A Main Project Report on

"FATIGUE ANALYSIS OF FRONT AXLE OF TRUCK"

Submitted to

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY KAKINADA

In Partial Fulfillment of the Requirements for the

Award of the Degree of

Bachelor of Technology

In

MECHANICAL ENGINEERING

BY

A. BHOGESH	**18485A0317**
B. SUSMITHA	**17481A0309**
CH.E.N.S.M.KUMAR	**17481A0317**
D.SRI KRISHNA	**18485A0319**

Under the Guidance of

Sri A. RAJESH, M. Tech (Ph.D)

Assistant Professor

DEPARTMENT OF MECHANICAL ENGINEERING

GUDLAVALLERU ENGINEERING COLLEGE

(An Autonomous Institute with Permanent Affiliation to JNTUK, Kakinada)

SESHADRI RAO KNOWLEDGE VILLAGE

GUDLAVALLERU- 521356

2020-2021

GUDLAVALLERU ENGINEERING COLLEGE

(An Autonomous Institute with Permanent Affiliation to JNTUK, Kakinada)

SESHADRI RAO KNOWLEDGE VILLAGE

GUDLAVALLERU- 521356

DEPARTMENT OF MECHANICAL ENGINEERING

CERTIFICATE

This is to certify that the main project report entitled "**Fatigue Analysis Of Front Axle Of Truck"** bonified record of work carried out by

A. BHOGESH	**18485A0317**
B. SUSMITHA	**17481A0309**
CH.E.N.S.M. KUMAR	**17481A0317**
D.SRI KRISHNA	**18485A0319**

Under my guidance and supervision and submitted to Jawaharlal Nehru Technological University Kakinada, Kakinada, in partial fulfillment of the Degree of Bachelor of Technology in Mechanical Engineering during the academic year **2020-2021.**

Sri A.RAJESH	**Dr. A. JAWAHAR BABU**
Project Guide	Head of the Department

External Examiner

ACKNOWLEDGEMENT

We would like to express our deep sense of gratitude and sincere thanks to **A. RAJESH**, Assistant Professor, in Mechanical engineering department, for his constant guidance, supervision and motivation in completing the projectwork.

We are very much indebted to **Dr. A. JAWAHAR BABU**, professor and head of the department of Mechanical engineering for giving his valuable suggestions and encouragement throughout the course of our project.

We would like to take this opportunity to express our profound sense of gratitude to our beloved principal **Dr. G.V.S.N.R.V. PRASAD**, for providing us all the requirement facilities.

We also thankful to faculty members of the department who directly or indirectly helped us in completing the project.

We also wish to express my indebtedness to my parents as well as my family members whose blessings and support always helped me to face the challenges ahead

Project associates

A. BHOGESH	**18485A0317**
B. SUSMITHA	**17481A0309**
CH.E.N.S.M.KUMAR	**17481A0317**
D.SRI KRISHNA	**18485A0319**

ABSTRACT

Front axle carries the weight of front part of the Automobile, as well as it facilitates steering and absorbs shocks due to road surface variations. The front axle is designed to transmit the weight of the Automobile front the spring to the front wheels, turning right and left as required. So proper design of front axle beam is extremely crucial. The axles in a system must bear the weight of the vehicle plus any cargo. The front axle beam is one of the major parts of vehicle suspension system. About 35 to 45 percent of the total vehicle weight is taken up by the front axle. Hence proper design of the front axle beam is extremely crucial.It experiences fatigue loading due various factors such as engine vibrations, bumps on the road, etc. therefore it becomes very crucial to design and analyse the axle for its fatigue life. In this project selection of material and doing fatigue analysis for analysing the life, damage, factor of safety and fatigue sensitivity are major factors.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

GRAPHS

CHAPTER-1

INTRODUCTION

The auto industry is one of the key sectors of the Indian economy. The Industry comprises of automobile and the auto components sectors and encompasses commercial vehicles, multi-utility vehicles, passenger cars, two wheelers, three wheelers and related auto components. During last few decades due to global economic scenario optimum vehicle design is major concern. To accomplish the need to design a moderate car, the structural engineer will need to use imaginative concepts. The demands on the automobile designer increased and changed rapidly, first to meet new safety requirements and later to reduce weight in order to satisfy fuel economy requirements. Experience could not be extended to new vehicle sizes, and performance data was not available on the new criteria. Mathematical modeling was therefore a logical avenue to explore. Most recently, the finite element method, a computer dependent numerical technique, has opened up a new approach to vehicle design

During the vehicle operation, road surface irregularity causes cyclic fluctuation of stresses on the axle, which is the main load carrying member. Therefore it is important to make sure whether the axle resists against the fatigue failure for a predicted service life. Axle experiences different loads in different direction, primarily vertical beaming or bending load due to curb weight and payload, torsion. Due to drive torque, cornering load and braking load. In real life scenario all these loads vary with time.

Vertical beaming is one of the severe and frequent loads on an axle Due to their higher loading capacity; solid axles are typically used in the heavy commercial vehicles. During the vehicle service life, dynamic forces caused by the road surface roughness produce dynamic stresses and these forces lead to fatigue failure of axle housing, which is the main load carrying part of the assembly. Front axle can experience a 3G load condition when vehicle goes on the bump.

The front axle of the vehicle carries a heavy load and also experience cyclic loading because of various factors such as engine vibrations, bumps on the road, etc. Due to fatigue loading, the axle beam may fail at stress values way below the yield or ultimate strength of the material. Therefore, it becomes very crucial to design and analyse the axle for its life.

Most of the axle beams fail due to fatigue loads acting on them which lead to fatigue crack growth generation, which then propagate and cause failure (breakage of axle in two parts).

Therefore, in this work, a finite element model of a front axle beam of a heavy duty truck is established, by providing boundary and loading conditions, the fatigue life of the axle beam is predicted with and without a crack, so that in reality, the beam could be replaced before failure, thus avoiding catastrophic accidents

The fatigue life of axle beam can be approximated close to practical values by using finite element analysis along with proper boundary conditions and loading conditions. For this purpose, ‘‘fatigue tool” in ANSYS workbench is utilized.

The theory of failure used for designing is ‘‘Goodman’s criteria for design under fatigue loading”; this theory is best suitable for design of ductile materials under fatigue loading. The material properties are fed into the system, these properties are then used to generate the S-N curve.

For design purpose the front axle beam of Ashok Leyland 1612 truck was choosen. All standard axles have an I cross section in the middle (spring seat to spring seat) and circular or elliptical cross sections at the ends. The front axle beam will have I cross section in the middle and circular cross sections at the ends. An axle is usually a forged component for which a higher strength to weight ratio is desirable. The I cross section has lower section modulus and hence gives better performance with lower weight. This type of construction produces an axle that is lightweight and yet has great strength.

The I-beam axle is shaped so that the center part is several inches below the ends. This permits the body of the vehicle to be mounted lower than it could be if the axle were straight. A vehicle body that is closer to the road has a lower center of gravity and holds the road better. On the top of the axle, the springs are mounted on flat, smooth surfaces or pads. The mounting surfaces are called spring seats and usually have five holes.

The four holes on the outer edge of the mounting surface are for the Ubolts which hold the spring and axle together. The center hole provides an anchor point for the center bolt of the spring. The head of the center bolt, seated in the center hole in the mounting surface, ensures proper alignment of the axle with the vehicle frame. A hole is located in each end of the I-beam section. It is bored at a slight angle and provides a mounting point for the steering knuckle or kingpin.

A small hole is drilled from front to rear at a right angle to the steering knuckle pinhole. It enters the larger kingpin hole very slightly. The kingpin retaining bolt is located in this hole and holds the kingpin in place in the axle. The steering knuckle is made with a yoke at one end and a spindle at the opposite end. Bronze bushings are pressed into the upper and lower arms of the yoke, through which the kingpin passes. These bushings provide replaceable bearing surfaces. A lubrication fitting and a drilled passage provide a method of forcing grease onto the bearing surfaces of the bronze bushings. The spindle is a highly machined, tapered, round shaft that has mounting surfaces for the inner and outer wheel bearings. The outer end of the spindle is threaded. These threads are used for installing a nut to secure the wheel bearings in position.

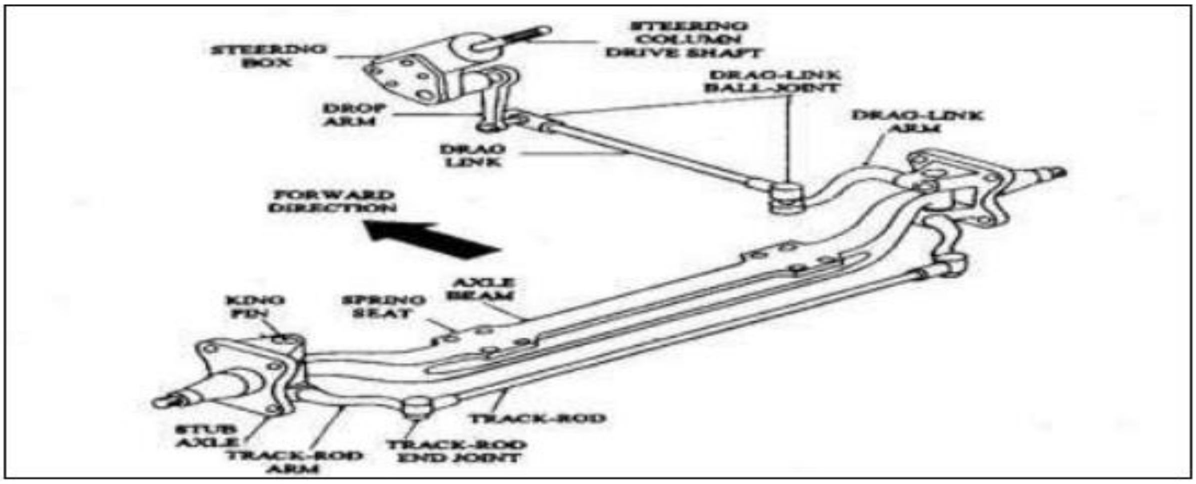

FIG-1Layout of Front Axle

CHAPTER-2

LITERATURE REVIEW

[1] **Kulkarni Vipul V, Prof. Ghodake Arvind P** have found that the steel is the traditional material for the axle and it can be replaced by composite materials like glass fiber or the epoxy resin. The most important factors were discussed in this thorough study. The epoxy resin is also better material but the glass fiber is the better than the epoxy resin also.

[2] **Pravin R.Ahire1, Prof.K. H. Munde** have found that the maximum deflection in front axle is 11.93mm for 27C15 materials, so, that 27C15 is better material for manufacturing of axle. Also in the present paper we have established a satisfactory co relation between hand calculations done analytically and the FEA results. The deflection in the FEA gave the confidence that the boundary conditions for beam are correctly simulated.

[3] **Ketan Vijay Dhande, Prashant Ulhe** has absorbed that the maximum deflection in axle is for SAE 1020 materials but at the same time the maximum stress distribution is low for SAE 1020 than Ductile Cast Iron. So, that SAE 1020 is better material for manufacturing of axle than Ductile Cast Iron. Also in this present thesis we have established a satisfactory co relation between hand calculations done analytically and the FEA results.

[4] **Mr. Khot S.S Padmashri Dr. Vithalrao Vikhe Patil** College of Engineering Ravindra **R Navthar Dr. Vithalrao Vikhe Patil** College of Eng has absorbed that Front Axle design of light truck application has been studied for different materials. Stresses of all materials are less than material yield and ultimate tensile/compressive strength and hence it is material are meeting static analysis acceptance limit. Axle with steel, CI and AL Alloy material have FOS safety of 1.44 , 1.57 and 1.62 respectively. Axle life of steel is observed to be least & AL Alloy observed life of axle highest among all material.

[5] **Pravin R.Ahire1 , Prof. K. H. Munde2** , “Design and analysis of front axle for heavy commercial vehicle”. Their paper deals with design and analysis of front axle. The same analysis with help of FE results were compared with analytical design. For which paper has been divided in to two steps. In the first step front axle was design by analytical method. For this vehicle specification – its gross weight, payload capacity, braking torque used for subject to matter to find the principle stresses & deflection in the beam has been used. In the second step front axle were modelled in CAD software & analysis in ANSYS software.

[6] **Hemant L. Aghav**, “Life Prediction for Vertical and Braking loading of Front Axle of Heavy-Duty Truck”. Aim of their study is to find stress analysis and predict life of front axle for vertical and braking loading case using analytical, experimental and FEA method. The fatigue life of front axle is estimated by strain life approach method. Here, Coffin-Manson and Smith Watson Topper equations used for calculating the fatigue of front axle beam. Stress analysis is performed by ANSYS workbench and fatigue analysis is performed by NCODE design life software under vertical and braking loading, and results are compared with the experimental test. Fatigue life of axle obtained by analytical method is more than 4 x 103,

which is considered as safe for vertical loading case, and the same is validated by FEA method. Stresses occurred in the axle are very high for braking loading case, so life of axle is also very low as compared to other loading condition i.e. vertical and cornering case. The location of Fatigue failure for vertical and braking loading obtained is in the goose neck of axle due to highest stress of that region.

[7] **Kiran Maddewad1 , Trupti Jadhav2** , “Optimization of Front Axle for Heavy Commercial Vehicle by Analytical and FEA Method”. Their paper focuses on design, analysis and optimization of front axle. The approach in this research paper has been divided into two steps. The First step involves design of front axle by Analytical method. For this, the vehicle specification gross weight and payload capacity is used to determine the stress and deflection in the beam. Second step involved further modeling of front axle using CATIA-V5 and ANSYS software. For model optimization, FEA results were compared with analytical design.

[8] **Vinay Dilliwar1 , Sankalp Verma2 ,** “Modification of Manufacturing Process of Kingpin for Steering Assembly of Heavy Motor Vehicles and its Analysis”. Conventional design of kingpin comes with a cross hole in the center of pin. But when the vehicle got loaded and travelled across hilly areas it was found that vehicle developed field failure from the center of kingpin. This was due to low core hardness required in the cross-hole area. In the proposed design this cross hole is eliminated which was used only for greasing. By eliminating this cross-hole manufacturing cost also got reduced and improved lead of manufacturing also. New kingpin without cross-hole is manufactured using standard manufacturing techniques which includes process flow diagrams and PFMEA. Nondestructive testing of new design of kingpin is also conducted using MPI testing.

[9**] J.B. Marcomini1 , C.A.R.P. Baptista2** , “Failure Analysis of a Hot Forged SAE 4140 Steel Kingpin”. The fifth wheel and kingpin connection system, a critical part of heavy vehicles, provides the link of tractor and trailer. The kingpin is usually manufactured by hot forging. A part manufactured by this process was assembled in a fifth wheel of an off-road truck and presented an early failure. The truck was used in a quarry until the kingpin failure, three months later. One process issue that can occur in hot forged products is a poor grain structure due to overheating, burning and cavitation. The Scanning Electron Microscopy (SEM) analysis of the failed part showed the presence of cavitation. However, the failure analysis results evinced that cavitation was not the main cause of the fracture, but a combination of wear, impact fatigue and overload.

CHAPTER-3

SELECTION OF MATERIAL

From the journals we have observed that, there is little focus on the composite materials. In this work we are going to focus on the taking the composite material.we have taken for the analysis of front axle are not taken in that journals. From our studies, we suppose that the materials we have taken are suit for the purpose of the front axle design. So we are gone through the analysis of the axle with the materials we have taken here. The general material used for the design of front axle is Mild steel. The results obtained for the other 3 materials are then compared with the Mild steel material.

1. Mild steel

2. Structural steel

3. AlSiC

4. Gray Cast Iron

5. Nickel steel

3.1 Mild steel: Mild steel usually contains 40 points of carbon at most. One carbon point is 0.01 percent of carbon in the steel. This means that it has at most 0.4 percent carbon. Most steels have other alloying elements other than carbon to give them certain desirable mechanical properties. 1018 steel, a common type of mild steel, contains approximately 0.6 percent to 0.9 percent manganese, up to 0.04 percent phosphorus, and up to 0.05 percent Sulphur. Varying these chemicals affects properties such as corrosion resistance and strength.

Mild steel is very strong due to the low amount of carbon it contains. In materials science, strength is a complicated term. Mild steel has a high resistance to breakage. Mild steel, as opposed to higher carbon steels, is quite malleable, even when cold. This means it has high tensile and impact strength. Higher carbon steels usually shatter or crack under stress, while mild steel bends or deforms.

Benfits:

- High tensile strength.
- High impact strength.
- Good ductility and weldability.
- A magnetic metal due to its ferrite content.
- Good malleability with cold-forming possibilities.
- Not suitable for heat treatment to improve properties.

3.2 Structural steel: Structural steel integrates ductility with strength, enabling it to absorb large amounts of energy prior to failing. Structural steel has high levels of elasticity, strength, ductility, and corrosion resistance that helps in vehicle building. This robust material is used for engineering and manufacturing of transport vehicles such as trucks, aircrafts, ships, trains, etc. Structural steel is also used in manufacturing other transport related components such as rails, ship and jet engine components, aircraft undercarriages, anchor chains, etc.

- speed of construction.
- safety.
- value for money.
- robustness and ductility.
- prefabrication.
- reduced weight.
- architectural expression.
- configuration adaptability.

3.3 Aluminum-Silicon Carbide (Al-SiC) is a metal-ceramic composite material consisting of silicon carbide particles dispersed in a matrix of aluminum alloy. It combines the benefits of high thermal conductivity of metal and low CTE (coefficient of thermal expansion) of ceramic. With its composite features, Al-SiC is an advanced packaging material for high technologythermal management. Al-SiC is compatible with a wide range of metallic and ceramic substrate and plating materials used in microelectronic packaging for aerospace, automotive, microwave applications.

Benefits:

- tailor-made coefficient of thermal expansion (eliminates thermal cycling fatiguedue to CTE mismatch).
- high thermal conductivity (conducts heat almost like aluminum)
- light weight and strong(almost as light as aluminum but stronger, it is excellent foraerospace, automotive and other mobile applications).

3.4 Gray cast iron:

It is made by remelting pig iron. It is an alloy of Carbon and Iron. Small amounts of Silicon, Phosphorus, Manganese and Sulfur are also present in it. The reasons behind its popularity are: ability to make complex structures and low cost. In addition, the excellent properties of Grey Cast Iron have made it one of the most widely used alloys. Its properties are

- High compressive strength
- Tensile strength
- Resistence to deformation
- Low melting point

- Resistance to oxidation

3.5 Nickel steel: The 9% nickel steel is a low-carbon nickel alloy steel with nominal composition of 9Ni, which is particularly used for low-temperature or cryogenic pressure vessels and plants for processing, transportation, and storage of LNG down to -196°C [-320°F]. The steel is originated from International Nickel Company's lab in 1940's. It has relatively higher strength and allowable stress than austenitic stainless steels, aluminum and copper. The 9% nickel steel exhibits excellent fracture toughness even at cryogenic temperatures. It also has high melting point and retention of strength at elevated temperatures, which provide fire-safe integrity for the LNG pressure vessels. Besides, the relatively low thermal expansion, low thermal conductivity and good corrosion resistance make the steel applicable for ethylene, methane, LNG, oxygen, and nitrogen applications.

3.6 Material Properties:

S.no	**Material**	**Density (kg/m^3)**	**YoungsModulus (GPa)**	**Possion ratio**	**Tensile Ultimate Strength (MPa)**
1.	Mild steel	8870	205	0.29	340
2.	Structural steel	7850	200	0.3	460
3.	AlSiC (Al-85%,SiC-15%)	2784	99.97	0.292	310
4.	Gray Cast Iron	7200	110	0.28	240
5	Nickel steel	7860	186	0.3	770

Table-A Material properties of all Materials

CHAPTER-4

MODELLING OF FRONT AXLE

4.1 Introduction to Solid works:

SolidWorks (stylized as SOLIDWORKS) is a solid modelling Computer Aided Design (CAD) and Computer Aided Engineering (CAE) computer program that runs onmicrosoft Windows. SolidWorks is published by Dassault Systems

According to the publisher, over two million engineers and designers at more than 165,000 companies were using Solid Works as of 2013. Also according to the company, SolidWorks is a solid modeller, and utilizes a parametric feature based approach to create models and assemblies. The software is written onparasolid-kernel.

The Solid works software is mechanical design automation application that takes advantage of Microsoft Windows graphical interface. This software make it possible for designers to quickly sketch out ideas, experiment with features and dimensions, and produce models and detailed drawings

Parameters refer to constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric parameters, such as line lengths or circle diameters, or geometric parameters, such as tangent, parallel, concentric, horizontal or vertical, etc. Numeric parameters can be associated with each other through the use of relations, which allows them to capture design intent.

Design intent is how the creator of the part wants it to respond to changes and updates. Design intent determines how you want your model to react as a result of any changes you need to make to the model. Design intent is primarily about planning. Deciding how to create the model determines how changes affect the model. The closer your design implementation is to your design intent, the greater the integrity of the model.

Various factors contribute to the design process, including:

Current needs. Understanding the purpose of the model to design it efficiently.

Future considerations. Anticipate potential requirements to minimize redesign efforts when changing the model.

The design process usually involves the following steps:

- Identify need
- Conceptualize model based on identified needs
- Develop model based on the concepts
- Prototype the model
- Construct the model
- Edit the model if needed

Features refer to the building blocks of the part. They are the shapes and operations that construct the part. Shape-based features typically begin with a 2D or 3D sketch of shapes such as bosses, holes, slots, etc. This shape is then extruded or cut to add or remove material from the part. Operation-based features are not sketch-based, and include features such as fillets, chamfers, shells, applying draft to the faces of a part, etc

Building a model in SolidWorks usually starts with a 2D sketch (although 3D sketches are available for power users). The sketch consists of geometry such as points, lines, arcs, conics (except the hyperbola), and splines. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity, and concentricity. The parametric nature of SolidWorks means that the dimensions and relations drive the geometry, not the other way around. The dimensions in the sketch can be controlled independently, or by relationships to other parameters inside or outside of the sketch

The SolidWorks interface offers a wide range of tools. You'll find more than one way to do almost everything. There is no single best way to use the interface; this book generally shows the most standardized and quickest methods.

In this chapter, I start by displaying the entire default interface, but in the rest of the book, I will show only a reduced interface, mainly to save space and keep the focus on the graphics window.

After you have mastered the various interface elements and customized your SolidWorks installation, working with the software will become much more efficient and satisfying.

4.2 File Format:

SolidWorks files (previous to version 2015) use the Microsoft Structured Storage file format. This means that there are various files embedded within each SLDDRW (drawing files), SLDPRT (part files), SLDASM (assembly files) file, including preview bitmaps and metadata sub-files. Various third-party tools (see COM Structured Storage) can be used to extract these sub-files, although the subfiles in many cases use proprietary binary file formats.

SolidWorks allows saving 3D Model information in *.step format, which lets the model be displayed and modified in other platforms from other vendors.

4.3 Modeling of Front Axle:

- The model (Fig. 1) is constructed according to the scope of the project considering the spring seat for the application of force and the ends with circular holes where the axle is fixed.
- The axle is designed using Solidworks 2020, where the basic design and the extreme constraints of the model were considered.
- The I-section of the axle is constructed by using the line command in the right plane and dimensions are given using the smart dimensions.

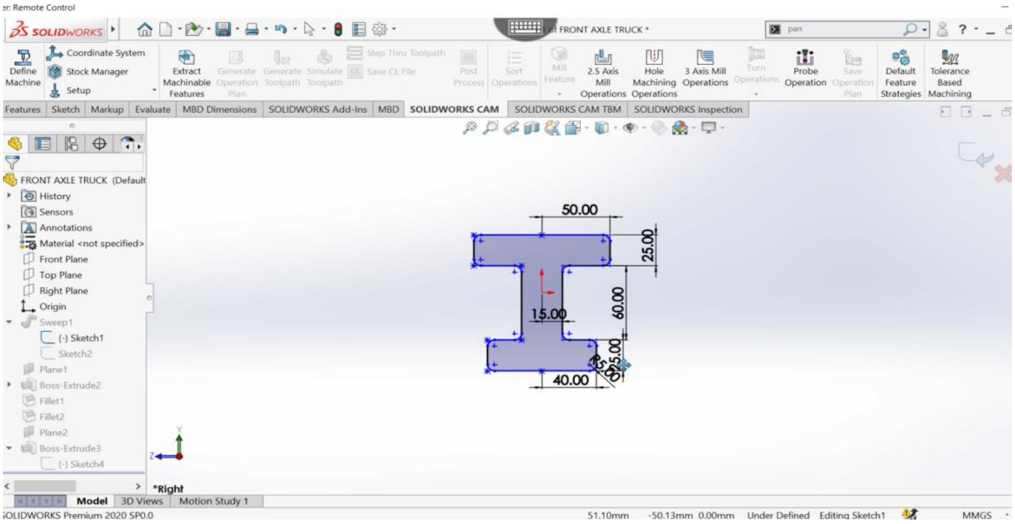

Figure-2 Cross section of Front Axle

- Now select the front plane and draw the sweep way to the I-section as the dimensions shown in the figure.

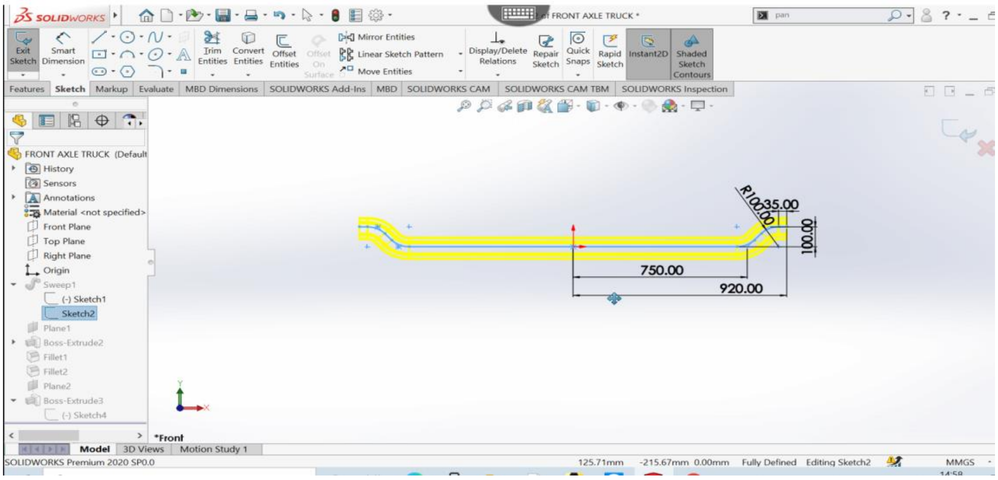

Figure-3 Length of Half portion of Front Axle

- After that, select the sweep option and select the I-section and the way to sweep and click on the OK.
- Then mirror the swept area to the other side inorder to get the complete I-section of the axle.
- For the circular holes select the reference plane which is on the one end of the I-section.
- Then draw as shown the figure below.

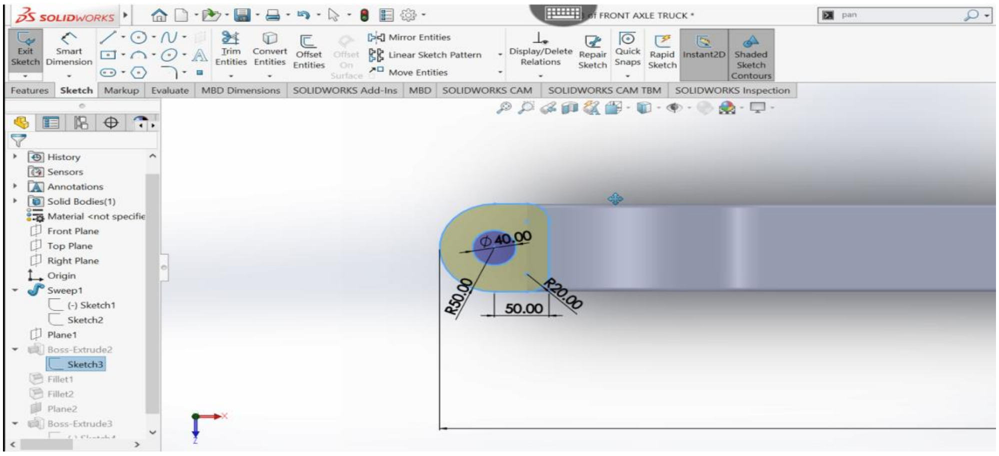

Figure-4 Dimensions of Rib

- Now extrude the sketch to the length of the I-section.
- Mirror the extruded portion to the otherside by using the mirror command.
- Now use the fillet command and fillet the sharp edges having a radius of 10mm.
- Then the modeling of front axle is completed.
- Save the model in iges format.

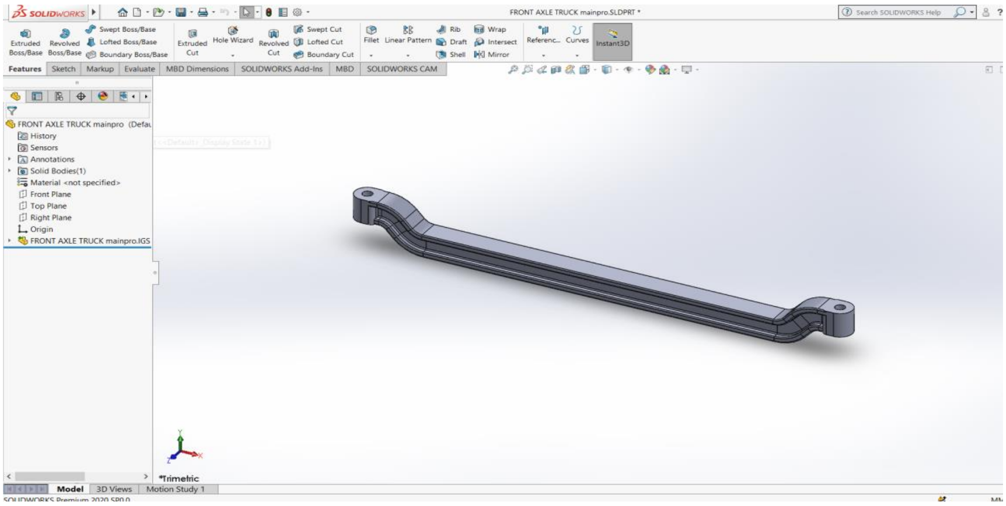

Fig- 5 Model of Front Axle

4.4 METHODLOGY

Calculation of weight

For Light Truck

Laden Weight = 6000 kg.

Actual weight coming on axle is,

Considering 3g condition (bump load)	=	3 *6000	=	18000 Kg
Total Weight on axle is given by	=	18000 * 9.81	=	176.58 kN
Weight on each spring seat	=	176.28/ 2	=	88.29 KN

CHAPTER-5

FATIGUE ANALYSIS OF FRONT AXLE

5.1 Introduction to ANSYS

Introduction:

ANSYS software, used to simulate interactions of all disciplines of physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic for engineers.

5.2 ANSYS WORKBENCH:

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behavior of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

ANSYS is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. ANSYS is also used in Civil and Electrical Engineering, as well as the Physics and Chemistry departments.

ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping.

With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of a design on the whole behavior of the product, be it electromagnetic, thermal, mechanical etc.

Workbench is a common platform for solving engineering problems. Typical tasks you can perform in Workbench are:

- Importing models from a variety of CAD systems.

- Conditioning models for design simulations using the Design Modeler.
- Performing FEA simulations using Simulation.
- Optimizing designs using DesignXplorer or DesignXplorerVT.

The underlined words above are the names of different processors within ANSYS Workbench. Basically, you will use the DesignModeler to create the geometry and the Simulation to set up the materials, FE-mesh, loads and boundary conditions, solve the problem and analyze the results. The standard interface ANSYS Classic (used in the first computer workshop) is still the core of ANSYS. ANSYS Workbench is a new modern interface with more up to date functions such as, for example, the integration of CAD geometries

5.3 Design Modeler

Design Modeler is designed to be used as a geometry editor of existing CAD models. Design Modeler is a parametric feature-based solid modeler designed so that you can intuitively and quickly begin drawing 2D sketches, modeling 3D parts, or uploading 3D CAD models for engineering analysis preprocessing.

Simulation

Use the Workbench Simulation module to define your model's environmental loading conditions, solve the simulation, and review results in various formats depending on the type of simulation.

Generic Steps to Solving Any Problem in Ansys workbench

Like solving any problem analytically, you need to define (1) your solution domain, (2) the physical model, (3) boundary conditions and (4) the physical properties. You then solve the problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in an otherwise too complex situation. Below describes the processes in terminology slightly more attune to the software.

In this section, the modeling and analysis of the chairless chair will be described. Start the ANSYS Product Launcher. Select a working directory for storing your model data and launch ANSYS Workbench. You will see the software outfit as Figure.

Topics Include:

- Workbench GUI
- Design Modeler
- Overview of FEA
- Engineering Data

- Meshing
- Parametric Modeling

5.4 Ansys Workbench Fatigue Module

While many parts may work well initially, they often fail in service due to fatigue failure caused by repeated cyclic loading. Characterizing the capability of a material to survive the many cycles a component may experience during its lifetime is the aim of fatigue analysis. In a general sense, Fatigue Analysis has three main methods, Strain Life, Stress Life, and Fracture Mechanics; the first two being available within the ANSYS Fatigue Module.

Total Life Approaches

- High cycle fatigue (Stress life)
- Low cycle fatigue (Strain life)

5.5 Fatigue life

- It is the avaliable life for the given fatigue analysis.
- Fatigue life can be over the whole model or on parts, surfaces, vertices and edges.
- If loading is of constant amplitude, this represents the number of cycles until the part will fail due to fatigue.
- If loading is non-constant, this represents the number of loading blocks until the failure.

5.6 Fatigue Damage:

- Fatigue damage is defined as the design life/available life.
- The default design life may be set manually.
- For Fatigue Damage, values greater than 1 indicate failure before the design life is reached.

5.7 Fatigue Safety Factor:

- It is the factor of safety with respect to a fatigue failure at a given design life.
- The maximum Factor of Safety is 15.
- For Fatigue Safety Factor, values less than one indicate failure before the design life is reached.

5.8 Fatigue Sensitivity:

- It shows how the fatigue results change as a function of the loading at the critical location on the model.
- Sensitivity may be found for life, damage, or factor of safety.
- The user may set the number of fill points as well as the load variation limits.

- For example, the user may wish to see the sensitivity of the model's life if the FE load was 50% of the current load up to if the load 150% of the current load.

5.9 Procedure to Solve in ANSYS Workbench:

Step 1: Launching ANSYS Workbench

The ANSYS installation has many packages included. For this tutorial, we will be using ANSYS Workbench.

• Start menu > ANSYS 18.2> Workbench 18.2

The Workbench Project Window will open.

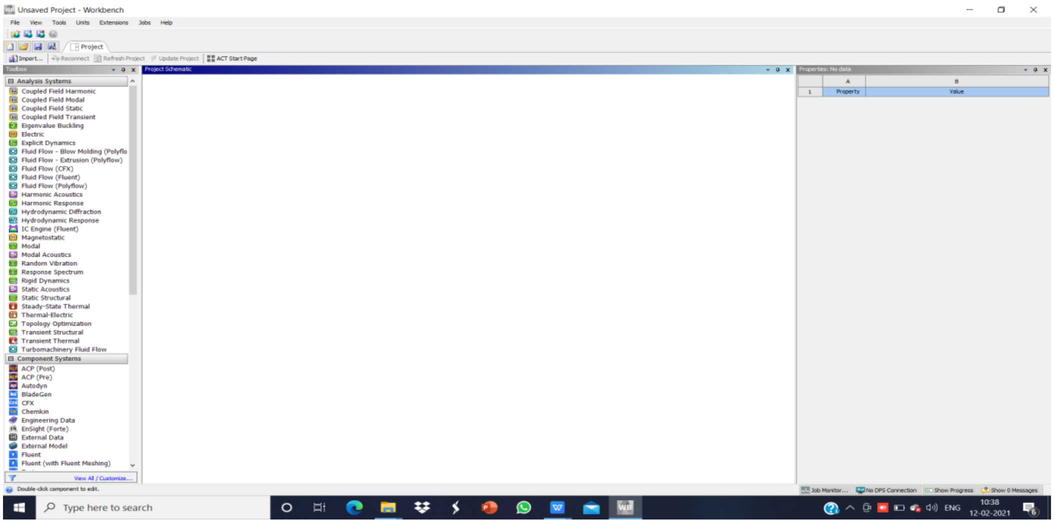

Figure-6 shows home page for the ansys workbench

Step 2: Pre-processing (Setting up the Model)

Our analysis is a Static Structural analysis. It can be found in the Toolbox on the left, and needs to be added to the Project Schematic by either double clicking it, or dragging it into the pane.

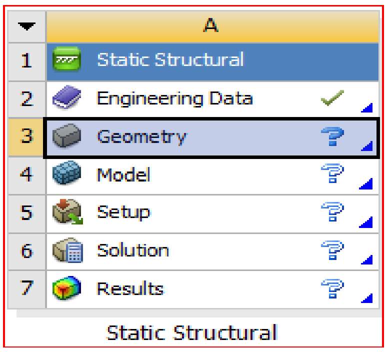

Figure-7 Tool box

The Static Structural component and all of its modules will be created. The modules are similar to those in ANSYS MAPDL. They outline the steps that are required to complete a finite element analysis.

- Engineering Data module is used to define the material properties.

- Geometry module opens the DesignModeler application, which can be used to import CAD models from other software like SolidWorks or to sketch a new 2D or 3D geometry.

- Model, Setup, Solution, and Results modules opens the Mechanical application, which can be used to set up and solve the simulation (includes meshing, load and boundary condition applications, solving, and results).

Step 2A: Engineering Data

- Double-click Engineering Data. What you see in this window may differ from the screenshot below. In here, you can add a new material by defining a new material entry for Mild Steel.

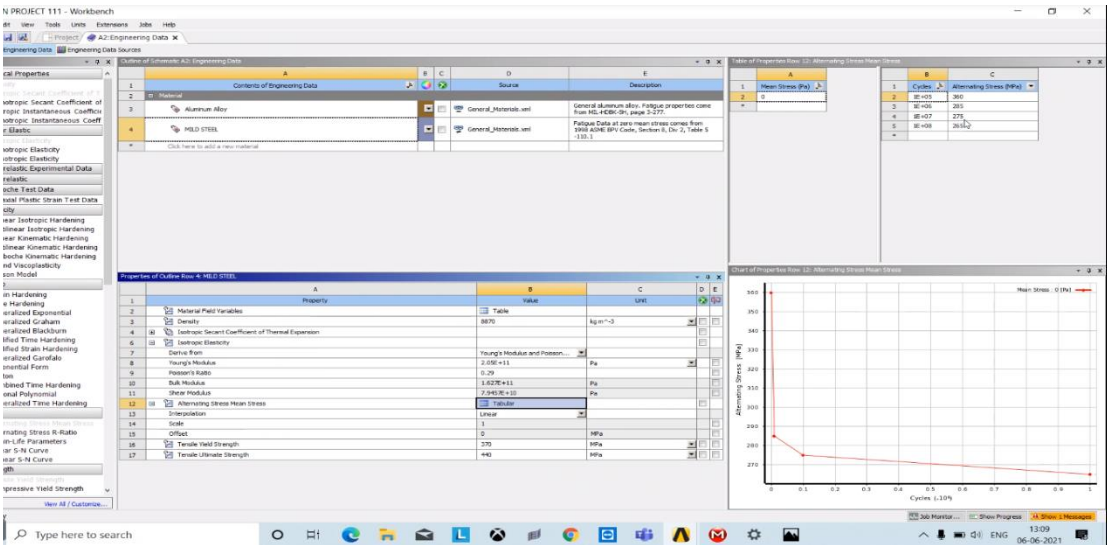

Figure-8 shows engineering data for the given material

- Exit Engineering Data by closing the tab at the top of the window and return to the main project.

Step 2B: Geometry

- By default, ANSYS Workbench will analyze the problem in 3D. In this problem, we

- are modelling a plane stress scenario, which allows us to reduce the analysis down to a 2D problem.
- Import the pre designed geometry in solid works in iges format directly.

- Enter the DesignModeler application by double-clicking on the Geometry module.

- Design Modeler is similar to a CAD program. Here you can work with the model and create sketches by clicking on the tabs on the left.

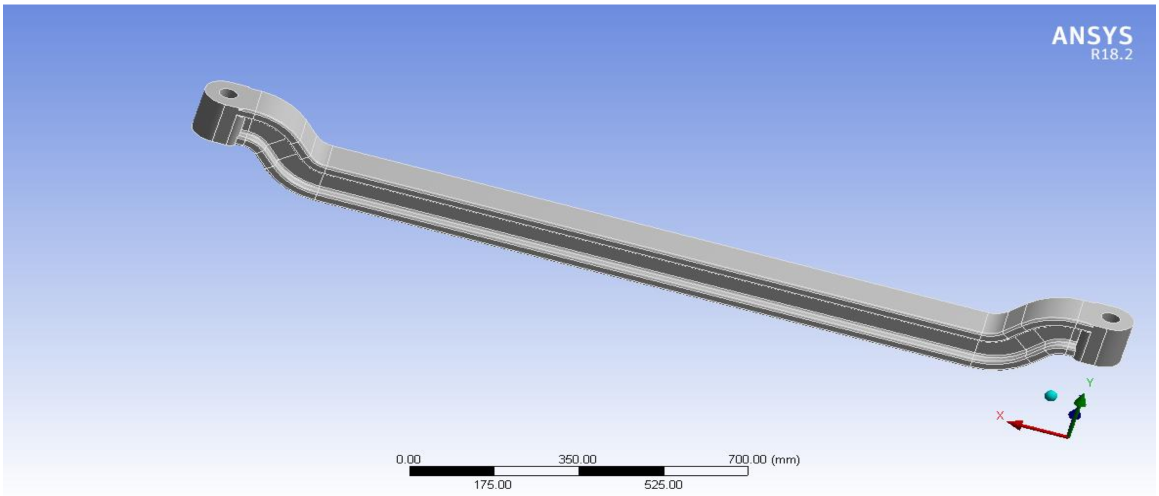

Figure-9 Imported Model of Front Axle

Step 2C: Model

Enter the Mechanical application by double-clicking on the Model module. At this point, Workbench should attach the geometry that was made in Design Modeler and make it available in the Mechanical application, where we will complete the configuration of this simulation and solve it. At this point, notice that the Mechanical application has two panes on the left: “Outline” and “Details”. The Outline pane contains a tree with all the settings you add to the model. The Details pane will provide options for each of these settings that you can change. After configuring Workbench to run this Static Structural simulation in 2D, the Mechanical application allows for the use of various 2D assumptions, including the plane stress and plane strain assumptions.

Step 2D: Meshing

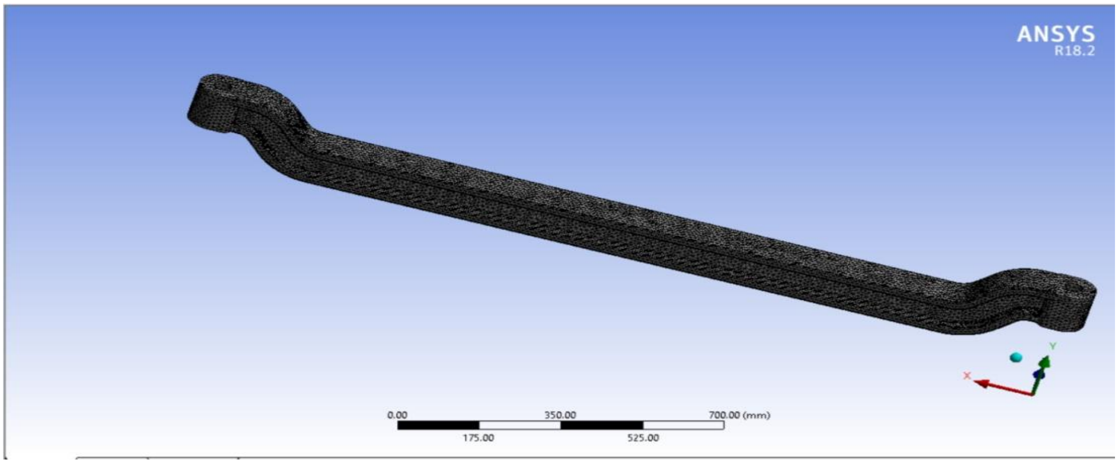

Figure-10 Model of Front Axle meshing of size 10mm

Applying the Boundary conditions:

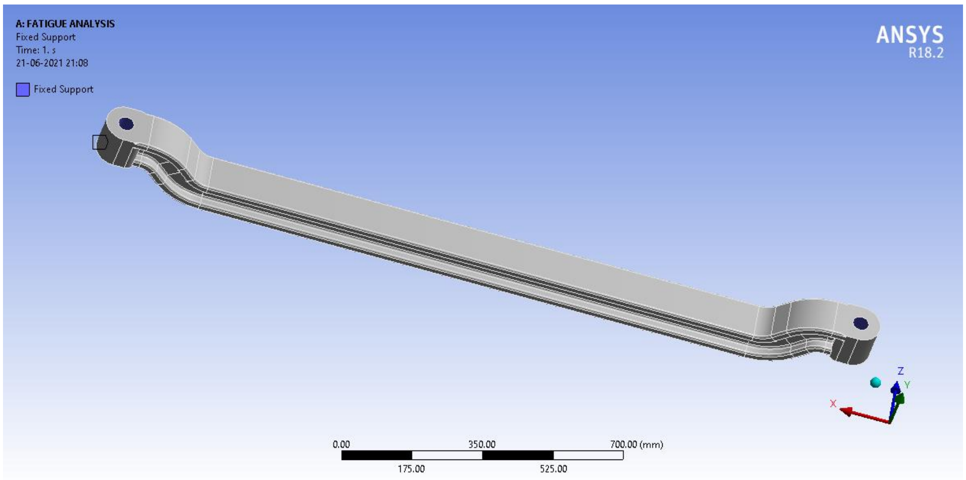

Figure-11 Fixed support at the two holes

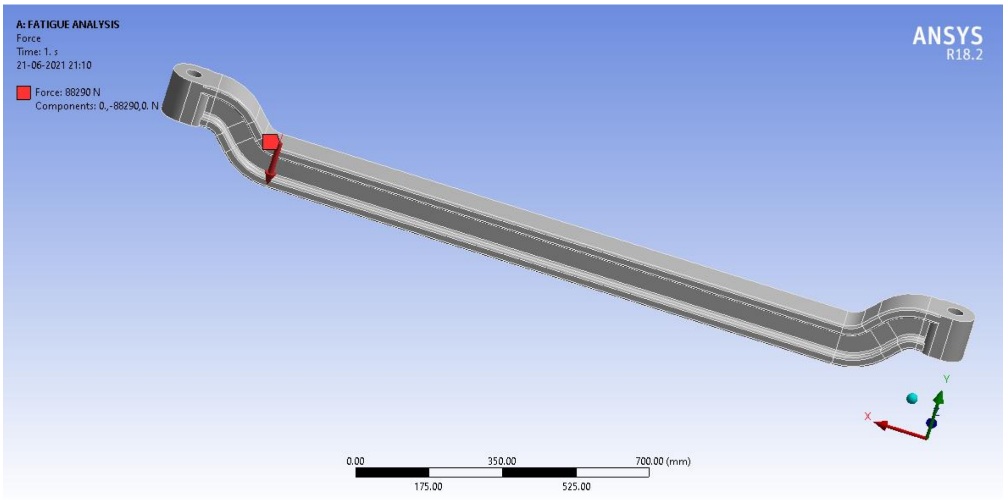

Figure-12 Force 1 at Right corner of front axle in negative y direction

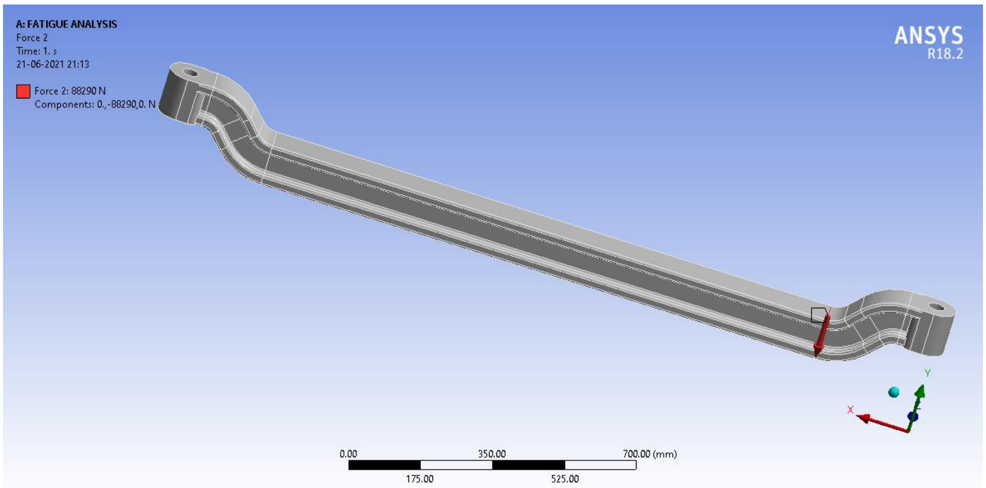

Figure-13 Force 2 at Left corner of front axle in negative y direction

CHAPTER-6

RESULTS AND DISSCUSION

TOTAL DEFORMATION FOR MILD STEEL

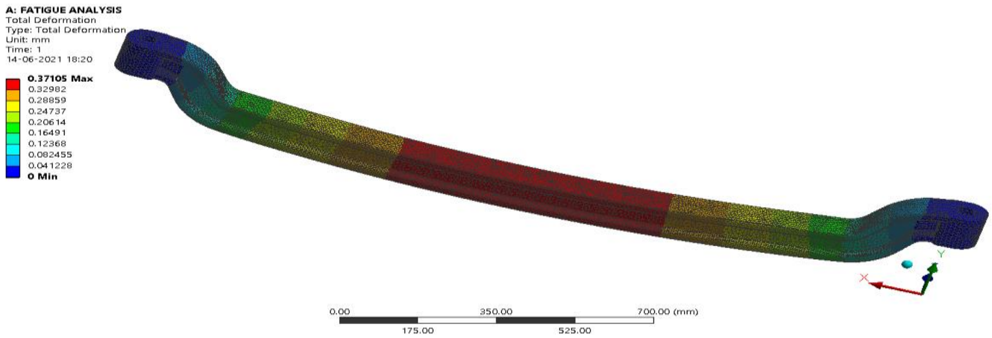

Figure-14 Shows the total deformation of material Mild steel. The maximum deformation is 0.37105 mm/mm.

EQUIVALENT ELASTIC STRAIN FOR MILD STEEL

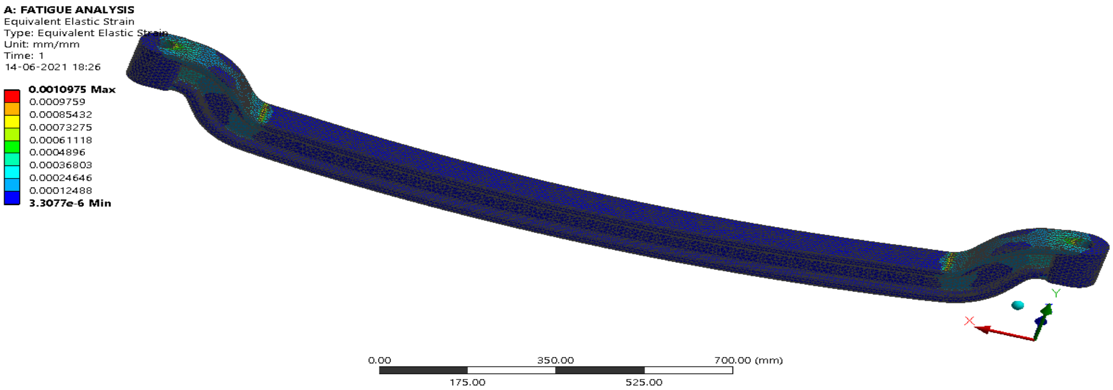

Figure-15 Shows the Elastic strain of material Mild steel. The maximum strain is 0.0010975mm/mm.

EQUIVALENT STRESS FOR MILD STEEL

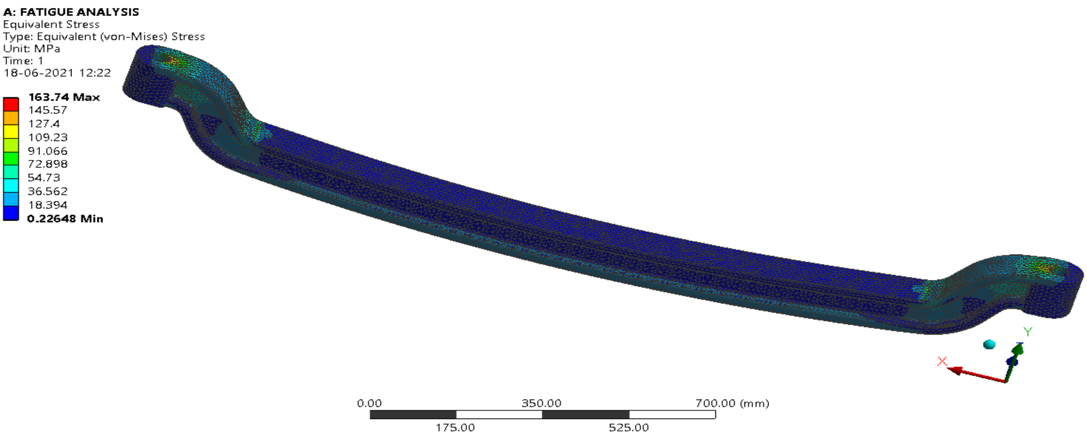

Figure-16 Shows the stress of material Mild steel. The maximum stress is 163.74MPa.

FATIGUE LIFE FOR MILD STEEL

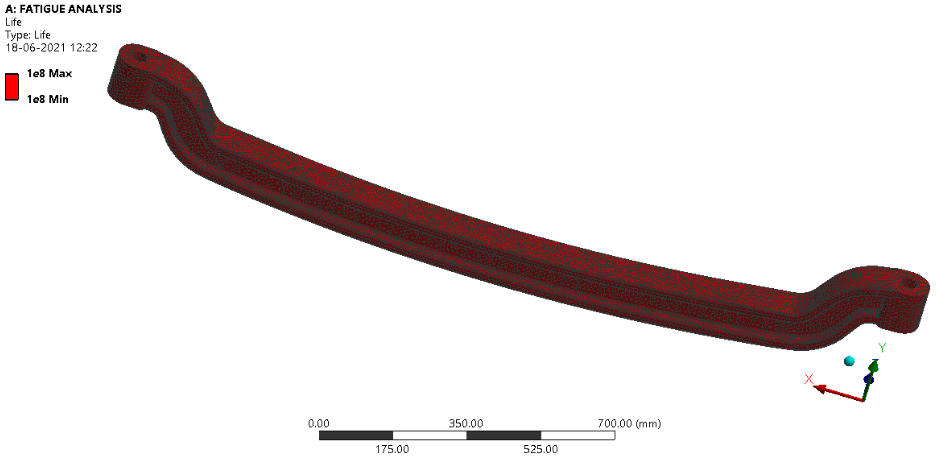

Figure-17 Shows the Fatigue Life of material Mild steel. The maximum value is1E8.

FATIGUE DAMAGE FOR MILD STEEL

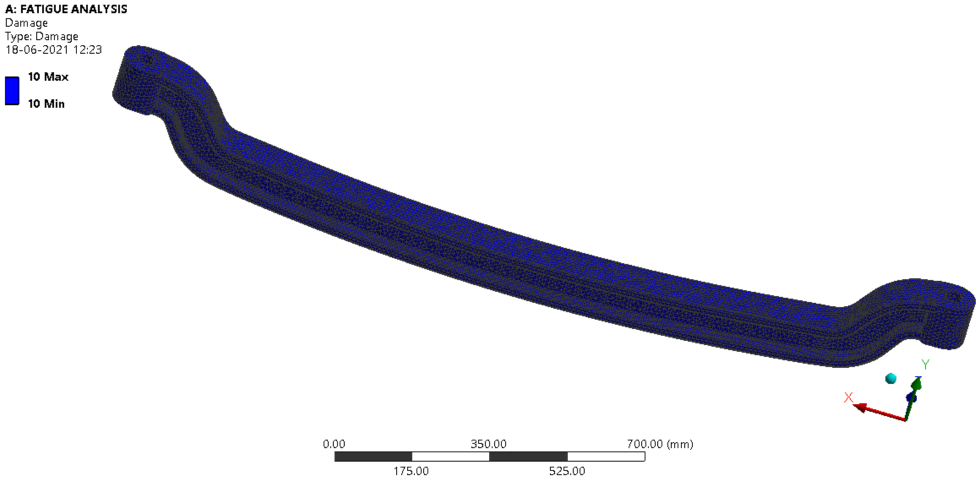

Figure-18 Shows the fatigue damage of material Mild steel. The maximum damage is 10Max.

SAFETY FACTOR FOR MILD STEEL

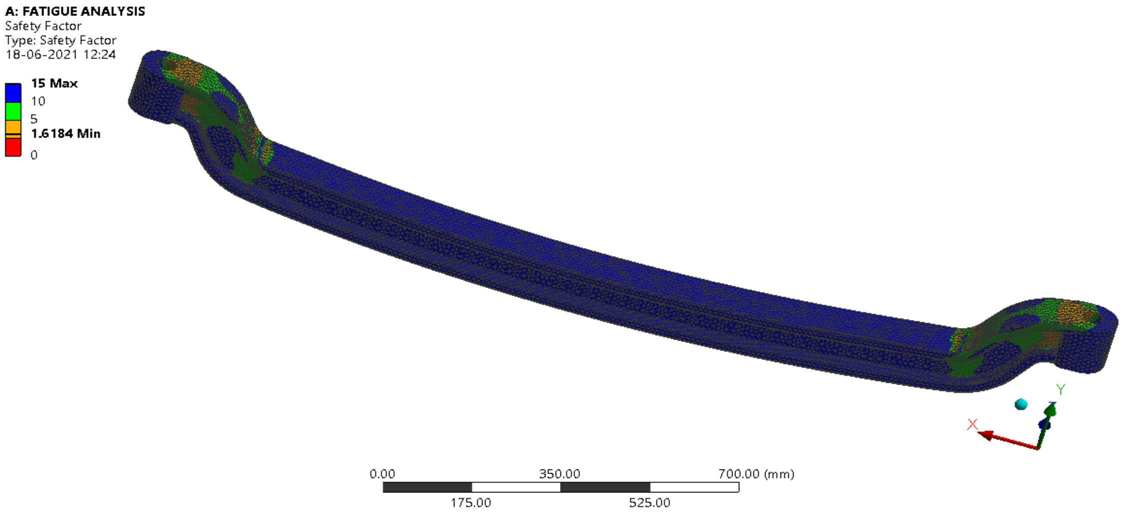

Figure-19 Safety factor of material Mild steel. The maximum safety factor is 15Max.

FATIGUE SENSITIVITY FOR MILD STEEL

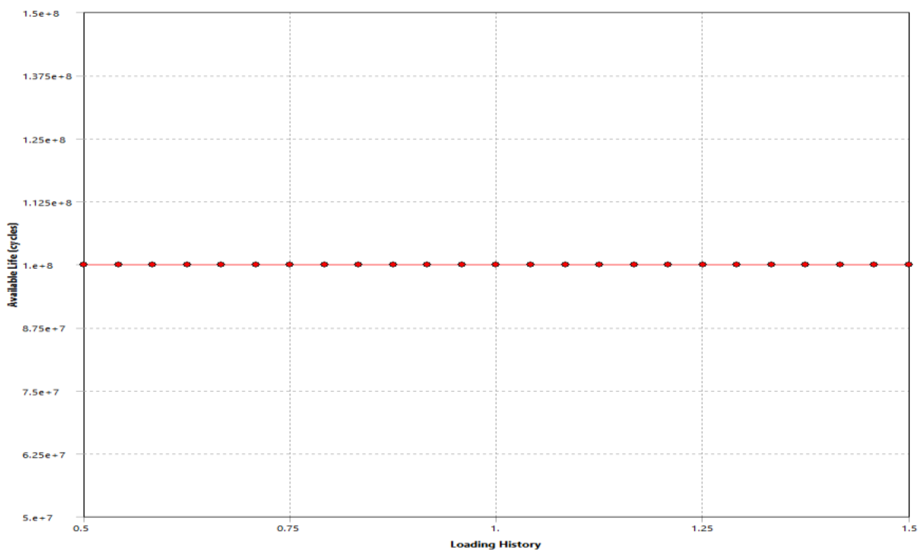

Figure-20 Fatigue Sensitivity for Mild steel

TOTAL DEFORMATION FOR STRUCTURAL STEEL

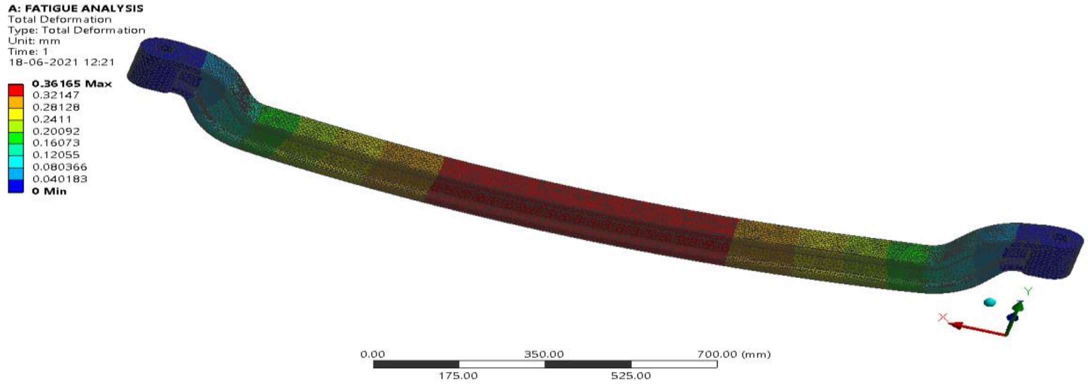

Figure-21 Shows the total deformation of material Structural steel . The maximum deformation is 0.36165 mm/mm.

EQUIVALENT ELASTIC STRAIN FOR STRUCTURAL STEEL

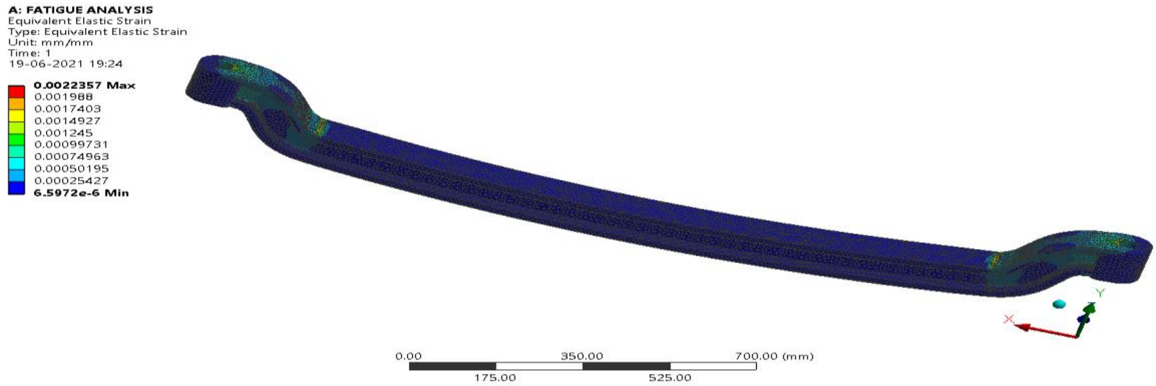

Figure-22 Shows the Elastic strain of material Structural steel. The maximum strain is 0.0022357 mm/mm.

EQUIVALENT STRESS FOR STRUCTURAL STEEL

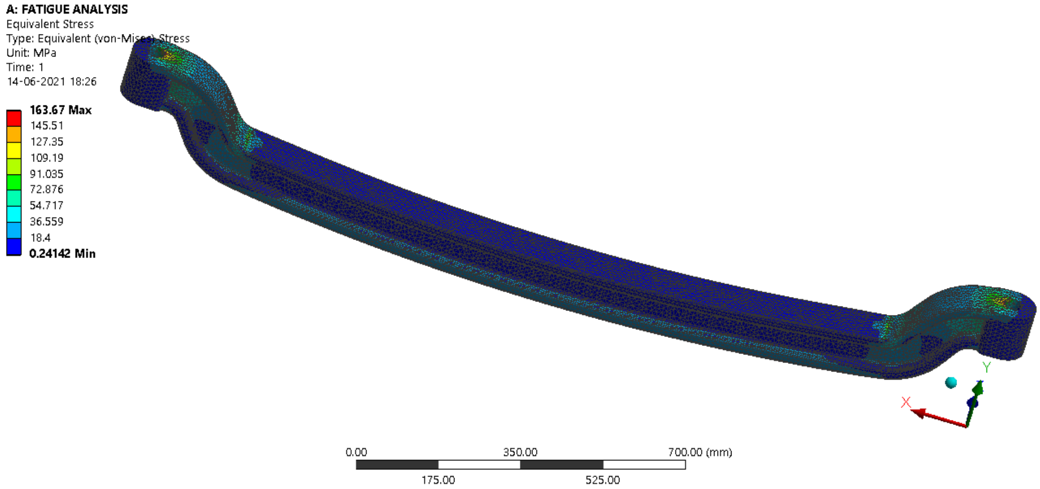

Figure-23 Shows the stress of material Structural steel. The maximum stress is 163.6Mpa

FATIGUE LIFE FOR STRUCTURAL STEEL

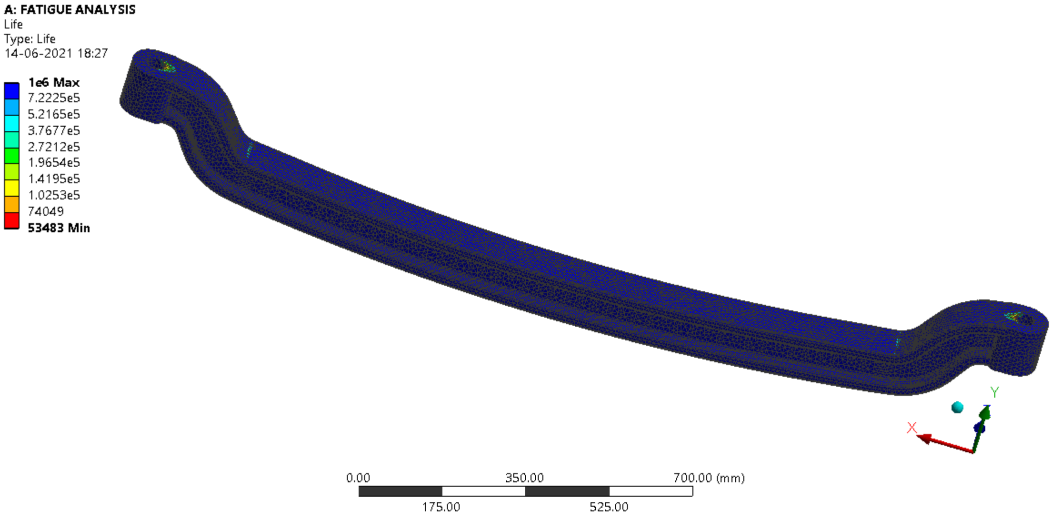

Figure-24 Shows the fatigue life of material structural steel. The maximum life is 1e6 Max

FATIGUE DAMAGE FOR STRUCTURAL STEEL

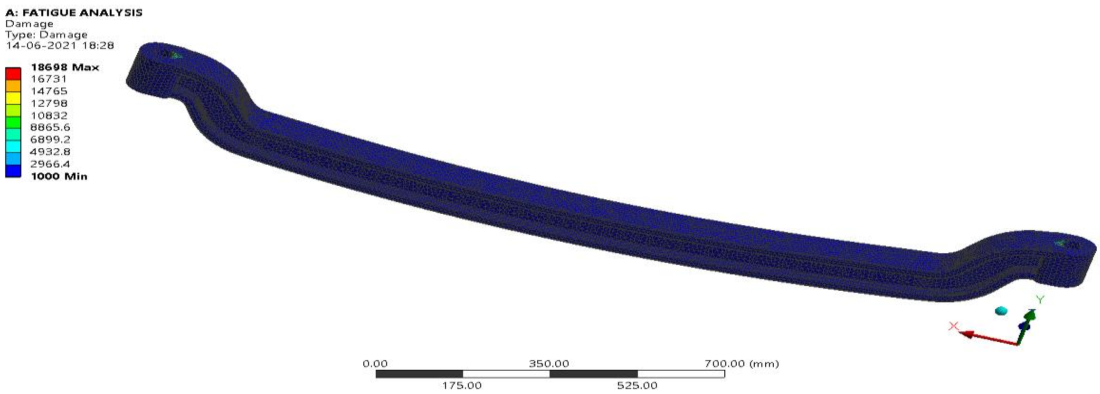

Figure-25 Shows the fatigue damage of material Structural steel. The maximum damage is 18698 Max.

SAFETY FACTOR FOR STRUCTURAL STEEL

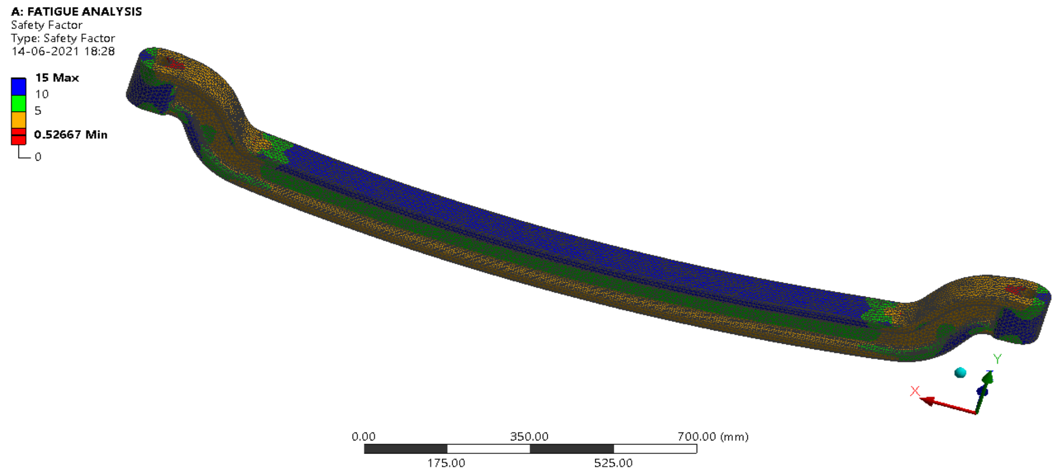

Figure-26 Shows the Safety factor of material Structural steel. The maximum safety factor is 15Max.

FATIGUE SENSITIVITY FOR STRUCTURAL STEEL

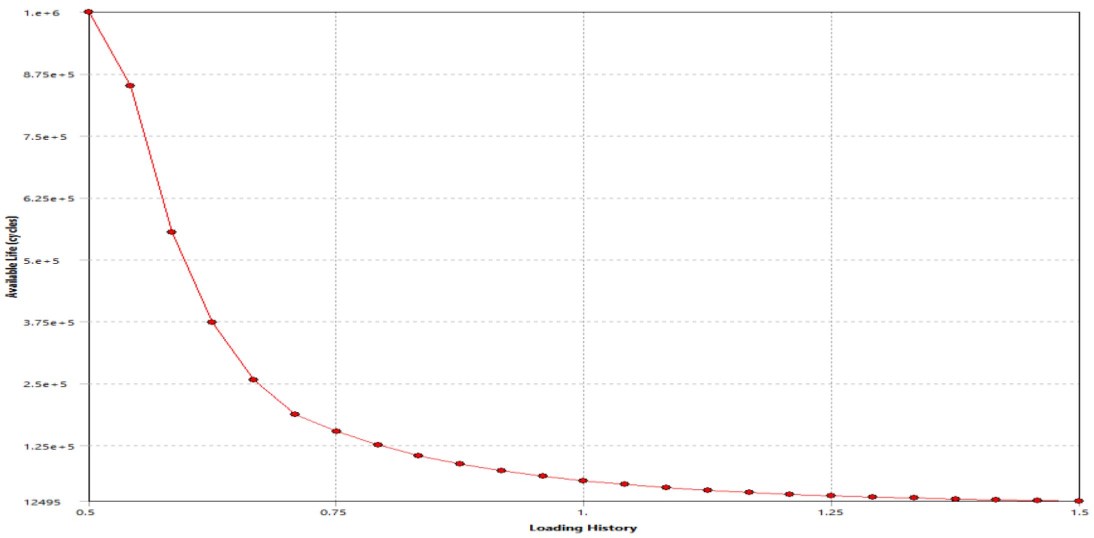

Figure-27 Fatigue Sensitivity For Structural Steel

TOTAL DEFORMATION FOR AlSIC

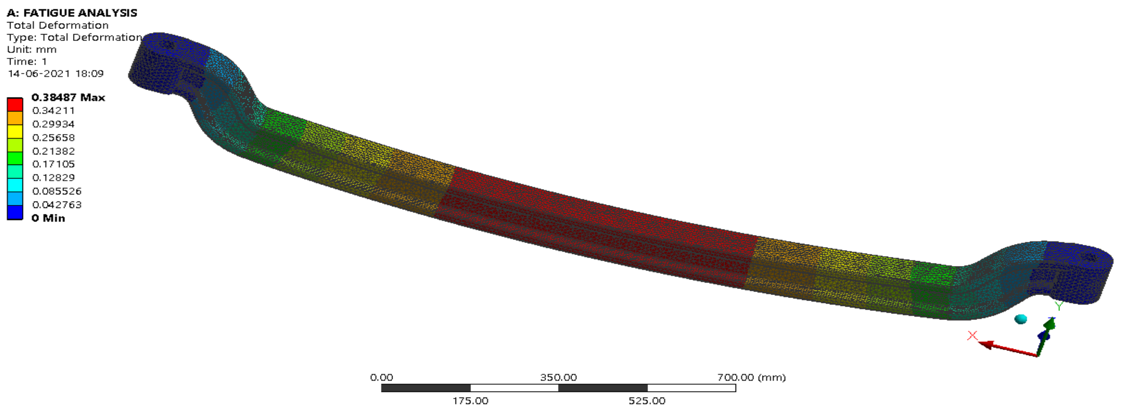

Figure-28 Total deformation of AlSiC .The Maximum deflection is 0.38487mm.

ELASTIC STRAIN FOR AlSiC

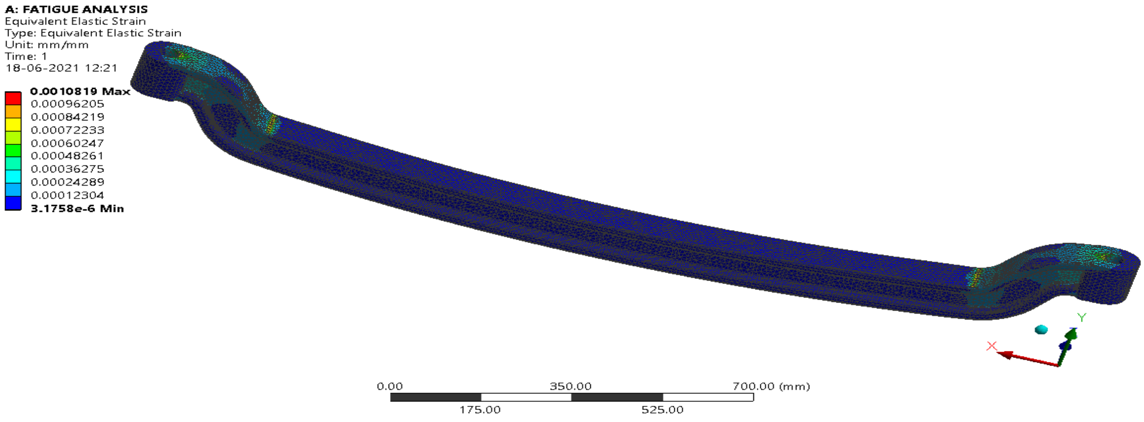

Figure-29 Elastic Strain of AlSiC. The Maximum Elastic strain is 0.0010819mm/mm.

EQUIVALENT STRESS FOR AlSiC

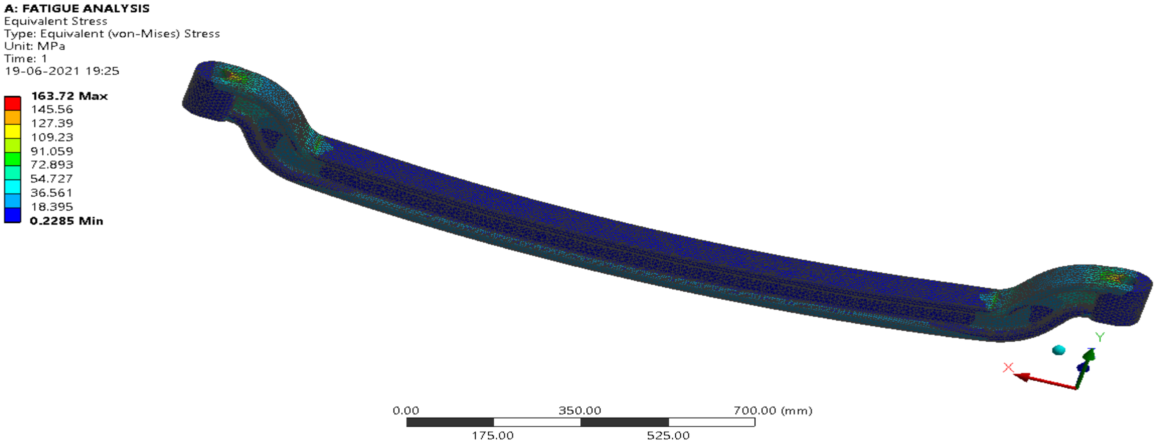

Figure-30 Equivalent Stress of AlSiC. The Maximum Stress is 163.72Mpa.

FATIGUE LIFE OF AlSiC

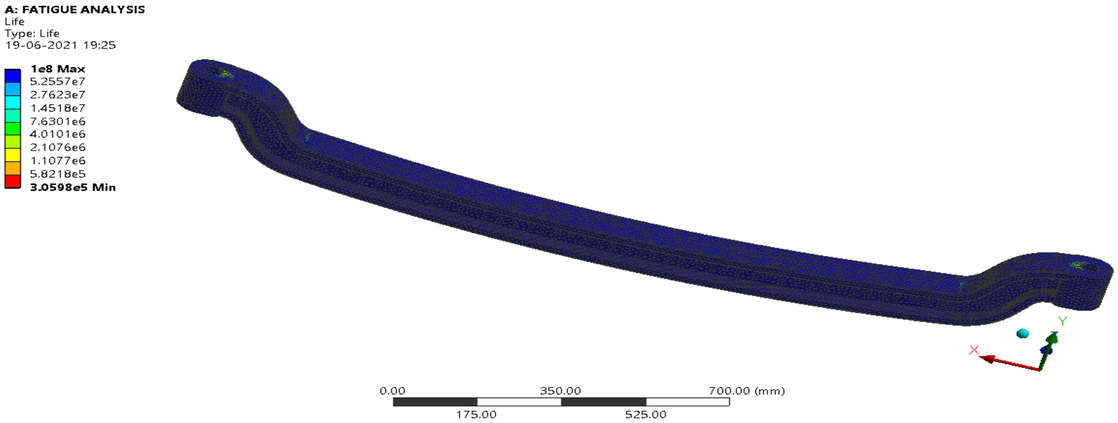

Figure-31 Fatigue Life of AlSiC. The maximum life is 1e8Max

FATIGUE DAMAGE OF AlSiC

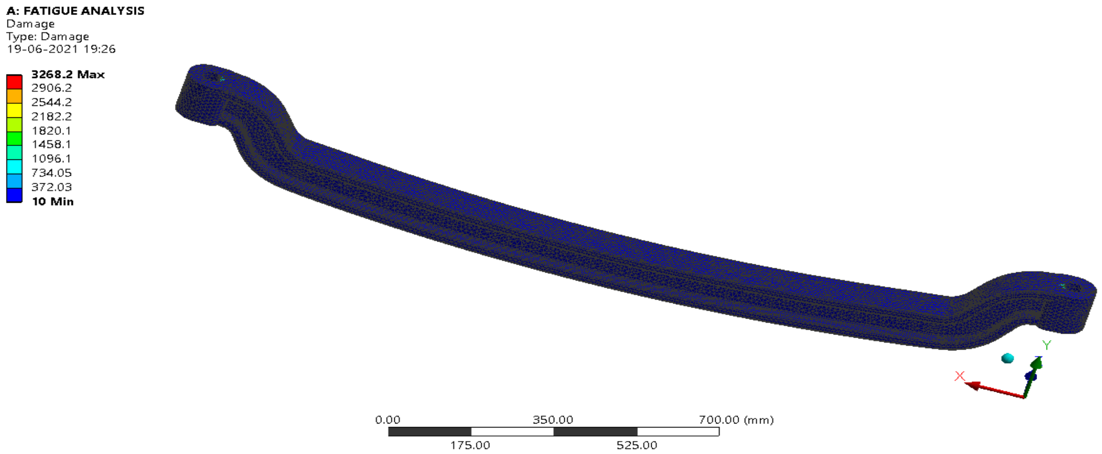

Figure-32 Fatigue damage of AlSiC. The maximum damage is 3268.2 Max

SAFETY FACTOR OF AlSiC

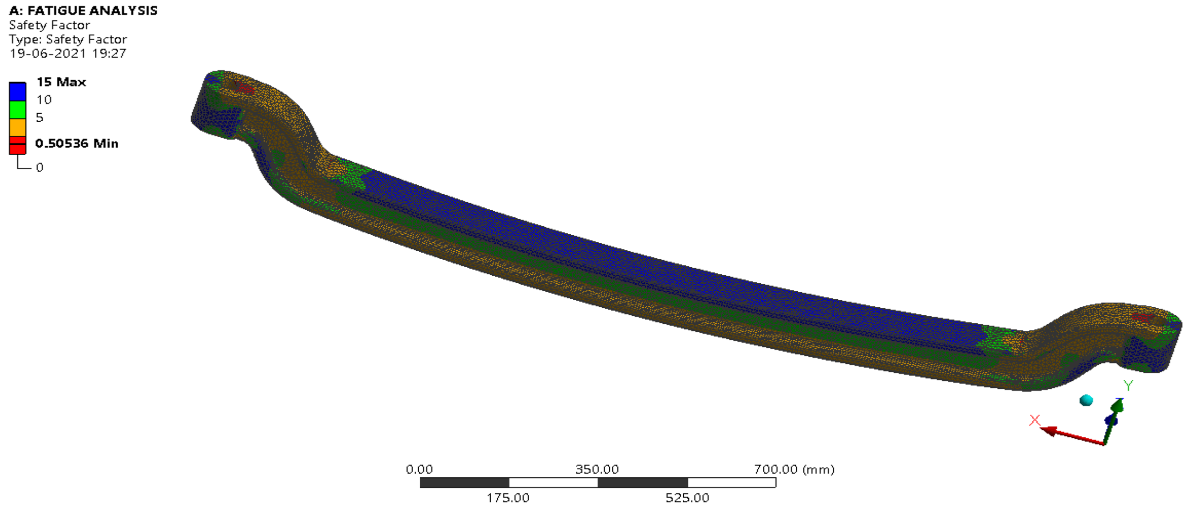

Figure- 33 Safety factor of AlSiC. The maximum Safety factor is 15Max

FATIGUE SENSITIVITY OF AlSiC

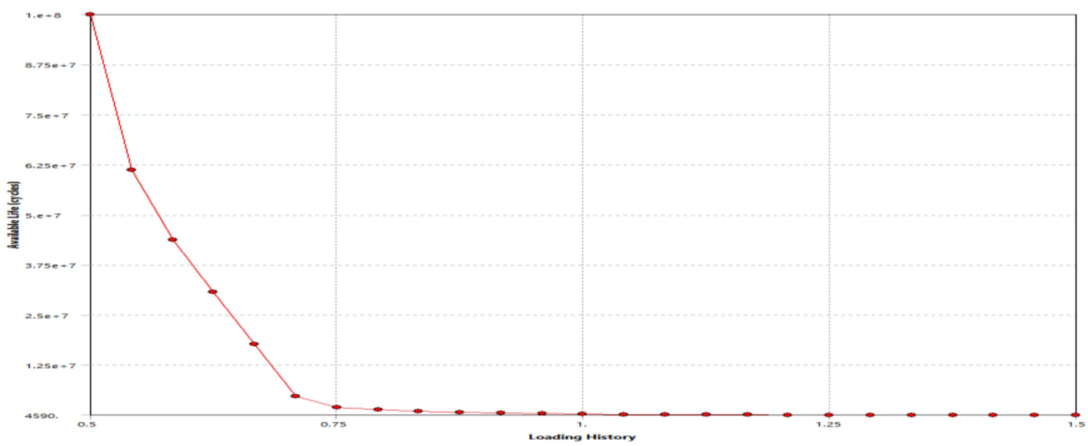

Figure-34 Fatigue Sensitivity of AiSic.

TOTAL DEFORMATION FOR GRAY CAST IRON

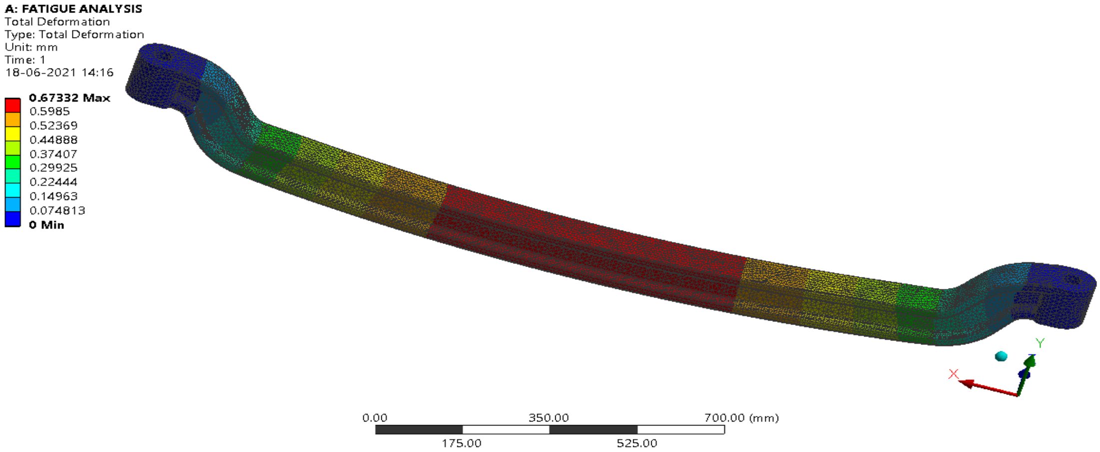

Figure-35 Total deformation of Gray Cast Iron. The maximum value is 0.67332Max

ELASTIC STRAIN OF GRAY CAST IRON

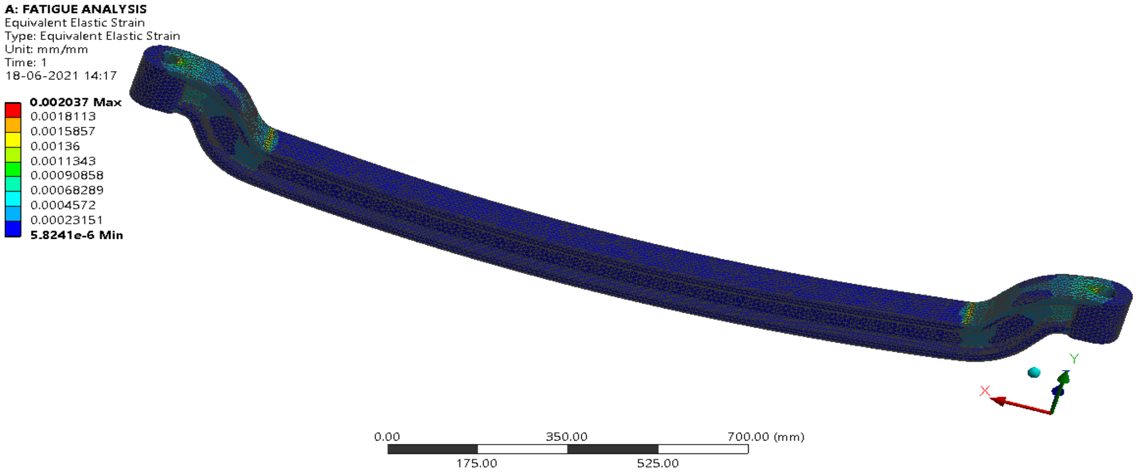

Figure-36 Elastic Strain of Gray Cast Iron. The maximum value is 0.002037Max

EQUIVALENT STRESS OF GRAY CAST IRON

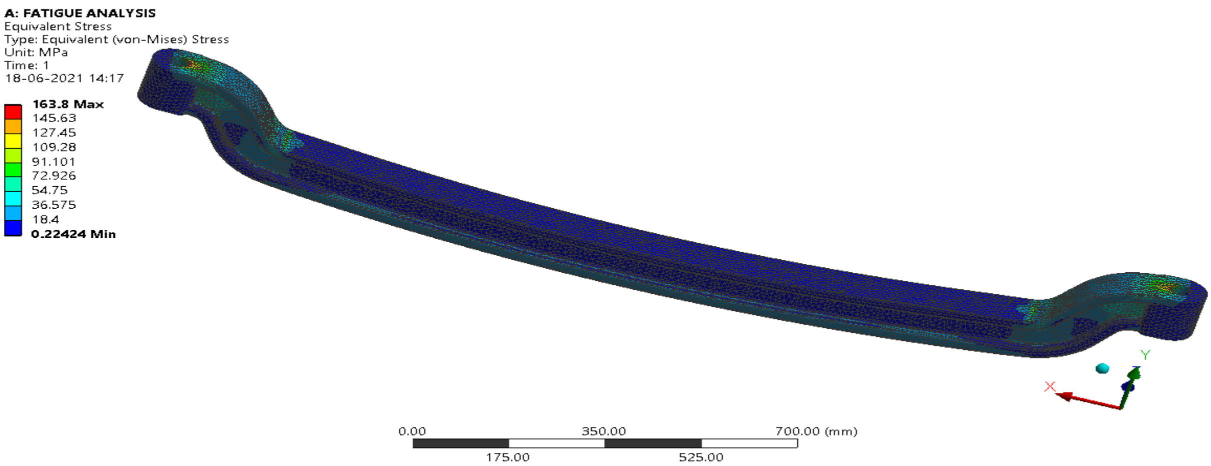

Figure-37 Equivalent Stress of Gray Cast Iron. The maximum value is 163.8 Max

FATIGUE LIFE OF GRAY CAST IRON

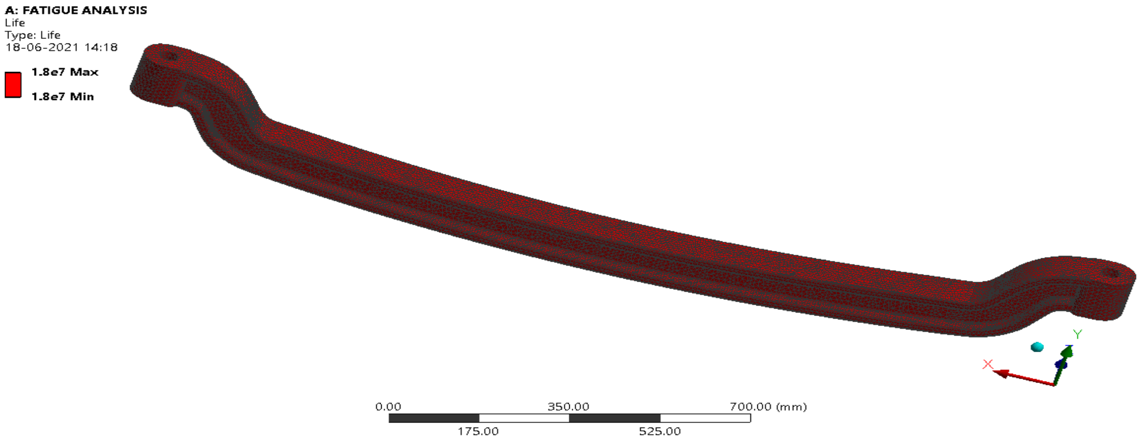

Figure- 38 Fatigue Life of Gray Cast Iron. The maximum value is 1.8e7 Max

FATIGUE DAMAGE OF GRAY CAST IRON

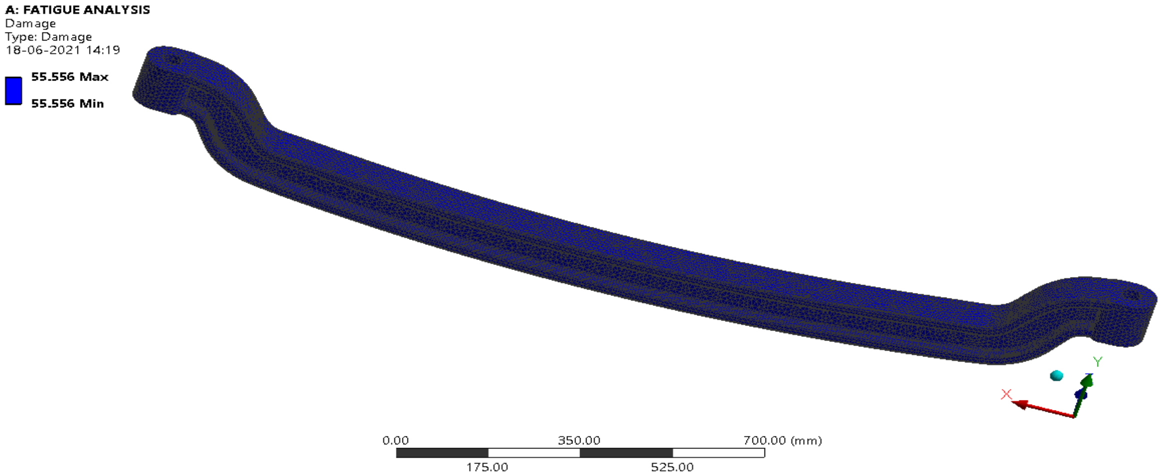

Figure-39 Fatigue damage of Gray Cast Iron. The maximum value is 55.556 Max

SAFETY FACTOR FOR GRAY CAST IRON

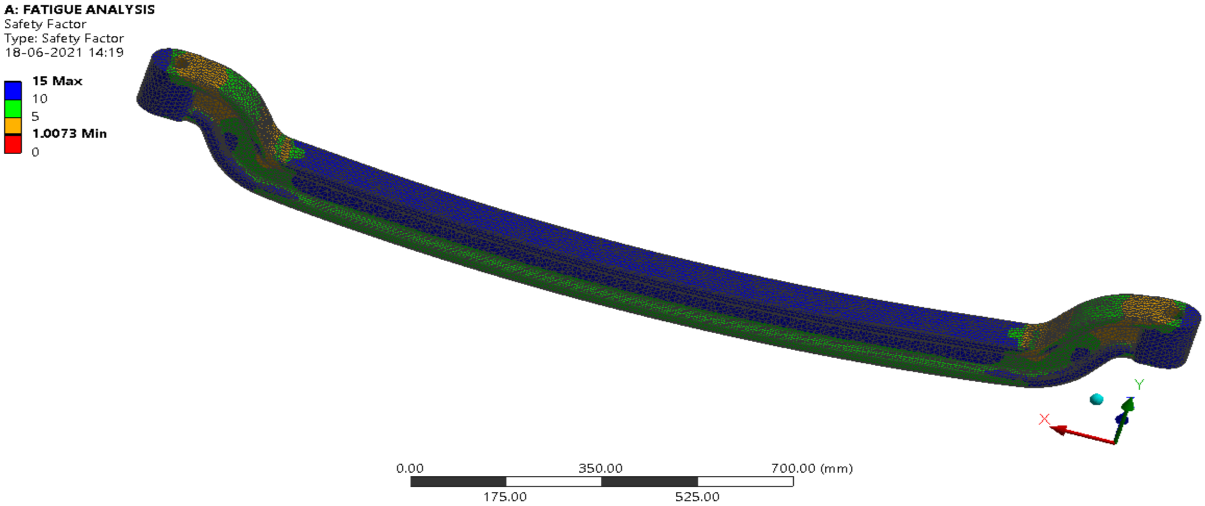

Figure-40 Safety factor for Gray Cast Iron. The maximum value is 15 Max

FATIGUE DAMAGE FOR GRAY CAST IRON

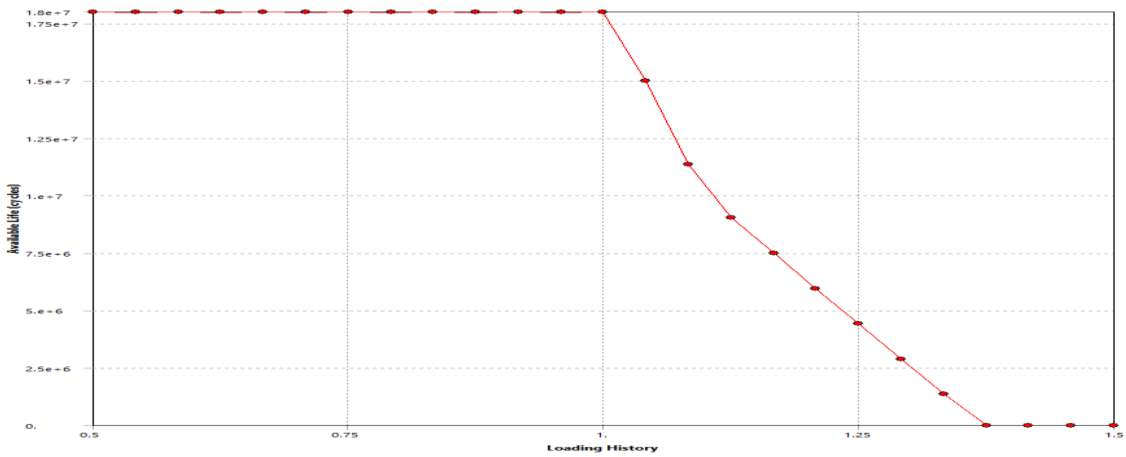

Figure-41 Fatigue sensitivity of Gray Cast Iron.

TOTAL DEFORMATION FOR NICKEL STEEL

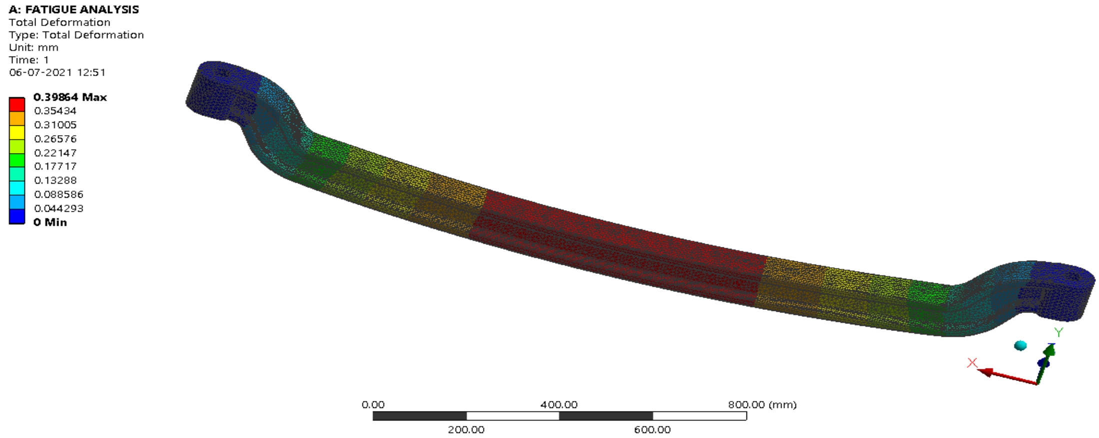

Figure-42 Total deformation for Nickel steel. The Maximum value is 0.39864 Max

EQUIVALENT ELASTIC STRAIN OF NICKEL STEEL

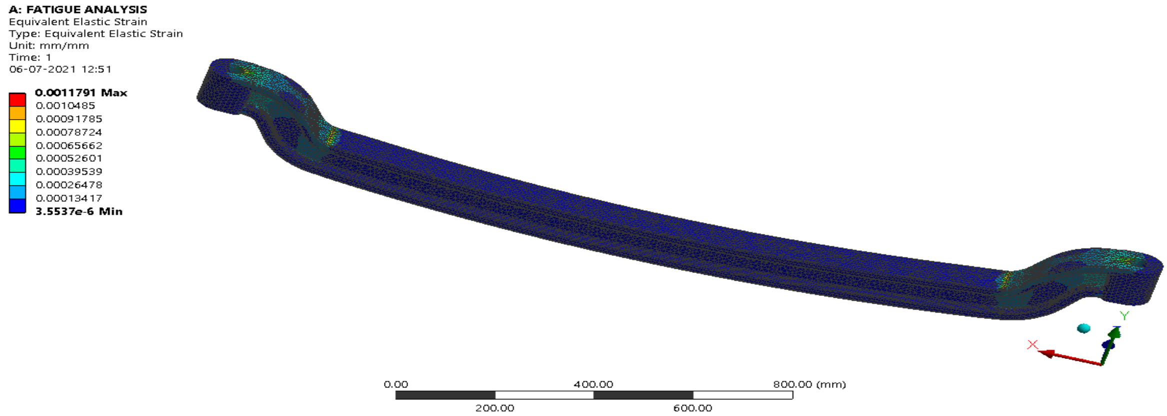

Figure-43 Equivalent Elastic Strain for Nickel steel. The Maximum value is 0.0011791 Max.

EQUIVALENT STRESS OF NICKEL STEEL

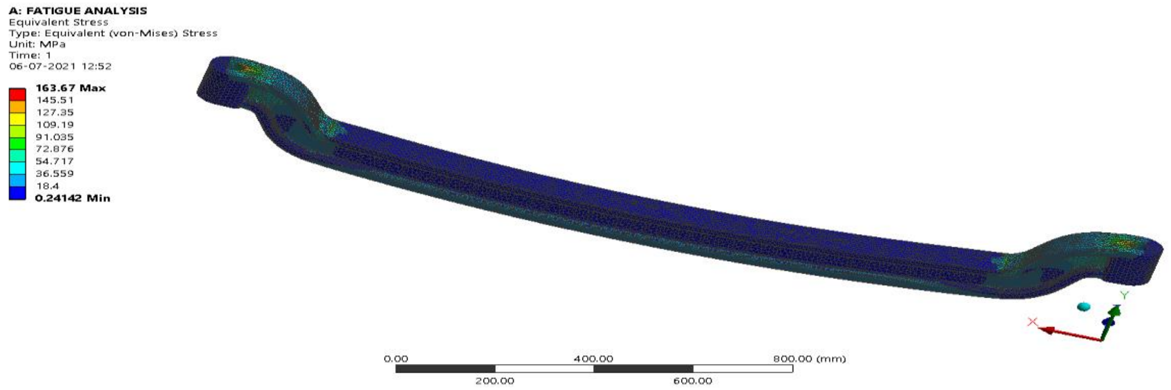

Figure-44 Equivalent stress for Nickel Steel. The Maximum value is 163.67 Max.

FATIGUE LIFE FOR NICKEL STEEL

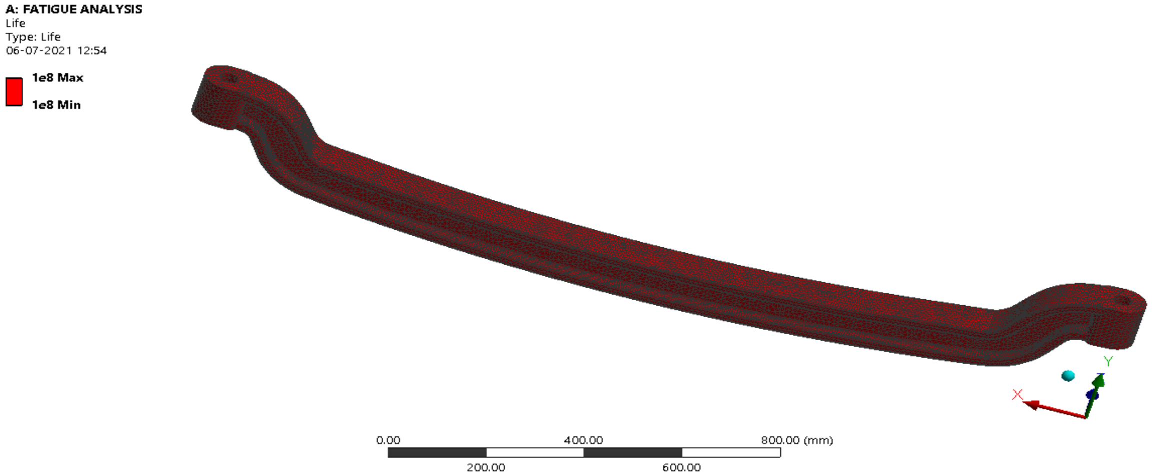

Figure-45 Fatigue life for Nickel Steel. The Maximum value is 1e8 Max.

FATIGUE DAMAGE OF NICKEL STEEL

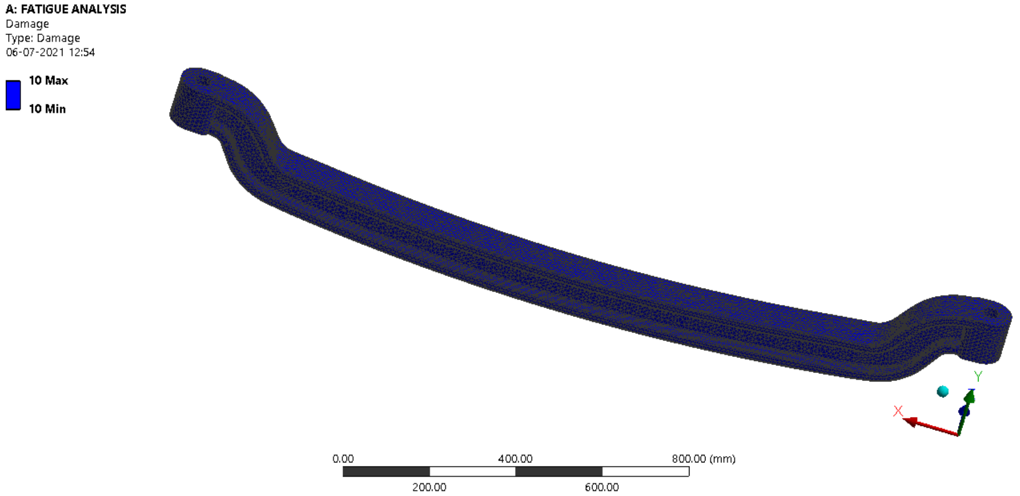

Figure-46 Fatigue damage for Nickel Steel. The Maximum value is 10Max.

SAFETY FACTOR FOR NICKEL STEEL

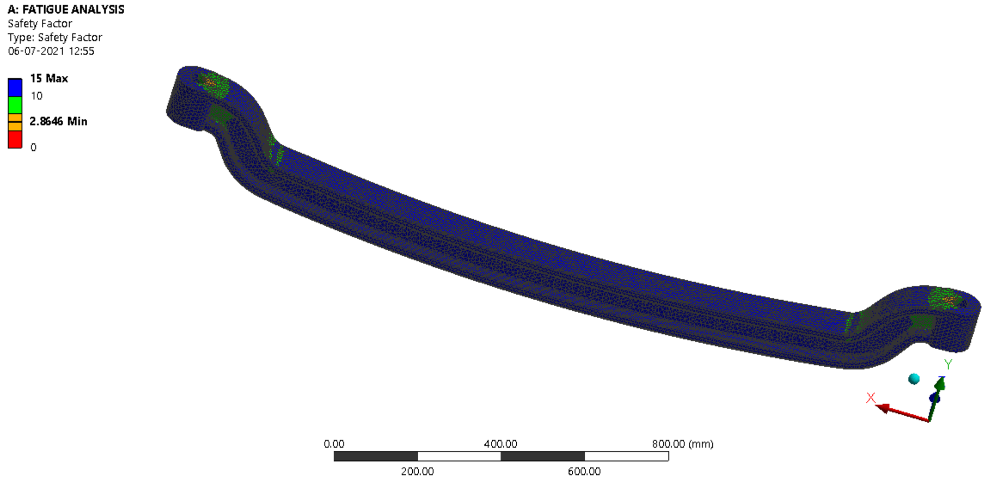

Figure-47 Safety factor for Nickel Steel. The Maximum value is 15Max.

FATIGUE SENSITIVITY FOR NICKEL STEEL

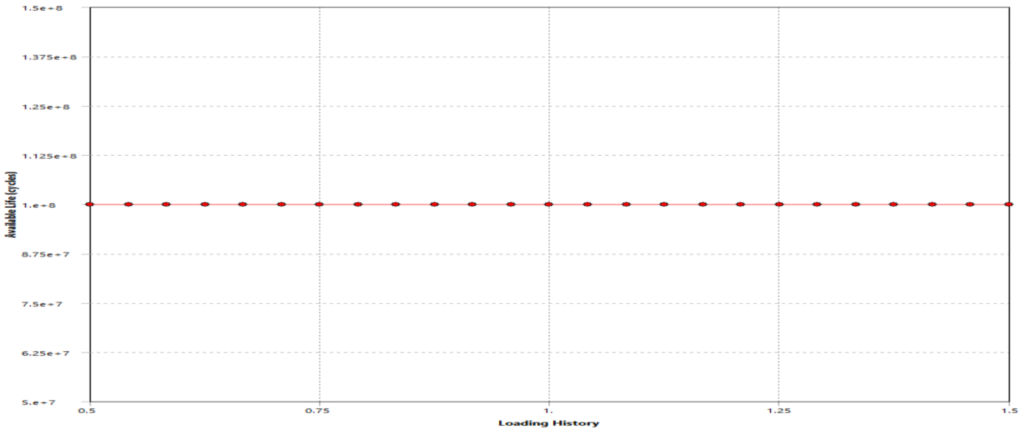

Figure-48 Fatigue Sensitivity for Nickel Steel.

COMPRASION TABLE

Table-B Total deformations for all Materials.

S.NO	MATERIAL	TOTAL DEFORMATION (mm)
1.	MILD STEEL	0.37105
2.	STRUCTURAL STEEL	0.749010
3.	AlSiC	0.36165
4.	GRAY CAST IRON	0.67332
5.	9% Nickel steel	0.39864

Table-C Elastic Strain for all Materials.

SL.NO	MATERIAL	ELASTIC STRAIN(mm/mm)
1.	MILD STEEL	0.0010975
2.	STRUCTURAL STEEL	0.0022357
3.	AlSiC	0.0010819
4.	GRAY CAST IRON	0.002037
5.	9% NICKEL STEEL	0.0011791

Table-D Equivalent Stress for all Materials.

SL.NO	MATERIAL	EQUIVALENT STRESS(MPa)
1.	MILD STEEL	163.74
2.	STRUCTURAL STEEL	163.69
3.	AlSiC	163.92
4.	GRAY CAST IRON	163.8
5.	9% Nickel steel	163.67

Table-E Fatigue Life for all Materials.

SL.NO	MATERIAL	FATIGUE LIFE(Cycles)
1.	MILD STEEL	1E8
2.	STRUCTURAL STEEL	1E6
3.	AlSiC	3.0598E5
4.	GRAY CAST IRON	1.8E7
5.	9% Nickel steel	1E8

Table-F Fatigue Damage for all Materials.

SL.NO	MATERIAL	FATIGUE DAMAGE
1.	MILD STEEL	10
2.	STRUCTURAL STEEL	18698
3.	AlSiC	3268.2
4.	GRAY CAST IRON	55.556
5.	9% Nickel steel	10

Table-G Safety Factor for all Materials.

SL.NO	MATERIAL	SAFETY FACTOR
1.	MILD STEEL	1.6184
2.	STAINLESS STEEL	0.52667
3.	AlSiC	0.50356
4.	GRAY CAST IRON	1.0073
5.	9% Nickel steel	2.8646

GRAPHS

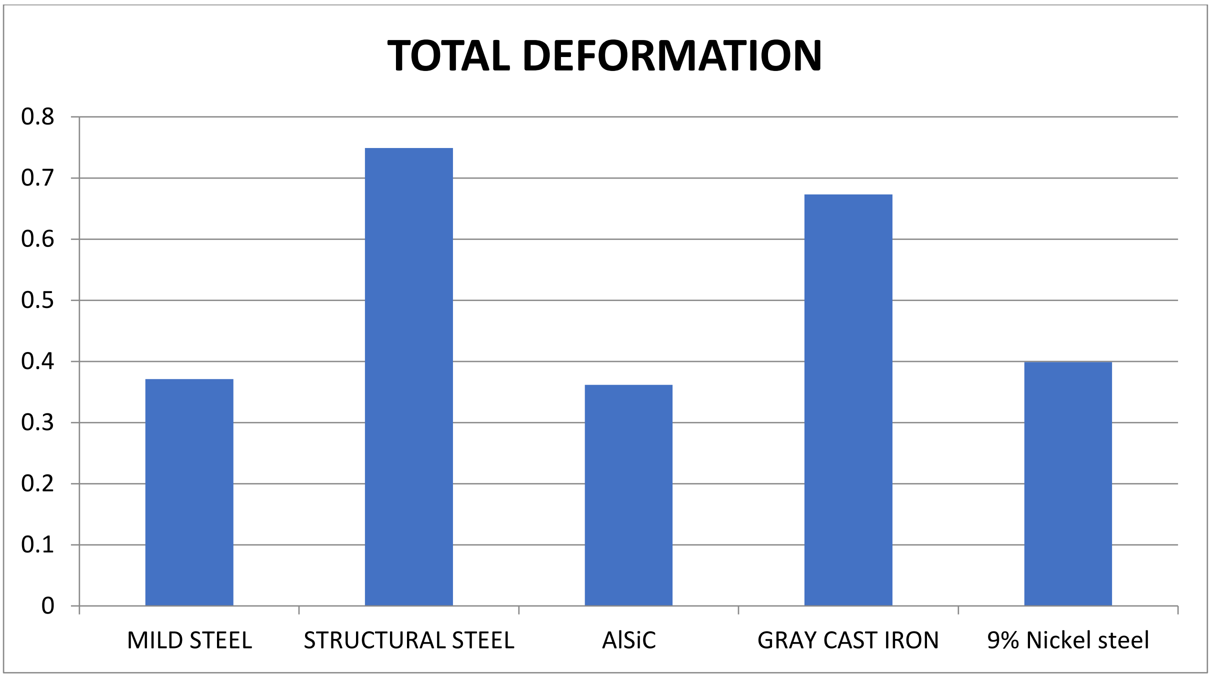

Fig-I TOTAL DEFORMATION FOR ALL MATERIAL

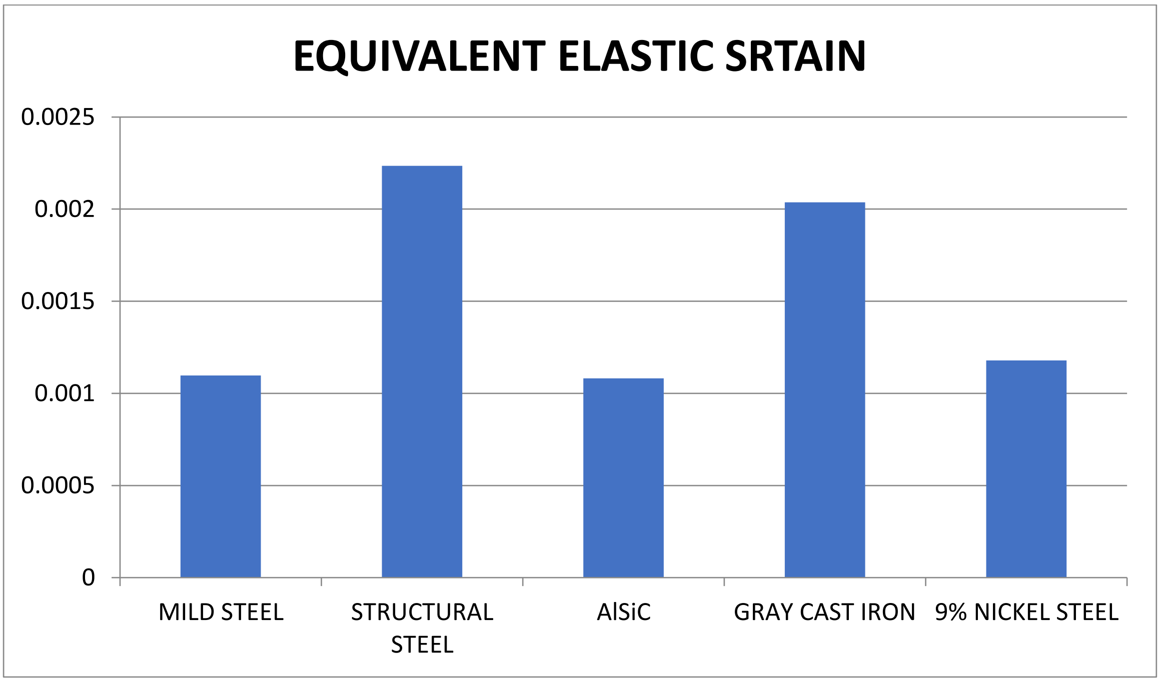

Fig-II EQUIVALENT ELASTIC STRAIN FOR ALL MATERIAL

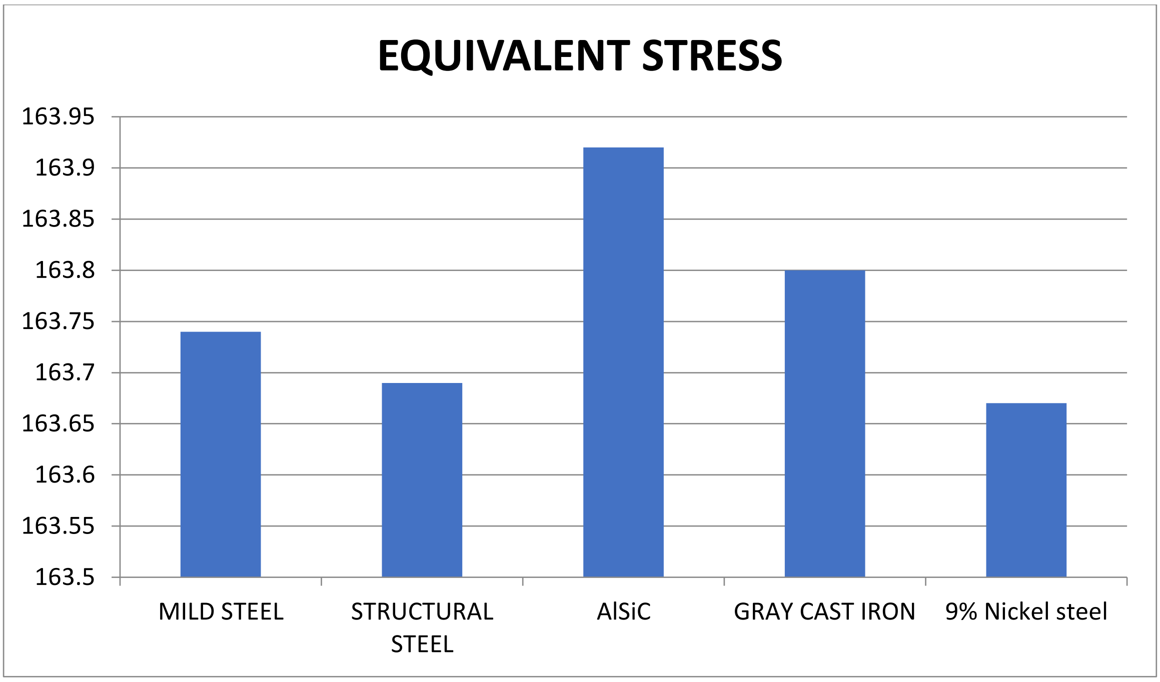

Fig-III EQUIVALENT STRESS FOR ALL MATERIALS

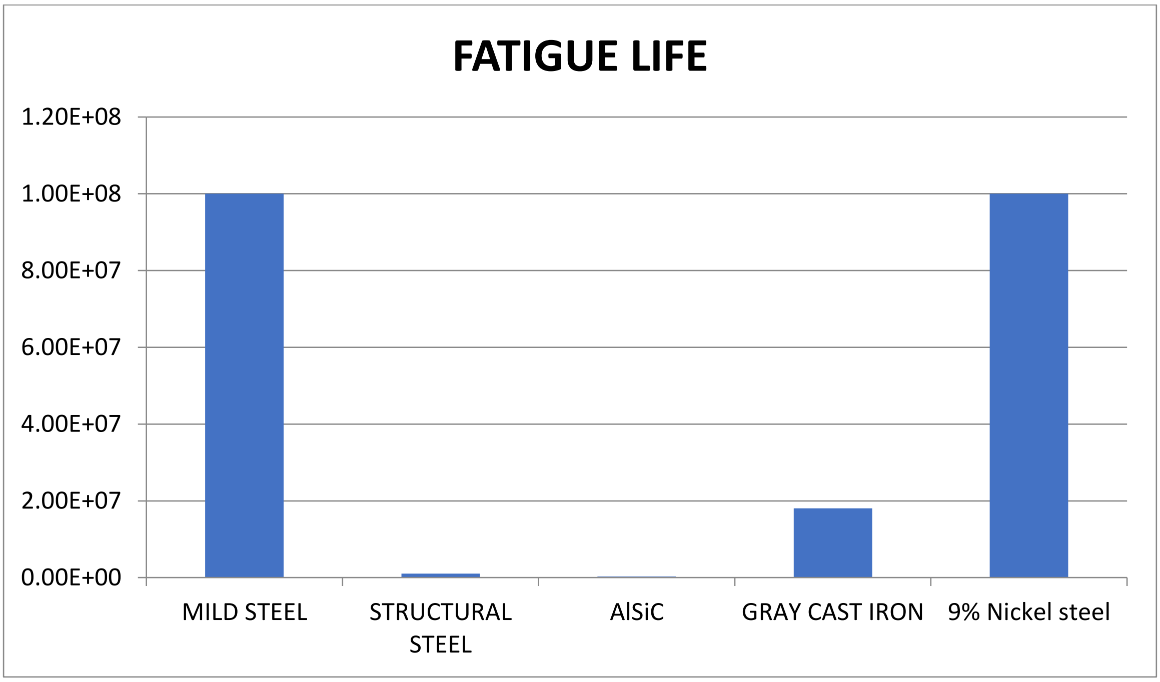

Fig-IV FATIGUE LIFE FOR ALL MATERIALS

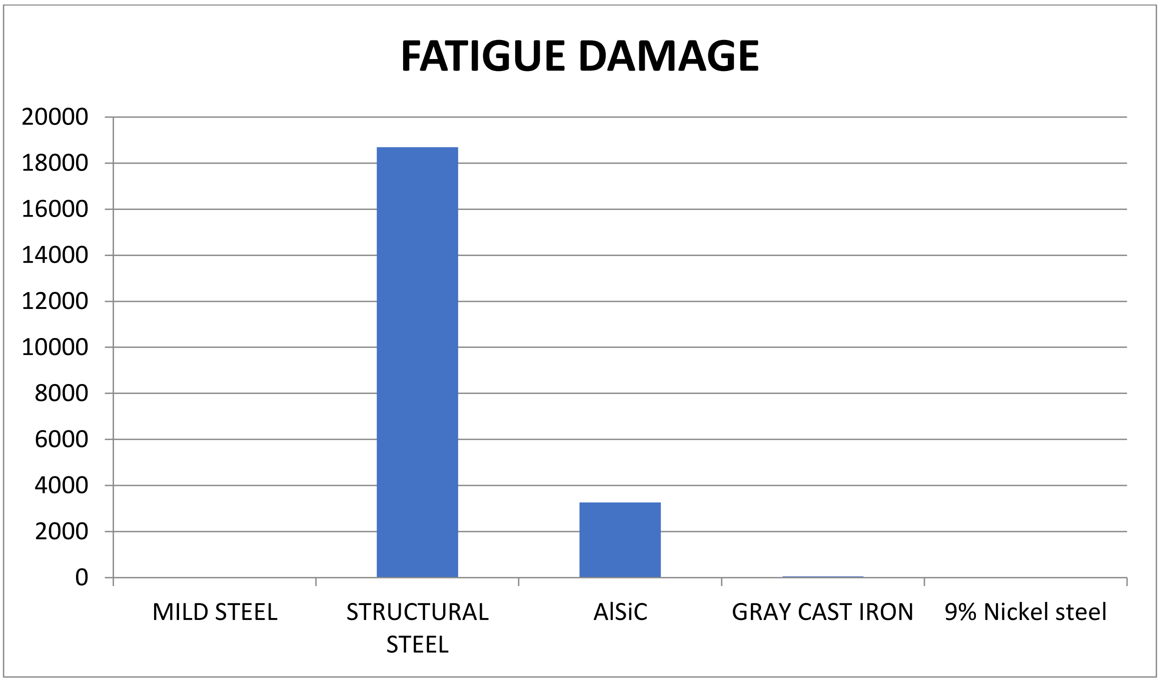

Fig-V FATIGUE DAMAGE FOR ALL MATERIALS

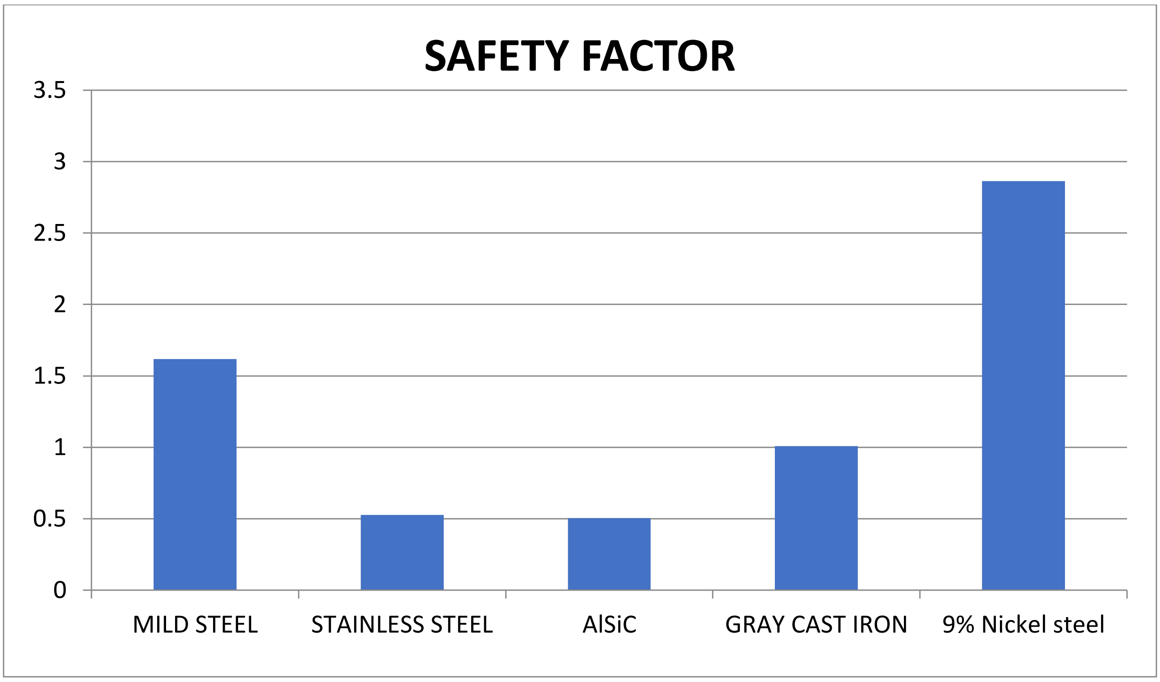

Fig-VI SAFETY FACTOR FOR ALL MATERIAL

CHAPTER-7

CONCLUSION

- The front axle of the truck is modeled and analyzed for its total deformation, elastic strain, equivalent strain and for the fatigue life, damage, safety factor and fatigue sensitivity.
- The analysis is done by using 3 different materials. Here Mild steel is the most commonly used material for the manufacturing of front axle.
- Analysis is done and the results obtained for 3 material Stainless steel, AlSiC and Gray Cast Iron are compared with Mild steel.
- After the comparison we got
    - For the Total Deformation point of view AlSiC is better.
    - For the Elastic strain point of view AlSiC is better.
    - For the Equivalent point of view Stress 9% Nickel steel is better.
    - For the Fatigue life point of view 9% Nickel steel is better.
    - For the Damage point of view 9% Nickel steel is better.
    - For the Safety factor point of view 9% Nickel steel is better.

On over all comparison other than mild steel 9% Nickel steel is best for the manufacturing of front axle in the Fatigue point of view.

REFERENCES:

- [1] Kulkarni Vipul V. *1, Prof. Ghodake Arvind P. *2 "DESIGN, ANALYSIS AND MANUFACTURING OF AXLE SHAFT BY ALTERNATE MATERIAL".
- [2] Pravin R.Ahire1, Prof.K. H. Munde2 "DESIGN AND ANALYSIS OF FRONT AXLE FOR HEAVY COMMERCIAL VEHICLE"
- [3] Ketan Vijay Dhande1, Prashant Ulhe2 DESIGN AND ANALYSIS OF FRONT AXLE OF HEAVY COMMERCIAL VEHICLE.
- [4] Mr. Khot S.S Padmashri Dr. Vithalrao Vikhe Patil College of Engineering Ravindra R Navthar Dr. Vithalrao Vikhe Patil College of Eng " DESIGN AND OPTIMIZATION OF FRONT AXLE OF HEAVY TRUCK".
- [5]. Pravin R. Ahire1, Prof. K. H. Munde2, "Design and analysis of front axle for heavy commercial vehicle". International Journal Of Engineering And Computer Science ISSN: 2319-7242 Volume 5 Issues 7 July 2016, Page No. 17333-17337
- [6]. Hemant L. Aghav, "Life Prediction for Vertical and Braking loading of Front Axle of Heavy-Duty Truck". International Journal of Engineering Science and Computing, July 2016 8207 http://ijesc.org/, DOI 10.4010/2016.1915, ISSN 2321 3361 © 2016 IJESC, Volume 6 Issue No. 7
- [7]. Kiran Maddewad1, Trupti Jadhav2, Ajinkya Bhosale3, "Optimization of Front Axle for Heavy Commercial Vehicle by Analytical and FEA Method". International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056 Volume: 04 Issue: 03 | Mar -2017 www.irjet.net p-ISSN: 2395-0072
- [8] Vinay Dilliwar, 2Sankalp Verma, "Modification of Manufacturing Process of Kingpin for Steering Assembly of Heavy Motor Vehicles and its Analysis". 2020 JETIR July 2020, Volume 7, Issue 7 www.jetir.org (ISSN-2349-5162)
- [9] J.B. Marcomini1*, C.A.R.P. Baptista2, J.P. Pascon3, "Failure Analysis of a Hot Forged SAE 4140 Steel Kingpin". International Journal of Engineering Research & Science (IJOER) ISSN: [2395-6992] [Vol-2, Issue-6 June- 2016]

Programme Outcomes (POs):

Engineering Graduates will be able to:

1. **Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals and an engineering specialization to the solution of complex engineering problems.
2. **Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3. **Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
5. **Modern tool usage:** Create, select, and apply appropriate techniques, resources and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one’s own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
12. **Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

Program Specific Outcomes (PSOs)

Engineering Graduates will be able to:

PSO-1: design and analyze various thermal systems used in power generation and human comfort.

PSO-2: design, analyze and develop products by adopting best manufacturing practices.

PSO-3: use various mechanical engineering software tools for design and analysis of various engineering components.

Gudlavalleru Engineering College

(An Autonomous Institute with Permanent Affiliation to JNTUK, Kakinada)

Seshadri Rao Knowledge Village, GUDLAVALLERU

Department of Mechanical Engineering

Class: IV B. Tech - II Sem **A.Y: 2020-2021**

Name of the Subject: Main Project **Batch: 2017-2021 (R17)**

Each CO is mapped with the POs and PSOs in three levels, ‘3’ indicates high, ‘2’ indicates moderate and ‘1’ indicates low level

MAIN PROJECT																
COURSE OUTCOMES	**PROGRAM OUTCOMES AND PROGRAM SPECIFIC OUTCOMES**															
	PO1	**PO2**	**PO3**	**PO4**	**PO5**	**PO6**	**PO7**	**PO8**	**PO9**	**PO10**	**PO11**	**PO12**		**PSO1**	**PSO2**	**PSO3**
CO1: Identify methods and materials to carry out experiments/develop code.																
CO2: Recognize the procedure with a concern for society, environment and ethics.																
CO3: Analyze and discuss the results to draw valid conclusions.																
CO4: Prepare reports and present effectively in oral and written form.																
CO5: Inculcate habit of lifelong learning independently to stay with the changes in technology.																
MAIN PROJECT																

Name of the Faculty: **Signature of the Faculty:**