Structural and Modal Analysis of Fuel Tank Bracket for Heavy Duty Trucks

Submitted in partial fulfilment of the requirements

of the degree of

Bachelor of Technology

by

Rushikesh Dilip Pabalkar
MIS: 111910085

under the guidance of Dr. S. S. Ohol.
Company guide - Mr. Prasanna Samaal.

Department of Mechanical Engineering

COLLEGE OF ENGINEERING, PUNE-411005

(2022-23)

CERTIFICATE

This is to certify that the thesis entitled 'Structural and Modal Analysis of Fuel Tank Bracket for Heavy Duty Trucks' submitted by Rushikesh Dilip Pabalkar (MIS 111910085), in the partial fulfillment of the requirement for the award of degree of Bachelor of Technology (Mechanical Engineering), affiliated to Savitribai Phule Pune University, is a record of their own work.

Dr. S. S. Ohol	**Dr. N. K. Chougule**
Project Guide	**Head of Department**
Department of Mechanical Engineering,	**Department of Mechanical Engineering,**
College of Engineering Pune,	**College of Engineering Pune,**
Shivajinagar, Pune - 05.	**Shivajinagar, Pune - 05.**

Date:
Place:

This report entitled

Structural and Modal Analysis of Fuel Tank Bracket for Heavy Duty Trucks

By

Rushikesh Dilip Pabalkar

MIS: 111910085

is approved for the degree of

Bachelor of Technology

of

Department of Mechanical Engineering

College Of Engineering, Pune

(An Autonomous institute of Govt. of Maharashtra)

Examiners	**Name**	**Signature**
1. External Examiner	----------------------------------	
2. Internal Examiner	----------------------------------	

Date:

Place:

DECLARATION

I declare that this written submission represents my ideas in my own words and where others' ideas or words have been included, I have adequately cited and referenced the original sources. I declare that I have properly and accurately acknowledged all sources used in the production of this report. I also declare that I have adhered to all principles of academic honesty and integrity and have not misrepresented or fabricated or falsified any idea/data/fact/source in my submission. I understand that any violation of the above will be a cause for disciplinary action by the Institute and can also evoke penal action from the sources which have thus not been properly cited or from whom proper permission has not been taken when needed.

Rushikesh Dilip Pabalkar
111910085

Date:
Place:

ACKNOWLEDGEMENT

I am extremely grateful to have had the opportunity to pursue my project at KPIT Technologies. I express my deepest gratitude towards Mr. Prasanna Kumar Samal, KPIT Technologies for being a constant source of inspiration and also allowing me to carry out my project at the esteemed organization. I express my sincere gratitude towards Mr. Sant Ranjan, KPIT Technologies for giving the necessary advice and guidance and keeping me on the right path.

My sincere gratitude and appreciation go to Dr S. S. Ohol sir for guiding me for the major project throughout the entire semester.

I express my sincere gratitude towards Dr. N. K. Chougule, HoD, Mechanical Department for his guidance.

I would also like to give my thanks and appreciation to my family, friends and colleagues who have been a part of this project directly or indirectly.

Abstract

Industries these days are able to produce a variety of products in a very lesser amount of time and their products and constructions are more reliable. This is possible with the help of computer-aided tools such as CAD, CAM, and CAE. One sub-branch of CAE is Finite Element Analysis (FEA). In this project, a Structural and Modal Analysis of a newly designed Fuel Tank Bracket is taken place, which is used for keeping the fuel tank of a heavy-duty truck in place when the vehicle is experienced with several forces due to acceleration in X, Y, Z directions. Generally, Fuel tank brackets have a very complex structure which makes the analysis of the structure challenging. Using structural analysis stresses and displacements occurring at various sections of the fuel tank bracket were studied. Modal analysis was done to find out the natural frequencies and the mode shapes of the system. These analyses are critically important, as inaccurate results will lead to a failed product and cause severe damage. Standard Industrial practices have been followed during this study to ensure proper results. The design of the fuel tank bracket is analyzed with three different materials Cast Iron, Aluminum alloy and Steel to find out the best-suited material for manufacturing of the bracket. After obtaining the results from the analysis necessary changes in the design were made and the best-suited material was concluded.

Keywords: CAD, CAM, CAE, FEA, Structural analysis, Modal analysis, Fuel tank, Bracket.

Contents

1 Introduction **2**

1.1 Concept of FEA 2
1.2 Advantages of FEA 3
1.3 Steps involved in FEA 4
1.4 Modal Analysis 6
1.4.1 The Eigenvalue problem 7
1.5 Need of Analysis 8
1.6 Background of the project 8

2 Literature review **10**

3 Problem Definition **12**

3.1 Problem Statement 12
3.2 Aim 12
3.3 Objectives 12

4 Proposed Methodology **13**

4.1 Steps carried out during Analysis of a system 13
4.2 Concerned Component 15
4.3 CAD Model 16
4.4 FE Model 17
4.5 Preprocessing 18
4.6 Meshing 18
4.6.1 Types of elements used in meshing 18
4.6.2 Quality of elements 19
4.6.3 Process of Meshing 21
4.6.4 Assigning materials and their specifications 25
4.6.5 Assigning properties 26
4.6.6 Connections 27
4.6.7 Loads and boundary conditions 28

4.6.8 Assumptions 28

5 Results and Discussions **29**

5.1 Material as Cast Iron 29

5.1.1 5G Loading in X Direction 29

5.1.2 5G Loading in Y Direction 29

5.1.3 5G Loading in Z Direction 30

5.1.4 Mode shapes 30

5.2 Material as Aluminum alloy 33

5.2.1 5G Loading in X Direction 33

5.2.2 5G Loading in Y Direction 33

5.2.3 5G Loading in Z Direction 34

5.2.4 Mode shapes 34

5.3 Material as Steel 37

5.3.1 5G Loading in X Direction 37

5.3.2 5G Loading in Y Direction 37

5.3.3 5G Loading in Z Direction 38

5.3.4 Mode shapes 38

5.4 Design changes 41

5.4.1 5G Loading in X Direction 43

5.4.2 5G Loading in Y Direction 43

5.4.3 5G Loading in Z Direction 44

5.4.4 Mode Shapes 44

6 Future Scope and Conclusion **48**

6.1 Limitations 48

6.2 Future Scope 48

6.3 Conclusion 49

List of Figures

1.1 Continuous problem converted to discrete 2
1.2 Steps in Finite Element Analysis 5
1.3 Steps in Finite Element Analysis 6
1.4 Mode Shapes 6

4.1 Methodology 14
4.2 Bracket(Concerned Component) 15
4.3 CAD Model of Truck Chassis 16
4.4 FEA Model of Fuel Tank Assembly 17
4.5 Importance of Meshing 18
4.6 Warp Angle 19
4.7 Aspect Ratio 20
4.8 Jacobian 20
4.9 Tet collapse 21
4.10 CAD Model of Fuel Tank Bracket 22
4.11 3D Tetra-meshed Model of Fuel Tank Bracket 23
4.12 CAD Model of a Cross Member 23
4.13 2D Mixed Model of Cross Member 24
4.14 Rigids and Bar elements 27
4.15 SPC applied at nodes 28

5.1 Displacement and stress plot in X direction 29
5.2 Displacement and stress plot in Y direction 30
5.3 Displacement and stress plot in Z direction 30
5.4 Modal frequencies of mode 1 and 2 31
5.5 Modal frequencies of mode 3 and 4 31
5.6 Modal frequencies of mode 5 and 6 32
5.7 Modal frequencies of mode 7 and 8 32
5.8 Modal frequencies of mode 9 and 10 33
5.9 Displacement and stress plot in X direction 33

5.10 Displacement and stress plot in Y direction 34
5.11 Displacement and stress plot in Z direction 34
5.12 Modal frequencies of mode 1 and 2 35
5.13 Modal frequencies of mode 3 and 4 35
5.14 Modal frequencies of mode 5 and 6 36
5.15 Modal frequencies of mode 7 and 8 36
5.16 Modal frequencies of mode 9 and 10 37
5.17 Displacement and stress plot in X direction 37
5.18 Displacement and stress plot in Y direction 38
5.19 Displacement and stress plot in Z direction 38
5.20 Modal frequencies of mode 1 and 2 39
5.21 Modal frequencies of mode 3 and 4 39
5.22 Modal frequencies of mode 5 and 6 40
5.23 Modal frequencies of mode 7 and 8 40
5.24 Modal frequencies of mode 9 and 10 41
5.25 Addition of another set of Bracket and strap 42
5.26 Adding ribs to Cross members at possible locations 42
5.27 Adding ribs to Cross members at possible locations 43
5.28 Displacement and stress plot in X direction 43
5.29 Displacement and stress plot in Y direction 44
5.30 Displacement and stress plot in Z direction 44
5.31 Modal frequencies of mode 1 and 2 45
5.32 Modal frequencies of mode 3 and 4 45
5.33 Modal frequencies of mode 5 and 6 46
5.34 Modal frequencies of mode 7 and 8 46
5.35 Modal frequencies of mode 9 and 10 47

List of Tables

4.1 Material Properties 15
4.2 List of Components 17
4.3 Type of Mesh and Element Size 21
4.4 Material Specifications 25
4.5 Properties Assigned 26

Chapter 1

Introduction

1.1 Concept of FEA

There are typically three methods to solve any problem Analytical, Numerical and Experimental. Analytical is classical approach and yields the accurate results. It is limited to standardized objects and can't handle real life complex objects well. It is time consuming and there are chances of involving human error. Experimental methods on the other hand are actual testing of object on the exactly required scenario, so they are mostly costly and time consuming. Numerical approach is well suited to handle the limitations of other two methods. In this approach approximate assumptions are made. Though it doesn't provide the 100% accurate solution, the results obtained by it are closest to actual values.

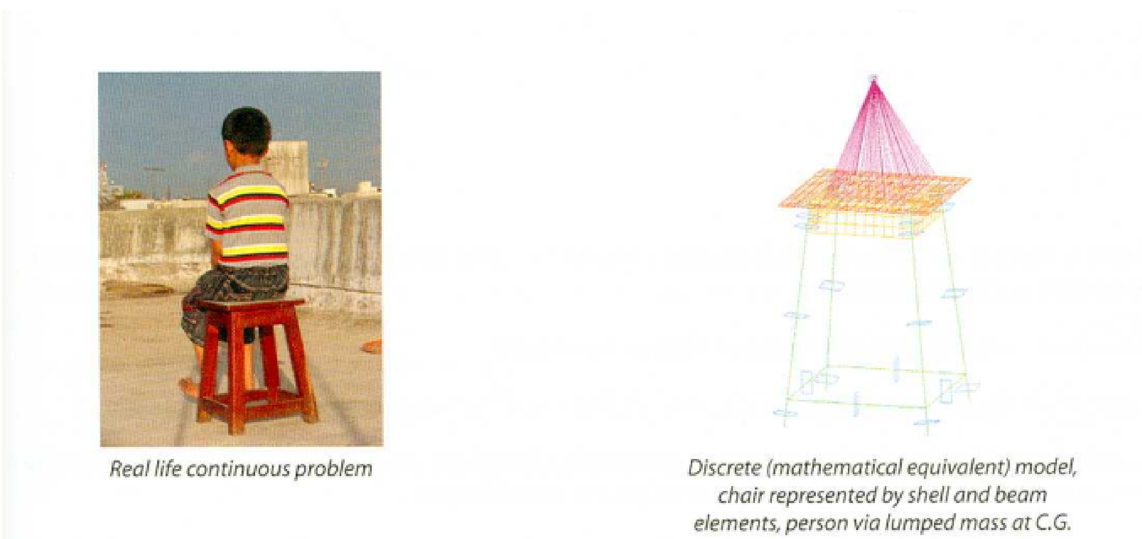

Figure 1.1: Continuous problem converted to discrete

Finite Element Analysis as known as FEA is one of the methods used in numerical type of analysis. It's first step is the discretization of an object into finite number of elements. Now question arises why to do so? Any real-life object is continuous in nature. All particles say items or molecules are connected to each other with strong attraction forces. But, in numerical approach we calculate the movement at each particle with simple mathematical equations. Now imagine infinite number of particles having some degrees of freedom leading to infinite equations with infinite unknowns. Is it possible to solve? That's why we consider continuous object as a discrete and perform calculations on limited number of points. Here degrees of freedom can be defined as minimum number of parameters required to define the position of any entity completely in the three-dimensional space.

1.2 Advantages of FEA

FEA has following advantages because of which it has gained the popularity in the industry.

- Visualization of results: It is easy to visualize point of maximum stress and displacement for simple geometries like cantilever beam or simply supported beam subjected to point load. But real-life objects or assemblies are made up of complex parts, contained with discontinuities loaded with complex load. In addition to this joint such as spot weld as well as residual stresses are present in them. These things make the life of engineers difficult. But, if modelled with appropriate fashion using FEA tools, one can obtain stress contour plots displaying regions of high stress and displacement with minimum efforts.

- Reduction in design cycle time: Validation of the design is one of the most import task in the design cycle. In earlier days it used to be a very time consuming and task. It either needed the expertise of reviewing team or needed to test it actually to get a glance of design's performance. Later on, changes were suggested in this way number of iterations needed reliable design at cost of time and money. But with FEA tools failure prediction is observed in minimum possible time. This leads the minimization of the iteration time and we may get more time to perform a greater number of iterations to deliver the best possible design.

- Reduction in number of protypes and testing: As this type of analysis provides us the response of our assembly to real life conditions, we get a better idea about the performance of the assembly. Because of this in most of the cases prototyping and testing is required only in some of last design iterations. This significantly saves time, labour and most importantly money. That's what any organization wants to maintain its competitive position in the industry.

- Optimum Design: Any designer always walks on an edge of a knife to do trade-off between various attributes of design that require comprehensive knowledge of all the simultaneous fields. Which is critical and tedious job to do. FEA is cutting edge technology equipped with modern algorithms to handle these kinds of problems easily.

1.3 Steps involved in FEA

1. Preprocessing:
   (a) CAD data: A CAD model is prepared of the component or assembly in specialized CAD software. This CAD model is then imported in the software used for FEA in .step format.
   (b) Meshing: Individual components are meshed and the type of elements are decided based on the geometry of the component.
   (c) Boundary conditions: The boundary conditions are applied at required locations in the model depending on the requirements of the analysis.
2. Processing: In this step the software carries out matrix formations, inversions, multiplication and solution for unknowns such as displacement and stress, strain for static analysis.

   The governing equation of FEA is given as below:

   $$[F] = [K] \, . \, [u] \tag{1.1}$$

   Where [F] is the external applied forces matrix, [K] is the Global stiffness matrix, [u] is the displacement response to be determined.

   Inversion of the stiffness matrix can lead to finding the resulting displacement matric due to the applied force on the system.

   $$[u] = [K]^{-1} \, . \, [F] \tag{1.2}$$

Stiffness is defined as force required to produce unit displacement. It depends on geometry and material properties.

$$K = \frac{F(Force)}{D(Displacement)} \tag{1.3}$$

$$K = \frac{AE}{L} \tag{1.4}$$

Where, A is Area of the element, E is Young's Modulus of the material and L is Length of the element.

There are 3 methods for deriving a stiffness matrix.

(a) Direct Method

(b) Variational Method - Rayleigh - Ritz Method

(c) Weighted Residual Method - Galerkin Method.

3. Post processing: This step involves viewing results, verification, conclusions and discussing about what steps could be taken to improve the design.

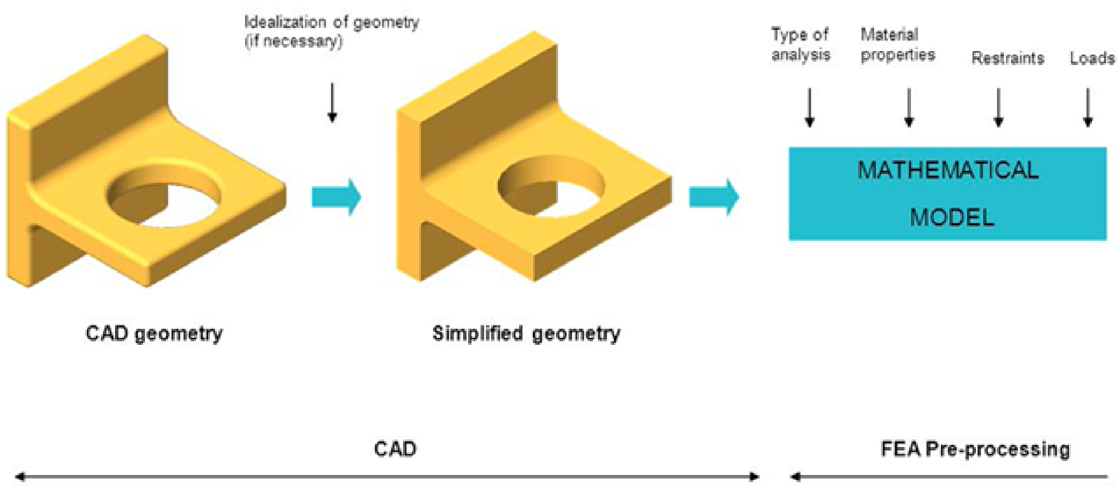

Figure 1.2: Steps in Finite Element Analysis

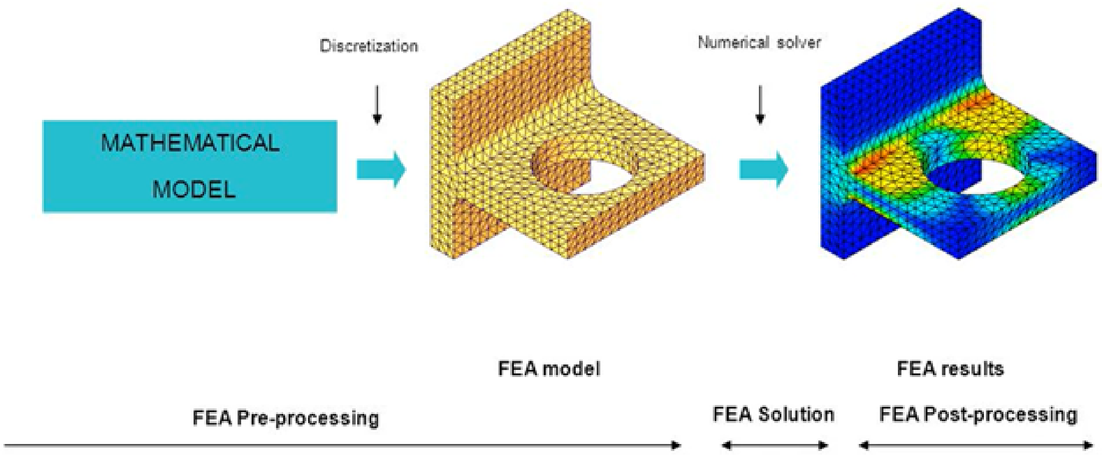

Figure 1.3: Steps in Finite Element Analysis

1.4 Modal Analysis

Modal analysis is method used to evaluate the dynamic properties of system in frequency domain. The goal of modal analysis in structural mechanics is to determine the natural mode shapes and their corresponding natural frequencies of vibrations. Natural frequency is frequency with which object oscillates on its own without any external disturbance when disturbed in its free state. Mode shape is simply the how the object oscillates in 3-D space. It refers to the deformation patterns at certain natural frequencies. Below figure gives the idea about mode shape.

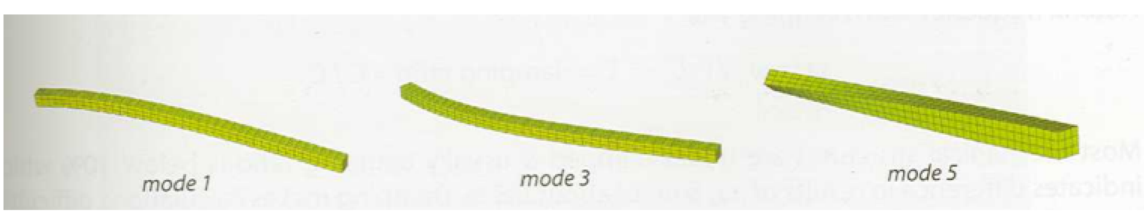

Figure 1.4: Mode Shapes

It is important type of analysis which lays a foundation for further studies of system like Response spectrum analysis, Random vibration analysis and Harmonic analysis. It gives basic idea about excitation frequencies which may resonate with system and cause the severe damage to system. Its governing equation is as given below:

$$[M]\{\ddot{u}\} + [K]\{u\} = [0] \tag{1.5}$$

In above equation M is mass, u is acceleration, K is stiffness and u is displacement. The above equation is an expression of equilibrium in the time domain. With modal analysis however, we are interested in the behavior of systems in the frequency domain. In order to do this, we assume that every point in the system is undergoing simple harmonic motion.

Simple harmonic motion is an oscillatory motion with a restoring force whose magnitude is directly proportional to the displacement from the position. The most common example of SHM is an ideal mass-spring system. SHM can be described by the following equation.

$$x = Asin(\omega t + \theta) \tag{1.6}$$

Where, x is displacement from the equilibrium position, A is amplitude. ω is angular frequency and θ is phase angle.

1.4.1 The Eigenvalue problem

We can use the expression for velocity and acceleration for SHM in the governing equation of motion as follows,

$$\{u\} = \{\phi\}_i sin(\omega_i + \theta_i) \tag{1.7}$$

$$\{\ddot{u}\} = -\omega_i^2\{\phi\}_i sin(\omega_i + \theta_i) \tag{1.8}$$

$$[M]\{\ddot{u}\} + [K]\{u\} = [0] \tag{1.9}$$

$$([K] - \omega_i^2\,[M])\{\phi\}_i = [0] \tag{1.10}$$

Here the two unknowns in the equation are omega and phi. To solve for unknowns equation can be converted to an eigenvalue problem with the eigenvalue being the eigenvector.

The angular frequency ω_i is related to the natural frequency f$_i by$;

$$f_i = \frac{\omega_i}{2\pi}(Hz) \tag{1.11}$$

The eigenvector ϕ_i gives the mode shape corresponding to f$_i$.

1.5 Need of Analysis

1. FEA plays a vital role in validating the design and provides close to practical results.

2. The component can be tested for additional load cases occurring at different situation as well.

3. The material of the component can be easily changed and iterated for analysis. It increases the material accuracy while designing a product.

4. Apart from structural characteristics the modal behavior of a structure is very critical to study. If gives idea about excitation frequencies of the system.

5. If neglected, any natural frequency of the structure may resonate it and cause severe damage.

1.6 Background of the project

In heavy-duty trucks, fuel tank mounting brackets play an import role in securely holding the fuel tanks in place during transportation. These brackets are subjected to a range of loads and stresses, including vibrations and impacts caused by uneven road surfaces and sudden movements of the truck. Therefore, it is essential to analyse the structural and modal behavior of fuel tank mounting brackets to ensure their reliability and safety.

Structural analysis is the process of determining the stresses within a structure due to applied loads. It involves using mathematical equations and computer simulations to predict the behavior of a structure under different conditions. In the case of fuel tank mounting brackets, the structural analysis is essential to ensure that the brackets can withstand the loads and stresses imposed on them without failure. Factors such as material properties, geometry, and loading conditions need to be considered during the analysis.

Modal analysis, on the other hand, is the process of determining the natural frequencies and modes of vibration of a structure. It helps in understanding the dynamic behavior of the structure and identifying any potential issues that may arise due to vibrations. In the context of fuel tank mounting brackets, modal analysis is crucial in identifying any potential resonant frequencies that may cause excessive vibrations and lead to failure.

The fuel tank mounting brackets for heavy-duty trucks are typically made of steel or aluminium and have a complex geometry that makes their analysis challenging. Moreover, the loading conditions for these brackets vary depending on the type of truck and the nature of the cargo being transported. Therefore, a detailed analysis is necessary to ensure that the brackets can withstand the loads and stresses imposed on them.

In recent years, computer-aided engineering (CAE) tools such as finite element analysis (FEA) have become popular for simulating the response of any component to the real-world situations. FEA involves dividing the structure into small elements and using mathematical equations to determine the behavior of each element. The results are then combined to obtain the overall behavior of the structure. FEA allows for a more accurate and detailed analysis of the brackets and helps in identifying potential failure modes and designing more efficient and reliable brackets.

Chapter 2

Literature review

Performed the structural and modal analysis of the Chassis ladder frame using FEA method. The Structural analysis was used to determine the stresses in the body and to identify the critical regions. Modal analysis was carried out to calculate the natural frequencies. [1]

Carried out the structural analysis of the chassis frame using FEA and found out critical regions in the assembly. Then they modified the design according to critical regions. After that they optimised the weight of the assembly using the different materials. [2]

Performed FEA on the assembly to understand the modal behavior of the chassis. To get the better results she altered the combination of cross sections and materials. This led to the optimum design. [3]

Carried out the analysis of engine mounting bracket in the work "Finite element analysis and natural frequency optimization of engine bracket" used various methods to optimize the natural frequency of the bracket. [4]

Performed the structural analysis of the fuel tank mounting bracket. They considered various acceleration loads for example load due to sudden acceleration, load due to sudden braking, load due to bumping of vehicle on the road, etc. They iterated the design for the best results on given conditions. [5]

Carried out design and optimization of HTV Fuel Tank Assembly by Finite Element Analysis. Additionally, they carried out the modal analysis. They compared the performance of the system with various rib patterns on the bracket which led to some interesting results in modal frequency and strength of the bracket. [6]

Normal modal analysis for base model was carried out to find the first natural frequency. Normal modal analysis was carried for all the design iteration to improve the first natural frequency of the fuel tank bracket. Static analysis was carried out for all modified designs to find out the Maximum displacement and von misses stress at critical location. Maximum principal stress and minimum principal stresses were carried out. [7]

Nitin G. in his book “Practical Finite Elements Analysis” Finite to infinite has described all the fundamentals of FEA along with good practices to be followed while working on a FE model. Additionally he has mentioned how analysis are done at Industry level to bring down time and cost factors. [8]

The research papers and publications reviewed covered a variety of aspects which must be considered for a better finite element modelling of truck chassis and fuel tank along with their brackets. Symmetry boundary conditions are found to be widely useful such analysis for obtaining quick and efficient results. It allows us to obtain results for the complete model, while just modelling a half or quarter portion of the complete model.

These publications have also discussed what type of meshing for 2D and 3D elements may help to obtain better results, and different ways to apply loads and boundary conditions, that one may find more appropriate to obtain more accurate and close to real life results. what necessary changes can be made in the design to make the design better and stiffer. Additionally, how analysis are done are done at industrial level to bring down the time and cost factor.

Chapter 3

Problem Definition

3.1 Problem Statement

To analyze the Structural and Modal behavior of Fuel Tank Brackets for Heavy duty trucks using Finite Element Analysis for Validation of the New design against 5G loading conditions and finding Mode shapes at their respective frequencies.

3.2 Aim

To improve the design and find out the most suited material for the fuel tank brackets.

3.3 Objectives

1. Analyze the displacements, stresses occurring on the fuel tank mounting bracket under 5G loading conditions for validation of new design.

2. To run the modal analysis and find mode shapes and their natural frequency of the component.

3. Run the analysis for three different materials Cast iron, Steel and Aluminium alloys to find the best suited material for the fuel tank bracket.

4. Review the obtained results and make necessary changes in the design.

Chapter 4

Proposed Methodology

4.1 Steps carried out during Analysis of a system

For the analysis of the fuel tank bracket, Hypermesh 2022 software was used and the following steps were carried out.

1. Import geometry in 'STEP' format: The CAD model is imported in .step format in the Hypermesh software.

2. Geometry review : The CAD model is reviewed properly by studying the geometry and understanding the requirements and instructions provided. The components which are not required for the analysis purpose are suppressed.

3. Geometry cleanup : It is done to close gaps in between surfaces, combine or split surfaces into meshing regions, merging vertices which are close to each other, also checking for free edges, Scar lines, Duplicate surfaces, small fillets, small holes, beads, intersection of parts (in case of assembly of components) helps in building a better mesh.

4. Meshing : Meshing of individual components is carried out. Considering the geometry and dimensions of the component the type of mesh and the element size is decided.

5. Defining material and properties to each component : The materials and properties which will be assigned to different components are defined. These materials and properties are then individually assigned to the respective components.

6. Defining boundary conditions and application of loads : Boundary conditions and the load application on the model is defined.

7. Running the Solver: After checking for errors, the Optistruct solver is run. The software internally carries out matrix formations, inversions, multiplication and solution for displacement, strain and stress for static analysis.

8. FE modal review and results review

9. Post processing : The obtained results are viewed, verified and studied. Conclusions are drawn from these results for what steps can be taken for improvements in the design.

10. Report generation and submission of report

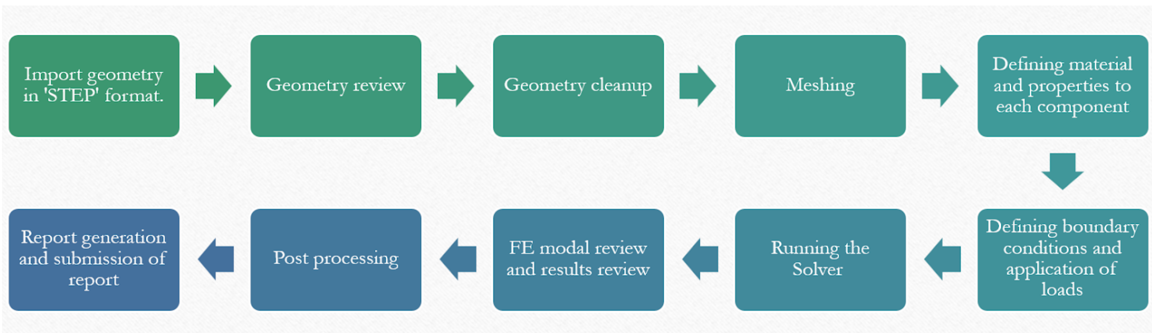

Figure 4.1: Methodology

4.2 Concerned Component

In this analysis, Bracket is the concerned component for analysis. Bracket is used for clamping the fuel tank to the truck chassis with the help of a strap, which is connected at both the ends with the help of bolts. The bracket has 4 mounting holes, using which it is mounted to the Long member using bolts. This bracket will be analyzed using three different materials, Cast Iron, Aluminum and steel alloys.

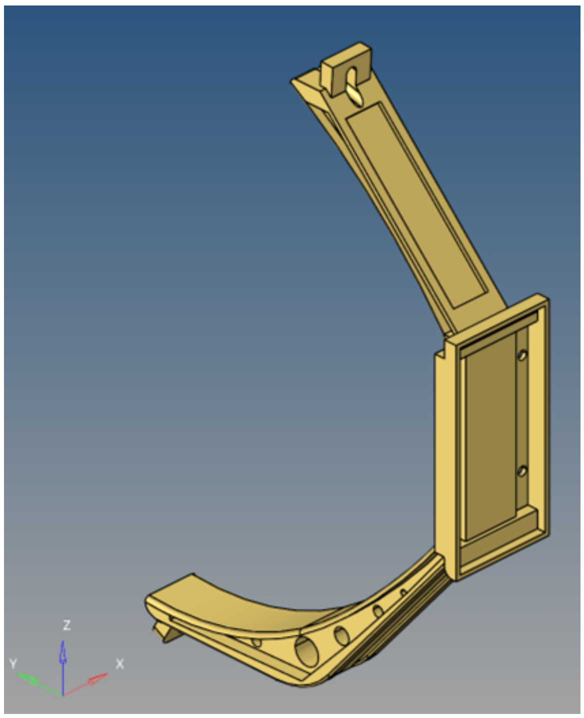

Figure 4.2: Bracket(Concerned Component)

Sr. No.	Material	Young's Modulus (in MPa)	Poission's Ratio	Density (in tonnes/mm^3)
1	Cast Iron	80000	0.27	6e-09
2	Aluminium Alloy	71000	0.33	2.9e-09
3	Steel Alloy	210000	0.3	7.85e-09

Table 4.1: Material Properties

4.3 CAD Model

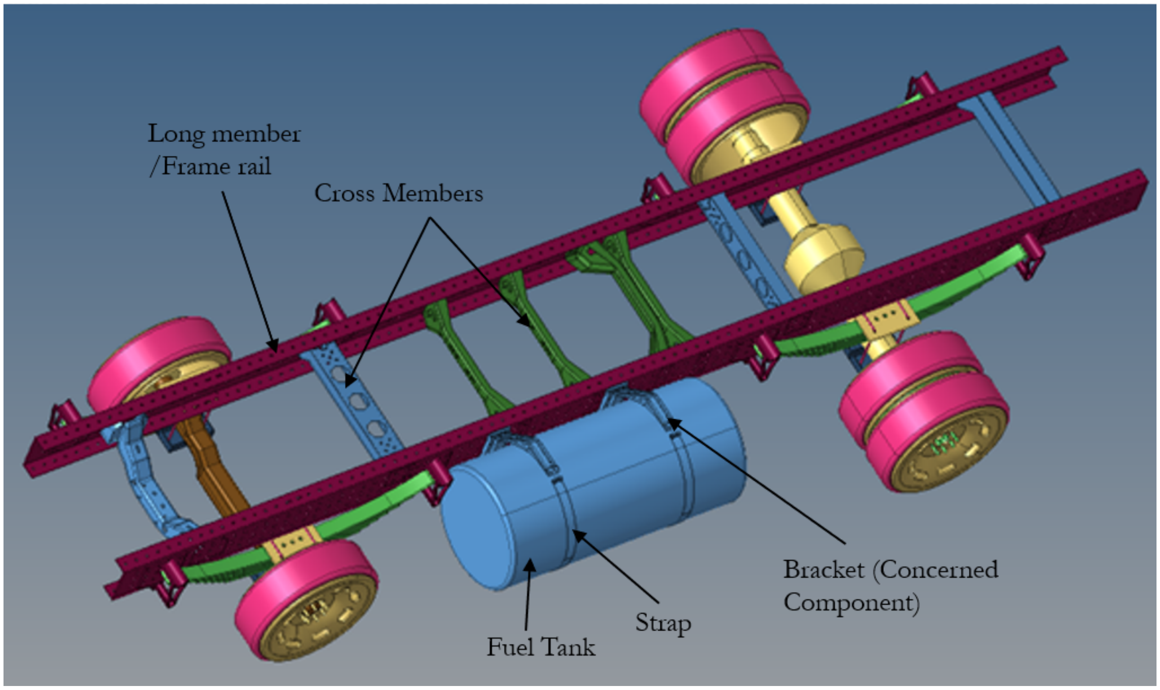

Figure 4.3: CAD Model of Truck Chassis

In this analysis, the chassis of a heavy duty truck is considered. The CAD model consists of several components which are not required for the analysis of fuel tank bracket, which is the concerned part in this study. So, other parts which are not required in performing the analysis are suppressed/removed and only the below parts are considered and meshed accordingly for the analysis of bracket.

1. Bracket
2. Fuel tank
3. Strap
4. Cross members
5. Long members

4.4 FE Model

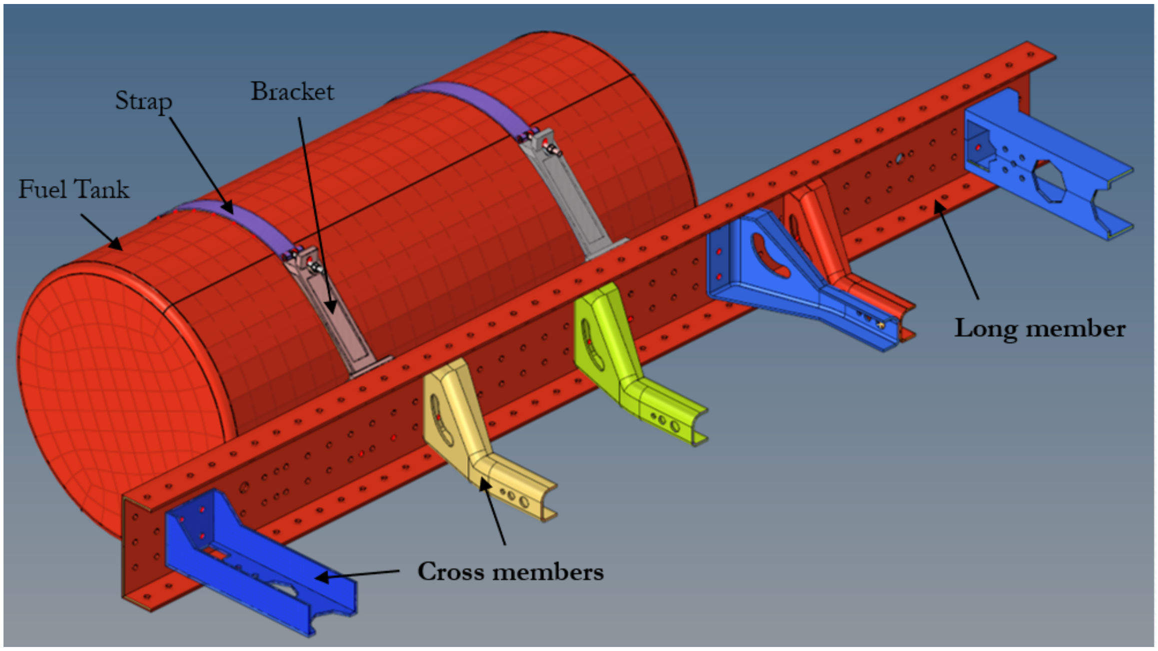

Figure 4.4: FEA Model of Fuel Tank Assembly

Sr. No.	Name of the Component	No. of Components in the assembly
1	Fuel Tank	1
2	Bracket	2
3	Strap	2
4	Cross Members	6
5	Long Members	1

Table 4.2: List of Components

4.5 Preprocessing

Geometry cleanup is performed for each part individually. Free edges, duplicate surfaces, small fillets and surfaces are combined or split into different meshing regions to build a better mesh.

4.6 Meshing

Reason for carrying out meshing is to reduce the number of degrees of freedom of an object from infinite to finite with the help of discretization i.e., meshing (nodes & elements). This reduces the number of equations and unknown, which also reduces the computational cost. The figure below illustrates the importance of meshing.

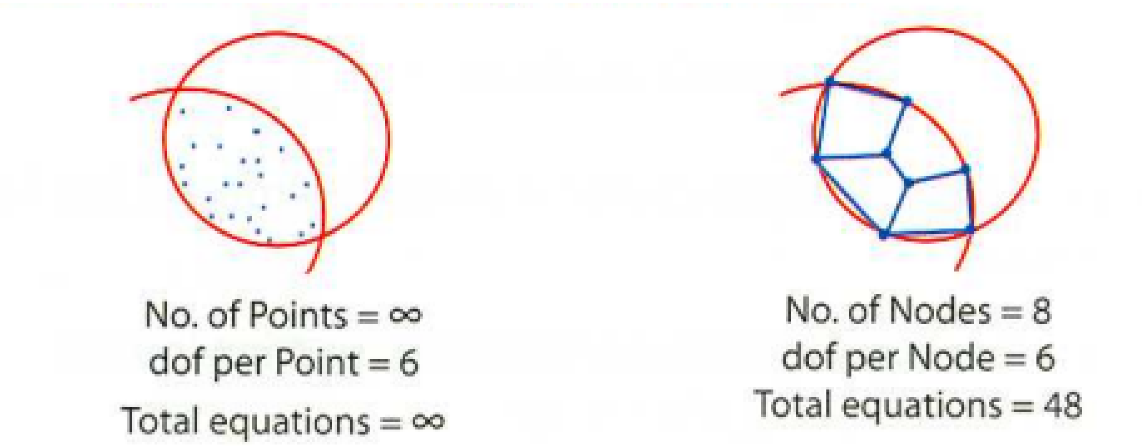

Figure 4.5: Importance of Meshing

The calculations are made at limited number of points known as nodes. The entity joining nodes and forming a specific shape such as quadrilateral or triangular etc, is known as an element.[8]

4.6.1 Types of elements used in meshing

There are basically three types of elements used in meshing and namely 1-D, 2-D and 3-D. The type of element to be used is decided based on geometry of the component.

If one of the dimensions is greater compared to rest of the two then 1D meshing is used. Remaining two dimensions are taken as additional input from i.e., cross section

area of the geometry. These elements include rod, bar, beam, pipe, axi-symmetric shell, etc. Element have a shape of a line.
When two of the dimensions are very large compared to third then 2-D elements are used. These types of elements are created at midsurface of geometry and thickness is provided as additional input from user. In this case elements are either of quad or tria shape. Element's type includes thin shell, plate, membrane, plain stress, plane strain, axi-symmetric solid, etc. The 3-D elements are used when all three dimensions are comparable. No additional data is required from user in this case. Element type is solid. Shape includes tetrahedral and hexahedral.

4.6.2 Quality of elements

Result quality is totally dependent on element quality. If quality is good then we can obtain solution close to the practical value. It is so much important to maintain good quality of element in the finite element model that if not done, it may happen that simulation won't work out i.e., it won't be even initiated. There are various quality checks available. Some of the important are listed below.

- Warp Angle: It is out of plane angle. Only applicable for quad elements. It is defined as angle between normal to two planes formed by splitting the quad element along diagonals. Maximum angle out of two possibilities is reported as warp angle.

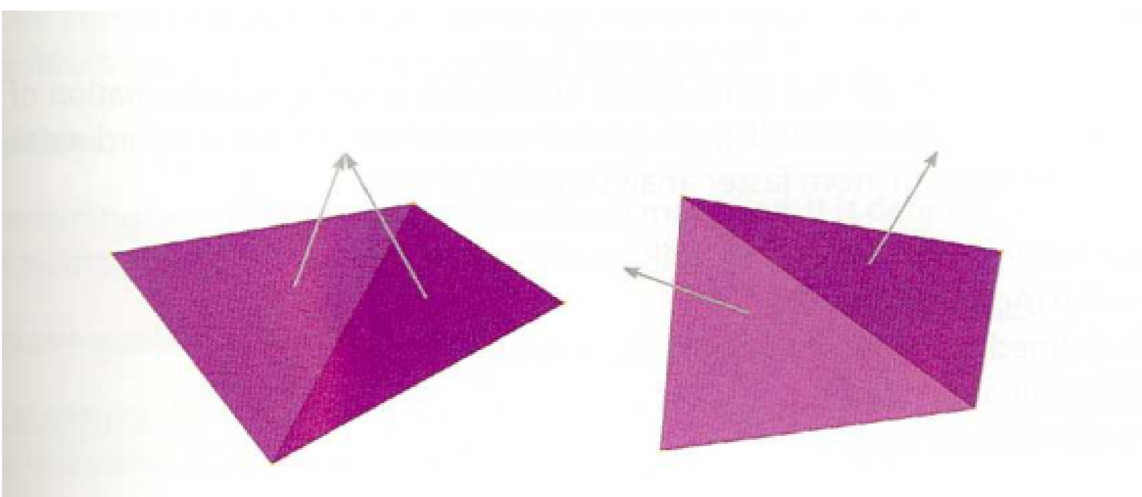

Figure 4.6: Warp Angle

- Aspect Ratio: It is the ratio of maximum element edge length to minimum element edge length.

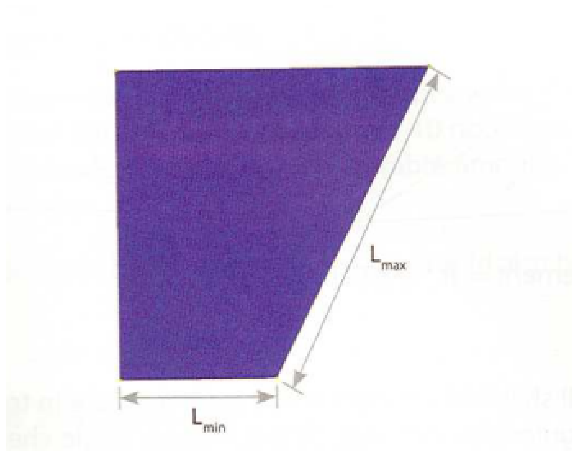

Figure 4.7: Aspect Ratio

- Skew: It is calculated differently for quadrilateral and triangular elements. For quadrilateral elements skew is difference between 900 and minimum angle between two lines joining opposite mid-sides of the element. For triangular elements skew is calculated as difference between 900 and minimum angle between the lines from each node to the opposing mid-surface and between the two adjacent mid-sides at each node of the element.

- Jacobian: In simple words we can define Jacobian as scale factor raising due to transformation of co-ordinate system. Elements are transformed form global coordinates to local coordinates for faster calculations.

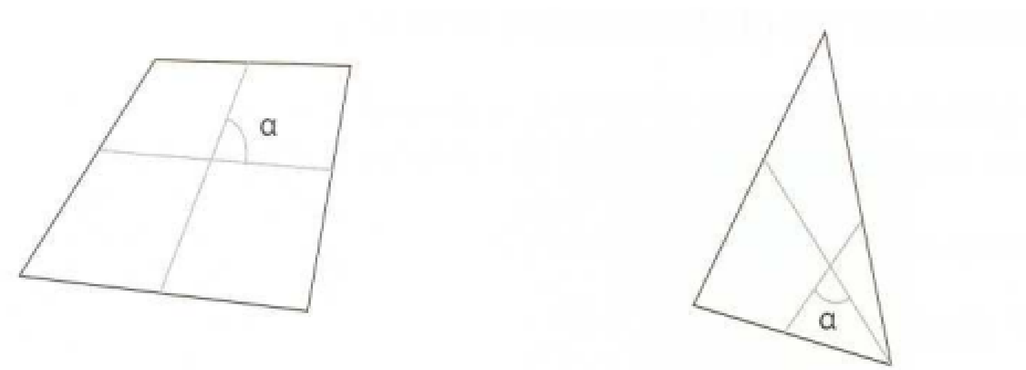

Figure 4.8: Jacobian

- Tet collapse: It is one of the most important quality checks for 3-D elements. The height of the tetra element is measured from each of the four nodes to its opposite face, and then divided by the square root of the face's area. The minimum of the four resulting values (one per node) is then normalized by dividing it by 1.24

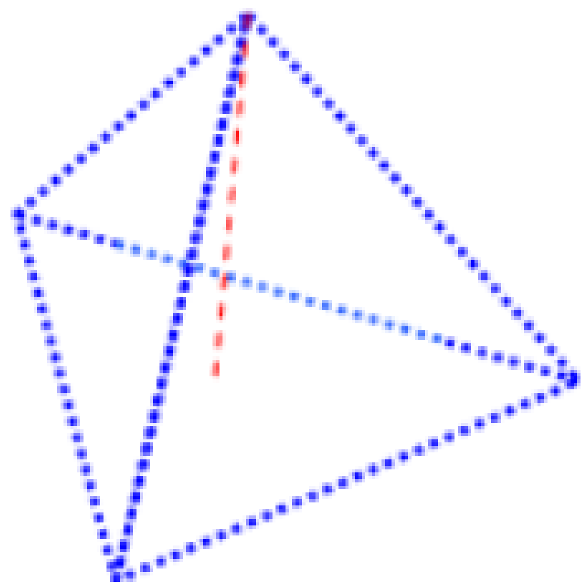

Figure 4.9: Tet collapse

4.6.3 Process of Meshing

Meshing is carried out for each component individually. For the components having 2D and 3D geometry, mixed and tetramesh are carried out respectively.

Sr. No.	Name of the Component	Type of Mesh	Element Size(in mm)
1	Fuel Tank	2D Mixed Mesh	30
2	Bracket	3D TetraMesh	10
3	Strap	3D TetraMesh	15
4	Cross Members	2D Mixed Mesh	10
5	Long Members	2D Mixed Mesh	15

Table 4.3: Type of Mesh and Element Size

For performing the 2D meshing of the components, the mid surface of the component is extracted and the mixed mesh forms an automatic mesh on the mid surface consisting of Quad and Tria elements. Quad element is better than tria element because of better interpolation function. Tria elements are stiffer when compared to quad elements, hence they result in lesser stress and displacement values if used in critical locations. So, it is attempted to reduce the number of trias as much as possible to obtain accurate results. These elements are also checked for minimum and maximum element size, wrap angle, warpage, Jacobian and aspect ratio values. Initially all the surfaces of the component are meshed using R-tria elements. These elements are checked for minimum length criteria by using the check elements tool. If some elements are found to have their minimum length less than 0.1mm then their length is manually increased by using the translate tool. Once the minimum length criteria is satisfied the tetramesh is performed on these 2D elements to obtain a 3D Tetramesh.

The generated tetramesh is then checked for tet collapse, which is a very important criteria for the failure of elements. The elements having the tet collapse value less than 0.3 are repaired using the fix elements tool available in the tetramesh panel. After fixing of these elements if still some elements are failing then they are isolated using the mask tool and their height is manually increased normal to the surface plane.

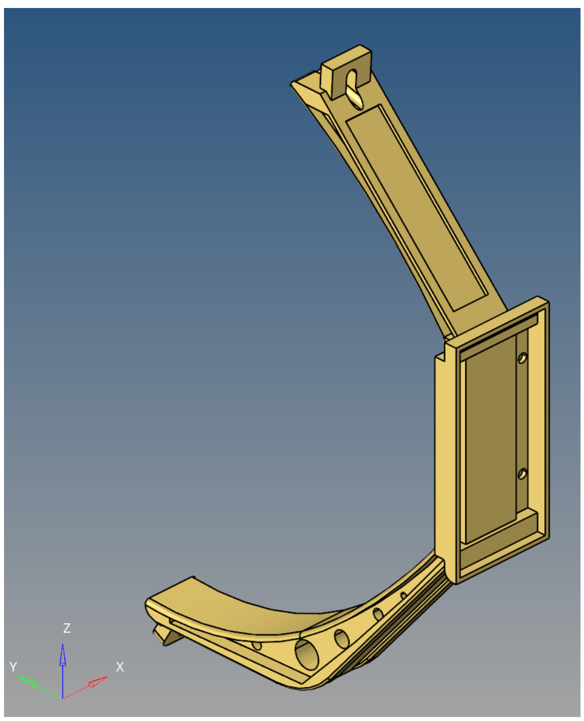

Figure 4.10: CAD Model of Fuel Tank Bracket

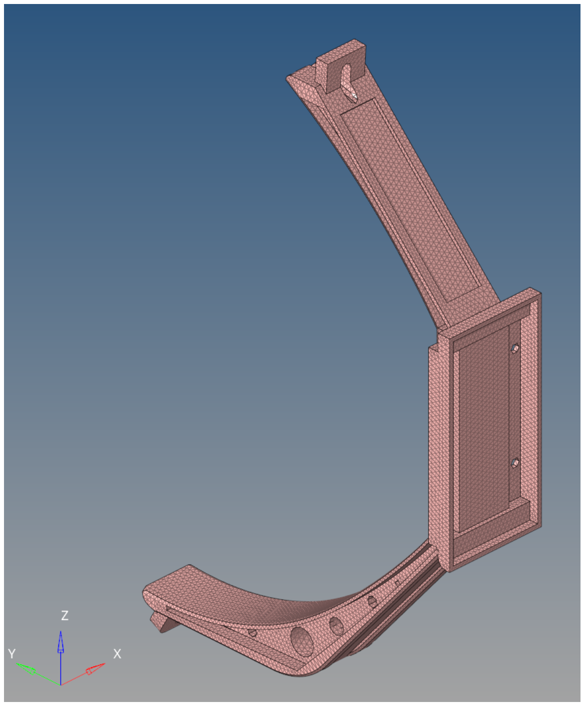

Figure 4.11: 3D Tetra-meshed Model of Fuel Tank Bracket

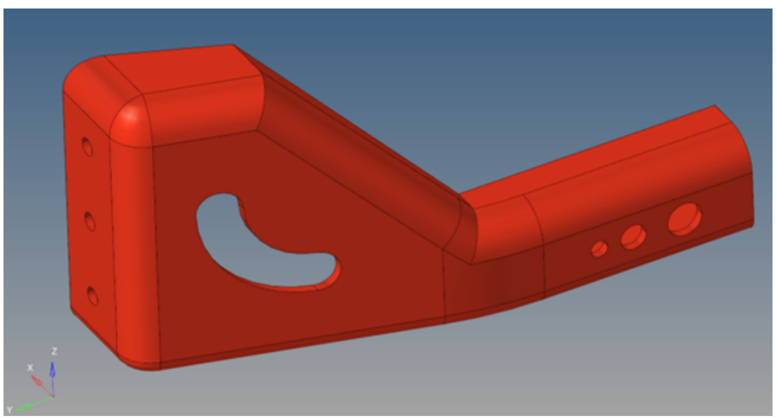

Figure 4.12: CAD Model of a Cross Member

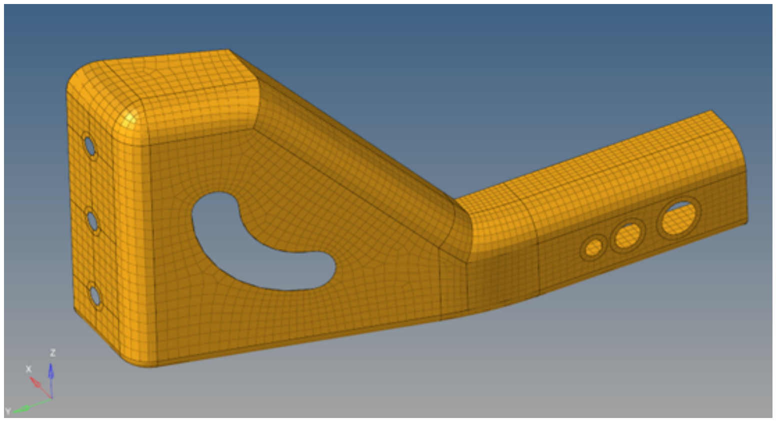

Figure 4.13: 2D Mixed Model of Cross Member

4.6.4 Assigning materials and their specifications

Once the meshing and element quality check of all components is completed. Materials are assigned to respective components according to their specifications and requirements.

Sr. No.	Name of the Component	Material Used	Dimensions (in mm)	Weight (in kg)
1	Tank	Steel	D: 656; L: 1400; t: 3	150
2	Bracket	Cast Iron, Aluminium Alloy, Steel Alloy	Max. radius: 350	Cast iron - 8.92, Aluminium - 4.31, Steel - 11.68
3	Cross Members and Long Members	Steel		
4	Strap	Steel	W: 50; t: 4	1.4

Table 4.4: Material Specifications

4.6.5 Assigning properties

After material cards are ready, properties cards are created. Properties are unique for each component, it consists of Thickness of 2D components, material, and card image which defines whether the component is Solid / Shell / Beam.

Sr. No.	Name of the Component	Type of Element	Card Image	Thickness (in mm)	Material
1	Fuel Tank	2D	PShell	3	Steel
2	Bracket	3D	PSolid		Cast Iron, Aluminium Alloy, Steel Alloy
3	Strap	3D	PSolid		Steel
4	Cross Members	2D	PShell	7.9 and 6.35	Steel
5	Long Members	2D	PShell	7.9	Steel
6	Bolts	1D	PBeam	Diameter-12	Steel

Table 4.5: Properties Assigned

1. PSolid: It is assigned to 3D elements of solids.
2. PShell: It is assigned to 2D elements and the 3rd dimension is stored as a thickness.
3. PBeam: It is assigned to 1D elements by storing the 2nd dimension as a separate beam section property.

4.6.6 Connections

Making connections is a very important step before the model is necessary for analysis. Connections are made to connect components with each other. It must be made sure that all the elements are connected in the model. This is achieved by using rigids and bars.

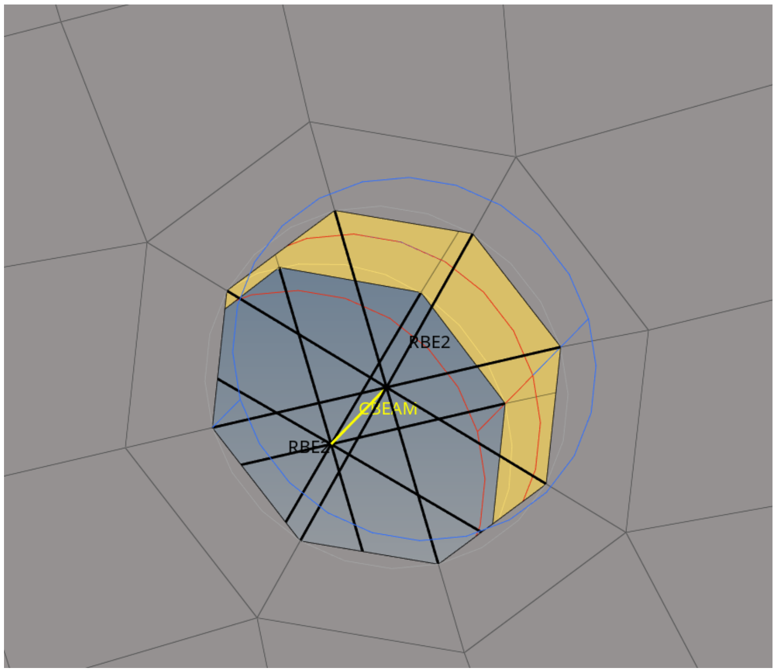

Figure 4.14: Rigids and Bar elements

1. Rigid elements (REB2) are 1D elements created in space between two nodes of a model where a rigid connection is desired.
2. Bars (CBEAM) are created to connect a rigid of one component to the rigid of another component as it is supposed to be connected using a bolt.

4.6.7 Loads and boundary conditions

1. Symmetric boundary conditions are applied at sections cut at cross members. This allows modeling half portion of the truck chassis which reduces the number of nodes, elements, and computational costs.

2. 5G loading along all three X, Y, and Z axis for forces occurring due to :

   (a) Vertical acceleration: impact due to wheel passing over a speed breaker or potholes - 3g

   (b) Lateral acceleration: Cornering force, acts when the vehicle takes a turn on curvatures - 0.5 to 1 g

   (c) Axial acceleration: Braking sudden acceleration - 0.5 to 1 g

3. Using SPC (Single point contacts) applied at nodes, DOF for translation along the Y axis is restricted and rotational DOF along the remaining two axis namely X and Z are restricted.

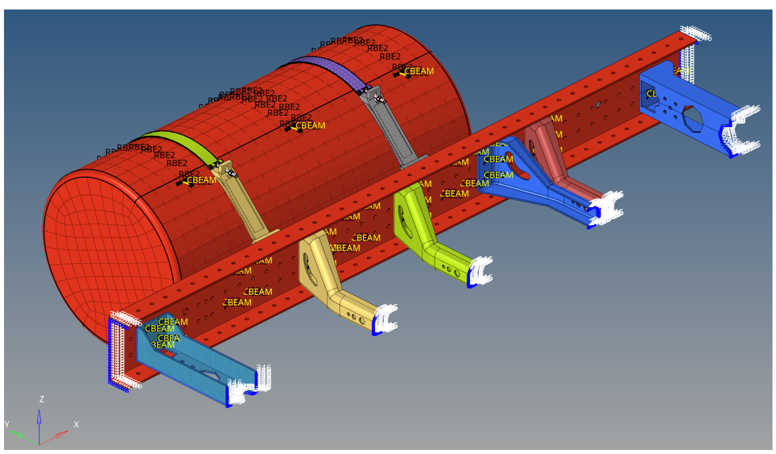

Figure 4.15: SPC applied at nodes

4.6.8 Assumptions

1. Materials are considered linear materials.

2. Bolts are modeled using 1D elements.

3. No continuous contact between components.

Chapter 5

Results and Discussions

5.1 Material as Cast Iron

5.1.1 5G Loading in X Direction

The below plot shows the results of 5G loading in X direction, the assigned material is Cast Iron.
Maximum Displacement: 3.79 mm
Maximum Stress: 72.49 MPa

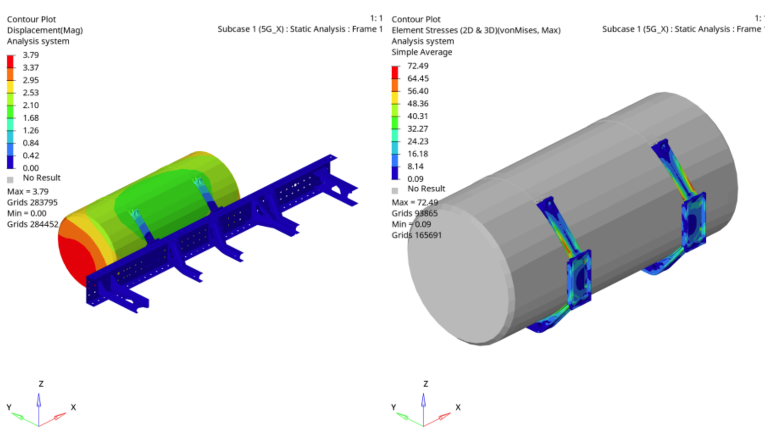

Figure 5.1: Displacement and stress plot in X direction

5.1.2 5G Loading in Y Direction

The below plot shows the results of 5G loading in Y direction, the assigned material is Cast Iron.
Maximum Displacement: 1.45 mm
Maximum Stress: 46.78 MPa

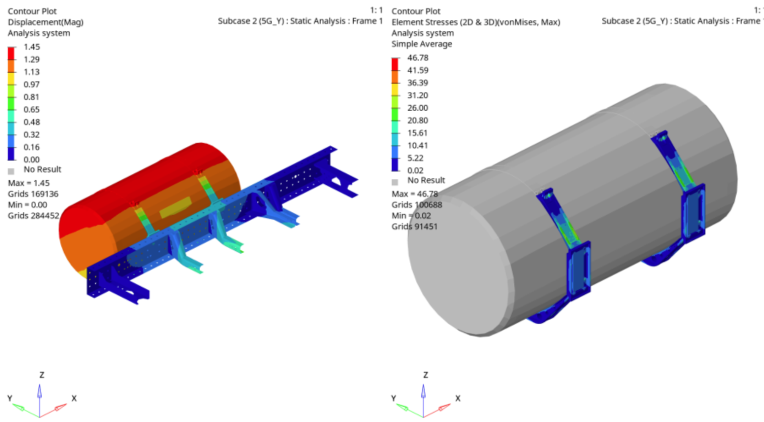

Figure 5.2: Displacement and stress plot in Y direction

5.1.3 5G Loading in Z Direction

The below plot shows the results of 5G loading in Z direction, the assigned material is Cast Iron.

Maximum Displacement: 17.01 mm

Maximum Stress: 161.38 MPa

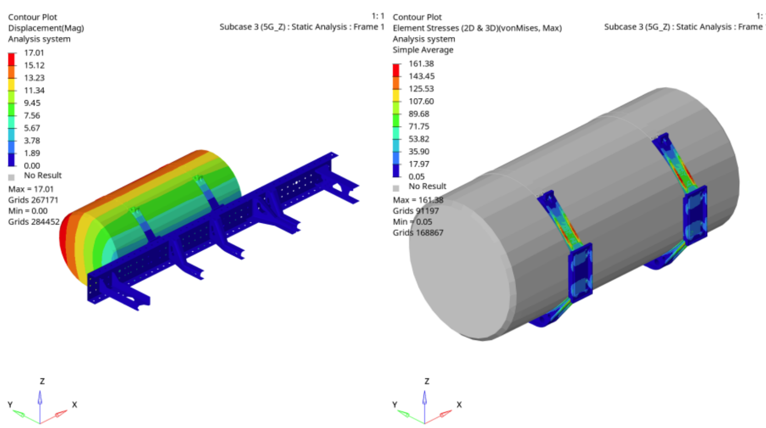

Figure 5.3: Displacement and stress plot in Z direction

5.1.4 Mode shapes

The below plot shows the 1st and 2nd mode shape of the system.

f1 = 9.33 Hz

f2 = 18.2 Hz

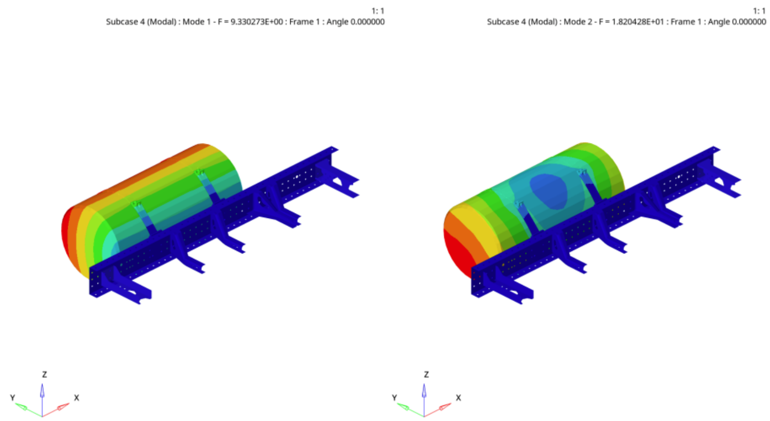

Figure 5.4: Modal frequencies of mode 1 and 2

The below plot shows the 3rd and 4th mode shape of the system.

f3 = 29.65 Hz

f4 = 33.42 Hz

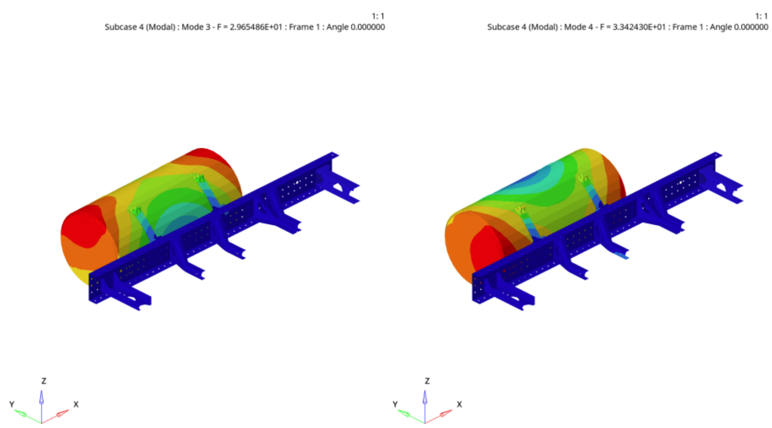

Figure 5.5: Modal frequencies of mode 3 and 4

The below plot shows the 5th and 6th mode shape of the system.

f5 = 35.07 Hz

f6 = 51.65 Hz

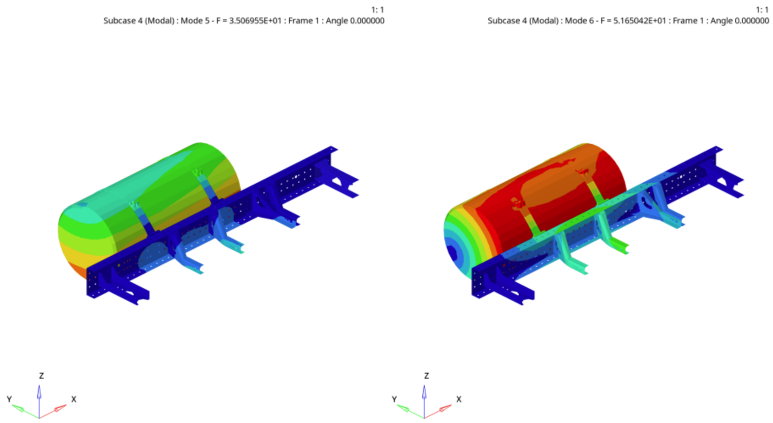

Figure 5.6: Modal frequencies of mode 5 and 6

The below plot shows the 7th and 8th mode shape of the system.
f7 = 61.42 Hz
f8 = 61.53 Hz

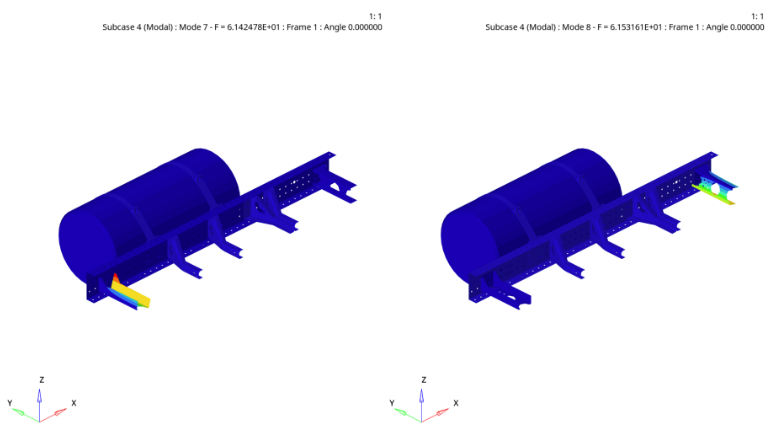

Figure 5.7: Modal frequencies of mode 7 and 8

The below plot shows the 9th and 10th mode shape of the system.
f9 = 66.08 Hz
f10 = 70.03 Hz

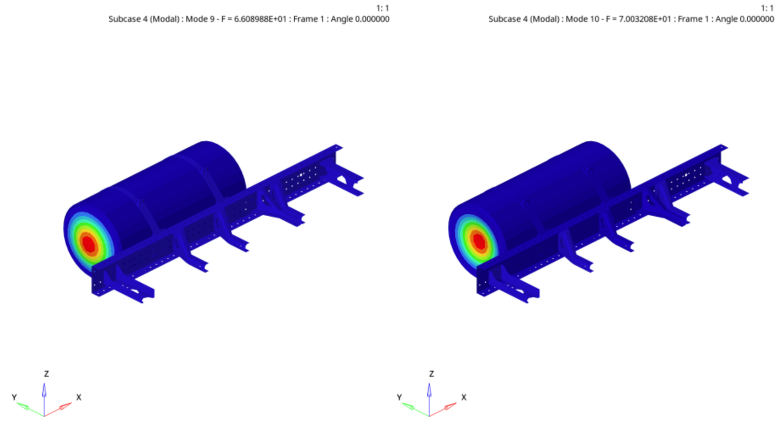

Figure 5.8: Modal frequencies of mode 9 and 10

5.2 Material as Aluminum alloy

5.2.1 5G Loading in X Direction

The below plot shows the results of 5G loading in X direction, the assigned material is Aluminum alloy.
Maximum Displacement: 3.92 mm
Maximum Stress: 69.88 MPa

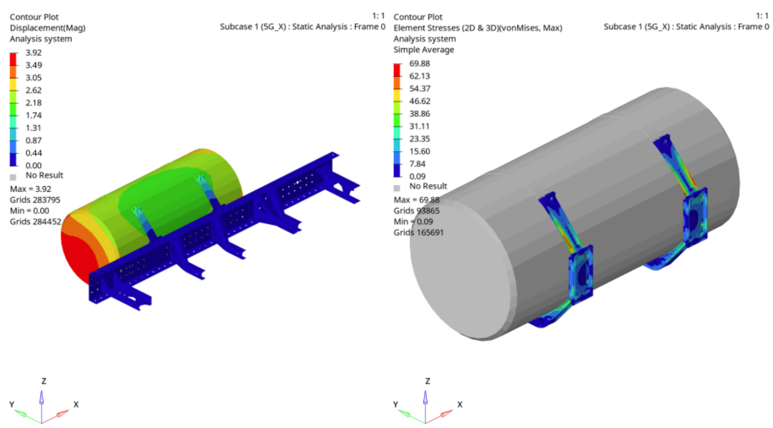

Figure 5.9: Displacement and stress plot in X direction

5.2.2 5G Loading in Y Direction

The below plot shows the results of 5G loading in Y direction, the assigned material is Aluminum alloy.
Maximum Displacement: 1.57 mm

Maximum Stress: 44.07 MPa

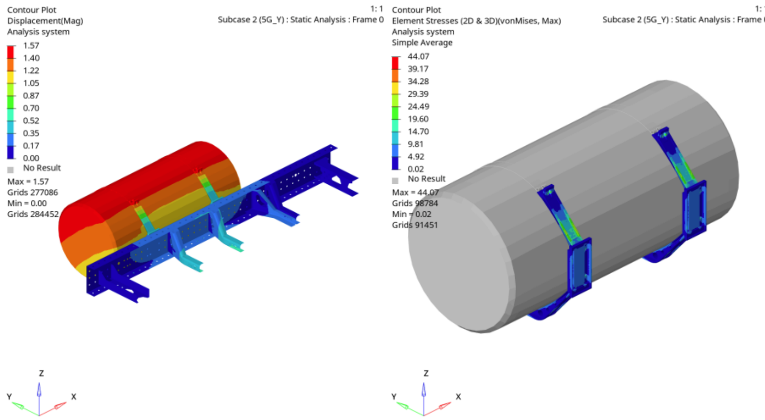

Figure 5.10: Displacement and stress plot in Y direction

5.2.3 5G Loading in Z Direction

The below plot shows the results of 5G loading in Z direction, the assigned material is Aluminum alloy.
Maximum Displacement: 17.93 mm
Maximum Stress: 156.37 MPa

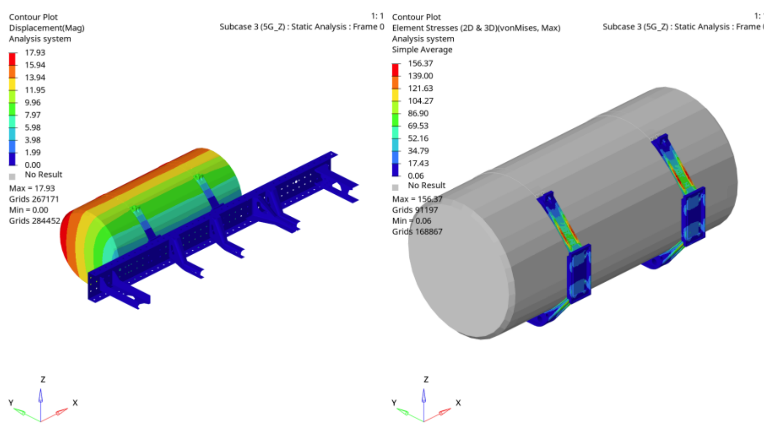

Figure 5.11: Displacement and stress plot in Z direction

5.2.4 Mode shapes

The below plot shows the 1st and 2nd mode shape of the system.
f1 = 9.07 Hz
f2 = 17.78 Hz

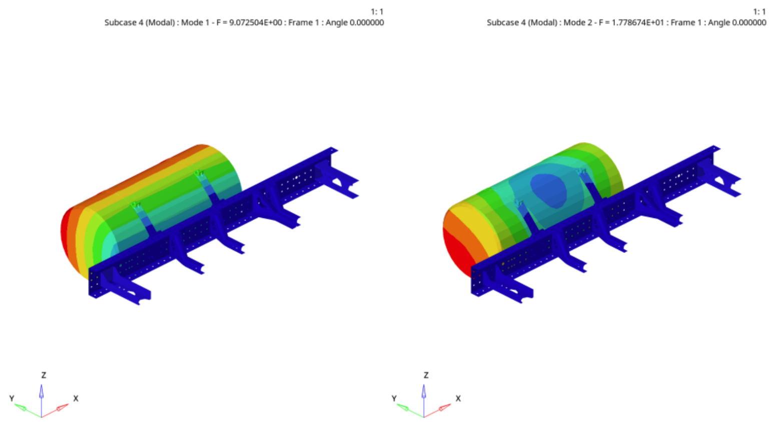

Figure 5.12: Modal frequencies of mode 1 and 2

The below plot shows the 3rd and 4th mode shape of the system.
f3 = 29.11 Hz
f4 = 32.95 Hz

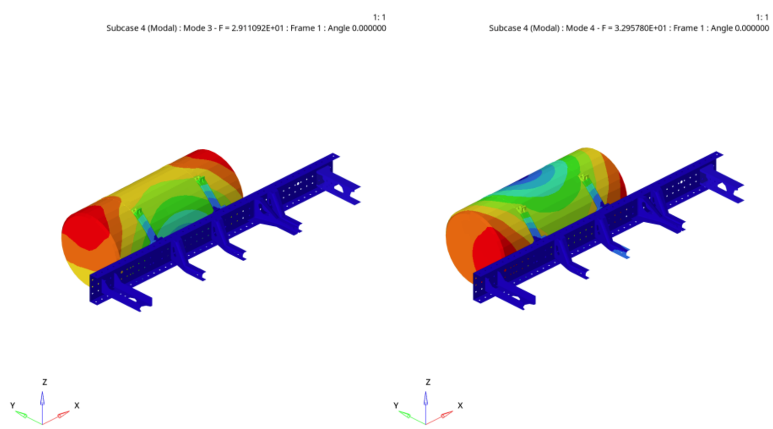

Figure 5.13: Modal frequencies of mode 3 and 4

The below plot shows the 5th and 6th mode shape of the system.
f5 = 35.07 Hz
f6 = 52.00 Hz

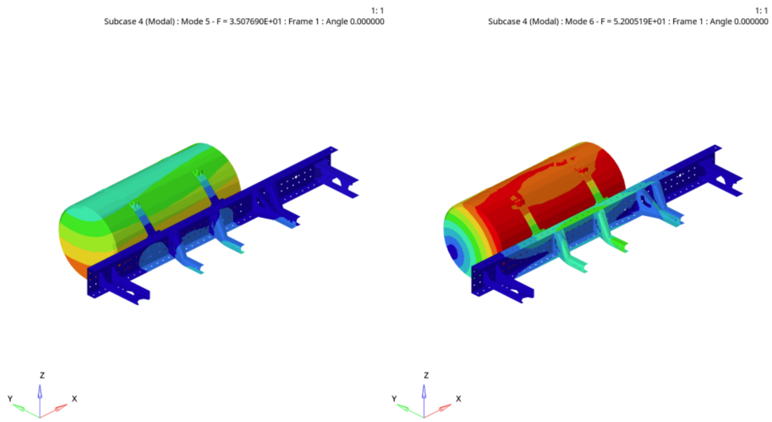

Figure 5.14: Modal frequencies of mode 5 and 6

The below plot shows the 7th and 8th mode shape of the system.
f7 = 61.42 Hz
f8 = 61.53 Hz

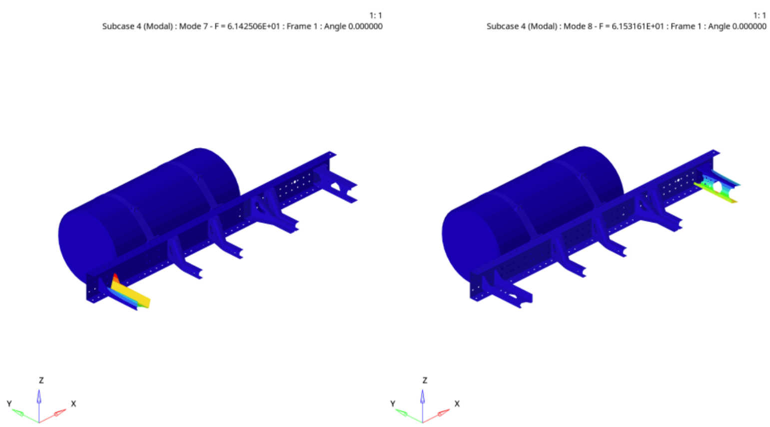

Figure 5.15: Modal frequencies of mode 7 and 8

The below plot shows the 9th and 10th mode shape of the system.
f9 = 66.08 Hz
f10 = 70.05 Hz

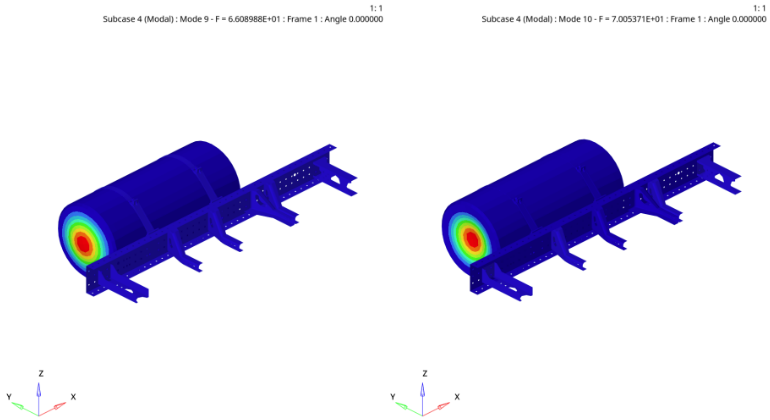

Figure 5.16: Modal frequencies of mode 9 and 10

5.3 Material as Steel

5.3.1 5G Loading in X Direction

The below plot shows the results of 5G loading in X direction, assigned material is Steel.
Maximum Displacement: 2.57 mm
Maximum Stress: 74.03 MPa

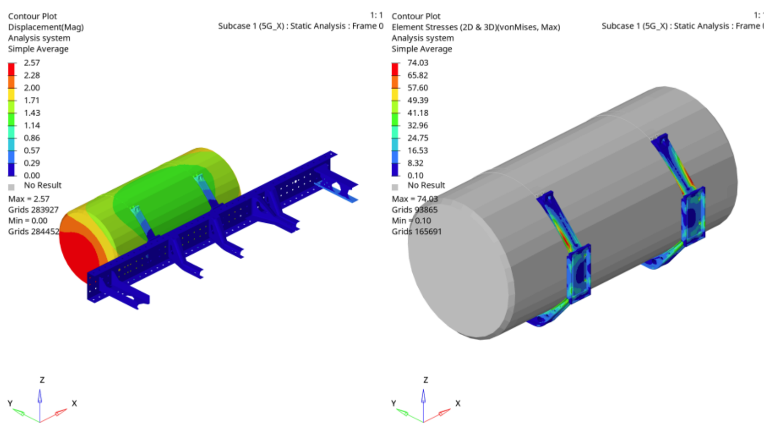

Figure 5.17: Displacement and stress plot in X direction

5.3.2 5G Loading in Y Direction

The below plot shows the results of 5G loading in Y direction, assigned material is Steel.
Maximum Displacement: 1.04 mm

Maximum Stress: 44.81 MPa

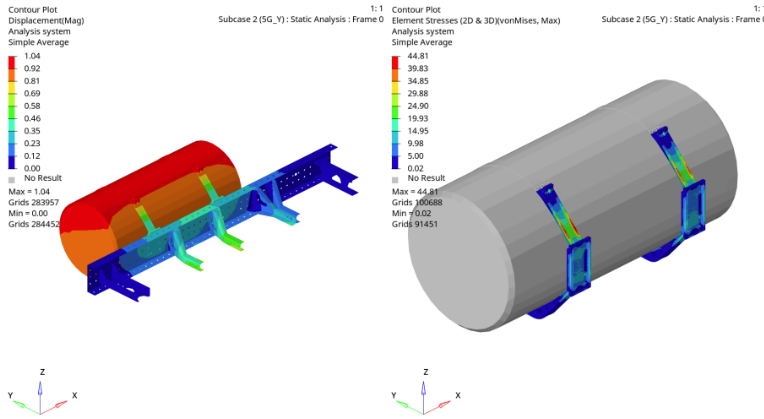

Figure 5.18: Displacement and stress plot in Y direction

5.3.3 5G Loading in Z Direction

The below plot shows the results of 5G loading in Z direction, assigned material is Steel.
Maximum Displacement: 10.06 mm
Maximum Stress: 161.62 MPa

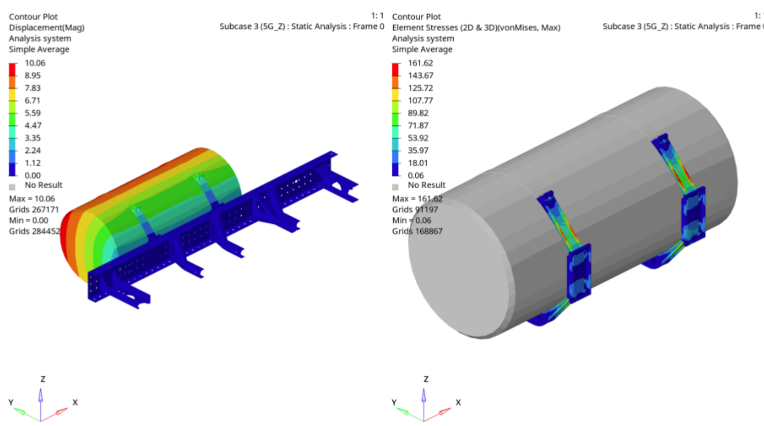

Figure 5.19: Displacement and stress plot in Z direction

5.3.4 Mode shapes

The below plot shows the 1st and 2nd mode shape of the system.
f1 = 12.04 Hz
f2 = 22.4 Hz

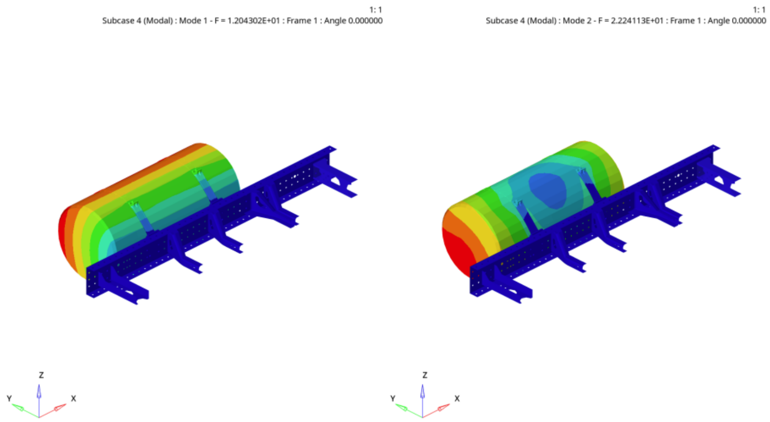

Figure 5.20: Modal frequencies of mode 1 and 2

The below plot shows the 3rd and 4th mode shape of the system.

f3 = 33.72 Hz

f4 = 38.94 Hz

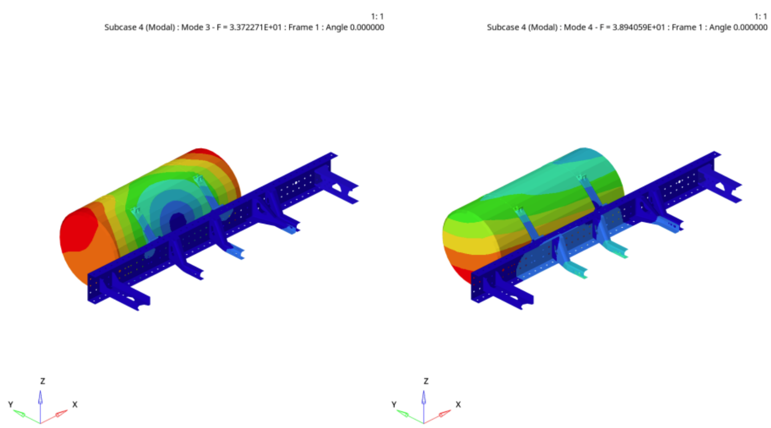

Figure 5.21: Modal frequencies of mode 3 and 4

The below plot shows the 5th and 6th mode shape of the system.

f5 = 40.14 Hz

f6 = 53.59 Hz

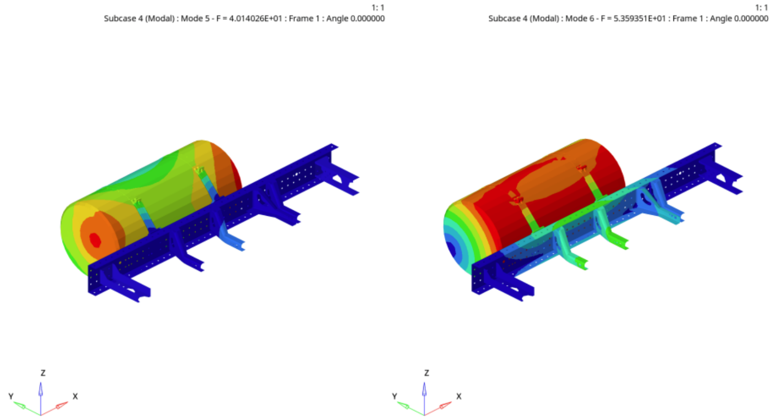

Figure 5.22: Modal frequencies of mode 5 and 6

The below plot shows the 7th and 8th mode shape of the system.
f7 = 61.42 Hz
f8 = 61.53 Hz

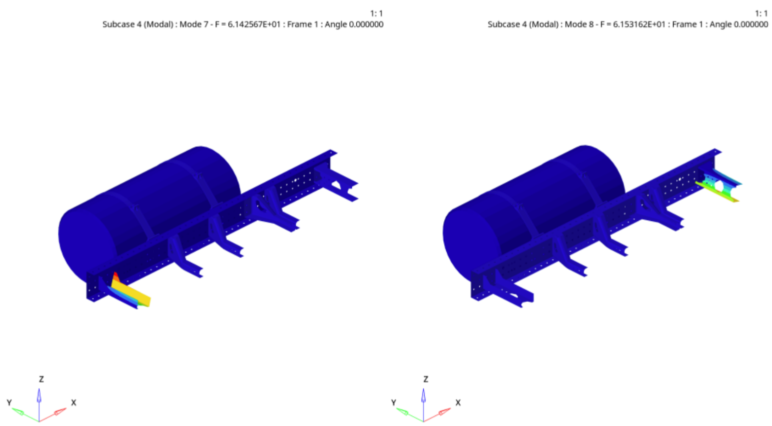

Figure 5.23: Modal frequencies of mode 7 and 8

The below plot shows the 9th and 10th mode shape of the system.
f9 = 66.08 Hz
f10 = 70.04 Hz

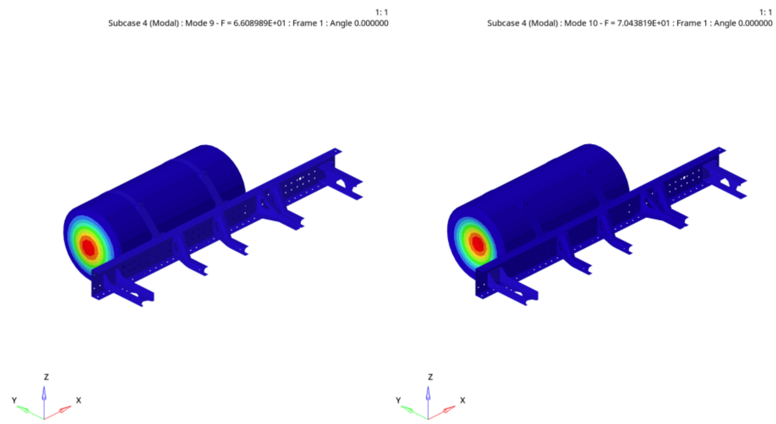

Figure 5.24: Modal frequencies of mode 9 and 10

5.4 Design changes

The Displacement and Modal frequency values for the three materials carried out are not favorable. This results show higher displacement values: 5G loading in Z direction: Cast iron - 17.01 mm, Aluminium 7075 - 17.93 mm, Steel - 10.16 mm.
The 1st mode shape should have a frequency close to 15 Hz or above. The highest 1st mode shape frequency obtained is 12.04 Hz for Steel.
Hence, now we need to do some necessary changes in the design to make the fuel tank assembly stiffer. Some common practices that are used for satisfying this purpose are addition of components, reducing or increasing weight at some specific areas, adding ribs at possible locations, etc. By doing such suitable changes in the design the system becomes stiffer which results in better analysis results.
In this model the required design changes made to make the system stiffer are as follows:

1. Addition of another set of Bracket and strap.

2. Adding ribs in two Cross members at possible locations, increasing the section modulus of the cross members.

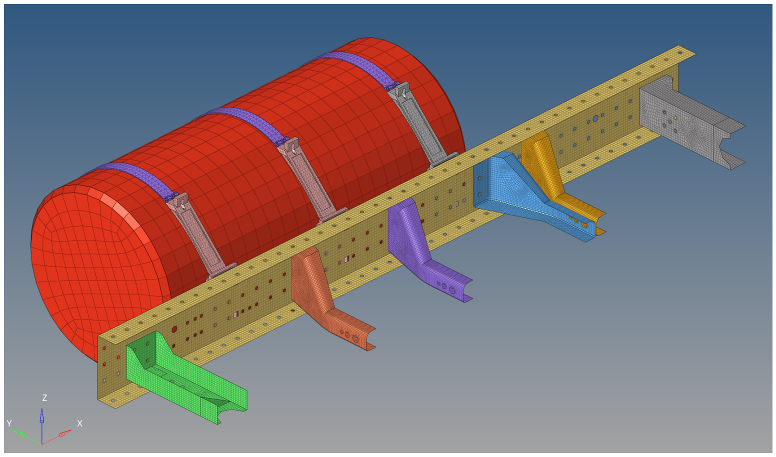

Figure 5.25: Addition of another set of Bracket and strap

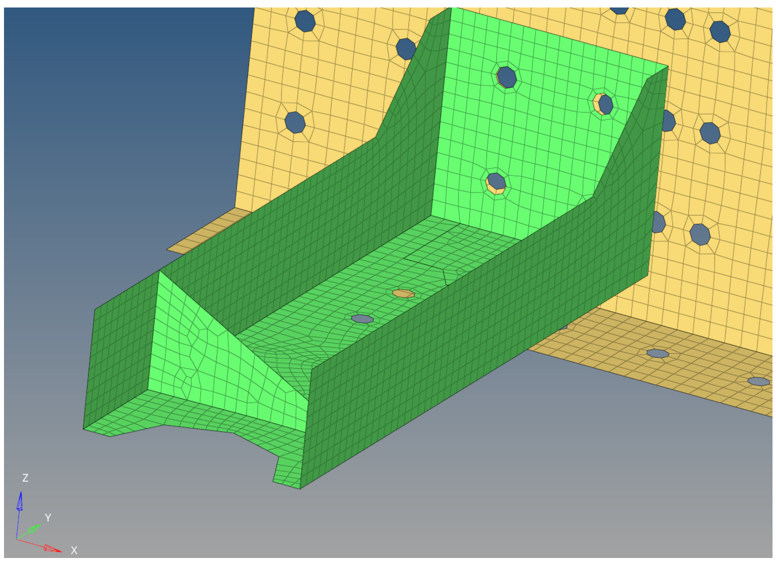

Figure 5.26: Adding ribs to Cross members at possible locations

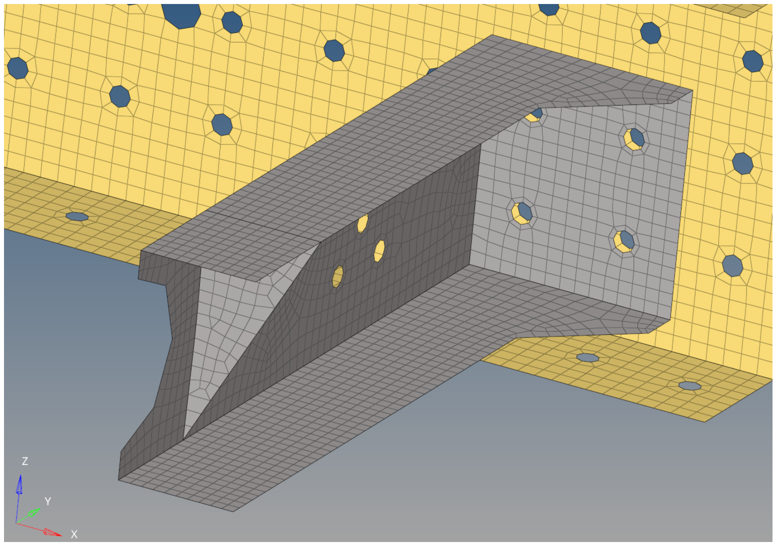

Figure 5.27: Adding ribs to Cross members at possible locations

5.4.1 5G Loading in X Direction

The below plot shows the results of 5G loading in X direction after doing the necessary changes in the design, assigned material is Steel.
Maximum Displacement: 1.52 mm
Maximum Stress: 48.7 MPa

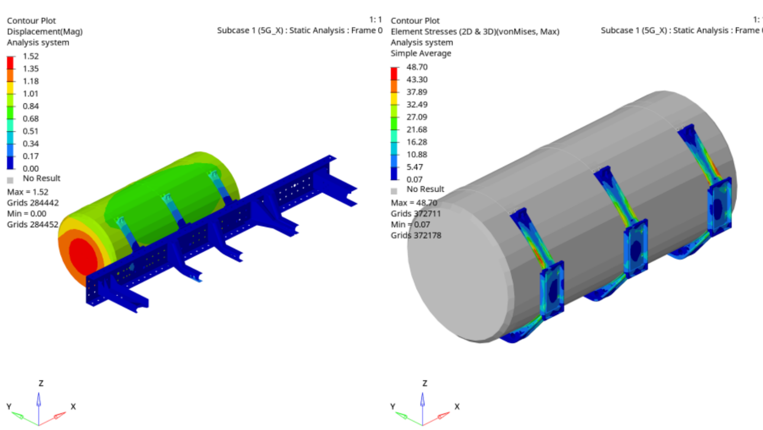

Figure 5.28: Displacement and stress plot in X direction

5.4.2 5G Loading in Y Direction

The below plot shows the results of 5G loading in Y direction after doing the necessary changes in the design, assigned material is Steel.
Maximum Displacement: 0.67 mm

Maximum Stress: 40.72 MPa

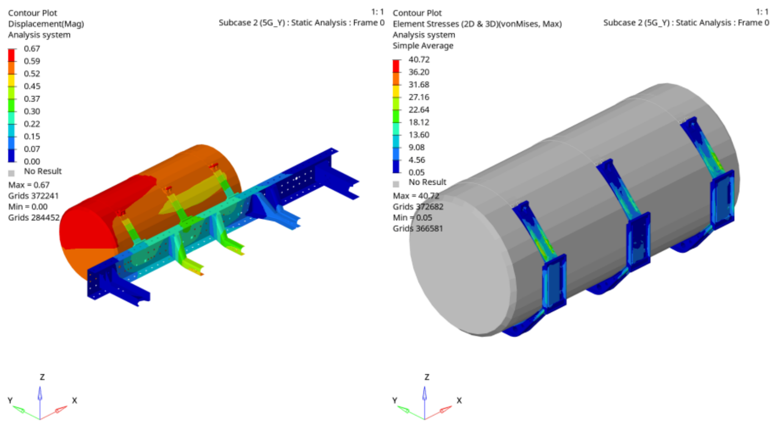

Figure 5.29: Displacement and stress plot in Y direction

5.4.3 5G Loading in Z Direction

The below plot shows the results of 5G loading in Z direction after doing the necessary changes in the design, assigned material is Steel.
Maximum Displacement: 0.89 mm
Maximum Stress: 143.56 MPa

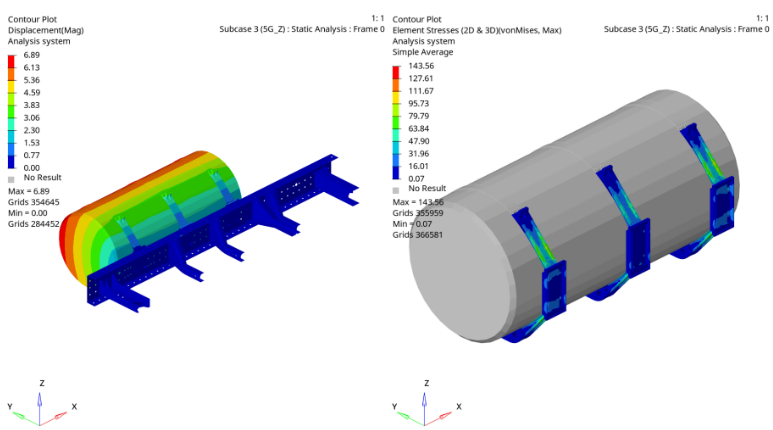

Figure 5.30: Displacement and stress plot in Z direction

5.4.4 Mode Shapes

The below plot shows the 1st and 2nd mode shape of the system.
f1 = 14.457 Hz
f2 = 32.33 Hz

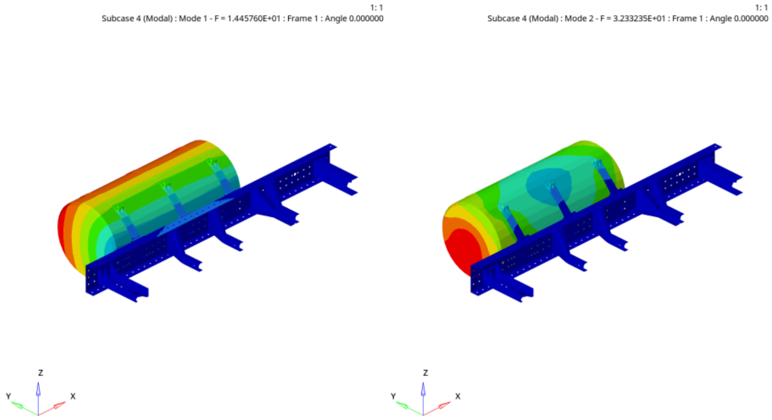

Figure 5.31: Modal frequencies of mode 1 and 2

The below plot shows the 3rd and 4th mode shape of the system.

f3 = 48.897 Hz

f4 = 50.227 Hz

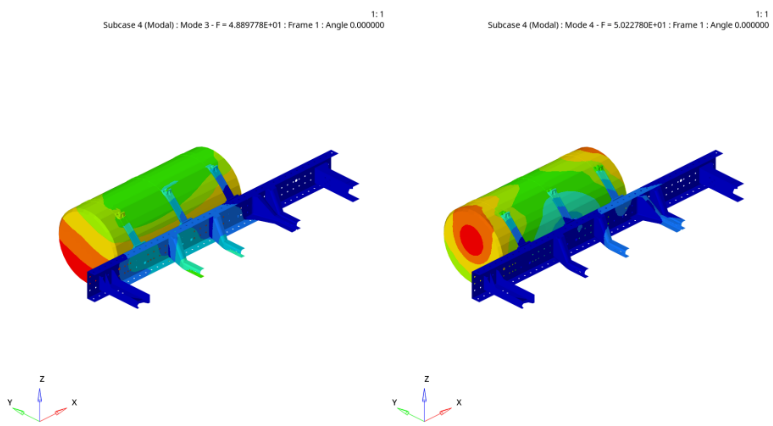

Figure 5.32: Modal frequencies of mode 3 and 4

The below plot shows the 5th and 6th mode shape of the system.

f5 = 55.162 Hz

f6 = 65.432 Hz

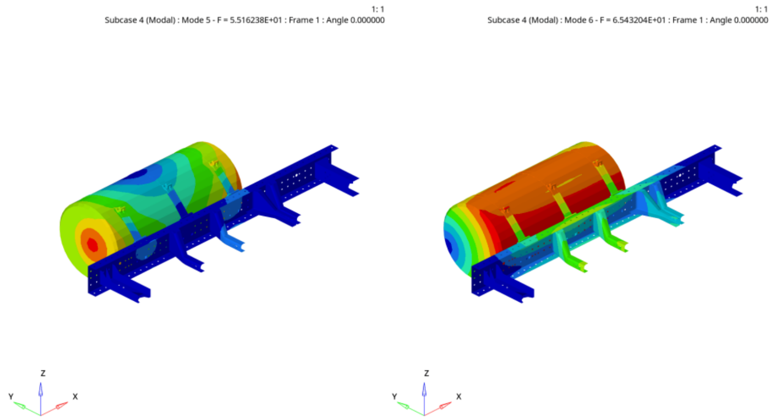

Figure 5.33: Modal frequencies of mode 5 and 6

The below plot shows the 7th and 8th mode shape of the system.

f7 = 66.091 Hz

f8 = 71.441 Hz

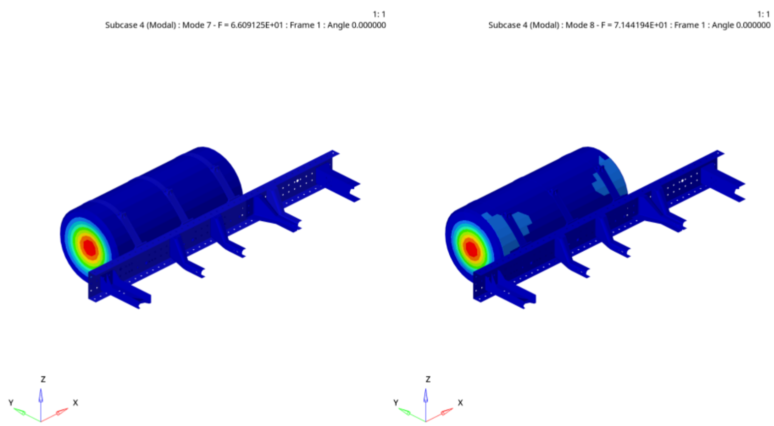

Figure 5.34: Modal frequencies of mode 7 and 8

The below plot shows the 9th and 10th mode shape of the system.

f9 = 122.753 Hz

f10 = 129.928 Hz

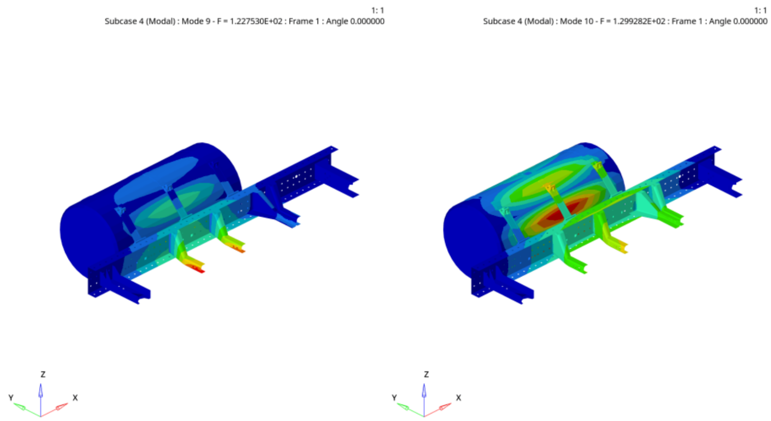

Figure 5.35: Modal frequencies of mode 9 and 10

The results obtained after the design changes are:

1. The stresses in all the three directions X, Y, Z are well within the fatigue limit of steel 207 MPa.

2. The least modal frequency of the first mode is 14.45 Hz. This frequency is within the standard values referred by Industries.

Chapter 6

Future Scope and Conclusion

6.1 Limitations

1. FEA is one of the Numerical methods, as there are few assumptions the results may not be 100% accurate to practical results. Hence, at the end stages of the design testing of prototypes is recommended.

2. As the materials are considered as linear material. But in reality no material is linear in nature. Also, If the material properties do not match the actual properties of the material, the analysis will produce incorrect results.

3. The quality and accuracy of the mesh can greatly affect the results of the FEA. There are some limitations with the number of elements and nodes, hence the features cannot be accurately captured. Therefore, results of the analysis may not be 100 percent accurate to real life.

6.2 Future Scope

1. The FEA model can be built having a better mesh quality by making a finer mesh. This may help to produce more accurate, closer to real life results.

2. Inclusion of other components which are mounted on the Long members and Cross members of the truck chassis may make the assembly stiffer and help to obtain better results.

6.3 Conclusion

1. Initially the fuel tank bracket was iterated by assigning three different materials, Cast Iron, Aluminum alloy and Steel. It was observed that steel has the better results considering the Displacements, Stresses on the bracket. Stresses: 156.37 MPa (Aluminum alloy) $<$ 161.38 MPa (Cast Iron) $<$ 161.62 MPa (Steel) in Z Direction (3g acceleration). Displacements: 10.06 mm (Steel) $<$ 17.01 mm (Cast Iron) $<$ 17.93 mm (Aluminum alloy) in Z Direction (3g acceleration).

2. The above results for Steel were better than the other two materials but were not acceptable as Modal frequencies of Steel is 12.04 Hz and is not close to 15 Hz or above.

3. To make the design fail safe, necessary changes, such as addition of another set of brackets and adding ribs at possible locations in two cross members, were made in the design of the fuel tank assembly which made the assembly stiffer.

4. After doing the required design changes, the stress values in all three directions 48.7 MPa (X), 40.72 MPa (Y), 143.56 MPa (Z) are within the fatigue limit of steel with is 207 MPa, the first modal frequency 14.45 Hz is close to 15 Hz which is an acceptable range according to industry practices for chassis mounted fuel tank.

5. So, it can be concluded that Steel can be used as a material for Fuel tank bracket if the required design changes are met.

References

[1] Madhu Ps and Venugopal T R, "Static Analysis, Design Modification and Modal Analysis of Structural Chassis Frame", IJERA, Volume-4, Issue-5, 2012.

[2] Shrivastava, S., Tiwari, R. and Sharma, S., 2019. Design and Analysis of Heavy Commercial Vehicle Chassis Through Material Optimization. International Journal of Engineering Trends and Technology (IJETT)–Volume, 67.

[3] Agrawal, M.S. and Razik, M., 2015. Finite element analysis of truck chassis frame. International Research Journal of Engineering and Technology, 2(3), pp.1949-1956.

[4] Ghorpade, U.S., Chavan, D.S., Patil, V. and Gaikwad, M., 2012. Finite element analysis and natural frequency optimization of engine bracket. International journal of mechanical and industrial engineering (IJMIE), 2(3), pp.1-6.

[5] Abhishek P, Hardeep S M, Sarvocch G, Anil L, "Finite Element Analysis of Fuel Tank Mounting Bracket", IRJET, Volume-5, Issue-8, 2018.

[6] Baseera Banushaik, Prasanna I., "Design and Optimization of FTV Fuel Tank Assembly by Finite Element Analysis", IJMETMR, Volume-4 Issue-2, 2015.

[7] Gajendra, G., Badrinarayan, K.S., Noor Ahmed, R. and Prakash, A.M., Design and Optimization of HTV Fuel Tank Assembly to Improve the Performance: A.

[8] Nitin Gokhale, Sanjay Deshpande, Sanju Bedekar, Anand Tithe "Practical Finite Element Analysis" Finite to Infinite 2008