ADAMA SCIENCE AND TECHNOLOGY UNIVERSITY

SCHOOL OF MECHANICAL, CHEMICAL AND MATERIALS ENGINEERING

Design of body structural frame of mini-electric train for ASTU campus

A Thesis Submitted in partial fulfilment of the requirements for the award of the degree of Master of Science in Automotive Engineering

By

Hailemariam Markos

Advisor: - Mr. Mekonnen Liben (Associate Professor)

MECHANICAL SYSTEMS AND VEHICLE ENGINEERING PROGRAM

June, 2017

Adama-Ethiopia

CANDIDATE'S DECLARATION

I hereby declare that the work which is being presented in the thesis entitled "**Design of body structural frame of mini-electric train for ASTU campus**" in partial fulfilment of the requirements for the award of the degree of Master of Science in Automotive Engineering is an authentic record of my own work carried out from November, 2016 to June, 2017 under the supervision of Mr. Mekonnen Liben, Associate Professor, Department of Mechanical and Vehicle Engineering, Adama Science and Technology University, Ethiopia.

The matter embodied in this thesis has not been submitted by me for the award of any other degree or diploma. All relevant resources of information used in this thesis have been duly acknowledged.

Name	Signature	Date
Hailemariam Markos	____________	________________
Candidate		

This is to certify that the above statement made by the candidate is correct to the best of my knowledge and belief. This thesis has been submitted for examination with my approval.

Name	Signature	Date
______________________	______________	________________
Advisor		

ADAMA SCIENCE AND TECHNOLOGY UNIVERSITY

SCHOOL OF MECHANICAL, CHEMICAL AND MATERIALS ENGINEERING

MECHANICAL SYSTEMS AND VEHICLE ENGINEERING PROGRAM

Design of body structural frame of mini-electric train for ASTU campus

By

Hailemariam Markos

Approved by board of Examiners

___________________________________ ____________

Chairman, Department graduate committee

___________________________________ ____________

Advisor

___________________________________ ____________

Internal Examiner

___________________________________ ____________

External Examiner

ACKNOWLEDGEMENT

All praises to the ALMIGHTY OF GOD for His unseen hand, which guided and strengthened us, even at times when we felt; we were not capable of doing it ourselves.

I would like to express my special thanks to my advisor, Mr. Mekonnen Liben (Associate Professor), Adama Science Technology University, for his valuable suggestion ,comments, continuous support and guidance throughout the entire thesis work.

Finally, my sincere thanks should be forwarded to my beloved family and I would like to thank all my friends and classmates who are either directly or indirectly involved during the process of this thesis was conducted.

TABLE OF CONTENTS

Contents pages

ACRONYMS AND ABBREVIATIONS

ANSYS	Analysis System
RHS	Rectangular Hollow Section
CAD	Computer Aided Design
FEM	Finite Element Method
FEA	Finite Element Analysis
CATIA	Computer Aided Three Dimensional Interactive Applications
3D	Three Dimensional
2D	Two Dimensional
DOF	Degree Of Freedom
FS	Factor of Safety
FEM	Finite Element Method
IE	Impact Energy
FRP	Fibre reinforced plastic
HSS	High strength steel
YS	Yield Strength
UTS	Ultimate Tensile Strength
δ_{max}	Maximum deformation
σ_{max}	Maximum stress
τ_{xy}	Shear stress
V	Shear force
M	Moment
σ_{v}	Von-misses stress
W	Weight

mass	mass
w	Distributed load
C.G	Centre of Gravity
E	Young's Modulus
ε	Strain
ρ	Density
v	Poisson's Ratio
I	Second Moment of Inertia
E_t	Energy Transferred
F	Force
PSQCA	Pakistan Standards Quality Control Authority
ASTU	Adama Science and Technology University
ERC	Ethiopian Railway Corporation

LIST OF FIGURES

Pages

LIST OF TABLES

Pages

ABSTRACT

Railway vehicle structures generally consist of shells, plates and beams, and behaviour of these members directly affect static and dynamic structural behaviour of these vehicles. In this paper the body structure of a small train built in ASTU is improved by modification and analysing its structural design. Improper design of the train structure and shape leads to failure in strength, reduction of speed, high power consumption and reducing the overall performance of the train. The objective of this research is to improve the body structure of a small train built for ASTU campus. It assess specifically the structures of the mini-train built in ASTU, prepare the model of the existing structure and the improved structure, analyse the structure for strength, deflection and compare the results. The scope of this study includes modelling of the train body using CATIA software, analysing structural integrity (strength and stiffness) to observe the structure strength (stress and deformation) using finite element analysis software, dynamic analysis (crash analysis) of the model using the proper finite element analysis, re-modelling original structure with newly replaced members of the structure. The research commences by modelling the train structure which is designed and manufactured in ASTU. This model is then analysed for strength i.e. stress and deformations. With the defects found on the previously built small train, we re-modelled the small train again for static structural analysis and crash analysis. It is found that, in the existing model all the total displacement values obtained are above 16.752mm whereas, in the new model designed using space frame the maximum displacement obtained is 0.49726mm. This indicates that the stiffness values are improved. Finally, the two results are compared whether structurally safe. Finally, we draw a conclusion that replacing the RHS of 40×40×1mm by 30×30×3mm, selecting space frame and reducing the number of elements gives improved and better structural strength and stiffness than the existing mini-electric train based on static structural analysis by ANSYS 15.0 workbench. Further studies also recommended for refined body structure.

Keywords: *railway vehicle structure, structural design, train structure, mini-train, structural integrity, finite element analysis, static analysis, dynamic analysis, strength, deflection, crash analysis*

CHAPTER ONE

1. INTRODUCTION

Structural design is the methodical investigation of the stability, strength and rigidity of structures. The basic objective in structural analysis and design is to produce a structure capable of resisting all applied loads without failure during its intended life. The primary purpose of a structure is to transmit or support loads. If the structure is improperly designed or fabricated, or if the actual applied loads exceed the design specifications, the device will probably fail to perform its intended function, with possible serious consequences. A well-engineered structure greatly minimizes the possibility of costly failures. The purpose of this thesis is to design body structure for the mini-electric small train built in Adama Science and Technology University (ASTU) using ANSYS software. In a very wide compound area like ASTU, if the track infrastructure of the rail vehicle is well designed, adapting mini-electric rail vehicle is very significance as the researchers in the said university are proposed and built small train for transporting staffs, students and other employees around the campus [3].Therefore this thesis will study further on the body structure of the mini-electric train for structural strength since the researchers were focused on adapting and designing of power transmission unit and mechanism of working, not on its body structures and frames. The need of researching a strengthened car body is quite clear that it protects the occupants mainly and connect other components of all the mechanical and electrical main parts install on it.

1.1. Background of the study

A train is a form of rail transport consisting of a series of vehicles that usually runs along a rail track to transport cargo or passengers. Motive power is provided by a separate locomotive or individual motors in self-propelled multiple units. Although historically steam propulsion dominated, the most common modern forms are diesel and electric locomotives, the later supplied by overhead wires or additional rails. Other energy sources include horses, engine or water-driven rope or wire winch, gravity, pneumatics, batteries, and gas turbines sometimes the concept of car body is limited to only the load carrying structure of the vehicle [2].

Railroad transportation gets attention as a very energy-efficient form of public transport. Currently, development of technology is underway to meet the varied needs of the increasingly diverse range of forms that railroad transportation is taking and there is a need to take action toward creating new value in order to achieve further progress in railroad transportation technology [7].

The core element of any car is the body structure. The car body connects all the different components; it houses the drive train and most importantly carries and protects passengers and cargo. The body structure needs to be rigid to support weight and stress and to securely tie together all the components. Furthermore, it must resist and soften the impact of a crash to safely protect the occupants. In addition, it needs to be as light as possible to optimize energy economy and performance [8]. Therefore, Car bodies of modern rail vehicles are designed as lightweight structures with the aim to minimize mass and thus operational energy demand.

There are several studies on car body structural flexibility. The models range from simple beam models to detailed finite-element model. In most cases, models obtained from experimental model analysis and finite-element calculation are used in the analysis. Modern general-purpose software for the simulation of railway vehicle system has included features that enable efficient dynamic analysis of the railway vehicles. The dynamic behaviour of railway vehicles relates to the motion or vibration of all the parts of the vehicle and is influenced by the vehicle design. The structural design of railway vehicle bodies depends on the loads they are subjected to and the characteristics of the materials they are manufactured from. The mass reduction and the associated lightweight structures are gaining importance in rail vehicles. The reasons for this are diverse and depend on the requirements which are demanded of the vehicle. For example, the forces exerted by the train on the track in a vertical direction are limited. This is the main reason for lightweight construction of high speed trains. Limiting the maximum axle loads often presents a challenge and demands rigorous lightweight construction. Further benefits of vehicles with reduced mass include, for example, reduced driving resistance and, as a result, significant energy saving [4].

Therefore, in order to make improvement on the design of the structure, it is basic to analyse the structure for stresses and deformations to find parts which are subjected to maximum and minimum stresses and deformations. This indirectly tells whether parts are highly loaded or not.

1.2. Problem statement

In the modelling and design of small mini-electric train built in ASTU, which can be used for transporting of passengers (students and staffs) in 'Adama Science and Technology University' is very useful. In the previous project work, a prototype of light train was manufactured and demonstrated. They stated also, how it works and tried to explain how it would be helpful if it is implemented well. But the structural design of the body was not according to static and dynamic load considerations of mechanical design analysis. So this paper will focus on design of the body structure of small train and will indicate the shortcomings of that design using ANSYS software. This shortcoming include structural failure, as a result an improved train body structure based on finite element analysis will be proposed.

The result of the analysis and the implementation of the design will contribute to the improvement of the locally designed body structure of small train in ASTU.

1.3. Objectives of the Study

1.3.1. General objective

The general objective of this research is to design the body structural frame of mini-electric train for ASTU campus.

1.3.2. Specific objectives

The specific objectives of this thesis are;

- To study the structures of the mini-train built in ASTU.
- To prepare the model of the existing structure and the improved structure.
- To analyse the structure for strength and deflection.
- To compare the results.
- To recommend the possible and relevant improvements in the structural design of small train.

1.4. Scope of the project

The scope of this project includes the following analysis,

- Modelling of the train body using CATIA software.
- Analysis of structural integrity (strength and stiffness) for previous design to observe the structure strength (stress and deformation) using finite element analysis software.
- Re-modelling of the original structure with newly replaced members of the structure.
- Analysis of structural integrity (Strength and stiffness) for the new structure to observe the structure strength (stress and deformation) using finite element analysis software.

1.5. Significance of the study

It is widely recognized that reliable railway system is a key to achieving sustainable economic growth and the prosperity of society. Many researchers have done an optimum design in the field of railway technology to fulfil the public demand and to be competent with other modes of transport. Therefore this thesis draws a great contribution in designing an optimum body structure for the mini-electric train built in ASTU campus. As a result of this study;

- Can be a resource for designing an optimum body structure;
- Directs designers how to design a rigid structure capable of supporting different loads;
- Can be a reference for designing body capable of resisting impacts and enhancing safety;
- It reduces energy consumption as a result of light weight as much as possible;
- Finally, it opens a door for other researchers to study and achieve optimum body structural design of a rail car.

1.6. Limitations of the study

The limitation that might be encountered when conducting the research includes; ignoring the effect of other property like thermal analysis and vibration and being theoretical only because of lack of full information in the stated field and MATLAB SimuLink.

1.7. Organization of the thesis

The research is organized in five chapters. The first chapter is the introduction part which states the background of the research, statement of problem, objective, scope and limitation of the study.

Chapter two focuses on Literature review on the structural analysis, related literatures and theoretical basis of rail vehicle car body. And also the alternative materials used and their general properties are briefly described.

Chapter three shows method and materials. Under here, the 3D modelling of structural body of small train, the mathematical and finite element modelling of the original car body and the newly modified are prepared for static analysis. Test analysis and simulation result interpretation is done using numerical approach. Chapter four stated about result and discussion of the existing model and the new modified model based on the simulation result from FEM. Conclusion and recommendation of the research is discussed and forwarded in chapter five.

CHAPTER TWO

2. LITERATURE SURVEY

2.1. Introduction

The design of vehicle body has been evolved from a simple, all steel structure that meets the basic requirement of strength and functionality, to the current day complex and efficient structure. Lightweight composite materials, such as glass-fibber reinforced polymers, have been used to replace traditional steel and aluminium components. This is because composites offer significant opportunities for enhancement of product performance in terms of strength, stiffness and energy absorption, combined with weight reduction and space saving. Today, design procedures of vehicles body that ensures reliability and road worthiness is well established. However, as a result of advancements in the areas of material, production methods, computational and design tools, optimization technique, etc., further improvement on vehicle structural design is still an active topic of research. In conventional practice the passenger car body is produced from overlapping sheet metal, fastened by a multiplicity of spot welds often robotically applied [1][2].

This chapter is all about reviewing several literatures and exercising different software that have been used in the field so far. For this reason, related literature such as books, articles, proceeding papers and some other sources that are retrieved from the internet are consulted, so as to understand the domain knowledge, concepts, principles and methods that are important for developing knowledge-based systems.

2.1.1. History and development of railways

This document is taken from [7]

i. The precedent of railways

Even during the Renaissance, Leonardo da Vinci conceived, but never to realize his project, the first machine capable of moving without resorting to force an animal. Then in mid eighteenth century, the French inventor Jacques de Vaucanson, who had dedicated their efforts to design robots, conceived a sort of vehicle powered by a system similar to the clockwork. Soon after, a Swiss national priest J. H. Genevois, planned a similar device, powered by a somewhat strangely unconventional procedure, two windmills, which provided small on top.

ii. RAILWAY DEVELOPMENT

In the eighteenth century, workers from different mining areas in Europe found that loaded wagons moved more easily if the wheels were turning guided by a rail made of metal plates, because in this way reduced friction. The lanes for cars only served to move the product to the nearest waterway, which was then the main form of transporting large volumes. The start of the Industrial Revolution in Europe in the early nineteenth century, demanded more effective ways of bringing raw materials to the new factories and moved from such finished products.

The two mechanical principles, guiding wheels and use of motive power, were combined for the first time by the English mining engineer Richard Trevithick, who on February 24, 1804 succeeded in adapting the steam engine, used since the early eighteenth century to pump water, to pull off a locomotive was run at a speed of 8 km / h pulling five wagons, loaded with 10 tons of steel and 70 men on a road 15 km from the smelting of Pen-y-Darren, in South Wales.

Two decades passed during which they developed the cast-iron rails that supported the weight of a steam locomotive. The power required to pull trains, rather than one or two cars, secured by placing a steam locomotive on two or more axles with wheels attached by rods.

Finally, in 1825 was opened to the public the first steam railway: a set of wagons pulled by a locomotive used this power, covering the distance between the English towns of Stockton and Darlington Five years later it was opened on Liverpool Manchester flight which said regular traffic of goods and passengers between the two locations, the locomotive, the famous Rocket, was built by the said Stephenson. With appropriate improvements, the prototype would be used in future machines.

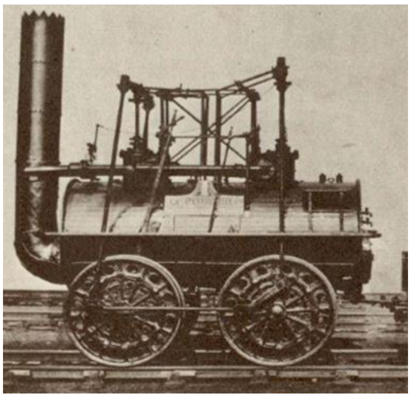


Figure 2.1. First Railway [7]

A mid-nineteenth century were built many miles of track, around 1850 the steam railway had reached every continent.

The builders of Europe and North America in general adopted the width of 1,435 mm (56 inches and a half) of the proposed George Stephenson, which was based on the lines of way for mining trucks from their place of origin had been shown empirically dimension was the most suitable for towing by human or horses. International standardization of this broad did not occur until the Berne Conference of 1887.

iii. NEW FORMS OF ENERGY APPLIED TO RAIL

Despite all the progress made, excess weight and volume of the steam engine and the high cost of facilities to supply fuel and water, coupled with the low performance and high degree of contamination of these machines, and favoured the emergence new technologies. Around 1940 the manufacture of steam engines was interrupted in America and Europe.

The first trains that used power from the late nineteenth century, the first railway of its kind was launched in 1881, near Berlin. However, the first regular rail service was the one who joined in 1895 Baltimore and Ohio in the United States.

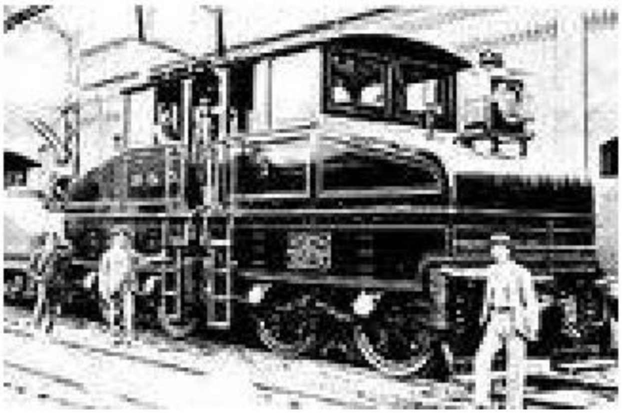


Figure 2.2. The First regular rail service [7]

2.1.2. Railway Development in Ethiopia past and current stage [6]

Over 100 years old diesel railway (781 km) was owned jointly with the Government of Djibouti operated by CDE (Chemin de fur Djibouti Ethiopian). In the present Status it is almost abandoned due to its age - deterioration and malfunctioning. The main function was:

- It was major Service Provider for passenger and freight transport to the Eastern Ethiopia

- Played a role in establishment and expansion of major economically active urban centres along its line

Modern and reliable railway system was needed to sustain the economic growth momentum of the country by supporting the demand of freight and passenger mobility. Recently, Ethiopian Railways Corporation (ERC) was established in 2007 by Regulation No. 141/2007 to:

- Develop railway infrastructure
- Provide freight and passenger railway transport services and
- Engage in other related activities necessary for the attainment of its purpose.

The Ministry of Transport of Federal Democratic Republic of Ethiopia was found to be prudent to develop Railway System in the country. The following train overview of light rail of AART helps to have a knowledge on rail.

Addis Ababa light rail transportation

i. Train over view

AALRT train is a 70% low-floor tram of Chinese brand, at 750V DC, with a capacity of 317 passengers per train (standing capacity: 8 persons/m^2), and the maximum speed of 70km/h.

ii. Classification of Rail vehicle

1. **Passenger rail vehicle:**

Examples of this vehicles are light rail and high speed trains. Their main function is to transport passenger in a city, across a country and from one country to other country.

Figure 2.3: Body of passenger rail car [6]

2. **Freight Rail Vehicle:**

Rail flatbed cranes, towed by rolling stock, can be coupled into groups for cooperative operations or used for lifting operations separately to lift and transport materials or machines and tools.

2.2. Related literature

2.2.1. Adapting improved mini electric train for wide area compound, the case of ASTU campus [3]

A mini-electric train with modified and improved feature was developed [3] to solve the demand of transportation inside the campus i.e., to save time and energy wastage due to long and tiresome walks among offices and workshops, students from the gate of the university to class rooms, dormitory, stadium and other locations inside, in wide area compound with scattered construction like ASTU. There are fuel powered and electric vehicles to solve the stated problem but are not enough and less important in some ways when compared with track line mini-electric trains. Therefore, it needs a new solution and they proposed and built a mini-electric train.

a. Mini-electric train and working principles

The researcher used locally available materials for construction of the rail and train structure as well as the overhead caternary. The train has electrical component parts, rail, caternary, poles and bow collector. DC brushless motor is installed at rear of train for rear drive while invertor and control card are located on the front of the train.

AC public electric grid was used for their work to change into DC using convertor to make overhead caternary with out shock and inventor is used the DC back to AC inter DC card again changing to 200 volts fed to the motor.

Figure 2.4: 3D picture of the train [3]

b. Body structure of the mini-electric train

The researcher mentioned CATIA software for drawing the structure of the train, figure below sows the shape and stricture of the train taken from his paper a photo taken from the exixting body train near manufucturin engineering shop (in front of machine shop) in ASTU.

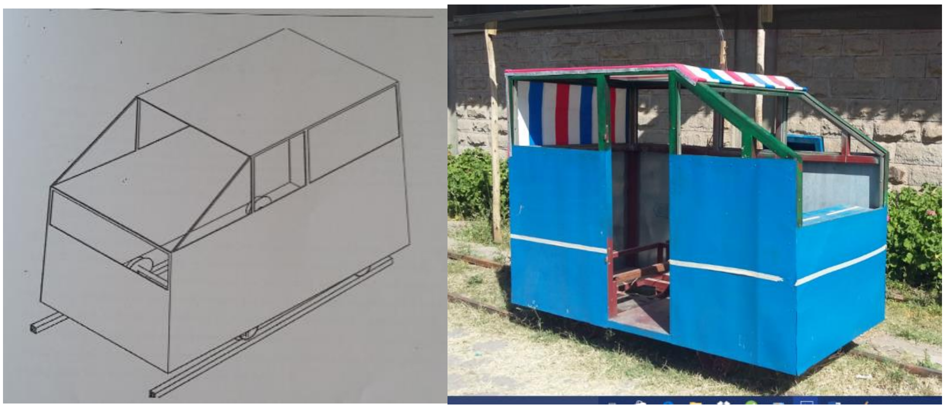


Figure 2.5: shape of the existing train

In their study, they mainly concerned on the generation of power for mini-electric train and it was quite appreciating and good initiation for home grown railway technology. But it was not good regarding their body structure i.e., whether the body structure is safe design or not, it needs further study. The structure was simply welded frame to frame Therefore it needs to check for structural analysis using ANSYS software.

c. Design works of the previous mini-electric train body structure

The researcher uses Dc motor with 0.75 kW power, 2000rpm and 3.5 Nm torque and this motor is connected by V-belt with pulley which is fixed at the rear axle centre point as shown below.

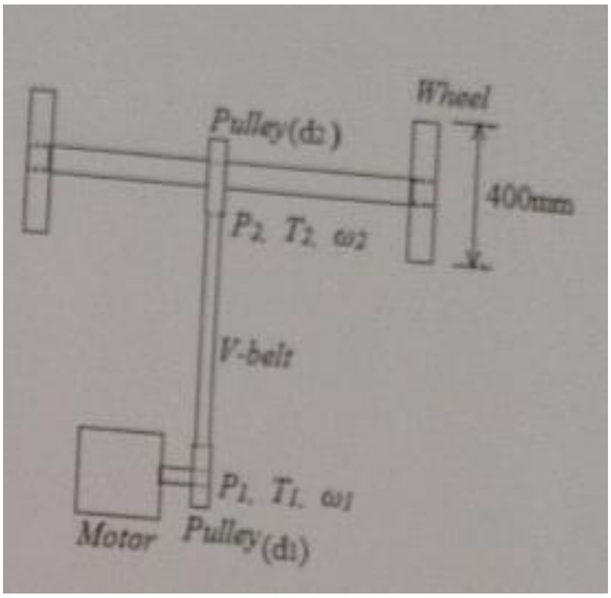


Figure 2.6: Power transmissions

1. **Longitudinal and vertical forces**

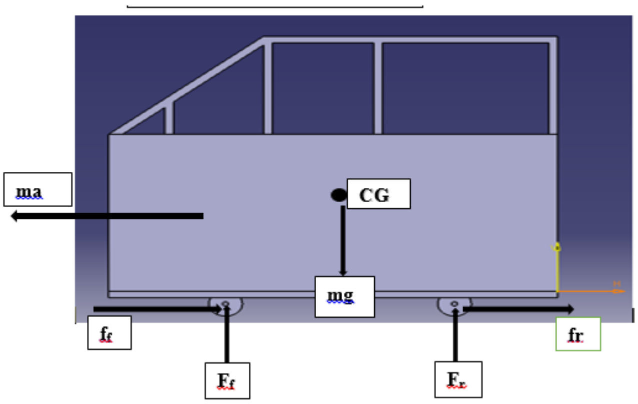


Figure 2.7: forces acting on the body train

$$\sum \boldsymbol{Fx} = \boldsymbol{ma} \text{.. .... (3.1)}$$

$$\sum \boldsymbol{Fy} = \boldsymbol{0} \text{.. (3.2)}$$

$$\sum \boldsymbol{Mr} = \boldsymbol{0} \text{... (3.3)}$$

Where, $\mathbf{F_x}$ and $\mathbf{F_y}$ are the forces in the direction of x and y respectively, $\mathbf{M_r}$ is the moment at rear axle of the train, **m** is mas of the train structure and **a** is acceleration.

2.2.2. Modelling and Structural Analysis of Railway Vehicle Body with Finite Element Method [2]

In this literature, a structural model was developed by using CATIAV5-R20 modelling and for analysis, and this model was imported with a file format such as IGES, ACIS, toANSYS-14.5 for final result analysis. The purpose of the structural sketch was to define the primary car body structure in advance of formal stress analysis and structural drawings. With the basic understanding of how finite element programs work, a finite element model was created with appropriate parameters such as dimensions, loads, constraints, element choice, mesh selection, etc. The analysis of their work was done for different types of loading purpose and it is summarised below [28].

Static structural analysis

The static analysis was concerned with determination of the response of the rail vehicle car body to steady state loads. The corresponding stresses and strains were obtained using strain-displacement and strain-stress relations. The deflections between two nodes were determined using assumed shape functions. Their solution to the equation gave nodal translational and rotational displacements [32].

In their analysis, the situation considered was the stand still condition of the vehicle on a level road loaded to the maximum payload. The other loads considered include dead weight of the engine, and weight of the vehicle itself.

2.2.3. Numerical static and dynamic stress analysis on railway passenger and freight car models [5]

In this work, they defined railway vehicle structures generally consist of shells, plates and beams, and behaviour of these members directly affects static and dynamic structural behaviours of these vehicles. In this study, numerical results on stress, vibration and crash analyses of railway passenger and freight car structures made of steel members was presented. In their analysis, Finite element (FE) method was used to assess the static and dynamic structural behaviour of railway vehicles.

a) Railway passenger and freight car models

Railway passenger and freight cars called respectively “N13-type” and “SGS-type” used by Turkish State Railways was examined in their study. The passenger car body is mainly made of five different steel materials (i.e., stainless steel, St 12, Stw 24, St 37 and St 52), composed of two sidewalls, underframe, roof frame and two end walls while freight car is mainly made of St 52 and composed only underframe.

b) Static analysis of the railway passenger and freight car models

Railway vehicles are subjected to different loading scenarios such as symmetrical compression force and tensile force. Loading and boundary conditions was adapted to FE simulations and to validate the FEM model. In their analysis they stated as, Von Mises of the tensile force for freight car and symmetrical compression force for passenger car, and displacement results obtained in the middle point of the passenger car body by using experimental measurements agree well with FE results.

c) Crash analysis of the railway passenger and freight car models

The main aim of crash analysis is to provide optimum crashworthiness design and to prevent loss of life when impact occurs. Nowadays, passive protection design approach is used in rail vehicle design. According to this approach, when collision occurs, the rail vehicle should deform and collapse in a controlled manner in such way structural deformation progresses from the vehicle end to its interior, without damaging the passenger's living regions [21].

For the deformation to progress in a controlled manner it is necessary that deformation of structural members such as extruded beams and plates are also controlled as the amount of energy absorbed in a vehicle is directly related to the yield strength, work-hardening behaviour, thickness and geometry of the structural members. Several crash scenarios are used to determine crashworthiness characteristic of railway vehicles such that, single-car impact with rigid wall, two-couples-car impact with rigid wall and rollover of vehicles. Rigid wall impact simulation is mostly used to reveal the general characteristics of impact behaviour of car for its simplicity [25].

To summarize this paper; as a result of relevant simulations, static and dynamic structural characteristics and structural weaknesses of designs are determined and working with the FE models saves cost and time during the development processes of railroad vehicles and complex simulations can be performed easily at all stages of design.

2.2.4. Modelling and Dynamic Analysis of Bus Body Built in Ethiopia [4]

In their paper, the structural strength and speed was a fundamental concern. Unnecessary weight of the bus structure leads to reduction of speed, high fuel consumption and reducing the overall performance of bus. As per objective of this work, the primary task was to improve the bus structure weight. The improvement was carried out by analysing the structure for two main loading cases, bending load and torsional stiffness; and they examined the stresses and deformation responses of a typical bus structure during application of the load in service. The work was started with modelling and importing the assembled body structure from CATIA V5R16 to ANSYS 12 workbench for analysis [22][29].

i. Welding effect in finite element modelling of the structure

In their work welding effect of FEM was considered. Because, the model constitutes assemblies of individual members to form the whole structure. When the model is imported in to ANSYS 12 workbench, the ANSYS 12 workbench was automatically recognized the members in contact. The contact surfaces in reality are the places where welds are found. But to incorporate the weld effects in to the analysis, they were in need to use ANSYS 12 workbench weld representative element. Therefore, in ANSYS workbench, there are two ways of considering welded assemblies. The first one is contact element where it is used when we have a solid model. The other one called spot-weld connection is used when we have model of surface bodies assembled together.in our analysis we assume contact element weld assemblies but we did not studied them their effects in this paper.

The assumption in their paper in ANSYS 12 workbench was that contact elements transfer structural loads and structural effects between solid body parts and used for stress, displacement, frequency, strain, and solutions.

ii. Static structural analysis

Static analysis was carried out on the model to get the weight of the bus structure, stress, deformation and performed for two travelling road conditions. The first one, bending load case, is used to demonstrate the bus travelling on a smooth road and extract the deformation and stress values. The second case, twisting moment case, is used to find the values of stress and deformation while the wheels face bumpy road. In this case, the front right side wheel is subjected to bump. But in our case, road condition is not a concern for our problem even though the body of bus and rail car vehicle are similar, because our vehicle is traveling on a track which is guided by rails and no bump and less friction. The need of our work and bus structure modelling is we equally share the methodology and type of analysis which is static structural analysis.

iii. Improvement process

Based on the objective of their work, the strength of bus structure is the most important thing to be considered in the design process. The bus model used in their study for simulation was developed with the same dimensions of a real bus manufactured in Ethiopia.

The results of the original structure analysis show the stresses found are small, which directly implies the bus structure is overdesigned. It was seen that the highest displacement occur at the roof section of the bus structure because the members used to build the roof are smaller in cross section than other members.

This gives them a room to further improve the bus structure by redesigning the body structure where the members of original structure was replaced with reduced thickness and cross section profile. It is found that the stress is raised in some members, however looking at the overall structure, the stresses found are still small.

2.3. Theoretical basis

A train running along a track is one of the most complicated dynamical and static systems in engineering. Many bodies comprise the system and so it has many degrees of freedom. The bodies that make up the vehicle can be connected in various ways and a moving interface links the vehicle to the track. This interface involves the complex geometry of the wheel tread and the rail head and non-conservative frictional forces generated by relative motion in the contact area [2].

BELETE JIRU in 2015 [2] also defines as, car body of rail vehicles refers to the load carrying structure, doors and windows, interior with seats etc., inner lining and so-called comfort systems for lighting, heat, ventilation and sanitation. The technical equipment for propulsion, braking etc. is not definition not include in the car body, even though this equipment usually is attached or hinged with car body. Sometimes the concept of car body is limited to only the load carrying structure of the vehicle.

Before putting a railway vehicle into service, it must be certified. Both Static and dynamic tests are performed according to international standards during the certification procedures. Some standards, specifications and requirements for static stress tests, vibration and crashworthiness of rail vehicle structures can be found. Within the scope car body standard, it is intended to provide a uniform basis for structural design of the vehicle body. The loading requirement for the vehicle body structure design and testing are based on proven experience supported by the evaluation of experimental data and published information.

Numerical methods and experimental measurements are used to determine the static behaviour of railway vehicles. Experimental measurements are very time consuming, expensive and they cannot be used at all stages of design. Hence, today numerical methods are important tools in static analyses of railway vehicles. Although numerical simulations do not have aforementioned disadvantages of experimental methods, they must be verified by experiments to obtain realistic results. FE method is a powerful numerical engineering analysis tool, and widely used in static stress analyses of railway vehicles.

Half-width or half-length and half-width or full-length modelling approaches are used to determine static structural, vibration depending on the symmetry of vehicle. Nevertheless, half-width or full length and modelling approaches can be used only for certain types of simple static structural loading such as static symmetrical compression and symmetrical tension loading conditions and cannot be validated for complex static structural loading and dynamic loading conditions. Hence, full-length simulations are important for the validation of designs and to provide the greatest possible accuracy.

To obtain static structural behaviour of the railway vehicles, i.e. stress and strain distribution, different static loading cases defined in [2] can be used in finite element analyses and are validated against experimental measurements; i.e. most common-base references that are taken into account are for static analysis and for dynamic analysis. Finite element method is used to assess the static and dynamic structural behaviour of railway vehicles.

2.4. Train body structure

The main purpose of a train body structure is to support all the major components and sub-assemblies making up the complete vehicle; and carry the passengers and/or payload in a safe and comfortable manner.

2.4.1. General Car body description

The car body shall be a lightweight integrated structure designed in harmony with the trucks and the coupler or draft gear system in regards to vibration damping and collision resistance.

Emphasis shall be placed on a structural design which allows maximum energy absorption in a collision by means of plastic deformation, transferring minimum forces to the passenger.

All car types supplied shall be of identical design and construction, except for the special requirements of the cab end of the cab car and food Service car window arrangement. The maximum amount of parts commonality between all car types is desired.

Car bodies of modern rail vehicles are designed as lightweight structures with the aim to minimize mass and thus operational energy demand. The central structural design requirements are given by the main static load. Railway vehicle structures generally consist of shells, plates and beams and behaviour of these members directly affect static and dynamic structural behaviours of these vehicle. The structural design of railway vehicle bodies depends on the loads they are subject to and the characteristics of the materials they are manufactured from [7].

The weight reduction of railway vehicles has different impacts: [2]

- One result is the energy saving. Simulations have shown that the potential for savings through reduction of vehicle mass is dominated by the characteristics of the service profile. There is a particularly high potential for savings with service profiles that have short distances between the stations and low maximum speeds. As the maximum speed increases, the proportion of energy needed to overcome aerodynamic resistance increases and the potential for savings goes down.
- Second effects can be seen in the reduction of rail damage and superstructure optimization according to the axle load.

In terms of weight distribution, a railway carriage can be divided into equipment, bogies and car body. Due to the equipment systems complexity and necessity to address the various subsystems separately and due to the bogies many and security-relevant a requirement, their weight saving potential is limited. The global weight saving potential resulting of the rail vehicle equipment is limited. Nonetheless the car body structure is in an interaction with different parameters. This means the car body structure has a direct influence on the weight saving potential of these parameters. Development in the field of railways goes in particular through decreasing the weight of rolling stock vehicles car body structure [28].

As a general rule, in rolling stock vehicle design the use of lightweight principles is indispensable.

The principles can be divided into: [2]

- Using of lightweight material function and system lightweight optimization (lightweight system configuration) form and shape design lightweight optimization

Currently different concepts of car body are in use:

- Differential concept (metallic framework planked with blank sheets)
- Integral concept (locally reinforced aluminium extruded profiles welded in longitudinal direction)
- Hybrid concept (different sections uses potential of different materials)

A railway carriage can be divided into three main assemblies: the running gears, Car body structure and equipment (like propulsion, interior, car body completion, etc.)

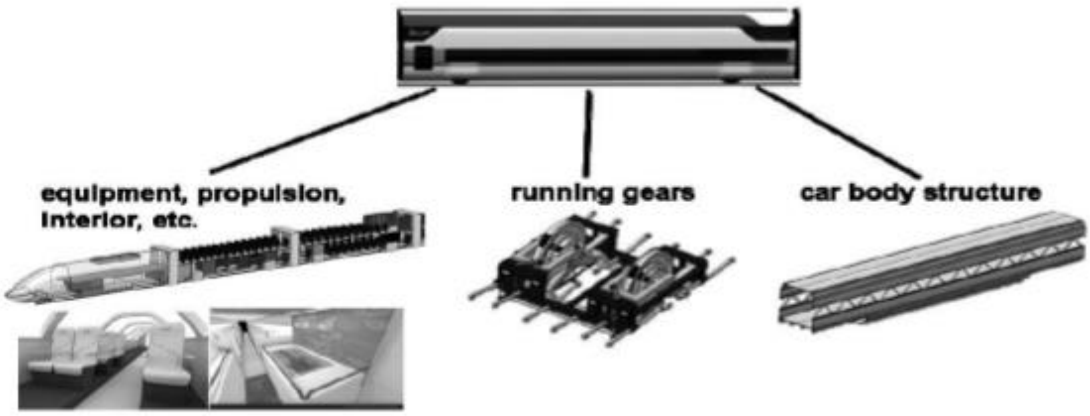


Figure 2.8: Vehicle body carriage subassemblies [2]

2.4.2. Passenger rail vehicles

To this group belong all types of railway vehicles intended for the transport of passengers, ranging from main line vehicles, suburban and urban transit stock to tramways. As a general requirement; rail vehicle car body should have limited weight. Trains exert forces on the infrastructure due to their weight which results in wear of the tracks. High weight also increases the wear on wheels, axles, brakes, shock absorbers, etc. The car body should also be sufficiently stiff. From a safety perspective the car body should not flex outside the track gauge during operations or show significant vertical displacement due to the passenger load.

Different research has been done on the materials used for the rail vehicles to get the materials advantage to minimized the stress, vibration, minimize the material deformation and other related to maximize the static nature of the rail car and also done to reduce the weight of the vehicle body.

For passenger rail car or any other vehicles, it is very important to have a chassis (Bogie) or frame for many purposes. Bogie is a chassis or frame work carrying wheels, attached to a vehicle this serving as a modular subassembly of wheels and axles. The purpose is

- Support of the rail vehicle body
- Stability on both straight and curved track
- Improve ride quality by absorbing vibration and minimizing the impact of centrifugal forces when the train runs on curves at high speed.
- Minimizing generation of track irregularities and rail abrasion.

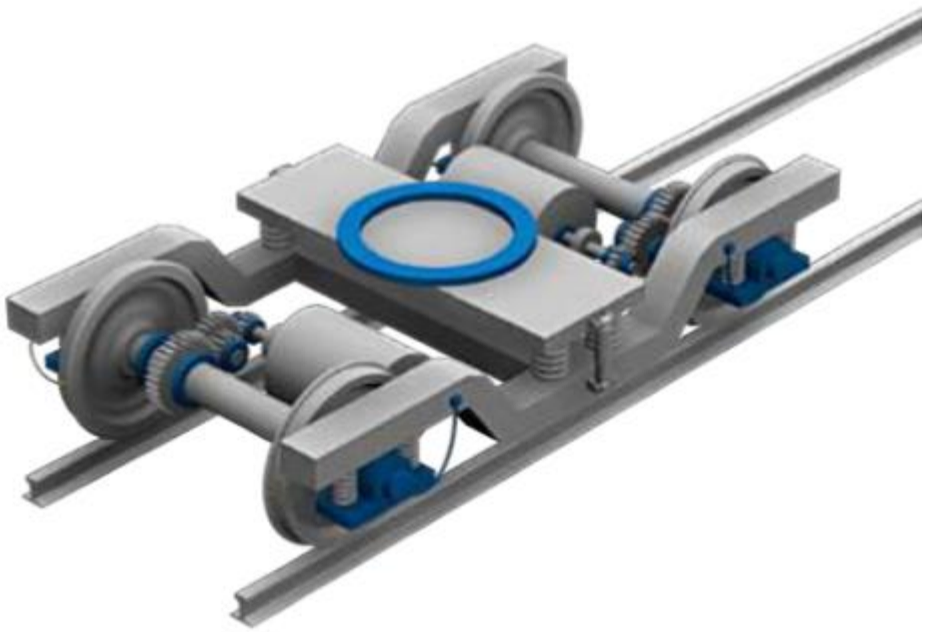


Figure 2.9: railway bogie

Since our model (small train) is very low speed and small power using (25 km/h top speed and 0.75 KW DC motor), we have to modify our small train with the following one of two best selected chassis frames. Each one has advantages and disadvantages [1], [8] and [9].Therefore, let's have a brief explanation on the following vehicle frame structure.

Space frames

Space frame is the oldest structure system used and nowadays it only applies in the construction of commercial vehicles and off road vehicles. It consists of two different structures: the frame and the bodywork [24][20].

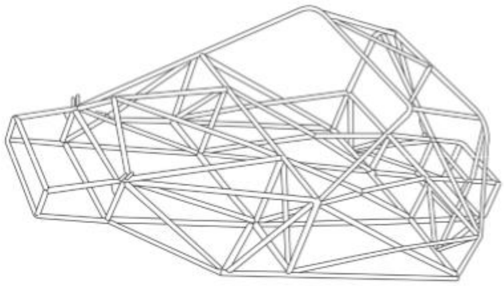


Figure 2.10: space frame

The frame is a structure consisting of one or more metal beams along the vehicle, connected by crossbars welded, screwed or riveted, arranged transversely or diagonally. This element has a high strength and the mechanical components and the bodywork are attached on it. The body forms the outer shell of the vehicle and is screwed to the frame. It has not any strength feature.

This system allows some advantages. It has a great strength to carry heavy loads and a high rigidity to withstand high elastic and dynamic stresses. This system also protects the bodywork from bending or torsion efforts. The body on frame system has a great facility to use different bodies in the same frame. Reforms in the body can also be performed easily. In all frames till now length in one dimension is very less compared to the other two dimensions. Increasing depth increases bending strength. Beam elements carry either tension or compressive loads.

Ladder frame

A ladder frame is the simplest and oldest frame used in modern vehicle construction. [19][20] It was originally adapted from "horse and buggy" style carriages as it provided sufficient strength for holding the weight of the components. Larger beams could be used if there were higher weight capacity required. Their constructions consist of two longitudinal rails interconnected by many lateral/cross braces, typically made from round or rectangular tubing or channel. The longitude members are the main stress member. They deal with the load and also the longitudinal forces caused by acceleration and braking. It can use straight or curved members. The lateral and cross members provide rigidity to the structure because it provides resistance to lateral forces and further increase torsion rigidity. Body mounts are usually integral outriggers from the main rails, and suspension points can be well or poorly integrated into the basic design.

2.5. Joining methods of the frame members

The processes that may be employed to join the side and cross- members are welding, riveting, and bolting. Welding gives the most rigid joint, but this method has so far been generally uneconomic for the production members involved, especially since machine welding would be required to guarantee the necessary weld uniformity of individual cross- members. Riveting is widely used for joining the frame members, and it may be done cold rather than hot so as to avoid the rivets contracting during cooling and developing clearances in their holes. Bolting is also popular and can be more effective than riveting as regards rigidity because the tightness with which the frame members are clamped together can be accurately controlled. A possible disadvantage is that screwed connections may loosen if not properly installed. A combination of riveting and bolting is sometimes employed [1].

2.6. Materials

One of the most important decisions is to choose the type of material to be used. Depending on the manufacturing process, the material used will vary. We can distinguish basically two groups of parts with different requirement in the vehicle structure. The structural beams, whose goal is to get a structure as strong and robust as possible, and the sheets, whose aim is, apart from providing resistance to the whole, enclose the interior volume to protect it from the outside [26][27].

2.6.1. Mechanical Properties of Metals

The mechanical properties of the metals are those which are associated with the ability of the material to resist mechanical forces and load. These mechanical properties of the metal include strength, stiffness, elasticity, plasticity, ductility, brittleness, malleability, toughness, resilience, creep and hardness [10].

2.6.2. Requirements of the materials in vehicle

The materials used in vehicle industry need to full fill several criteria before being approved. Some of the criteria are the results of regulation and legislation with the environmental and safety concerns and some are the requirements of the customers. In many occasions different factors are conflicting and therefore a successful design would only be possible through an optimized and balanced solution [2]

1. **Lightweight**

As there is a high emphasis on greenhouse gas reductions and improving fuel efficiency in the transportation sector, all car manufacturers, suppliers, assemblers, and component producers are investing significantly in lightweight materials research and development and commercialization. All are moving towards the objective of increasing the use of lightweight materials and to obtain more market penetration by manufacturing components and vehicle structures made from lightweight materials. Because the single main obstacle in application of lightweight materials is their high cost, priority is given to activities to reduce costs through development of new materials, forming technologies, and manufacturing processes. Rail vehicle manufacturers are investigating the reduction of the weight in a viable economical way [2].

2. **Cost**

One of the most important consumer driven factors in car body industry is the cost. Since the cost of a new material is always compared to that presently employed in a product, it is one of the most important variables that determine whether any new material has an opportunity to be selected for a vehicle component. Cost includes three components: actual cost of raw materials, manufacturing value added, and the cost to design and test the product. This test cost can be large since it is only through successful vehicle testing that the product and manufacturing engineers can achieve a level of comfort" to choose newer materials for application in a high-volume production program. Aluminium and magnesium alloys are certainly more costly than the currently used steel and cast irons that they might replace. The ability to approach the total cost of the competition, therefore, must be associated with lower component manufacturing costs. Compared to cast irons and steel, cast aluminium and magnesium components are potentially less costly. This is based on their reduced manufacturing cycle times, better much inability, ability to have thinner and more variable wall dimensions, closer dimensional tolerances, reduced number of assemblies, more easily produced to near net shape (thus decreasing finishing costs, and less costly melting/metal-forming processes).However, wrought aluminium and magnesium components are almost always more costly to produce than their ferrous counterparts. Since cost may be higher, decisions to select light metals must be justified on the basis of improved functionality. Government regulations mandate reductions in exhaust emissions, improved occupant safety, enhanced fuel economy, reductions in workplace emissions, increased safety requirements, for toxic materials handling and disposal [2].

3. Safety, crashworthiness

The ability to absorb impact energy and be survivable for the passengers is called the “crashworthiness” of the structure in vehicle [21]. There are two important safety concepts in vehicle industry to consider, crashworthiness and penetration resistance. Crashworthiness is defined as the potential of absorption of energy through controlled failure modes and mechanisms that provides a gradual decay in the load profile during absorption. However, penetration resistance is concerned with the total absorption without allowing projectile or fragment penetration.

The current trend of materials in car industry is towards replacing metal parts more and more by polymer composites in order to improve the fuel economy and reduce the weight of the vehicles. The behaviour of composite failure in compression is the opposite of metals. Most composites are generally characterized by a brittle rather than ductile response to load. While metal structures collapse under crush or impact by buckling and/or folding in accordion type fashion involving extensive plastic deformation, composites fail through a sequence of fracture mechanisms involving fibre fracture, matrix crazing and cracking, fibre-matrix de-bonding, de-lamination and interplay separation. The actual mechanisms and sequence of damage are highly dependent on the geometry of the structure, lamina orientation, and type of trigger and crush speed, all of which can be suitably designed to develop high energy absorbing mechanisms. Several aspects are considered in design for improved crashworthiness including the geometrical and dimensional aspects which have key role in different stages of crash. However the materials deformation and progressive failure behaviour in terms of stiffness, yield, strain hardening, elongation and strain at break are also very important in the energy absorption capacity of the vehicle. Their crashworthiness behaviour is of fundamental importance in the safety design of the whole vehicle because their plastic collapse is the mechanism that is used to dissipate the kinetic energy of the vehicle in an accident. The mechanism of plastic collapse should be reliable and its evolution during the crash regular so that the desired quantity of absorbed energy, a low load uniformity and the required level of deformation load can be achieved without increasing danger for the vehicle passengers [25].

2.6.3. Structural beam materials

Materials used for beams should be strong, so metals are the most adequate material. Common materials of structural beams are [1]:

- **Conventional steel**: It is a ductile material that can be manufactured with complex forms and it present good welding properties. It is very economical. Nevertheless, it is necessary to use greater thicknesses than other materials, which leads to a weight increasing.
- **HSS (High speed steel):** It has higher strength than conventional steel, so it is possible to reduce the weight and increase the rigidity of the structure. They are lightly more expensive and less ductile.
- **Boron alloyed steel**: This kind of steel might be formed by hot-stamping. This method allows good ductility during the process and high final rigidity. It is more expensive than conventional steel.
- **Aluminium**: It is lighter than steel and gets more strength with less weight. It presents better corrosion resistance. However is more expensive than steel and it is more difficult to weld. Many people think that the frame (chassis) which built from aluminium is the path to the lightest design, but this is not necessarily true. Aluminium is more flexible than steel. In fact, the ratio of stiffness to weight is almost identical to steel, so an aluminium chassis must weigh the same as a steel one to achieve the same stiffness. Aluminium has an advantage only when there are in very thin sheet sections where buckling is possible but that are not generally the case with tubing. The uses of aluminium and Fibre reinforced plastic (FRP) in designing the frame are helped to reduce overall vehicle weight, thereby reducing energy consumption as well.

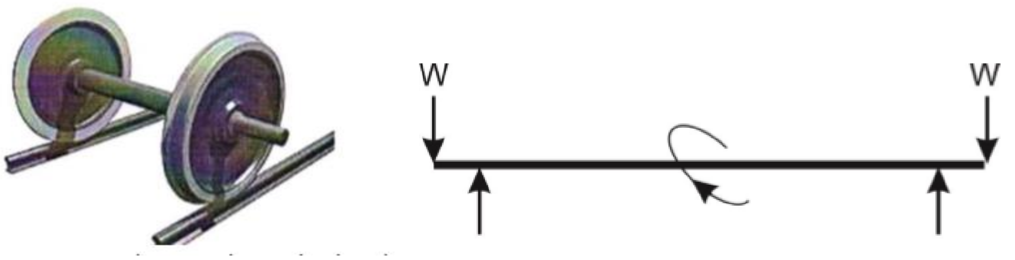
2.6.4. Exterior sheets materials

Materials used for external sheets have less structural requirements [1].

- **Plastic material**; It is a cheap material and easy to manufacture, but cannot be welded to the structure and do not provide any strength to the assembly.
- **Fiberglas**s; It is a light material, but harder than plastic. It cannot be welded to the structure and it is more expensive than plastic or sheet metal.
- **Sheet metal**; It is a cheap material, provides strength and can be welded the assembly. However, it is heavier than plastic.

2.7. Design for Static Load

A static load may be defined as an external force, which is gradually applied to a mechanical element, and does not change its magnitude or direction. See figure below [15] [23] [31]. The figure 2.11 (a) below shows axle and wheel on their tracks while figure (B) shows the load applied on the axle shaft as a result of train weight (W).



A) B)

Figure 2.11: A, shows railway axle and wheels and B shows bending moment under loading as a result of weight.

2.7.1. Strength and Modes of Failure

A primary consideration in designing any component is to make it strong enough so that it will not fail in service. Actual breaking of the component is no doubt a failure; however, in designing we consider a broader perspective that an element is considered failed as and when it is unable to perform its function satisfactorily, due to any one of the following modes:

- Failure by actual breaking or fracture (brittle material, ductile material under certain conditions).
- Failure by general yielding (ductile materials).
- Failure by elastic elongation or deflection. (columns, shafts)

Generally used theories for Ductile Materials (Failure by general yielding (ductile materials)) are:

1. **Maximum shear stress theory**

   States that failure occurs when the maximum shear stress from a combination of principal stresses equals or exceeds the value obtained for the shear stress at yielding in the uniaxial tensile test.

2. **Maximum Equivalent (von-misses) stress theory**

In this case, a material is said to start yielding when its von Mises **stress** reaches a critical value known as the yield strength. The von Mises stress is used to predict yielding of materials under any loading condition from results of simple uniaxial tensile tests.

2.7.2. Strength of Materials and Failure Theories

State of Stress

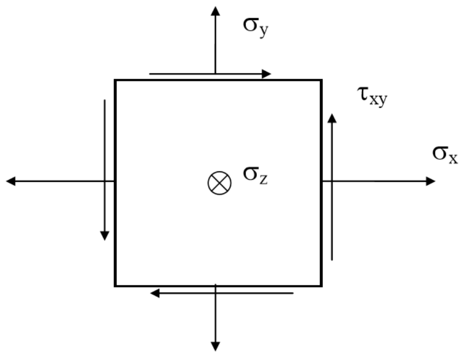


Figure 2.12: 2D state of stress

This is a 2D state of stress – only the independent stress components are named. A single stress component σ_z can exist on the z-axis and the state of stress is still called 2D and the following equations apply. To relate failure to this state of stress, three important stress indicators are derived: Principal stress, maximum shear stress, and Von-misses stress. See figure 2.12 above.

Principal stresses: The principal stresses are the components of the stress tensor when the basis is changed in such a way that the shear stress components become zero.

$$\sigma_1, \sigma_2 = \frac{\sigma_x+\sigma_y}{2} \pm \sqrt{(\frac{\sigma_x+\sigma_y}{2})^2 \pm {\tau_{xy}}^2} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (3.4)$$

Where, σ_1, and σ_2 are principal stresses, σ_x *and* σ_y are component stresses on their x and y respective axes and τ_{xy} is shear stress in the xy-plane.

The Von Mises stress: In this case, a material is said to start yielding when its von Mises **stress** reaches a critical value known as the yield strength. The von Mises stress is used to predict yielding of materials under any loading condition from results of simple uniaxial tensile tests.

$$\sigma_V = \sqrt{\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_1-\sigma_3)^2}{2}} \quad (3.5)$$

Where, σ_V is von-misses stress and σ_3 is principal stress.

Strain: a geometrical measure of deformation representing the relative displacement between particles in a material body.

Total strain definition

$$\varepsilon_{total} = \varepsilon_t = \frac{\Delta L}{L} \quad (3.6)$$

Where, ε_{total} is strain total, L is original length before deformation and ΔL is the change in length.

a. **Bending stress and bending moment**

Many structures can be approximated as a straight beam or as a collection of straight beams. For this reason, the analysis of stresses and deflections in a beam is an important and useful topic.

- Bending stress for bending about the Z-axis

$$\sigma_x = \frac{M_z y}{I_z}, \text{ and } M_z = F_y L \quad (3.7)$$

Where, M_z is bending moment about Z-axis, F_y is the load in the direction of Y-axis, L is the length in which the load is located and I_z is area moments of inertias about the Z and represents resistance to rotation about Z-axis.

- Bending stress for bending about the Y-axis:

$$\sigma_x = \frac{M_y z}{I_y}, \text{ and } M_y = F_z L \quad (3.8)$$

Where, I_y is area moments of inertias about the y and represents resistance to rotation about y axis.

b. Impact

i. Frontal impact

For cases of frontal impact, there has to be a frontal crush zone that will absorb and distribute the forces when the train crashes head-on, whether into another train, a large tree, or a bridge abutment. Structurally, the frontal impact should not be shoved rearward and impinge into the front-door hinges and door-latch system [10][17][21] and [25].

ii. Side impact

For side impacts, the train must have strong frame members that are at the outermost periphery of the vehicle body, or if the main structural frame members are further inboard, serve as extensions. The doors should have internal reinforcement beams, with strong hinges and door-latch system [10][17].

iii. Rear impact

For cases of rear impact, the seats should be designed with high backrests and integrated head restraints to help keep the head, neck, and upper torso in safe alignment during the dynamics of a crash [10][17].

c. Software

CATIA (computer aided three dimensional interactive application) is used to model the small train vehicle 3D drawing. It is used to do both part drawing and assembly drawing [18].

The Finite Element Method

The finite element method (FEM) is the dominant discretization technique in structural mechanics and it is used for seeking an approximated solution of the distribution of field variables in the problem domain that is difficult to obtain analytically. Known physical laws are then applied to each small element, each of which usually has a very simple geometry. A continuous function of an unknown field variable is approximated using piecewise linear functions in each sub-domain, called an element formed by nodes. The unknowns are then the discrete values of the field variable at the nodes.

The basic concept in the physical FEM is the subdivision of the mathematical model into disjoint (non-overlapping) components of simple geometry called finite elements or elements for short. The response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function, or functions, at a set of nodal points. The response of the mathematical model is then considered to be approximated by that of the discrete model obtained by connecting or assembling the collection of all elements.

Element Attributes

The procedure involves the separation of elements from their neighbours by disconnecting the nodes, followed by referral of the element to a convenient local coordinate system. After that we can consider generic elements: a bar element, a beam element, and so on.

From the standpoint of the computer implementation, it means that you can write one subroutine or module that constructs, by suitable parameterization, all elements of one type, instead of writing a new one for each element instance.

Dimensionality

Elements can have intrinsic dimensionality of one, two or three space dimensions. There are also special elements with zero dimensionality, such as lumped springs or point masses. The intrinsic dimensionality can be expanded as necessary by use of kinematic transformations. For example a 1D element such as a bar, or beam may be used to build a model in 2D or 3D space.

Nodes

Each element possesses a set of distinguishing points called nodal points or nodes for short. Nodes serve a dual purpose: definition of element geometry, and home for degrees of freedom. When a distinction is necessary we call the former geometric nodes and the latter connection nodes. For most elements studied here, geometric and connector nodes coalesce.

Degrees of Freedom

The element degrees of freedom (DOF) specify the state of the element. They also function as “handles” through which adjacent elements are connected. DOFs are defined as the values (and possibly derivatives) of a primary field variable at connector node points.

A simple definition of "degrees of freedom" is - the number of coordinates that it takes to uniquely specify the position of a system.

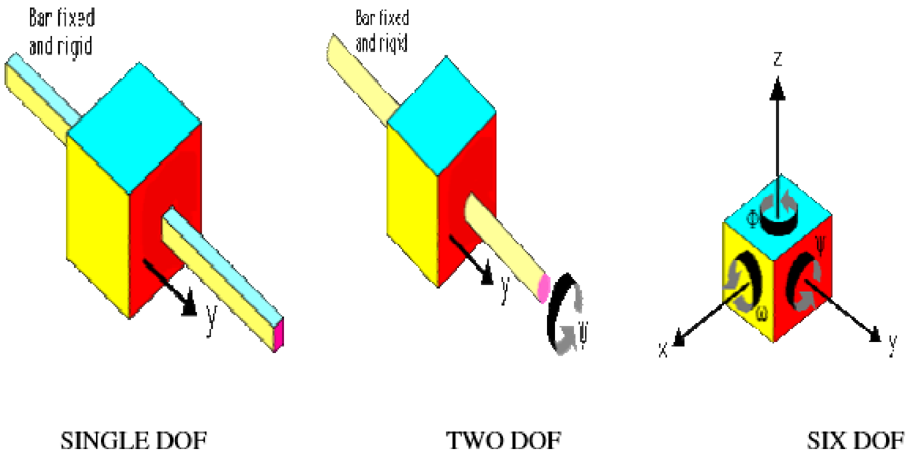


Figure 2.13: degree of freedom

For mechanical elements, the primary variable is the displacement field and the DOF for many (but not all) elements are the displacement components at the nodes.

If the number of degrees of freedom is finite, the model is called discrete, and continuous otherwise. Because FEM is a discretization method, the number of DOF of a FEM model is necessarily finite. They are collected in a column vector called **“u”**. This vector is called the DOF vector or state vector. The term nodal displacement vector for **u** is reserved to mechanical applications.

Nodal Forces

There is always a set of nodal forces in a one-to-one correspondence with degrees of freedom. In mechanical elements the correspondence is established through energy arguments.

Assembly

The assembly procedure of the Direct Stiffness Method for a general finite element model follows rules identical in principle to those discussed for the truss.

d. Modelling

Traditionally, formal modelling of systems has been via a mathematical model, which attempts to find analytical solutions to problems and thereby enable the prediction of the behaviour of the system from a set of parameters and initial conditions. While computer simulations might use some algorithms from purely mathematical models, and can combine simulations with reality or actual events, such as generating input responses, to simulate test subjects who are no longer present. Many research works had been done on vehicle structures, computer aided design of various engineering structures and assembly. And the unique specialty of the work is the integration of computer aided design, modelling and simulation through the use of relevant computer programs to aid the assembly procedure of train vehicle structure. This will help the accuracy, save time and make it flexible thereby improving quality and efficiency of the assembly procedure [1].

CHAPTER THREE

3. MATERIALS AND METHODS OF THE STUDY

3.1. General introduction

Generally, the study of this paper is carried out through the following steps.

Literature review: A lot of paper and journal has been read and a part of it are considered in this thesis. Also, different reliable websites, and other relevant literature review is done.

Data collection: reading the previously designed manufactured mini-electric train in ASTU, visit to Addis Ababa light rail transportation and measuring each dimension of the mini-electric train components of the structure.

Modelling of the structure: this is made by using the data recorded from the existing mini-electric train built and collected data. CATIAV5-R20 software is used to build the 3D model.

Finite element (FE) analysis: analysing of structure for the required results using ANSYS Workbench 15.0 software. Finally, the vehicle structure is analysed based on the ANSYS result of static structural analysis, i.e. stress and displacement.

Materials required: Materials used to accomplish this research are software like, CATIA and ANSYS workbench ; measuring instruments like, meter; Flash, digital Camera; stationary materials; books and journals; Computer and other complimentary services.

The following figure 3.1 shows the entire procedure of the work.

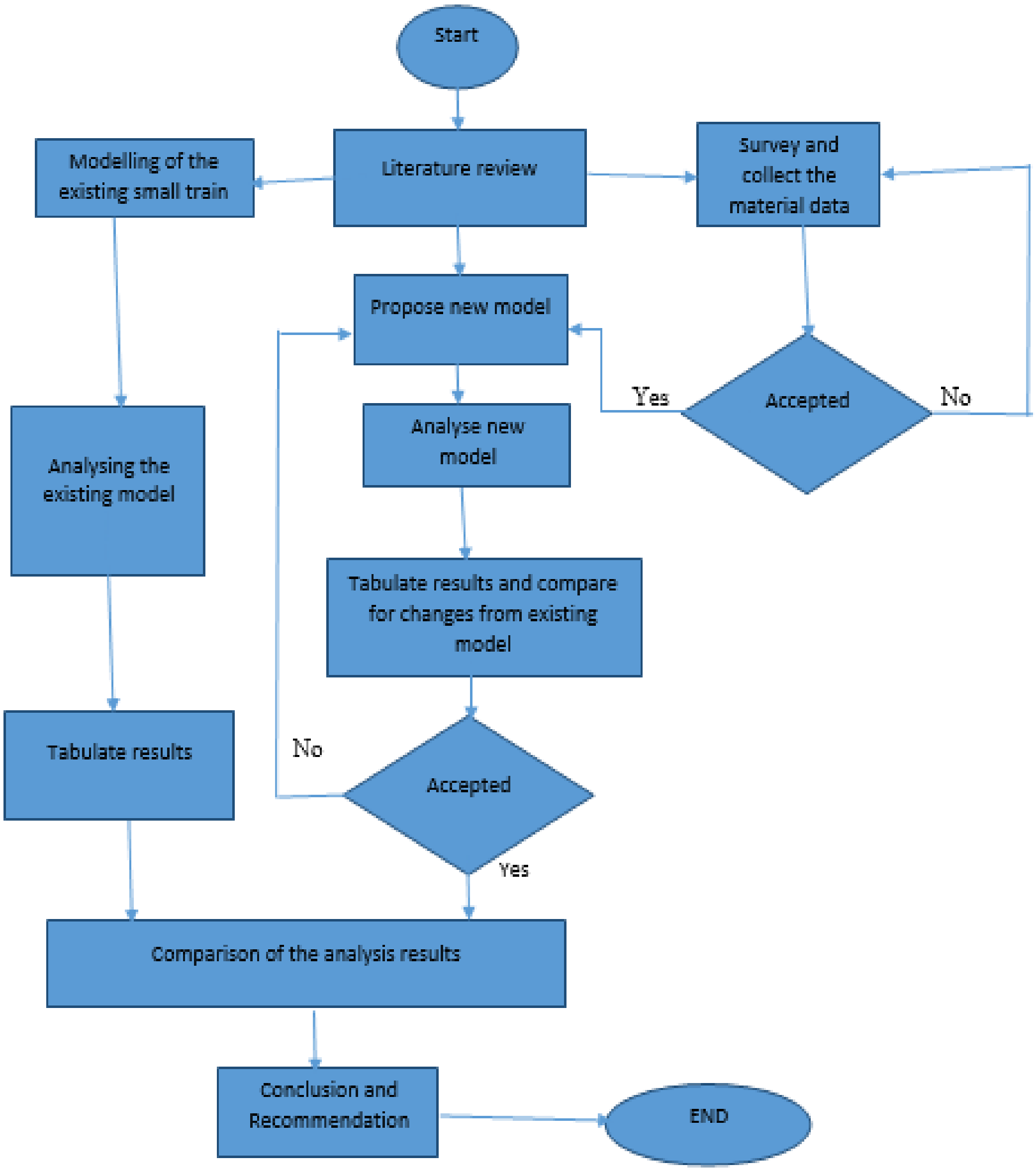


Figure 3.1: Flow diagram for analysis

3.2. Introduction to small train body modelling

To define the car body structure, a structural model is developed by using CATIAV5-R20 modelling and analysis software. This model was imported with a file format such as IGES, or ACIS to ANSYS-15.0 for final analysis. The purpose of the structural sketch is to model the primary train body structure in advance of formal stress analysis and structural drawings. With the basic understanding of how finite element programs work, a finite element model must be created with appropriate parameters such as dimensions, loads, constraints, element choice, mesh selection, etc.

A computer aided design (CAD) or finite element (FE) analysis can give the mass, centre of gravity position, and mass moments of inertia. However, in most applications the mass of the car body structure is less than half of the car body mass. The corresponding mass might also be combined with the car body structure model [2].

3.3. Modelling and analysis of the existing small train vehicle

Here, the frame and body are built as one unit. In this project, small train built and manufactured in Adama Science and Technology University is considered. The main dimensions/specification of car body of rail vehicle that are taken from the builder is as follows in table 3.1 below.

Table 3.1: Basic dimension of the mini-electric train

Body and chassis	
Overall length	2000mm
Width	1000mm
Height	1600mm
Wheel base	660mm
Curb weight	2300N
Maximum top speed	25 km/h
Electrical components	
DC traction motor	½ HP, 1800rpm, 200V
DC drive control card	1 hp,220V
Invertor	100W
Convertor	100W, 12-40V out put
Circuit breaker	25A, 200-250V

Table 3.2: Materials used for the existing mini-electric train as per the researcher selection.

No.	Parts	Quantity	Material	Dimension
1	wheel	4 pcs	Mild steel	Diameter (D=150mm)
2	axle	2 pcs	Mild steel (round hollow tube)	D_o D_i ,L (32mm, 27mm, 740mm)

3	Angle pipe for the structure	24 meters	Steel, RHS	40*40*1mm thickness
4	Body front, rear and both sides cover	4 pcs	Sheet metal, aluminium	1 mm thickness
5	Body top cover	2.5 meters	Plastic sheet	1 mm thickness
6	Pulley	2	Aluminium cast	Motor side (D1=70mm) Axle side (D2=350mm)
7	Bearing	2	steel	6308-2Z, ball bearing

3.3.1. CATIA Modelling of existing small train vehicle

Any problem becomes complex, if the real situations are considered, it becomes very difficult to analyse the problem with such complexities. In order to simplify the problem, some assumptions have to be made. In the present analysis the following assumptions are considered.

1. The wheel are considered as solid disc with stiffness in vertical direction alone.
2. The mass of the DC motor and other components are lumped at appropriate nodes.
3. Some small members which do not have any significant load carrying function are neglected.

The CATIA V5R20 is used for model of the existing small train vehicle. It is made by using cross-section of 40x40x1mm steel beam rectangular hallow section (RHS).

Generally the model has been done using the overall dimensions of 2000mm Length, 1000mm

Width and 1600mm Height. Sheet metal made of aluminium with 1mm thickness and plastic material is used for both sides, front and rear sides of the train, and for roofing purpose have been modelled in CATIA. Finally each part assembled together before going to finite element analysis.

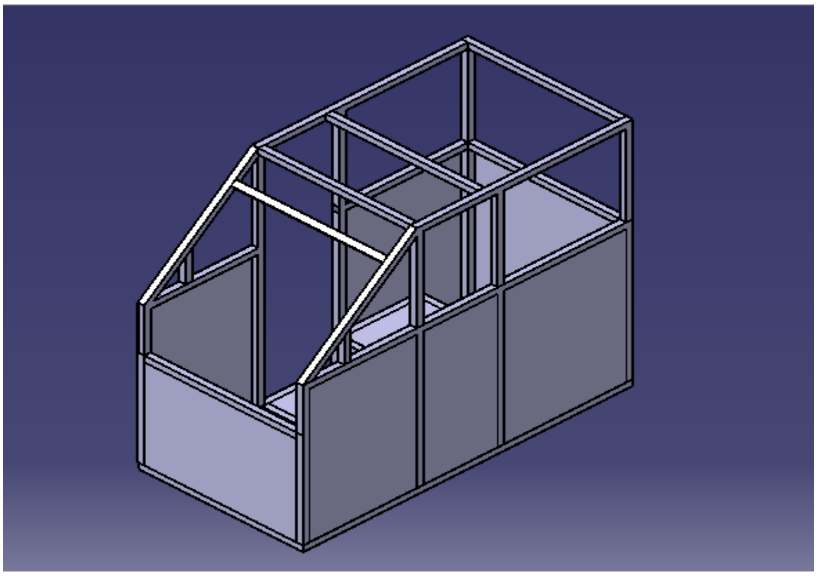


Figure 3.2: 3D CATIA drawing of the existing small train structure

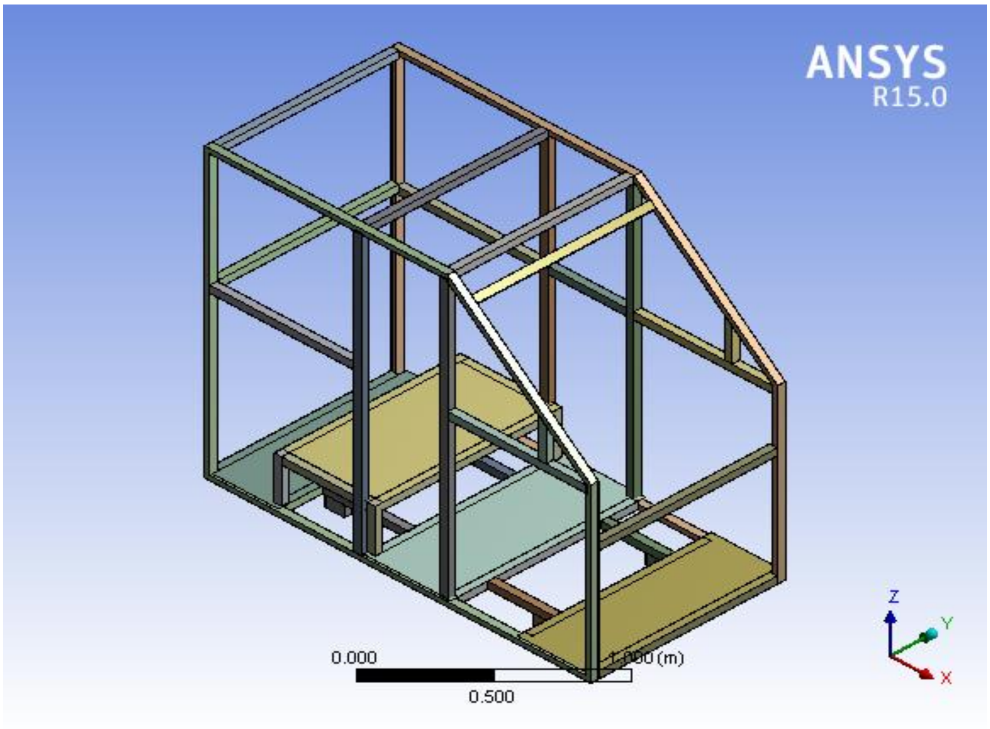


Figure 3.3: ANSYS 3D structure of the existing small train imported from CATIA

3.4. Finite element analysis of the existing small train structure

The reason for using a FEA software program is to analyse and compare the static structural and impact simulation of varying designs, with the process of transferring the model from the CAD program CATIA to the FEA program ANSYS. Next, proper principles are followed to establish equations for the elements, after which the elements are tied to one another. This process leads to a set of linear algebraic simultaneous equations for the entire system that can be solved easily to yield the required field variable.

The FEA of the model, once imported into ANSYS is a simple process, consisting of a number of simple steps;

1. selecting the element type(s) to use,
2. Inputting material properties,
3. Entering real constants, thickness values or section properties,
4. Meshing elements
5. Applying loads and boundary conditions,
6. Running Analysis and viewing results.

3.4.1. Materials selection for the existing mini-electric

Material selection plays crucial part in providing the desired strength, endurance, safety and reliability to the vehicle [10]. To choose the optimal material we need an extensive study on the properties of different materials. The material for the existing small train vehicle model is the structural steel ASTM 106a grade B with the following parameters shown in table below.

Table 3.3: Mechanical properties of mild steel

No	material	Yield strength	Ultimate tensile strength	Young's modulus(E)	Poisson's ratio(v)	Elongation
1	structural steel	250 Mpa	460 Mpa	200 Gpa	0.3	20%

3.4.2. Static analysis

A static strength analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. This analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. For carrying out the FEM Analysis of body as per standard procedure first it requires to create merge part for assembly to achieve the connectivity and loading, and constraining is required to be applied. Simplified CAD model of existing mini-electric train vehicle frame is created using CATIA and it is imported in ANSYS as an external geometry file.

Meshing

An important process of any FEA is to refine the mesh that is used in the analysis, a process that involves reducing mesh or element size a number of times and plotting the results as to ensure the analysis is stable. The mesh influences the accuracy, convergence and time required for analysis. Furthermore, the time it takes to create a mesh model is often a significant portion of the time it takes to get results from an ANSYS solution. Tetrahedral mesh elements are used in meshing of the small train vehicle structure. A tetrahedron element is 3-D solid element which has four triangle faces and six triangle cross-sections.

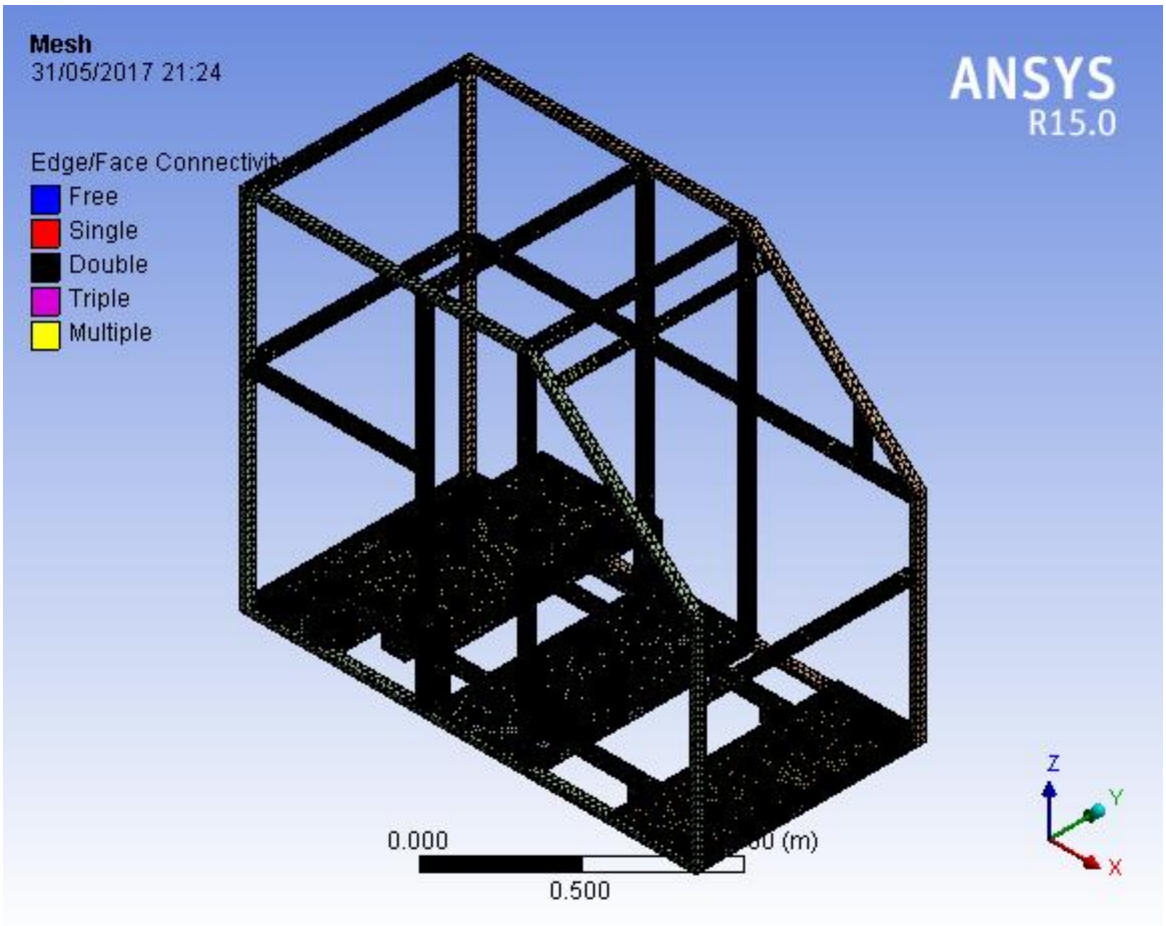


Figure 3.4: meshing of the existing mini-electric train structure

Element and Nodes

The meshed existing small train model has number of elements and number of nodes. The element is tetrahedral. In order to get a better result, locally finer mesh is applied in the region which is suspected to have the highest stress. Finally, 161803 numbers of Nodes and 37887 numbers of Elements has been found.

3.4.3. Loading and Boundary condition

Total weight, the weight of the mini-electric rail car having all components without occupant or baggage is, 230 kg or 2254.3818N. This will be applied at the centre of gravity of the frame. And Maximum payload including driver is 195kg applied at seat points of each passenger and driver. For each will have 65kg (637.108N) weights. The small mini-electric train and frame structure model is loaded by static forces from its own body and load from passengers, driver, and other baggage weights. For this model, the maximum loaded Gross vehicle weight is 425 kg (including curb weight, driver and passengers). The load is assumed as a uniform distributed obtained from the maximum loaded weight divided over the total length of chassis frame.

Maximum axle weight is calculated as [13]:

$$\text{Weight transferred to front/rear axle} = \frac{\text{(Total weight) x (Distance C.G. is ahead of the rear/front axle)}}{\text{(Wheelbase)}}$$

We have inertia centre from CATIA drawing as x=905.287mm, y= 465.675mm and z= 487.312mm, wheel base, b=1200mm and total weight, W=425kg.

Distance C.G of the rear axle, a_2=505mm, distance C.G of the front axle, a_1=695mm and height of the centre of gravity from the ground is H=487mm

Now, weight transferred to front axle = $\frac{W \times a2}{b} = \frac{425 \times 9.8066 \times 0.505}{1.2}$ = 1754 N

Weight transferred to front axle = $\frac{W \times a1}{b} = \frac{425 \times 9.8066 \times 0.695}{1.2}$ = 2412 N

Where, W is gross vehicle weight, b is wheel base and C.G is centre of gravity

Then the maximum front axle weighs 877N each, which is the part of the Gross vehicle weight that loaded on front wheel axle left and right wheels, so that it uses as a vertical reaction force at each front wheels. Maximum rear axle weight is 1206N each, which is the part of the Gross vehicle weight that loaded on rear left and rear right wheels, so that it uses as a reaction force. Earth gravity is also considered for the chassis frame as a part of loading. The location of the CG of the driver, passengers and small train frame are calculated using the small mini-electric train vehicle drawing in the x, y and z coordinate system. These are as follows in x, y and z axis order.

- Driver load is 65 kg ×9.8066= 637.429N @ [250, 500,400] mm
- Passenger 1 load is 65 kg ×9.8066= 637.429N @ [890, -330, 400] mm
- Passenger 2 load is 65 kg ×9.8066= 637.429N @ [890, -660 , 400] mm
- Body standard gravity at C.G is 9.8066m/s^2 @ [905., 466,487] mm

There are four displacement boundary conditions of model. The first and second boundary conditions are applied in front part of the chassis frame, the third and the fourth boundary conditions are applied in rear part of chassis frame. For all displacement boundary conditions the Y component of the displacement had set to zero i.e. assuming vertical loads only to test for deformation of the structure as we can see in figure 3.5 below.

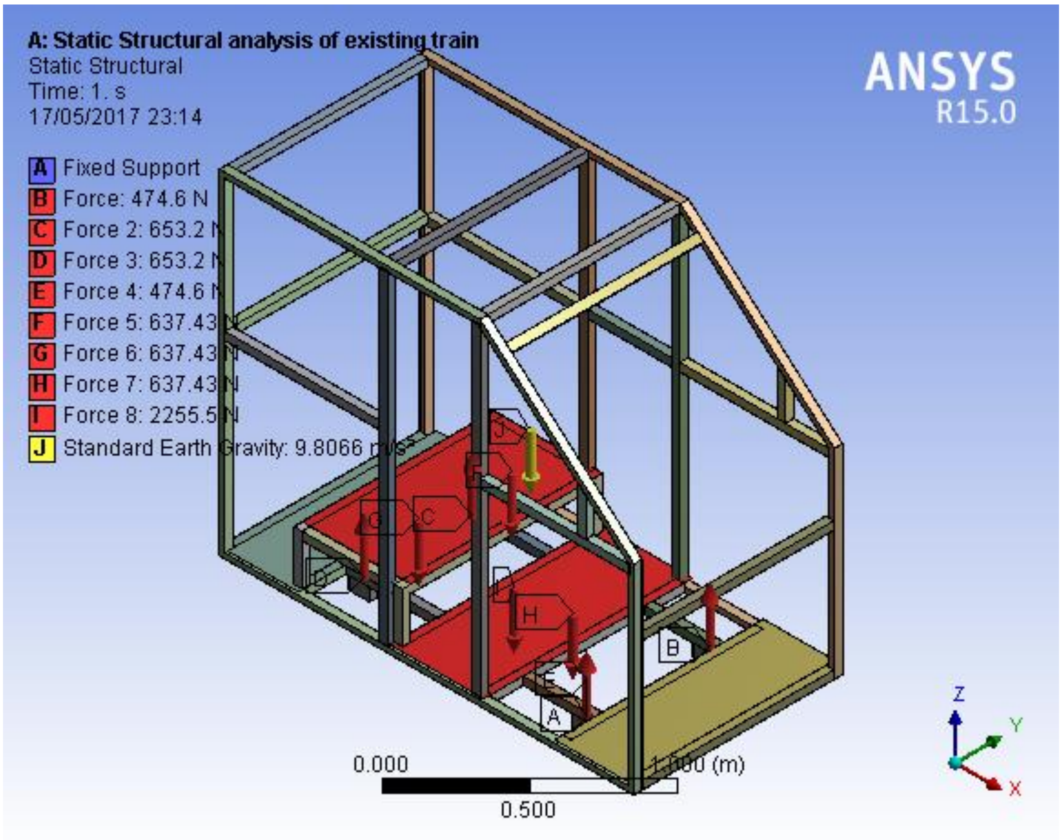


Figure 3.5: loads and boundary conditions of the existing model

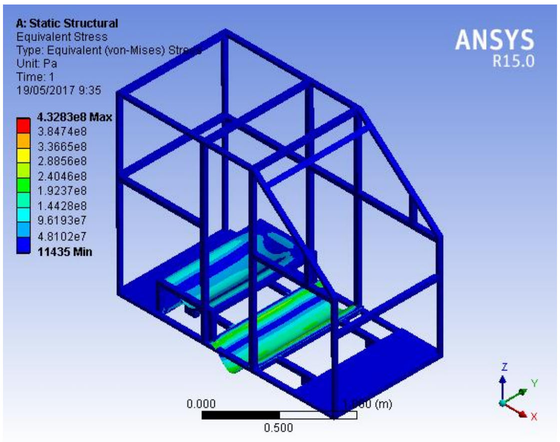
3.4.3.1. Simulation Results

The overall solution progress is indicated by a status bar. In addition we can use the Solution information object which is inserted automatically under the Solution folder. This object allows us to view the actual output from the solver, graphically monitor items such as convergence criteria for nonlinear problems and diagnose possible reasons for convergence difficulties by plotting Newton-Raphson residuals.

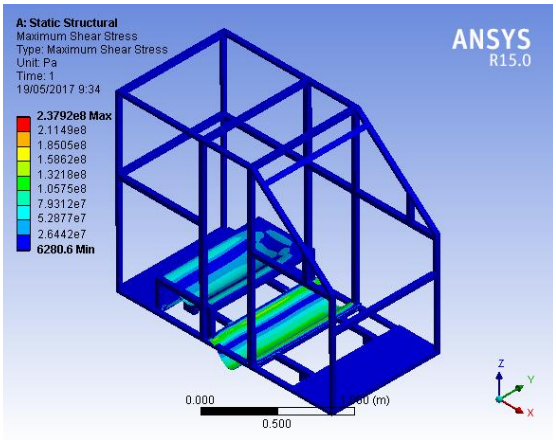
The analysis type determines the results available for us to examine after solution. For this project the analysis type is structural analysis and we are interested in equivalent stress results or maximum shear results. The result types section lists various results available to us for post processing. In this case the solution shown below in table 3.4 is obtained.

Table 3.4: simulation results of the static structure analysis

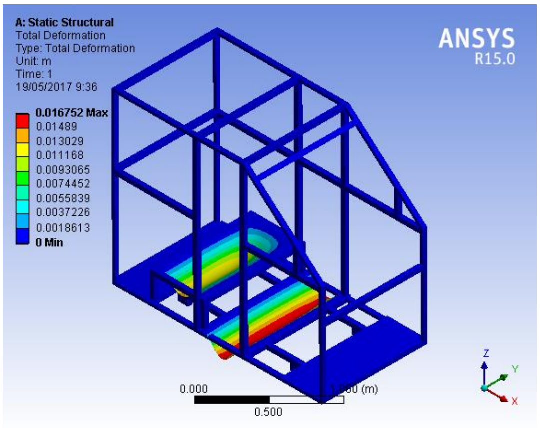
Result type	Equivalent or Von-misses stress in MPa	Maximum shear stress in MPa	Maximum principal stress in MPa	Safety factor	Total deformation in m	Directional deformation in m
Minimum	1.1435e-02	6.2806e-02	-50.396	0.5776	0	-4.5805e-004 m
Maximum	432.83	237.92	449.21	0.5776	1.6752e-002	1.156e-004



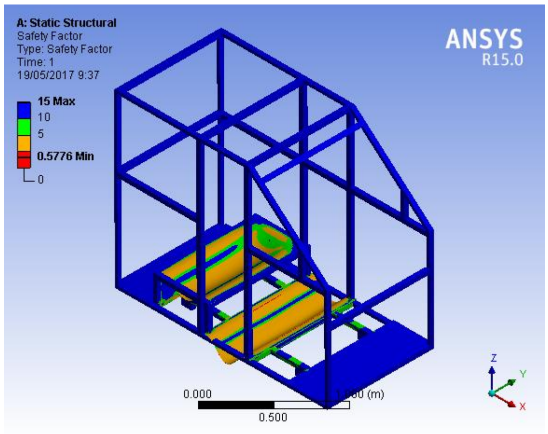
A)



B)



C)



D)

Figure 3.6: results of A) equivalent stress B) maximum shear stress C) total deformation D) Safety factor respectively

Maximum Von Misses stress and maximum shear stress

The Maximum Equivalent Stress safety tool is based on the maximum equivalent stress failure theory for ductile materials, also referred to as the von Mises-Hencky theory. The theory states that 'a particular combination of principal stresses causes failure if the maximum equivalent stress in a structure equals or exceeds a specific stress limit'. The Maximum Shear Stress safety tool is based on the maximum shear stress failure theory for ductile materials. The theory states that 'a particular combination of principal stresses causes failure if the Maximum Shear equals or exceeds a specific shear limit, where the limit strength is generally the yield strength of the material'.

From the table 3.4, the maximum Von Misses stress magnitude obtained is 432.83 MPa and the maximum shear stress magnitude is 237.92 MPa of critical point is around the floor, passenger and driver seat surfaces of the small train structure as shown in figure 3.6 (a) and (b). But factor of safety is 0.5776 as shown in figure 3.6 (D) and this is not reliable value. The floor frame is not in safe mode to resist the load applied, i.e. the stress results (von-misses stress) is above frame material yield strength.

Displacement results

The maximum displacement of frame structure is 14.89 mm obtained around the passengers and driver seat (welded sheet metal over frames). The magnitude of displacement is moderate, but passengers and driver seats should not be deformed that much, hence the model needs some improvement around the seats and floor thickness.

3.4.4. Impact simulation

The force applied on the front head direction is pretty big, because it is simulating a front impact.

Then we need a very tough structure to resist this kind of impact. We can get the result of the test that the maximum displacement under these conditions is occurred by using the following dynamic load. For a perfectly inelastic collision, energy transferred is given by the following equation.

$$E_t = \frac{m_1 \times m_2 \times (u_1 - u_2)^2}{2 \times m_1 \times m_2} = \frac{425 \times 425 \times (\frac{7m}{s})2}{2 \times (425+425)} = 5206.25\text{Nm}$$

Where, m_1 & m_2 are the two colliding vehicle masses with velocities u_2 & u_1 respectively. Since both m_1 & m_2 are two small train vehicles with similar masses & the vehicle m_2 is at rest. Implies that, $m_1 = m_2$ & $u_2 = 0$ then,

$$E_t = 5206.25\text{Nm}$$

Now, $F = \frac{Et}{t}$ where, t is impact time which is taken for 1second [1] and F is impact force.

$$F = 5206.25\text{N}$$

Also the design is for no plastic deformation, the vehicle should remain in the elastic region. The design factor of safety should remain above unity taking in to account uncertainty in the nature of forces.

1. **Front impact FEM analysis**

To do the FEA analysis of the front Bulkhead, we are going to apply the load of 5206.25N at the front in addition to static loads we have seen before in the static condition analysis. The point of application of these loads will be the actual attachment points between the impact front heads (figure 3.7). The frame will be fixed displacement but not rotation of the bottom nodes of both sides and both locations. Model used is full model of the small train vehicle with boundary conditions at bearing mounting points $U_z = U_y = 0$ and at rear corner points all DOF=0.

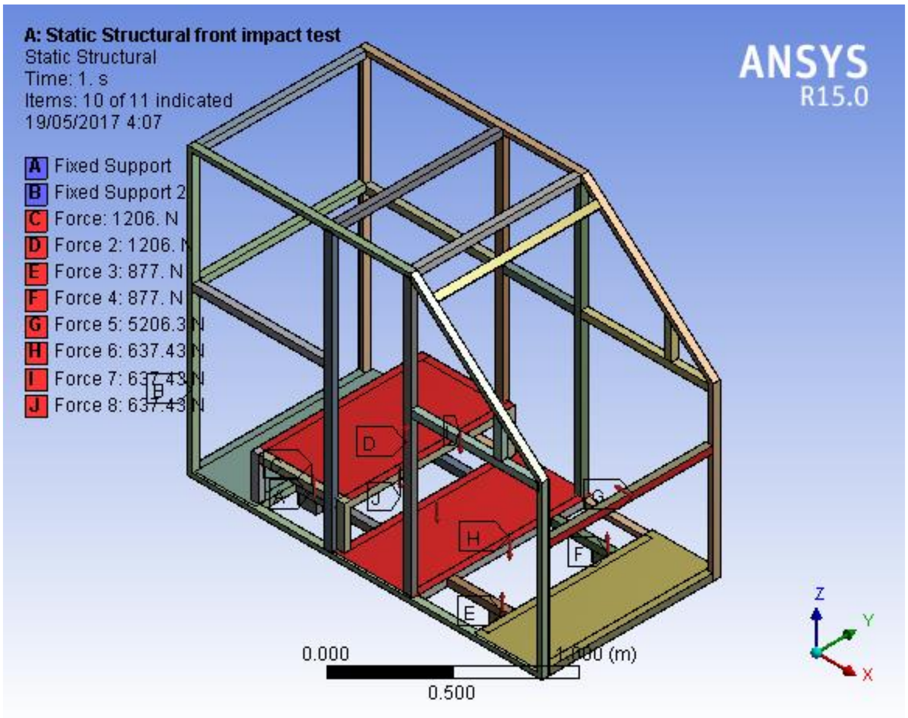


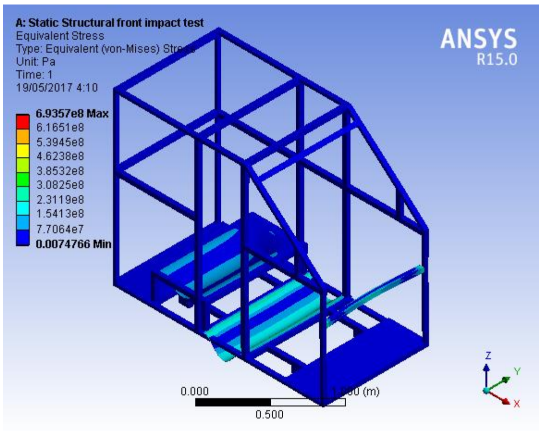
Figure 3.7: loads and boundary conditions for front impact

Stress results

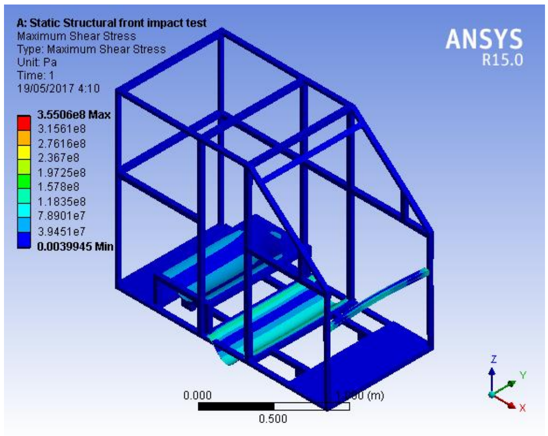
Since the force applied on the front frame is pretty big, the location of maximum Von Misses stress and maximum shear stress are at front. The Von Misses stress magnitude of critical point is 693.57 MPa and the maximum shear stress magnitude is 355.06 Mpa as we see from table 3.5 below. The result of maximum displacement under these conditions is 16.325 mm, which is totally unacceptable. This result has happened because the floor sheet metal is less in thickness and not giving a high rigidity to the frame. Safety factor is 0.36045. Hence; the structure will not safe under Frontal Impact. Figure 3.8 (d) below shows in a very clear way.

Table 3.5: simulation results of front impact case

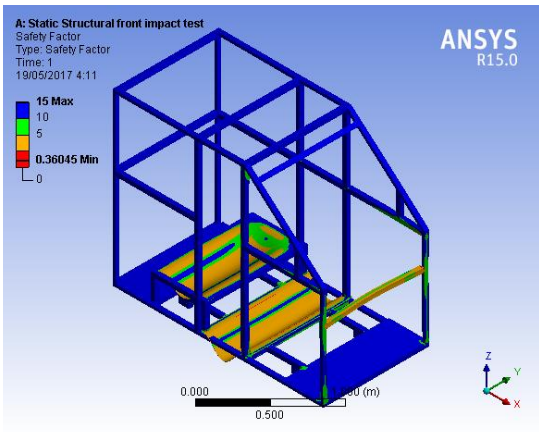
Analysis type	Equivalent (von-Mises) Stress in MPa	Maximum Shear Stress in MPa	Maximum principal stress in MPa	Safety factor	Total deformation in m	Directional deformation in m
Minimum	7.4766e-009	3.9945e-009	-52.113	0.36045	0. m	-4.1932e-003
Maximum	693.57	355.06	446.81	0.36045	1.6325e-002	1.4606e-004



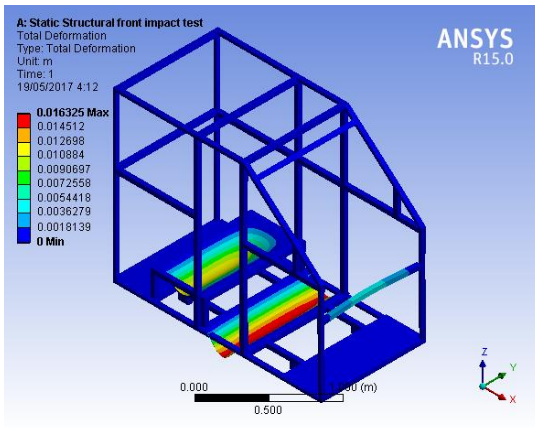
A)



B)



C)



D)

Figure 3.8: shows A) equivalent stress B) maximum principal stress C) safety factor D) total deformation of front impact test respectively

2. **Rear impact FEM analysis**

To do the FEA analysis of the rear head impact, we are going to apply the load of 5206.25N at the rear (figure 3.9) in addition to static loads we have seen before in the static condition analysis.

The boundary conditions are used at Suspension mounting points Uy = Uz = 0 and at front corner points all DOF=0.

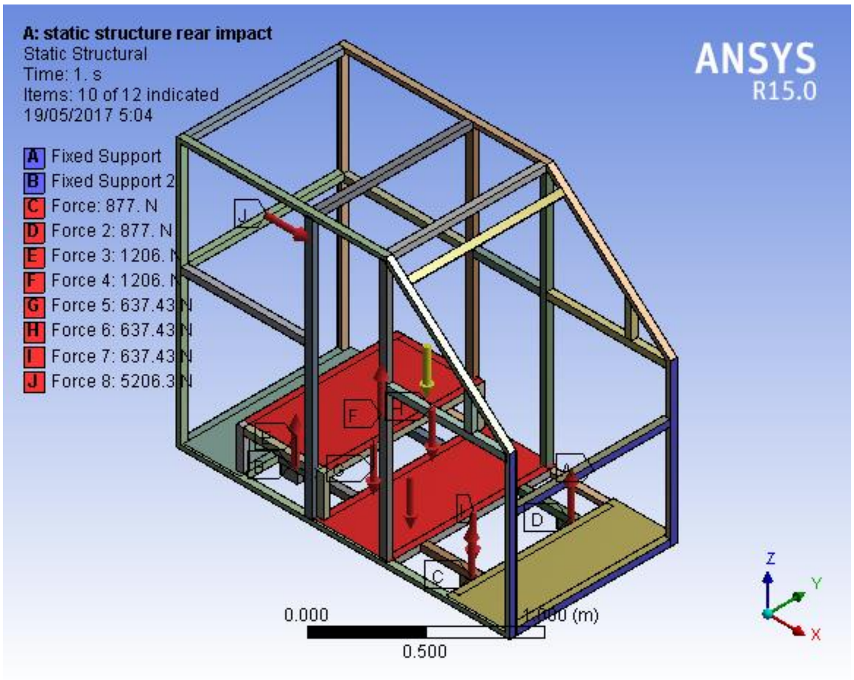


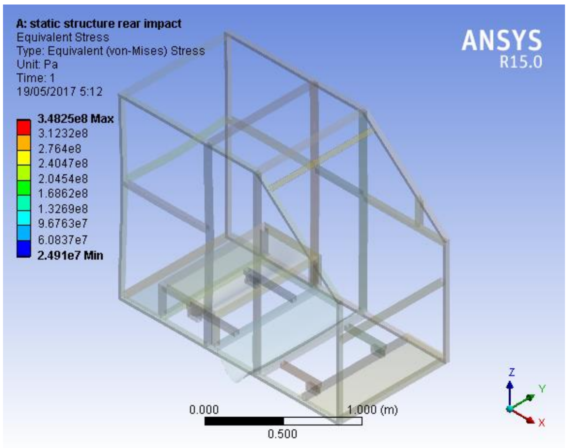
Figure 3.9: loads and boundary conditions for rear impact

Simulation results

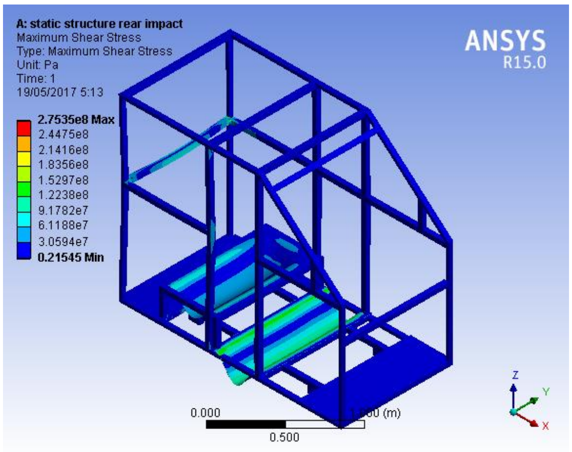
The location of maximum Von Misses stress and maximum shear stress are at rear. Table 3.6 shows the Von Misses stress magnitude of critical point of 351.35 MPa and the maximum shear stress magnitude is 277.62 Mpa due to Impact. As we can see the table, the result of the test is that the maximum displacement under these conditions is 13.302 mm, which seems good compared to front impact, but it is better to minimize it. In this result, the frame could not resist the load, i.e. has not enough rigidity. Safety factor is 0.48001 shown in figure 3.10 (d), hence, the structure will not safe under rear impact.

Table 3.6: simulation results of rear impact case

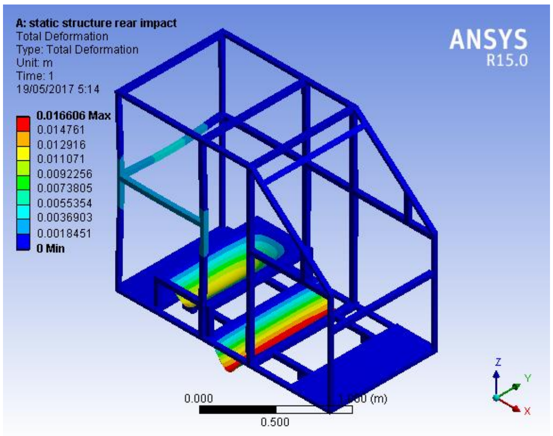
Analysis type	Equivalent or von-misses stress in MPa	Maximum shear stress in MPa	Maximum principal stress in MPa	Safety factor	Total deformation in mm	Directional deformation in mm
Minimum	24.707	0.12863e-006	-29.013	0.48001	0	-1.7573e-004
Maximum	351.35	277.62	370.89	0.48001	1.3302e-002	4.2018e-003



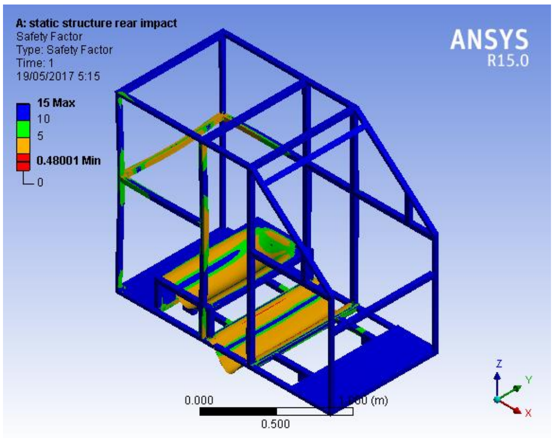
A)



B)



C)



D)

Figure 3.10: shows A) equivalent stress B) maximum shear stress C) total deformation D) safety factor of rear impact test respectively.

3. Side impact FEM analysis

To do the FEA analysis of the side impact, we are going to apply the Load of 5206.25N at the side in addition to static loads we have seen before in the static condition analysis. As shown in figure 3.11, the point of application of these loads will be the actual attachment points between the impact front and side heads. Boundary conditions applied at bearing mounting points are Ux = Uz = 0 and at rear corner points all DOF are zero.

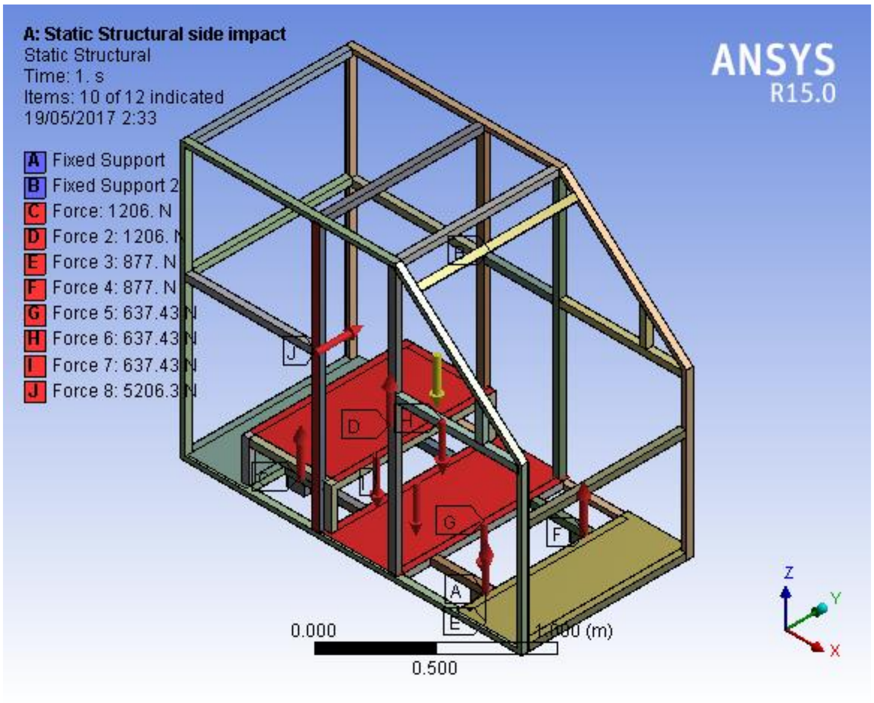


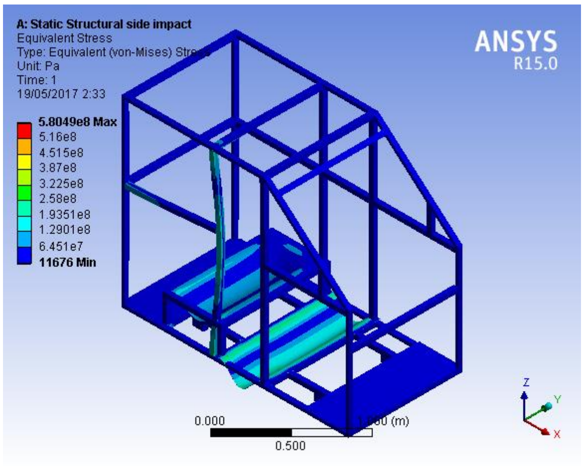
Figure 3.11: loads and boundary conditions for side impact

Simulation results

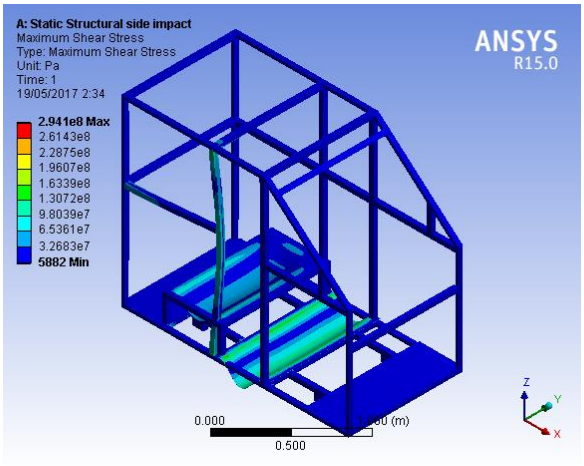
The location of maximum Von Misses stress and maximum shear stress are at sides around passenger's seat right side. The Von Misses stress magnitude of critical point is 580.49 MPa and the maximum shear stress magnitude is 294.1 MPa obtained. As we can see, the result of the test is that the maximum displacement under these conditions 16.718 mm, which is totally unacceptable. Safety factor is 0.43067. Hence, the structure will not safe under side impact. See table 3.7 and figure 3.12 for details.

Table 3.7: simulation results of side impact case

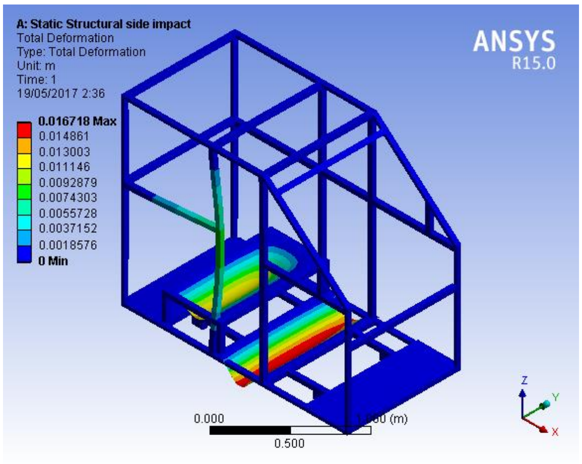
Analysis type	Equivalent (von-Mises) Stress in MPa	Maximum Principal Stress in MPa	Maximum Shear Stress in MPa	Safety factor	Total Deformation in m	Directional Deformation in m
Minimum	1.1676e-002	-60.617	5.882e-003	0.43067	0. m	-4.271e-004
Maximum	580.49	465.43	294.1	0.43067	1.6718e-002	2.4111e-004



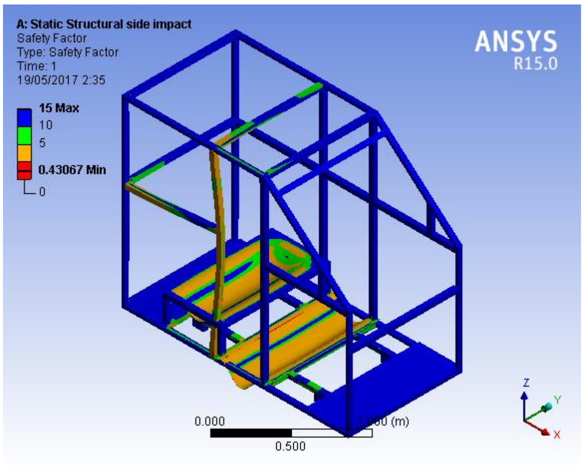
A)



B)



C)



D)

Figure 3.12: shows A) equivalent stress B) maximum shear stress C) total deformation D) safety factor

3.5. Result discussion and conclusion for the existing model

The highest von-miss stress occurred is 693.57 MPa by FEM analysis in case of front impact. The material yield strength is 250 MPa. The result of FE analysis is bigger 36% than the material strength and the minimum safety factor is 0.36045 which is at the front impact of the structure. The maximum displacement of numerical simulation result is 16.718 mm in case of side impact. These values tell us that, the model could not resist the crash loads and could not give a protection to the passengers and to driver also. It indicates that it is necessary to model a new small train vehicle structure.

3.6. Modelling and analysis of new small train model

3.6.1. Frame design criteria

A frame of a vehicle can be defined as a structure that has the finality to connect in a rigid way and at the same time brings anchorage points for the different parts of the vehicle to protect the driver and passengers in case of impact or rollover. The design of a frame is a commitment between rigidity, weight and volume, without forgetting the final economic cost have to be considered. In this thesis, we only takes into account the static stresses and the impacts, leaving for a higher level of complexity the fatigue calculations. In the frame design, regarding to stiffness, there are some aspects to take in to count [2].

- Items that are not part of the structure that give stiffness to the frame.
- Although is interesting to the frame to allow deformation due to an impact, the part that protects the legs has to be totally rigid.

Rigidity criteria

We must differentiate between bending stiffness and torsion stiffness. The first refers to the deformation of the frame under the weight of the different elements. Not a problem while designing the frame. The second refers to the frame deformation when it is under an asymmetric load, such as when one wheel goes through a hole or turning.

Several studies show that the deformation due to axial forces is much smaller than due to bending moments and torsion. That is why it is better to load the bars with axial efforts than with bending moments or torsion moments. Regard with the axial effort type, is better the compression efforts than the traction efforts to avoid buckling problems [2].

Mass distribution criteria

- It is interesting to maintain the rigidity of the frame, trying to get the overall weight of the frame lower as possible to take advantage of the motor power.

Space criteria

- Leave space for the DC motor, pulley, belts and other components.

Cost criteria

- The selection of the bars has not to be very varied in terms of diameter and thickness
- Will be bending the low number of bars and trying to bend only bars of one determinate diameter.

- Minimum number of joints.

In general the requirements of bodies for various types of vehicles, i.e. the body of the most vehicles should fulfil the following requirements [9]:

- The body should be light.
- It should have minimum number of components.
- It should provide sufficient space for passengers and luggage.
- It should withstand vibrations while in motion.
- It should offer minimum resistance to air.
- It should be cheap and easy in manufacturing.
- It should be attractive in shape and colour.
- It should have uniformly distributed load.
- It should have long fatigue life
- It should provide good vision and ventilation.

3.6.2. Basic decisions

Some basic decisions must be taken at the beginning of the project development, such as the type of structure, the manufacturing processes and the materials used. These three decisions must be taken together. The materials must be adequate to use the manufacturing processes selected.

Type of structure: Space frame structure is selected. Using this structure the small train will be safer and adequate for mass production.

The space frame design is a very popular option for chassis type, as it is a low cost, simple to design and easy to construct. The design is governed by the Pakistan Standards and Quality Control Authority (PSQCA) for three wheeler auto vehicles as our structure is also similar to this kind of vehicle because of low weight. PSQCA rules, although they leave enough freedom to see a wide verity of space frame shapes, sizes and design techniques.

One of the main advantages of using the space frame design is its easy and logical construction process, of which can be performed by person with intermediate knowledge and experience using basic welding and metal working equipment [1]. Generally, the space frame is the most suitable frame type used in the model car construction in world compared to others chassis types, hence we decide to apply this concept in this thesis.

There are many advantages obtained from space frame. Since the space frame systems are triangulated format, it will provide maximum strength and minimum deflection of the design compared to the other chassis types due to the support from tubular pipes [24]. Space frame chassis systems are lighter than traditional steel. Therefore, it provides significant economy in foundation costs. Using a space frame chassis in a small train car, the high torsion rigidity can be achieved as well as its light weight. It means that, space frame chassis designs will enhance the rigidity per weight ratio.

3.6.3. New small train vehicle 3D CATIA model

The CATIA V5R20 is used for modelling of the new small train vehicle. It is made by using rectangular hallow section (RHS) of 30x30x3mm steel beam welded together for obtaining better model. Chassis is a space frame type with the same length width and height value as previous as shown in table 3.8 below.

Table 3.8: Basic dimensions of the new model

Parameter	Value in mm
Length	2000
Width	1000
Height	1600

We decide to share the safety rules or standards as three wheeled vehicle which is listed under ‘The Pakistan standard of three wheeled vehicle’ [12] that has fulfilled during the design of small train structure design. These rules are listed below in the table and used in this thesis as guidance.

Table 3.9: Rules and standards taken from Pakistan standards for small train [12]

Rule number	**Name**	**Specification**
1	Seating capacity	Three (3) persons including driver
2	Safety bar	There shall be a partition between the driver and Passengers and with safety bars.
3	Provision of doors	Optional

4	Protection from weather	The passengers shall be provided protection from weather by means of fiberglass or any suitable water-resistant material
5	Seat	Every passenger shall be provided with at least 38 cm (15″) x 38 cm (15″) seats.
6	Seating height	Shall not be less than 30 cm (12 inches) measured from floor board
7	Backrest	Every passenger shall be provided with backrest of 40 cm (16 inches) height minimum.
8	Roof	Watertight roof shall be provided which shall cover completely the area from windscreen to the extreme rear edge of the vehicle.
9	Height of roof	The internal height of the roof measured from the seat level to any point vertically above each seat shall not be more than 1060mm
10	Leg room back	Every passenger shall be provided with legroom not less than 32.5 cm (13 inches)
11	Floor board	Shall be strong made of suitable material so closely fitted covered to prevent smoke or dust from entering in.
12	Mirrors	Shall be provided internally and externally so fitted as to enable the driver to have view of the road in the rear of the vehicle

Using data of basic dimension of small train and rules and standards of Pakistan in the above table, the CATIA model of the new vehicle structure has been developed. This model in figure 3.13 below shows the model of the new small train vehicle that is ready for FEM analysis.

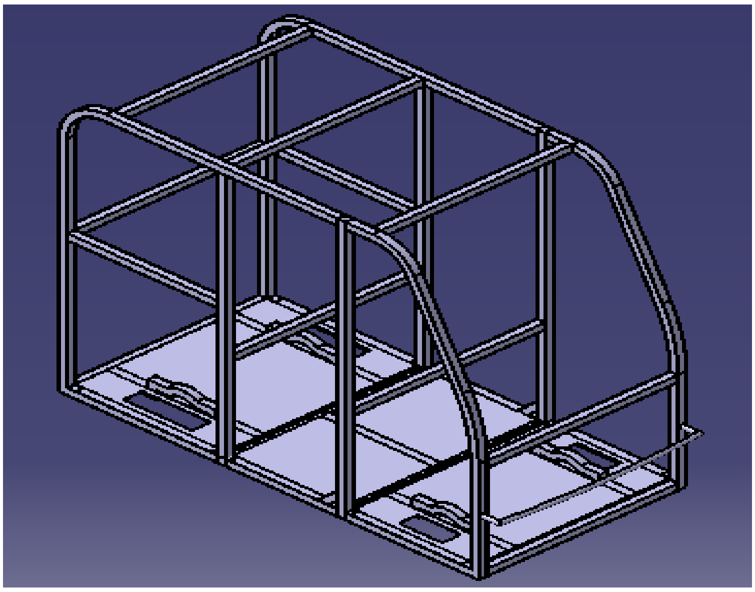


Figure 3.13: 3D CATIA modelling of the new train body structure using RHS 30x30x3mm

3.6.4. FEM analysis of the new small train vehicle model

The element types selected were BEAM 189 for the tubing sections and shell element being used for skin floor. ANSYS only required the input of Young’s modulus and Poisson's ratio, with the values of 200GPa and 0.3 used respectively; these were used for both the tubing and sheet steel.

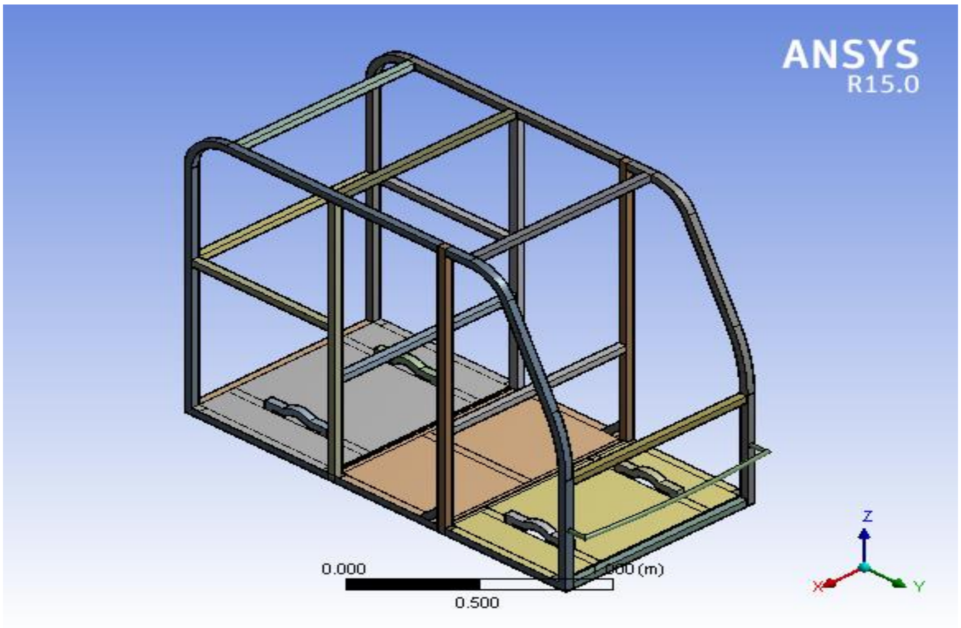


Figure 3.14: 3D ANSYS imported from CATIA Drawing

An important process of any FEA is to refine the mesh that is used in the analysis, a process that involves reducing mesh or element size a number of times and plotting the results as to ensure the analysis is stable as shown in figure 3.15 below.

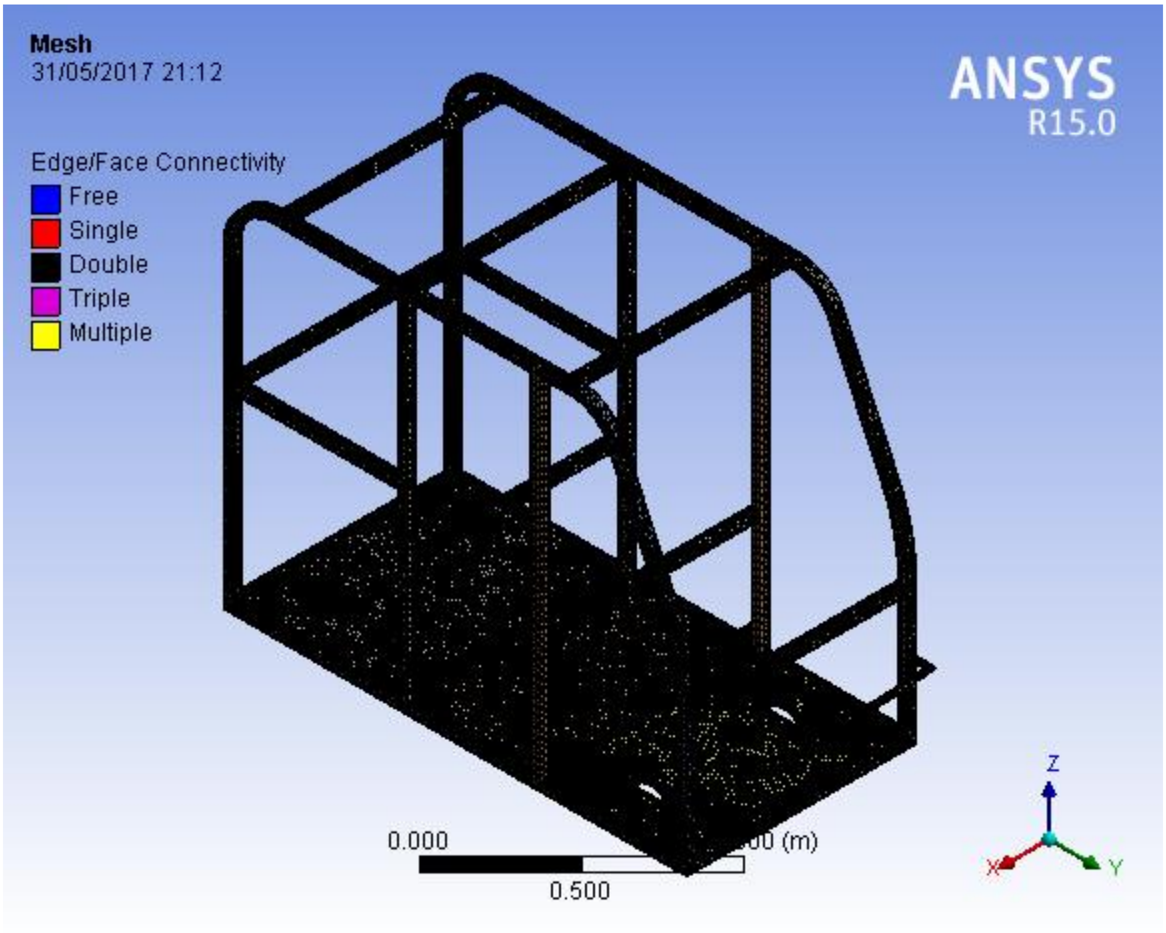


Figure 3.15: FEM mesh of the new model

Element and Nodes

The meshed new model of small train chassis has number of elements and number of nodes. The element is BEAM189 :(three nodded quadratic version of BEAM188. This is pretty much the ultimate beam element that can do everything well).

In order to get a better result, locally finer meshing applied in the region which is suspected to have the highest stress that is around the passenger and driver seats. Finally the model has 228956 numbers of Nodes and 86090 numbers of Elements.

3.6.5. Static analysis

Loading and Boundary condition

Total weight, the weight of small train vehicle having all components without occupant or baggage is, 230 kg or 2255.52N. This has been applied at the centre of gravity of the frame. And maximum play-load including driver is 195kg applied at seat points of each passenger and driver (figure 3.16 below). For each will have 65Kg (637.43N) weights. Actually no modification applied to loads and boundary conditions since our aim is to study and get better strength and stiffness in comparison to the existing model. And we have also the same axle load as before.

- Front axle load= 2x877 N
- Rear axle load = 2x1206 N

And the material for the structure is structural steel the same mechanical and chemical properties as before for the existing small train.

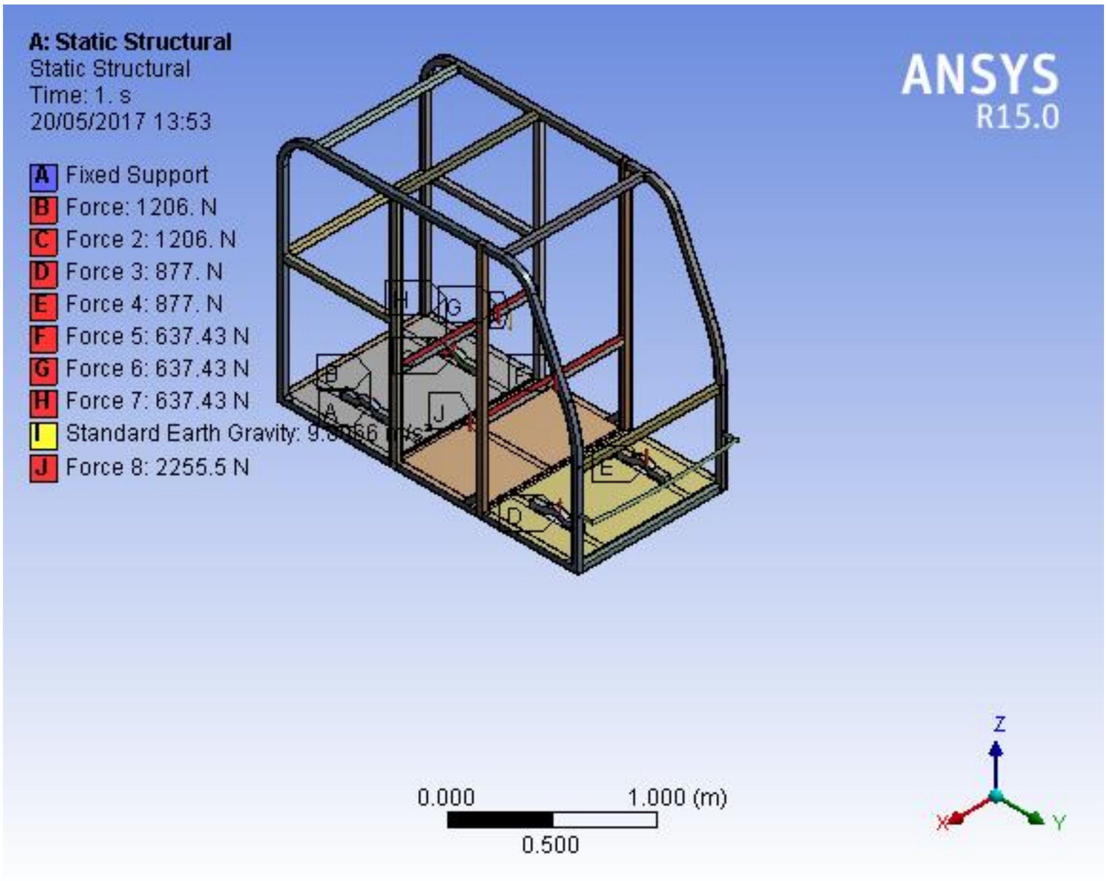


Figure 3.16: loads and boundary conditions for new model static analysis

Analysis results for static structure

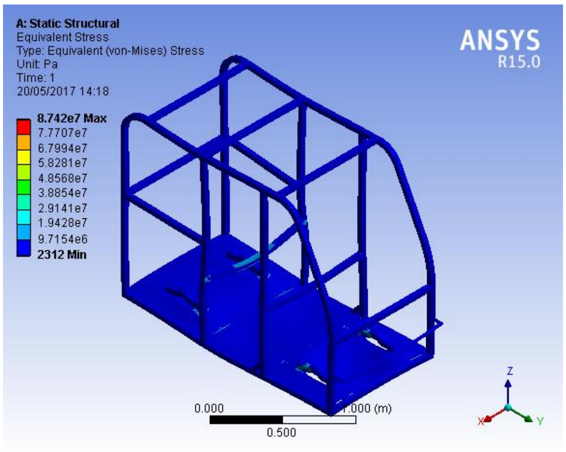
The frame body structure provides necessary support to the vehicle components placed on it. Stress analysis using finite element method (FEM) can be used to locate the critical point which has the highest stress. This critical point is one of the factors that may cause the fatigue failure. The magnitude of the stress can be used to predict the life span of the vehicle structure. The accuracy of prediction life of vehicle structure is depending on the result of its stress analysis.

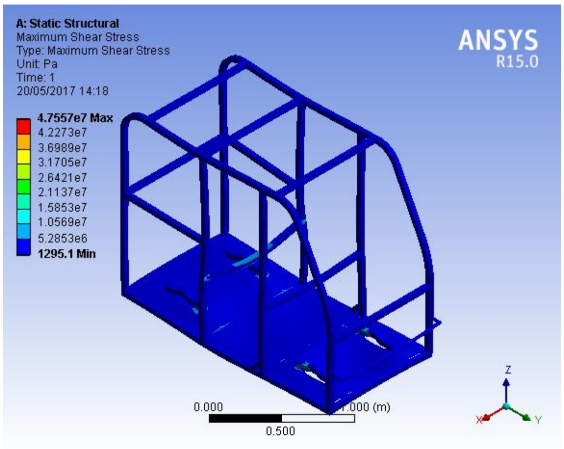
Maximum Von Misses stress and maximum shear stress

For the new model, the maximum Von Misses stress magnitude obtained is 87.42MPa (figure 3.17(a) and table 3.9). Its critical point is around passenger seats and the maximum shear stress magnitude is 47.557MPa (figure 3.17(b)). But factor of safety is 2.8598 (figure 3.17(c)). So it is safe for stiffness and thickness of the structure at reliable stress level. Therefore, it indicates as it is necessary to accept a new small train body structure of vehicle.

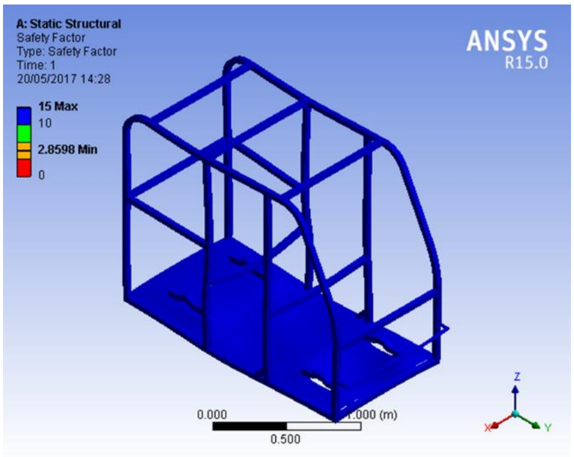
Table 3.10: static structural simulation results of new model

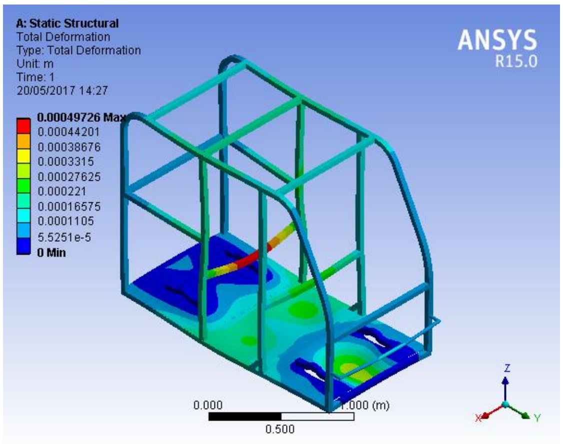
Analysis type	Equivalent or von-misses stress in MPa	Maximum shear stress in MPa	Maximum principal stress in MPa	Safety factor	Total deformation in m	Directional deformation in m
Minimum	2.312e-003	1.2951e-003	-34.999	2.8598	0. m	-8.2631e-005
Maximum	87.42	47.557	120.09	-	4.9726e-004	8.0843e-005





a) **b)**





c) **d)**

Figure 3.17: shows a) equivalent stress b) maximum shear stress c) safety factor d) total deformation

3.7. Crash analysis of new small train model

1. Front crash simulation

To do the FEA analysis of the front impact for the new small train, we are going to apply the load of 5206.25N at the front in addition to static loads. The frame has displacement boundary conditions at bearing mounting points Ux = Uz = 0 and at rear corner points all DOF=0.see figure 3.18 below.

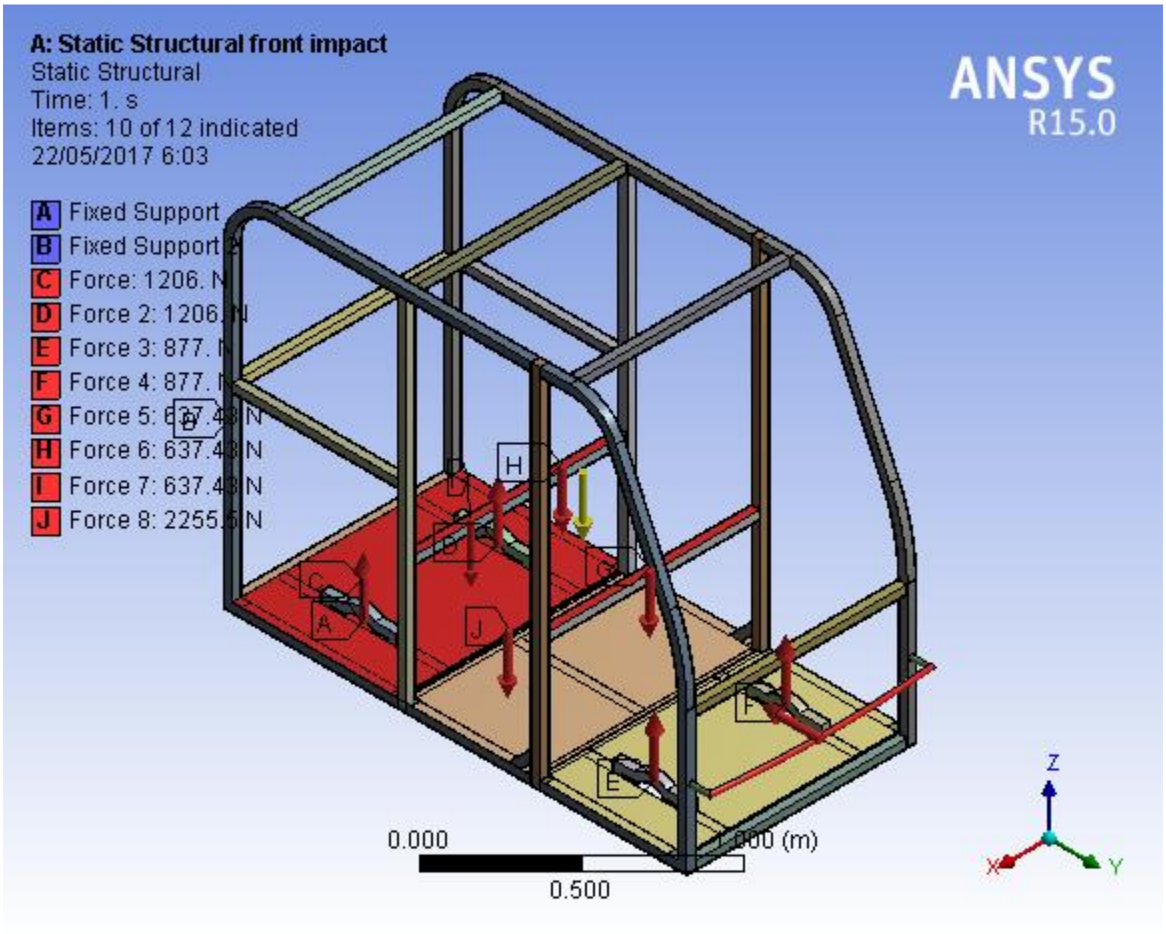


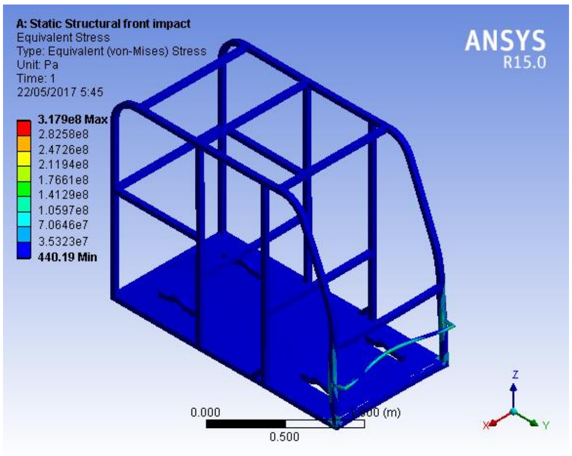
Figure 3.18: loads and boundary conditions for front impact analysis of the new small train model

Analysis stress results

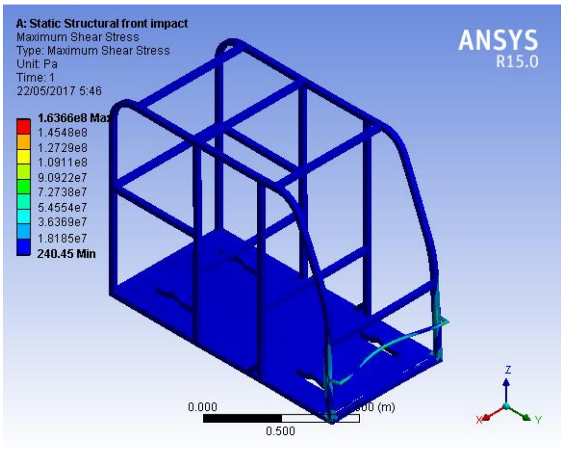
Since the force applied on the front head, the location of maximum Von Misses stress and maximum shear stress occurred at front (figure 3.19). The Von Misses stress magnitude of critical point is 317.9 MPa and the maximum shear stress magnitude is 163.66 MPa (table 3.11). The maximum shear stress agree with maximum shear stress theory of failure but not the maximum Von-misses stress. Even if the stress values obtained are above the material strength we should not forget as the analysis is for collision. Safety factor is 0.7864. Hence, the structure is a little bit not safe under frontal impact but compared to the existing model it is more reliable under crashing condition.

Table 3.11: front crash results of new model

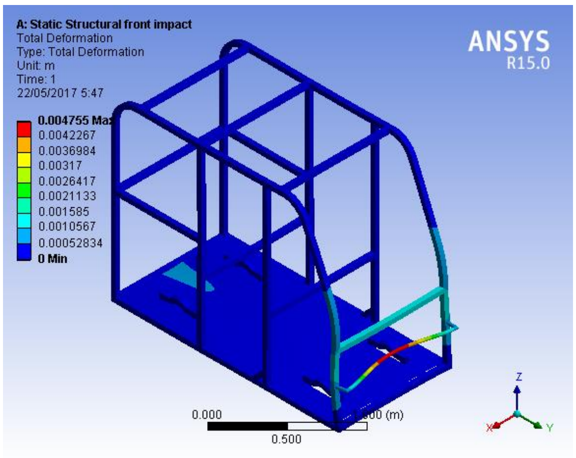
Analysis type	Equivalent or von-misses Stress in MPa	Maximum Principal Stress in Mpa	Maximum Shear Stress in MPa	Safety factor	Total Deformation in m	Directional Deformation in m
Minimum	4.4019e-004	-91.772	2.4045e-004	0.7864	0	-4.2845e-004
Maximum	317.9	182.24	163.66	-	4.755e-003	4.2845e-004



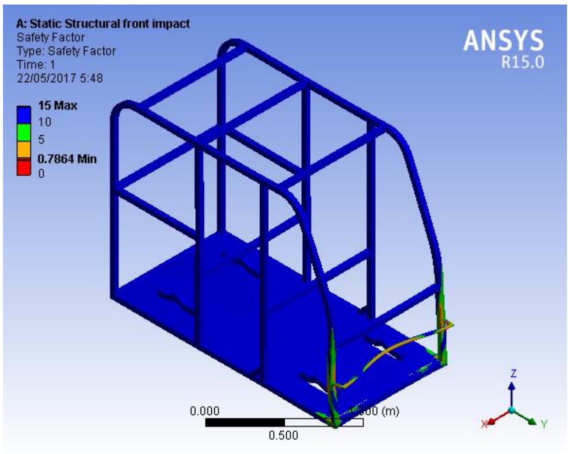
a)



b)



c)



d)

Figure 3.19: shows a) equivalent stress b) maximum shear stress c) total deformation d) safety factor

Deformation results

As we can see in figure 3.19 (c), the maximum displacement under these conditions is 4.755 mm, which is almost acceptable. This figure shows us the front beams are deflected inward due to the large amount of load applied in case of collision.

2. **Rear crash simulation**

To do the FEA analysis of the rear impact load applied is 5206.25N at the rear in addition to static loads as we applied before in the static condition analysis. The point of application of these loads will be the actual attachment points between the impact front and rear heads and boundary conditions are: at bearing mounting points Ux = Uz = 0 and front corner points all DOF=0.

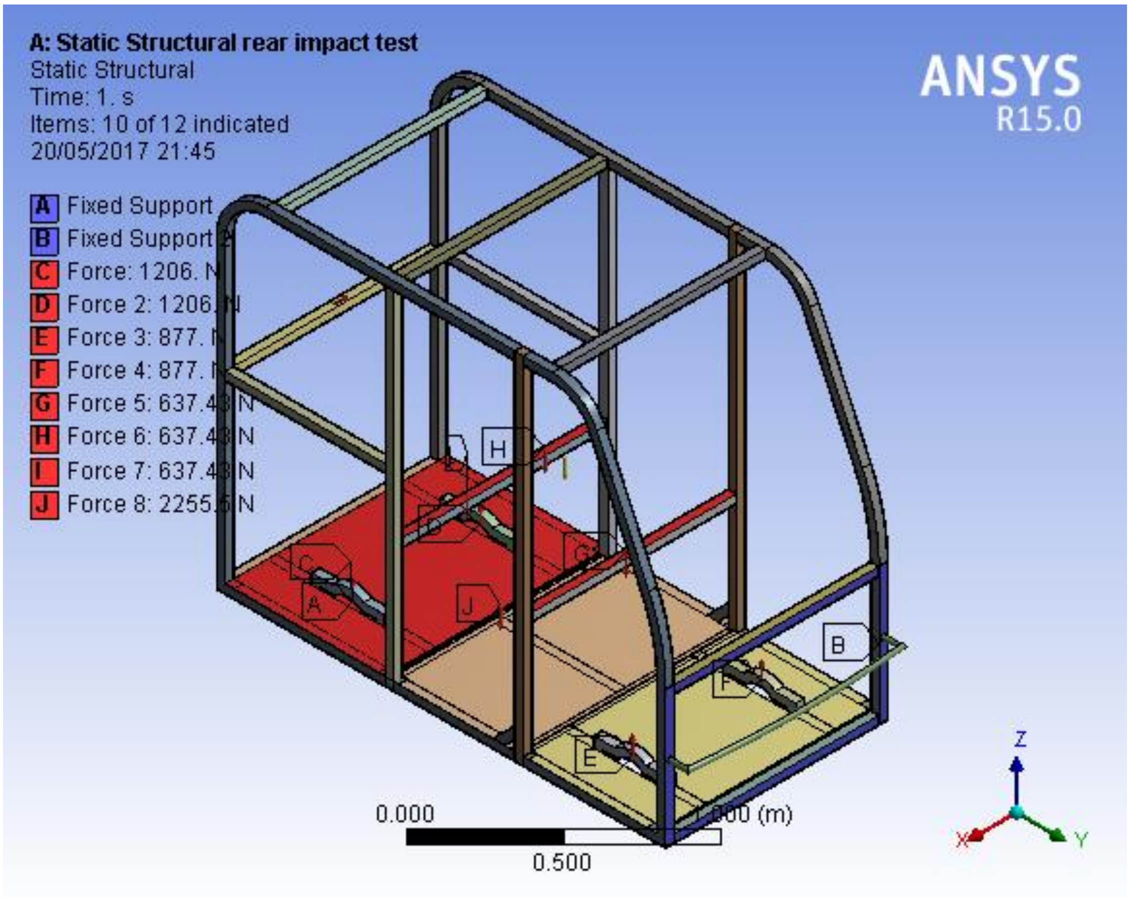


Figure 3.20: Rear crash loads and boundary conditions of new mode

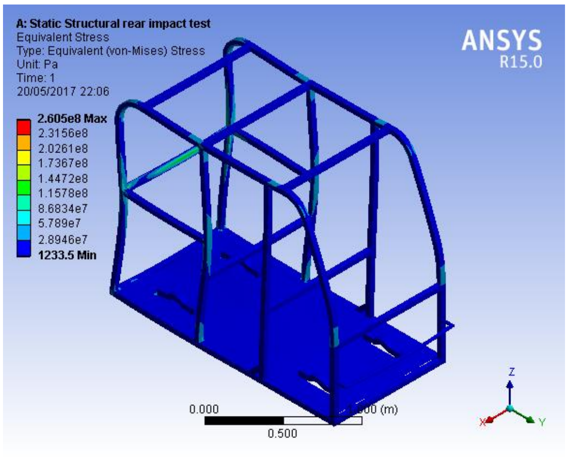
Analysis results

The location of maximum Von Misses stress and maximum shear stress are at rear frames (figure 3.21 a and b).

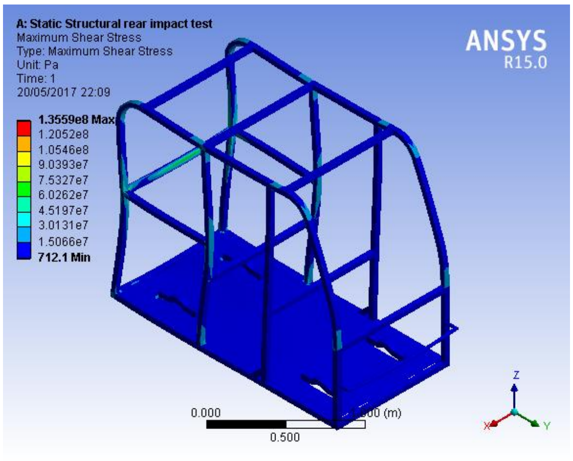
The Von Misses stress magnitude of critical point is 260.5MPa and the maximum shear stress magnitude is 232.04MPa. As we can see table 3.12, the result of the test is that the maximum displacement under these conditions is 4.2005e-003 m or 4.2005 mm, which is acceptable. This result will be done because the frame could resist the load, and giving a high rigidity. Safety factor is 0.95969, hence; the structure is almost safe under rear impact.

Table 3.12: Analysis results of rear crash for new model

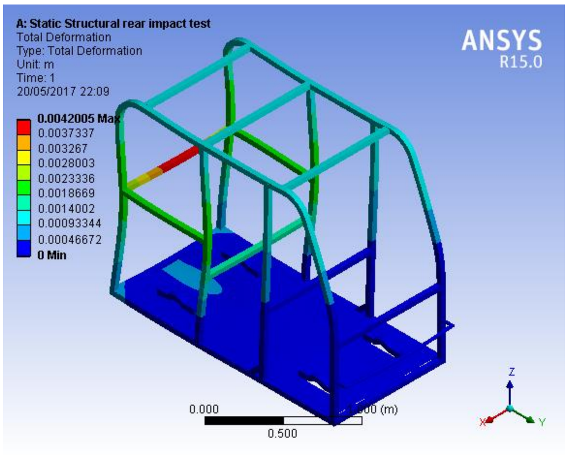
Analysis type	Equivalent or von-misses stress in MPa	Maximum principal stress in MPa	Maximum shear stress in MPa	Safety factor	Total deformation in m	Directional deformation in m
Minimum	1.2335e-003	-34.95	7.121e-004	0.95969	0. m	-4.0447e-004
Maximum	260.5	232.04	135.59	-	4.2005e-003	4.0245e-004



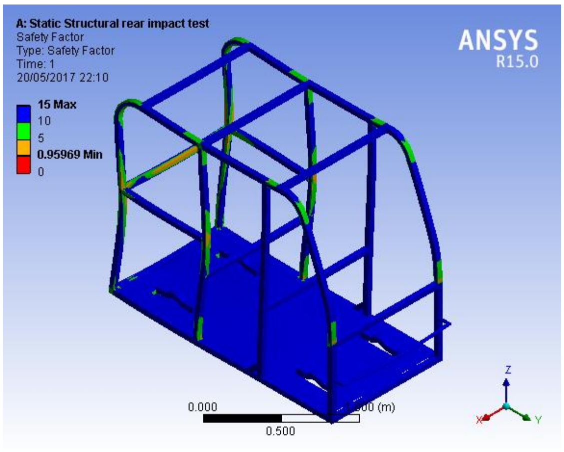
a)



b)



c)



d)

Figure 3.21: a) equivalent stress b) maximum shear stress c) total deformation of analysis d) safety factor results of rear impact

3. Side impact FEM simulation

Side impact of the vehicle may occur due to crash one vehicle with another vehicle that comes perpendicular to the side of this vehicle. To do the FE analysis of the side impact, we are going to use the load applied of 5206.25N at the side. The model used is full model of the small train vehicle and boundary conditions at bearing mounting points are Uy = Uz = 0 and at side corner Points All DOF=0, see figure 3.22.

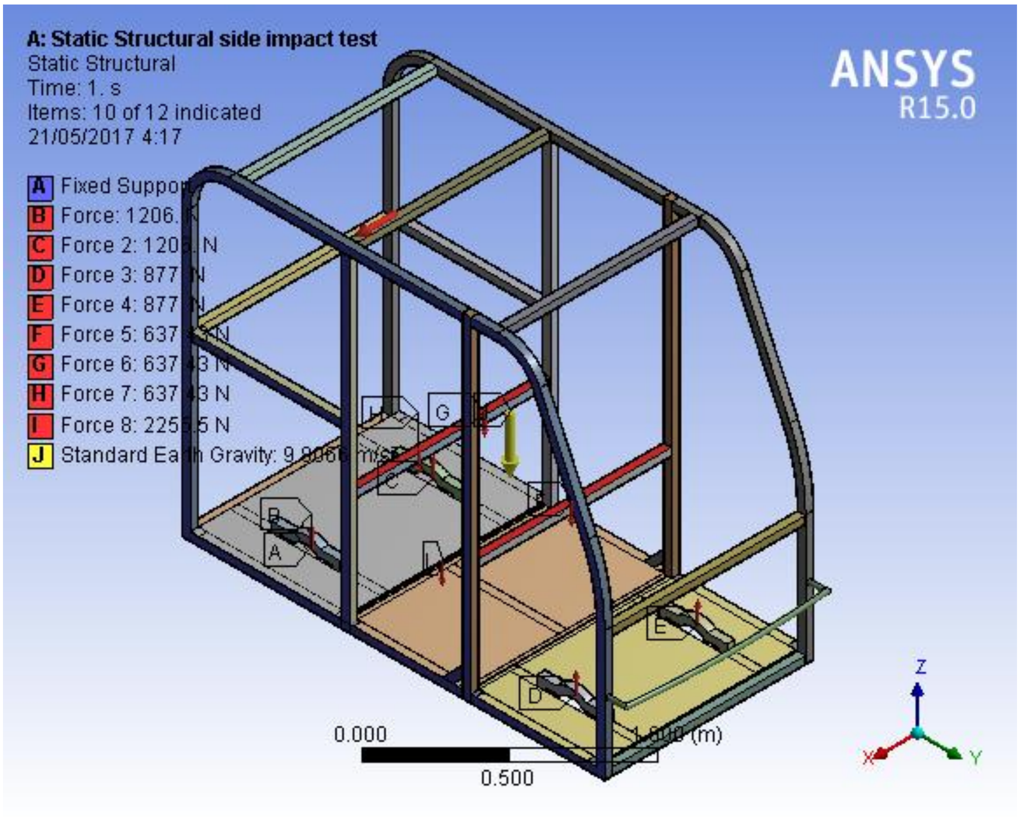


Figure 3.22: shows loads and boundary conditions of side impact simulation for new model analysis results

Analysis results

The location of maximum Von Misses stress and maximum shear stress are at top sides around passenger's seat (figure 3.23). The Von Misses stress magnitude of critical point is 81.142MPa and the maximum shear stress magnitude is 111.82MPa (table 3.13).

Table 3.13: Analysis results of side impact

Analysis type	Equivalent or von-misses stress in MPa	Maximum shear stress in MPa	Maximum principal stress in MPa	Safety factor	Total deformation in m	Directional deformation in m
Minimum	3.1642e-002	-11.419e	1.6624e-002	3.081	0	-4.0206e-005
Maximum	81.142	111.82	44.091	-	3.3598e-004	1.6463e-004

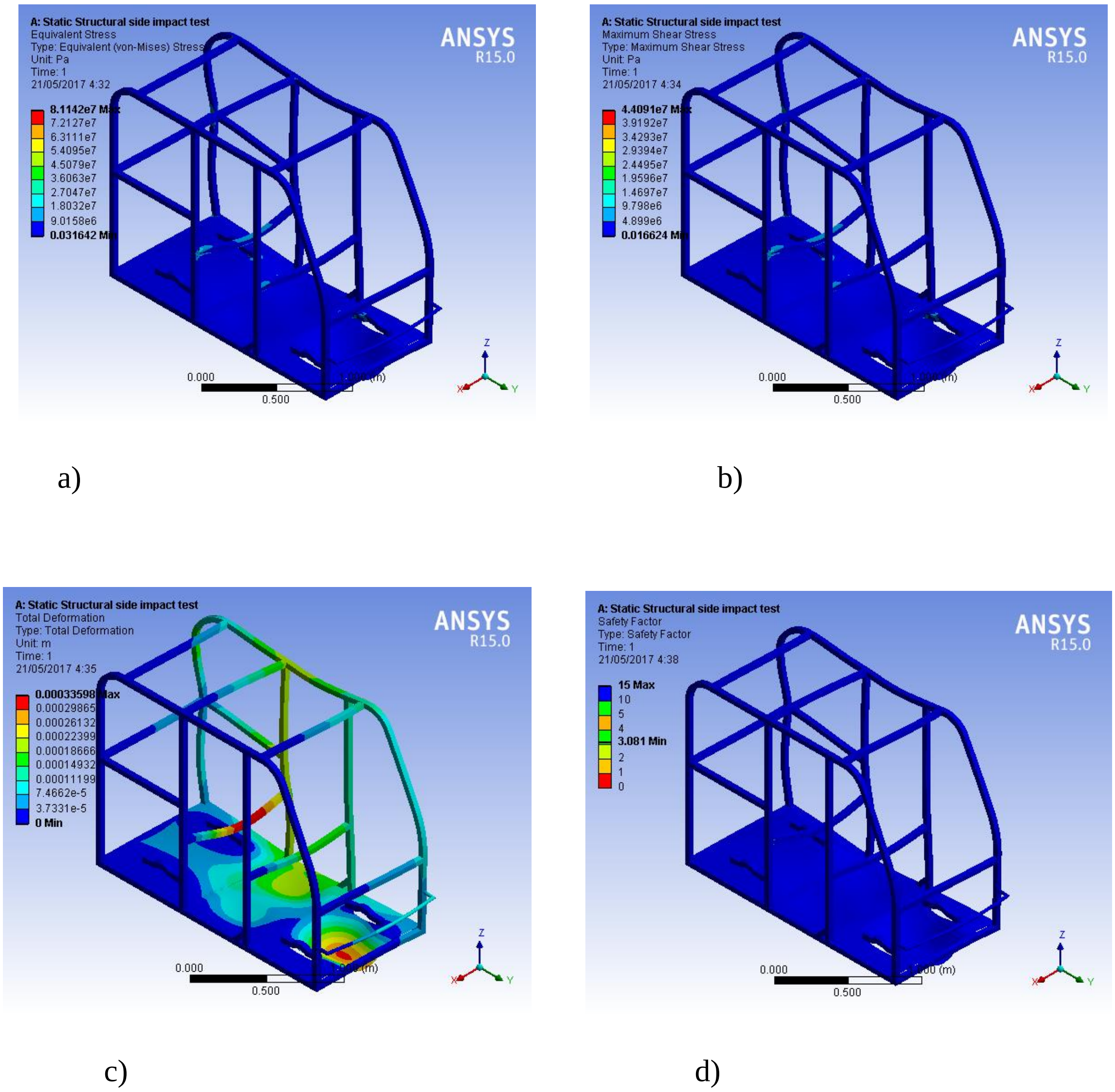


Figure 3.23: shows a) equivalent stress b) maximum shear stress c) total deformation d) safety factor

As we can see, the maximum displacement under these conditions is 3.3598e-004m or 0.33598mm, which is acceptable. This result tells that, the bars are in a correct position, giving a high rigidity to the frame. Safety factor is 3.08, hence, the structure is safe under side Impact.

3.8. Analytical Verification

The following method is used as verification for the method we have used during our analysis. The method is verified by using a simply supported overhanging beam over the wheels of cross section used in the structure. Total load acting on chassis is the sum of capacity of the chassis and Weight of body and DC motor, which is 425 kg**.** Chassis has two beams. So load acting on each beam is half of the total load acting on the chassis. That is 212.5kg mass.

Calculation for reaction

Chassis is a Simply Supported Beam with uniformly distributed load acting on entire span of the beam and is assumed to carry a load of 212.5kg (2083.9N). Length of the beam is 1200 mm and uniformly distributed load is 2083.9N / 2000mm =1.043 N/mm. Side bar of the chassis are made from rectangular channels with 30mm x 30 mm x 3 mm.

- Front Overhang (a) =400 mm
- Rear Overhang (c) = 400 mm
- Wheel Base (b) = 1200 mm
- Total length (l) = 2000 mm

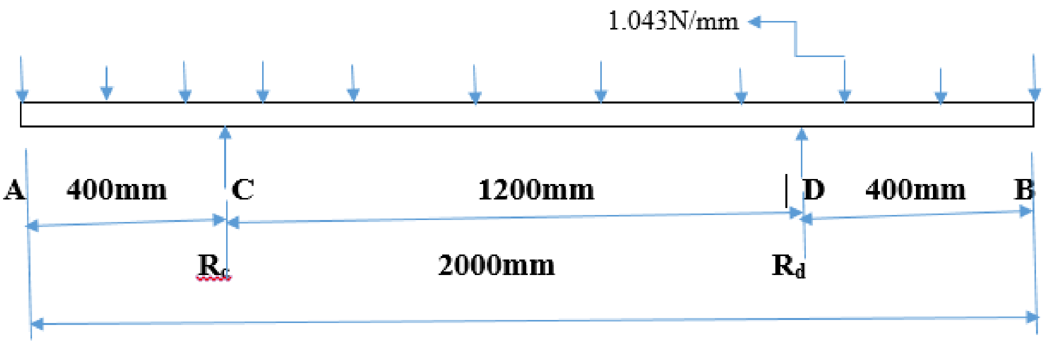


Figure 3.24: Chassis as a simply supported beam with overhang

1. **Reaction force calculation**

Now, taking the moment around the support D, we have:

$$R_c = \frac{w \times l \times (l-c)}{2 \times b}$$

$$= \frac{1.043 \times 2000 \times (2000-400)}{2 \times 1200}$$

$$=1043 \text{ N}$$

To find the moment at the reaction c, we follow the same principle. And it is obtained as:

$$R_d = \frac{w \times l \times (l-a)}{2 \times b}$$

$$= \frac{1.043 \times 2000 \times (2000-400)}{2 \times 1200}$$

$$=1043 \text{ N}$$

2. Shear force and bending moment calculation

a. Shear force calculation:

$V_1 = w \times a$

$=1.043 \text{ N/mm} \times 400\text{mm}$

$=417.2 \text{ N}$

$V_2 = R_c - V_1$

$= 1043 \text{ N} - 417.2 \text{ N}$

$=625.8 \text{ N}$

$V_4 = V_1$, because of a=c

$=417.2 \text{ N}$

$V_3 = R_d - V_4$

$=1043 \text{ N} - 417.2 \text{ N}$

$=625.8 \text{ N}$

Where, R_c and R_d are the reaction forces at point c and d, w is uniformly distributed load, V_1, V_2, V_3 and V_4 are shear forces.

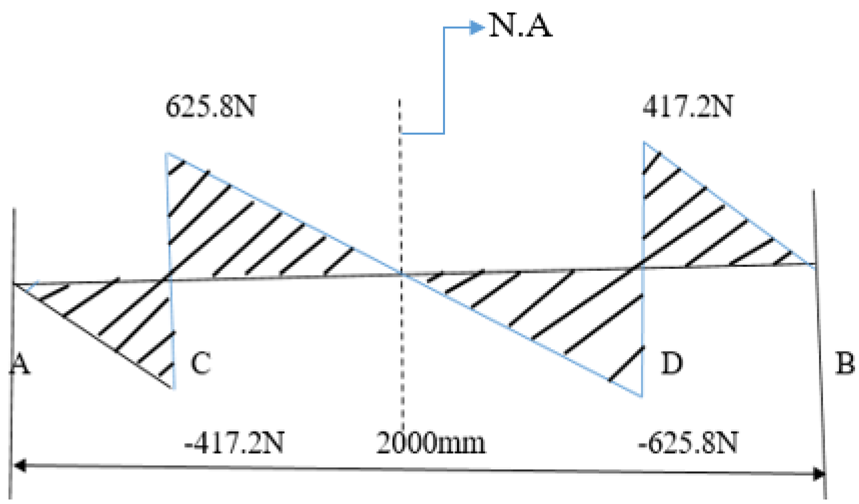


Figure 3.25: Shear force diagram

b. Bending moment calculation

i. Bending moment between A and C:

$$M_1 = \frac{-W \times a2}{2}$$

$$= \frac{-1.043 \times (400)2}{2}$$

= -83,440 Nmm

ii. Bending Moment between D and B:

$$M_2 = \frac{-w \times c2}{2}$$

$$= \frac{-1.043 \times (400)2}{2}$$

=83440 Nmm

iii. Bending moment between C and D:

$$M_3 = R_c \times \left(\frac{Rc}{2 \times w} - a\right)$$

$$= 1043 \times \left(\frac{1043}{2 \times 1.043} - 400\right)$$

= 104, 3000 Nmm

Where, M_1, M_2 and M_3 are bending moments at their corresponding points.

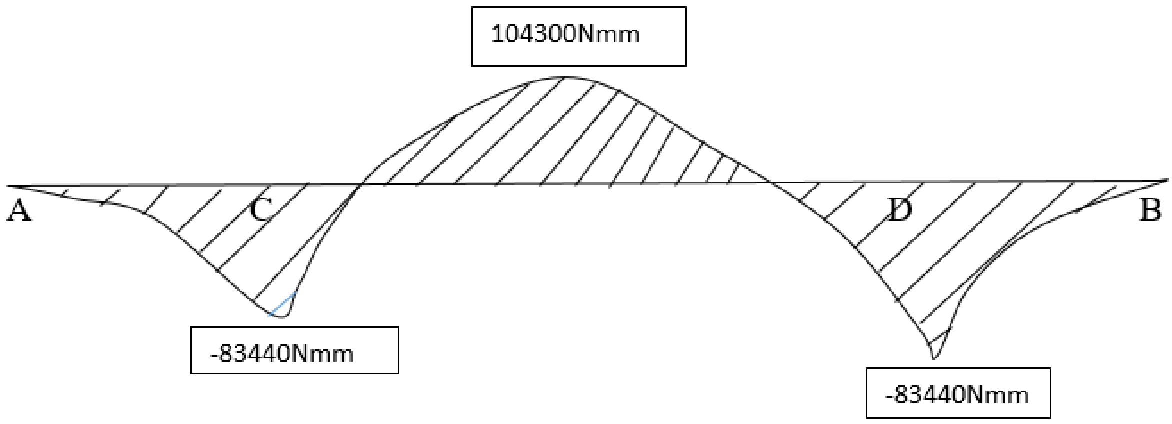


Figure 3.26: bending moment diagram

3. **Stress calculation**:

$$\sigma = \pm \frac{M*Y}{Iz},$$

Where, I_z is second moment of area and σ is the stress at distance y from the N.A. (z-z axis) of the beam cross-section.

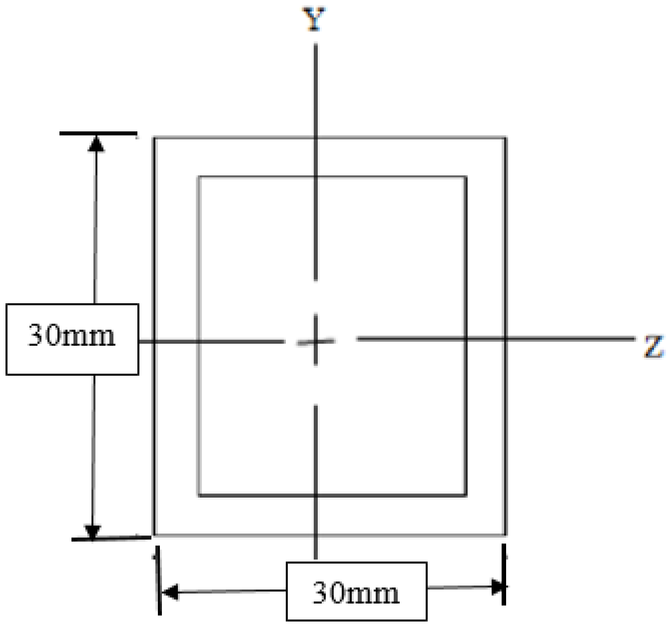


Figure 3.27: Cross section of the beam used in the verification

The second moment of area is given by

$$I_z=\frac{b\times h^3}{12}$$

$$I_z=\frac{30\times 30^3}{12}-\frac{24\times 24^3}{12}$$

$$=101,972\ mm^4$$

The maximum tensile stresses occur at bottom periphery of the cross section, i.e. y = -15mm whereas the maximum compressive stress occurs at the top periphery of the cross section, i.e. y=15mm.

$$\sigma=\pm\frac{104300\times 20}{101,972}$$

$$=\pm 20.46Mpa$$

The compressive stress occurs at the top side of the cross section and its value is calculated to be -20.46Mpa. The tensile stress occurs at the bottom section of the beams cross section and its value is found to be +20.46Mpa. Now to find the von miss stress (equivalent stress), we should first find the principal stresses.

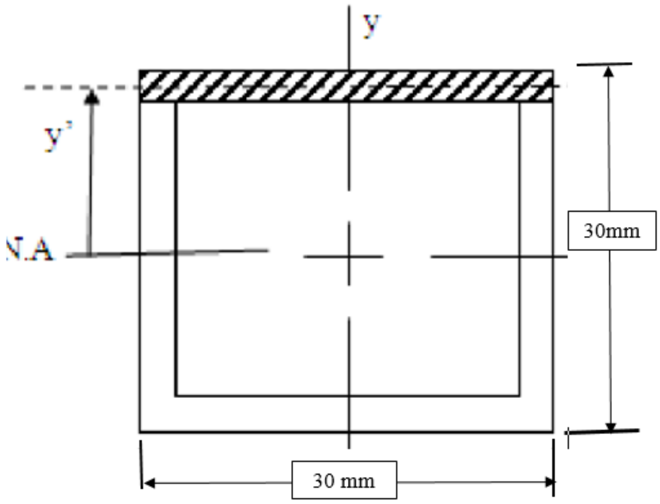


Figure 3.28: Cross section of beam for shear stress

For the shear stress at section, considering the shaded area above we have:

$$Q = y' \times A'$$

Where, y' is the distance from the neutral axis to the centroid of the area considered and A' is the area of the considered section.

As we have seen from shear force diagram shear force at the middle between points C and D of the beam becomes zero.

From the maximum shearing stress theory we have:

$\tau_{xy} = \frac{V*Q}{I}$ or $= \frac{Sy}{2}$, where S_y is yield strength in shear.

Since shear force has a zero value at the neutral axis, the shear stress becomes zero, i.e. $\tau_{xy} = 0$. To find the principal stresses, the following equation is used.

$$\sigma_1, \sigma_2 = \frac{\sigma x + \sigma y}{2} \pm \sqrt{\left(\frac{\sigma 1 - \sigma 2}{2}\right)^2 2 + \tau x y^2}$$

$\sigma_1 = \sigma_x = -20.46$ MPa, where, $\sigma_2, \sigma_y = 0 = \tau_{xy}$

$$S_y = \sigma_1 - \sigma_3$$

$$\sigma_3 = -250\text{Mpa} + 20.46\text{MPa}$$

$$=229.54 \text{ MPa}$$

$$\sigma_{\text{von-misses}} = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$$

=220.025 MPa is the von-miss stress at the middle point between supports.

4. Analytical verification for deflection of the beam

Macaulay's method (the double integration method) is a technique used in structural analysis to determine the deflection of Euler-Bernoulli beams, i.e. used to find the bending moment at any general section X-X of the beam.

The Macaulay's method helps to find a single expression for bending moment throughout the complete span of the beam. It is used to cover all loading conditions. Finding moment at x-x, taking clockwise direction as positive.

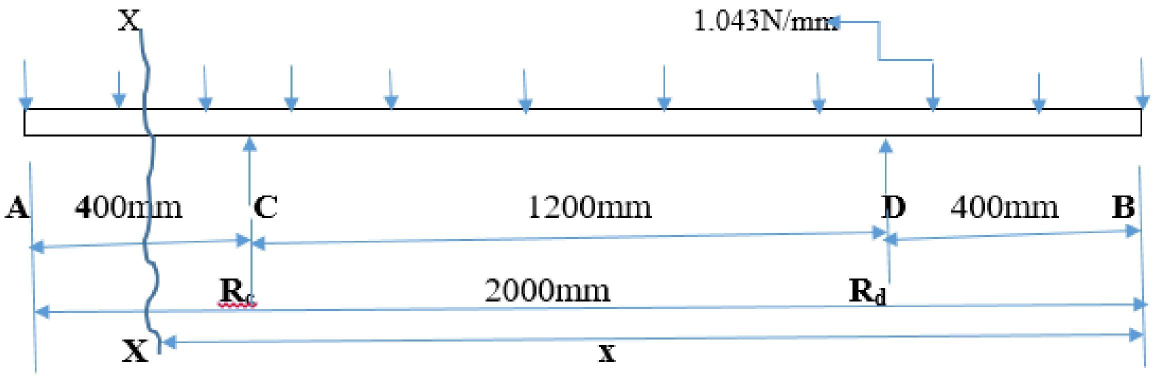


Figure 3.30: Section X-X of Beam

$M_{X\text{-}X}$= -1.043N/mm× (2000-x)-RC×(x-1600)-Rd×(x-400) +1.043N/mm×(x/2)

=-1.043N/mm× (2000-x)-1043N×(x-1600)-1043×(x-400) +1.043N/mm×(x/2)

= -2086+1668800+417200+1.043X-1043X-1043-+0.5215X

= 2083914 – 2084.4355X

But the bending moment is given by the following equation:

$$M=EI\frac{d^2y}{dx^2}$$

By integrating this equation, slope is found

$$\theta = \frac{dy}{dx}$$

The second integration of this equation gives deflection (δ) of the beam.

$$\delta=y$$

Therefore, integrating twice the above bending moment equation will provide the following deflection equation along the length of the beam.

$$EI\delta=AX+B+1041957X^2 - 347.406X^3$$

$$=-347.406X^3+1041957X^2+ AX+B$$

Where, A and B are integrating constants to be solved from the boundary conditions.

When x=400,y=0 and x=1600,y=0 Therefore, the values of A and B is

$$A= - 9.1625\text{x}10^8 \text{ and } B=22.2\text{x}10^{10}$$

$EI=200\times10^7\times101972=2039440000 N/mm^2$

Substituting these values in the deflection equation, the maximum deflection can be found at the centre between supports C and D at x=1000mm

$$2039440000*\delta = -347.406(1000)^3+1041957(1000)^2+ (-9.1625\text{x}10^8)(1000)+22.2\text{x}10^{10}$$

$$\delta = 5.8476mm$$

3.9. ANSYS verification

Using this beam of 2000 mm in length and having zero displacement values at its supports attachment points the ANSYS verification is done. The result extracted is the stress and displacement values at the middle point between supports because the analytical verification is done at this point. This happens since we are verifying our analytical results and ANSYS results in comparison. The following figure shows finite element mesh of the beam.

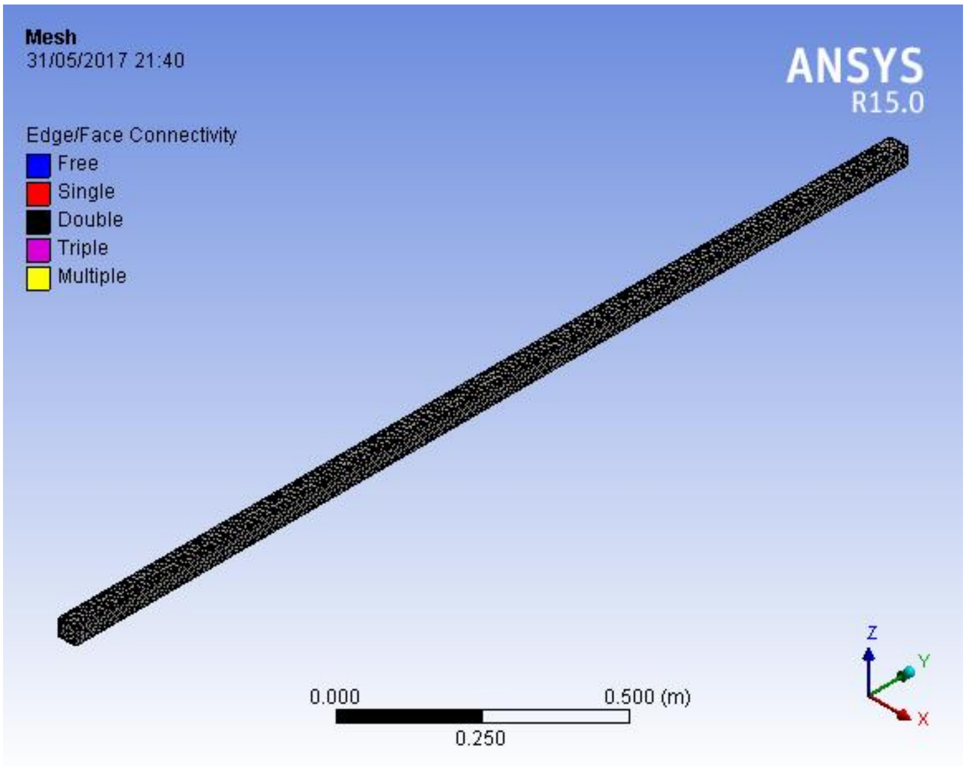
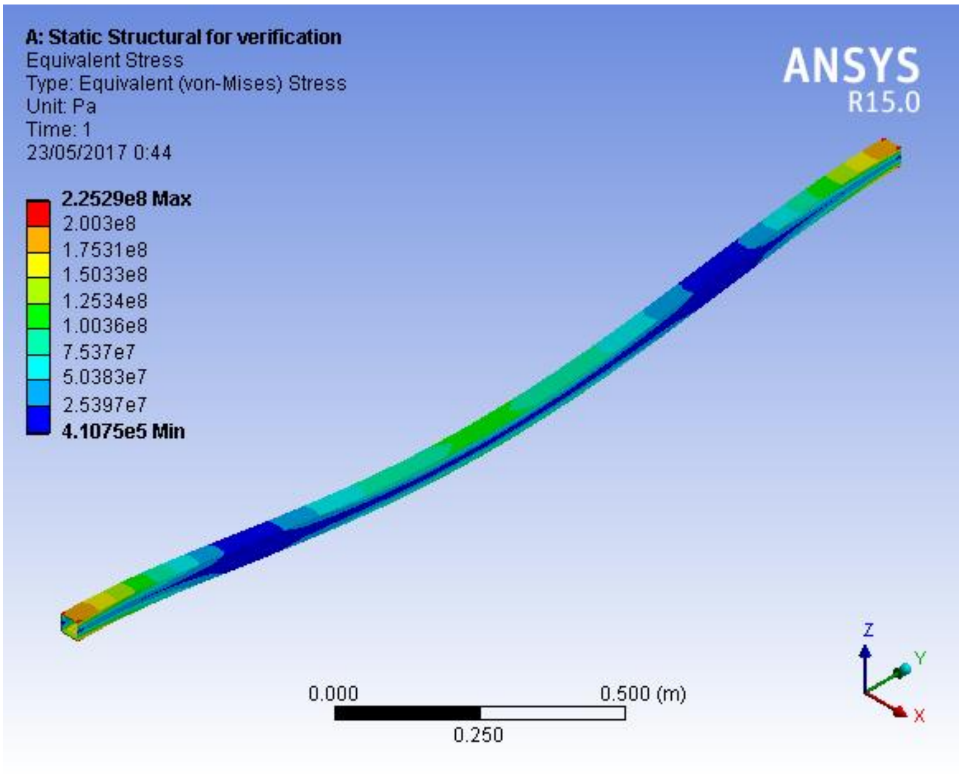
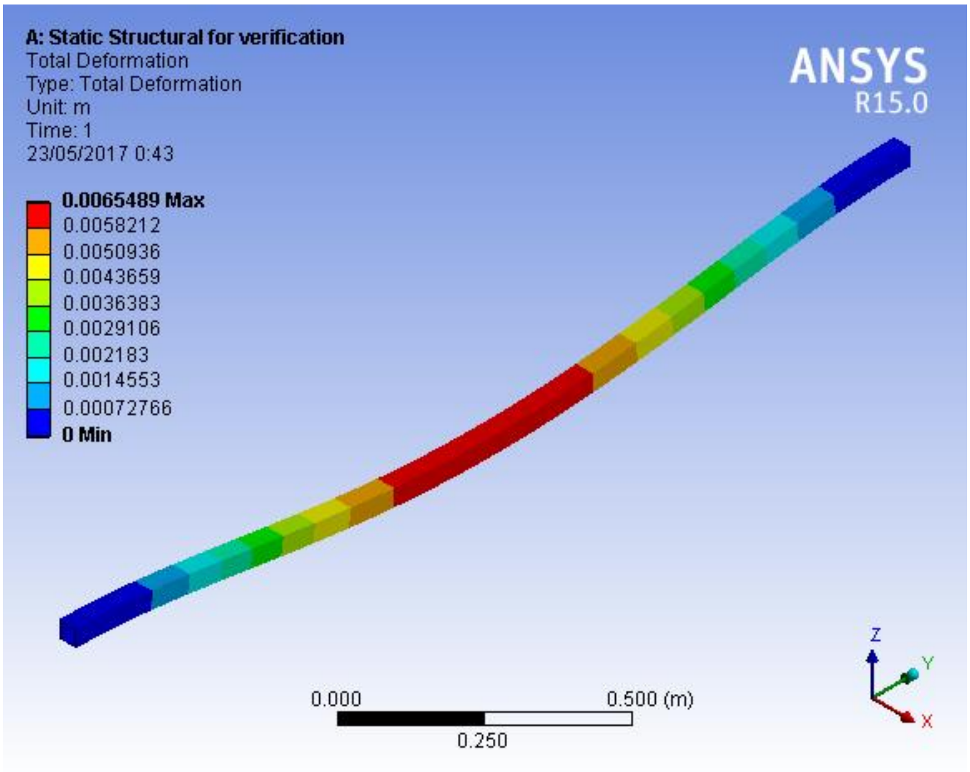


Figure 3.31: Finite element mesh of the beam



a)



b)

Figure 3.32: shows a) equivalent stress & b) deformation of ANSYS verification results

The ANSYS result gives a maximum equivalent stress of 225.29 MPa (figure 3.29).this result is a little greater than the analytical result of 220.025 MPa. And the maximum displacement obtained in analytical calculation is 5.8476mm but ANSYS gives the result of 6.5489mm (figure 3.29b).The difference is caused by simplification of model and uncertainties of numerical calculation but values approaches to each other. Therefore, this implies that our analysis is correct.

CHAPTER FOUR

4. RESULTS AND DISCUSSIONS OF THE MODELS

Here we will see the results of the existing model and the new model in comparison. Different cases of the analysis are tabulated and compared in each detail cases and remarks have been given. The remark is given to the new model strength status when compared to the existing model strength.

4.1. In terms of static case results

As we can see the results in table 4.1 below the results of equivalent stress, maximum shear stress and maximum principal stress are above the material strength for the existing model. Whereas, for the new models is below the material strength. Even though the values of the new model stresses are higher than the existing model stress values the safety factor value indicates that the model is safe. This shows that the new model is designed to have high strength without material wastage. But when we see the displacement values the new model is extremely reliable than the existing model.

Table 4.1: Static case result

Analysis type	Existing model	New model	Remark
Von-misses stress in MPa	432.83	87.42	Safe
Maximum principal stress in MPa	449.21	120.09	Safe
Maximum shear stress in MPa	237.92	47.557	Safe
Total deformation in mm	16.752	0.49726	Better
Safety factor	0.5776	2.8598	Safe

4.2. In terms of impact case

1. Front impact

Table 4.2 shows that the new model is much safer than the existing model in the case of front crash. Even if the new model stress level (von-misses stress) is above the material strength, the safety factor is about two times of the existing model. When we see the deformation results, there is difference of 11.57mm between them. This indicates that the new model design using the space frame, with increased thickness 3mm, is advanced one step forward in front crash case.

Table 4.2: Front impact result

Analysis type	Existing model	New model	Remark
Von-misses stress in MPa	693.57	317.9	Better but not safe
Maximum principal stress in MPa	446.81	182.24	Safe
Maximum shear stress in MPa	355.06	163.66	safe
Total deformation in mm	16.325	4.755	Better
Safety factor	0.36045	0.7864	Better

2. Rear impact case

All stress results of the new model except for von-misses stress are under the material resistance. As we remember from the figure of the existing model result the rear part of the model was diminished. But in the new model this case is extremely changed. Even the safety factor becomes approaches to one. From table 4.3 we can observe the deformation difference that is unbelievable.

Table 4.3: Rear impact result

Analysis type	Existing model	New model	Remark
Von-misses stress in MPa	351.35	260.5	Better but a little not safe
Maximum principal stress in MPa	370.89	232.04	Safe
Maximum shear stress in MPa	277.62	135.59	Safe
Total deformation in mm	13.302	4.2005	Better
Safety factor	0.48001	0.95969	better

3. Side impact case

In this case the analysis is somewhat different from the previous results. All the stress results of the new model are very low compared to the existing model result and the material strength. The safety factors values of existing model is bellow one. Whereas, for the new model it is greater than one (3.081) we see table 4.4 the total deformation result in the side is highly improved.

Table 4.4: Side impact result

Analysis type	Existing model	New model	Remark
Von-misses stress in MPa	580.49	81.142	Safe but seems over design
Maximum principal stress in MPa	465.43	44.091	Safe but seems over design
Maximum shear stress in MPa	294.1	111.82	Safe
Total deformation in mm	16.718	0.33598	Better
Safety factor	0.43067	3.081	Better

CHAPTER FIVE

5. SUMMARY, CONCLUSIONS AND RECOMMENDATIONS

5.1. Summary

In this thesis a small train body structure is designed and modelled in 3D CATIA software. The dimension of the small train body structure are taken from the previously built mini-electric train in ASTU campus. The original body is redesigned by replacing the RHS of 40x40x1mm with 30x30x3mm, increasing the thickness, selecting space frame type and reducing the number of elements with same material used so that the structure is safe in strength stiffer than the original one. Static structural and impact simulation is done on both the original body structure and newly remodelled body structure to determine the strength of the structure. Analysis is done in ANSYS 15 Workbench.

5.2. Conclusion

By observing the analysis result, the displacement and stress values are within their limits and strength of the newly remodelled train body structure is better. So we can conclude that:

- An attractive option is to be performed using the space frame that greatly facilitates the assembly and may even lead us to manufacture easily.
- The space frame is a better option in terms of strength and stiffness and cost for the small train frame and body design built in ASTU.
- In the existing model all the total displacement values obtained were above 16.752mm whereas, in the new model designed using space frame the maximum displacement obtained was 0.49726mm. This indicates that the stiffness value has been improved.
- Also in terms of the crash analysis the maximum von-misses stress obtained in the existing model was about 36% times larger than the results obtained in the new model. This shows that the new model can withstand impact loads so as the risk level has been minimized.
- According to failure by general yielding for ductile material, the new model is better in strength and stiffness, and safe structure.
- In general, we can conclude that, by replacing the RHS of 40x40x1mm with 30x30x3mm, increasing the thickness of the material and selecting space frame yields better results than the original model.

5.3. Recommendation and future works

As a recommendation and future work, we put here to researchers on this field and to the society of ASTU

- To invest their knowledge on further study of the proposed small train i.e. detail component design, design power and the maximum vehicle speed.
- To study with different materials and get optimum body structure.
- To analyse the train structure again both static and dynamic analysis.
- The sheet metal which is not included in our study should be included in the finite element analysis in order to find a more refined result.
- Welding effects should be studied
- To manufacture and finally give service in the campus to eliminate the wastage of time and extra cost.

REFERENCES

1. Damtie Enawgew, Strength analysis of three wheeled vehicle’s chassis and body frame assembled in Ethiopia, A thesis submitted to School of Mechanical and Industrial Engineering, Addis Ababa Institute of Technology, Addis Ababa, Ethiopia, October 2013.
2. Belete Jirru, Modelling and Structural Analysis of railway Vehicle body with Finite Element method, Thesis submitted to the Addis Ababa Institute of Technology in partial fulfilment of the requirements for the degree of Master of Science in Mechanical Engineering (Railway Engineering Stream), School Of Mechanical and Industrial Engineering, Addis Ababa, Ethiopia, March 2015.
3. Ahmed Abdulahi, , A thesis submitted to Adama Science and Technology University Technology in partial fulfilment of the requirements for the degree of Master of Science in Automotive Engineering, School of Mechanical, Chemical and Materials Engineering, Adama, Ethiopia, November 2016.
4. Besufekad Getachew, Modelling and Dynamic Analysis of Bus Body Built in Ethiopia, Thesis submitted to the Addis Ababa Institute of Technology in partial fulfilment of the requirements for the degree of Master of Science in Mechanical Engineering (Mechanical Design Stream), Addis Ababa Institute of Technology (AAiT) School of Graduate Studies, Addis Ababa, Ethiopia, February 2012.
5. C. Baykasoglu a,b, E. Sunbuloglu a, S. E. Bozdag a, F. Aruk a, T. Toprak a and A. Mugan, Numerical Static and Dynamic Stress Analysis on Railway Passenger and Freight car Models : Istanbul Technical University, Faculty of Mechanical Engineering, 34437,Istanbul, Turkey, April 2012.
6. Operation and Maintenance Manual for 70% Low Floor Light Rail Vehicle Project in Addis Ababa, Ethiopia
7. Beatriz Tena Villar, Railway design, simulation and analysis by ADAMS-Rail software, AKADEMIA GÓRNICZO-HUTNICZA, June 2011.
8. The Aluminium Automotive Manual, Applications-Car body (Body structures), Version 2013 © European Aluminium Association.
9. Sri. N.R.HEMA KUMAR, A Text Book on Automobile Chassis and Body Engineering, Lecturer in Vocational, Department of Vocational Education, Government Junior College, PALAMANER.

10. Kebede Zegeye, Modification and Analysis of Frontal Frame Structure of a Small Car with Different Composite Materials for Improving Self Protection in a Frontal Crash, A thesis submitted in partial fulfilment of the requirements for the award of the degree of Master of Science in Automotive Engineering, DEPARTMENT OF Mechanical and Vehicle Engineering, Adama, Ethiopia, September, 2015.
11. Fausto Capelli, Vittario Boneschi and Paolo Vigano, Stainless steel, A New Structural Automotive Material, 9th International Conference and Exibition, Milan, Florence ATA 2005-May 11, 12, 13 2005.
12. Pakistan Standards and Quality Control Authority (PSQCA) Standards Development Centre, Pakistan standard specification for three wheeler auto vehicles (SDC), paper for standards and rules, Pakistan, 2005.
13. Isuzu Truck, weight distribution and Restrictions concepts, a lecture in automotive engineering, 2013.
14. J.Sugumaran, P.hd Scholar, Design and Analysis of Bus Body Structure by using Composite Materials, International Journal of Research in Engineering, Science and Technologies (IJRESTs), Mechanical Engineering of Mailam engineering college Mailam, ISSN-2454-5392.
15. Sandip Godse, Prof. D.A.Patel, Static Load Analysis Of Tata Ace Ex Chassis And Stress Optimisation Using Reinforcement Technique, International Journal of Engineering Trends and Technology (IJETT) - Volume4 Issue7- July 2013.
16. S.S Sane, Ghanashyan Jadhav and Anandaraj.H, Stress analysis of a light commercial vehicle chassis by FEM.
17. Mr. Nitin S. Motgi , Prof. S. B. Naik and Prof.P.R.Kulkarni " Impact Analysis of Front Bumper" International Journal of Engineering Trends and Technology (IJETT) - Volume 6 Number 5- Dec 2013.
18. David Botana Olañeta, Part Realization Process from CAD Models to Physical Parts, paper for Master Thesis, Lulea, May 2012.
19. Jornsen Reimpell, Helmut Stoll, and Jurgen W. Betzler, The Automotive Chassis: Engineering Principles, second edition, book translated from the German by AGET Limited, 2001.
20. Formula Student DIT Team, frame and body design, Institution of Mechanical Engineers (I. Mech. E), paper submitted for competition, Europe, 1999.

21. EN 15227 ,2007, Railway applications–crashworthiness requirements for railway vehicle bodies‟, European Committee for Standardization (CEN), CEN/TC 256, Final Draft 9/2007.
22. D.Purushottam Reddy[1], S.Sardar ali [2], D.Srikanth[3], JV.Mohanachari[4], DESIGN AND ANALYSIS OF BUS BODY SUPERSTRUCTURE, IJRDO-Journal Of Mechanical And Civil Engineering ISSN: 2456-1479.
23. Dr. Vinod K. Banthia, Loads, Design and Structure, M.S. Ramaiah School of Advanced Studies, Bengaluru, PEMP AME2510.
24. Brendan. J. Waterman, Design and Construction of a Space-frame Chassis, School of Mechanical and Chemical Engineering University of Western Australia, 20348559.
25. Harish BM, Mallesh AP, K. Prakash and Sudharshan M, Crashworthiness Analysis of Metro Cars (As per International Railway Standard EN 15227), Altait Technology Conference, India 2015.
26. NORSOK Standard, Material Data Sheets For Structural Steel, Norwegian Technology Center, M-120 Rev. 3, Dec. 2000.
27. David Roylance, Mechanical Properties of Materials, 2008.
28. David Wennberg, Light-Weighting Methodology in Rail Vehicle Design through Introduction of Load Carrying Sandwich Panels, Centre for Eco2 Vehicle Design Department of Aeronautical and Vehicle Engineering, TRITA AVE 2011:36 ISSN 1651-7660 ISBN 978-91-7501-002-1.
29. Pankaj Chandna, Gian Bhushan and V.P. Singh, Finite Element Analysis of Bus Body structure using CAE tools.
30. M. A. Maleque and M. S. Salit, Materials Selection and Design, Springer Briefs in Materials, DOI: 10.1007/978-981-4560-38-2_2, 2013.
31. Module 3, Design for static loading, Version 2 ME, IIT Kharagpur.
32. Dario Croccolo, Massimiliano De Agostinis and Nicolò Vincenzi, Structural Analysis of an Articulated Urban Bus Chassis via FEM: a Methodology Applied to a Case Study, Strojniški vestnik - Journal of Mechanical Engineering 57(2011)11, 799-809, DIEM, Department of Mechanical Engineering, University of Bologna, Italy, 2011.