DESIGN AND STRUCTURAL ANALYSIS OF REAR AXLE CASING IN AUTOMOBILE

A PROJECT REPORT

Submitted by

M. Sathish kumar - 71206402009

In partial fulfillment for the award of degree
Of

MASTER OF ENGINEERING
In
CAD/CAM

DEPARTMENT OF MECHANICAL ENGINEERING

KUMARAGURU COLLEGE OF TECHNOLOGY

COIMBATORE – 641 006

ANNA UNIVERSITY:: CHENNAI 600 025

JUNE - 2008

ANNA UNIVERSITY :: CHENNAI 600 025

BONAFIDE CERTIFICATE

Certified that this project report entitled "**Design and Structural Analysis of Rear Axle Casing in Automobile**" is the bonafide work of

Mr. M. Sathish kumar **- Register No. 71206402009**

Who carried out the project work under my supervision.

Signature of the HOD **Signature of the Supervisor**

6/6/08

Internal Examiner **External Examiner**

Department of Mechanical Engineering

KUMARAGURU COLLEGE OF TECHNOLOGY

COIMBATORE – 641 006

KUMARAGURU COLLEGE OF TECHNOLOGY

COIMBATORE , TAMILNADU

DEPARTMENT OF MECHANICAL ENGINEERING & KCT-TIFAC CORE

ADVANCES IN MECHANICAL SCIENCES

CERTIFICATE

This is to certify that Mr/Ms/Mrs M . SATHISHKUMAR

has participated and presented a paper titled Structural analysis of rear axel casing of HMT tractor in the 2nd National Conference on "ADVANCES IN MECHANICAL SCIENCES" during 27- 28, March 2008.

Dr.P.PALANISAMY CO-ORDINATOR	**Dr.C.SIVANANDAN** CONVENOR & DEAN	**Dr.JOSEPH V.THANIKAL** PRINCIPAL

AIMMS '08

Sri SAI RAM ENGINEERING COLLEGE

[NBA ACCREDITED & ISO 9001 : 2000 CERTIFIED INSTITUTION]

SAI LEO NAGAR, WEST TAMBARAM, CHENNAI - 600 044.

DEPARTMENT OF MECHANICAL ENGINEERING

National Conference on Advancements In Manufacturing and Management Systems

Certificate

This is to certify that Mr./Ms. M. SATHISH KUMAR from KUMARAGURU COLLEGE OF TECHNOLOGY, COIMBATORE has presented a paper titled DESIGN AND STRUCTURAL ANAYSIS OF DIFFERENTIAL HOUSING IN A TRACTOR in AIMMS '08, a National Conference, conducted on 17th April 2008.

ORGANIZING SECRETARY	HOD	PRINCIPAL	DEAN	CHAIRMAN

ABSTRACT

In order to gain competitive advantage in global market it is necessary to produce a quality product with good safety. It is important to industries to introduce into the market as quickly as possible the new product or new models of existing products. To attain the above said designer should design each and every component in an automobile with very safely and the product should have long life and also be economical.

In this project work rear axle casing which is used in HMT tractor considered for design improvements. Currently the problem occurs in that rear axle casing is it deforms before the warranty period. This work is intended to increase the life time of the component by resolving the failure by making some design modification, considerably it reduces total service cost which includes part replacement, labor.

PRO-E WILDFIRE 3.0 is used for modeling the component. Static and modal analyses are carried out by using ANSYS 10.0 software is used to rectify the problem.

This project work was done in the following sequence.

- Initially data related to the rear axle casing were collected and tabulated as required

- Modeling of the rear axle casing was carried out using parametric modeling software PRO-E Wildfire 3.0. Analysis were carried out for the failure component (rear axle casing), and then we find out main cause of the failure. This was done in two stages. The first stage was the static analysis, and the second is the modal analyses are carried for the same component.

- After found out the problem. Redesigns were carried out by modifying the existing design of the rear axle casing according to theoretical knowledge. Then this design had been taken into reanalysis by adopting the same

- procedure and also used the same software to resolve the problem which occurs in component.

- Moreover, by comparing the both analysis results i.e. (actual and redesign) which ensure whether the design is to be safe or not.

From this design improvements life time of the rear axle casing is increased.

ஆய்வு சுருக்கம்

இன்றைய உலக சந்தையில் தரமான மற்றும் பாதுகாப்பான பொருட்களை தயாரிப்பதன் மூலம் நல்ல நிலையை அடைய முடியும். தற்காலக்கட்டத்தில் வாகனங்கள் வெவ்வேறு பயனுக்கு பயன்படுத்தப்பட்டு வருகின்றன. இதன் பாகங்களில் ஏற்படும் பழுதால் விபத்து நேரிடுவதை காண்கின்றோம். மேற்கூறிய இவைகளை நிவர்த்தி செய்ய வடிவமைப்பாளர் இயந்திர பாகங்களை மிகவும் பாதுகாப்பானதாகவும், நீண்ட ஆயுளை கொண்டதாகவும் மற்றும் பொருளாதார முறைக்கு ஏற்றதாகவும் வடிவமைக்க வேண்டும்.

இந்த ஆய்வில், ஹச். எம். டி டிராக்டரில் உபயோகபடுத்தபடும் பின் தண்டின் குப்பியின் வடிவமைப்பின் மேம்படுத்துவதை கருத்தில் கொண்டுள்ளது. தற்பொழுது இந்த பின் தண்டு குப்பி அதன் உத்திரவாத காலத்திற்குள்ளேயே பழுதடைந்துவிடுகிறது. இந்த ஆய்வின் நோக்கம் யாதெனில் அப்பாகத்தின் வடிவமைப்பு மாற்றுவதன் மூலம் அதில் உள்ள குறைபாடுகளை நீக்கப்படுகிறது. இதன் மூலம் அப்பாகத்தின் ஆயுட்காலம் அதிகரிக்கப்படுகிறது மற்றும் பாகத்தின் பராமரிப்பு செலவு குறைக்கபடுகிறது.

பின் தண்டு குப்பியை கணிப்பொறி மூலம் வடிவமைக்க புரோ/இஞ்னியர் வைல்டு பையர் 3.0 என்ற மென்பொருள் பயன்படுத்தப்படுகிறது. இதனை பகுப்பாய்விற்கு உட்படுத்த ஆன்சிஸ் 10.0 என்ற மென்பொருள் பயன்படுத்தப்படுகிறது.

முதற்கட்டமாக பாதிக்கப்பட்ட பின் தண்டு குப்பி, மென்பொருளை தொகுப்பாய்விற்கு உட்படுத்தி எப்பகுதியில் என்பதும் எந்த இடத்தில் பாதிப்பு ஏற்பட்டது எனக் கண்டறியப்பட்டது. இதன் தொடர்ச்சியாக பாதிக்கப்பட்ட அளவுகுகளை விஞ்ஞான தத்துவ விளக்கங்களை கொண்டு அதன் அளவுகளில் மாற்றம் செய்யப்பட்டு மீண்டும் பகுப்பாய்விற்கு உட்படுத்தப்பட்டு அதன் குறைகள் நிவர்த்தி செய்யப்பட்டது. இதனால் மேற்கூறியபடி குறைபாடுகள் நீக்கப்பட்டு, அதனின் ஆயுட்காலம் அதிகரிக்கப்பட்டு அதனுடைய பராமரிப்பு செலவு குறைக்கபடுகிறது.

ACKNOWLEDGEMENT

I take this opportunity to publicly express my gratitude to all those people who have supported my work during this project.

First and foremost, My heartfelt gratitude to my guide, **Dr. V. VeL Murugan**, Professor, Department of Mechanical Engineering, Kumaraguru College of Technology, for contributing to my project work in numerous ways through his constant source of motivation, wise counsel and encouragement which has been an inspiration throughout the work for the successful completion of this project work.

I am grateful to **Dr. C. Sivanandan**, Dean and Head of the Department, Department of Mechanical Engineering for his valuable suggestions and timely help towards my project. I wish to express my deep sense of gratitude to **Dr. Joseph V. Thanikal**, Principal, Kumaraguru College of Technology for patronizing me, besides providing all assistance.

Words are inadequate in offering my thanks to **Dr. P. Palanisamy**, our project coordinator and the **faculty members** of the Department of Mechanical Engineering, Kumaraguru College of Technology for their encouragement and cooperation in carrying out my project work. My sincere thanks also to all my **friends** for their help in all the matters during my project work.

I would be failing in my duty if I do not express my gratitude of **all those authors** who had contributed abundant literature through various research publications.

Finally, I would like to thank my dearest ones, my father **Mr. G. Mohanraj**, my mother **Mrs. J. Sulochana,** for their support and encouragement throughout the different stages of my project work.

Last but not least, I express my heartfelt thanks to the Almighty God for the blessings. Without His permission and blessings it would not have been possible to complete this project work.

CONTENTS

Title		Page No.
Certificate		iii
Abstract		v
Acknowledgement		viii
List of Tables		xiii
List of Figures		xiv
List of Nomenclature		xv
CHAPTER 1	**INTRODUCTION**	01
1.1	Introduction	01
1.2	Rear axle housing	01
1.2.1	Banjo type	02
1.2.2	Split type	02
1.3	Problem Chosen and Its Importance	03
1.4	Methodology	05
1.5	Detailed view	06
CHAPTER 2	**LITERATURE REVIEW**	07
CHAPTER 3	**FINITE ELEMENT ANALYSIS**	11
3.1	A Brief History	11
3.2	Finite Element Method Concept	12
3.2.1	Finite Element Analysis- General Procedure	13
3.2.2	General Structure of Fem Procedure	14
3.2.3	Structural Analysis	15
3.3	The Reason for Using the FEA Method in this Study	19
3.4	Generation of the Geometry	19
3.5	Material Properties	19
3.6	Load Conversion	20
3.7	Design Constraints	20

3.8 Element Details 21
3.9 Stress Concentration 22
3.10 Meshing 24

CHAPTER 4 ANALYSIS RESULTS OF THE FAILED REAR AXLE CASING 25
4.1 Actual Model 25
4.2 Structural Analysis 25
4.2.1 Static Analysis 25
4.2.1.1 Displacement Analysis 26
4.2.1.2 Stress Analysis 26
4.3 Time Stepping Analysis 26
4.3.1 Displacement Result 28
4.3.2 Stress Result 28
4.4 Modal Analysis 31
4.4.1 Free-Free Analysis 31
4.4.2 One End Fixed 32

CHAPTER 5 ANALYSIS RESULTS OF REDESIGNED REAR AXLE CASING 34
5.1 Redesigned Component 34
5.2 Structural Analysis 34
5.2.1 Static Analysis 34
5.2.1.1 Displacement Analysis 35
5.2.1.2 Stress Analysis 35
5.3 Time Stepping Analysis 35
5.3.1 Displacement Result 38
5.3.2 Stress Result 38
5.4 Modal Analysis 40
5.4.1 Free-Free 40
5.4.2 One End Fixed 41

CHAPTER 6 COMPARISION OF RESULTS AFTER ANALYSIS 43

6.1 Comparison of Static Analysis Results For (Actual and Redesign) 43

6.1.1 Displacement 43

6.1.2 Stress 43

6.2 Time Stepping Analysis 44

6.2.1 Displacement Results 44

6.2.2 Stress Results 44

6.3 Comparison of Modal Analysis Results For (Actual And Redesigned component) 46

6.3.1 Free-Free Results 47

6.3.2 Fixed End Results 47

CHAPTER 7 CONCLUSIONS 49

REFERENCES 50

LIST OF TABLES

Table No.	Details	Page No.
3.1	Material properties of rear axle casing	20
3.2	Design constraint for analysis	21
4.1	Time stepping result for displacement of failed rear axle Casing	28
4.2	Time stepping result for stress of failed rear axle casing	31
4.3	Modal free-free analysis results for failed component	32
4.4	Modal one end fixed analysis results for failed Component	32
5.1	Time stepping displacement results for redesigned rear Axle casing	38
5.2	Time stepping stress results for redesigned rear axle casing	40
5.3	Modal free-free analysis results for redesigned component	41
5.4	Modal one end fixed analysis results for redesigned Component	41
6.1	Comparison displacement results for both failed and redesigned Rear axle casing	43
6.2	Comparison stress results for both failed and redesigned rear Axle casing	44
6.3	Comparison of time stepping displacement results for both failed And redesigned rear axle casing	45
6.4	Comparison of time stepping stress results for both failed and Redesigned rear axle casing	46
6.5	Comparison of modal free-free results for both failed and Redesigned rear axle casing	47
6.6	Comparison of modal one end fixed results for both failed and Redesigned rear axle casing	48

LIST OF FIGURES

Figure No.	Details	Page No.
1.1	Banjo type RAC	3
1.2	Split type RAC	3
1.3	Rear axle casing	5
3.1	Analysis Element geometry	21
4.1	Failed rear axle casing 3D model	27
4.2	Displacement analysis for failed rear axle casing	27
4.3	Stress analysis for failed rear axle casing	29
4.4	Time stepping displacement results for failed rear axle Casing	29
4.5	Time stepping stress results for failed rear axle casing	30
4.7	Modal free-free analysis results for failed component	33
5.1	Redesigned rear axle casing 3D model	36
5.2	Displacement analysis for redesigned rear axle casing	36
5.3	Stress analysis for redesigned rear axle casing	37
5.4	Time stepping displacement analysis result for redesigned Rear axle casing	37
5.5	Time stepping displacement analysis result for redesigned Rear axle casing	39
5.6	Time stepping stress analysis result for redesigned rear Axle casing	39
5.7	Modal free-free analysis results for redesigned component	42
5.8	Modal one end fixed analysis results for redesigned component	42
6.1	Displacement comparison graph for both models	45
6.2	Stress comparison graph for both models	46

LIST OF NOMENCLATURES

NOMENCLATURE	DEFINITION
FEA	Finite Element Analysis
RAC	Rear Axle Casing
HZ	Hertz
K	Stress concentration factor
[K]	Stiffness matrix
F	Force vector
q	Nodal displacement vector

Chapter 1

Introduction

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Today, Automobiles are used everywhere in all over the world. Designer should design each and every component in the automobile with very safely and the product should have long life, and also be economical. Normally automobile carries heavy load in the real time application, some failure occurs due to the carrying of heavy load or external environment, while stationary or dynamically. By using the software, the failures can be avoided by redesigning and analyzing the components effectively.

Here, the rear axle casing is taken as automobile component, which is used in tractors. Axle housing is one of the significant components that lead to great performance of the vehicle. It may be available in one-piece or has spilt type construction that is also known as banjo type construction. Solid axle housing was used by rear suspension in earlier models. They were positioned with two lower control arms and pan hard rod having shock absorbers inside the coil springs. Solid axle housing was positioned in later models by two lower control arms, two upper control arms and a pan hard rod.

1.2 REAR AXLE HOUSING

A British term indicating a tubular housing, which encloses the differential and half-shafts along with their bearings. The US term is "axle housing." Rear axle differential: A differential situated in the final drive of the transmission assembly in a conventional rear-wheel drive vehicle. Axle housing is one of the significant components that lead to great performance of the vehicle. It may be available in one-piece or has spilt type construction that is also known as banjo type construction. Axle housings are usually comprised of double lip seals and over size bearings. This results in more capacity and enhanced surface area contact over axle. Both the front and rear openings have center housing. Differential carrier closes the front opening. On the other hand, spherical cover plate is used for closing the rear. Solid axle housing was used by rear suspension in earlier models. They were positioned with two lower control arms and pan hard rod having shock absorbers inside the coil springs. Solid axle housing was positioned in later models by two lower control arms, two upper control arms and a pan hard rod.

In order to carry a heavy load of the vehicle, such as trucks and tractors, axle housing is manufactured using heavy cast unit. Light duty trucks and tractors have axle housing comprising of a great combination of steel tube and cast. The cast or machined units are used in center or differential and final drive case. On the other hand, welded or extruded steel tubing is used in axle housings. Items that may be welded riveted or cast in axle housings are brake-backing plates, mounting flanges, spring mounting plates and accessory unit. Rear axle housing, the component which connects the drive shaft to the axle shafts. Normal types of rear axle housing used are given below

1. Banjo type
2. Split type

1.2.1 Banjo Type:

A rear axle housing from which the differential unit may be removed while the housing remains in place on the vehicle. The housing is solid from side to side it shown in fig 1.1

1.2.2 Split Type:

Rear axle housing made up of several pieces and bolted together. The housing must be split apart to remove the differential. The split type shown in fig 1.2

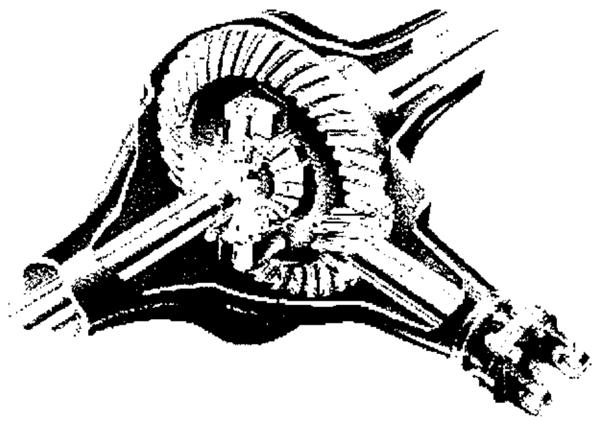

Fig 1.1 Banjo type

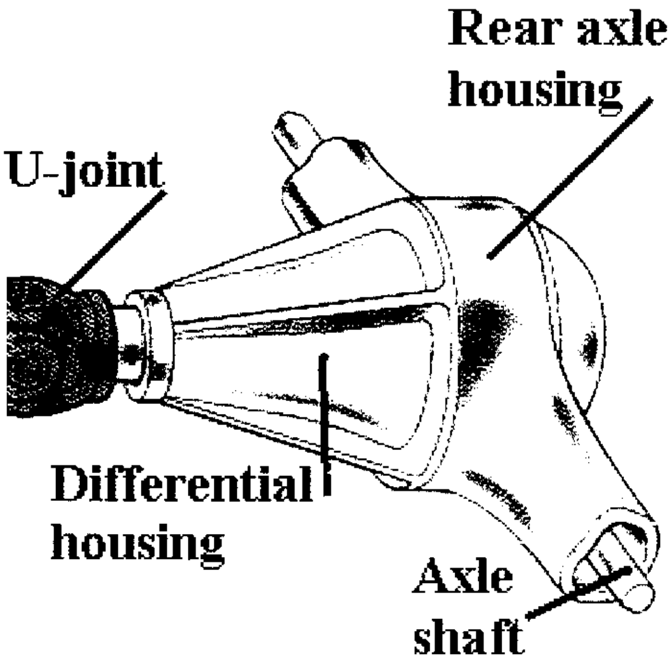

Fig 1.2 Split type

1.3 PROBLEM CHOSEN AND ITS IMPORTANCE

In this competitive world today, where nothing but change is constant, Automobiles are used everywhere in all over the world. it is very much necessary to gain and maintain a competitive advantage. For the same, designer should have to introduce new product or new models of existing products as quickly as possible. Designer should design each and every component in the automobile with very safely and the product should have long life, and also be economical. Normally automobile carries heavy load in the real time application. some failure occurs due to the carrying of heavy load or external environment, while stationary or dynamically. By using the software, the failures can be avoided by designing and analyzing the components effectively.

An improvement in design of the components in an automobile industry has become essential in the current scenario. In this project work the differential housing of tractor is considered for design and improvements. Differential housing is a critical part in an automobile structure which is used in HMT tractors shown fig 1.3. This component is a very complex element to design. Predicting strength of these parts is of great practical interest, since improper design may cause structural problems of the differential housing. Currently the industries facing critical problem in differential housing, it fails before the completion of the warranty period In order to carry a heavy load of the vehicle, such as trucks and tractors, axle housing is manufactured using heavy cast unit. Light duty trucks and tractors have axle housing comprising of a great combination of steel tube and cast. The cast or machined units are used in center or differential and final drive case. On the other hand, welded or extruded steel tubing is used in axle housings. Items that may be welded riveted or cast in axle housings are brake-backing plates, mounting flanges, spring mounting plates and accessory unit.

The main objective of this project is to increase the component life, by analyzing the component in order to avoid the failure. Here, the rear axle casing is taken as automobile component, which is used in HMT tractor. In the real time application, load acts continuously while the vehicle is in stationary or dynamic position; it causes Failure.

This project focus to find out the main reason to cause the failure of the rear axle casing whiles it working in different condition. After the problem identification to avoid these failures by making some design improvements and modification and analyzing these for static load.

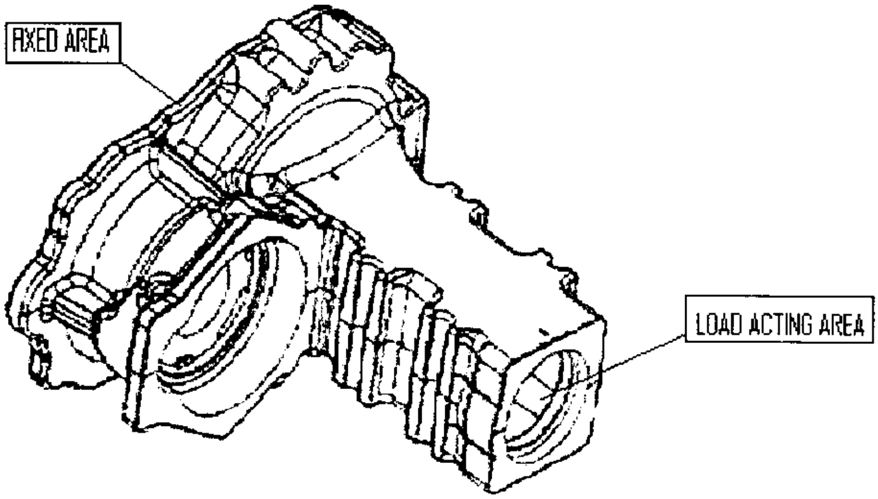

Fig 1.3 Rear axle housing

1.4 METHODOLOGY

This project work was done in the following sequence.

- Initially data related to the rear axle casing were collected and tabulated as required

- Modeling of the rear axle casing was carried out using parametric modeling software PRO-E Wildfire 3.0. Analysis were carried out for the failure component (rear axle casing), and then we find out main cause of the failure. This was done in two stages. The first stage was the static analysis, and the second is the modal analysis are carried for the same component. 1

- After found out the problem which was the main cause of the failure. Redesigns were carried out by modifying the existing design of the rear axle casing according to theoretical knowledge. Then this design had been taken into reanalysis by adopting the same procedure and also used the same software to resolve the problem which occurs in component.

- Moreover, By comparing the both analysis results i.e (actual and redesign) which ensure whether the design is to be safe or not.

1.5 DETAILED VIEW:

The detailed view of the rear axle casing is helpful for design and analysis process. The views are given below.

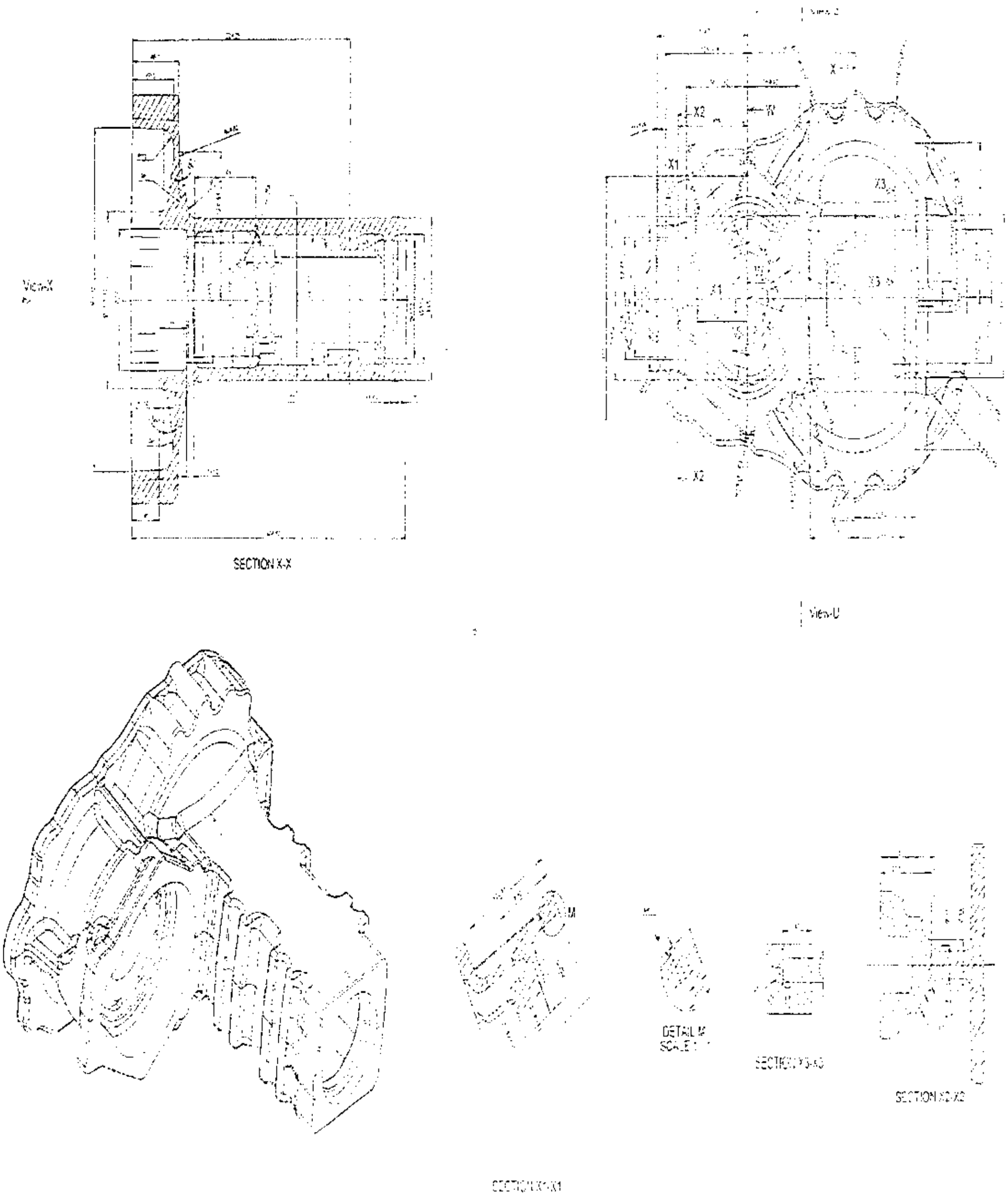

Fig1.4 Detailed View

Chapter 2

Literature Review

CHAPTER 2

LITERATURE REVIEW

Following are the overview of the relevant work done earlier related to the problem identified and the methodology to be adopted to solve the chosen problem for this work. It gives the description of literature reviewed from various research papers published in international and national journals, proceedings of various conferences and books.

Corbin [1993] explained Solder Ball Connect (SBC) is a second-level surface mount electronics packaging technology in which ceramic modules containing one or more chips are joined The solder structures accommodate the bulk of the strain If the solder structures not properly designed, the thermal strain can be a source of premature fatigue failure Designed experiment techniques are used to systematically evaluate the thermal strain sensitivity to structural variables. Results are used to identify an optimally reliable structure theist robust in terms of assembly-process variables.

Antonio Barbosa et al. [1998] this paper considers the practical application of nonlinear models in the analysis of reinforced concrete structures. The results of some analyses performed using the reinforced concrete model of the general purpose finite element code Ansys are presented and discussed. The differences observed in the response of the same reinforced concrete beam as some model that is always basically the same are emphasized. The consequences of small changes in modeling are discussed and it is shown that satisfactory results may be obtained from relatively simple and limited models.

Milan Vasek [2002] provided a report giving the means of putting design procedures of certain structural elements in standard form. The design of more efficient and light structures leads to the nonlinear analysis of deflections and stresses. Some parts of structure could reach the yield point under the design load.

Some parts of structure could lose their local stability. These complexes problems of a different nature are possible to solve by FEM methods. In some types of connections the contact between surfaces is changed within the loading process and the distribution of deformations and stresses is highly dependent on the area of contact and acting force. Some experiments are necessary to provide to calibrate the computer solution.

Mutamba et al. [2002] presented the recent progress in the fabrication of microoptical devices and the maturity reached in the integration of electromechanical functions. The deformation state of the fabricated multilayer membrane structure depends on the material combination and the geometry. Differences in the material properties of individual layers and the quality of multilayer interfaces can give rise to built-in stresses leading to an ffset in the response. Taking these effects into account one can find an appropriate membrane geometry Analyses have been carried out with the ANSYS FEM (Finite Elements Method) Program for two types of electrostatically actuated Fabry-Pérot filters. This is based on an approximation replacing the electrostatic force which bends the membrane by a pressure applied on the electrode surfaces of the moving part. Structural analyses have been carried out for two electrostatically actuated Fabry-Pérot filters based on two different material systems and membrane design. A mechanical model replacing the tuning electrostatic force with a mechanical pressure on the mirror electrode surfaces was used.

Kermanidis et al. [2002] presented a Methodology for three-dimensional progressive damage model was developed in order to simulate the damage accumulation of bolted single-lap composite joints under in-plane tensile loading. This model is capable of predicting the residual strength and residual stiffness of laminates with arbitrary lay-ups, geometries and bolt positions. The parametric study includes stress analysis, failure analysis and material property degradation. Stress analysis of the three dimensional geometry was performed using the ANSYS FE code. Failure analysis and degradation of material properties were implemented using a progressive damage model, which is incorporated in an ANSYS macro-routine. The progressive model utilizes a set of stress-based Hashin-type criteria and a set of appropriate degradation rules. A parametric study was performed to examine the effect of bolt position and friction upon residual strength and damage accumulation.

David Johnson [2003] this report details the construction and safety features. The vehicle is designed to be as light and robust as possible. It is important to note that since many of the systems analyzed in this vehicle use the same materials, the entire materials library for the vehicle uses trailing arms for its two rear wheels. These wheels do not steer and they carry a combined 50-60% of the total vehicle weight. The suspension is provided by two Rock Shox 7.5" long SID Rear Race air shock absorbers. The structural members are formed by two triangular sections that support the wheel in double shear. The titanium was welded together using the same process as the front suspension.

He Liu [2004], Liquid storage tanks are considered essential lifeline structures. Water storage tanks in particular, are important to the continued operation of water distribution systems in the event of earthquakes. the computer program ANSYS was selected to develop a Finite Element Analysis (FEA) model of a ground level, cylindrical steel shell and roof tank structure with contained fluid under seismic load. The ANSYS program was selected for its ability to include shell and structural steel elements, contained fluid elements, fluid-structure interactions, material and geometric nonlinearities, and contact type elements. For purposes of this study, analysis results from a linear elastic, small deformation fixed base model are compared with an elasto -plastic material property model with large deformation assumptions. Results show the significant difference in results based on the assumptions used and indicate that current design code based values may not be conservative in resultant loading calculations.

Schmid et al. [2005] in this study, mechanical structures based on polycrystalline diamond lms grown on silicon substrate are investigated. In contrast to all-silicon MEMS, additional problems arise due to the unusual properties of diamond. These effects are investigated by modeling diamond-based micro-mechanical devices and comparing the results to measurements on processed devices. All simulations were carried out using the standard simulation tool ANSYS and measured modeling parameters

Nick Cristello et al. [2006], this paper presents the shape optimization of an automotive universal joint Part modeling and analysis is conducted using the Finite Element Analysis package ANSYS. The results show Pareto frontiers for both the flange and weld yoke, constructed using the Adaptive Weighted Sum technique.

Lianyou Yu et al. [2007] provided a review of modular design methodologies. An analysis of the literature showed that the efficient for dynamically loaded lightweight structures strength is an important design criterion. In this paper a modeling approach to study the characteristic of railroad car body is shown, which is based on the experimental and numerical modal analyses. Major issues related to modal analyses of open-top wagon are resented along with respective results. Moreover, a complex numerical model of the railway vehicle suitable for reproducing its dynamic behavior is set up. In order to reduce the car body's weight, an optimal design was developed. The strength, stiffness and the models of the car body were calculated and compared by means of finite element method. The lightening plan is successfully applied to new car design, and is adopted by road tests.

Boada et al. [2007] In this paper a structural optimization of a simplified bus structure is proposed. The proposed optimization, as part of bus design methodology, has been carried out by means of genetic algorithms, and structural behavior of the bus structure has been done using the finite element method. Results show that an optimized structure, in weight and tensional stiffness, has been achieved.

Wilhelm Rust. Presented an general properties of implicit Finite Element analysis using ANSYS and explicit analysis it is shown when and how quasi-static limit load analyses can be performed by a transient analysis using explicit time integration. Then we focus on the remaining benefits of implicit analysis and how a proper combination of ANSYS and can be used to prepare the transient analysis by common preprocessing and static analysis steps. Aspects of discretization, solution control, and consideration of imperfections and methods of checking the results are outlined.

Chapter 3

Finite Element Analysis

CHAPTER 3

FINITE ELEMENT ANALYSIS

3.1 A BRIEF HISTORY

The finite element method is a numerical procedure that can be applied to obtain approximate solutions to a variety of problems in engineering. Steady, transient, linear, or nonlinear problems in stress analysis, heat transfer, fluid flow, and electromagnetism problems may be analyzed with the finite element method. The idea of representing a given domain as a collection of discrete parts is not unique to the finite element method. It was recorded that ancient mathematicians estimated the value of by noting that the perimeter of a polygon inscribed in the circle approximates the circumference of the latter. They predicted the value of accuracies of almost 40 significant digits by representing the circle as a polygon of a finitely large number of sides. In modern times, the idea found a home in aircraft structural analysis, where, for example, wings and fuselages are treated as assemblages of stringers, skins, and shear panels.

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and variational calculus to obtain approximate solutions to structural analysis. In the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense and nuclear industries. Due to the rapid decline in the cost of computers and the phenomenal increase in computing power, current FEA models have usually much greater number of elements and thus increased precision. Present day supercomputers are now able to produce more accurate results in much shorter amounts of time. At the same time, modern desktop computers can be used to analyze larger and more complex problems than could be done by the mainframes of yesteryear. FEA systems now have powerful graphics capabilities, automated functionality, and advanced user interfaces that make the technology considerably faster and easier to use. These improvements notwithstanding, however, full-blown advanced FEA still requires considerable time and the expertise of a dedicated analyst with the knowledge necessary to apply proper mesh densities, element types, and boundary conditions. These expert analysts also must know how to go about translating CAD geometry into proper format for building the FEA model as well as correctly interpreting plots and other output information. One of the driving forces in manufacturing companies is the continuing demand for reduction in product development time and cost to maintain profitability and competitiveness. Over the years, this requirement has prompted organization in a wide range of industries to find different ways of making product development more efficient. Advancement in the entire spectrum of CAD/CAM/CAE tools in particular have automated many design, engineering, and analysis tasks to shorten development cycles, mostly as a labor savings to minimize overhead costs. Progressive manufacturers are now investigating ways to further reduce design cycle-time by evaluating and changing the product development process itself. The goal here is not so much economic savings in the engineering department but rather a broad business advantage in getting product innovations to customers faster and thereby increasing a company's market share. FEA allows designers and engineers to quickly iterate back and forth in performing basic conceptual "what-if" studies to evaluate the merit of different ideas, compare alternatives, and filter out design weaknesses before more detailed analysis, prototype testing, and production planning are conducted.

3.2 FINITE ELEMENT METHOD CONCEPT

FEM is the powerful numerical method used for any type of engineering analysis. FEM will be used to analysis the designed rear axle casing under static loading condition and also to find out the natural frequency of the rear axle casing. This gives the design a great deal of strength to predict it self under the real time operating condition, which is difficult to be model or arrive in analytical methods. The finite element method is defined as discretization whole region (model) in to small finite number of elements. These small elements connected to each other at node points. Finite element analysis grew out of matrix methods for the analysis of structure when the widespread availability of the digital computer made it possible to solve systems of hundreds of simultaneous equations using FEA software like Nastran, Ansys etc.

3.2.1 FINITE ELEMENT ANALYSIS- GENERAL PROCEDURE:

The following steps summarize finite element analysis procedure.

Step 1:

The continuum is a physical body, structure, or solid being analyzed. Discretization may be simply described as the process in which the given body is subdivided into an equivalent system of finite elements. The finite elements may be triangles, group of triangles or quadrilaterals for a two dimensional continuum. The collection of the elements is called finite element mesh. The elements are connected to each other at points called nodes. The choice of element type, number of elements and density of elements depend on the geometry of the domain, the problem to be analyzed.

Step 2:

The selection of the displacement models representing approximately the actual distribution of the displacement. The three interrelated factors, which influence the selection of a displacement model, are

(i) The type and degree of displacement model
(ii) Displacement magnitudes and
(iii) The requirements to be satisfied which ensuring correct solution.

Step 3:

The derivation of the stiffness matrix that consists of the coefficients of the equilibrium equation derived from the material and geometric properties of an element. The stiffness relates the displacement at nodal points to the applied forces at nodal points.

[K] {q}	=	{F}
[K]	-	Stiffness matrix
{F}	-	Force vector

{q} - Nodal displacement vector

Step 4:

Assembly of the algebraic equation for the overall discretized continuum includes the assembly of the overall stiffness matrix for the entire body from individual element stiffness matrices, and the overall global load vector from the elemental load vectors. The most commonly used technique was direct stiffness method.

The overall equilibrium relations between the total stiffness matrix [K], the total force vector {R}, and the node displacement vector for the entire body {r} can be expressed as

$$[K]\{r\} = \{R\}$$

Step 5:

The algebraic equations assembled in step 4 are solved for unknown displacements. In linear equilibrium problems, this is a relatively straightforward application of matrix algebra techniques.

Step 6:

In this step, the element strains and stress are computed from the nodal displacements.

3.2.2 GENERAL STRUCTURE OF FEM PROCEDURE:

The analysis of a structure during its design process is accomplished by the solution of the partial differential equations, which describe the given model. This involves the following three steps.

1. **PRE-PROCESSING**
   - Type of analysis
   - Element type
   - Material properties
   - Real constants

- ❖ Build the model
- ❖ Element size
- ❖ Meshing
- ❖ Loads and boundary conditions (constraints).

2. SOLUTION

- ❖ To obtain solution (Displacement, stress etc)

3. POST-PROCESSING

- ❖ Review the results
- ❖ Graphical display of Displacements and stresses.
- ❖ For quick and easy Interpretation, sorting and Printing of results.

3.2.3 STRUCTURAL ANALYSIS:

The primary faction of a structure is to receive loads usually known as service loads at certain points and transmit them safely to some other points. For instance, a building frame receives occupancy loads of the building besides the self – weight of the structural components of the building and transfers them safely to the foundations. Similarly, a highway bridge has to support the live load due to the traffic and the dead load of the bridge itself besides several other loads. The structural system of the bridge has to be designed so as to transmit these loads safely through the supporting piers and abutments to the foundations. In performing this primary function of receiving service loads at certain points and transferring them safely to some other points, the structure develops internal forces in its component members known as structural elements. It is the responsibility of the structural engineer to design all the structural elements of a structural system in such a way that they perform their functions adequately. The inadequacy of one or more structural elements may lead to the malfunctioning or even collapse of the entire structure. The object of structural analysis is to determine the internal forces and the corresponding displacements of all the structural elements as well as those of the entire structural system. The safety and proper functioning of the structure can be ensured only through a thorough structural analysis. The importance of a correct structural analysis for the proper functioning and safety of the structure cannot, therefore, be over – emphasized. A systematic analysis of structural systems can be carried out by using matrices. The matrix approach for the solution of structural problems is also eminently suitable for a solution using modern digital computers. Hence, the advantages of using the matrix approach for large structural problems in evident.

3.2.4 DEFINITION OF STRUCTURAL ANALYSIS:

Structural analysis is probably the most common application of the finite element method. The term structural (or structure) implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

- **Types of Structural Analysis**

The seven types of structural analyses available in the ANSYS family of products are explained below. The primary unknowns (nodal degrees of freedom) calculated in a structural analysis are displacements. Other quantities, such as strains, stresses, and reaction forces, are then derived from the nodal displacements.

Structural analyses are available in the ANSYS/Multiphysics, ANSYS/Mechanical, ANSYS/Structural, and ANSYS/Professional programs only.

- **Harmonic Analysis** - Used to determine the response of a structure to harmonically time-varying loads.
- **Transient Dynamic Analysis** - Used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed.
- **Spectrum Analysis** - An extension of the modal analysis, used to calculate stresses and strains due to a response spectrum or a PSD input (random vibrations).

Buckling Analysis - Used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigen value) buckling and nonlinear buckling analyses are possible.

- **Explicit Dynamics Analysis** - ANSYS provides an interface to the LS-DYNA explicit finite element program and is used to calculate fast solutions for large deformation dynamics and complex contact problems. In addition to the above analysis types, several special-purpose features are available:

  - Fracture mechanics
  - Composites
  - Fatigue
  - p-Method
  - Beam Analyses

Static Analysis

Used to determine displacements, stresses, etc. under static loading conditions. Both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyperelasticity, contact surfaces, and creep.

We encounter structural linearity on a routine basis. For instance, whenever you staple two pieces of paper together, the metal staples are permanently bent into a different shape. Heavy load Dropped into the wooden slab or Metal plate. As weight is added to a car or truck, the contact surfaces between its pneumatic tires and the underlying permanent change in response to the added load. If you were to plot the load-deflection curve for each application, you would discover that they all exhibit the fundamental characteristic of nonlinear structural behavior - a changing structural stiffness.

Modal Analysis:

Used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available. You use modal analysis to determine the natural frequencies and mode shapes of a structure. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. You can do modal analysis on a prestressed structure, such as a spinning turbine blade. Another useful feature is modal cyclic symmetry, which allows you to review the mode shapes of a cyclically symmetric structure by modeling just a sector of it. Modal analysis in the ANSYS family of products is a linear analysis. Any nonlinearities, such as plasticity and contact (gap) elements, are ignored even if they are defined. You can choose from several mode-extraction methods: Block Lanczos (default), subspace, Power Dynamics, reduced, unsymmetrical, damped, and QR damped. The damped and QR damped methods allow you to include damping in the structure. Details about mode-extraction methods are covered later in this section.

The procedure for a modal analysis consists of four main steps:

1. Build the model.
2. Apply loads and obtain the solution.
3. Expand the modes.
4. Review the results

When building your model, remember these points:

- Only linear behavior is valid in a modal analysis. If you specify nonlinear elements, they are treated as linear. For example, if you include contact elements, their stiffness are calculated based on their initial status and never change.

- Material properties can be linear, isotropic or orthotropic, and constant or temperature-dependent. You must define both Young's modulus (EX) (or stiffness in some form) and density (DENS) (or mass in some form) for a modal analysis. Nonlinear properties are ignored. If applying element damping, you must define the required real constants for the specific element type.

Apply Loads

After defining master degrees of freedom, apply loads on the model. The only "loads" valid in a typical modal analysis are zero-value displacement constraints. (If you input a nonzero displacement constraint, the program assigns a zero-value constraint to that DOF instead.) Other loads can be specified, For directions in which no constraints are specified, the program calculates rigid-body (zero-frequency) as well as higher (nonzero frequency) free body modes

3.3 THE REASON FOR USING THE FEA METHOD IN THIS STUDY

The theoretical concept implies that the higher stress concentration is measured near the location where cracks had been reported was substantially higher than at other locations. As it was mentioned in the previous paragraph, the fastest and most cost-effective approach to investigate this problem, would be based on the Finite Elements Method. Once the mesh is developed, it is relatively easy to study the effect of the material for applying load. Changes in shape require substantially more effort, but not nearly as much as fabricating a new prototype, by using a computational analysis of rear axle casing to easily found the strength of component by avoiding the failure.

3.4 GENERATION OF THE GEOMETRY

The drawings of rear axle casing of the HMT tractor products are made using the PRO-E Wildfire 3.0 drafting package. The detailed 3D drawings of the front and back view of the rear axle casing were transferred to the ANSYS 10.0 CAE package. This package performs preprocessing functions, such as meshing, defining contact conditions. The rear axle case has been treated as solid in the model. The thicknesses of different areas were taken from the detailed drawings, but in the areas of special interest, such as at the location of the deformation.

3.5 MATERIAL PROPERTIES

The next step was assigning material properties to component. every material having a certain property based upon this material property engineering analysis can be carried out. Here rear axle casing is a casting component so normally used and assigned material for rear axle casing is (CAST IRON sg 500). The property is given in the following table 3.1

Table 3.1 Material properties of rear axle casing

Parameters	Value
Modulus of elasticity (E)	$100*10^3$ N/mm^2
Poisson ratio (μ)	0.23 to 0.26
Density (ρ)	(7.85*10-6 kg/mm^3
Maximum load (L)	7000N

3.6 LOAD CONVERSION

The approach in the FEA study was to use the static load acting of the rear axle casing as the loading. But the major constraints in FEA, load is not applied directly on the unit area so we have to convert that into pressure by using a mathematical formula and then applies according to boundary condition.

- Load = 5000N
- Area = 192.5 mm^2
- Pressure $= \frac{load}{unit\ area}$ N/mm^2

  = (5000 /192.5) N/mm^2

  = 26 N/mm^2

3.7 DESIGN CONSTRAINTS

For every engineering material had some material properties and design constraints. In this project we choose a cast iron sg 500 for rear axle casing. For these material had some design constraints analysis results should not exceed these if exceed failure happens. The design constraints are mention in the table 3.2.

Table 3.2 Design constraint

Stress	**165 N/mm^2**
Displacement	**(0.3 to 0.4) mm**
Load	**5000 N**
Pressure	**26 N/mm^2**

3.8 ELEMENT DETAILS

SOLID92 Element is chosen for analysis; SOLID92 has a quadratic displacement behavior and is well suited to model irregular meshes (such as produced from various CAD/CAM systems). See SOLID92 for a 20-node brick shaped element. The element is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element also has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

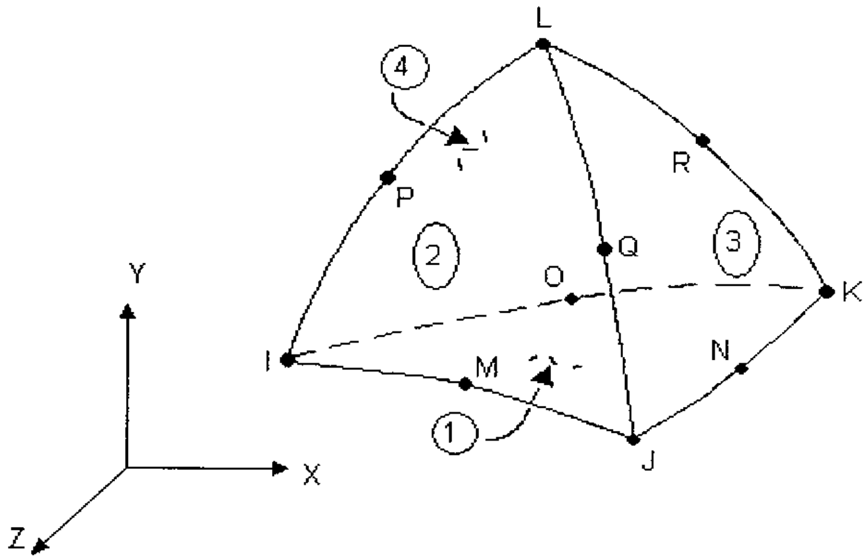

Fig 3.1 Analysis Element geometry (SOLID92 Geometry)

The geometry, node locations, and the coordinate system for this element are shown in Figure 3.1 "SOLID92 Geometry". Beside the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems. Element loads are described in Node and Element Loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers "SOLID92 Geometry". Positive pressures act into the element. Temperatures and fluences may be input as element body loads at the nodes. The node I temperature T(I). If all other temperatures are unspecified, they default to T. If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures.

3.9 STRESS CONCENTRATION:

The fracture of a material is dependent upon the forces that exist between the atoms. Because of the forces that exist between the atoms, there is a theoretical strength that is typically estimated to be one-tenth of the elastic modulus of the material. However, the experimentally measured fracture strengths of materials are found to be 10 to 1000 times below this theoretical value. The discrepancy is explained to exist because of the presence of small flaws or cracks found either on the surface or within the material. These flaws cause the stress surrounding the flaw to be amplified where the magnification is dependent upon the orientation and geometry of the flaw. Looking at fig. 1, one can see a stress profile across a cross section containing an internal, elliptically-shaped crack. One can see that the stress is at a maximum at the crack tip and decreased to the nominal applied stress with increasing distance away from the crack. The stress is concentrated around the crack tip or flaw developing the concept of stress concentration. Stress raisers are defined as the flaws having the ability to amplify an applied stress in the locale.

DETERMINATION OF THE MAXIMUM STRESS AT THE CRACK TIP:

If the crack is assumed to have an elliptical shape and is oriented with its long axis perpendicular to the applied stress, the maximum stress, □$_m$ can be approximated at the crack tip by Equation 1. Eqn. 1: Determination of the maximum stress surrounding a crack tip.The magnitude of the nominal applied tensile stress ; the radius of the curvature of the crack tip; and ***a*** represents the length of a surface crack, or half the length of an internal crack.

$$\sigma_m = 2\sigma_0\left(\frac{a}{\rho_t}\right)^{1/2}$$

DETERMINATION OF STRESS CONCENTRATION FACTOR:

The ratio of the maximum stress and the nominal applied tensile stress is denoted as the stress concentration factor, K_t, where K_t can be calculated by Equation 2. The stress concentration factor is a simple measure of the degree to which an external stress is amplified at the tip of a small crack.

$$K = \frac{\sigma_m}{\sigma_0} = 2\left(\frac{a}{\rho_t}\right)^{1/2}$$

Eqn. 2: Determination of the stress concentration factor.

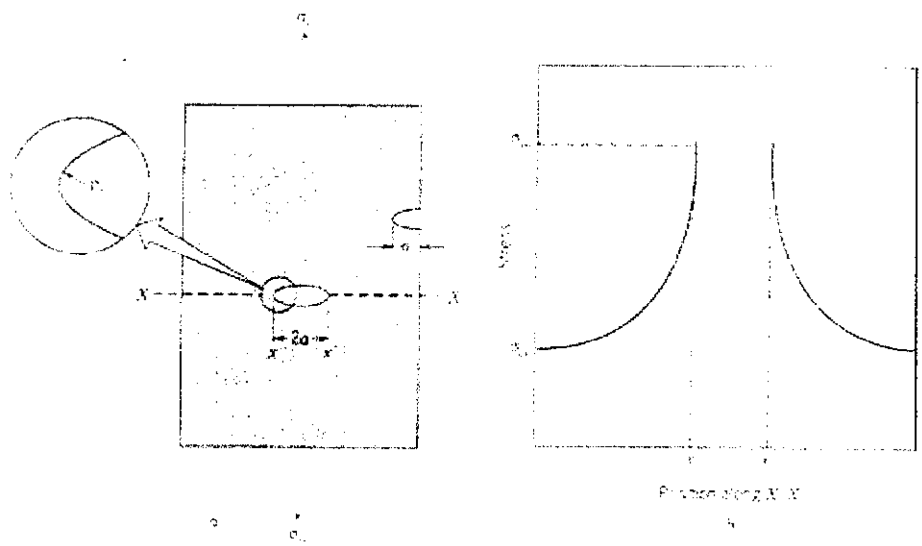

Fig. 3.2(a) The geometry of surface and internal cracks. (b) Schematic stress profile along the line X-X' in (a), demonstrating stress amplification at crack tip positions.

STRESS CONCENTRATION CONSIDERATIONS:

It is important to remember that stress amplification not only occurs on a microscopic level (e.g. small flaws or cracks,) but can also occur on the macroscopic level in the case of sharp corners, holes, fillets, and notches. Fig. 2 depicts the theoretical stress concentration factor curves for several simple and common material geometries. Stress raisers are typically more destructive in brittle materials. Ductile materials have the ability to plastically deform in the region surrounding the stress raisers which in turn evenly distributes the stress load around the flaw. The maximum stress concentration factor results in a value less than that found for the theoretical value. Since brittle materials cannot plastically deform, the stress raisers will create the theoretical stress concentration situation.

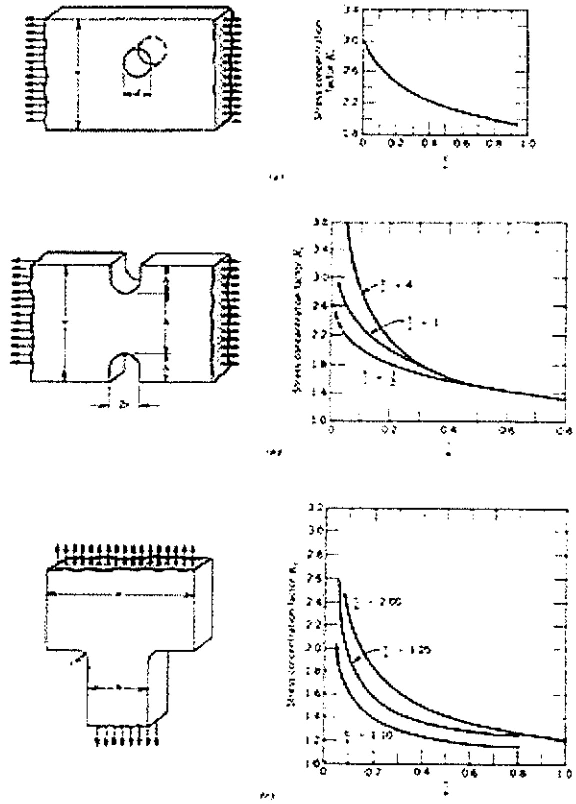

Fig. 3.3: Stress concentration factor plots for three different macroscopic flaw situations.

3.10 MESHING

Meshing generation is the back bone of the FEA. Mesh generation refers to the generation of nodal coordinates and elements. It also includes the automatic numbering of nodes and elements based on a minimal amount of user-supplied data. Automatic mesh generation reduces errors and saves a great deal of user time, therefore reducing the FEA cost. In this work, meshing was done automatically with the help of the software. Here quadratic element is used for free mesh.

Chapter 4

Analysis of Failed Rear Axle Casing

CHAPTER 4

ANALYSIS OF THE FAILED REAR AXLE CASING

4.1 ACTUAL MODEL

It is considered here failure model is the actual model these was modeled by using the geometrical data which is shown in fig 1.4. The 3D MODEL is modeled by using the parametric software PRO-E WILDFIRE 3.0. This can be shown in fig 4.1 based upon this model the analysis were carried out.

4.2 STRUCTURAL ANALYSIS

Structural analysis is probably the most common application of the finite element method. The term structural implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical and mechanical structures such as ship hulls, aircraft bodies, and machine housing, as well mechanical components and tools. The primaries unknown (nodal degrees of freedom) calculated in a structural analysis are displacements. Other quantities, such as strains, stresses, and reaction forces, are then derived from the displacements. Here the rear axle casing is taken into account to encounter the reason for failure while its working in maximum loading condition it was carried out in two ways static and modal analysis.

4.2.1 STATIC ANALYSIS:

Static analysis is used to determine displacements, stresses, etc. under static loading conditions which includes both linear and nonlinear characteristics. Nonlinearities can include plasticity, stress stiffening. Rear axle casing is rigid component it encloses a differential unit assembly, this projects describes the effect of maximum static load which acts on this component while it works in different condition. Static analysis were carried out in a two section displacement, stress

4.2.1.1 DISPLACEMENT:

For every engineering analysis displacement plays a major role to determine the maximum deflection for applying load. Here the rear axle casing is fixed in one end; load is acting on the other end. Design specification and constraints for analysis are shown in the table 3.1 After analysis the obtained maximum deflection of the failured rear axle casing is that 0.27mm shown if fig 4.2 , according to design constraints the deflection should not not be exceeded 0.4mm. by refering that it ensures the failure doesn't occur due to the displacement.

4.2.1.2 STRESS:

Stress analysis is one of the very important analysis for any structure because in this stage to know the capability of the material to withstand in maximum load. Here the load is not directly applied to an unit area so we have to convert the load into pressure by using mathematical formula. After computational analysis the maximum stress obtained for the rear axle casing is (235.96 N/mm^2), according to design constraints the stress limit should not be exceed to (165 N/mm^2). The fig 4.3 shows stress is high in the fixed area. it ensures that due to stress concentration the rear axle casing deform based on theoritical concept it considered redesign it will be discussed later.

4.3 TIME STEPPING ANALYSIS

Time stepping is a non- linear analysis mainly it is used to find out at what load the failure happens. So that the applied maximum load is divided into many steps using the computer module as our requirement. These analyses were carried out for both displacement and stress to find out the failure.

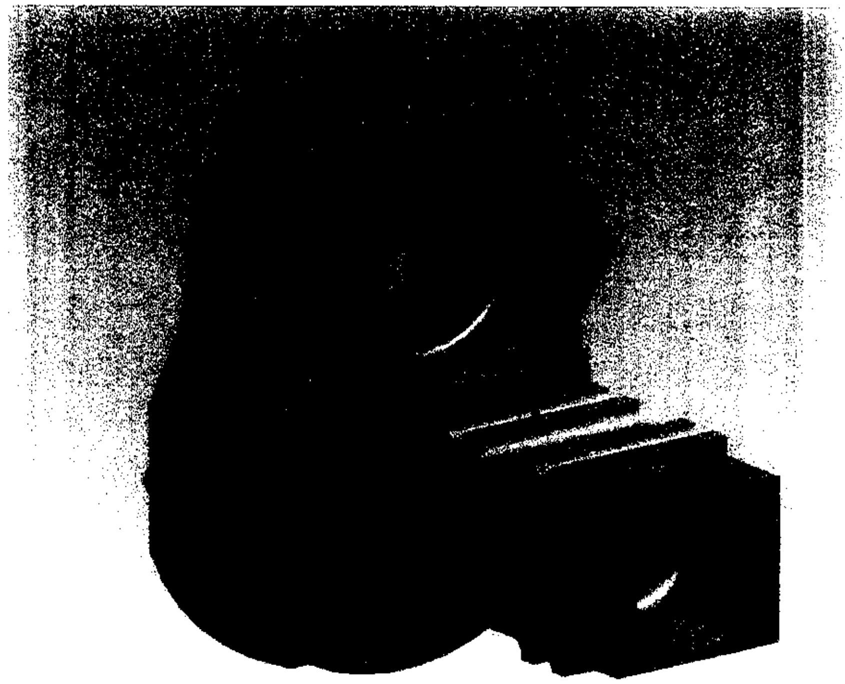

Fig 4.1 Failed rear axle casing 3D model

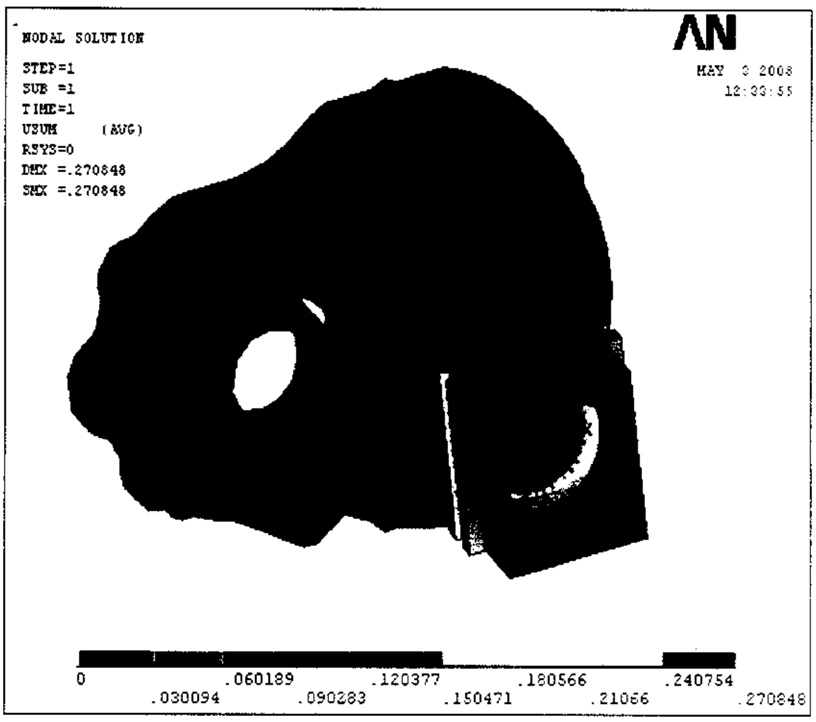

Fig 4.2 Displacement analysis for failed rear axle casing

4.3.1 DISPLACEMENT RESULT:

As said in the last paragraph according to design constraints, the obtained results shows in table the deflection doesn't exceed. it ensure that component is safe. The tenth step result shown bellow in fig4.4

Table 4.1 Time stepping displacement result for failed rear axle Casing

	PRESSURE (N/mm2)	DISPLACEMENT (mm)
STEP 1	2.6	0.027
STEP 2	5.2	0.054
STEP 3	7.8	0.081
STEP 4	10.4	0.108
STEP 5	12	0.135
STEP 6	15.6	0.162
STEP 7	18.2	0.189
STEP 8	20.8	0.216
STEP 9	23.4	0.243
STEP 10	26	0.270

4.3.2 STRESS RESULT:

Time stepping analysis for stress the result shows in the below table .clearly the results entails that deformation takes place after the seventh step. Upto step7 the component is safe but increasing the load to next step8 20.8 the stress beyond the assigned stress level it ensures that deformation occur by applying load 20.8 N/mm^2. the ANSYS results shown in the following fig 4.5 and fig 4.6.

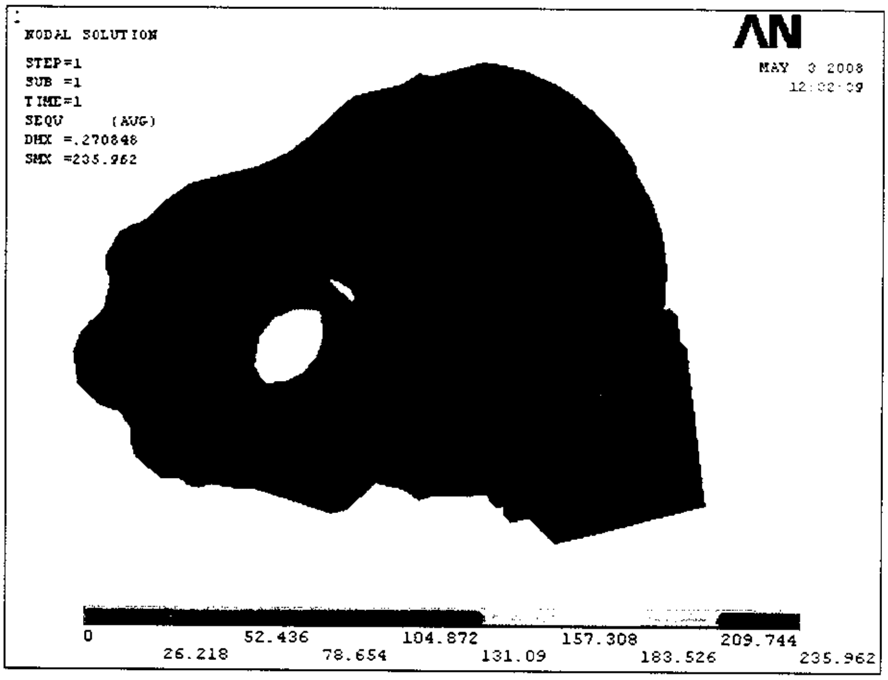

Fig 4.3 Stress analysis for failed rear axle casing

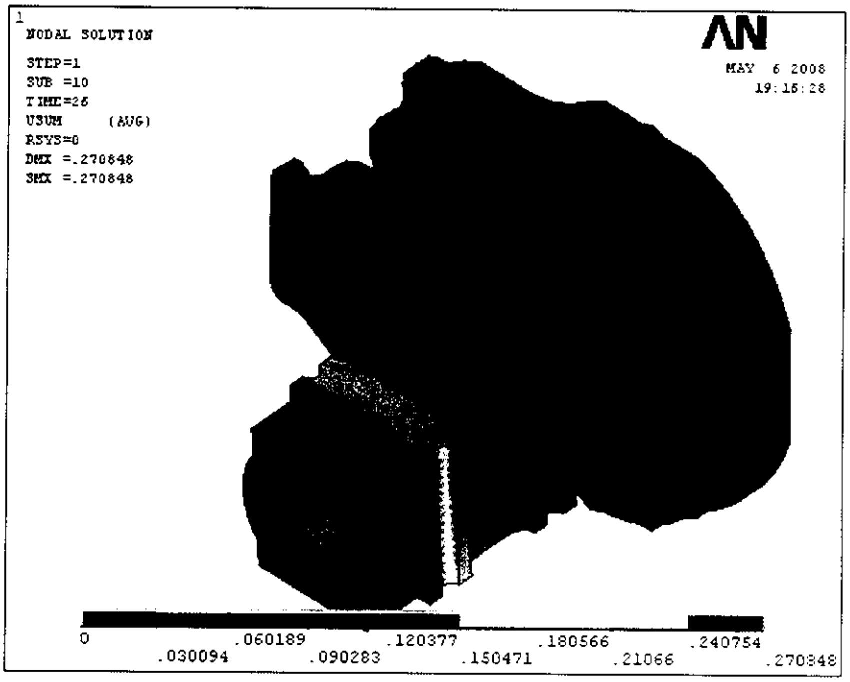

Fig 4.4 Time stepping displacement results for failed rear axle Casing (step 10)

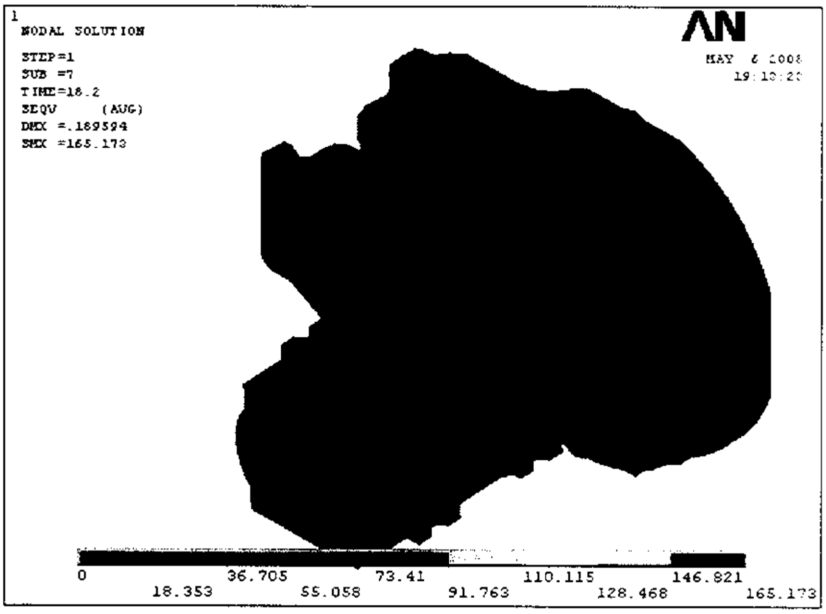

Fig 4.5 Time stepping stress results for failed rear axle casing (step 7)

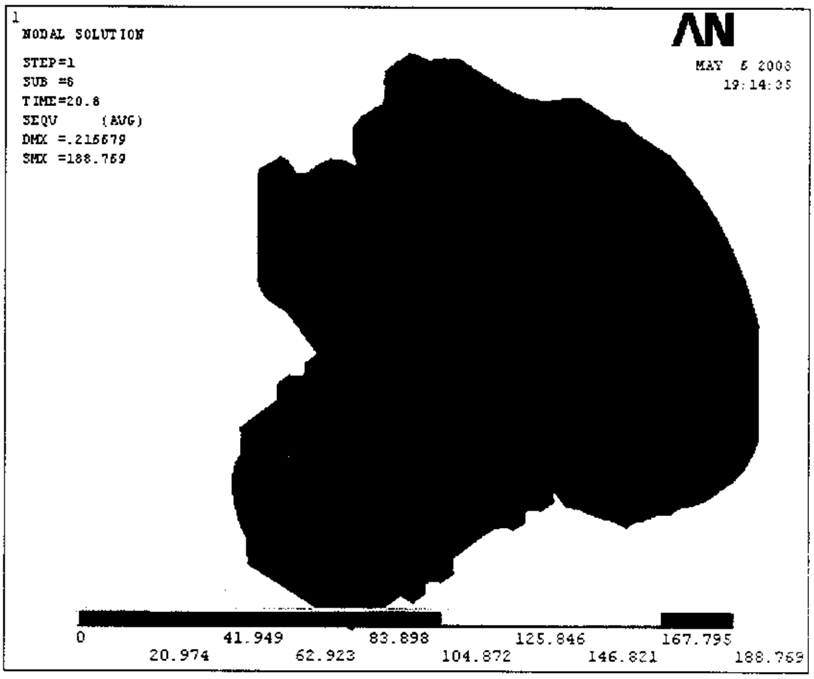

Fig 4.6 Time stepping stress results for failed rear axle casing (step 8)

Table 4.2 Time stepping stress result for failed rear axle casing

	PRESSURE (N/mm2)	STRESS (N/mm2)
STEP 1	2.6	23.596
STEP 2	5.2	47.192
STEP 3	7.8	70.789
STEP 4	10.4	94.385
STEP 5	12	117.981
STEP 6	15.6	141.577
STEP 7	18.2	165.173
STEP 8	20.8	188.769
STEP 9	23.4	212.366
STEP 10	26	235.962

4.4 MODAL ANALYSIS

Modal analysis is used to determine the natural frequencies and mode shapes of a structure. The natural frequencies and mode shapes are important parameters in the design of a structure for static and dynamic loading conditions. Modal analysis is carried out on a prestressed structure, such as a rear axle casing. Another useful feature is modal cyclic symmetry, which allows you to review the mode shapes of a structure by modeling just a sector of it.. Details about mode-extraction methods are given in the table

4.4.1 FREE-FREE ANALYSIS:

The natural frequencies and mode shapes are important parameters in the design of a structure for static or dynamic loading conditions. For every component or structure before used in real time application considered for vibration analysis. Initially they go free-free analysis to find out the original natural frequency. Based on frequency vibrational analyses are done. After analysis the natural frequency for rear axle casing is (6.099 HZ). After applying boundary conditions the frequency should not be exceed the natural frequency. Mode results are show in the table 4.3 and the figure 4.7 shows ANSYS result by computational analysis the natural frequency occurs in mode7.

Table 4.3 Modal free-free analysis results for failed component

MODE	NATURAL FREQUENCY (HZ)
Mode 1	0
Mode 2	0
Mode 3	0
Mode 4	0
Mode 5	0
Mode 6	$0.679*10^{-5}$
Mode 7	6.099

4.4.2 ONE END FIXED:

Here the boundary conditions are one end is fixed and other load Is applied. By applying this condition we obtain the frequency for maximum load. Here the failure happened at the MODE3 so the safer limit of the component is 2. The results are shown in the table 4.4 and ANSYS result shown in fig 4.8.

Table 4.4 Modal one end fixed analysis results for failed component

MODE	NATURAL FREQUENCY (HZ)
Mode 1	3.879
Mode 2	4.615
Mode 3	8.843
Mode 4	14.192
Mode 5	15.466
Mode 6	16.475
Mode 7	17.066

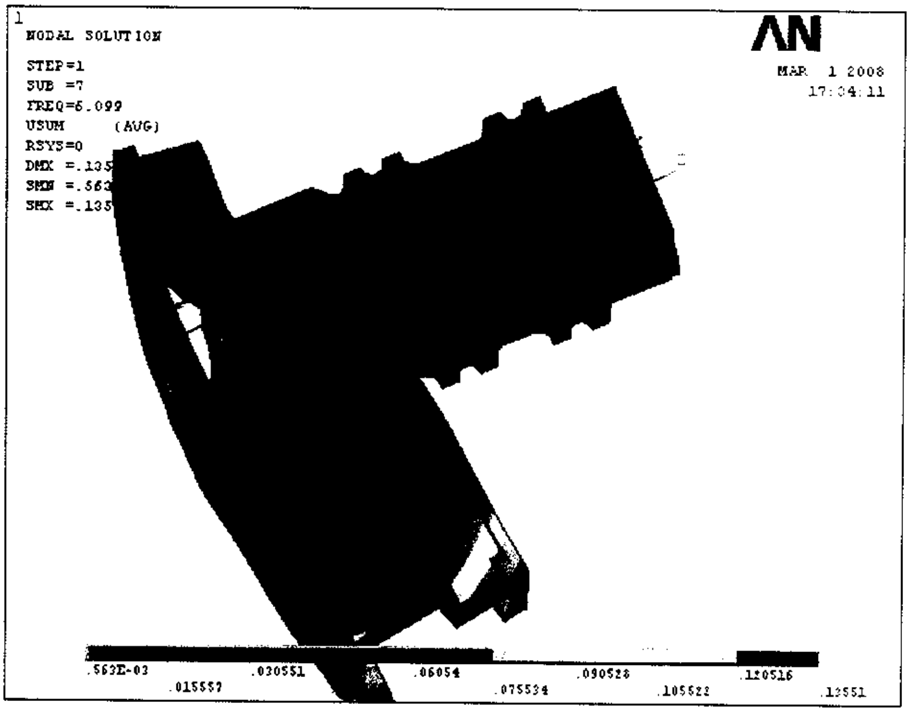

Fig 4.7 Modal free-free analysis results for failed component

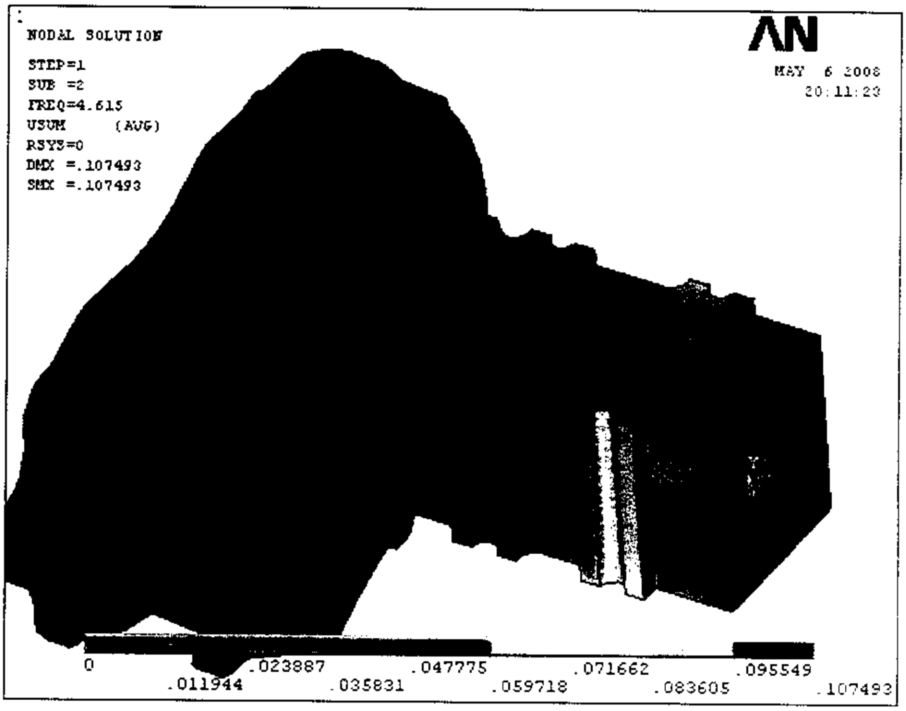

Fig 4.8 Modal one end fixed analysis results for failed component

Chapter 5

Analysis of the Redesigned Rear Axle Casing

CHAPTER 5

ANALYSIS OF REDESIGNED REAR AXLE CASING

5.1 REDESIGNED COMPONENT

Analyzed are done in different module to found out the problem for failure. By referring some theoretical concepts failed component is considered for redesign and analysis. Here redesigned are done by some modification i.e. by adding some ribs in appropriate area to withstand the load is shown in fig 5.1. This 3D MODEL was done using same software. Further it taken into analysis in a different module by using analysis software ANSYS 10.0.

5.2 STRUCTURAL ANALYSIS

Structural analysis is probably the most common application of the finite element method. Here the redesigned rear axle casing is taken into analysis to resolve the failure while its working in maximum load condition. it was carried out by adopting the same analysis procedure and software as done earlier. The primaries unknown (nodal degrees of freedom) calculated in a structural analysis are displacements. Other quantities such as stress, natural frequency etc.

5.2.1 STATIC ANALYSIS:

Static analysis is used to determine displacements, stresses, etc. under static loading conditions which includes both linear and nonlinear characteristics. Redesigned rear axle casing is taken for computational static analysis to resolve the problem which is the main cause of the failure while working in different working condition. By using the same analysis procedure the static analysis it were carried out in a two ways displacement and stress.

5.2.1.1 DISPLACEMENT:

For every engineering analysis displacement plays a major role to determine the maximum deflection for applied load. Here the redesigned rear axle casing is considered for analysis by using same method and applying the same boundary condition to resolve the problem. Design specification and constraints for analysis are shown in the table After analysis the obtained maximum deflection of the redesigned rear axle casing is that (0.45mm) shown if fig 5.2 , according to design constraints the deflection should not be exceeded to (0.4mm) here it exceeds some few fractions it is negible by refering that it ensures the failure doesn't occur due to the displacement.

5.2.1.2 STRESS:

Stress analysis is one of the very important analysis for any structure because in this stage only to know the capability of the material to withstand in maximum load. Here redesigned casing is considered for analysis. After computational analysis the maximum stress obtained for the rear axle casing is (158 N/mm^2), according to design constraints the stress limit should not be exceed to (165 N/mm^2). The fig 5.3 shows stress is not high in the fixed area. it ensures that stress concentration removes by making some modification in the rear axle casing design.

5.3 TIME STEPPING ANALYSIS

Time stepping is a non- linear analysis mainly it is used to find out at what load the failure happens. So that the applied maximum load is divided into many steps using the computer module as our requirement. Here redesigned rear axle casing is considered for analyses for both displacement and stress to find out whether the redesigned component would satisfy the design constraints or not.

Fig 5.1 Redesigned rear axle casing 3D model

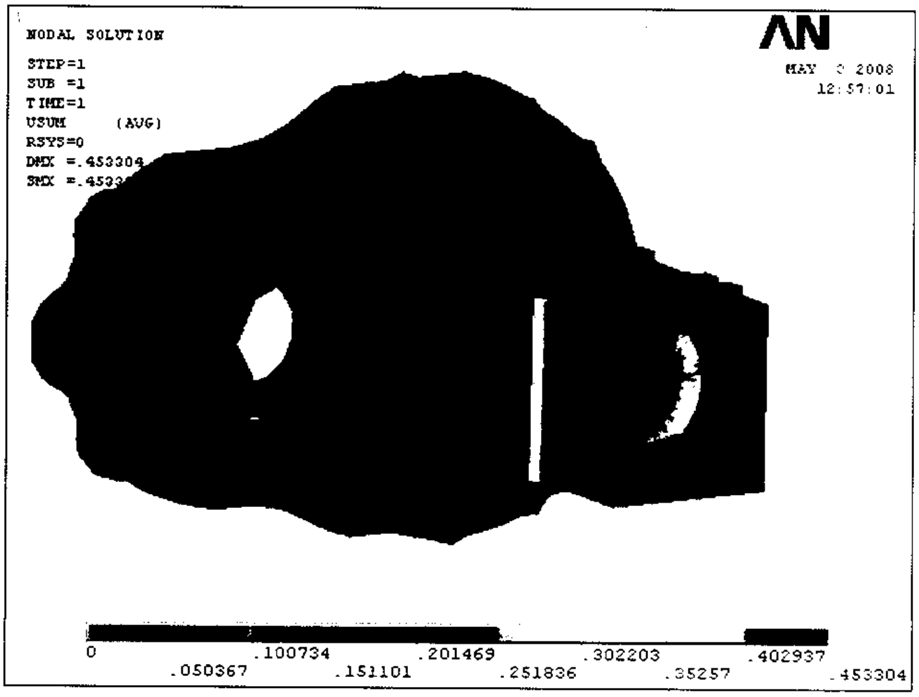

Fig 5.2 Displacement analysis for redesigned rear axle casing

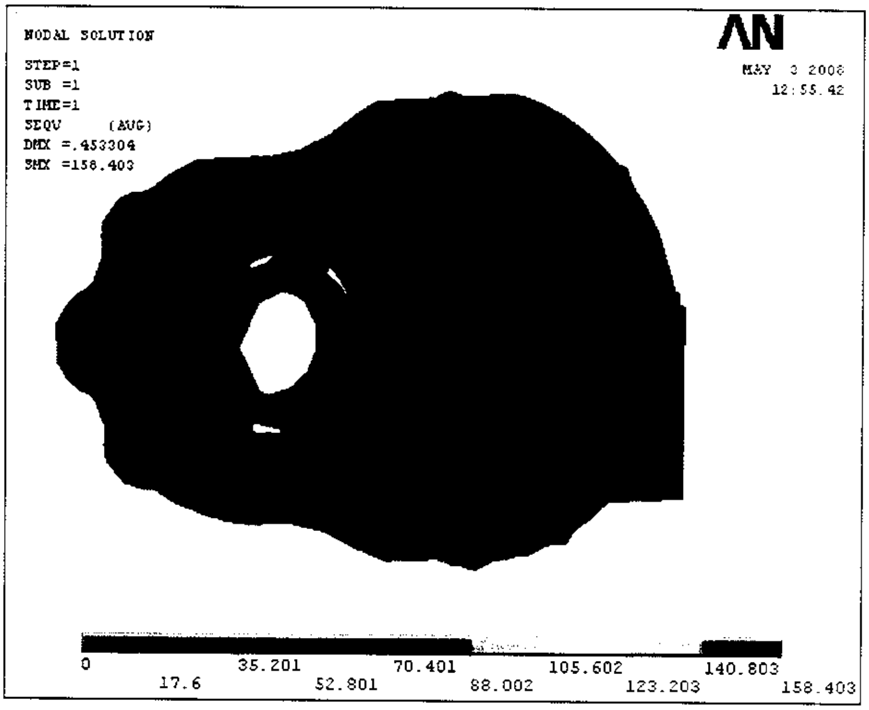

Fig 5.3 Stress analysis for redesigned rear axle casing

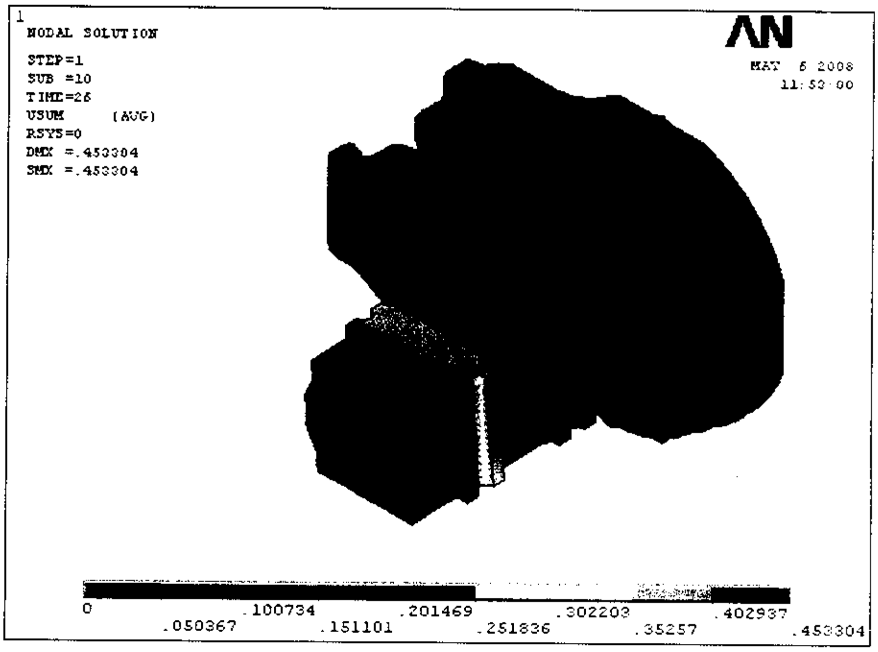

Fig 5.4 Time stepping displacement analysis result for redesigned Rear axle casing (step 10)

5.3.1 DISPLACEMENT RESULT:

As said in the last paragraph according to design constraints, the obtained results shows in table 5.1 the deflection doesn't exceed. it ensure that component is in safer limit. The tenth step result shown bellow in fig 5.4. Here at step10 the deflection doesn't exceed its in safer limit i.e. merely equal to constraint.

Table 5.1 Time stepping displacement results for redesigned rear Axle casing

	PRESSURE (N/mm2)	DISPLACEMENT (mm)
STEP 1	2.6	0.045
STEP 2	5.2	0.090
STEP 3	7.8	0.135
STEP 4	10.4	0.181
STEP 5	12	0.226
STEP 6	15.6	0.271
STEP 7	18.2	0.317
STEP 8	20.8	0.362
STEP 9	23.4	0.407
STEP 10	26	0.453

5.3.2 STRESS RESULT:

After computational time stepping analysis the maximum stress obtained for the rear axle casing is (158 N/mm^2), according to design constraints the stress limit should not be exceed to (165 N/mm^2). The fig 5.5 shows stress is not high in the fixed area. it ensures that stress concentration removes by making some modification in the rear axle casing. Here at step10 stress limit does't exceed it proves design of that rear axle casing is safer shown in the table 5.2.

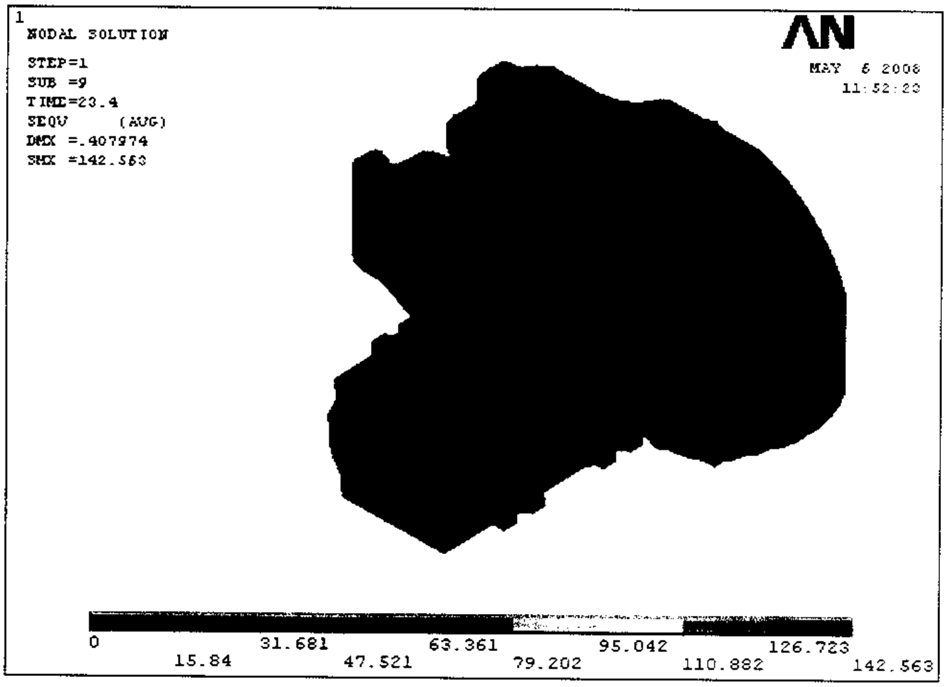

Fig 5.5 Time stepping stress analysis result for redesigned rear Axle casing (step 9)

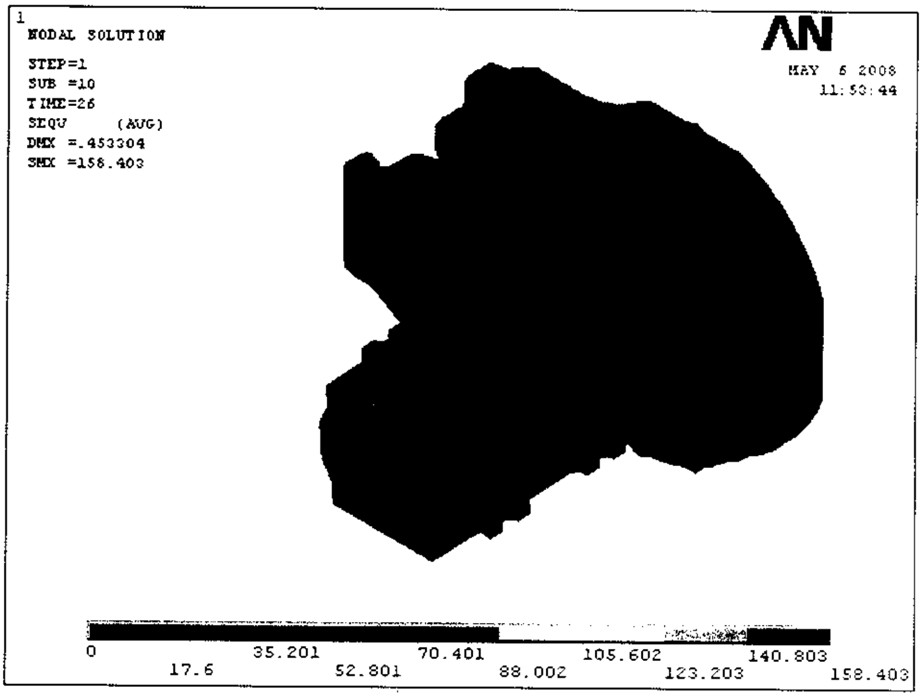

Fig 5.6 Time stepping stress analysis result for redesigned rear Axle casing (step10)

Table 5.2 Time stepping stress results for redesigned rear axle casing

	PRESSURE (N/mm2)	STRESS (N/mm2)
STEP 1	2.6	15.84
STEP 2	5.2	31.681
STEP 3	7.8	47.521
STEP 4	10.4	63.361
STEP 5	12	79.202
STEP 6	15.6	95.042
STEP 7	18.2	110.882
STEP 8	20.8	126.723
STEP 9	23.4	142.563
STEP 10	26	158.403

5.4 MODAL ANALYSIS

Modal analysis is used to determine the natural frequencies and mode shapes of a structure. The natural frequencies and mode shapes are important parameters in the design of a structure for static and dynamic loading conditions. Here modal analysis for redesigned rear axle casing to know the natural frequency for the new one Details about mode-extraction methods are given in the table

5.4.1 FREE-FREE:

The natural frequencies and mode shapes are important parameters in the design of a structure for static or dynamic loading conditions. For every component or structure before used in real time application considered for vibration analysis. Initially they go free-free analysis to find out the original natural frequency. Based on calculated frequency analyses are carried out. Here the redesigned component considered for free-free analysis to know the natural frequency after analysis the natural frequency for rear axle casing is (6.48 HZ). By applying boundary conditions the frequency should not be exceed these natural frequency. Mode results are show in the table and the figure 5.7shows ANSYS result the natural frequency always obtained in MODE7.

Table 5.3 Modal free-free analysis results for redesigned component

MODE	NATURAL FREQUENCY (HZ)
Mode 1	0
Mode 2	0
Mode 3	0
Mode 4	0
Mode 5	$0.198*10^{-5}$
Mode 6	$0.615*10^{-5}$
Mode 7	6.485

5.4.2 ONE END FIXED:

In the real time application, the real axle casing is fixed in one end and the other is load carrying. After the free-free analysis by applying the boundary condition the modal analysis is carried out for fixed end. In this section easily identified the factor of safety. In the computational modal analysis redesigned casing is fixed an one end in the other end load is applied by using this constraint the analysis are done the mode results are shown in table and ANSYS results in fig 5.8. At the MODE3 deformation takes place it exceeds the original natural frequency of this model. At mode2 the component withstand the vibration these ensures it is a safer limit.

Table 5.4 Modal one end fixed analysis results for redesigned component

MODE	NATURAL FREQUENCY (HZ)
Mode 1	4.369
Mode 2	5.214
Mode 3	10.269
Mode 4	14.302
Mode 5	15.462
Mode 6	15.848
Mode 7	16.718

Fig 5.7 Modal free-free analysis results for redesigned component

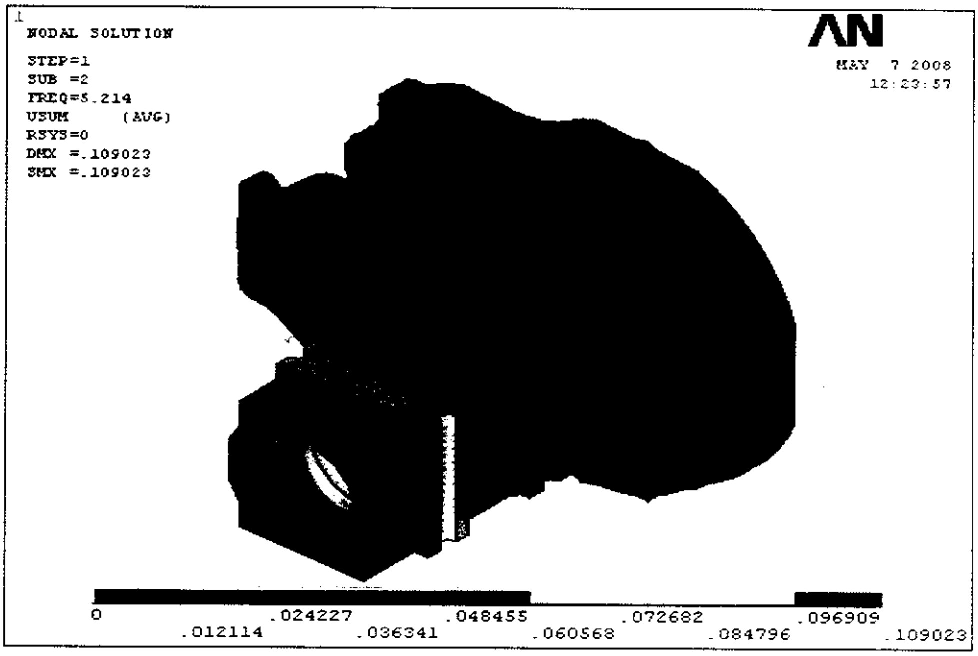

Fig 5.8 Modal one end fixed analysis results for redesigned component

Chapter 6

Comparison of Results after Analysis

CHAPTER 6

COMPARISON OF RESULTS AFTER ANALYSIS

6.1 COMPARISION OF STATIC ANALYSIS RESULTS FOR (ACTUAL AND REDESIED)

Thus far, Discussed the analyzed results of the failed and redesigned model of the rear axle casing individual. In this chapter we compare both the results to conclude whether the design is safe or not in the following sequence.

6.1.1 DISPLACEMENT:

Displacement is one of the significant factors while designing any engineering structures as like differential housing, In rear axle casing maximum load 5000N is acting on that its working in different condition. Here we compare the results for actual and modified design the results shows in that table 6.1. The calculated maximum displacement for the actual is (0.270mm) and for redesign is (0.453mm) according to design constraints shown in the table. the displacement doesn't exceeds obviously the design is safe while working in different condition.

Table 6.1 Comparison displacement results for both failed and redesigned Rear axle casing

PRESSURE (N/mm2)	DISPLACEMENT (mm)	
	ACTUAL	REDESIGNED
26	0.270	0.453

6.1.2 STRESS:

Due to stress concentration the failure happens as it found earlier analysis and how it to be resolve. Here comparing the stress value obtained after analysis for actual is (253.96 N/mm2), and for redesign is (158.40 N/mm2).it ensures that after redesign the failures are remove the result shows the design is safe according to design constraints. The results in below table 6.2

Table 6.2 Comparison stress results for both failed and redesigned rear Axle casing

PRESSURE (N/mm2)	STRESS (N/mm2)	
	ACTUAL	REDESIGNED
26	235.962	158.403

6.2 TIME STEPPING ANALYSIS

Time stepping analysis were carried out for both the models, analysis is carried out step by step by divided the load. This method adopts for both the models the comparison table as shown in below.

6.2.1 DISPLACEMENT RESULTS:

After analysis the results obtained for actual is 0.27mm, and then redesigned rear axle casing is 0.45mm. By comparing this results for displacement, the deflections are within the limit for applying load it satisfies the design constraints. Failure doesn't occur due to deflection. However a design constraint satisfies, each stepping load, deflection varies with each other for the same load. The comparison result and below table 6.3 and graph is shown fig 6.1.

6.2.2 STRESS RESULTS:

Stress is calculated for each step at divided load shown in table. The actual model results of the rear axle casing shows in table the deformation takes place at the load 20.8 N/mm2 the obtained stress value beyond the design constraints. But the redesign analysis result shows that the component withstand maximum load the obtained stress value doesn't exceed the design constraint.

Table 6.3 Comparison of time stepping displacement results for both failed and redesigned rear axle casing

STEP	PRESSURE (N/mm2)	DISPLACEMENT (mm)	
		ACTUAL	REDESIGNED
STEP 1	2.6	0.027	0.045
STEP 2	5.2	0.054	0.090
STEP 3	7.8	0.081	0.135
STEP 4	10.4	0.108	0.181
STEP 5	12	0.135	0.226
STEP 6	15.6	0.162	0.271
STEP 7	18.2	0.189	0.317
STEP 8	20.8	0.216	0.362
STEP 9	23.4	0.243	0.407
STEP 10	26	0.270	0.453

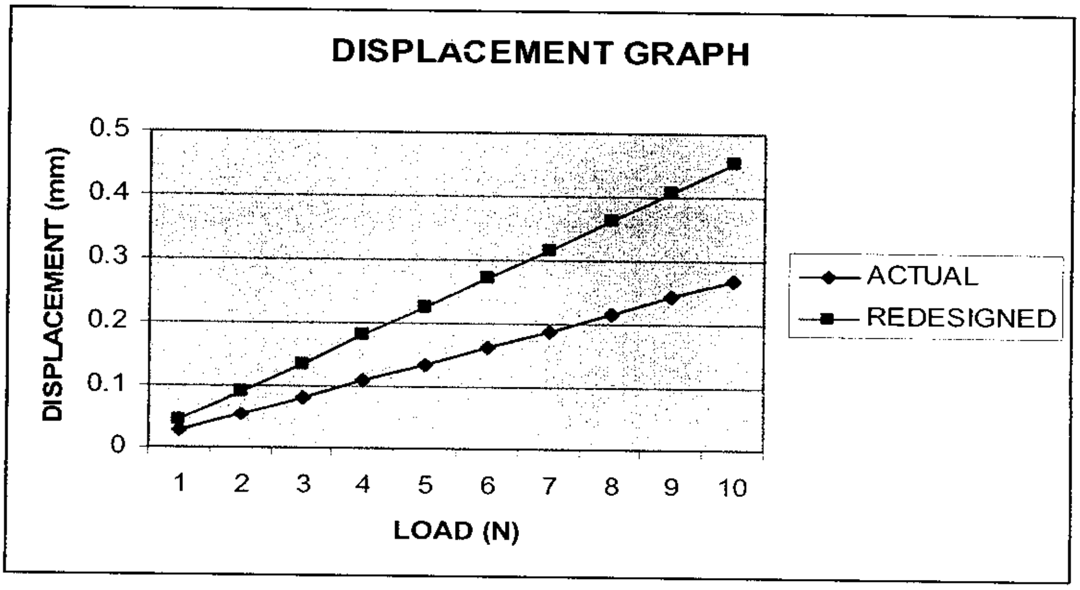

Fig 6.1 Displacement comparison graph for both models

Obviously the comparison results shown in table 6.4 ensure that the redesigned rear axle casing is under the safer limit. The plotted graph is shown in fig 6.2

Table 6.4 Comparison of time stepping stress results for both failed and Redesigned rear axle casing

STEP	PRESSURE (N/mm2)	STRESS (N/mm2)	
		ACTUAL	REDESIGNED
STEP 1	2.6	23.596	15.84
STEP 2	5.2	47.192	31.681
STEP 3	7.8	70.789	47.521
STEP 4	10.4	94.385	63.361
STEP 5	12	117.981	79.202
STEP 6	15.6	141.577	95.042
STEP 7	18.2	165.173	110.882
STEP 8	20.8	188.769	126.723
STEP 9	23.4	212.366	142.563
STEP 10	26	235.962	158.403

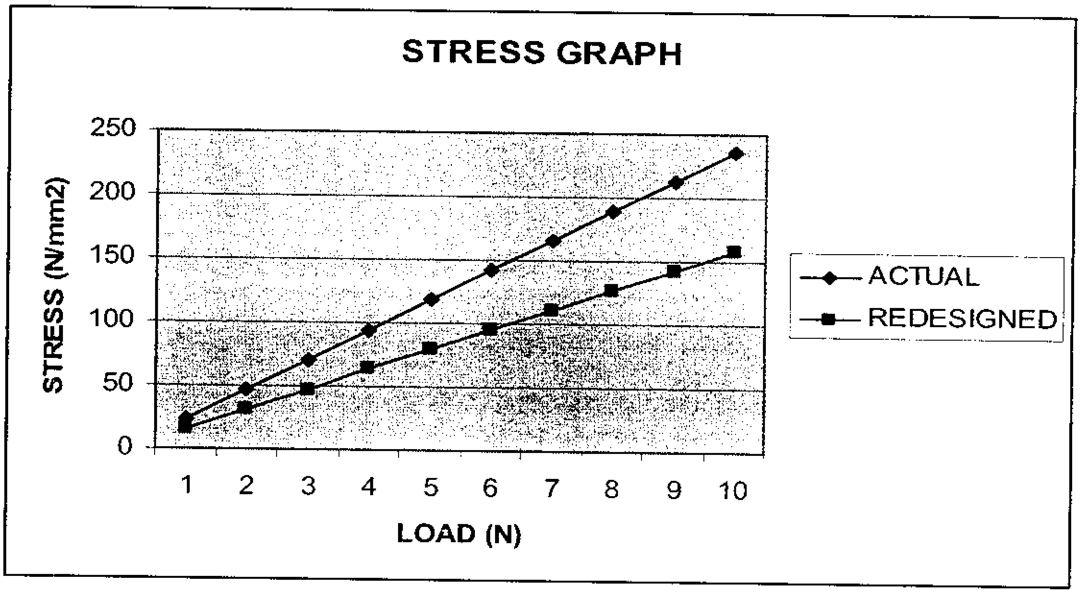

Fig 6.2 Stress comparison graph for both models

6.3 COMPARISION OF MODAL ANALYSIS RESULTS FOR (ACTUAL AND REDESIED)

So far, we discussed the comparison results for structural analysis of the failed and redesigned model of the rear axle casing. The natural frequencies and mode shapes are important parameters in the design of a structure for static or dynamic loading conditions But here the modal analysis results are comparing to ensure the designed model is in under the safer limit or not.

6.3.1 FREE-FREE RESULTS:

Natural frequency for the actual and the redesigned differential housing after modal analysis results shown in the below table 6.5 normally the free-free analysis is carried out to know the original natural frequency of the component. The result shows the max frequency for actual model is (6.09 HZ) and for the redesign the obtained frequency is (6.485 HZ). By comparing these results the deformation takes place if the calculated frequency beyond it but here both frequency are merely equal so the redesign is well and good.

Table 6.5 Comparison of modal free-free results for both failed and Redesigned rear axle casing

MODE	NATURAL FREQUENCY (HZ)	
	Actual	Redesigned
Mode 1	0	0
Mode 2	0	0
Mode 3	0	0
Mode 4	0	0
Mode 5	0	$0.198*10^{-5}$
Mode 6	$0.679*10^{-5}$	$0.615*10^{-5}$
Mode 7	6.099	6.485

6.3.2 FIXED END RESULTS:

In the real time application, the real axle casing is fixed in one end and the other is load carrying. After the free-free analysis by applying this boundary condition the modal analysis is carried out for fixed end. By this analysis easily identified the factor of safety. The obtained results describes that natural frequency for the actual model is beyond in the MODE3, it explains that natural frequency is exceed the deformation takes place after the MODE2. At the same time the frequency beyond for the redesigned model for maximum load in MODE3. By comparing these results we conclude that according to design constraints both are deform after the MODE2. So it ensures the factor of safety for both model is 2 so the design is safe. The results are shown in table 6.6

Table 6.6 Comparison of modal one end fixed results for both failed and Redesigned rear axle casing

MODE	NATURAL FREQUENCY (HZ)	
	Actual	Redesigned
Mode 1	3.879	4.369
Mode 2	4.615	5.214
Mode 3	8.843	10.269
Mode 4	14.192	14.302
Mode 5	15.466	15.462
Mode 6	16.475	15.848
Mode 7	17.066	16.718

By comparing and analyzing the results we conclude that design is safe.

Chapter 7

Conclusion

CHAPTER 7

CONCLUSION

In order to gain competitive advantage in global market it is necessary to produce a quality product with good safety. Today most of the accident occurs due to the failure of the component. it is one of the major Constraint .so to over come this problem the designer should considered the above point while designing the component in a safer limit.

Here rear axle casing is considered for design improvements. In this work the problem occurring in the RAC of the tractor was analyzed and the better solution for solving the problem has been discussed. The, problem currently faced in that RAC was the failure of the component before the warranty period. During analysis it has been found that the main cause for the failure of the RAC is due to the stress concentration at edges. Redesign has been done by including ribs in appropriate places in the component and again the analysis has been done.

After comparing the results with design constraints. This concludes the design is in safer limit the following benefits are acquired after redesign. Strength of the component increased, Component life is increased, Reduced service and product cost

References

REFERENCES

1. Antonio F. Barbosa and Gabriel O. Ribeiro, (1998) "Analysis of Reinforced concrete structures using ansys nonlinear concrete model", *computational mechanics new trends and applications Department of Structural Engineering*, Barcelona, pp 1-7.

2. Boada B.L., Gauchıa B.L., Boada,V. D´ıaz, (2007)"A genetic-based optimization of a bus structure as a design methodology", *12th IFToMM World Congress*, Besanc,on, June 18-21.

3. Corbin J. S. (1993), "Finite element analysis for Solder Ball Connect (SBC) *structural design optimization*", ibm j. Res. Develop. Vol. 31 no. 5, pp 585- 596.

4. David Johnson (2003), "Mechanical Systems Analysis" *American association of mechanical engineering Structural Report.*

5. He Liu (2004), "Effects of Nonlinear Geometric and Material Properties on the Seismic Response of Fluid/Tank Systems" *University of Alaska Anchorage.*

6. Kermanidis Th, Labeas. G, Tserpes K.I and Pantelakisi Sp, (2002), "Finite Element Modeling of Damage Accumulation in Bolted Composite Joints Under Incremental Tensile loading", *European Congress on Computational Methods in Applied Sciences and Engineering*, vol-21,pp. 1-13.

7. Lianyou Yu, Hengjun Zhu, Jiangtian Yang ((2007), "A Modeling Approach for the Lightweight Design of Railroad Truck", *12th IFToMM World Congress, Besançon (France)*, June18-21, pp 26-30.

8. Milan Vasek, (2002) "The non-linear problems of the steel civil engineering structures solved by ansys program", *proceedings of Czech Technical University, Prague*, pp 1-15.

9. Nick Cristello, and Il Yong Kim (2006), " Design optimization of an automotive universal joint considering manufacturing cost", *Proceedings of 17th International Conference on Modelling and Simulation*, pp499-506.

10. Robort matouser Munich(1995). "*Engineering design of a systematic approach* " Black and Son Limited, London

11. Schmid .P, Adamschik. M, Ertl. M, Gluche.V and Kohn. E, (2005) "Modeling Aproach for CVD-Diamond based mechanical structures" *Department of Electron Devices and Circuits, University of Ulm 89069 Ulm, Germany*, pp 28-30

12. Wilhelm Rust, "Finite element limit load analysis of thin-walled structures by Ansys (implicit), LS-DYNA (explicit) and in combination", *Journal of Mechanical Engineer*, pp. 3-9.

13. Sadhu singh (1988) "*Strength of materials*" khanna publication, delhi