Prediction of Universal Joint Behavior Under Small Axial Loading Using FEM

Rahul Sharma*, Sumit Kumar Shaw, S.Somesh, Mukul Bhatia, Joginder Singh and Jai Prakash Sharma

Department of Mechanical Engineering, Manav Rachna University, Faridabad, Haryana, India; rswvt2898@gmail.com, sumit.sct65@gmail.com, somesh.01.03@gmail.com, imukulbhatia23@gmail.com, joginder@mru.edu.in, jpsharma@mru.edu.in

Abstract

Objectives: The present investigation aims to design a structural steel universal joint and analyze its behavior under coaxial compressive loading. **Methods/Statistical Analysis:** In the present investigation, CATIA V5 was used to design the Universal joint of a steering column, and its finite element analysis was carried out using ANSYS software. Structural steel and its properties were used for design as well as analysis purpose. In ANSYS, static structural analysis was carried out to determine the von-Mises stress, von-Mises strain and total structural deformation. **Findings:** The steering column is an integral section of an automobile for its stability and steady movement. A universal joint is typically employed to transmit the rotational energy of a steering column to a shaft inclined at a particular angle. In universal joints, two forked shaft are connected using a cross-shaped intermediate member. This section is very susceptible to shear deformation owing to torsion, and fatigue failure may occur due to its cyclic nature. After evaluation of nature and stress characteristics, it can be concluded that the universal joint made up of structural steel is safe below and at the applied load. **Application/ Improvements:** The collapsible steering column is an effective mechanism for safe and comfortable driving experience, especially in the all-terrain vehicle. For this purpose, the steering column components should be able to bear sudden compressive and tensile forces.

Keywords: Finite Element Analysis, Universal Joint, ANSYS, CATIA

1. Introduction

The moving direction of the vehicle is controlled by the steering wheel. The directional accuracy and manoeuvre capability of an automobile are mainly reliant on how accurately steering response is communicated to wheels. Regardless of vehicle type, the steering mechanism consists of several parts like steering wheel, steering shaft, tie rods, steering arms etc. It is nearly impossible to align all the aforementioned components in a single axis due to design constraints. Therefore, universal joints are employed to distribute the steering response in a designated direction. Apart from automobile, universal joints are also integral parts of aircraft control mechanisms, medical instruments etc¹. Universal joint (Figure 1) has the capability to join two offset shafts and transmit rotational motion of one shaft to another in synchronized manner. If aligned perfectly, the synchronous movement of the two shafts will have consistent speed. Practically, it is not feasible to attain similar speed by both the shafts due to the presence of joint friction and joint angle. In practice, bending moments affect the life of the universal joint system severely². Various types of universal joints are used in the automobile sector such as Hook type joint, Hardy-Spicer joint, Layrub joint, Cross type joint, Block type joint, Doughnut rubber coupling etc^{3.4}. Inmost commonly used universal joint assembly, two forked

*Author for correspondence

shafts are connected using a cross-shaped intermediate component. It frequently subjected to torsional as well as bending stresses and unwanted vibrations. Therefore, it might fail by torsion induced deformation and fatigue.

Fierce competition in the market forced the manufacturer to reduce the time-to-market duration of a component and provide a well-designed reliable and durable product. In this short duration of time span, design, development and optimization operations along with up-front analysis need to be conducted to ensure the quality of the product. With the development of faster workstations or computers and more sophisticated design and analysis software, it is possible to inculcate a more realistic design, and analyze its real-time operation or performance. In the present investigation, a Computer Aided Design (CAD) software named CATIA is used to design a universal joint which is further analyzed in finite element analysis software named ANSYS. CAD software is equipped with multiple application modules; one of them is geometric modeling. It is a subdivision of applied mathematics and computational geometry. It provides a mathematical description of the geometry of objects using different algorithms. A CAD program performs the numerical analysis of a contour and illustrates it as a 3D image on the computer screen. Therefore, a CADmodel describes the intent design, where geometrical characteristics, parameters and constraints are entered. It also provides the geometrical restrictions for the part⁵.



Figure 1. A schematic diagram of commonly used bench vice $\frac{21}{2}$.

Table 1. Chemical composition of structural steel ^{Δ}
--

A geometric modeling in CAD framework generally consists three steps. In the first step, necessary geometric elements are constructed using multiple commands such as point, line, circle etc. In the second step, the geometric features were transformed using commands such as scaling, translation, rotation, cutting etc. In that way, it is possible to create a library of different profiles that replicate themselves. In the third step, a desired shape of the object is obtained by integrating already created or transformed elements of profiles in previous steps^{6.7}. CATIA is a commonly used CAD framework for geometric modeling of various components frequently used in mechanical devices⁸⁻¹¹. In the Finite Element Analysis (FEA), computational methods are used to perform various engineering analysis such as structural, heat and mass transfer, fluid flow etc. It is a potent tool in which an investigated section is subdivided into very small subsections named finite elements. These finite elements were modelled using simpler equations, and all these equations were clubbed to form a significant equation for modelling the whole investigated section. ANSYS is very commonly used FEA software in engineering¹².

Universal joints used in automobile steering section did not become directly subject to axial loading. As already mentioned it is a type of hinged joints and subjected to rotational motion. In collapsible steering columns, there is a minute chance that the column will exposed to axial forces (mostly compressive) during an accidental hit. No study explained the effect of tensile or compressive force in the axial direction of universal joints. Therefore, the present investigation aims to design a structural steel universal joint and analyze its behavior under co-axial compressive loading.

2. Materials and Methods

2.1 Material

Universal joint fabricated from structural steel was utilized for the present investigation. Table 1 contains the detailed chemical composition of commonly used structural steel.

Elements	Manganese (Mn)	Silicon (Si)	Phosphorus (P)	Carbon (C)	Iron (Fe)
Weight % (max)	1.60	0.50	0.025	0.12	Remaining

2.2 Design and Analysis Software

CATIA is a powerful tool for 3D interactive modeling. The 3D design of connecting rod was modeled on CATIA V5R20 software. ANSYS with Finite Element Analysis (FEA) tool is utilized to solve complex engineering problems, design optimization etc. It is also assisting in reducing the expenditure of physical testing and material loss. The design of the connecting rod was imported from the CATIA and analyzed in ANSYS 17.2 software using various boundary conditions. The characteristics of materials employed for the analysis are listed in Table 4.

3. Results and Discussion

3.1 Modeling of Universal Joint in CATIA

Solid modeling is started with the creation of a 2D geometry of the profile using sketcher module. The profile sketch tool or product structure tool is used to create the basic outline of the base body design. All the necessary limitations or restrictions (like coincidence, contact, offset, angle, fix etc.) required for the design are imposed using constraints toolbar. After exiting sketcher module, the solid model is generated in part design module. Pad tool in part design module was used to prepare a solid 3D model. The unwanted portion of the solid model was removed using the pocket tool. The hole, chamfer, shaft etc. tools are used to create the relevant contour profile at an appropriate place in the design. Figure 2 is representing the solid 3D model of the base body formed in part design module. The fork body, center block, pin and collar as shown in figure are designed separately using sketcher module, part design module, and pad/ pocket tools as per the requirement. The last step of the geometric modeling is to assemble all the scattered parts into one. The positioning of multiple parts at suitable places is carried out in assembly design module using manipulation tool. In addition, the relationships between the selected components are established using constraints such as coincidence (contact, offset, and angle) and fix. Conventional elements accessible from the library are

(a) (b) (c) (c) (d)

Figure 2. The basic form of the (a) Fork body (b) Center block (c) Pin (d) Collar in part design module.



Figure 3. Design of universal joint.

utilized to give a final finish to the geometrical design. The final design of the universal joint is shown in Figure 3. The necessary material properties for performing proper virtual analysis are listed in Table 2.

Tab	le 2.	Mes	hing o	detail	s of	various	sections
-----	-------	-----	--------	--------	------	---------	----------

Universal Joint Parts	Collar 1	Collar 2	Pin 1	Pin 2	Fork Section 1	Fork Section 2	Center Block
Nodes	773	773	1166	1166	2244	2244	1641
Elements	357	357	557	557	1152	1152	789

3.2 Finite Element Analysis in ANSYS

A 3D model developed in CATIA was imported to the design modeler of ANSYS 17.2 software for analysis. The analysis was carried out using static loading condition. Before starting the virtual analysis, meshing of the designed part is an essential operation to obtain reliable results. Details of the parameters used in meshing operation along with other parameters are included in Table 3. The compressive load of 100 N was applied to the fork

S. No.	Properties	Values	
1.	Density	7.85×10 ⁻⁶ kg/mm ³	
2.	Coefficient of thermal expansion	1.2×10 ⁻⁵ /°C	
3.	Specific heat	4.34×10⁵mJ/kg °C	
4.	Thermal Conductivity	6.05×10 ⁻² W/mm °C	
5.	Resistivity	1.7×10 ⁻⁴ ohm mm	
6.	Compressive Yield Strength	250 MPa	
7.	Tensile Yield Strength	250 MPa	
8.	Tensile Ultimate Strength	460 MPa	
9.	Zero Thermal Strain Reference Temperature	22 °C	
10.	Young Modulus	2×10 ⁵ MPa	
11.	Poisson Ratio	0.3	
12.	Bulk Modulus	1.6667×10 ⁵ MPa	
13.	Shear Modulus	76923 MPa	
14.	Relative Permeability	10000	

Table 3. Material properties

section 2, and fork section 1 was fixed. The variation in equivalent strain (von-Mises strain), equivalent stress (von-Mises stress) and the deformation in universal joint are shown in Figure 4-6. The propensity of stresses, strains and deformations are represented by multiple colors in the given contour of the universal joint. Red color represents maximum whereas, blue color represents a minimum.

Figure 4 revealed the von-Mises stress profile for the designed universal joint. The von-Mises stress is utilized to predict the appropriateness of the material for a particular set of loading conditions. If the von-Mises stress is higher than the yield strength, the material is said to be crumpled¹³. It is generally used to study the failure behavior of the ductile components. The von-Mises theory considers all types of stresses responsible for failure.

Table 4. Parameters used for FEA analysis

S. No.	Parameters	Values	
1.	Materials used	Structural steel	
2.	Volume	$2.8802 \times 10^{6} \text{ mm}^{3}$	
3.	Mass	22.609 kg	
4.	Scale factor value	1	
5.	Total number of bodies	7	
6.	Total number of active bodies	7	
7.	Total number of nodes	10007	
8.	Total number of elements	4921	
9.	Mesh metric	None	
10.	Applied compressive load	100 N	



Figure 4. Von-Mises strain profile of universal joint at 100 N loading.



(ii)

Figure 5. Von-Mises strain profile of universal joint at 100 N loading.



Figure 6. Total deformation in universal joint at 100 N loading.

On the other hand, Tresca theory omits the intermediate principal stress during analysis¹⁴. From Figure 5, it would be conferred that the maximum stress was observed at the nearby region of fork and center block joint. From the observation, it could be estimated that the area near to the periphery of fork and center block joint is susceptible to failure if an excessive crushing load is imposed during accidental conditions. This area is also vulnerable to fatigue failure due to cyclic nature of the stresses¹⁵. However, the maximum stress on universal joint is well

below the yield strength. Therefore, the rupture does not take place for the loading conditions used in the study¹⁶. To evaluate the maximum von-Mises stress is essential concerning fatigue because the high tensile stresses mostly initiate fatigue cracks at certain points^{17,18}. At maximum loading conditions, the load shared by various sections of the connecting rod resulted in high-localized pressure. Owing to this localized pressure or stresses, the universal joint may or may not face catastrophic failure during active service life.

However, it can act as active sites for wear or fatigue and reduce the useful life of the part¹⁹. The von-Mises stress values for structural steel universal joint exhibited that the jointissafe to use for loading conditions used in the current study.

Figure 5 exhibited the von-Mises strain profile for the designed universal joint. The maximum von-Mises strain was occurred in center block more precisely near to the region where fork section is being attached with the center block.

The deformation in universal joint might take place due to the bending or buckling under severe loading. Figure 6 revealed the deformation profile of universal joint at 100 N compression loading condition. The maximum deformation occurred at the fork Section 1.

The maximum value of von-Mises stress or equivalent stress was lower than the yield strength. The calculated factor of safety for structural steel universal joint is very high for 100 N loads. Therefore, the design of universal joint is safe for applied load in static loading condition.

The collapsible steering column is an effective mechanism for safe and comfortable driving experience, especially in the all-terrain vehicle. For this purpose, the steering column components should be able to bear sudden compressive and tensile forces. It can not only provide safety to the driver but also reduce the damage to the vehicle during minor crashes. The energy generated in the steering column can be dissipated through joints. As per the results obtained, the universal joint (design and material) studied in the present work is safe for minor crashes or accidentally generated axial compressive load.

4. Conclusions

In this work, the design and finite element analysis of universal joint were accomplished. We focused on the effect of axial compressive loading on the universal joint. The design of the universal joint was prepared using CATIA software, and the static analysis of the model was performed in ANSYS. In the existing model, the noted von-Mises stress was 0.14 MPa for 100 N axial forces which is well below the yield strength of structural steel. The maximum stress and maximum strain were generated at and around the joint of the Fork Sections (1 and 2) and the Center Block. However, the possibility of maximum deformation was observed in the Fork Section-1 due to its restraint movement. From the analysis, it can be concluded that the existing design and material of universal joint is safe against accidentally generated axial loading. The behavior of Universal joint under axial loading may assist in developing the more efficient collapsible steering columns in future. To further ensure the safety of the existing universal joint, this design must be tested for higher axial compressive loads. Apart from axial loading test, the combination of axial as well as rotational load needs to apply, and behavior must be analyzed. The combination of loads on the universal joint will be able to provide more realistic or practically feasible data than the single loading approach.

5. Acknowledgements

The support extended by Accendere Knowledge Management Services, CL Educate, New Delhi during the manuscript preparation is greatly acknowledged.

6. References

- Hummel SR, Chassapis C. Configuration design and optimization of universal joints with manufacturing tolerances. Mechanism and Machine Theory.2000; 35(3):463–76. https://doi.org/10.1016/S0094-114X(98)00092-5
- Sheu PP, Chieng WH, Lee AC. Modeling and analysis of the intermediate shaft between two universal joints. Journal of Vibration and Acoustics.1996; 118(1):88–99. https://doi. org/10.1115/1.2889640
- Whipp J, Brooks R. Universal joints/propeller shafts/drive arrangements. Transmission, Chassis and Materials; 1983. p. 49–60. https://doi.org/10.1007/978-1-349-06704-6_4
- 4. Universal Joints (Automobile). Available from: http://whatwhen-how.com/automobile/universal-joints-automobile/
- Olofsson J, Salomonsson K, Johansson J, Amouzgar K. A methodology for microstructure-based structural optimization of cast and injection moulded parts using knowledge-based design automation. Advances in Engineering Software. 2017; 109:44–52. https://doi.org/10.1016/j.advengsoft.2017.03.003
- Geometric modeling. Available from: https://books.google. co.in/books/about/Geometric_modeling.html?id= TPVQAAAAMAAJ&redir_esc=y
- 7. Computer graphics and geometric modeling for engineers. Available from: https://dl.acm.org/citation.cfm?id=530066
- Choi G-H, Mun D, Han S. Exchange of CAD part models based on the macro-parametric approach. International Journal of CAD/CAM. 2002; 2(1):13–21.
- 9. Integrating advanced computer-aided design, manufacturing, and numerical control: Principles and implementations.

Available from: https://www.igi-global.com/book/integr ating-advanced-computer-aided-design/611

- Mun D, Han S, Kim J, Oh Y. A set of standard modeling commands for the history-based parametric approach. Computer-Aided Design. 2003; 35(13):1171-9. https://doi. org/10.1016/S0010-4485(03)00022-8
- 11. Computer aided design and manufacturing. Available from: https://www.inc.com/encyclopedia/computer-aided-design-cad-and-computer-aided-cam.html
- The finite element method: linear static and dynamic finite element analysis. Available from: https://www.amazon.in/ Finite-Element-Method-Mechanical-Engineering-ebook/ dp/B00CB2MK8K
- Wahono B, Praptijanto A, Santoso WB, Nur A. Modeling and static analysis of a connecting rod in range extender engine. IEEE Proceedings of Sustainable Energy Engineering and Application (ICSEEA); 2016. p. 17–21. PMid: 26458737
- Ali HMM, Haneef M. Analysis of fatigue stresses on connecting rod subjected to concentrated loads at the big end. Materials Today: Proceedings. 2015; 2(4–5):2094–103. https://doi.org/10.1016/j.matpr.2015.07.205
- 15. Ilman MN, Barizy RA. Failure analysis and fatigue performance evaluation of a failed connecting rod of recip-

rocating air compressor. Engineering Failure Analysis. 2015; 56:142–9. https://doi.org/10.1016/j.engfailanal.2015.03.010

- Zhu X, Xu J, Liu Y, Cen B, Lu X, Zeng Z. Failure analysis of a failed connecting rod cap and connecting bolts of a reciprocating compressor. Engineering Failure Analysis. 2017; 74:218–27. https://doi.org/10.1016/j.engfailanal.2017.01.016
- Witek L, Stachowicz F, Zalęski A. Failure investigation of the crankshaft of diesel engine. Procedia Structural Integrity. 2017; 5:369–76. https://doi.org/10.1016/j.prostr. 2017.07.184
- Charkha PG, Jaju SB. Analysis and optimization of connecting rod. IEEE Proceedings of Second International Conference on Emerging Trends in Engineering and Technology (ICETET); 2009. p. 86–91.
- Khare S, Singh OP, Dora KB, Sasun C. Spalling investigation of connecting rod. Engineering Failure Analysis. 2012; 19:77–86. https://doi.org/10.1016/j.engfailanal.2011.09.007
- 20. Structural Steel 460- Chemical composition, mechanical properties and common applications. Available from: https://www.azom.com/article.aspx?ArticleID=8067
- 21. How to make design and assembly of universal joint 3D modeling in solidworks design engineering. Available from: https://solidworks99.blogspot.com/2017/09/how-to-make-design-and-assembly-of.html