical arm), two millimetres thickness, 20 millimetres extruded bracket length, and five millimetres diameter for all the clearance holes (four clearance holes). In addition, the material we used was AISI 4340 steel, normalized for the bracket, Balsa wood for the shelf, Hex Screw Grade AB ISO 4041 to fix the L-bracket to the shelf. Also, the shelf dimensions are 1000 millimetres length, 300 millimetres width and 30 millimetres height (thickness). Finally, for the load we assumed the weight to be placed on the shelf to be 50 kilograms. Therefore, the load is 490.5 Newtons ($P= m \cdot g= 50kg \cdot 9.81m/s^2$).

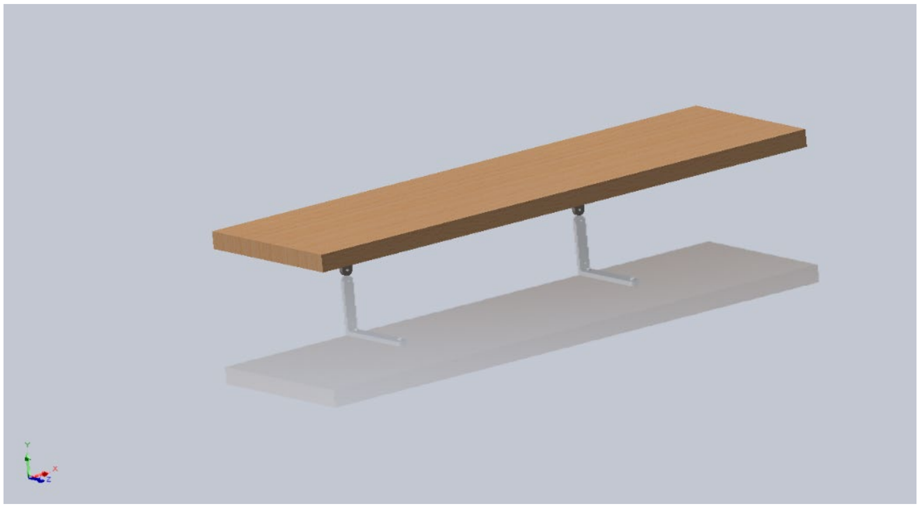

Figure 1: Assembly of Our Model (Shelf and L-Brackets)

6. Simulation

For the simulations and analysis, we structured a plan to be able to visualize the effect of each parameter clearly by simulating once using a smaller value than the value of the reference model, and another time using a value larger than the value of the reference model; however, for the thickness we demonstrated its influence on the bracket by using values larger than that of the reference model twice to be able to show clear results since the reference model is two millimetres thick. First and foremost, we ran a simulation on the reference model to have the results as a reference for conducting comparisons. Secondly, for the thickness we ran the simulation once using four millimetres thickness and once using six millimetres thickness. For the length of the protruded arm, we simulated once using 125 millimetres length and once using 175 millimetres length. For the diameter of the clearance hole, we used four millimetres diameters and six millimetres diameters. Moreover, we ran the simulation with a model that does not contain a fillet edge and another time with a model that contains a gusset. Lastly, we ran a simulation with eight millimetres thickness and no gusset to minimize the use of material whilst also maintaining a safe and reliable design.

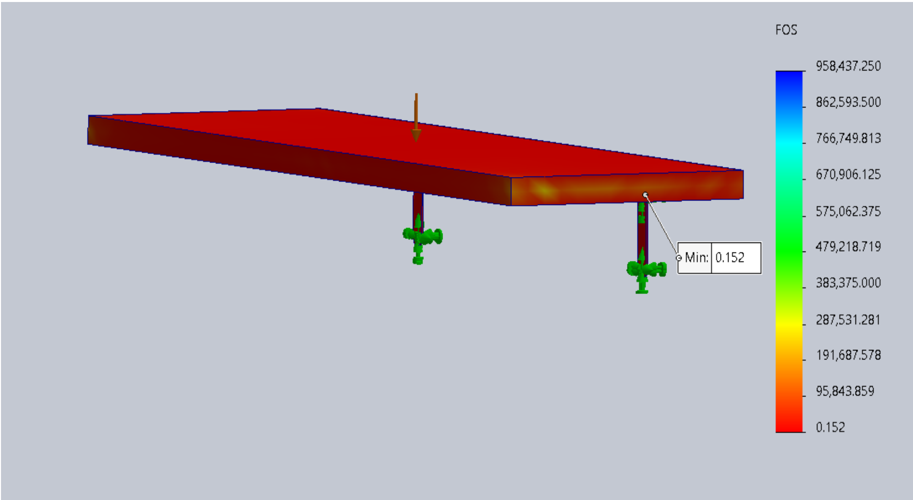

Figure 2: Factor of Safety Plot of Reference Model

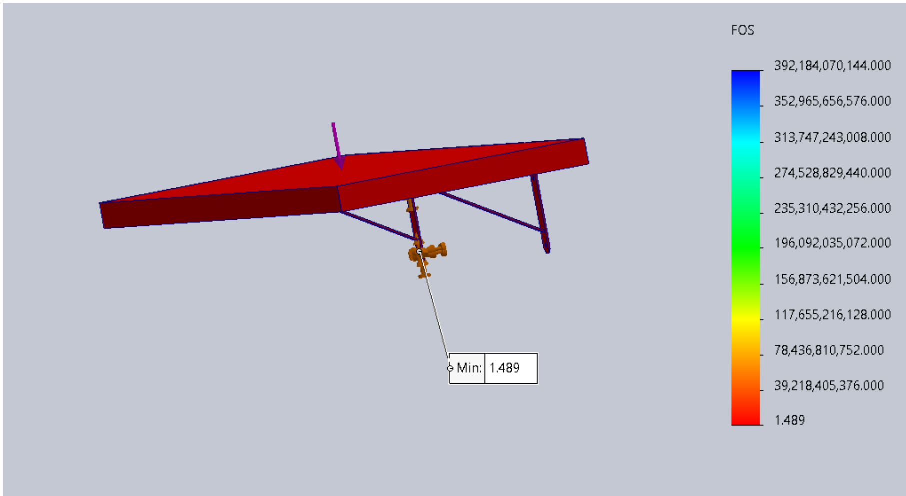

Figure 3: Factor of Safety Plot of the L-bracket with a Gusset

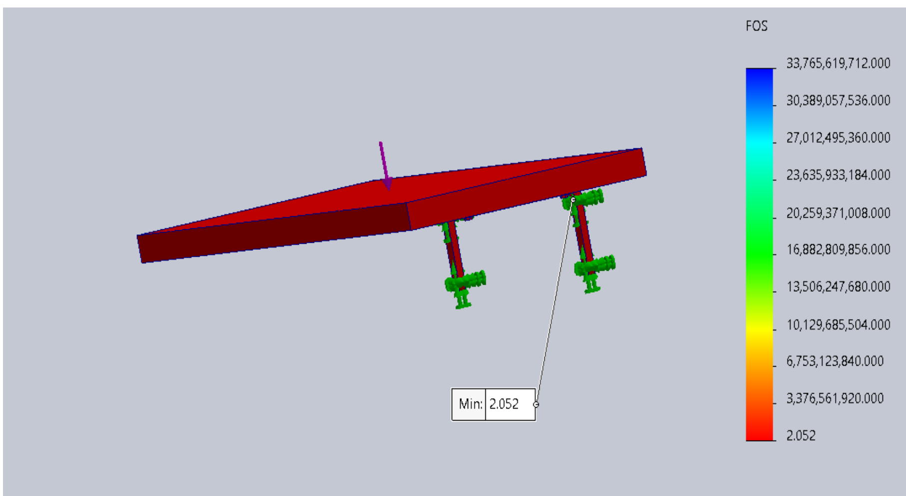

Figure 4: Factor of Safety Plot of L-Bracket with 8mm Thickness

7. Results and Discussion

After conducting simulations on all the previously mentioned parameters, we have come to understand that each parameter affects the bracket differently in which some parameters had a very minor effect when changed. For instance, when we changed the length of the arm that is anticipating the shelf (the protruded arm),we realized that the change that occurred was negligible due to the values we obtained for Von-Mises stress, displacement, factor of safety; therefore, it has very little influence on the bracket and would not be considered a decisive factor that would play a significant role in determining the reliability of the L-bracket. When we changed the diameter of the clearance holes to four millimetres, the Von-Mises stress and displacement decreased but very slightly, however, we came to realize that as we increased the diameter to six millimetres the values of stress and displacement decreased, and the minimum factor of safety increased but not substantially. When excluding the fillet edge from our design, the values of Von Mises stress and displacement decreased slightly and the factor of safety remained around the same value making it a negligible factor. When changing the thickness to four millimetres, the maximum stress dropped remarkably from 4684.452MPa to 1101.092MPa, the displacement dropped notably from 58.403 millimetres to 5.993 millimetres and the factor of safety increased from 0.152 to 0.645; In addition, when increasing the thickness to six millimetres, the stress decreased to

590.799MPa, displacement decreased to 1.923mm and the factor of safety increased to 1.202 making the design safe but not satisfactory; but, we understood the vast importance and influence of changing the thickness on our design. However, when applying a gusset to our reference model the Von-Mises stress decreased significantly from 4684.452MPa to 476.906MPa, the displacement also decreased dramatically from 58.403mm to 0.572mm, and the factor of safety increased from 0.152 to 1.489 making the gusset the parameter with the greatest influence on the safety and dependability of the design. Finally, we increased the thickness to 8mm without applying a gusset and achieved a factor of safety of 2.052 with the stress decreasing to 345.952Mpa and the displacement decreasing to 0.898; regardless of the significant influence the gusset depicted, our goal was to achieve the optimal design whilst remaining within the design constraints by minimizing material and maintaining a safe and reliable design with a adequate factor of safety of minimum two.

Table 2: Results for All Simulations

L-Bracket Models	Von Mises Stress (Maximum)	Factor of Safety (Minimum)	Displacement (Maximum)
Reference Model	4684.452 MPa	0.152	58.403mm
4mm Thickness	1101.092 MPa	0.645	5.993mm
6mm Thickness	590.799 MPa	1.202	1.923mm
125mm Length	4260.741 MPa	0.167	57.071mm
175mm Length	4316.682 MPa	0.164	57.028mm
4mm Diameter	4521.031 MPa	0.157	58.480mm
6mm Diameter	3823.907 MPa	0.186	49.748mm
No Fillet	4047.940 MPa	0.175	44.836mm
Gusset	476.906 MPa	1.489	0.572mm
Optimal Design 8mm Thickness	345.952 MPa	2.052	0.898mm

8. Conclusion

L brackets are an unembellished but effective rigid mechanical structure employed to support structures and components such as shelves in which brackets can come in different sizes and dimensions depending on their use and application. After completing research and applying what we know and the skills we possess, Also, we designed an optimal safe design with a factor of safety equivalent to two whilst remaining within the design constraints and without compromising on strength and safety.

9. References

[1] X. Chen and Y. Liu, *Finite Element Modeling and Simulation with ANSYS Workbench*, 2nd ed. Taylor & Francis Group: CRC Press, 2019, pp. 1-16.

[2] M. Aisha and I. H. Shanono, "Strength analysis and structural optimisation of an l-shaped bracket," *Nigerian Journal of Technological Research*, vol. 13, no. 2, p. 14, Dec. 2018, doi: 10.4314/njtr.v13i2.2.

[3] S. DIVYA and M. R.N., "STRUCTURAL ANALYSIS OF L-BRACKET USING ANSYS," *i-manager's Journal on Mechanical Engineering*, vol. 7, no. 2, p. 17, 2017, doi: 10.26634/jme.7.2.13432.

[4] A. B. Raghunandan, D. S. Chiniwar, S. Hiremath, P. Sondar, and H. M. Vishwanatha, "Modelling and Comparative Analysis of Epoxy-Fly-Ash Composite with Alloys for Bracket Application," *Journal of Composites Science*, vol. 6, no. 12, p. 358, Nov. 2022, doi: 10.3390/jcs6120358.

[5] S. Suryawanshi and A. Hange, "Comparative Analysis of Tetrahedrons and Hexahedrons for Finite Element Mesh during Static Structural Analysis of Guide Bracket," *International Research Journal of Innovations in Engineering and Technology (IRJIET)*, vol. 5, no. 5, Art. no. ISSN (online): 2581-3048, May 2021, doi: 10.47001/IRJIET/2021.505002.

[6] A. S. Adkine, Prof. G. P. Overikar, and Prof. S. . S. Surwase, “Modal Analysis of Engine Supporting Bracket using Finite Element Analysis,” *International Journal of Advanced Engineering Research and Science*, vol. 4, no. 3, pp. 55–63, 2017, doi: 10.22161/ijaers.4.3.9.

[7] N. R. E. Domingues, R. D. S. G. Campilho, R. J. C. Carbas, and L. F. M. da Silva, “Experimental and numerical failure analysis of aluminium/composite single-L joints,” *International Journal of Adhesion and Adhesives*, vol. 64, pp. 86–96, Jan. 2016, doi: 10.1016/j.ijadhadh.2015.10.011.

[8] A. R. Jaware and Dr. K. B. Kale, “Vibration Analysis and Weight Reduction for Compressor Mounting Bracket Using Composite Material ,” *International Research Journal of Engineering and Technology (IRJET)* , vol. 5, no. 8, Art. no. e-ISSN: 2395-0056, Aug. 2018.

[9] C. Ramana, P. V. Kumar, Y. K. Rao, and Dr. V. N. Rao, “STRUCTURAL OPTIMIZATION OF AN L-SHAPED BRACKET SUBJECTED TO DIFFERENT LOADING CONDITIONS,” *International Research Journal of Modernization in Engineering Technology and Science*, vol. 2, no. 10, Art. no. e-ISSN: 2582-5208, Oct. 2020.