

Modeling & FEA of Universal Coupling of an Automobile Truck

Purvash Shah

Department of Machine Design

S.S.G.B.C.O.E.T., Bhusawal, North Maharashtra University

Prof. A. V. Patil

Department of Machine Design

S.S.G.B.C.O.E.T., Bhusawal, North Maharashtra University

Abstract

Universal joint is defined as a mechanical device that can transmit a power or rotational motion from one shaft to another at fixed and varying angles of intersection of the shaft axis. Here a re-design of a Universal coupling which will allow power transmission between two misalign axis is carried out. As compare to existing design it has much better strength to withstand against given load. New Universal coupling designs will allow the engineer to transmit the motion of the driver shaft to drive shaft at misalign axis (more than one axis). 3-D model of the universal coupling is prepared by using CAD software for modal and stress analysis. A finite element based optimization is used to optimize the design of Universal coupling. The forces will apply to the modeled universal coupling in CAD software for analysis. These forces and moments are used as the boundary conditions for finite element model of the universal coupling.

Keywords: CAD software, Universal Coupling, ANSYS software

I. INTRODUCTION

The power produced from an engine of automobile can be transferred to the drive wheel by power transmission system. Each automobile has different power transmission system constructive features depend on the vehicle's driveline concept. (H.Bayrakceken et al., 2006) To transmit the driving torque from the engine or gear unit to the wheels, most of passenger car and light vehicle driven by combustion engine has at least two driveshaft as a basic requirement (Amborn, P. 1995).

During operation, torsional stress and bending stress was experienced by driveshaft due to the weight of the car or misalignment of journal bearing (Asi, 2006). In order to meet the requirements of one of the most highly stressed components in automotive assembly, a failure investigation must be conducted. Finite element method was used as stress analysis to determine the stress conditions at the failed section. Nearly all of driveshaft are metal shafts or metal tubes that has special joint at each end called universal joint (Birch and Rockwood2005). Power transmission system of vehicles consist several components which sometimes encounter unfortunate failures. Some common reasons for the failures may be manufacturing and design faults, maintenance faults, raw material faults, material processing faults as well as the user originated faults. In this study, fracture analysis of a universal joint yoke and a drive shaft of an automobile power transmission system are carried out. For the determination of stress conditions at the failed section, stress analysis is also carried out by the finite element method. The common failure types in automobiles and revealed that the failures in the transmission system elements cover 1/4 of all the automobile failures. Some common reasons for the failures may be manufacturing and design faults, maintenance faults, raw material faults as well as the user originated faults.

A. Basic Types of Universal Joints:

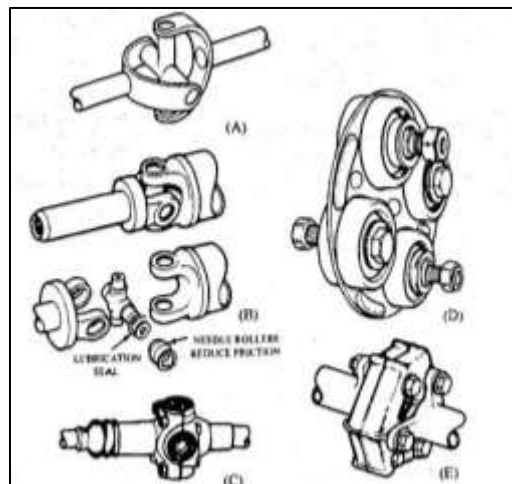


Fig. 1.1: Types of universal joints.

- 1) Hooked type joint
- 2) Cross type joint
- 3) Crossed type with rubber bushing
- 4) Layrub
- 5) Doughnut rubber coupling

II. LITERATURE REVIEW

A. History:

Many person have studied about the Universal Joint, some of them studied about the design, some of them studied about weight & stress optimization, some of studied about material property and some of studied about failure of the joint and their factors. This chapter will explain all the details of the research and reviews of Universal joint from the material, the benefits and also its functions. This information or research will be used as guidance for selection method of analyzing. Based on the fact and theory, the final method will be decided.

B. Review of Papers:

Siraj Mohammad Ali Sheikh [1] in his paper titled analysis of universal coupling under different torque condition has studied about the Drive shafts are one of the most important components in vehicles. It generally subjected to torsional Stress and bending stress due to weights of components. Thus, these rotating components are susceptible to fatigue by the nature of their operation. Common sign of driveshaft failure is vibration or shudder during operation. Driveshaft mainly involves in steering operation of vehicle. Drivers will lose control of their vehicle if the drive shafts broke during high speed cornering. Because of this human life can be in great danger if we don't know when, where and how the drive shaft will failed. It is very important to know the accurate prediction for the drive shaft to fail.

Swati N. Datey , S.D. Khamankar, Harshal C. Kuttarmare [2] studied in this paper,Finite Element analysis of rigid flange coupling is carried out with the help of ANSYS Software for different torque and load condition and it verify by manual calculation.In this preproject seminar report, analysis of rigid flange coupling is carried out which is similar to the universal joint. In this Finite Element Method analysis of rigid flange coupling with the help of ANSYS Software for different torque and load condition and it verify by manual calculation. In order to meet the requirements of one of the most highly stressed components in automotive assembly, a failure investigation must be conducted. Finite element method was used as stress analysis to determine the stress conditions at the failed section. Nearly all of driveshaft are metal shafts or metal tubes that has special joint at each end called universal joint.

Kamal Kashyap,D.G. Mahto [6] Analyze of Hooks Joint using Ansys by Von-Mises Method. The aim of this paper is to present a finite element analysis predicting the behavior of hooks joint under different loads on different parts. The software package ANSYS is used to model the joint. This paper may help to improve the quality of hooks joint. In this paper FEA analysis of Hooks joint is carried out and concluded that the total deformation that came out is maximum which take place at the center block of the entire assembly and the minimum deformation takes place at the Hub.

III. PROBLEM DEFINITION

Due to pin (fork) wear occurs, at the mating surface of the universal coupling it becomes noisy when rotate at high speed. In existing design cross section area is less, so that strength of the universal coupling is less.

A. Objectives:

- To design a universal coupling with the help of FEA so that it can be replaced existing design of universal coupling.
- To reduce wear of the material of in universal coupling.
- By applying new design we can reduce the intensity of noise produce during high speed rotation of the universal coupling.
- For improvement of the strength of the universal coupling with the help of shape change and applying the chamfer feature on the coupling joint.
- Smoothly operation during high speed rotation.

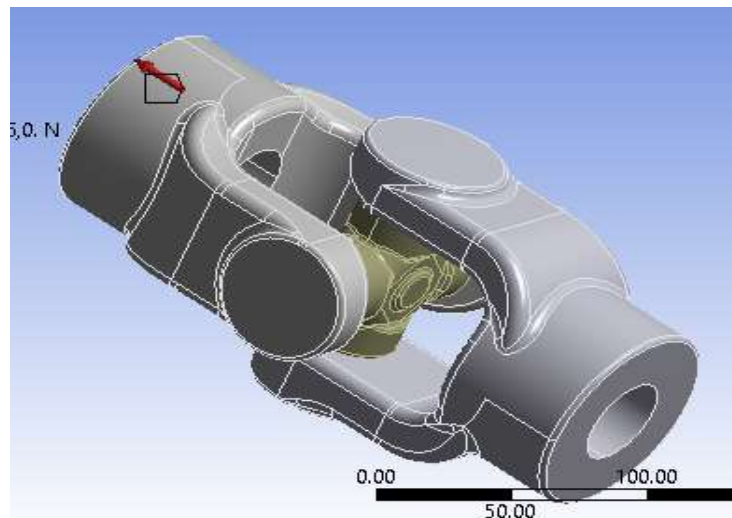


Fig. 3.1: Existing Design Assembly

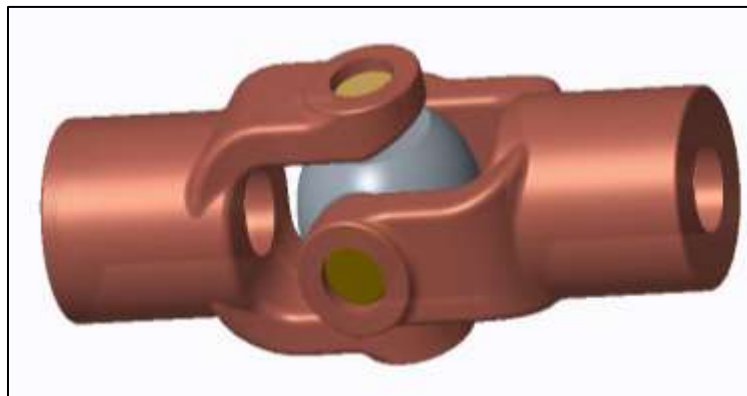


Fig. 3.2: New Design Assembly

IV. THEORETICAL ANALYSIS

A. Modeling of Existing Design:

- 1) The CAD models of existing design of universal coupling prepared in CREO software.
- 2) Load calculation and stress strain calculation of existing design is carried out with the help of manual calculation in EXCEL.
- 3) Stress analysis of each part of existing data has been done in Finite Element Method using ANSYS software.
- 4) New CAD model of universal coupling as per the load calculation.
- 5) Meshing of new CAD file in software.
- 6) Analysis of new model will carry out in Analysis software.
- 7) Induced stresses obtained from analytical calculations will be compared with the results obtained by manual calculation.

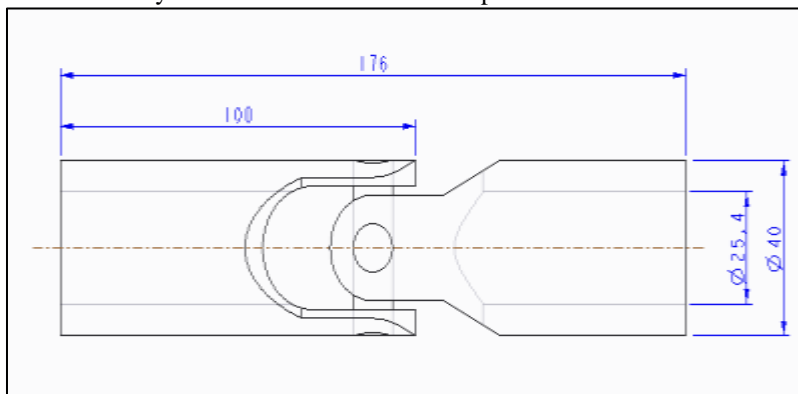


Fig. 4.1: Dimensions of Existing Design

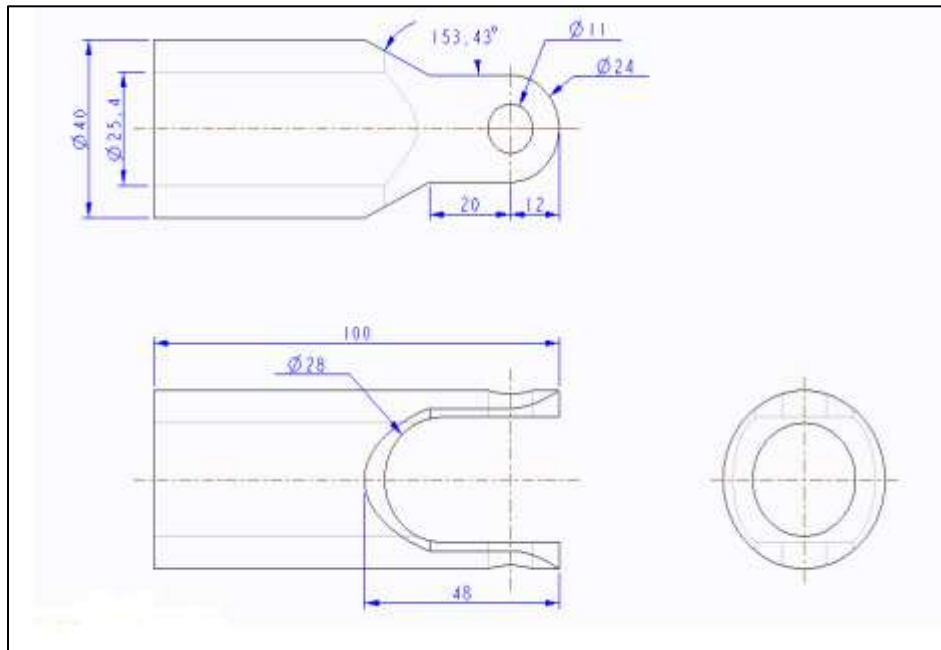


Fig. 4.2: Detailed Drawing of Yoke

B. Modeling of Proposed Design

CREO is a suite of design software supporting product design for discrete manufacturers and is developed by PTC. The suite consists of apps, each delivering a distinct set of capabilities for a user role within product development. CREO runs on Microsoft Windows and it provides apps for 2D design, 3D CAD parametric feature solid modeling, 3D direct modeling, Finite Element Analysis and simulation, schematic design, technical illustrations, and viewing and visualization. 2D drawing of proposed universal coupling is shown in figure 3. Modeling of hub, pin & assembly is done in CREO software as shown in figure

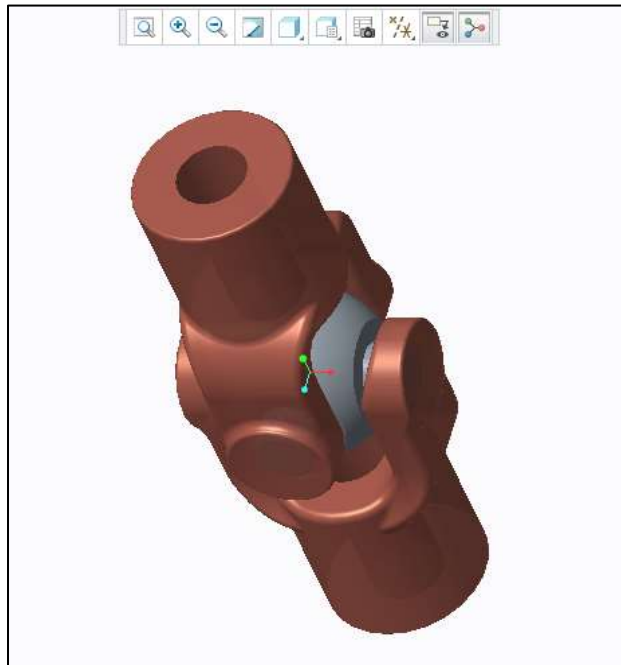


Fig. 4.3: Isometric Design of Proposed Coupling

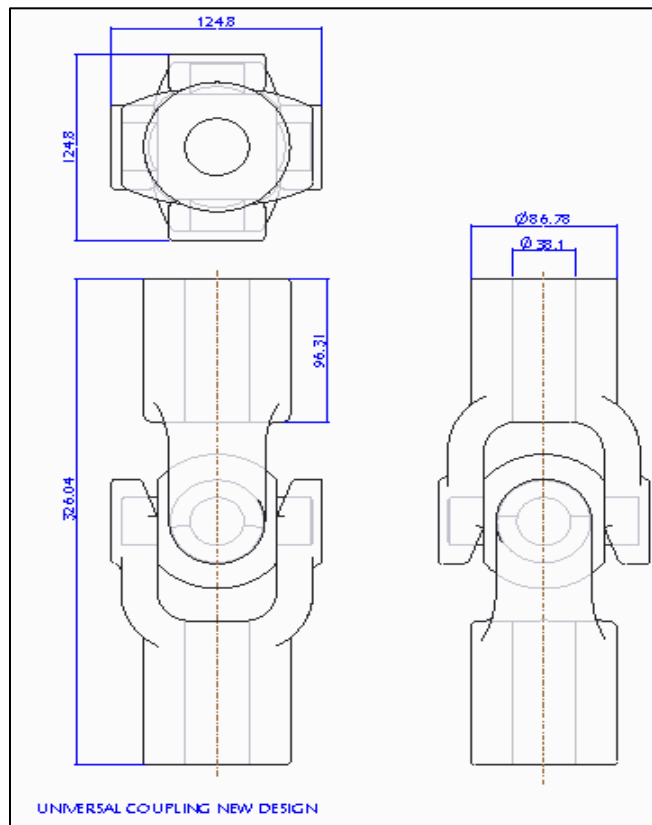


Fig. 4.4: Dimensions of an Assembly

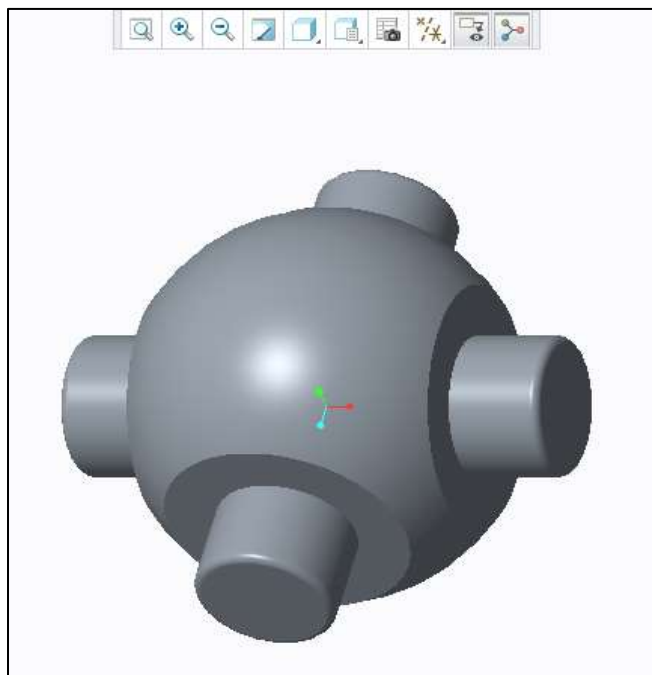


Fig. 4.5: New Ball Isometri.

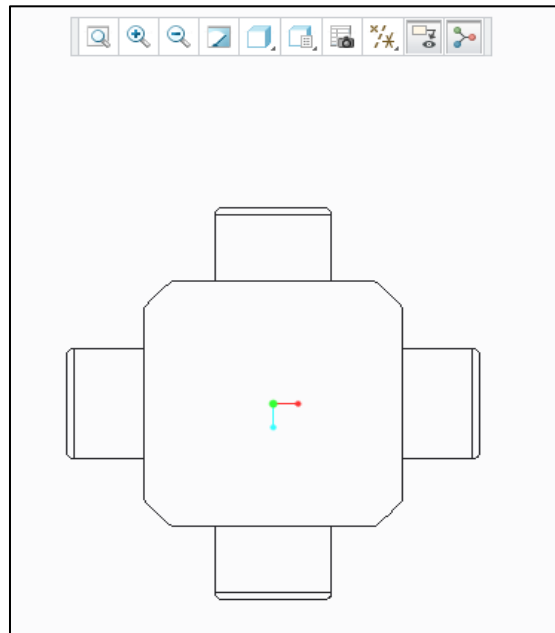


Fig. 4.6: New Ball Top View

C. Change in New Design As Compared To Existing Design:

- 1) In new design with the effect of increment in the area of universal Ball the stress can reduce.
- 2) The length of total assembly changed from 244.36 mm to 326.64 mm so connection to the rotating shaft is more reliable.
- 3) Design of hub remains same in new model.

Material use for manufacturing of universal coupling is EN-24 material.

<i>En-24</i>		
<i>Tensile Strength (ULTIMATE)</i>	850	MPA
<i>Modulus of Elasticity</i>	210	GPA
<i>Shear Modulus</i>	86	GPA
<i>Thermal Conductivity</i>	42	W/m-K
<i>Specific Heat Capacity</i>	0.46	J/g-°C
<i>Elongation</i>	13	%
<i>Tensile Strength, Yield</i>	635	MPA
<i>Density</i>	7.85	g/cc

V. FINITE ELEMENT ANALYSIS

A. Introduction to Analysis:

Here we come with the analytical software Ansys-14. ANSYS offers engineering simulation solution sets in engineering simulation that a design process requires. Companies in a wide variety of industries use ANSYS software with the help of this software we will do our remaining work.

B. Creating and Measuring the FE:

Modeling is based on a Conceptual understanding of the physical system and judgment of the anticipated behavior of the structure. A Model is an assembly of finite elements, which pieces of various sizes and shapes. A suitable mesh should minimize the occurrences of high aspect ratio and excessive skewers. In addition, the mesh must have enough elements to provide accurate results without warning time in processing and in interpreting the results.

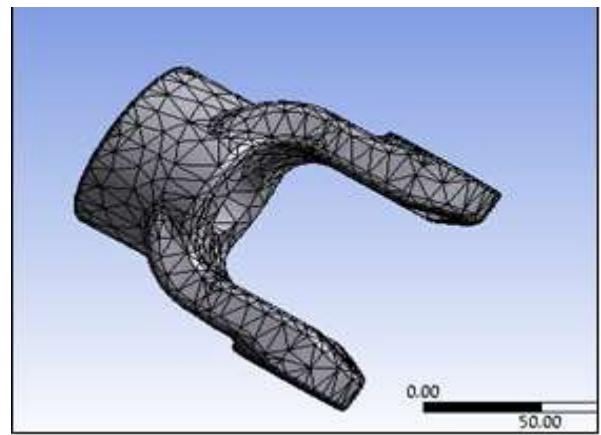
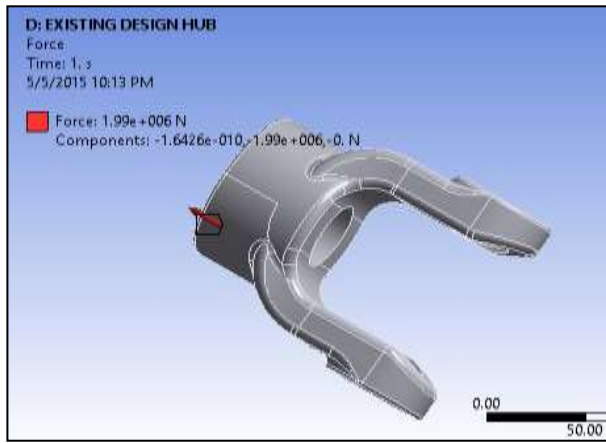


Fig. 5.1: Loading & Meshing of Hub for Existing Design

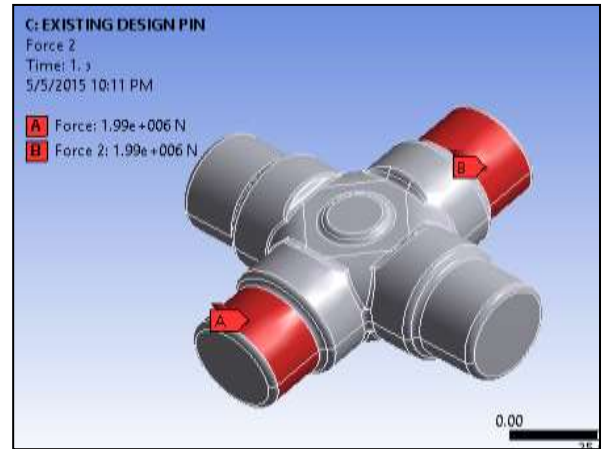
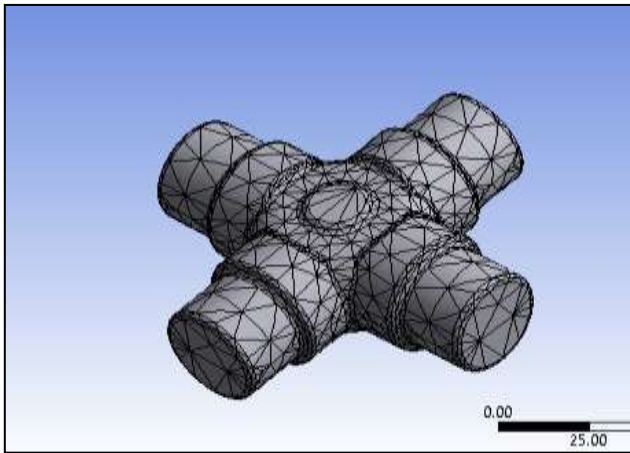


Fig. 5.2: Loading and measuring of pin for existing design

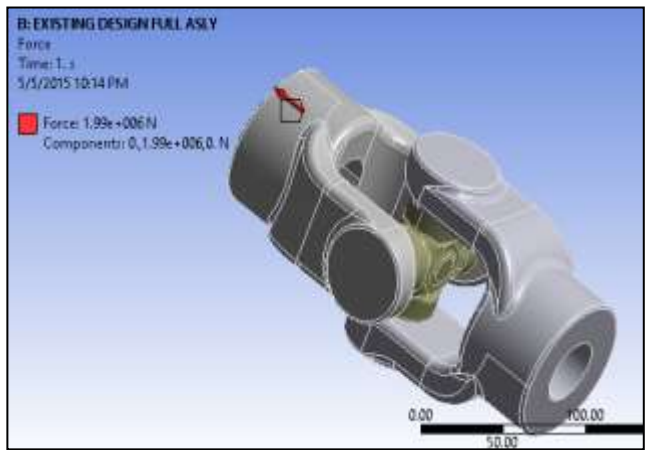
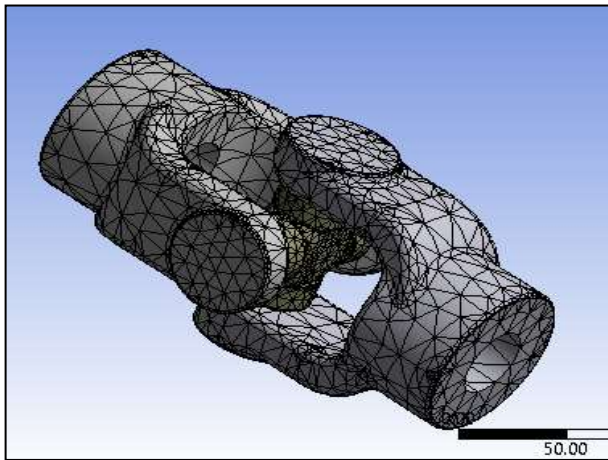


Fig. 5.3: Loading & Meshing of Assembly of Existing Design

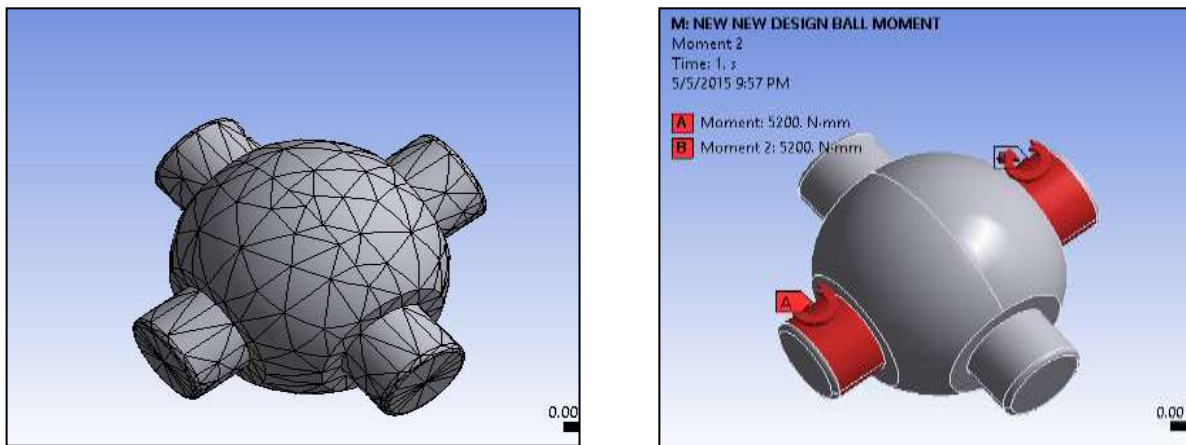


Fig. 5.4: Loading and measuring of pin for new desing

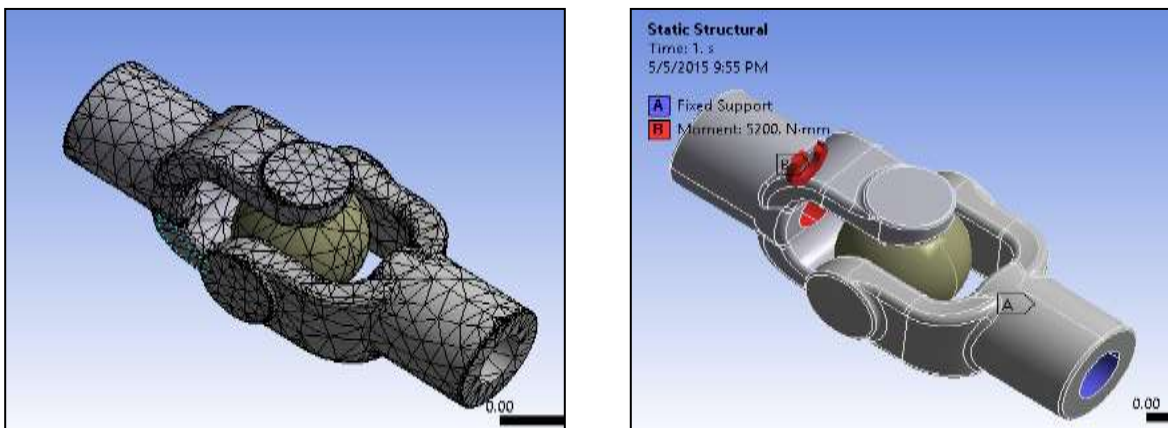


Fig. 5.5: loading and measuring of assembly of new design

C. Assumption in the Analysis:

- The actual loading in the Universal Joint was idealized using statically equivalent loads
- Perfect Welding was assumed between the structural components.
- The Geometry of the joint was simplified to accommodate the analysis within the limitations of the ANSYS.
- There may be some error due to linear Analysis.

Below figures show the results obtained by Given Boundary Condition and Loading Condition for case1 and case 2. Hear from above two case we can see that In the first case and the second case the stress result is almost same and the total deformation which we want to reduce is discussed, the total deformation in the first case is more and in the second case the deformation is less compare to the first one.

D. Existing design anylisis data

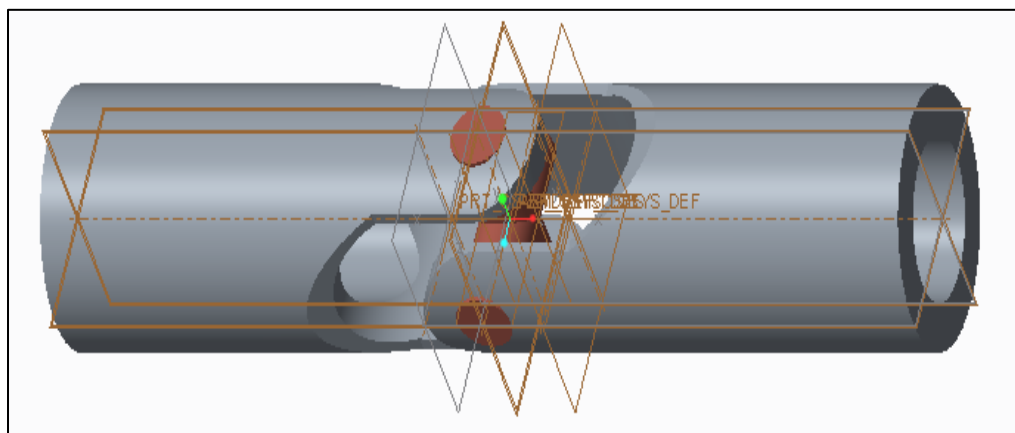


Fig. 5.6: Modeling of Existing Design

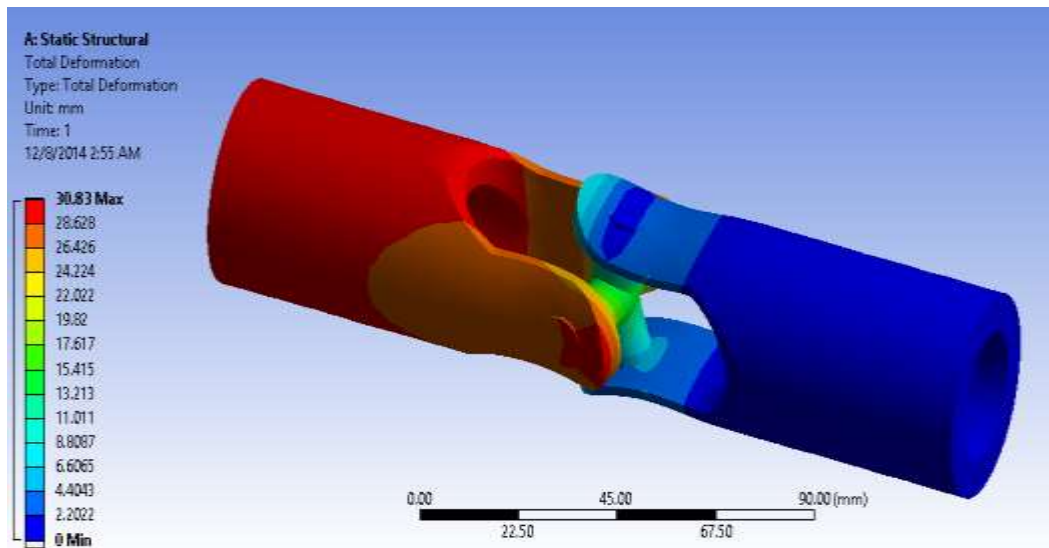


Fig. 5.7: Total Deformation of Existing Coupling

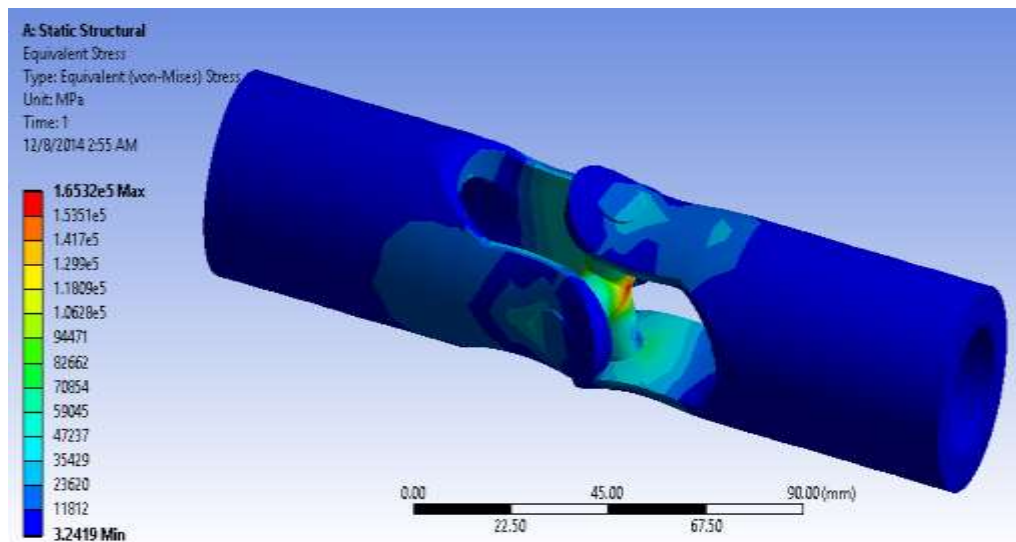


Fig. 5.7.1 Stress Analysis

E. New design analysis data

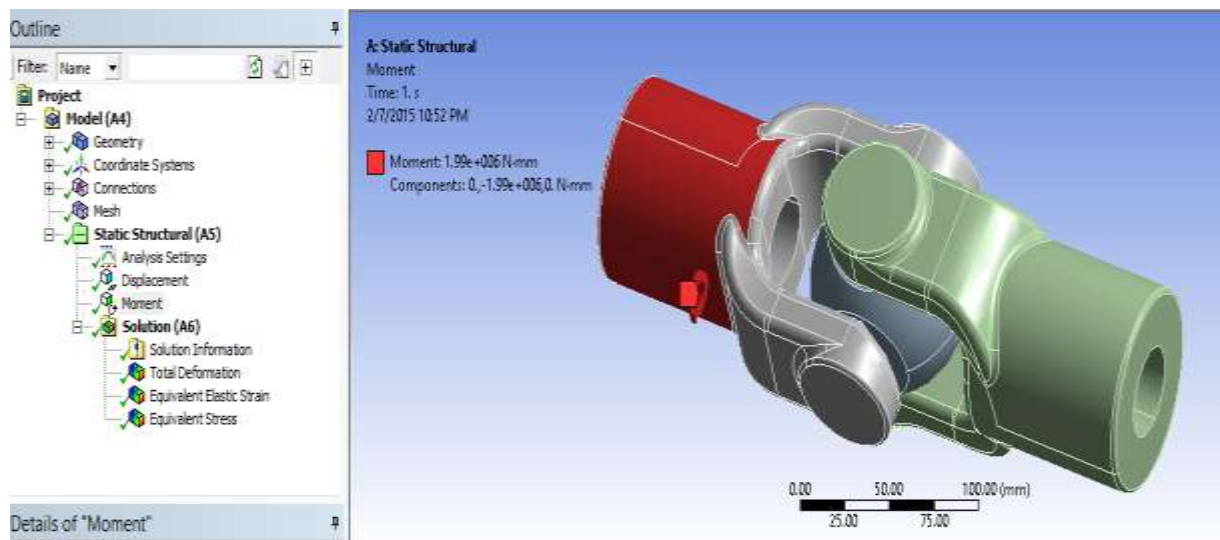


Fig. 5.8: Torque applied for Dynamic analysis

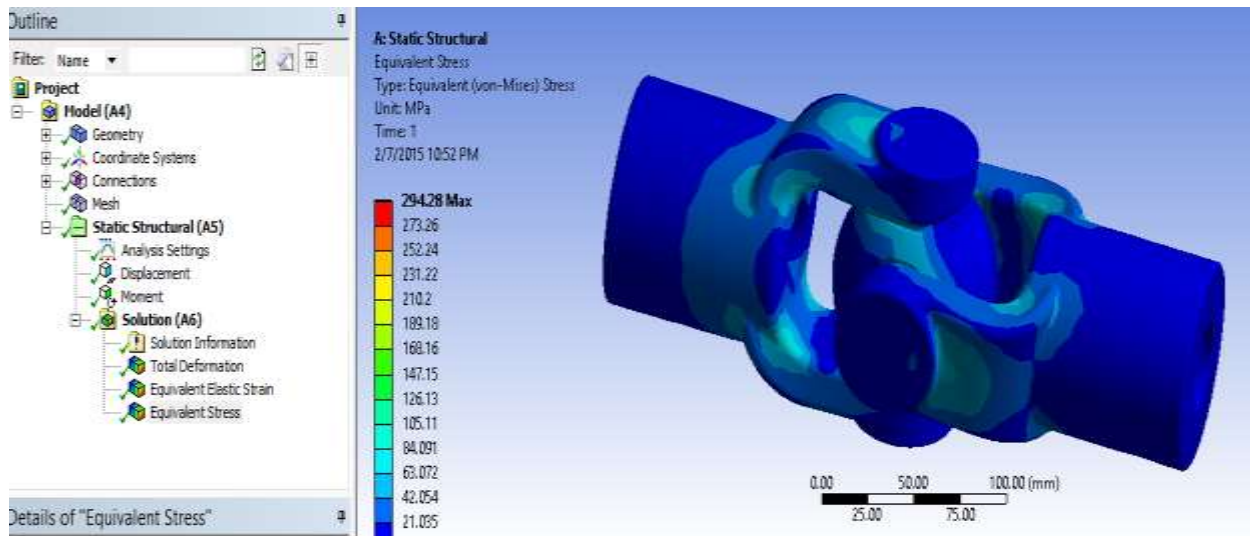


Fig. 5.9: Stress Analysis

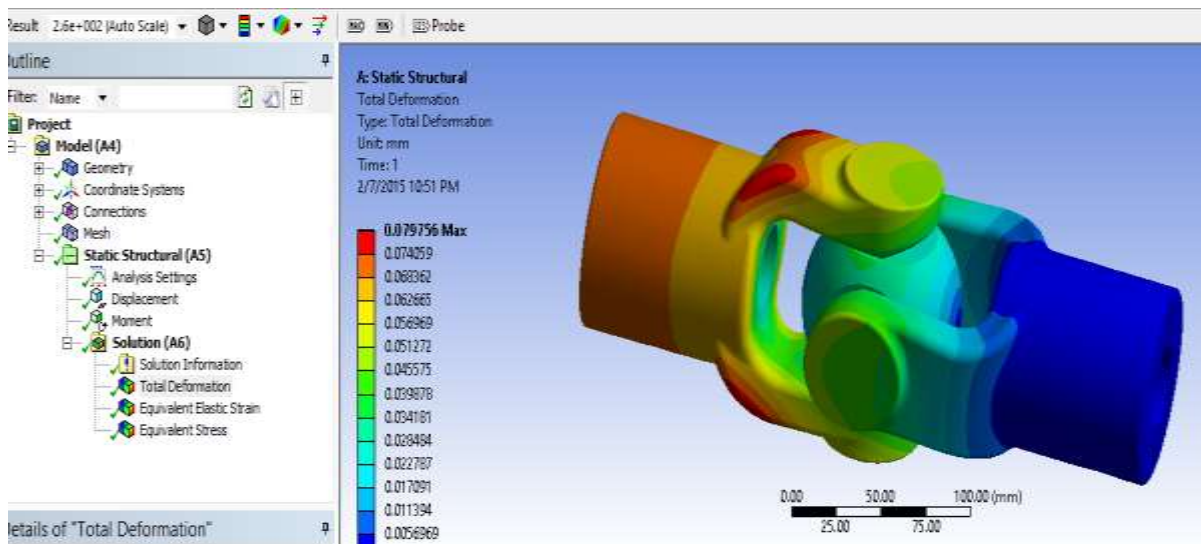


Fig. 5.10: Total Deformation in New Design

Table - 5.1
Surface Area and Weight Comparison

SURFACE AREA AND WEIGHT COMPARISON			
NEW VS OLD			
PARTICULARS	SURFACE AREA HUB x 2	SURFACE AREA BALL x 1	TOTAL WEIGHT
EXISTING DESIGN	12800	23000	9.05
NEW DESIGN	160078	401000	13.7
DIFFERENCE	147278	378000	4.65

VI. RESULT AND DISCUSSION

A. Analysis Summary:

The modeling of proposed design is done by using CREO software & static and dynamic analysis is done in ANSYS software. The comparison between Existing Design with Proposed Design is as below.

Table - 5.1

Comparison of Existing Design & Proposed Design - Analysis Data for Von Mises Stress

Comparison				
Existing Design And Proposed Design Analysis Data For Von Mises Stress				
Sr	Part / Assembly Name	Existing Design	Proposed Design	Difference
1	Full Assembly Stress	704.71	241.56	463.15

2	Hub Stress	46.19	42.67	3.52
3	Pin / Ball Stress	23101	5980.3	17120.7

Table - 5.2

Comparison of Existing Design & Proposed Design - Analysis Data for Shear Stress

Comparison				
Existing Design And Proposed Design Analysis Data For Shear Stress				
Sr	Part / Assembly Name	Existing Design	Proposed Design	Difference
1	Full Assembly Stress	351.3	120.04	231.26
2	Hub Stress	23.41	21.63	1.78
3	Pin / Ball Stress	11507	3530.8	7976.2

The comparison of manual data between existing designs with proposed design is as below.

Table - 5.3

Comparison of Existing Design & Proposed Design - Manual Data for Von mises Stress Comparison

Existing Design And New Design Manual Data For Von Mises Stress				
Sr	Part / Assembly Name	Existing Design	Proposed Design	Difference
1	Full Assembly Stress	608.46	207.93	400.53
2	Hub Stress	48.22	42.32	5.9
3	Pin / Ball Stress	19930	6115.52	13814.48

Table - 5.4

Comparison of Existing Design & Proposed Design - Manual Data for Shear Stress

Comparison				
Existing Design And New Design Manual Data For Shear Stress				
Sr	Part / Assembly Name	Existing Design	Proposed Design	Difference
1	Full Assembly Stress	304.23	103.96	200.27
2	Hub Stress	24.11	21.16	2.95
3	Pin / Ball Stress	9965	3057.76	6907.24

Table - 5.5

ANSYS Comparison

COMPARISON				
EXISTING DESIGN AND NEW DESIGN ANALYSIS DATA				
SR	PART / ASLY NAME	EXISTING DESIGN	NEW DESIGN	DIFFERENCE
1	FULL ASLY STRESS	704.71	294.28	410.43
2	HUB STRESS	46.19	45.74	0.45
3	PIN / BALL STRESS	23101	4517.7	18583.3

Table - 5.6

Mathematical Comparison

COMPARISON				
EXISTING DESIGN AND NEW DESIGN MANUAL DATA				
SR	PART / ASLY NAME	EXISTING DESIGN	NEW DESIGN	DIFFERENCE
1	FULL ASLY STRESS	645.06	480.53	164.53
2	HUB STRESS	65.82	63.74	2.08
3	PIN / BALL STRESS	24761.72	6260.74	18500.98

VII. CONCLUSION

Analysis of the Universal Joint clearly shows that by a small modification in the existing design the strength of the part can be increased significantly. Also with the same changes we obtain a small amount of weight reduction in the design. The maximum stress values generated are significantly reduced and the stress is evenly distributed over the entire part.

In this work design & finite element analysis of universal coupling is carried out. The failure of component is occur due to manufacturing and design fault, shear failure, improper assembly, raw material faults, maintenance faults, material processing faults, drivable joint angle, cyclic load, wear, noise etc. The main objective of this work is to reduce shear failure. The modeling of proposed design is done by using CREO software & static and dynamic analysis is done in ANSYS software. In existing design von mises stress and shear stress are 704.71 MPa & 351.3 MPa respectively. After the modification in pin's design von mises stress and shear stress are reduced to 241.46 MPa & 120.04 MPa respectively. By the comparison of both the result it is found that the von mises stress is reduced from 704.71MPa to 241.46MPa & shear stress is reduced from 351.3MPa to 120.04MPa. So shear failure is automatically reduced.

REFERENCES

- [1] Siraj MohammadAli Sheikh. "ANALYSIS OF UNIVERSAL COUPLING UNDER DIFFERENT TORQUE CONDITION". International Journal of Engineering science & Advanced Technology, Vol-2, Issue-3, 690-694
- [2] Swati N. Datey, S.D. Khamankar, Harshal C. Kuttarmare, "Finite Element Analysis of Universal Joint". IOSR Journal of Mechanical and Civil Engineering, Vol-11, Issue-3, PP 64-69

- [3] Farzad Vesali, Mohammad Ali Rezvani and Mohammad Kashfi, "Dynamics of universal joints, its failures and some propositions for practically improving its performance and life expectancy". *Journal of Mechanical Science and Technology*, 26 (8) (2012) 2439-2449
- [4] S G Solanke and A S Bharule, "AN INVESTIGATION ON STRESS DISTRIBUTION FOR OPTIMIZATION OF YOKE IN UNIVERSAL JOINT UNDER VARIABLE TORQUE CONDITION: A REVIEW". *International Journal of Mechanical Engineering and Robotics Research*, Vol-3, Issue-2, PP 136-142
- [5] Mr. Anuj A. Muley, Dr. M. J. Sheikh, "Failure Analysis of Yoke Joint Assembly of Transmission Drive Shaft: A Review". *International Journal for Research in Applied Science & Engineering Technology*, Vol-3, Issue-XII, PP 46-49
- [6] Kamal Kashyap, D.G.Mahto, "Analysis of Hooks Joint Using Ansys by Von-Mises Method", *International Journal of Engineering and Advanced Technology*, Vol-3, Issue-3, PP 304-305.
- [7] Naik Shashank Giridhar, Sneha Hetawal, Baskar P. , " Finite Element Analysis of Universal Joint and Propeller Shaft Assembly" , *International Journal of Engineering Trends and Technology*, Vol-5, Issue-5, PP 226-229
- [8] Saurav Rajgadia, Debayan Das, Pawan Jaiswal, Ankit Basnet, Anupam Raj Jha, Rakesh Jaiswal, Anush Karki, Rabindra Nath Barman," Design and Stress-Analysis of a Rigid Flange Coupling using FEM" , *International Journal of Innovative Research in Science, Engineering and Technology*, Vol-4, Issue-10, PP 9599-9609
- [9] Prof. Salunkhe R.T., Mr. Patil N.T., Mr. Lokhande V.S., Mr. Bandelkar D.M., Mr. Patil R.D., "Design, Modelling, Analysis of Propeller Shaft and Couplings", *International Journal of Advanced Scientific and Technical Research*, Vol-5, Issue-3, PP 580-590
- [10] Avinash C Vasekar, Ranjitsinha R. Gidde, "FAILURE ANALYSIS AND OPTIMIZATION OF UNIVERSAL JOINT YOKE SUBJECTED BY TORSION AND SHEAR", *International Research Journal of Engineering and Technology*, Vol-2, Issue-7, PP 229-235