© INNOVATIVE RESEARCH THOUGHTS | Refereed | Peer Reviewed | Indexed
ISSN : 2454 – 308X | Volume : 04 , Issue : 07 | October - December 2018

Study of Design and Development of D-post Gusset

JAFFARSADIK MULLA
LECTURER
DEPT OF AUTOMOBILE ENGG
GOVT POLYTECHNIC BIJAPUR
jmmulla11@gmail.com

SAYEEDA
Lecturer, Mechanical Engineering,
Government Polytechnic, Belagavi.
sayeedaa.sekhaji@gmail.com

VIJAY. M.P
Designation: Lecturer
Department :Electrical and Electronics Engineering
College:Government Polytechnic Chamarajanagar
Email:mpsvijay1@gmail.com

SHIVAKUMAR MD
Designation:Lecturer in science
Government polytechnic, chamarajanagar-571313
Email:shivakumarmd79@gmail.com

Publication : 30 October 2018 **DOI:** https://doi.org/10.36676/irt.v4.i7.1555

Abstract

The current study is needed for the evaluation of customizations, ergonomics, safety, high performance, durability & customer satisfaction made to have focus point of an industry as we serve the customers directly. Gusset or a Gusset plate is one of the components of (BIW) which is on a broader perspective is a sheet metal component that acts as an anchor for various components and sometimes used to connect various components. D-Post Gusset, despite being a minor component, it is crucial in connecting the seat belt mount. This present study we have chosen appropriate material based on the application & model is created accordingly in Catia V6. The required analysis is carried out in Ansys software and results are discussed. Technologies are improving day by day, as technology has improved; it is evident that choice has become a comfort and this comfort has become an absolute driving factor in almost all the industries. Our beloved Automotive Sector isn't immune to this factor as well. Owning an automotive was a privilege then, now it's a fashion, necessity & whatnot. With the above thoughts one can infer & safely agree that passenger safety has taken up a driving seat in the design of an automotive. This is where Gusset comes into picture as it anchors and connects many internal components in a car. Parts ranging from chassis to seat belt mounts & wheel arch are connected. A simple design modification can bring about noteworthy results in customer safety, overall performance, ease of manufacturing.

Key words: Design, Development, D-post, Gusset etc.

Introduction

Modern vehicles rely on mechanical systems that are essential to the vehicle's mobility and safety features. Different safety systems, like the Antilock Braking System (ABS) and passenger restraint systems, have been designed to guarantee the safety of the passenger in the case of an accident, whether it be head-on or of another kind. On the other hand, manufacturers also want their consumers to enjoy the driving experience, so they work to enhance the car's handling and flexibility. Comfort for passengers is ensured by electronic technologies including Cruise Control and active suspension systems. The numerous parts of a vehicle must be produced using cutting-edge techniques, tested, and inspected with extreme care in order to ensure optimal and safe driving. This way, any defective parts may be stopped from being distributed from the start of the supply chain. As a result, there is a lot of research and development being done on procedures that can extend the lifespan of the individual components. These techniques may be employed considerably more adaptably for various components, made up of various materials, and under various input circumstances with a strong research foundation. This will help the process become more profitable and enhance its value to the sector.

Requirements of BIW structure:

1. Comfortable space
2. Strength & stiffness
3. Low weight material
4. Corrosion resistant
5. Aerodynamic & good appealing structure: BIW joinery defines the exterior and interior look
6. Sufficient space for packaging and provisions made for inner trim and aggregates
7. Pedestrian and occupant protection during rollover accident
8. Structure should support.

Finite-life design

ILD has historically been the foundation for the design of automotive components, but the need for mass optimization has expanded the use of the finite-life design (FLD) criterion (e.g Ponomarev, &Tsybanev 2001). A component must safely endure the design-life, taking into account an acceptable design safety factor, according to FLD requirements. Although FLD needs substantially more design work to execute safely than ILD for complicated and unpredictable loading circumstances, such as those found in suspension components, it has the potential to save mass by:

lowering the modelling uncertainties that arise when the design-life is fewer than the number of cycles associated with the fatigue limit or endurance limit of the material under consideration, allowing safer use of materials with poorly defined fatigue limits, such as steel and aluminium.

Current study: The current study is needed for the evaluation of customizations, ergonomics, safety, high performance, durability & customer satisfaction made to have focus point of an industry as we serve the customers directly.

Gusset or a Gusset plate is one of the components of (BIW) which is on a broader perspective is a sheet metal component that acts as an anchor for various components and sometimes used to connect various components. D-Post Gusset, despite being a minor component, is crucial in connecting the seat belt mount., xyz, abc, etc.

This present study grants the interface for consumer a brief oversight into the process of design. As we examine the crucial elements that influence design, the space requirements and manufacturing limitations that frequently change the initial design of gussets, and lastly how it might deceive user safety and ergonomics.

Technologies are improving day by day, as technology has improved; it is evident that choice has become a comfort and this comfort has become an absolute driving factor in almost all the industries. Our beloved Automotive Sector isn't immune to this factor as well.

Owning an automotive was a privilege then, now it's a fashion, necessity & whatnot. With the above thoughts one can infer & safely agree that passenger safety has taken up a driving seat in the design of an automotive. This is where Gusset comes into picture as it anchors and connects many internal components in a car. Parts ranging from chassis to seat belt mounts & wheel arch are connected. A simple design modification can bring about noteworthy results in customer safety, overall performance, ease of manufacturing, etc. Let's dig in to Gusset Part a bit. This present study we have chosen appropriate material based on the application & model is created accordingly in Catia. The required analysis is carried out in Ansys software and results are discussed.

Geometry Details of the gusset Structure

Working on D-Post part which will come in UPPER BODY, which located in Rear panel and it is a Connecting member to ROOF and Rear quarter inner and outer panel. The part will mount on chassis, it will be spot welded. The important function of this part is, it should with stand suspension load, seat belt retractor, need to clear the weld gun issue for welding, need to do package study and need to give a good and feasible design and do the weight reduction and optimize the cost. The part should pass the CAE analysis and crash test. Need to check the material and thickness need to keep 17mm weld flat and to do the clash analysis with surroundings parts. Need to interact with other teams like harness team, seat belt team, underbody team. From scratch to till production need to take all responsibility. Getting master section and styling surface by using this and with the help of CATIA V5 software. Need to see the safety of the passengers. Additionally, by allowing for the safer application of materials without clearly defined fatigue limits, like steel and aluminium, as well as by lowering modelling uncertainties that arise when the design-life is less than the number of cycles associated with the material's fatigue limit or endurance limit.

The main aim for this study is to modal a part, conduct analysis & to find the failure load, maximum von-mises stress and deformation induced in the component for static structural case.

Fig. below shows the Geometry Details and location on BIW of the gusset Structure

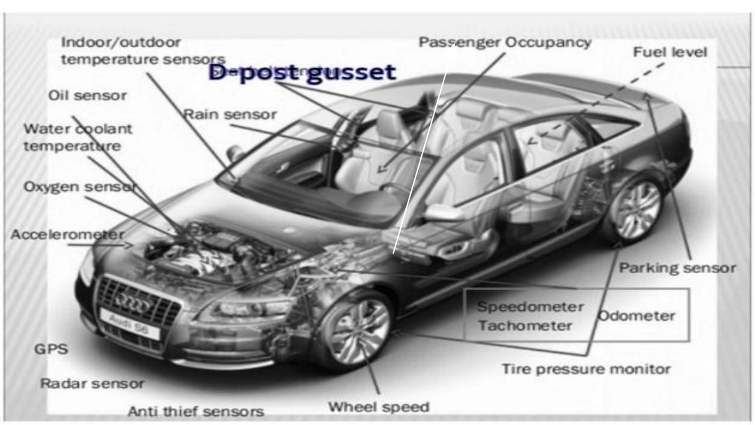


Fig. 1.1 Geometry Details and location on BIW of the gusset Structure

About Structural Design

A lot of prior works have focused on vehicle's structural design & optimization. The following is a summary of some of the previous work on vehicle structural design, analysis, and optimization. Among other things, the goal of structural design is to minimise component weight while satisfying load (stress) and stiffness requirements. Structural optimization refers to the process of creating the best structure with the best structural performance. Using finite element techniques, structural systems such as the chassis or the chassis may be easily examined for stress, stiffness, and so on. The constraints for optimization are the limits on stress, strength, and so on. In vehicle structural design, the finite element approach has been used for a variety of objectives, including design analysis and optimization. The finite element approach was used to analyze the stresses of a car BIW part D-Post Gusset.

Literature Review

Linli Tian and Yunkai Gao : For a B.I.W. exposed to different representative legislative crash loads, such as frontal impact, side impact, roof crush, and rear impact, topology optimization design flow has been proposed. The optimization process has been expanded to multiple load cases design by using the compromise programming method, in which the strain energy has been treated as the stiffness evaluation index of the B.I.W. and the relative displacements have been used as the compliance index of the B.I.W.

Pranav & Shinde et.al.: They examined the method for enhancing the functional qualities of a light weight body in white, explained how parameters affect light weight bodies in white, and compared various methods for resolving issues with light weight body in white structures.

Dinesh & Munjurulimana et.al.: In order to achieve considerable weight savings without sacrificing crash performance, they have concentrated on developing reinforcement solutions for body in white utilizing engineering thermoplastic materials put at the proper locations on a vehicle's body in white. Through CAE studies, various design and material configurations, including structural members made of plastic, metal-plastic, and composite-plastic mounted on the body in white, were assessed for a range of crash scenarios, including high-speed frontal crashes, side impacts, pole impacts, and rollovers.

Elena & Cischino et al.: They presented a project that would reduce the number of parts by utilizing cutting-edge lightweight materials and technologies. They thought that high performance aluminium alloys provided the opportunity to obtain parts with complex geometries and thin walls, fusing various parts into a single, distinctive element.

Ashish Kumar & Sahu et al.: A Sports Utility Vehicle's body in white was the subject of an attempt to maximize the body's bulk reduction (SUV). The optimization process took into account all the fundamental performance criteria necessary to reduce the body's bulk. Instead of mass reduction after vehicle launch, optimization during the early stages of body in white development helped to ensure the least B.I.W. weight

Prof V Kumar, and Atul Adhau : The goal of this paper is to provide readers with more information about the forming process, the materials that can be formed, the stresses that sheet metal experiences during forming, and the flaws that can occur when sheet metal is being formed. By applying compressive forces through a set of rolls, it is possible to reduce the thickness of a long work piece to create sheet metal. In general, sheet metals are sheets less than 6 mm thick, and plates are sheets larger than 6 mm.

Patel et al optimized the weight of the Truck Chassis Frame by carrying the structural analysis using different models of the frame. The frames were optimized using different width and height of the frames. Different chassis has been modelled by the use of Pro-E & structural analysis was done through ANSYS workbench. The parameters considered for comparisons were maximum stress deformation and weight. A reduction in truck chassis weight by 17% was achieved through optimization

M D Vijayakumar et al: They investigated the effects of type of material on truck's Tata 407 staircase using FE analysis. The analysis of chassis was carried using conventional steel and Carbon fiber. The parameters considered for comparison were weight, deformation and maximum stress. The usage of Carbon fibers reduced the weight of chassis by 80 % (From 456.9 kg to 91.379 kg) and the equivalent stress is almost same

Finite Element Analysis:

- **Engineering Simulation**

Engineering simulation is one of the most rapidly expanding fields of engineering in the world. In layman's words, it is the application of engineering software tools to the design process in order to simulate the operational performance of existing or new product or process designs. Physical prototypes for each design candidate must be produced and tested through physical experimentation in the absence of engineering simulation technologies. Small changes to a design may necessitate the creation and testing of an entirely new prototype, delaying development and increasing costs. Furthermore, a test may produce final results indicating that a design is successful but without providing any hint or explanation as to how and why this is the case.

The use of engineering simulation technologies in the design process reduces the requirement for extensive physical prototype and experimentation. Instead, several design candidates may be examined rapidly and efficiently under a wide range of conditions, some of which may be impossible to recreate experimentally. Results can be displayed in data formats or as three-dimensional still images or animations, allowing an engineer to perceive and comprehend how and why a design performs in any given environment. This knowledge enables a research and development team to swiftly optimize goods and processes, shortening design and development periods, lowering costs, and improving product efficacy. The process of performing an engineering simulation, whether multiphysics, structural (FEA), fluid flow (CFD), thermal, or electromagnetics, is divided into three steps:

1) Pre-Processing
2) Solving/processing
3) Post-Processing

Pre-Processing

To begin pre-processing, a computer model of the object or process is needed; typically, this is CAD geometry transferred into engineering simulation software. ANSYS software tools can be used to create a representative geometry from scratch. The CAD files occasionally import with rough, incorrect edges that must be smoothed up or the simulation would be wrong. ANSYS tools can help with this as well. The model's outside boundary is established, which limits simulation to a given area and speeds up the solving process.

Following that, a computational mesh is generated that encompasses every surface of the geometry as well as all volumes of space in and around the geometry up to the model's outer boundary. In other situations, the mesh can be made up of millions of individual cells. The complexity of the mesh is also an important consideration. When more accuracy of results is desired, such as in a boundary layer, smaller cells in a localized mesh must be used. To arrive at an overall simulated prediction for the model, the computer will solve sets of fundamental mathematical equations for each individual cell.

Next, a product or process-operational system's performance is studied by known parameters such as flow velocities, temperature, pressure, stress loads, and other considerations. The model is now ready to be solved, thanks to this step.

Solving/Processing

Engineers develop a model solution using engineering simulation software, which involves a series of computational methods including applied forces & element attributes. The process of resolving the problem begins with the click of a mouse button. Together, the software and hardware perform millions of complex calculations per second in order to assert their dominance.

Scientists have known the physics equations that underlie these simulations for more than a century. Until the 1940s, they were calculated by hand by a team of engineers. It is now possible to solve complex problems with a single computer, and depending on the size and complexity of the model to be simulated, a solution can be found within a few minutes. When it comes to solving a problem, the more cells there are in a simulation, the longer it will take. With more computing power, the solution speed can be increased.

Post-Processing

Simulation results could be analyzed in order to fully understand the the study's implications. Using graphical tools & numerical tools, the engineer can pinpoint to the exact location of the data, usually as stresses &deflections, on a structure or component.

In particular, engineers and analysts can benefit from tabular data because it provides precise numbers that can be used to meet design and manufacturing as well as regulatory and other needs. In order to communicate the simulation results to all stakeholders in the design process (engineers and non-engineers), visualization tools are available to help. There are a variety of still and animated graphs that can be used to identify problems and determine where further research should be focused.

The Finite Element Method (FEM) is a computational method used in engineering to tackle problems related to stress analysis, electromagnetic, heat transfer, dynamics, fluid flow, & a variety of other disciplines. Because it's a numerical method that can be used together Using a standard algorithm, it's most powerful unifying way of approach to solving the widest range of problems related to engineering. In the fig.3.1, below, the discretized dominion is depicted.

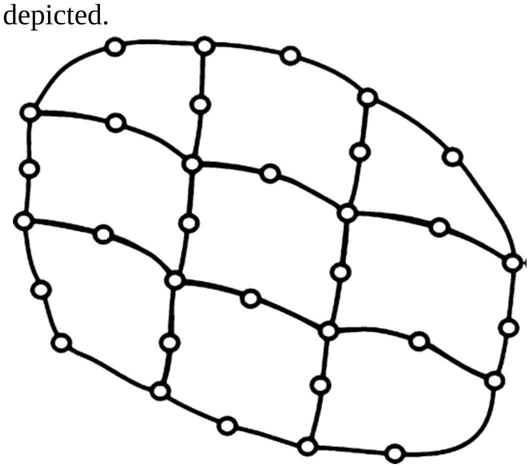


Fig: Discretized domain

The following complicated problems which may arise when attempting to derive the field governing equation characterizing the precise behavior of an engineering quantity in a domain:

© INNOVATIVE RESEARCH THOUGHTS | Refereed | Peer Reviewed | Indexed
ISSN : 2454 – 308X | Volume : 04 , Issue : 07 | October - December 2018

1) Geometry and boundary condition
2) Initial condition

To deal with this, FEA approximates the solution by partitioning the domain into small finite elements of simple shape. As a function of nodal values, the behavior of each element's field primary variable is determined. The governing differential equations for each element are integrated. The pieces are then put together to re-form the entire structure by equating the values of common nodes.

Basic steps in the Finite element method (FEM)

Pre-processing phase

Create & discretize the solution of domain into finite elements.

Breaking the problem down into nodes & elements accomplishes this. As demonstrated in Figure (b), FEM separates a complex domain into a number of simple finite elements and treats each element separately.

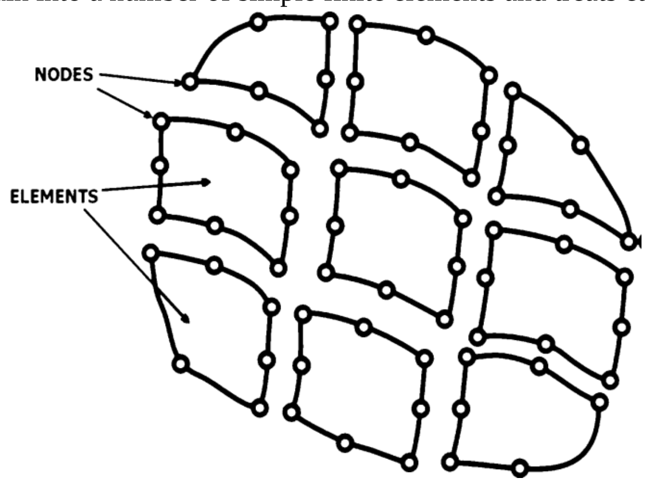


Fig: Domain broken up into finite elements

(i) Select interpolation or displacement model

A displacement model or function must be selected for each element. Within the element, the function is defined by using the nodal values of the element as inputs.

(ii) Assume a Shape Function to characterize the physical behavior of an element

An element's solution is believed to be a continuous function that is approximate. The field variable of each element is described by a shape functions corresponding to the nodes along the element's boundary.

(iii) Develop matrix equation for an element Each element generates a stiffness matrix, K, at each node of an elasto-static problem, which relates forces (F)& displacements (u). The stiffness matrix has the following size when the number of nodes per element is multiplied by the number of freedoms per node: $[F] = [K]\{u\}$

(iv) Assemble the elements to form a complete picture of the situation.

At this stage it's time to build the global stiffness matrix. The displacements at common nodes between neighboring elements are equated to generate a global matrix. Number of nodes multiplied by the number of nodes' freedoms gives the global matrix's dimension.

(v) Apply boundary conditions, initial conditions and loading.

The nodes on the structural boundary are then subjected to boundary conditions.

Solution Phase

To solve for all unknowns, use the global matrix in the solver. To attain nodal results, such as values of displacement at various nodes in a heat transfer problem, the solver simultaneously solves a set of algebraic equations, either linear or nonlinear. During an Eigen value problem, the Eigen values and vectors are evaluated to determine the solution.

Post-Processing Phase

This phase provides the output results. Interpolation of nodal values and charting can be used to generate displacement or temperature contours.

There is a wide range of advanced computer-aided design [CAD] tools available today which may be used in to the production of 3D Solid modelling of structure, later on automatically mesh the model &export it to FEA software for further handling. This is having a significantly decreased in the time required for the pre-processing. Likewise, post-processing has become fairly sophisticated, featuring 3D views of various contours such as stress & temperature contours, as well as mode form and kinematics animation. The majority of the major software companies have created FEA modules that can be operated on Windows-based systems. Many people may now afford to set up their own FEA lab. FEA software has been around for decades, but the capability of personal computers has allowed even the most basic models to be created.

ANSYS

ANSYS, Inc. is a significant producer of engineering simulation software in the globe. Its technology enables consumers to accurately forecast if their product designs will succeed in the real world. This is the company's primary goal which is to provide an unified platform of wholly integrated multiphysics software tools for optimizing a product that has a developmental processes in a variety of industries ,including automotive, aerospace, civil engineering, electronics, power, consumer products, chemical process, healthcare, environmental, marine, sports, & others .Design idea, validation and troubleshooting of current designs can be considerably accelerated by using ANSYS software, which reduces design and development times, saves money and provides valuable insights into the performance of products or processes.

ANSYS software not only increases productivity, but it also promotes innovation. The ability of technology to go beyond physical restrictions and execute simulated tests that would otherwise be impossible is crucial to investigating and expanding operational boundaries in the development of cutting-edge products and processes. Modeling and simulation can thus be used to drive new solutions rather than simply verifying old ones. This is referred to as Simulation-Driven Product Development by ANSYS. The most extensive versions will be run on them.

Linear Static Stress Analysis

This is the most often used form of analysis. When loads are been applied to a body, the body gets deformed& the stresses' properties are transferred throughout the body. To make balance between the applied external loads, the body will creates internal forces & responses at the supports to absorb the impacts of load. Linear static analysis is the computation of displacements, strains, and stresses in the presence of external loads under certain assumptions. The relationship between loads and reactions is linear. For example, if we double the amount of the loads, the model's response (displacements, strains, and stresses) will also double. The model's materials all satisfy Hooke's Law, which asserts that stress is proportional to strain perfectly. The generated displacements are modest enough that the change in stiffness caused by loading may be ignored. Load application has no effect on boundary conditions. The magnitude, direction, and distribution of loads must remain constant. They should not be altered while the model is deformed.

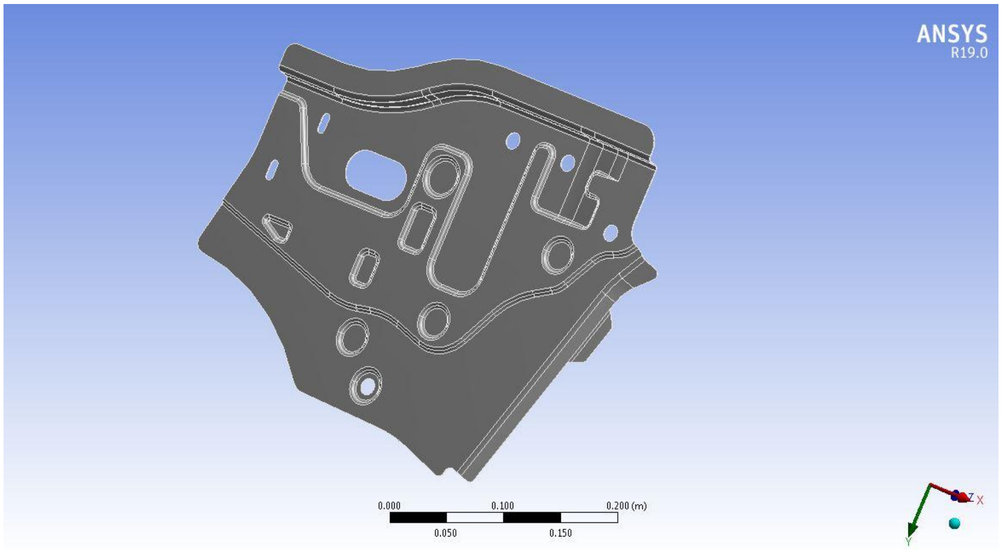


Fig: Gusset CAD Model imported in Design Modeler of Ansys Workbench

In the next stage, Model is edited in the Ansys Mechanical module to carry out Meshing and update analysis settings for the static structural case. Discretization is done using the medium size mesh under sizing options. Typical tetrahedral elements are used for meshing.

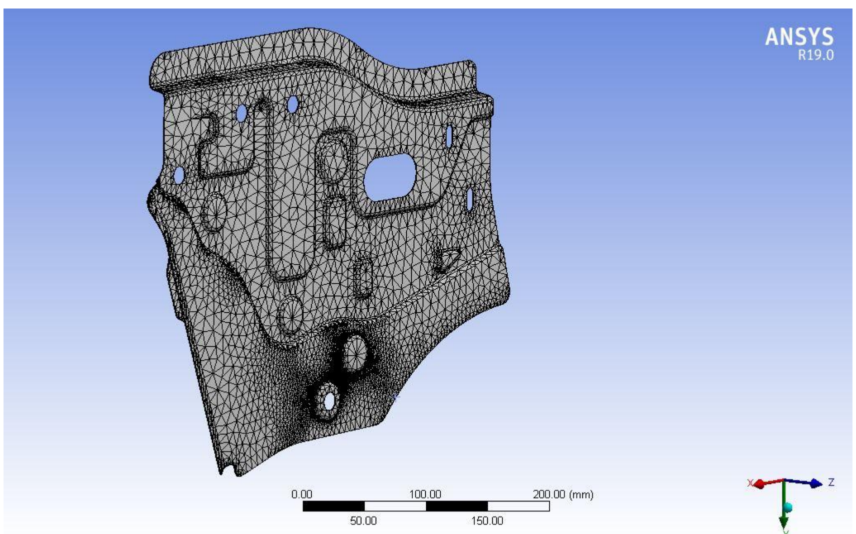
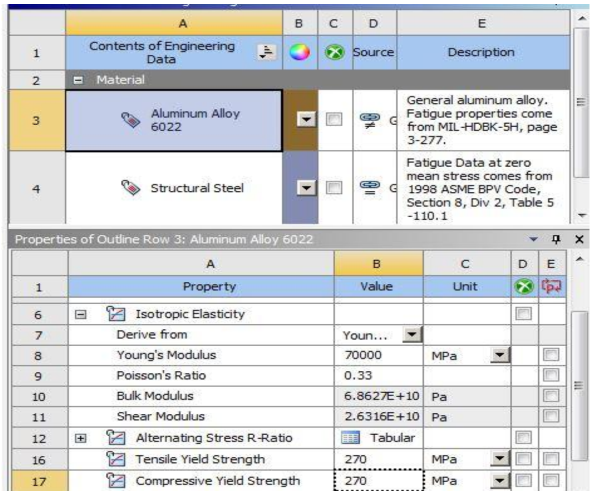


Fig: Gusset meshed model

Aluminum 6022 alloy is used as the material for conducting analysis for gusset component. Mechanical properties of the material are as mentioned below:

© INNOVATIVE RESEARCH THOUGHTS | Refereed | Peer Reviewed | Indexed
ISSN : 2454 – 308X | Volume : 04 , Issue : 07 | October - December 2018



	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	Aluminum Alloy 6022				General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
4	Structural Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1

Properties of Outline Row 3: Aluminum Alloy 6022

	A	B	C	D	E
1	Property	Value	Unit		
6	Isotropic Elasticity				
7	Derive from	Youn...			
8	Young's Modulus	70000	MPa		
9	Poisson's Ratio	0.33			
10	Bulk Modulus	6.8627E+10	Pa		
11	Shear Modulus	2.6316E+10	Pa		
12	Alternating Stress R-Ratio	Tabular			
16	Tensile Yield Strength	270	MPa		
17	Compressive Yield Strength	270	MPa		

Fig Aluminum 6022 Alloy - Material properties

Once the mesh is generated, deck preparation can be started. In this process all the necessary boundary conditions and loads on the defined locations. Tensile load is applied on all the clamping locations. Iteration wise the load is applied 50N, 100N, 150N, 200N, 250N, 275N, 300N to identify the failure load. The load at which vonmises stress induced is equivalent or greater than the tensile yield strength of Material is considered as the failure load or maximum load the gusset can withstand without plastic deformation. Fixed support and load applied on the component is illustrated in below image:

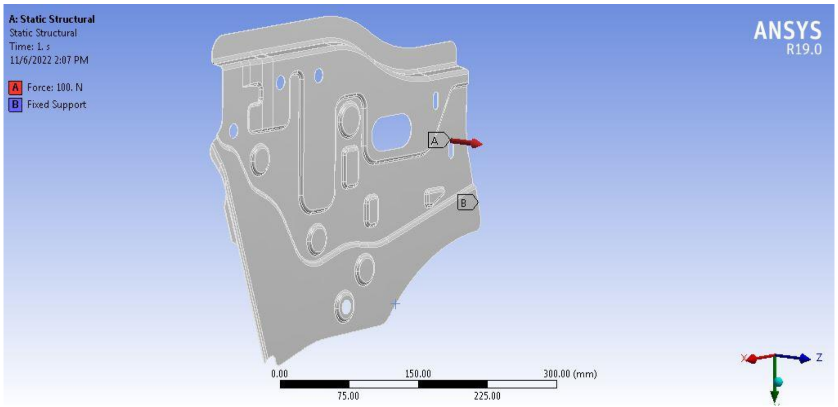


Fig- Fixed support applied on the outer boundary and application of 100N force.

Below Flow chart illustrates the process followed in conducting analysis:

Conclusion

The study on the Design and Development of D-Post Gusset has provided significant insights into its structural performance, material selection, and manufacturing feasibility. The research involved an in-depth analysis of the mechanical properties, stress distribution, and optimization of the gusset design to enhance load-bearing capacity and durability. Key findings indicate that the optimized D-post gusset design improves structural integrity by reducing stress concentrations and ensuring uniform load distribution. The selection of high-strength materials and advanced fabrication techniques further enhances the component's performance, making it suitable for various engineering applications. Finite element analysis (FEA) results confirmed that the redesigned gusset meets safety standards while minimizing material usage, leading to cost efficiency without compromising strength. Prototyping and experimental validation further demonstrated the practicality of the design, confirming its robustness under real-world loading conditions. Future work may explore additional modifications for weight reduction, improved manufacturability, and integration with different structural frameworks. Further experimental testing in diverse environmental conditions will enhance the applicability of the D-post gusset for industrial and automotive uses. In conclusion, the study successfully achieves its objective of developing an optimized D-post gusset, offering a balance of strength, efficiency, and cost-effectiveness. These findings contribute to the advancement of structural component design, paving the way for innovative applications in engineering and construction industries.

References:

1. *Sai AdityaPradeep[1]Rakesh K.Iyer[2]HakanKazan[3]SrikanthPilla[4]* : BIW Automotive Applications of Plastics: Past, Present, and Future.*" Applied Plastics Engineering Handbook (Second Edition)"* Processing, Materials and Applications, Plastics Design Library 2017, Pages 651-673
2. *Tian, Linli and Gao, Yunkai*, "Crashworthiness Design of Automotive Body in White using Topology Optimization" SAE Technical Paper 2016-01-1535, 2016.
3. *Shinde, Pranav, Ravi, K., Nehru, Nandhini, Pawar, Sushant, Balakrishnan, Balaji and Vinit Nair*, "Light Weight BIW Solutions for Improving Functional Properties: A Review" SAE Technical Paper 2016-018138, 2016.
4. *Munjurulimana, Dinesh, Kulkarni, Amit, Nagwanshi, Dhanendra, Thambi, Joel Luther, Winters, Ruud and Delaney, Matthew*, "Body In White Reinforcements for Light-Weight Automobiles" SAE Technical Paper 2016-01-0399, 2016.
5. *Cischino, Elena, Paolo, Francesca Di, Mangino, Enrico, Pullini, Daniele, Elizetxea, Cristina, Maestro, César, Alcalde, Estibaliz, Christiansen, Jesper deClaville,* "An Advanced Technological Lightweighted Solution for a Body in White" Transportation Research Procedia 14 2016-05-082, Elsevier, pp. 1021 – 1030, 2016
6. *Sahu, Ashish Kumar, Londhe, Abhijit, Kangde, Suhas and Shitole, Vishal*, "Body in white mass reduction through Optimization" SAE Technical Paper 2015-01-1352, 2015.
7. Prof V Kumar, and Atul Adhau," Latest Forming Techniques For BIW". International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 5, May – 2013.
8. Design Of Body In White. IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 7 Issue 3, March 2020 ISSN (Online) 2348 – 7968 | Impact Factor (2019) – 6.248.