STRUCTURAL VALIDATION AND OPTIMIZATION OF TRUCK CHASSIS USING FINITE ELEMENT ANALYSIS

By:

Jigar Patel

(21MMCC08)

DEPARTMENT OF MECHANICAL ENGINEERING

INSTITUTE OF TECHNOLOGY

NIRMA UNIVERSITY

AHMEDABAD-382481

MAY 2023

STRUCTURAL VALIDATION AND OPTIMIZATION OF TRUCK CHASSIS USING FINITE ELEMENT ANALYSIS

MAJOR PROJECT REPORT

Submitted in partial fulfilment of the requirements

For the degree of

MASTER OF TECHNOLOGY IN MECHANICAL ENGINEERING (CAD/CAM)

By:
Jigar Patel
(21MMCC08)

Guided by

Prof. Reena Trivedi

DEPARTMENT OF MECHANICAL ENGINEERING

INSTITUTE OF TECHNOLOGY

NIRMA UNIVERSITY

AHMEDABAD-382481

MAY 2023

Declaration

This is to certify that

- The Report comprises my original work towards the degree of Master of Technology in CAD/CAM Engineering at Nirma University and has not been submitted elsewhere for degree or diploma.
- Due acknowledgement has been made in the text to all other material used.

Jigar Patel

21MMCC08

Undertaking for Originality of the Work

I Jigar Patel (21MMCC08) give undertaking that the Major Project entitled ("Structural validation and optimization of truck chassis using finite element analysis ") submitted by me, towards the partial fulfillment of the requirements for the degree of Master of Technology in Mechanical Engineering (CAD/CAM) of Nirma University, Ahmedabad, is the original work carried out by me. I give assurance that no attempt of plagiarism has been made. I understand that in the event of any similarity found subsequently with any published work or any dissertation work elsewhere; it will result in severe disciplinary action.

Signature of Student

Date:

Place: Nirma University, Ahmedabad

CERTIFICATE

This is to certify that the Major Project Report entitled "**Structural Validation and Optimization of Truck Chassis using Finite Element Analysis.**" submitted by **Mr. Jigar Patel (21MMCC08)**, towards the partial fulfillment of the requirements for the award of Degree of **Master of Technology in Mechanical Engineering (CAD/CAM)** of School of Engineering, Nirma University, Ahmadabad is the record of work carried out by him under our supervision and guidance. In our opinion, the sub-mitted work has reached a level required for being accepted for examination. The result embodied in this major project, to the best of our knowledge, has not been submitted to any other University or Institution for award of any degree.

Date:

Prof. K M Patel
Head and Professor,
Department of Mechanical Engineering,
Institute of Technology and Engineering,
Nirma University, Ahmedabad

Industrial Guide:
Mr. Niraj Badhekha
Project Manager;
L&T Technology Services;

Prof. Reena Trivedi
Department of Mechanical Engineering,
Institute of Technology and Engineering,
Nirma University, Ahmedabad

ACKNOWLEDGEMENT

It gives me extremely privileged to express my gratitude to great many people who helped and supported me during completion of this project.

Apart from blessing of Gods and parents, I would like to express my deepest gratitude to both my internal and industrial guide Dr. Reena Trivedi (Professor, Mechanical Engineering, Institute of Technology, Nirma University) and Mr. Niraj Badhekha (Project Manager, L&T Technology and Services Ltd.), for guiding and correcting various actions and steps with attention and care. They have presented a good amount of attention throughout the project and make necessary correction as and when needed.

I am very much indebted to Dr. R. N. Patel (Additional Director General, Institute of Technology, Nirma University) for their valuable support and guidance. I am also very thankful to Dr.B. A. Modi (PG-coordinator, CAD /CAM Engineering, Institute of Technology, Nirma University) for providing support for project.

ABSTRACT

The chassis of a truck is a crucial component and it acts as a framework for the body and other components of the truck. Additionally, it must be sufficiently robust to handle pressures such as shock, twisting, vibration, and others. Having sufficient bending stiffness for improved handling characteristics is just as vital as strength when designing a chassis. Deflection, maximum stress, and maximum equilateral stress are crucial design factors for the chassis.

The work done to optimise the truck chassis is what this project. To determine stiffness and determine how to minimise weight while increasing stiffness, sub-modelling and static analysis is performed. Therefore, using real parameters, a correct finite element model of a truck chassis must be created. Applying appropriate loading and boundary constraints to the truck chassis model.

For Finite Element Analysis, pre-processing is done on the Hypermesh 21.1 and analysis is carried out in Optistruct package.

Key words: Finite element analysis, Optimization, Truck Chassis.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1 Introduction

1.1 Introduction to Chassis

Chassis is the vehicle's main support structure, also known as the 'Frame.' It bears all the stresses on the vehicle in both static and dynamic conditions. In a vehicle, it is analogous to the skeleton of a living organism. The origin of the word Chassis lies in the French language. For all class like two-wheeler, car, truck, every vehicle has a chassis-frame. However, its form, changes with the vehicle type.[1]

The Chassis has the following functions it:

1. Supports or carries the load of the vehicle body.
2. Provide the space and mounting location for various aggregates of vehicle.
3. Supports the weight of various systems of the vehicle such as engine, transmission etc.
4. Supports a load of passengers as well as the luggage.
5. Withstands the stresses arising due to bad road conditions.
6. Withstands stresses during braking and acceleration of the vehicle.

1.1.1 Types of Chassis

1. **Backbone Chassis:** The backbone of a car chassis must consist of at least one substantial central component connecting the front and rear of the frame. In the backbone chassis, the driveshaft is protected from damage by a rectangular cross-section cylindrical tube & which also makes it difficult to repair. It is relatively expensive to manufacture the backbone chassis.
   Such development enables a far superior, superior, higher, stronger, improved, and superior attachment of the axles to the ground. This increases the car's stability. Even in the unlikely event that the vehicle frequently bumps into the ground when off-road, the drive shaft is quite safe. The unbending nature is additionally higher and can withstand a lot of pressure.
2. **Monocoque Chassis:** Lightweight and compact development permit for the utilization indeed within the littlest of cars. The dealing with qualities is greatly upgraded since the complete vehicle is one compact structure and energetic powers are effortlessly exchanged out to preserve soundness around the corners.

Fig 1. 1 1 Monocoque Chassis [1]

Being used extensively, the fetched fabricating is very moo, which is why it may well be utilized for any car. The security viewpoint of the car and travellers is improved since of a cage-like development. A part of the fold zone can be included in the construction.

3. **Ladder Chassis:** A ladder frame is among the most conventional types. Its construction resembles that of a ladder and consists of two long beams supported by two short beams. Trucks, buses, and light commercial vehicles use them because they are easy to manufacture.

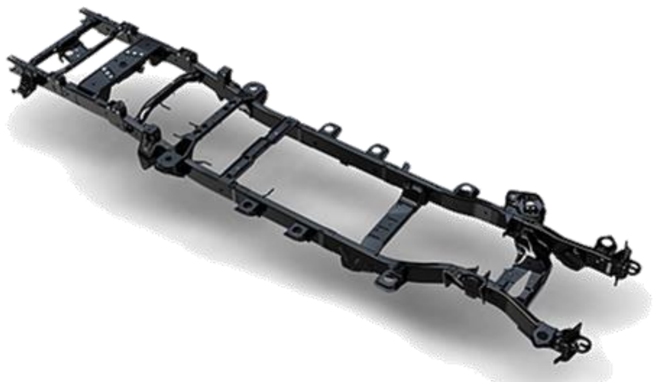

Fig 1. 2 2 Ladder Chassis [1]

Ladder Chassis development is all-inclusive, in the sense that different sorts of bodies can be joined to one kind of stepping stool frame. Being independently mobile from the body, the off-road capabilities are improved, and as the suspension seems to travel encourage and nearly autonomously of the body. These chassis are fantastically spry and flexible.

4. **Frameless Chassis:** This style of chassis, also known as a frameless chassis or uni-body chassis, does not have a ladder frame and instead uses the body as a frame. It also supports all of the vehicle's elements and components. In most modern cars, these are employed in almost the same way as conventional chassis.
5. **Hybrid Chassis:** Hybrid chassis is a chassis that take elements from different types to make the most suitable version for the chassis of that particular car. [2]

1.2 Introduction to Sub modelling

Submodeling is a finite element technique used to get more accurate results in a region of your model.

Submodeling is also known as the cut-boundary displacement method or the specified boundary displacement method. The cut boundary is the boundary of the submodel which represents a cut through the coarse model. Displacements calculated on the cut boundary of the coarse model are specified as boundary conditions for the submodel.

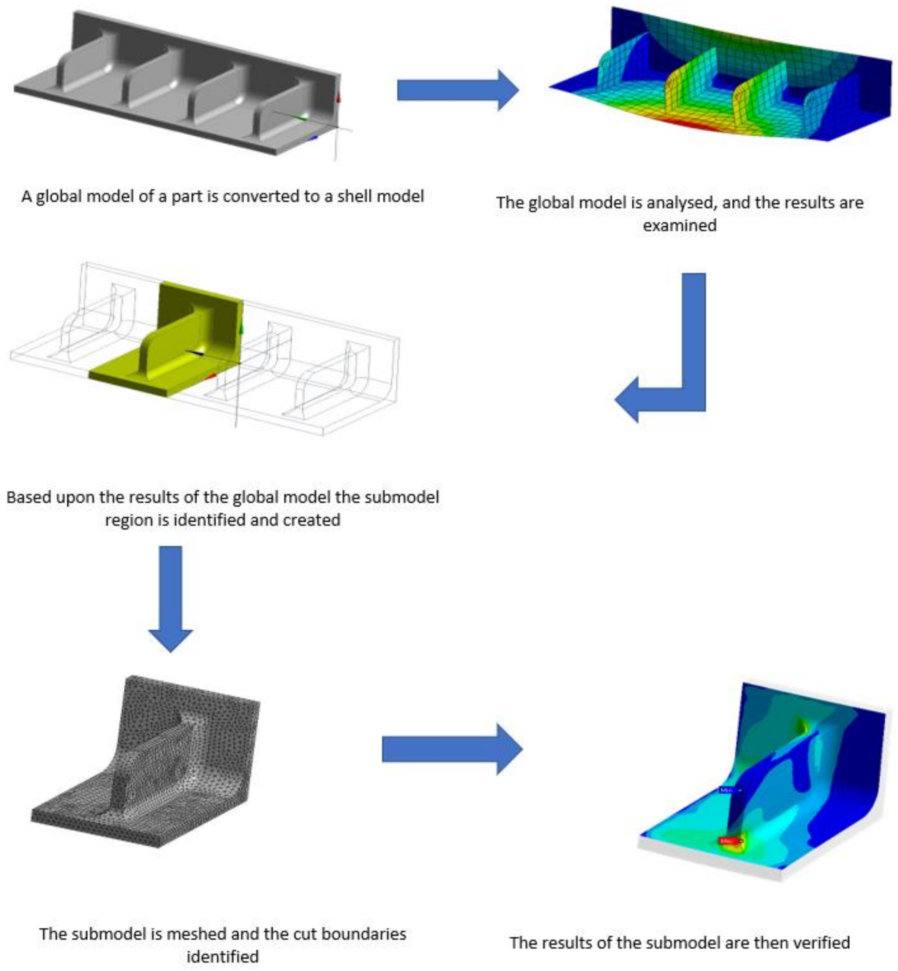

Fig 1. 3 Sub Modelling Example [3]

Submodeling is based on St. Venant's principle, which states that if an actual distribution of forces is replaced by a statically equivalent system, the distribution of stress and strain is altered only near the regions of load application. This implies that stress concentration effects are localized around the concentration; therefore, if the boundaries of the submodel are far enough away from the stress concentration, reasonably accurate results can be calculated in the submodel.[3]

1.3 Introduction to Optimization

1.3.1 Types of Optimization

1. **Topology Optimization**: Topology optimization (TO) is a mathematical method that optimises material layout within a given design space, for a given set of loads, boundary conditions and constraints with the goal of maximising the performance of the system.

The conventional topology optimization formulation uses a finite element method (FEM) to evaluate the design performance. The design is optimised using either gradient-based mathematical programming techniques such as the optimality criteria algorithm and the method of moving asymptotes or non-gradient-based algorithms such as genetic algorithms.

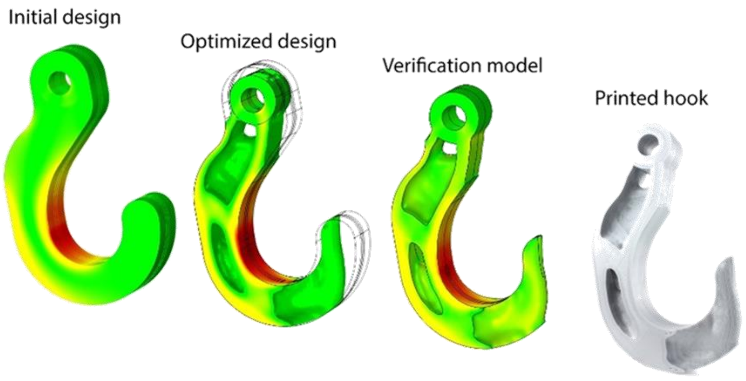

Fig 1. 4 Topology Optimization Example [4]

2. **Design Optimization:** Design optimization is an engineering design methodology using a mathematical formulation of a design problem to support selection of the optimal design among many alternatives.
   It involves the following stages:
   1) Variables: Describe the design alternatives
   2) Objective: Elected functional combination of variables (to be maximized or minimized)
   3) Constraints: Combination of Variables expressed as equalities or inequalities that must be satisfied for any acceptable design alternative
   4) Feasibility: Values for set of variables that satisfies all constraints and minimizes/maximizes Objective.

1.4 Introduction to Hypermesh

1.4.1 Hypermesh

Altair HyperMesh is a pre-processor for high fidelity modeling. With automatic and semi-automatic shell, tetra, and hexa meshing capabilities, HyperMesh simplifies the modelling process of complex geometries. It also accommodates a wide range of CAD and solver interfaces. One of HyperMesh's key strengths is its quick generation of high-quality mesh.

Altair HyperVie is a complete post-processing and visualization environment for finite element analysis, CFD and multi-body system data.

Hypermesh is a CAE software which comes in after a model has been developed in any CAD software. One should know it is an integral part of PLM (Product life cycle management) where the focus is to prepare the model to see how should it behave in a real life environment or simulation in other words.

1.5 About Company

L&T Technology Services (LTTS) is an engineering services company. It operates as a subsidiary of the conglomerate Larsen & Toubro (L&T). L&T Technology Services was founded in 2006 as L&T Integrated Engineering Services.

L&T Technology Services Limited (LTTS) is a global leader in Engineering and R&D (ER&D) services. With 868 patents filed for 57 of the Global Top 100 ER&D spenders, LTTS lives and breathes engineering. Our innovations speak for themselves – World's 1st Autonomous Welding Robot, Solar 'Connectivity' Drone, and the Smartest Campus in the World, to name a few.

LTTS' expertise in engineering design, product development, smart manufacturing, and digitalization touches every area of human lives - from the moment one wakes up till the time one goes to bed. With 89 Innovation and R&D design centers globally, we specialize in disruptive technology spaces such as 5G, Artificial Intelligence, Collaborative Robots, Digital Factory, and Autonomous Transport.

LTTS is a publicly listed subsidiary of Larsen & Toubro Limited, the $21 billion Indian conglomerate operating in over 30 countries.

It majorly provides ER&D services in the field of Manufacturing, Product, Operations and Consultancy. [5]

1.5.1 About CAE Department

At LTTS, product engineering is a forte. They have teams of specialists that can assist you in every step of your product development journey – starting with design & prototyping to VLSI, software, hardware and security engineering.

I am part of Team CAE in Mechanical Design sector. Where a wide range of services had been provided starting from

- STATIC ANALYSIS
- IMPLICIT DYNAMIC ANALYSIS
- EXPLICIT DYNAMIC ANALYSIS
- FATIGUE/ DURABILITY
- NVH ANALYSIS
- THERMAL ANALYSIS
- MULTI-BODY DYNAMIC SIMULATION

1.6 Project Statement

- Structural validation and optimization of truck chassis using finite element analysis
- Study the existing FE model and understand modelling techniques.

1.7 Objectives of project

- To Learn basic truck frame layout and Familiarize with assembly techniques used.
- To Learn Optistruct deck, simulation and troubleshooting.
- To Learn post processing, bolt evaluation, and how to summarize findings and write conclusions and recommendations.

CHAPTER 2 Literature Review

This chapter includes the description of number of research paper to optimize the existing truck chassis. Research paper is related to the major element of the chassis, design and optimization of the chassis. Research paper related to structural analysis has been studied. So, this chapter discusses the following paper related to mentions topics.

2.1 Chassis Definition

A chassis is the load-bearing framework of an artificial object, which structurally supports the object in its construction and role. An example of a chassis is a vehicle frame, the underpart of a motor vehicle, on which the body is mounted; if the running gear such as wheels and transmission, and sometimes even the driver's seat, are included, then the assembly is described as a rolling chassis.

2.2 Literature reviewed

Naveen G build a chassis of a car using CATIA V5 software. Meshing of the model is carried out in the hypermesh software where the element type is set to be SHELL63. Total 5793 elements are created with the node count of 5932. Material used was steel. They have performed self-weight analysis, bending analysis and torsion analysis.[6]

S. Mathivanan et al used pickup truck- 2015 Chevrolet Silverado for the analysis purpose where so many sub-assemblies like powertrain, nuts, bolts, name plate which is not considered foe the analysis because these parts do not add any significance to the study. They have imported the model in the ANSYS software. They have tested the crash analysis for different sections and different materials. After the comparision they have concluded that channel frame section is reduced in weight and safe under static crash analysis for optimum load and speed condition.[7]

M. S. Agrawal and Md. Razik used TATA 1612 chassis to calculate the stress of given boundary and loading conditions. They have calculated the pressure acting on the parts manually. Solid model of chassis is made through CATIA software and the material used is mild steel which is containing 0.05-0.3% of carbon. First they have performed the static analysis and check the total deformation and directional deformation. Modal analysis is performed and check different mode shapes and regarding that they have changed the thickness of the C – section wherever less load is acting and where there are less deformation.[8]

M D Vijayakumar et al has performed the stress analysis of the static force loaded heavy duty truck chassis. They have prepared the Tata 407 fire truck model in the PRO-E software. Material used is steel and carbon fibre. ANSYS workbench is used to mesh the model and for the analysis to check the equivalent stress, extreme elastic strain and whole deformation. Modal analysis is also performed for both the materials from that they have concluded that carbon fibre material can reduce the 80% of the weight.[9]

D A Bircan and A Yasar has made the truck chassis using the CATIA software. They have used the Taguchi design procedure for optimization.

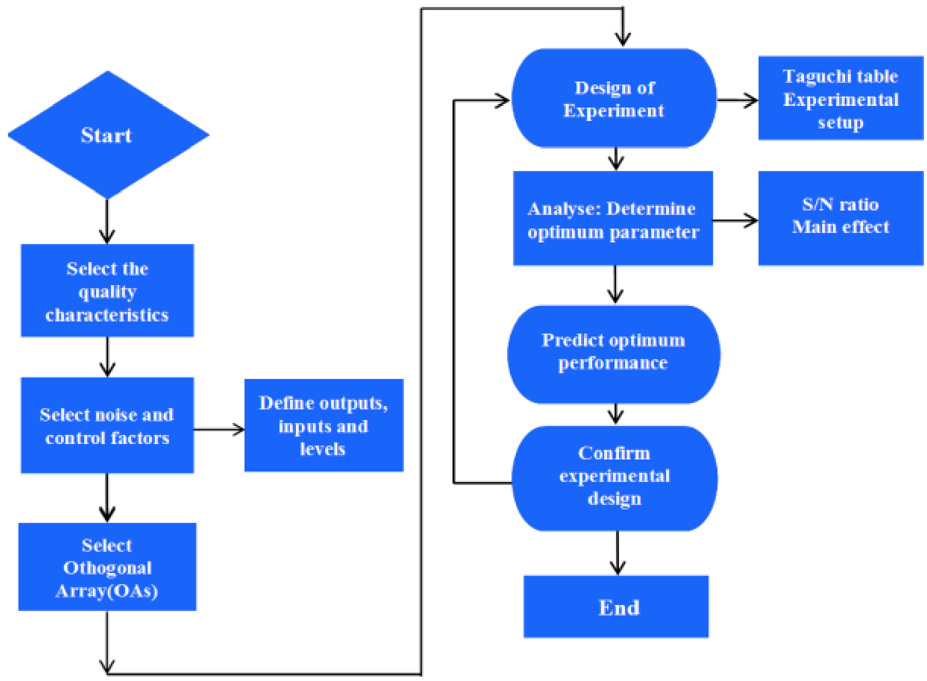

Fig 2. 1 Proposed Taguchi design procedure [5]

The material used are AISI 1006 Steel Cold drawn and ASTM Class 30, Gray Cast Iron. Profile used was U and I where U profile is having the supports whereas I does not have any supports. 100 KN load is applied on the chassis total deformation is calculated for both the chassis and analysed the results.[10]

O Kurdi et al have used Hino modle truck chassis. The material of chassis is ASTM Low Alloy Steel A 710 C . The meshed truck chassis model has 101466 elements and 37697 nodes. The element shape is tetrahedral and element type is 3D stress. The location of maximum Von Misses stress is at opening of chassis which is contacted with bolt. [11]

C. Karaoglu and N. S. Kuralay investigated stress analysis of a truck chassis with riveted joints using FEM. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally. If the thickness change is not possible, increasing the connection plate length may be a good alternative.[12]

B. R Naik and C. S have objective to analyse an automobile chassis for a 10 tonne vehicle. The modelling is done using Pro-E, and analysis is done using ANSYS. The overhangs of the chassis are calculated for the stresses and deflections analytically and are compared with the results obtained with the analysis software. Modal Analysis is also done to find the natural frequency of the chassis and seen that it is above than its excitation frequency. The Theoretical calculations and FE analysis results are compared and it is observed that they are within the material properties. [13]

C Karaoglu and N. S Kuralay has studied that the stress analysis of a truck chassis with riveted joints. They have derived the loading conditions acting on the chassis.

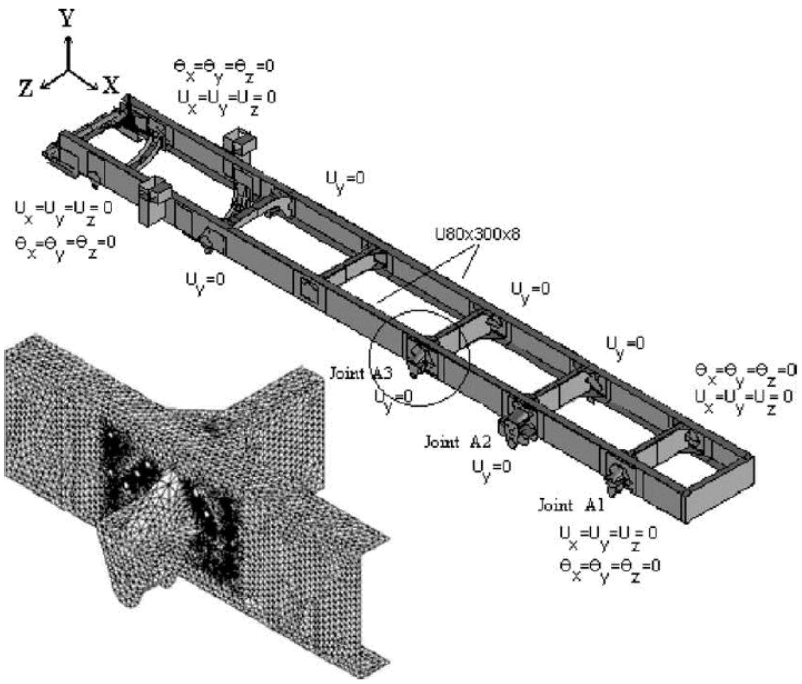

Fig 2. 2 Truck Chassis model [9]

They have made 3 types of connection where the long and cross members are in mating condition. They have analysed the von-mises with different thickness of plates and concluded that changing the plate thickness and adding local plates where local stress is occurring. [14]

R. B. Nallamothu have analysed the light weight chassis for different composite materials. Where they have used steel, E-glass and aluminium alloy materials. The behaviour for steel and aluminium alloy is Isotropic while E-glass is orthotropic. After analysing the results, they have concluded that E-glass is best suitable for the chassis material as it has less weight but strength is very high as compared to other 2 materials.[15]

Patel et al have investigated and optimised a chassis design for Weight reduction of TATA 2516TC chassis frame using Pro-Mechanica. Thy first find out the assembly weight, maximum stress, strain and displacement for the existing section of chassis (C, I and Box sections) by using ANSYS Software after then they modified the dimensions of existing C-sections and again find all and concluded that the existing “C” sections is better than all the sections with respect to the Stress, Displacement, Strain and Shear stress except the weight. For the weight consideration modified “C” section has less weight than the all sections which are studying in this paper. Finally, By the use of modified “C” section, 105.50 Kg (11 percent) weight is saved per chassis assembly and in

CHAPTER 3 Methodology

Here, for optimization below steps are used where baseline run is performed of the global model and then on the sub-model topology optimization is done.

3.1 Steps involved for Optimization

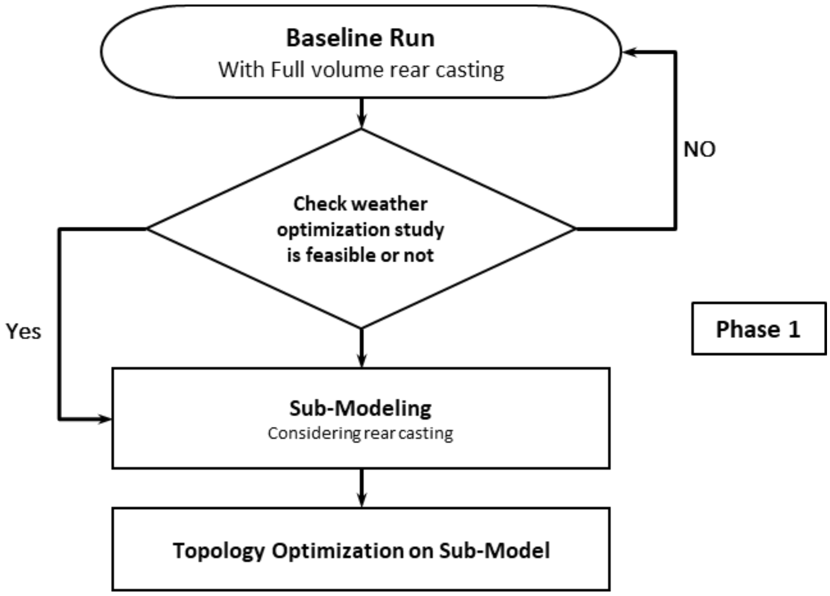

Fig 3. 1 Methodology Flowchart

- Firstly we have to perform Baseline run with the full assembly and loadcases.
- After that sub model run is performed forces and moments applied at the end nodes and validating with the global model results.
- Perform the analysis of sub-model with loading and boundary conditions.
- Optimization run is performed and get the optimized design.

CHAPTER 4 Design of Chassis

4.1 Design of Chassis

4.1.1 Chassis Specification

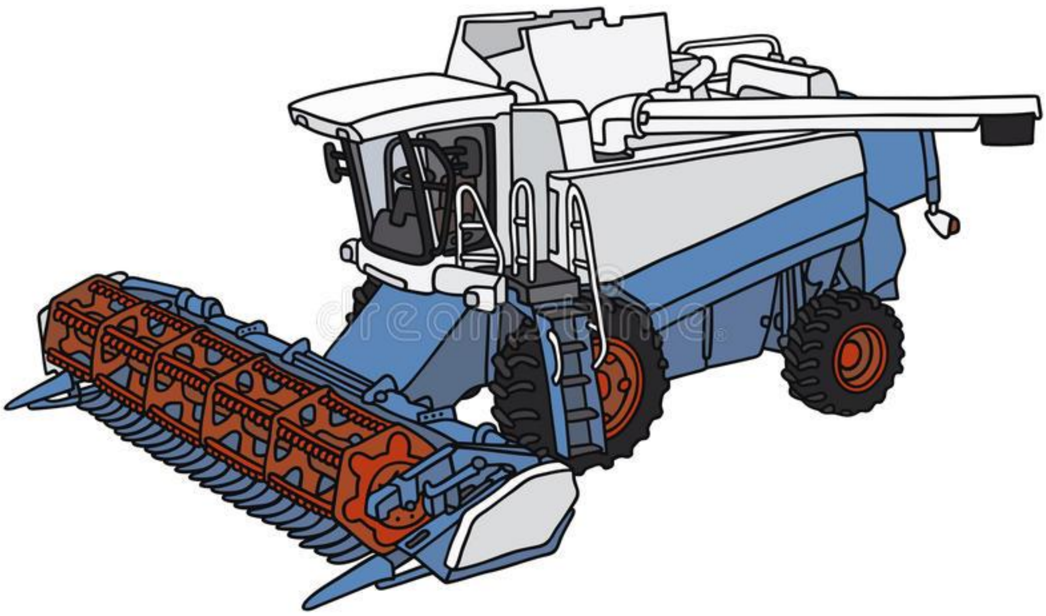

Fig 4. 1 Combine Harvester [17]

The chassis in use here comes from a combine harvester. The most advantageous multi-crop harvester in its class is the new Ideal for harvesting minor crops like wheat or barley, specialised crops like rice or sunflower, or huge grains like corn, soybeans, and a variety of pulse crops. The new five strawwalker model comes equipped with a top-of-the-line 1,300 mm wide and 607 mm diameter threshing drum, a double cascaded cleaning shoe with a pre-sieve, a 3500 litre grain tank with an enclosed 3.85 m unloading auger, and the ability to be linked with the 15 ft High-Capacity header. The vehicle also offers the segment's lowest total cost of ownership. The capacity is the highest in its segment as a result.

The chassis which we have used is consist of many sub assemblies named as Adapter, Cradle, Housing etc.

The whole feeder assembly is about 1500 kg in weight. It consist of 180 kg of cradle sub assembly. Cradle is basically the front portion of the feeder where the crop is collected. After that there is another sub assembly called Adapter is there.

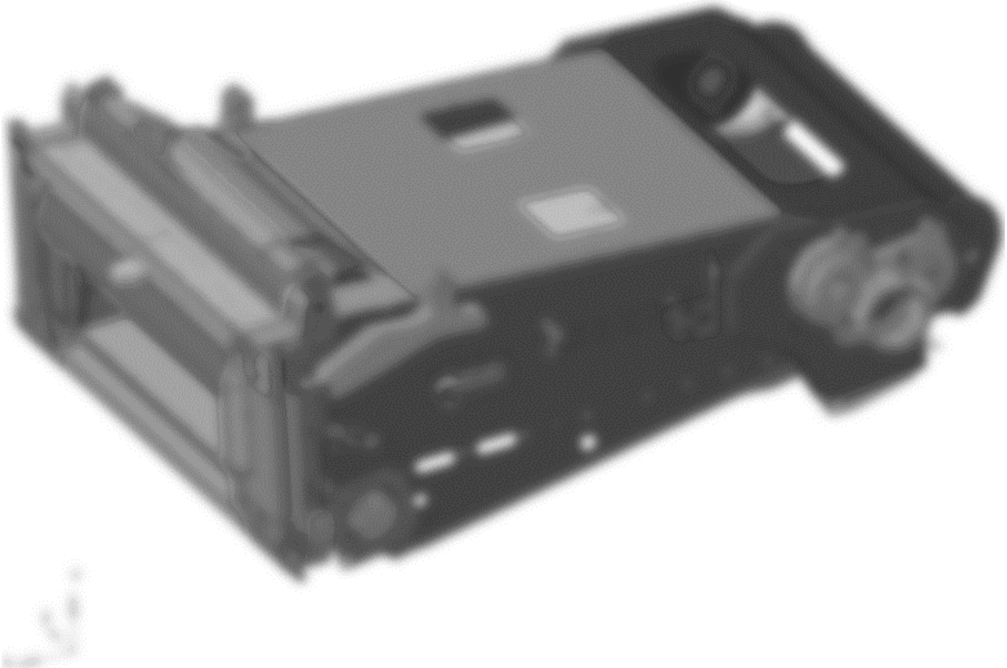

Fig 4. 2 Feeder Assembly

The main purpose of Adapter is to hold the cradle and conecction between the cradle and housing. The weight of the Adapter assembly is around 330 Kgs.

Fig 4. 3 Cradle

Fig 4. 4 Adapter

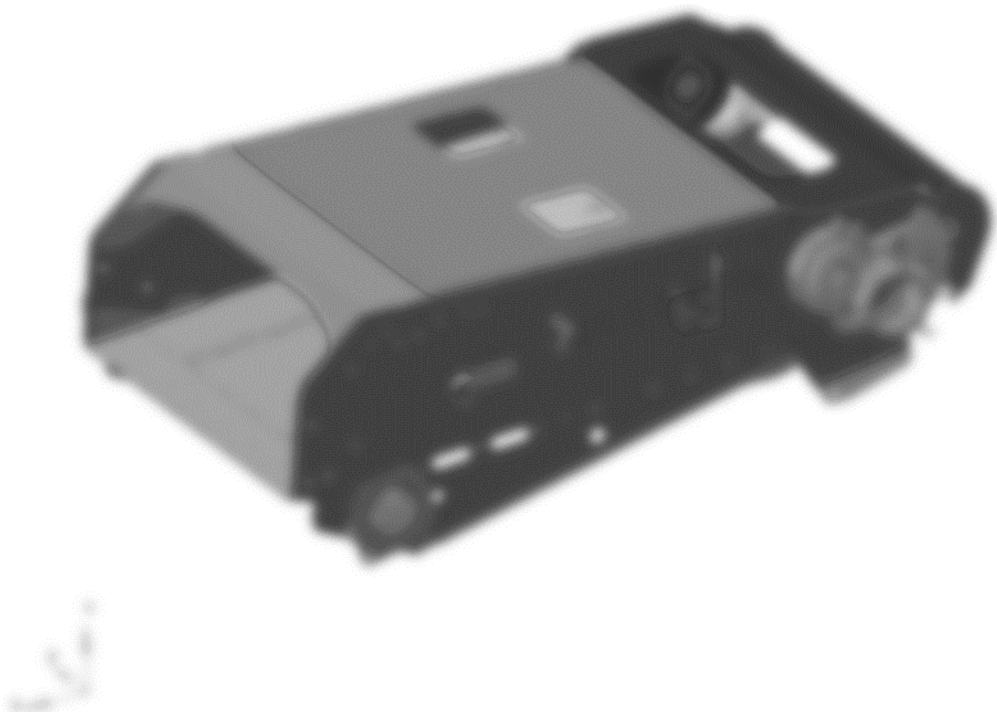

Fig 4. 5 Housing

Housing is the main part of the assembly. It consist of the conveyor belt through which the crop will get collected. Also the gear box is also mounted on this

Housing to circulate the conveyor belt. The total weight of Housing is around 620 Kgs. The thickness varies from 2.5mm to 19mm in some areas.

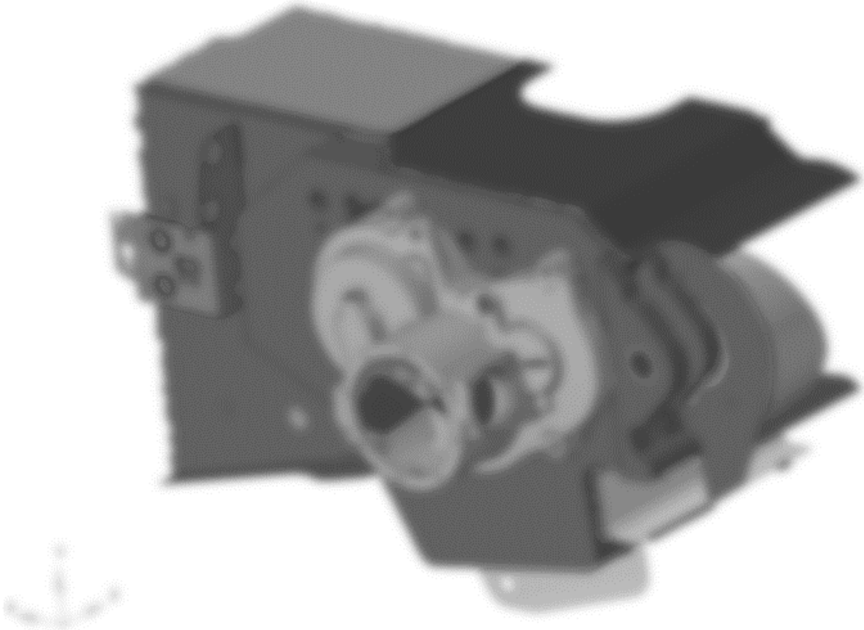

Fig 4. 6 Sub-Model

4.1.2 Geometry Clean-up

Designers have different priorities while creating CAD geometry than analysts who are seeking to use the data. Usually, a single plane surface is divided into smaller patches, each of which is a distinct mathematical face. And often, there will be spaces, overlaps, or other arrangement issues at the junction of two sides.

In geometry cleanup, analysts combine several faces onto a single smooth surface to make the geometry more appropriate for meshing. This makes it possible to create all of the elements simultaneously across the region and does away with the need for artificial or unintentional edges in the final mesh.

4.2 Pre-Processing

4.2.1 Types of Mesh

1. **Hex Mesh**: For casting we have used First order Hex mesh. This casting is basically to mount the gear box through bolted conecction on the Housing assembly. CHEXA

elements are used to model this casting. Around 1112000-1112200 CHEXA elements are used.

Fig 4. 7 HEX Mesh

2. **Tetra Mesh**: More complex parts can be meshed with the Second order tetra element. Here we have meshed the gear box and also some clamps with second order tetra element. The type of second order tetra elemet is CTETRA. 606600 CTETRA used to crearte this casting model.

Fig 4. 8 Tetra Mesh of Gear Box

3. **Shell Mesh**: Sheet metal parts or the parts where the thickness is constant is modelled with 2D Quad elements. Here in optistruct quad elements is named as CQUAD4. It utilizes 5 integration points. PSHELL property is assigned to this type of mesh. Proper thickness is assigned to this type of mesh. 298752 CQUAD elements are used to create this type of mesh.

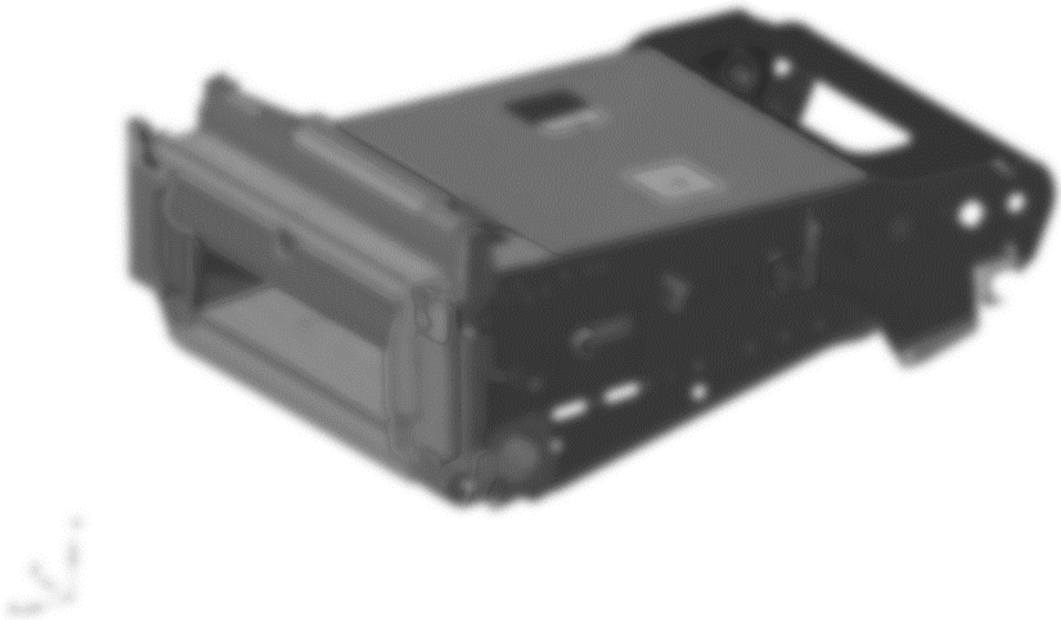

Fig 4. 9 Shell Mesh

4.3 Connectors

In some case the weld region is modelled with node to node conecction and in some cases it is modelled with 1D rigids. A rigid element is an element created in a space between two nodes of a model where a rigid connection is desired.

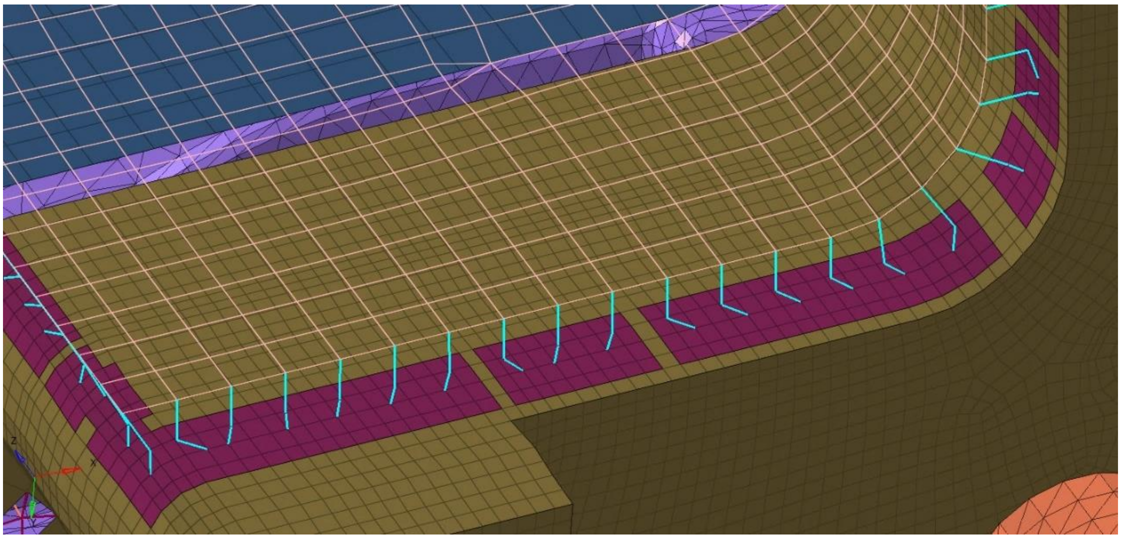

Fig 4. 10 Weld Connection

Rigid elements are element config 5 and are displayed as a line between two nodes with the letter R written at the centroid of the element.

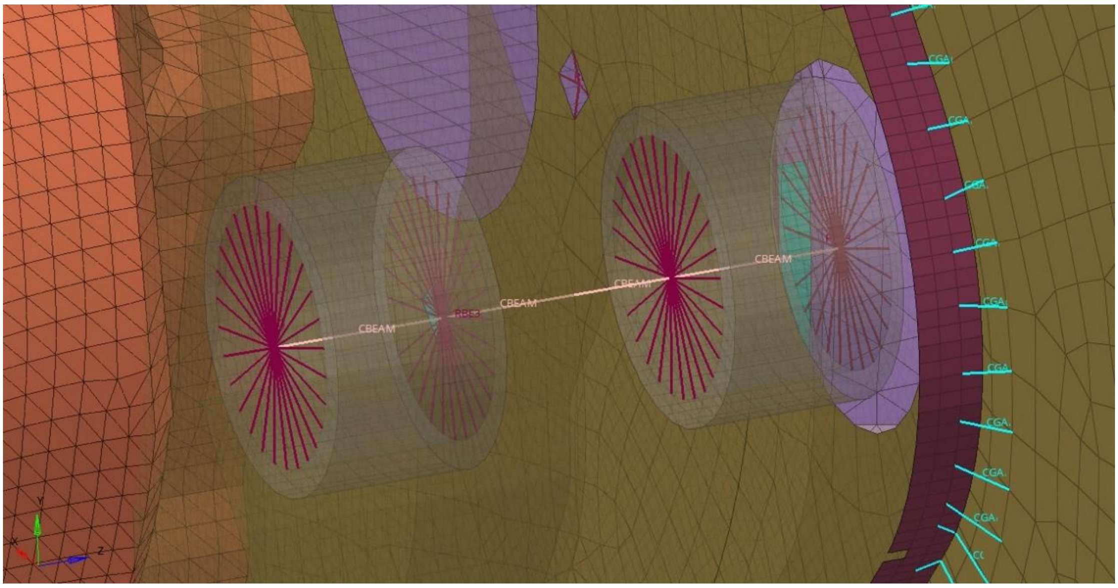

Fig 4. 11 Bolt Connection

Bolt conecctions are made with combination of RBE3 and CBEAM elements. RBE3's are interpolation constraint elements, where the motion of a dependent node is defined as the weighted average of the motions of a group of independent nodes. The weighting factors and dependent degrees of freedom can be manually entered or computed automatically using the geometry. RBE3 elements are element config 56 and are displayed as lines between the dependent node and the independent node(s) with RBE3 displayed at the dependent node of the element.

RBE3's are typically used to distribute loads applied on the dependent node amongst the selected independent nodes.

There is a cross section applied to the CBEAM element so it can represent as a bolt shank portion.

Also the front portion of the feeder assembly is modelled with the CBEAM element. We have to directly transfer the load to the cradle and adapter sub assembly through this front feeder head.

Fig 4. 12 CBEAM Elements

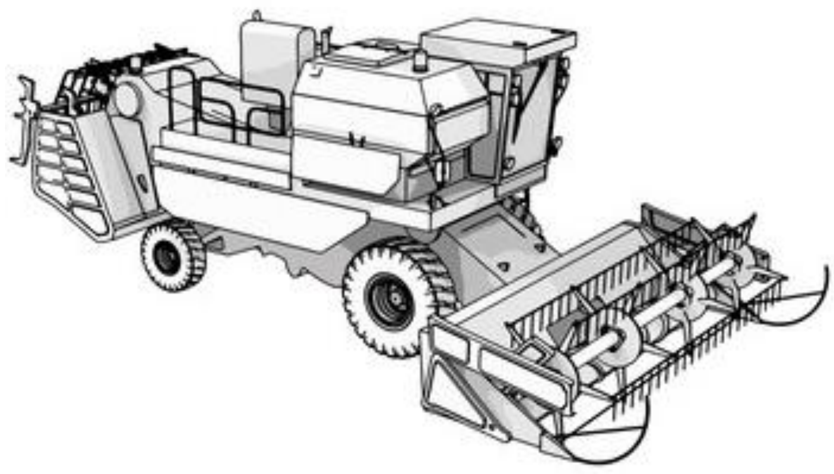

Fig 4. 13 Combine Harvester Sketch

4.4 Material Used

The material used for the construction of the chassis is usually carbon steel and Aluminium. Steel found it's way to cars that demands high durability and reliability.

Steel		
Mechanical Properties	**Value**	**Unit**
Young's Modulus	210	GPa
Poisson's Ratio	0.3	-
Density	7900	Kg/m^3

Table 4. 1 Steel Mechanical Properties

Aluminium		
Mechanical Properties	**Value**	**Unit**
Young's Modulus	703	GPa
Poisson's Ratio	0.34	-
Density	2660	Kg/m^3

Table 4. 2 Aluminium Mechanical Properties

4.5 Boundary and Loading Conditions

4.5.1 Boundary conditions

Boundary condition is basically to defined the constraints of the Degrees of Freedom (DOFs). Here on the left side it is fixed by 1,2 and 3 it means it does not translate in X,Y and Z direction. Whereas on the right side it is fixed in 1 and 2 only, means it can translate in Z direction and also on both side it can rotate in X,Y and Z direction. On the upper side it is fixed on 1,2 and 3.

All the bouandry conditions are defined where the housing is mounted on the full assembly.

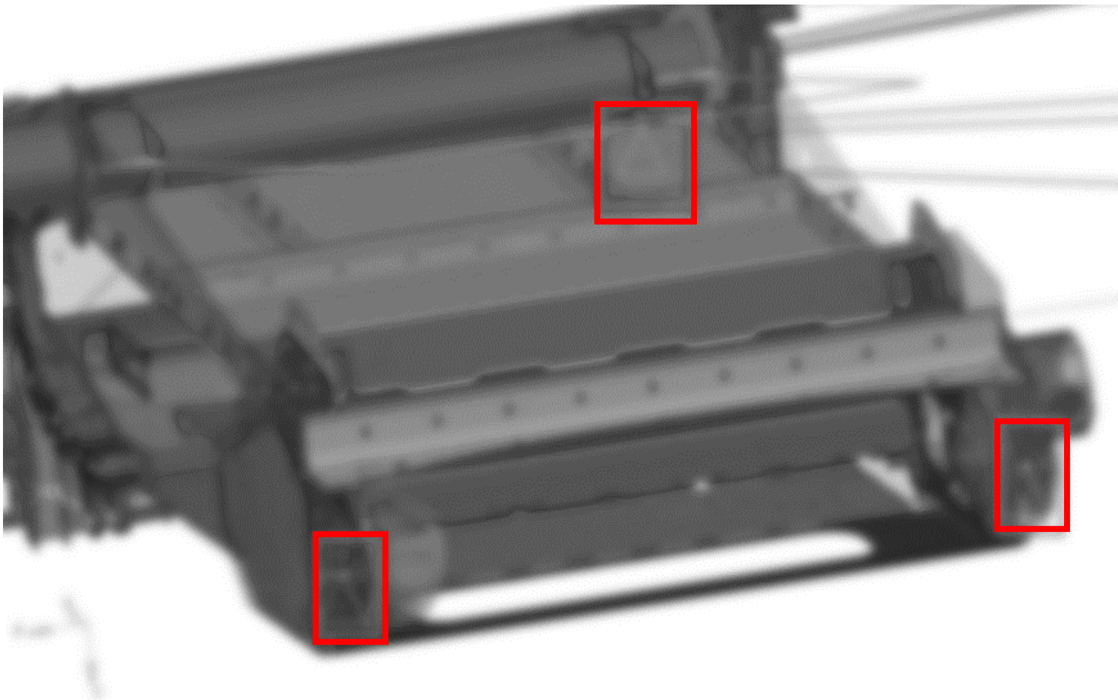

Fig 4. 14 Boundary Condition

4.5.2 Loading conditions

The chassis is having mainly five types of loading conditions. First one is 2G downward direction, second is moment of around 1.2e+08 in +X and +Y direction. Third one is of 0.2 times in Right direction. The fourth and fifth one is same as the second and third loadcase but in reverse direction respectively. Gear Box torque of 1222 Nm in +Z direction is applied in all the five load cases respectively.

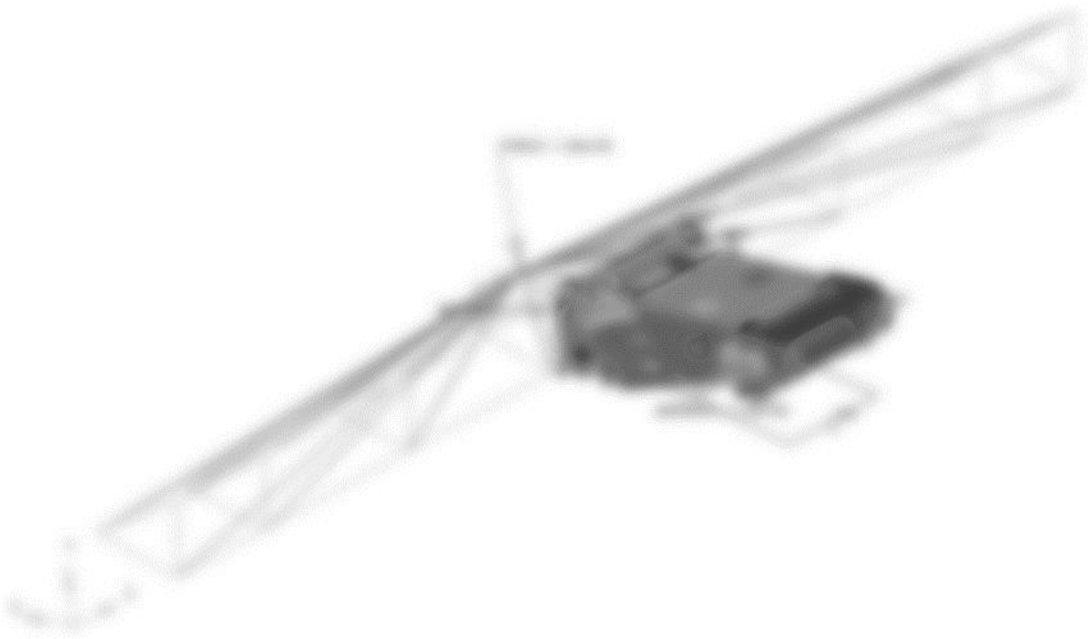

Fig 4. 15 Load Case 1: 2G Downward Direction

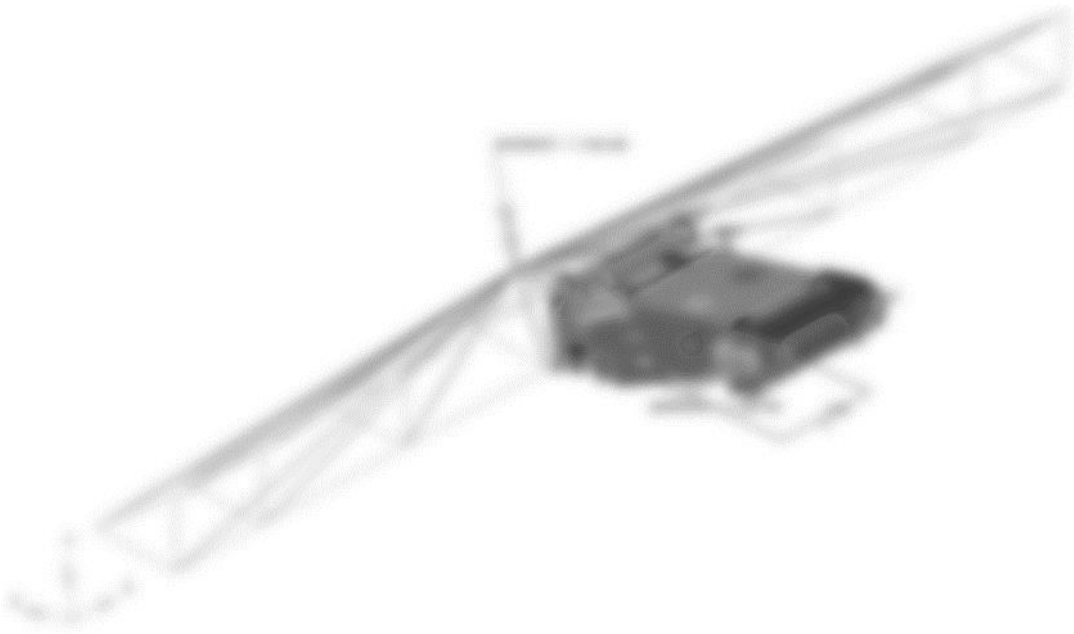

Fig 4. 16 Load Case 2: MX+ MY

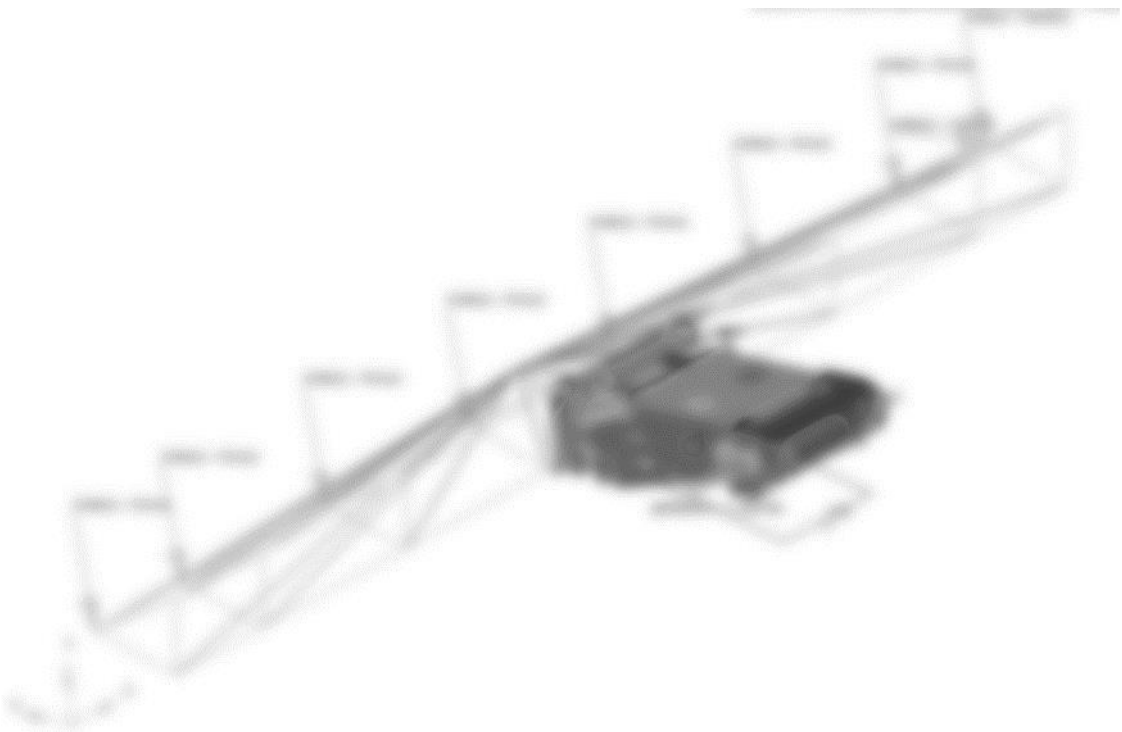

Fig 4. 17 Load Case 3: 0.2Right

Fig 4. 18 Load Case 4: -MX -MY

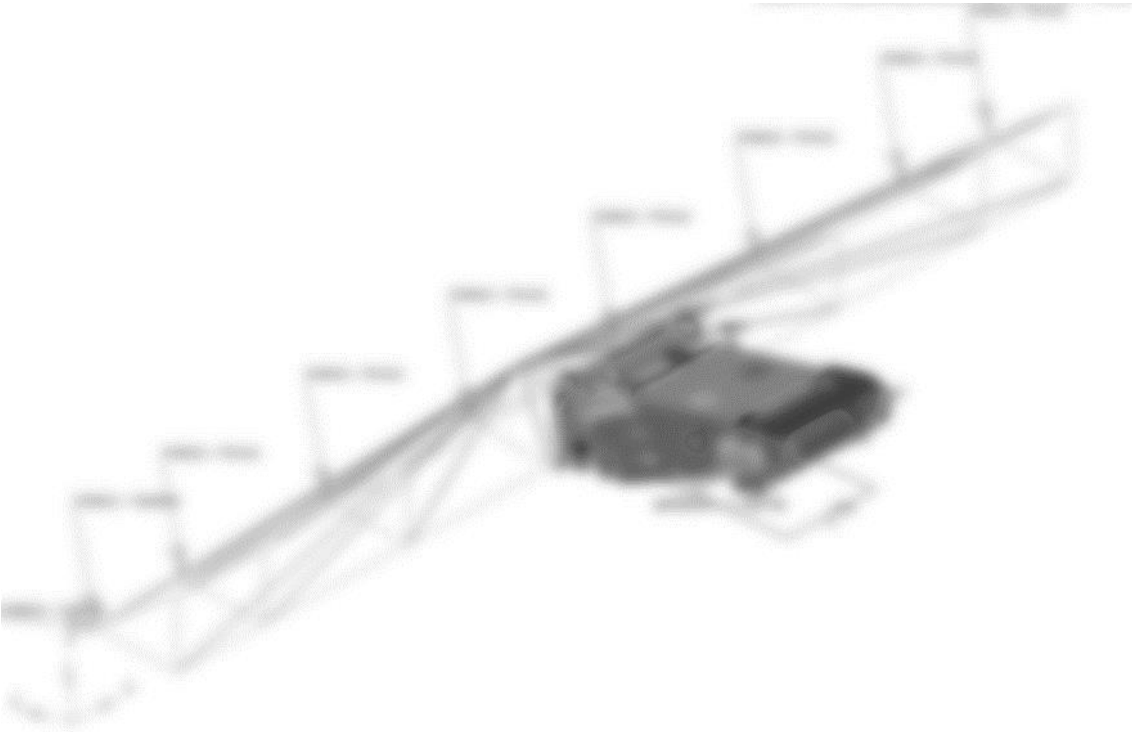

Fig 4. 19 Load Case 5: 0.2Left

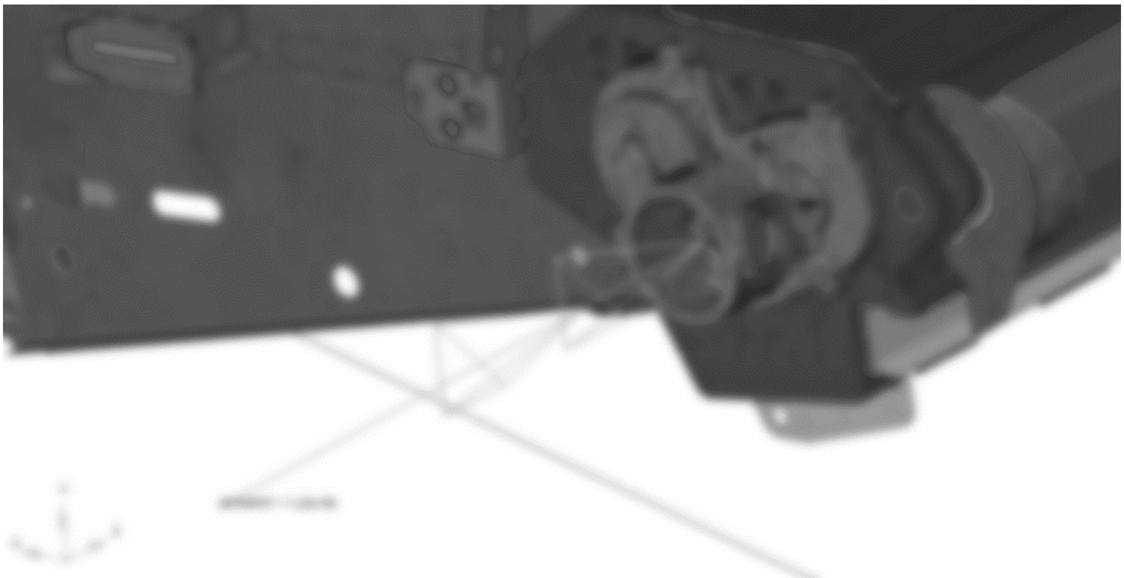

Fig 4. 20 Gear Box Torque

4.6 Analysis

Here submodelling is done to extract the forces and moments on the end nodes of the submodel set. Firstly element set is created at interested region to be analyzed for sub-model detailed analysis. For submodelling output request is created for GPFORCE for all end nodes in OUTPUT2 (.OP2) format. Free body diagram(FBD) forces are calculated for sub model end nodes. Exporting this FBD forces and assigning this FBD forces to the sub model we have created and rerunning the analysis.

The analysis is carried out in the Optistruct module. Where we have to import the .fem file and it will converge the model. After converging it will show the results as per the required output selected.

Optimization analysis is carried out by assigning the design and non-design portion to the chassis. Also assigning the stress constraint to the design portion of the chassis. Configuration is set as topology. Objective is to minimize the volume.

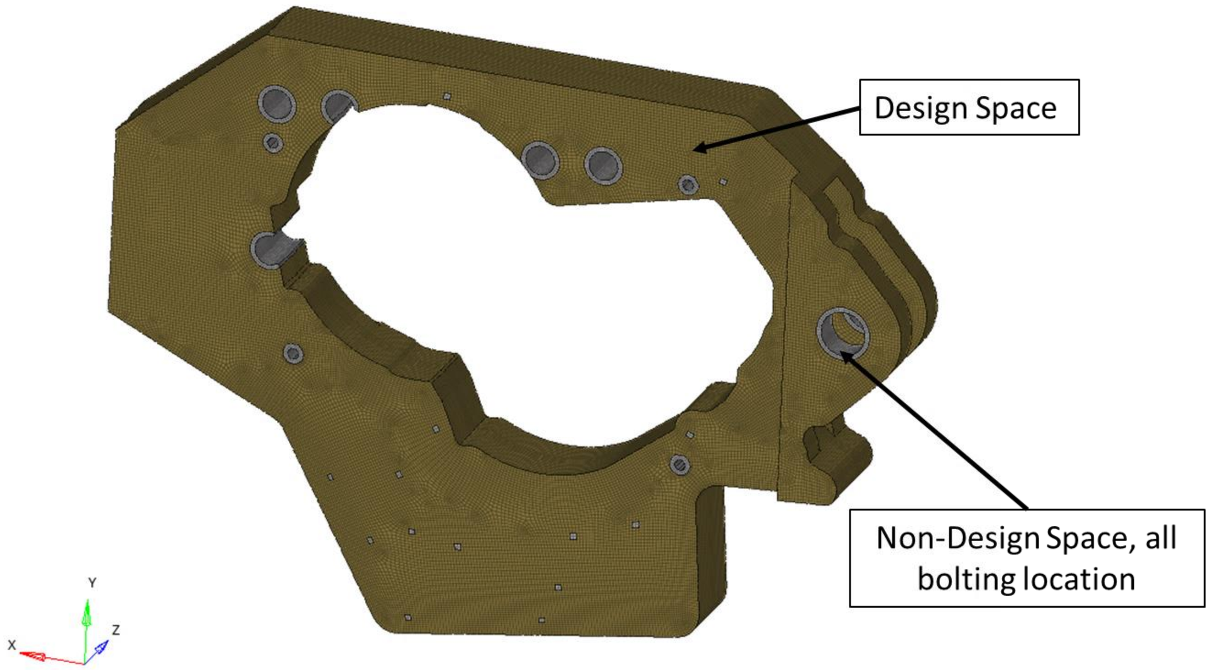

Fig 4. 21 Design Parameter

4.7 Post-Processing

4.7.1 Sub Model validation

Sub Modelling methodology is used for quick results of optimization. Below Procedure will be followed for preparing the sub model

1. Considering the rear casting along with gear box and their near by connection region for sub model.
2. After that extracted the FBD forces from the linear static analysis, and apply the same load on Sub model.
3. Static results of Sub model is matching with complete model for all the load cases.

For the global model the stress value is above the limit of 340 MPa but we can neglect that value because it is near the weld region.

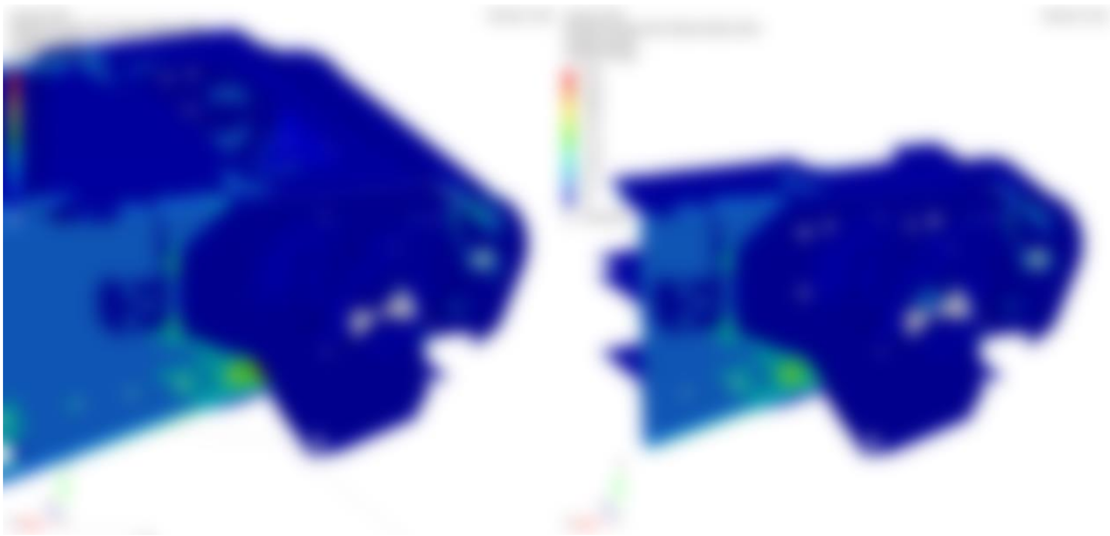

Fig 4. 22 Load Case 1: 2G Downward direction

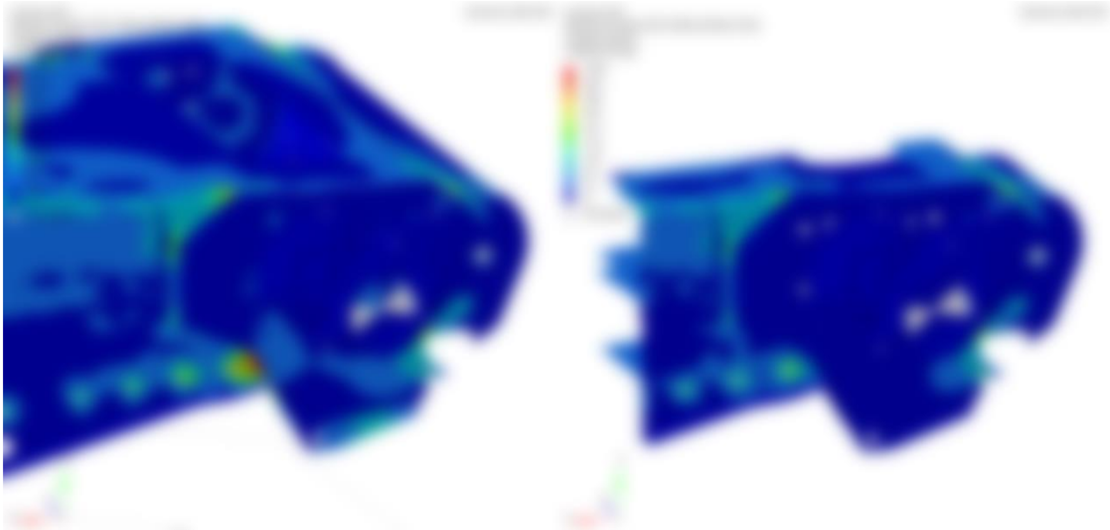

Fig 4. 23 Load Case 2: MX+ MY

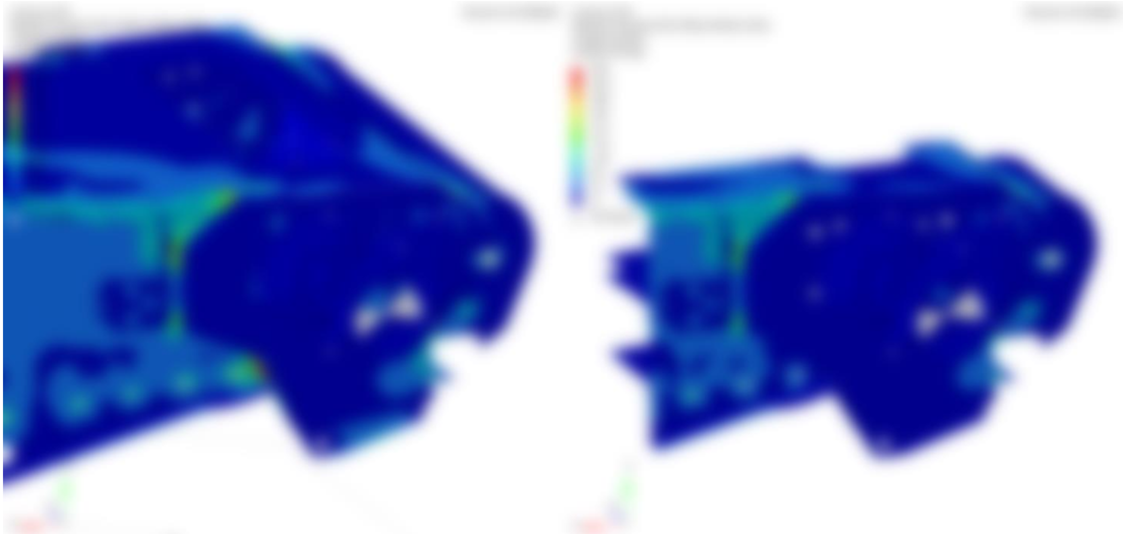

Fig 4. 24 Load Case 3: 0.2Right

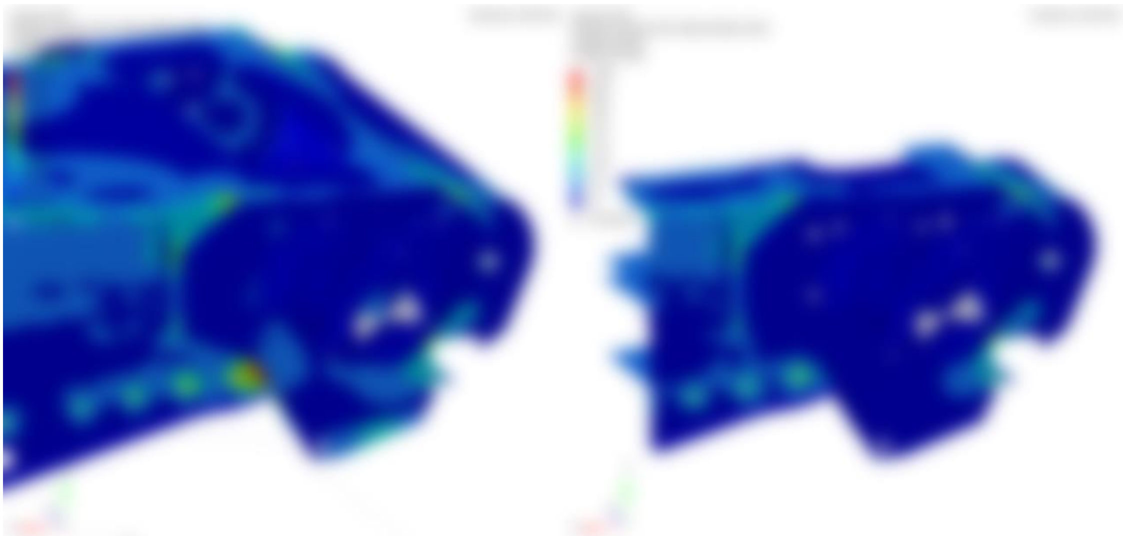

Fig 4. 25 Load Case 4: -MX -MY

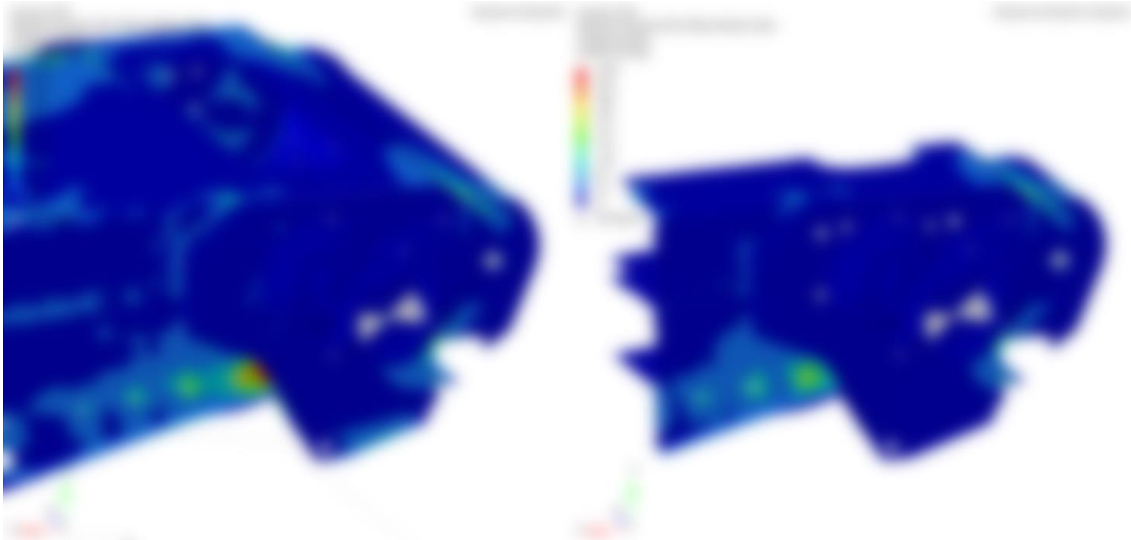

Fig 4. 26 Load Case 5: 0.2Left

4.7.2 Optimization

After the submodelling, optimization run is done. Where we got the results as per the input we have given like minimize volume. Here element density is set to be as 0.015. and we got the rough design of optimized casting.

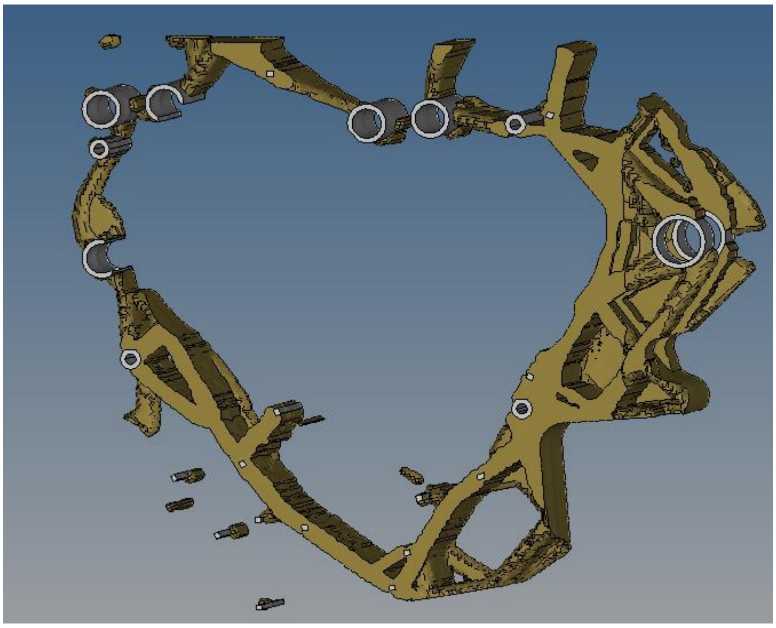

Fig 4. 27 Optimized Shape 1

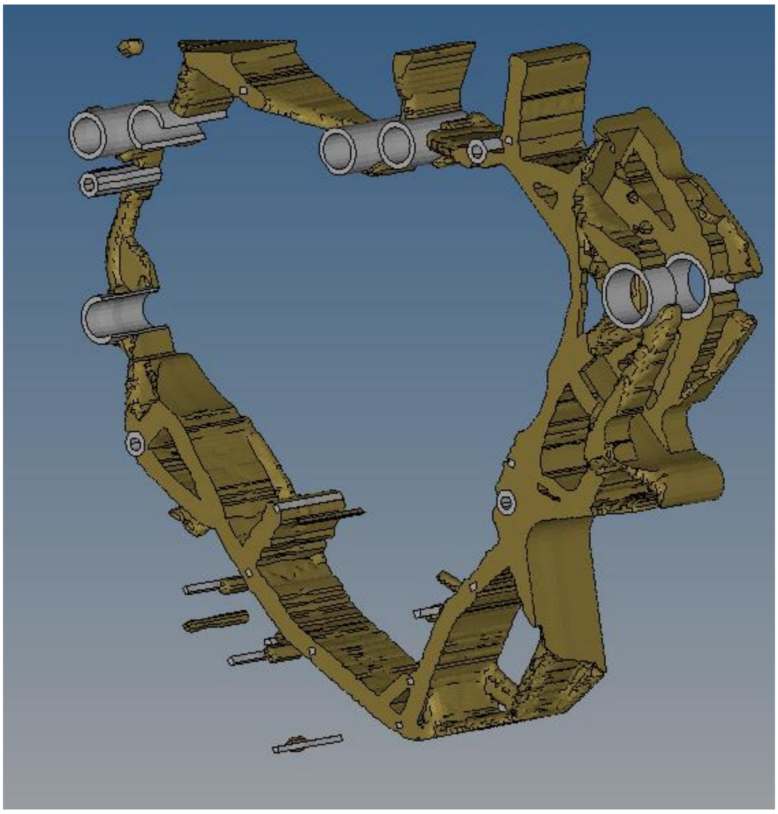

Fig 4. 28 Optimized Shape 2

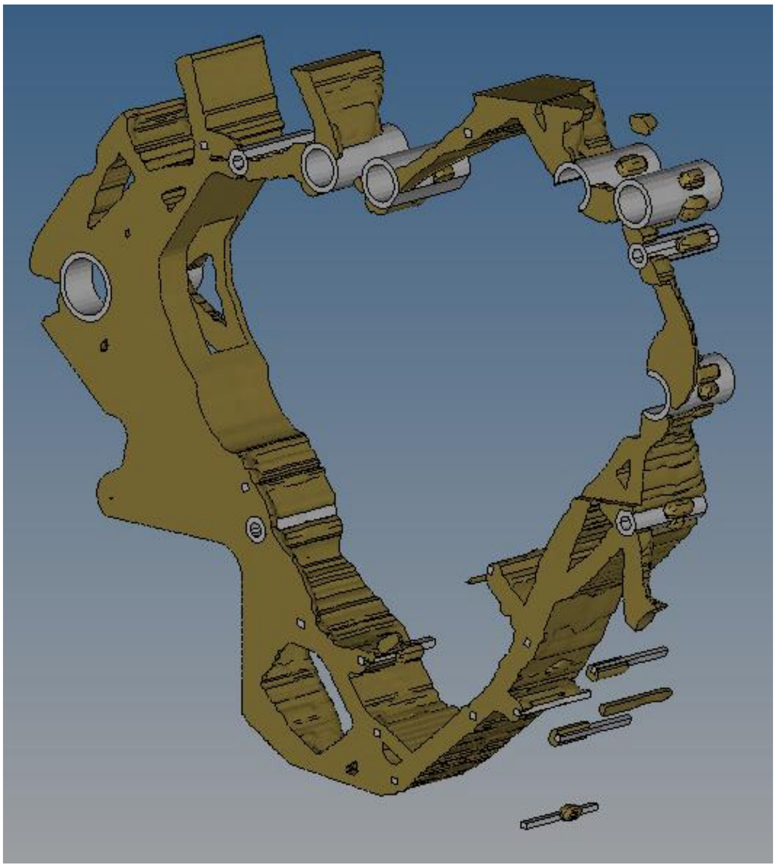

Fig 4. 29 Optimized Shape 3

CHAPTER 5 Conclusion and Future Work

5.1 Conclusion

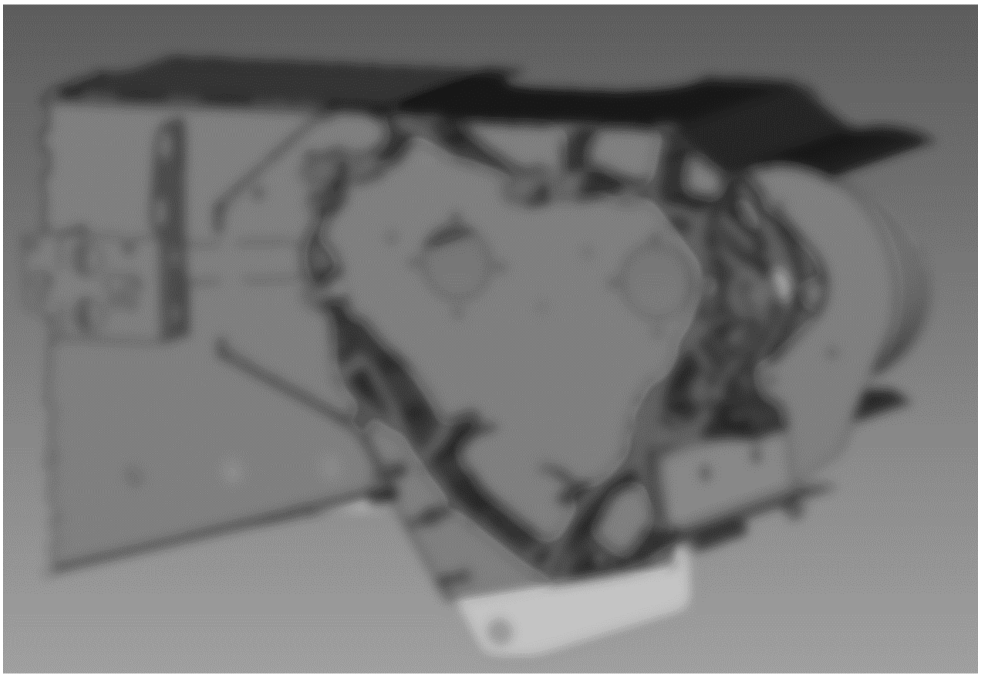

Fig 5. 1 Optimized Shape on Assembly

Sub model result is verified with the global model results, and it is nearly equals to the global model results. The maximum von-mises stress is around 400MPa which is crossing the limit of 340MPa. But we can neglect that stress value because it is stress singularity and it is on the weld region by proper mesh style we can neglect this. As per optimization result, it is recommended, marking red line location need to do the weld with plate, rest of the location bolt connects are enough. Currently the optimised shape, we have got is in Raw shape. Total wight of the casting component before optimization is around 250 kg and after optimization it is around 200 kgs. So we reduced the weight around 17% without compromising the stiffness.

5.2 Future Scope

For the future work we can work on this raw shape to create some best out of it design properly.

References

1. What Is A Chassis, And What Are Its Types? - CarBikeTech (Last used: 16-05-2022)
2. What is Car Chassis? Definition, Types, Function, Works - mechstudies.com (Last used: 16-05-2022)
3. digitallabs.edrmedeso.com/blog/elevating-your-simulations-using-submodeling (Last used: 16-05-2022)
4. www.3dnatives.com/en/topology-optimisation (Last used: 16-05-2022)
5. Company Profile | L&T Technology Services | L&T India (ltts.com) (Last used: 16-05-2022)
6. Naveen G, Dr. Prashanth A S Design and structural analysis of truck chassis International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 p-ISSN: 2395-0072
7. S. Mathivanan N. K. Mugesh, K. Praveenkumar Analysis and Optimization of Truck Chasis International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181.
8. Monika S. Agrawal, Md. Razik Finite Element Analysis of Truck Chassis international journal of engineering sciences & research technology Agrawal, 2(12): December, 2013]
9. M D Vijayakumar, C Ramesh Kannan, S Manivannan, J Vairamuthu, Samuel Tilahun and P M Bupathi Ram Finite Element Analysis of Automotive Truck Chassis IOP Conference Series: Materials Science and Engineering
10. Durmus Ali Bircan, Abdulkadir Yasar Design, Analysis and Optimization of Heavy Vehicle Chassis Using Finite Element Analysis nternational Journal of Scientific and Technological Research ISSN 2422-8702 (Online) Vol 1, No.6, 2015
11. O Kurdi, R Abd- Rahman, M N Tamin Stress analysis of heavy duty truck chassis using finite element method
12. C. Karaoglu and N. S. Kuralay, Stress Analysis of a Truck Chassis with Riveted Joints, Elsevier Science Publishers B. V. Amsterdam, the Netherlands, 2000, Volume 38, 1115 – 1130.
13. B. Ramana Naik and C. Shashikanth, Strength Analysis on Automobile Chassis.
14. Stress analysis of a truck chassis with riveted joints, Cicek Karaoglu, N. Sefa Kuralay, Finite Elements in Analysis and Design 38 (2002) 1115–1130

15. Ramesh Babu Nallamothu, Analysis of Composite Material Light Weight Chassis, International Journal of Mechanical Engineering, Vol. 7 No. 1 January, 2022.
16. Mr. Rahul L. Patel, Mr. Divyesh B. Morabiya and Mr. Anil N. Rathour "Weight optimization of chassis frame using Pro-Mechanica" ISSN: 2348-8360, SSRG International Journal of Mechanical Engineering (SSRG-IJME) - vol. 1 Issue 8, pp.4-9, December 2014.
17. www.dreamstime.com/illustration/harvester-thresher.html (Last used: 16-05-2022).
18. Bendsøe MP and Sigmund O. Topology optimization: theory, methods and applications. Berlin: Springer, 2003.
19. Olhoff N. On optimum design of structures and materials. Meccanica 1996; 31: 143–161.
20. Rajan SD. Sizing, shape, and topology design optimization of trusses using genetic algorithm. J Struct Eng 1995; 121: 1480–1487.
21. Degertekin SO. Improved harmony search algorithms for sizing optimization of truss structures. Comput Struct 2012; 92–93: 229–241.
22. Rozvany GIN, Zhou M and Birker T. Generalized shape optimization without homogenization. Struct Optim 1992; 4: 250–252.
23. Aziz MA, Owis FM and Abdelrahman MM. Design optimization of a transonic-fan rotor using numerical computations of the full compressible Navier-stokes equations and simplex algorithm. Int J Rotat Mach 2014; 2014: 743154.
24. Liu C, Du Z, Zhang W, et al. Additive manufacturingoriented design of graded lattice structures through explicit topology optimization. ASME J Appl Mech 2017; 84: 081008.
25. Dapogny C, Faure A, Michailidis G, et al. Geometric constraints for shape and topology optimization in architectural design. Comput Mech 2017; 59: 933–965.
26. Chen C-J and Young C. Integrate topology/shape/size optimization into upfront automotive component design. In: Proceedings of the 10th AIAA/ISSMO multidisciplinary analysis and optimization conference, Albany, NY, 30 August–1 September 2004, vol. 5, pp.3252–3259. Reston, VA: AIAA.
27. Rao JS. Optimization of aircraft structures and rotating machinery. Bangalore, India: Aero India, 2007.
28. Luo J and Gea HC. Optimal stiffener design for interior sound reduction using a topology optimization based approach. J Vib Acoust 2003; 125: 267–273.