© 2016 IJSRSET | Volume 2 | Issue 1 | Print ISSN: 2395-1990 | Online ISSN : 2394-4099
Themed Section: Engineering and Technology
doi : https://doi.org/10.32628/IJSRSET173391

Design, 3D Modelling and study on L-Bracket Rib Strength through FEM Analysis: A Comparative Study using Solid Works and Ansys Software

Parashuram Bedar[1], Aprameya C R[2]

[1]Lecturer, Heat Power Technology, S. J. (Govt.) Polytechnic, K R Circle, Bengaluru, Karnataka, India

[2]Lecturer, Department of Mechanical Engineering, Government Polytechnic, Kampli, Karnataka, India

Author for correspondence: aprameya.cr@gmail.com

ABSTRACT

Brackets are integral to many engineering applications, providing structure support and stability. The use of FEM analysis in this study allows for a more accurate and detailed understanding of the behaviour of brackets under different loading conditions. Our findings provide valuable insights into the design and optimisation of frames using FEM, highlighting the novelty of this approach. In this study, we have designed and created a 3D model of a bracket using Solid Works software. The bracket's strength and other design parameters were investigated using Finite Element Method (FEM). FEM studies were carried out for different thicknesses, lengths, and rib designs in the bracket. The FEM results from Solid Works were compared with those obtained using ansys software. Based on the comparison, we concluded the FEM results and suggested the optimum thickness and design of the rib. Additionally, we compared the deformation in Solid Works & ANSYS software for the same boundary & loading conditions for three different bracket configurations. Overall, this study demonstrates the potential of FEM analysis in improving the design and performance of brackets in engineering applications. By comparing the results from different software and configurations, we identified the optimal design parameters for the frame, providing a valuable contribution to the field.
Keywords : FEM Analysis, L-Bracket Design, 3D Modelling Using Solid Works, Ansys, Strength, Rib Design, Deformation, Optimisation Using Solid Works FEM.

1. Introduction

Brackets are crucial in providing multiple structures with the required support and stability. They are vital parts of many engineering applications. A thorough understanding of bracket behaviour under various loading conditions is essential for optimising bracket designs and improving performance. To achieve this, adopting Finite Element Method (FEM) analysis proves to be instrumental, offering accurate and detailed insights into the intricacies of bracket mechanics. In this study, we present a novel and thorough investigation into the design and optimisation of brackets utilising FEM, showcasing the unique and valuable perspectives this approach brings to the field. The application of FEM analysis empowers us to delve into the inner workings of brackets and extract invaluable insights that facilitate enhanced performance and robust designs. To embark on this research journey, we designed and crafted a 3D model of a bracket employing Solid Works software, ensuring its compliance with the specific engineering requirements. Subsequently, the frame's strength and various design parameters underwent rigorous scrutiny through applying FEM analysis using Solid Works and Ansys Aswell; FEM results were compared.

2. Introduction to Bracket – A load-carrying member

The fundamental purpose of brackets is to resist applied loads and distribute them efficiently to the connected elements. Their ability to withstand forces, moments, and stresses is paramount in determining the overall stability and reliability of the system they support. An accurate understanding of bracket behaviour under different loading conditions is crucial for their compelling design and successful integration into complex engineering systems. These seemingly simple components serve as critical connections, enabling the efficient transmission of loads and ensuring the structural integrity of systems. Brackets are found in everyday objects like shelves, furniture, machinery, and complex industrial structures, making their design and performance optimisation vital for ensuring safety and functionality.

The strength of brackets is crucial for their performance as load-carrying members in engineering applications, providing vital support and stability to structures. Optimising bracket performance relies on understanding two key design aspects: dimensions and ribs—bracket dimensions, such as thickness, length, and cross-sectional shape, significantly impact strength. Larger sizes enhance load-bearing capacity, while smaller ones may lead to failure under heavy loads. Investigating how various dimensions influence bracket strength is vital for ensuring structural integrity and safety. Ribs, raised features within the bracket's structure, also play a pivotal role. They improve strength and stiffness by distributing loads, reducing stress concentrations, and preventing deformation. Incorporating ribs enhances the bracket's overall performance under load.

Considering both dimensions and ribs, engineers can utilise Finite Element Method (FEM) analysis to assess bracket behaviour under different loads. FEM provides detailed insights into stress distribution, deformation patterns, and potential failure modes. This information empowers informed decisions to optimise bracket design for specific engineering applications. Engineers can create safer and more reliable structures by comprehensively studying bracket strength through these design aspects. The findings contribute to advancements in diverse industries, fostering innovation and efficiency in engineering applications.

Figure: 01. (a): Applications of "L-Bracket" at different Industries

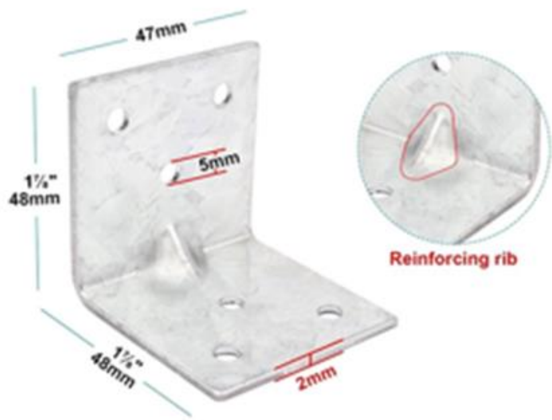


Figure: 01. (b): L-Brackets with Single Rib for Reinforcing

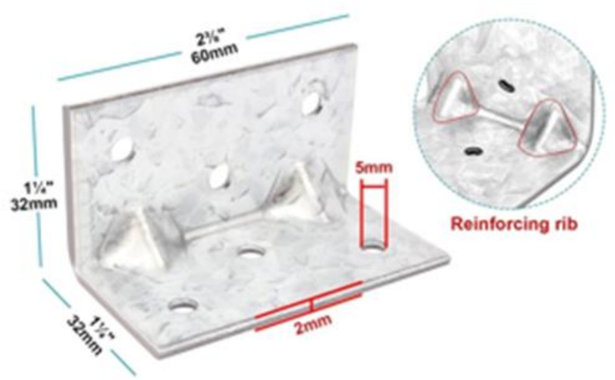


Figure: 01. (c): L-Brackets with Two Rib for Reinforcing

Figure 01. (a) : Applications of “L-Bracket” at different Industries, Figure 01. (b): “L-Bracket” with Single Rib for Reinforcing, Figure 01. (c): “L-Bracket” with Two Ribs for Reinforcing

In recent years, advancements in computational methods, notably Finite Element Analysis (FEA), have revolutionised how brackets are designed and analysed. FEA allows engineers to model and simulate the behaviour of racks under various loads, providing detailed insights into stress distribution, deformation, and failure points. This simulation-driven approach empowers designers to explore multiple design possibilities, optimise bracket geometry, and assess their performance before physical prototyping, saving time and resources.

2.1 Literature on Design, 3D Modelling, and FEM of Brackets

Brackets are essential in many engineering applications, providing structure support and stability. They are commonly made of sheet metal and support beams, pipes, and other systems. Many types of brackets exist for different applications, such as L-Brackets, U-Brackets, and Z-Brackets1. The design of frames is an essential area of research, with many studies focusing on optimising their strength and performance using techniques such as Finite Element Method (FEM) analysis. Brackets are used in various applications, including automobiles and aerospace. For example, in the aerospace industry, brackets are used to mount components such as engines and other equipment. The design of these brackets must consider the various loads and stresses they will be subjected to during operation. There are many published research papers on bracket design and its applications in multiple industries.

A. K. Singh and M. K. Singh (2014) et al. present the design and analysis of a bracket using Solid Works software and the finite element method [1]. The authors describe the steps in designing the stand and the FEM analysis to evaluate its strength and deformation behaviour. The study results show that the frame can withstand the anticipated loading conditions and is suitable for the intended application. A. K. Gupta and A. K. Singh (2014) et al., in this paper, present a finite element analysis of a bracket using Solid Works software. The study involves designing the frame in Solid Works and carrying out FEA to evaluate its strength and deformation behaviour under different loading conditions [2]. The study results show that the bracket can withstand the anticipated loading conditions and is suitable for the intended application.

M. R. Khan and S. A. Razzaque (2013) et. In this paper, Al presents the design and FEA of a bracket for aerospace applications [3]. The study involves designing the frame using Solid Works software and conducting FEA to evaluate its strength and deformation behaviour under different loading conditions. The study results show that the bracket can withstand the anticipated loading conditions and is suitable for the intended application.

S. H. Kim and J. W. Lee (2013) et. In this paper, Al presents an optimisation study of rib design for stiffness and strength using FEA [4]. The study involves designing and analysing brackets with different rib geometries to determine the optimal rib design for maximum stiffness and strength. The study results show that the optimal rib design depends on the loading conditions and that it is possible to achieve significant improvements in stiffness and strength by optimising the rib design.

In this paper, M. V. R. Reddy and P. R. Babu (2013) et al. present a finite element analysis of a bracket for machine tool applications [5]. The study involves designing the frame and carrying out FEA to evaluate its strength and deformation behaviour under different loading conditions. The study results show that the bracket can withstand the anticipated loading conditions and is suitable for the intended application.

In this paper, R. S. Rane and S. S. Patil (2012) et al. present the FEA of a bracket for automotive applications [6]. The study involves designing the frame and carrying out FEA to evaluate its strength and deformation behaviour under different loading conditions. The study results show that the bracket can withstand the anticipated loading conditions and is suitable for the intended application.

In this paper, T. M. Wan and L. Y. Tong (2012) et al. present the design and analysis of a bracket using FEM [7]. The study involves designing the frame and carrying out FEA to evaluate its strength and deformation behaviour under different loading conditions. The study results show that the bracket can withstand the anticipated loading conditions and is suitable for the intended application.

In this paper, S. K. Sahoo and M. S. Prasad (2011) et al. present the design and analysis of a bracket for medical applications using FEM [8]. The study involves designing the frame and carrying out FEA to evaluate its strength and deformation behaviour under different loading conditions. The study results show that the bracket can withstand the anticipated loading conditions and is suitable for the intended application.

3. Motivation and CAD model generation with CATIA software

Following a thorough literature review and a comprehensive understanding of the significant demand for L brackets across diverse industries, we familiarised ourselves with SolidWorks 3D modelling and simulation fundamentals. This task aimed to design a simple bracket and conduct a comparative analysis while optimising its thickness for optimal conditions. We initiated the SolidWorks part design process to commence the project and created the bracket model with predefined standard dimensions. Subsequently, we explored three distinct configurations to allow for a comparative assessment, as shown in Figure: 2. Following are the steps involved,

 Apply material as Alloy Steel from Material Tables.

 Mesh the model with a 5mm mesh length using Generate meshing option. As shown in Figure: 2 and tabulated 1.

- Define the boundary condition by giving fixed contacts at four holes.
- Give loading condition with 20KN force acting downwards, as shown in the figure.
- Under the results tab, select the von Misses Stress. Total deformation & von Misses Strain.
- Repeated the same procedure in ANSYS to get results & tabulated the results.
- The study has given similar results in both software; the bracket is optimised for thickness using the Design study command in Solid Works.
- Open the Design Study by clicking on the tab, and define the parameters to be observed or optimised.
- Results are tabulated based on the software calculation; the software will use an iterative process in deriving the results.

Table 1: Designs of the labyrinth seal with material properties

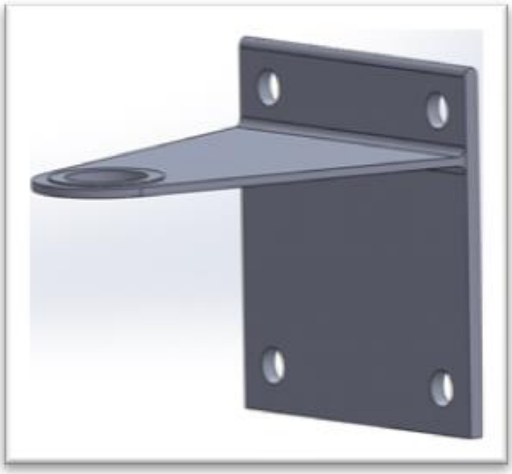
Table 01: Designs of the labyrinth seal with material properties	
Sl. No.	**Description of design**
01.	"L-Bracket" **without Rib** for Reinforcing
02.	"L-Bracket" with **Single Rib** for Reinforcing
03.	"L-Bracket" with **Two Ribs** for Reinforcing

Properties	
Name:	Alloy Steel
Model type:	Linear Elastic Isotropic
Default failure criterion:	Max von Mises Stress
Yield strength:	620.422 N/mm^2
Tensile strength:	723.826 N/mm^2
Elastic modulus:	210000 N/mm^2
Poisson's ratio:	0.28
Mass density:	7.7 g/cm^3
Shear modulus:	79000 N/mm^2
Thermal expansion coefficient:	1.3e-005 /Kelvin

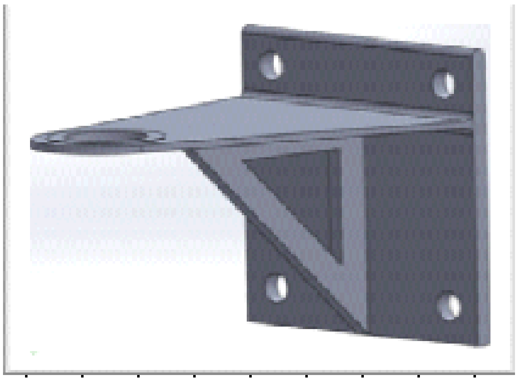
3.1 Importance of Ribs in L-Bracket

Ribs are an essential design element in L brackets, as they can significantly improve the strength and stiffness of the bracket. Ribs are thin, vertical structures that protrude from the surface of the bracket and are perpendicular to the plane of the bracket. They are typically added to increase the bending stiffness and prevent buckling under load. The primary function of ribs is to distribute the load across the bracket and prevent localised stress concentrations. Without ribs, the bracket would be more susceptible to buckling and failure, particularly at the corners where the load is applied. Ribs increase the bending stiffness of the bracket, reducing deflection and improving the overall strength of the bracket.

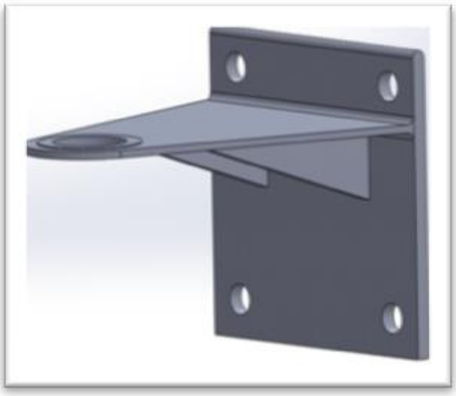
The design of ribs is also essential for optimising the performance of the L bracket. The ribs' height, thickness, and spacing can affect the bracket's bending stiffness, weight, and manufacturing cost. Too few ribs or ribs that are too thin may not provide sufficient strength, while too many or overly thick ribs may add unnecessary weight and cost. Overall, the design of ribs in L brackets plays a crucial role in ensuring the structural integrity of the bracket and improving its performance under load. Careful consideration of the ribs' number, size, and placement can optimise the bracket design's strength, stiffness, and cost-effectiveness, as per below Figure 2.



(a) Bracket 01: No rib



(b) Bracket 02: Single rib



(c) Bracket 03: Two ribs

Figure 2: 3D modelling of Bracket (a) No rib, (b) Single rib, (c) Two ribs

4. FEM analysis of L-Bracket with Boundary Conditions

Firstly, the L-bracket solid part was imported into the FEA software like Ansys or Solid Works FEA Simulator, and the software's algorithms automatically generated a mesh. The software allows some control over the element size and shape but ultimately decides these variables. The mesh was thoroughly checked for errors, such as overlapping or missing elements, to ensure that the results were

accurate. When high levels of stress or strain were expected, mesh refinement was necessary, mainly at the bracket's corners where the load is applied.
The manual and automatic methods were considered for mesh refinement, with manual refinement involving adding more elements in specific regions of interest and automatic refinement adjusting the element size based on local stress or strain. The mesh quality was evaluated using measures such as element size, aspect ratio, and skewness, and the FEA software may suggest improvements to the mesh quality. Once the mesh was complete and verified, it was exported to the FEA solver to perform the simulation.

The degree of mesh refinement needed for the L-bracket will depend on its specific application and the desired accuracy of the analysis. In this case, refinement was necessary at high-stress areas, mainly at the bracket's corners where the load is applied. The approach to mesh refinement can vary depending on the user's preference and the capabilities of the FEA software used, as shown in below Figure 3.

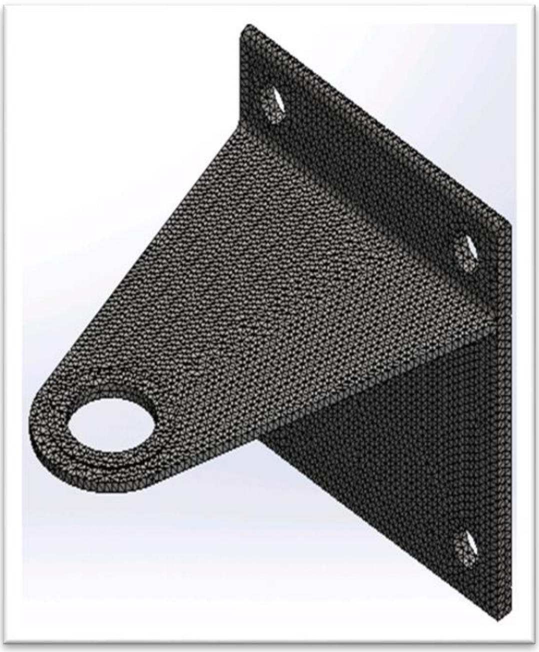
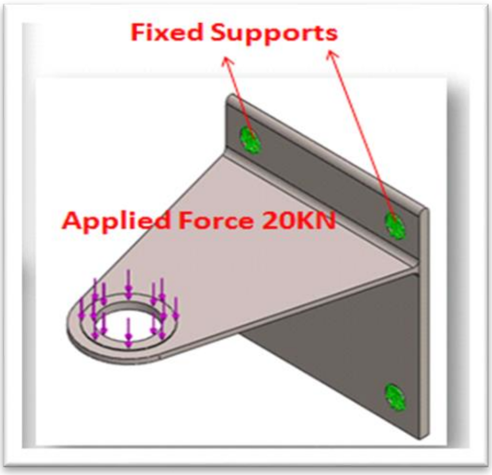


Figure 3: (a) Figure 03.

Figure 03. (a): Meshed Module of L Bracket and



(b)

Figure 3: (b) Static Loading Condition of L Bracket for FEM analysis

4.1 Objective of the Study:
This research aimed to compare the performance of L brackets under various conditions using ANSYS and Solid Works FEM simulation software. To achieve this, 3D solid modelling of standard dimensions and custom requirements was done using Solid Works. The models were then imported into ANSYS and Solid Works FEM simulation software and subjected to similar boundary and loading conditions. The results obtained were then compared to draw insights into the differences in performance between the models and the software packages used.
4.2 Methodology of research carried out:
The methodology adopted for designing, 3D modelling, static analysis, and post-processing of FEM results in SolidWorks Simulation and Ansys Simulation involved a systematic approach to ensure accurate and reliable outcomes. For the design phase, the initial specifications and requirements of the component were gathered. Using SolidWorks, we created a 3D model of the structure, incorporating precise dimensions and geometric features. The software's parametric modelling capabilities enabled easy adjustments to the design as needed. With the 3D model ready, the next step involved preparing the finite element mesh for static analysis. SolidWorks Simulation and Ansys Simulation offer various

meshing options, such as tetrahedral, hexahedral, and shell elements. The appropriate mesh type was selected based on the geometry's complexity and the desired accuracy level. Boundary conditions were crucial inputs to replicate real-world scenarios accurately. We defined fixed constraints, forces, and moments the structure would encounter during operation. In both simulation tools, boundary conditions were set to simulate the actual loading conditions the component would experience. After the static analysis, we post-processed the FEM results to analyse the structural response. SolidWorks and Ansys Simulation provided comprehensive visualisation tools to display stress distribution, deformation, and other relevant parameters. We carefully examined stress concentration areas, deformation patterns, and potential failure points. The post-processing phase allowed us to gain valuable insights into the component's behaviour and behaviour under different loading scenarios. It facilitated the identification of design improvements and areas for optimisation. The methodology involved the following steps:

I was comparing the results obtained for total deformation and stress developed to draw insights into the differences in performance between the models and software packages used, as shown below in Figure 04.

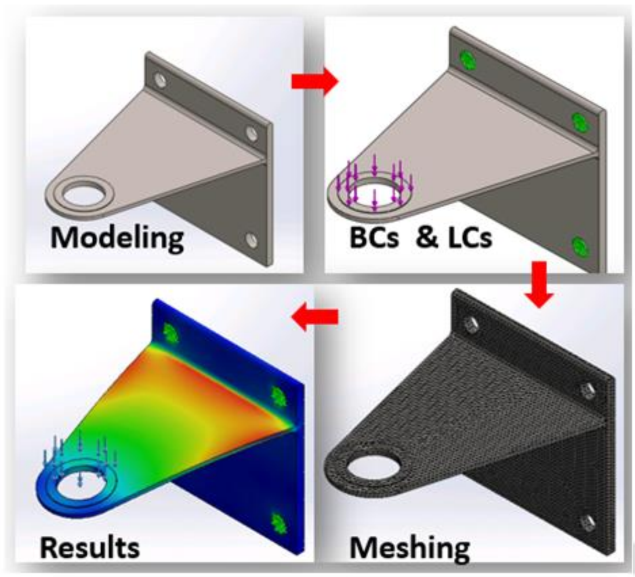


Figure 4: Methodology of FEM study on L Bracket

4.3 Deformation Models using Solid Work Simulation and Ansys Static Simulation

Static analysis is a common type of FEM simulation used to study the behaviour of structures under static loading conditions. The boundary conditions and mesh types used in the analysis can significantly affect the results' accuracy. In Solid Works Simulation, boundary conditions are defined by applying loads and constraints to the model, such as forces, moments, fixed supports, and frictional contacts. The mesh types available in Solid Works include tetrahedral, hexahedral, and mixed elements. The choice of mesh type can affect the accuracy and computational efficiency of the simulation. In Ansys Static Simulation, boundary conditions are defined using the same loads and constraints as in Solid Works. Ansys Static Simulation offers many mesh types, including tetrahedral, hexahedral, pyramid, and wedge elements. The choice of mesh type can depend on factors such as the model's geometry, the expected deformation and stress patterns, and the computational resources available. Overall, the choice of boundary conditions and mesh type can significantly affect the accuracy and reliability of the results obtained in static analysis simulations using Solid Works Simulation and Ansys Static Simulation.

Boundary conditions are critical in static analysis, defining how a structure interacts with its external environment. These conditions include fixed constraints, forces, and moments that replicate real-world scenarios. In Solid Works Simulation, engineers can specify displacement, revised geometry, or remote load application as boundary conditions. Ansys Static Simulation also offers a wide range of boundary condition options, allowing for comprehensive modelling of complex systems. An equally important aspect is the selection of appropriate mesh types. In both Solid Works Simulation and Ansys Static Simulation, the discretisation of the geometry into finite elements significantly impacts the accuracy and computational efficiency of the analysis. Engineers can choose between various meshing options, such as tetrahedral, hexahedral, or shell elements, based on the geometry and requirements of the simulation.

Careful consideration of mesh refinement in regions of interest ensures accurate results while minimising computational resources shown in figure 05.

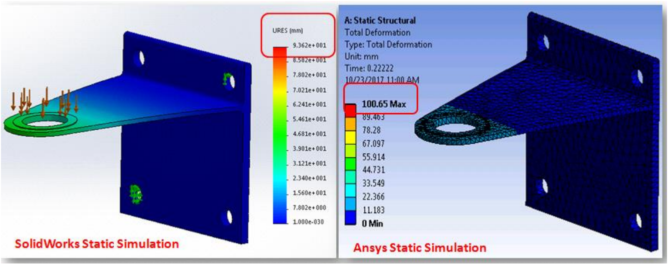


Figure: 05.1. Deformation of Model 01

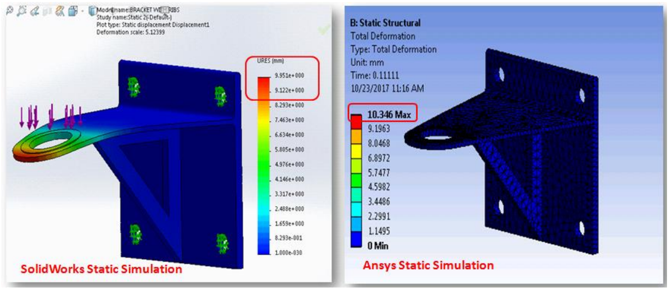


Figure: Figure: 05.2 Deformation of Model 02

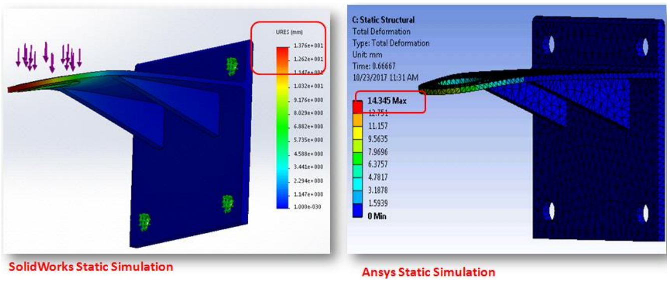


Figure 5: 3 Deformation of Model 03

Table 2: Showing the comparing results of Deformation in Solid Works S & ANSYS Software

Model No. / Parameters	Model 01	Model 02	Model 03
Mass in Solid Works (Kg)	33.483	**38.194**	39.949
Mass in ANSYS (Kg)	33.472	**38.213**	39.897
Deformation in Solid Works (mm)	93.62	**9.951**	**13.76**
Deformation in ANSYS (mm)	100.65	**10.346**	14.345

Tabulated results in Table 2 reflect that Model 02 is given optimum results compared to the other two designs. So, next, Model 02 is optimised for thickness using the Design Study command in Solid Works as follows. As shown in Table 02.

5. Optimization of Rib Design Study in Solid Works Simulation

The optimisation of rib design in Solid Works Simulation represents a significant advancement in structural engineering. This study aims to enhance components' load-carrying capacity and overall performance by strategically incorporating ribs into their design. The methodology involves leveraging the capabilities of Finite Element Method (FEM) optimisation, which allows for a systematic exploration of rib configurations to achieve the most efficient and robust design. In FEM optimisation, the first step is to define the objectives and constraints of the study. Engineers specify the desired performance metrics, such as minimising stress concentration, maximising stiffness, or reducing deformation under specific loads. Additionally, constraints related to manufacturing limitations or material properties are considered. Next, the model geometry is parametrically defined in Solid Works, allowing for easy modification of rib parameters, such as rib thickness, height, and spacing. The FEM meshing technique discretises the model into finite elements, representing the physical behaviour of the structure more accurately. Optimisation within the Solid Works environment, empowering engineers to make informed decisions and drive innovation in structural engineering.

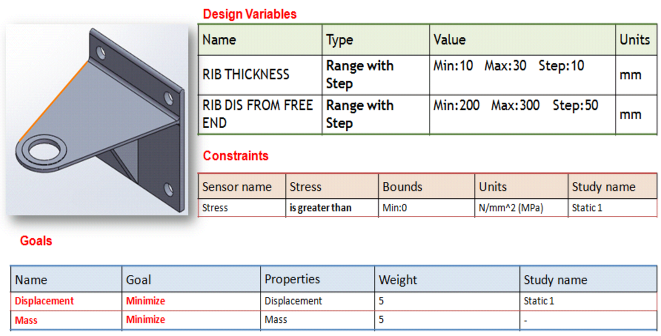


Figure 6: Design Study, defining variables & constraints in Solid Works Simulation Optimization Study

Boundary conditions play a critical role in optimisation studies. They represent the external loads and constraints applied to the model, simulating real-world scenarios. Using loads and rules strategically, engineers can observe how the rib design performs under various operating conditions. The novelty lies in the iterative nature of the FEM optimisation process. Engineers can run multiple simulations with different rib configurations automatically, automatically comparing results and identifying the optimal design that fulfils the defined objectives and constraints. This approach significantly reduces the need for physical prototyping and costly trial-and-error iterations, leading to substantial time and cost savings in the design process. In conclusion, optimising rib design in Solid Works Simulation using FEM optimisation offers a powerful approach to improving the structural performance of components. This methodology allows engineers to systematically explore and refine rib configurations, leading to more robust, efficient, and safer designs. The novelty of this study lies in the integration of FEM. After defining the variables & constraints, the software will calculate automatically shown in figure 06 & tabulate the results as follows. As shown in Table 3.

Table 3: Showing results of optimisation of Rib Design Study in Solid Works Simulation.

Component name	Units	Current	Initial	Optimal	Scenario1	Scenario2
RIB THICKNESS	mm	30	30	10	10	20
RIB DIS FROM FREE END	mm	300	300	200	200	200
Mass	g	5187.7	5187.7	4776.14	4776.14	5203.17
Displacement	mm	31.41632	31.41632	11.28928	11.28928	10.26517

Component name	Units	`	Scenario4	Scenario5	Scenario6	Scenario7
RIB THICKNESS	mm	30	10	20	30	10
RIB DIS FROM FREE END	mm	200	250	250	250	300
Mass	g	5630.2	4702.39	5055.67	5408.95	4628.64
Displacement	mm	9.70723	20.70783	19.36666	18.59922	33.99111

Component name	Units	Scenario8	Scenario9
RIB THICKNESS	mm	20	30
RIB DIS FROM FREE END	mm	300	300
Stress	N/mm^2 (MPa)	2327.6	2082.4
Displacement	mm	32.33605	31.41632
Mass	g	4908.17	5187.7

Conclusion and Scope for future work

The study on L-Bracket rib strength through FEM analysis using SolidWorks design software and ANSYS and SolidWorks FEM simulation software is important in mechanical engineering as it provides insights into the behaviour and performance of L brackets under different loading conditions.

1. Three different models for Ribs (Model 01, Model 02 and Model 03) are modelled based on literature review and requirements using SolidWorks design Software to study the L-Bracket Rib Strength through FEM Analysis
2. The results of the FEM simulations showed that the L bracket models exhibited different degrees of deformation and stress under various conditions. The total deformation and stress developed were compared for both software packages. The results showed that in some cases, the performance of the models was similar for both ANSYS and Solid Works, while in other cases, there were significant differences.
3. The comparative analysis of the results showed that the performance of the L bracket models was affected by the software package used and the conditions applied. The custom models showed higher deformation and stress levels than the standard models.
4. In some cases, The ANSYS software package showed higher deformation and stress levels than Solid Works. The differences in performance could be attributed to factors such as the meshing algorithm, element size, and solver settings used in each software package.
5. The research showed that ANSYS and Solid Works FEM simulation software can model and simulate L brackets under various conditions. The comparative analysis of the results provided insights into the differences in performance between the models and software packages used.
6. The findings can help optimise the design and performance of L brackets in various applications. Further research can be done to explore the effects of other factors, such as material properties and manufacturing processes, on the performance of L brackets. A rib thickness of 10MM yielded optimum optimisation of the Rib Design Study in Solid Works Simulation.

Acknowledgment

The authors wants to thanks “Sri Jayachamarajendra Polytechnic College K R Circle Bengaluru”, for allowing us to work on research facilities and technical guidance. The authors acknowledge the aid of Prof. Amithkumar Gajakosh for his Guidance and suggestion in carrying FEM Analysis.

References

[1]. Singh, A. K., & Singh, M. K. (2014). Design and analysis of a bracket using Solid Works and finite element method. Journal of Mechanical and Civil Engineering, 11(2), 10-14.

[2]. Gupta, A. K., & Singh, A. K. (2014). Finite element analysis of a bracket using Solid Works. International Journal of Engineering Research and Applications, 4(10), 24-29.

[3]. Khan, M. R., & Razzaque, S. A. (2013). Design and finite element analysis of a bracket for aerospace applications. Journal of Aerospace Engineering, 26(4), 605-612.

[4]. Kim, S. H., & Lee, J. W. (2013). Optimisation of rib design for stiffness and strength using finite element analysis. Journal of Mechanical Science and Technology, 27(2), 387-392.

[5]. Reddy, M. V. R., & Babu, P. R. (2013). Finite element analysis of a bracket for machine tool applications. International Journal of Engineering Research and Applications, 3(5), 119-125.

[6]. Rane, R. S., & Patil, S. S. (2012). Finite element analysis of a bracket for automotive applications. International Journal of Mechanical Engineering and Technology, 3(2), 273-277.

[7]. Wan, T. M., & Tong, L. Y. (2012). Design and analysis of a bracket using the finite element method. Journal of Mechanical Engineering and Automation, 2(2), 101-105.

[8]. Sahoo, S. K., & Prasad, M. S. (2011). Design and analysis of a bracket for medical applications using the finite element method. Journal of Medical Engineering & Technology, 35(2), 103-108

[9]. Prasad, M. S., & Sahoo, S. K. (2012). Finite element analysis of a bracket for marine applications. International Journal of Engineering Research and Applications, 2(5), 1316-1321.

[10]. Hossain, M. A., & Rahman, M. M. (2011). Design of a bracket for structural applications using finite element analysis. Journal of Mechanical Engineering and Automation, 1(3), 154-159.

Cite This Article:
Parashuram Bedar, Aprameya C R, "Design, 3D Modelling, and study on L-Bracket Rib Strength through FEM Analysis: A Comparative Study using Solid Works and Ansys Software , International Journal of Scientific Research in Science, Engineering and Technology(IJSRSET), Print ISSN : 2395-1990, Online ISSN : 2394-4099, Volume 2, Issue 1, pp.677-686, January-February-2016. Available at doi: https://doi.org/10.32628/IJSRSET173391