CAD TOOLS AND TECHNIQUE FOR UNIVERSAL JOINT

[1]Dhiraj V. Astonkar, [2]Dr. Shrikant S. Dandge, [3]Dr. Sanjay M. Kherde

Assistant Professor, Department of Mechanical Engineering, Dr. Sau. Kamaltai Gawai Institute of Engineering & Technology, Darapur, Dist. Amravati[1], Associate Professor, Department of Mechanical Engineering, Dr. Rajendra Gode Institute of Technology & Research, Amravati[2] , Principal-Professor, Department of Mechanical Engineering, Sipna College of Engineering & Technology, Amravati[3],

dhirajastonkar@gmail.com[1], shreesdandge@rediffmail.com ,sanjaykherde@gmail.com[3]

ABSTRACT

Vehicle transmission framework comprises a few parts which in some cases experience the ill effects of various burdens (disappointments). An all-inclusive joint comprise of two yokes, one on each pole, associated by cross-molded middle of the road part i.e. creepy crawly. Yoke get together are constantly subjected to torsion and shear. Era of stress, removal and strain in a general coupling has been broke down. Yoke get together are turning part and here and there experience the ill effects of weakness by use of variable torque. In this paper limited component investigation of the part is done to discover the stress and deformation of the last item. For demonstrating of the segment CAD software is utilized. Pre- handling work like cross section and examination work is done in ANSYS CAE software. Utilizing FEA investigation, we can recognize the nature and qualities of stresses following up on the Yoke and evaluate the impact of the load/mass geometry/boundary limit conditions over the yoke.

Keywords: CAD, CAE, Universal Joint.

1. INTRODUCTION

Recent trends in automobile development activities for reduction of lead-time and cost have led to a current situation where CAE (Computer Aided Engineering) techniques are fully used to skip conventional development steps for making and checking costly prototypes. Many automakers now use a computer simulation instead of preparing costly prototypes to analyze the strength and the collision resistance of a vehicle body. Recent use of computer simulation has been further expanded for a dummy model or vehicle interior accessories which are used for analyzing what and how much impact may occur to passengers. Some automakers are trying to use a so-called digital prototyping, where all design steps for a prototype are performed through computing operation. With such a trend for digital developments by automakers, vehicle component makers including KOYO, who are responsible for the development and mass-production of steering column products. (e.g.,a safety steering wheel and an electric power steering), must keep up with the trend by further improving their CAE analysis techniques for preproduction steps to reduce the number of redundant steps from prototyping to experiment evaluation and to provide drawings with higher accuracy. The current CAE analysis by Koyo includes four major functions of a vehicle (i.e., strength, noise/Vibration, vehicle motion, and collision), along with collision of a steering column assembly (hereinafter referred to as assembly) will be focused in this paper. Specially, this paper will use a collision model of steering column assembly to examine the consistency between the result of the CAE analysis model and the result of actual collision test of an actual assembly.

2. NEED FOR THE STUDY:

Recent trends in automobile development activities for reduction of lead time and cost have lead to a current situation where CAE (Computer Aided Engineering) techniques are fully used to skip conventional development steps for making and checking costly prototypes. The Steering System used predominantly in passenger cars today is the Rack and pinion type. A virtual prototyping approach by using a one degree haptic system makes it possible for the customer to test the virtual prototype of the steering unit in a direct and natural way, in early design phase. A comparison of CAE analysis results and Testing results for the Steering Column

Assembly and characteristics of the steering system can be evaluated properly using HIL A number of Analysis has been performed on virtual prototype of Steering column Assembly. But Static Rack Bending Analysis of Steering column Assembly has not been studied yet. Steering Rack is designed to sustain bending loads during vehicle running. The loads come from tire side and produce the bending loads on Steering Rack. Steering Rack Static Bending Analysis will be focused in this paper. This project is an attempt to design a Rack and Pinion with specifications minimizing swing torque, in ADAMS (Automatic Dynamic Analysis of mechanical Systems). This model helped to identify critical parameter which affects steering column. A number of Analyses has been performed on virtual prototype of Steering case Assembly. The loads come from tire side and produce the bending loads on Steering Rack. Steering Rack Static Bending Analysis will be focused in this paper.

3. CONCEPT OF CAD MODELING:

The stacking states of torsional minute and the rotational speed are kept for the outcome. The model of universal joint was investigated in ANSYS viewing the segment as comprised of Materials.

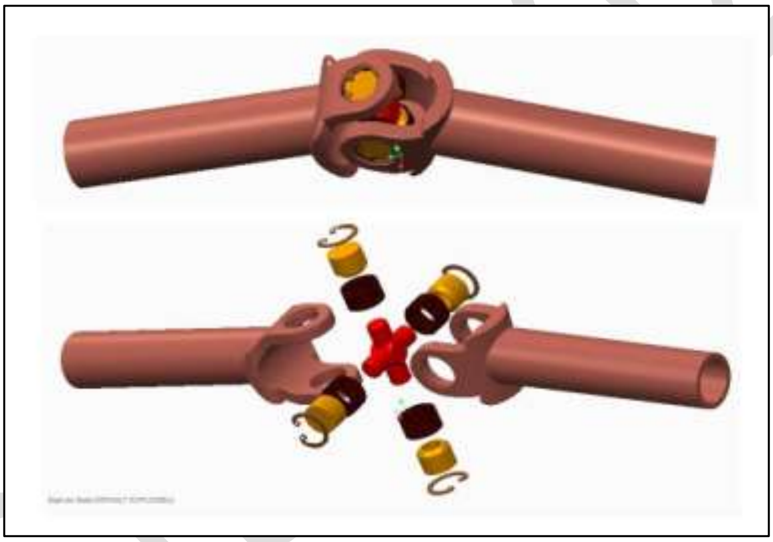

Fig. No. 01: CAD Assembly of Universal Joint

4. A GENERAL PROCEDURE FOR ANALYSIS TECHNIQUES:

Certain steps in formulating a finite element analysis of a physical problem are common to all such analysis, whether structural, heat transfer, fluid flow, or some other problem. These steps are embodied in commercial finite element software packages (some are mentioned in the following paragraphs) and are implicitly incorporated in this text, although we do not necessarily refer to the steps explicitly. The steps are described as follows.

A. Preprocessing :

The preprocessing step is, quite generally, described as defining the model and includes

- Define the geometric domain of the problem.
- Define the element type(s) to be used.
- Define the material properties of the elements.
- Define the geometric properties of the elements (length, area, and the like).
- Define the element connectivity (mesh the model).
- Define the physical constraints (boundary conditions).

- Define the loadings.

The preprocessing (model definition) step is critical. In no case is there a better example of the computer-related axiom "garbage in, garbage out." A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem.

B. Solution:

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow. As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a wave front solver, based on Gauss elimination, is commonly used. While a complete discussion of the various algorithms is beyond the scope of this text, the interested reader will find a thorough discussion in the Bathe book.

C. Post processing:

Analysis and evaluation of the solution results is referred to as post processing. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution.

Examples of operations that can be accomplished include:

· Sort element stresses in order of magnitude.

· Check equilibrium.

· Calculate factors of safety.

· Plot deformed structural shape.

· Animate dynamic model behavior.

· Produce color-coded temperature plots.

While solution data can be manipulated many ways in post processing, the most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable.

5. FEA SOFTWARE PACKAGES AND APPLICATIONS:

The rapid advance made in computer hardware and software led to significant developments in FEA software. FE programming gas emerged as a specialized discipline which requires knowledge and experience in the diverse areas such as FE technology including foundations of machines, and numerical analysis on the one hand and the computational skills in areas of software technology including programming techniques, data structure, data base management and computer graphics on the other hand. It requires several man years to develop general purpose finite element analysis software with a processing capability and facility for the user to have a wide choice of several types of elements, analysis for different types of problem-static, dynamic, material and geometric nonlinear, coupled situations, hear transfer, interaction problems etc. and pre and post-processing features. In 1975 the FEA software package was used for the analysis of aero plane parts. From the analysis it was predicted that the ansys packages gives better solution as compare to defining theory models. Thus the packages cover the industrial sector for analysis of various structural and mechanical parts. The main purpose of such packages is to obtain the solution without creating physical model. The figure shows memory allocation by ANSYS software package. The analysis software packages are applicable to static linear analysis, non-linear analysis, modal analysis, fluid flow analysis, thermal analysis, harmonic response analysis, steady state analysis, etc. As it is not possible here to review the capabilities and compare different commercially available finite element analysis packages only the names of some of the popular packages are given below: ABAQUS, ADINA, ANSYS, ALGOR, ASKA, COSMOS, GT-STRUDL, LISA, NISA, ASTRAN, PAFEC,PASTRAN,SAP.

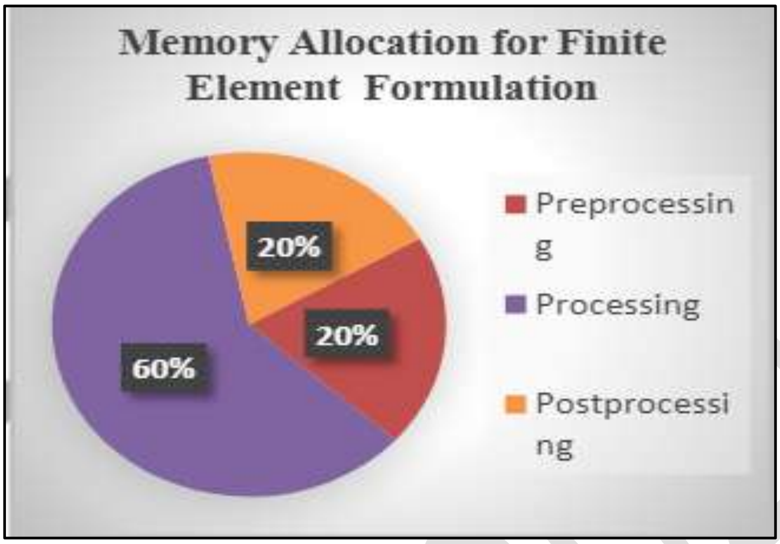

Fig. No. 02: Memory Allocations for Analysis Software Package

6. ANALYSIS TECHNIQUES ON ANSYS SOFTWARE:

Consider Twisting Moment for Steering Turn (1kgfm = 9.806N-m)

1) 50 kgfm for HMV (Truck) is converted to 490.33N-m.

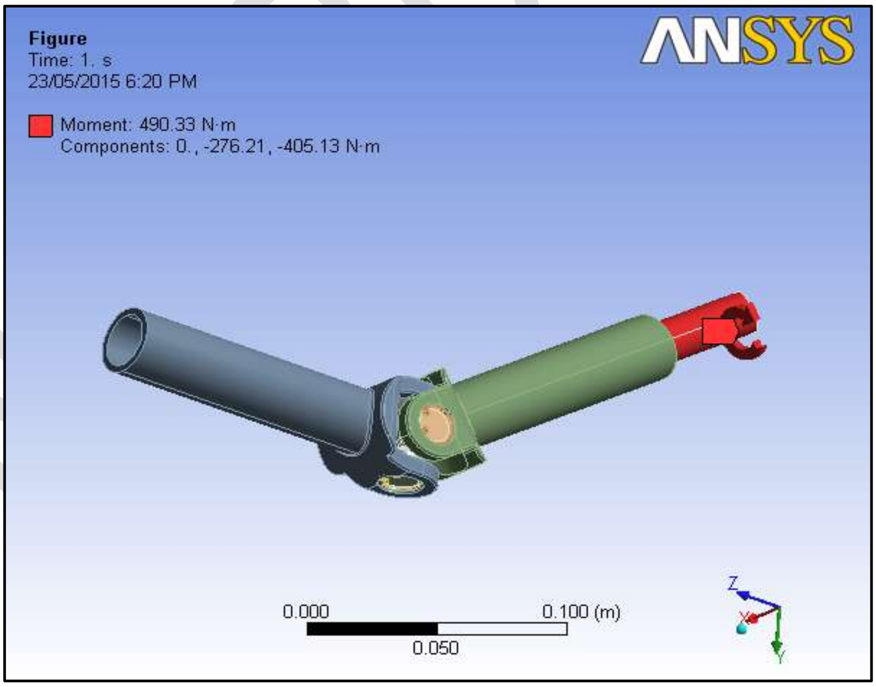

Fig. No. 03: Twisting Moment load applied on HMV

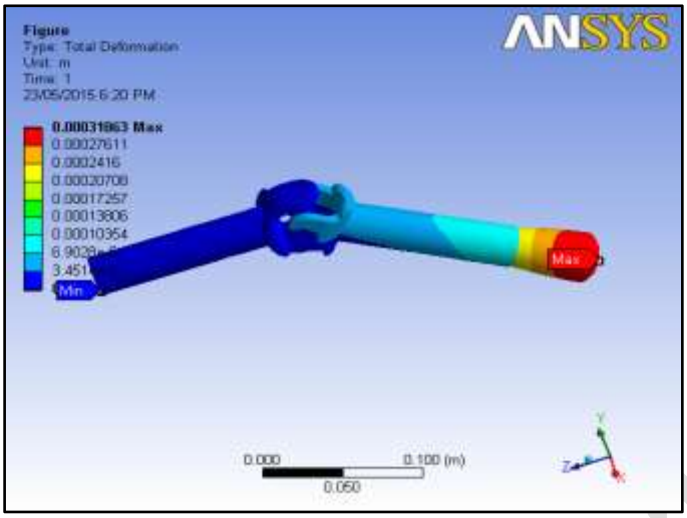

Fig. No. 04: Total Deformation result on HMV

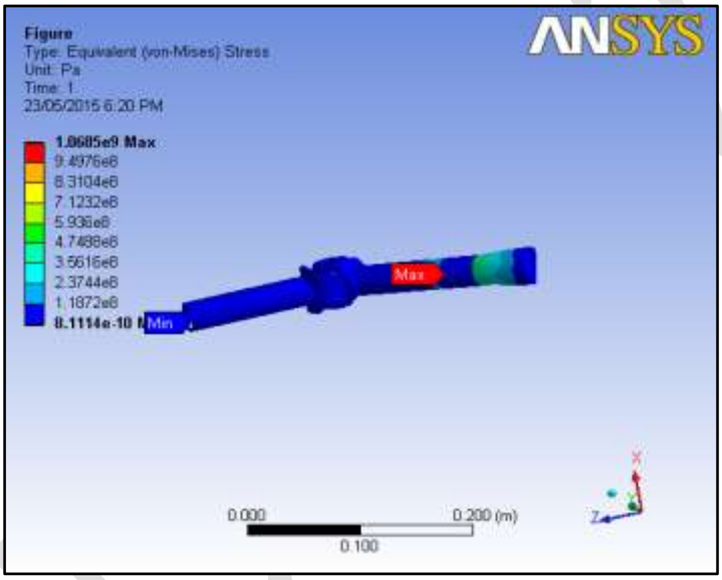

Fig. No. 05: Equivalent Stress result on HMV

2) 10 kgfm for LMV (Car) is converted to 98.06N-m.

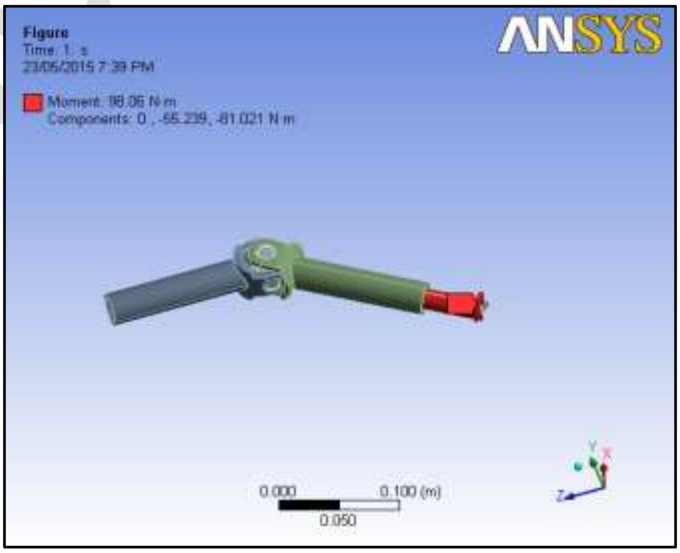

Fig. No. 06: Twisting Moment load applied on LMV

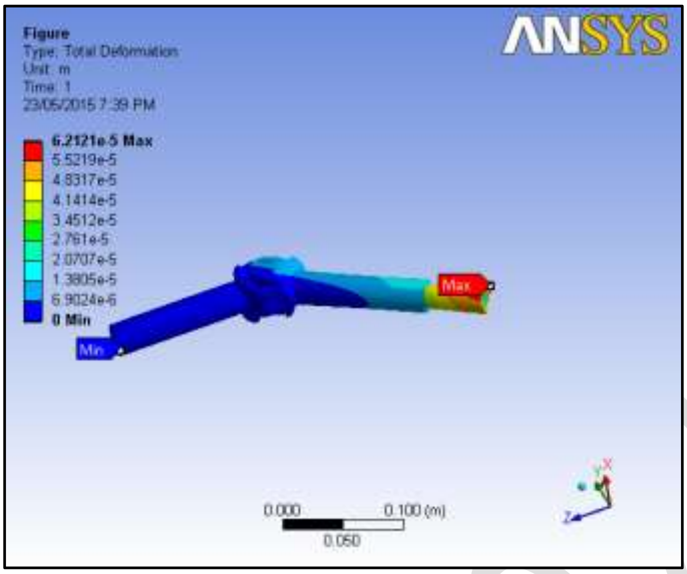

Fig. No. 07: Total Deformation result on LMV

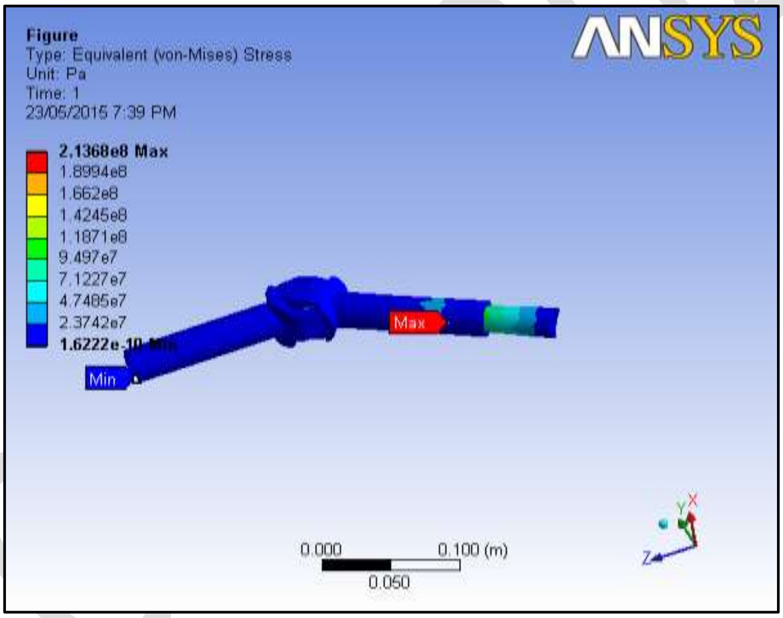

Fig. No. 08: Equivalent Stress result on LMV

Table No. 01: Comparison of Analysis Techniques Result with Different Vehicle Type

Vehicle Type	Twisting Moment	Total Deformation	Equivalent Stress (von-Mises)
HMV (Truck)	490.33 N-m	$3.1063e^{-4}$	$1.0685e^{9}$
LMV (Car)	98.06 N-m	$6.2121e^{-5}$	$2.1368e^{8}$

CONCLUSION

The outcomes were gotten are very positive which was normal. This outcome center the connection between the assembling expense and joint edge execution measures of a car widespread joint. From the outcomes acquired from FE Analysis, numerous dialogs have been made. The outcomes acquired are well in concurrence with the accessible existing outcomes. The model exhibited here, is well sheltered and under reasonable cutoff of stresses. It can be noticed that disappointment of segment is happen due assembling and configuration blame, crude material shortcomings, looks after issues, material preparing deficiencies, to evade this issues different methods, for example, either, change material or geometry and direct examination on the same lastly actualized in the arrangement or used to discover the best outline of Joint with considering the all the component, for example, weight, cost, Fatigue life, push dissemination, solidness, and so forth. The mechanical stress is discovered most extreme close to the sharp edges.

REFERENCES

1. Rahul Arora," Modeling and Failure Analysis of Universal Joint using ANSYS", on IJETER, Volume 5, Issue 8 Aug. 2017, pp 53-56.
2. Vishal Rathi and Nitin K. Mandavgade (2009) "Fem analysis of universal joint of Tata 407" Second International Conference on Emerging Trends in Engineering and Technology, ICETET-09, ISBN: 978- 0-7695-3884-6, PP. 98-103.
3. Nick Cristello, and Yong Kim (2006) "Design optimization of an automotive universal joint Considering manufacturing cost", 17th IASTED International Conference Modeling and Simulation, ISSN: 530-091 , PP. 499–504.
4. S.K.Chandole, M.D.Shende, M.K. Bhavsar" Structural Analysis Of Steering Yoke Of An Automobile For Withstanding Torsion/ Shear Loads",IJRET: International Journal of Research in Engineering and TechnologyVolume: 03 Issue: 03 | Mar-2014.
5. H. Bayrakceken, S. Tasgetiren, I. Yavuz (2007) "Two cases of failure in the power transmission system on vehicles: A universal joint yoke and a drive shaft", Engineering Failure Analysis, Vol. 14, ISSN: 1350- 6307, PP. 716- 724.
6. Majid Yaghoubi, Ali Jafary, Seyed Saeid Mohtasebi (2010) "Design, simulation and evaluation of a new universal joint with intersecting angle up to 100 degrees for farm machineries", AJAE 1(4), ISSN:1836- 9448, PP: 149-152.
7. Muhammad Ziaur Rahman, Debasish Adhikary & Tazmin Rashid Mumu (2013) "Failure Analysis of a Universal Coupling using Finite Element Method
8. P.G. Tathe, Prof. D.S. Bajaj and Swapnil S. Kulkarni, "Failure Analysis And Optimization In Yoke Assembly Subjected By Torsion And Shear", International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974. JulySept,2014.
9. Naik Shashank Giridhar, Sneh Hetawal and Baskar P.,"Finite Element Analysis of Universal Joint and Propeller Shaft Assembly", International Journal of Engineering Trends and Technology (IJETT) - Nov 2013 – Volume Number 5.
10. Ashish Bharatrao.Nitalikar, R.D.Kulkarni, Swapnil.S.Kulkarni, "Structural Analysis for a Cardon Joint in steering column assembly through FEA Techniques". Asst.Prof.TPCT college of Engineering,Osmanabad,Director-Able Technologies India Pvt.Ltd,Pune.IJAERS.E-ISSN2249-8974
11. "Model for localized failure with thermo-plastic coupling: Theoretical formulation and ED-FEM implementation" Van-Minh Ngo, Adnan Ibrahimbegovic, Delphine Brancherie, Ecole Normale Supérieure, LMT-Cachan, 61 Avenue du, Président Wilson, 94235 Cachan, France b Université de Technologie de Compiègne, Laboratoire Roberval, 60205 Compiègne, France article info Article history: Received 18 October 2012 Accepted 7 December 2012 Available online 20 March 2013.
12. Tae Hee, Lee Byung Ryul, Ham Seong Oh ,"Study on Steering Column Collapse Analysis using detailed FE Model", Hong Advanced Safety CAE Team / Hyundai Motor Co. Korea Paper No. 11-0262.

13. Dr. Satypal T. Warghat, Dhiraj V. Astonkar, Manish P. Bijwe,"Analysis of differential crown gear and pinion for axle with different materials, on IJSDR, Volume 4 (4), ISSN: 2455-2631, April 2019, pp 116-125