Graduate Journal of Interdisciplinary Research, Reports & Reviews
(Grad.J.InteR^3)
ark:/85846/JPR3.2025AR.03010016

ISSN(E): 2584-2919
Vol.03, No.01 (2025)
pp.16-23

Research Article

Design and Analysis of Universal Joint Center Block

Akash Singh & Santosh Tripathy

Department of Mechanical Engineering, GIET University, Gunupur, India.
Email: sktripathy@giet.edu (Corresponding Author)

(Received: 21-01-2025; Accepted: 15-05-2025; Published Online: 21-05-2025)

ABSTRACT:
Background: Universal joints are critical components in automotive power transmission systems, enabling torque transfer between misaligned shafts. One of the essential elements of this mechanism is the center block, which connects the driver and driven shafts and is subject to complex loading conditions.
Purpose: This study aims to analyze the structural performance of the center block by evaluating the effect of different material properties and design variations. The goal is to optimize the component in terms of strength, shape, size, and weight under real-time operating conditions.
Methods: Design and simulation tools including NX 12 and ANSYS Workbench 16 were used to create and analyze two center block designs. Fracture analysis was performed by applying appropriate moments while fixing other components to replicate realistic operational conditions. Four materials—structural steel, stainless steel, aluminum alloy, and grey cast iron—were tested in the simulation environment.
Results: The analysis revealed significant differences in performance based on material selection and design variation. Structural steel and stainless steel provided higher strength, while aluminum alloy offered considerable weight reduction. The optimized design demonstrated improved mechanical integrity and material efficiency.
Conclusions: Material selection and structural design significantly impact the mechanical behavior of the center block in a universal joint. Simulation-driven testing is effective in identifying optimal configurations, thereby enhancing the reliability and performance of automotive power transmission systems.

Keywords: Universal joint, NX 12, ANSYS 16, Finite element analysis

1. Introduction

The power produced from an automobile engine is transferred to the drive wheel by a power transmission system. To transmit the driving torque from the engine to the final drive by the propeller shaft, at least one or two universal joints are required. Universal joint is a positive mechanical joint used for connecting shafts, whose axes are inclined at an angle to each other. It is also known as universal coupling, U-joint, Cardan Joint and Hooke's Joint. It compensates angular misalignment between the shafts in any direction. Generally, in order to use a universal joint, the inclination between the shafts should be less than 30 [1]. The two fork ends are assembled co-axially with respect to the center block. The pins are assembled into the holes provided in the fork end. They are held in position by means of a collar and a collar pin. The assembled view of a universal joint is shown below.

For the determination of stress conditions at the stress sensitive sections, stress analysis is also carried out by the finite element method.

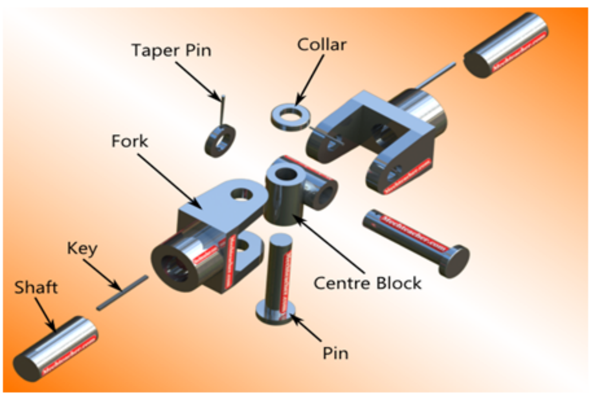


Figure 1: Components of universal joint

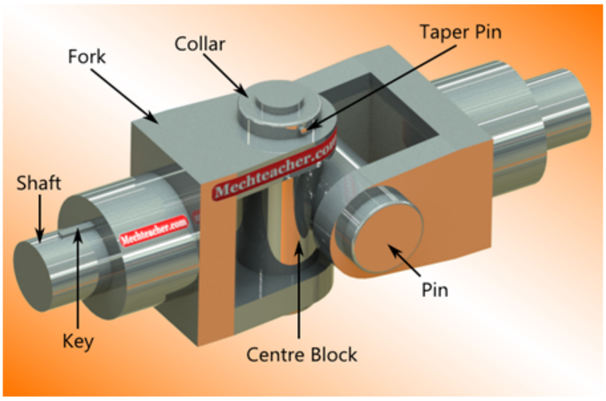


Figure 2: Assembly of components of universal joint

The finite element method (FEM) is a “numerical technique for solving problems which are described by partial differential equations or can be formulated as functional minimization. A Domain of interest is represented as an assembly of finite elements” [2]. The ANSYS software was developed in 1970 by Dr. John Swanson which was initially named as ANSYS Inc. The developer of simulation package John Swanson is regarded as the forefather of modern FEA. The capabilities of ANSYS software has increased manifold by expanding its domain of operations which include magneto statics, kinetic dynamics and piezoelectric, electronics, CFD, structural, vibration, fatigue and other optimization techniques. The ANSYS software aided to reduce production cycle time by reducing crash tests and other destructive forms of testing required in initial product development.

ANSYS Workbench employs 3 of the ANSYS solvers and automatically chooses the most appropriate or efficient solver for the job at hand. In addition to linear/static, ANSYS Workbench performs Coupled analysis types (thermal-stress, stress-modal, thermal-stress model) as well as some limited non-linear analysis types (thermal with temperature- dependent material properties and convection, geometric/contact with contact supporting lift- off). All solver settings and iteration propagations from one solve step to the next are performed automatically [3].

The power transmission from one shaft to another was initially proposed by Gerolamo Cardano who was an Italian scientist. The design proposed by him comprises a shaft connected with a joint which was known as cardan joint. The device was later reinvented by Christopher Polhem in 1730 A.D. which was named as Polhemsknut. The universal joints are used in automobile transmission systems and operate at much higher angles with misalignment couplers. These joints rotate the shafts which have laterally displaced axes. One of the major advantage is its ability to change angle while operating under varying loads [4].

Universal joints are typically used to transmit positive rotation through intentional offsets where the power source is some way from the load. It is important to note that a U/J is essentially a pivot and that it cannot accommodate any parallel displacement between shafts if used singly. In addition, the installation must allow for some adjustment when assembling the joint so that the shaft center lines can converge at the pivot point of the joint. This is necessary if the joint is to function properly and not cause excessive radial load on adjacent bearings. However, most applications require a universal joint pair. This brings several advantages including a constant output speed, a less critical installation procedure, a shared working angle (each joint works at 50% of the whole) and the ability to pass through offset shafts. A pair of joints can include a drive shaft with a U/J at each end or a double U/J for close-coupled applications.

Universal joints are selected for size based on torque transmission, rotational speed and working angle. These variables give rise to a performance graph from which the values can be read and the appropriate joint selected. Sometimes factors related to the nature of the power source or load apply. For example, a single-cylinder internal combustion engine is more punishing for the transmission than an electric motor. Uniform loading is less demanding than intermittent loading. In principle, the U/J works harder as the working angle increases. The larger the working angle, the lower the torque or speed that can be transmitted,or both [5]. In selecting the best type of universal joint for a given application, the intended service life and service life requirements are the determining factors. High speeds and/or working angles are best handled by U/J equipped with roller bearings. These are lubricated for the entire lifetime, but it is still a good idea to protect the moving surfaces with a sleeve that prevents the ingress of dust, moisture and other foreign substances. Roller bearing joints are generally intended where continuous speeds exceed 1200/1500 rpm.

2. Literature Review

In one paper study, fracture analysis of a universal joint yoke and a drive shaft of an automobile power transmission system were carried out. Spectroscopic analyses, metallographic analyses and hardness measurements were carried out for each part. For the determination of stress conditions at the failed section, stress analysis is also carried out by the finite element method.

In another paper the main objective of the paper was to reduce shear failures by modification of pin (cross) in the existing design of universal coupling. The modeling of proposed design is done by using CREO software & static and dynamic analysis is done in ANSYS software & results are compared with existing design. One paper says that the universal joint is connected with a rigid rod that allows the rod to bend in any direction, and is commonly used in shafts that transmit rotary motion. Due to pin (fork) wear occurs, at the mating surface of the universal coupling. It becomes noisy when rotating at high speed. The main aim of their work was to replace traditional single joint universal coupling with modern one which has a high strength to withstand desired motion, reduce weight & cost and save time during assembly of components [2].

They have conducted numerical investigations on universal joints connected to rotating shafts. The shafts are intersecting and non-parallel connected using “spider”. From the numerical investigation the critical zones of high stresses are determined and the effect of operating angle on strength of shaft is evaluated on the basis of stresses and deformation [6].

In this paper it researches steering column universal joints using techniques of Finite Element Method. The boundary conditions are applied on universal joints which are high torque and high load. The research findings have shown that by altering the certain design parameters of universal joints the performance and fatigue life of yoke can be significantly improved [7].

The numerical investigation on a universal joint consisting of two yokes (connected to each shaft) with the help of a spider. From the FEA analysis critical regions of high stresses and deformation are obtained and the effect of different mass geometry, loading conditions on yoke geometry is evaluated. The research findings have shown that maximum equivalent stress obtained from the analysis is 70MPa which is well below the yield strength (250 MPa) and is therefore safe. The topological optimized design is also safe and well within the safe limit.

The other numerical investigation on flange coupling using JAVA coding platform. The dimensional computing of the flange couple is done in very short duration. With this program any one can “compute the dimensions of the coupling within a fraction of seconds. The provision given to change the flange, shaft, key and bolt material will be an added advantage for the tool. Since the simplicity in operation and possibility of changing the material this tool can be used in all design and fabrication industries” [5].

The numerical investigation on yoke to investigate the effect of different design parameters on performance and structural stability. The loading type applied on the yoke are tensile and compressive in order to evaluate the stress concentration region. The FEA results have shown various design parameters (width and thickness of joint) have significant effect on stress concentration and thus affecting the performance. From the review, it can be noted that failure of component is occur due manufacturing and design fault, raw material faults, maintains faults, material processing faults, drivable joint angle, cyclic load to avoid this problems various method such as a topology optimization method, Weight reduction method, Shape optimization method, manufacturing method [8].

Focuses on increasing the performance and improving the life expectancy of the cardan joint. The use of the intermediate spring and damper increased the torque arm, increasing the degree of freedom in order to remove the regularity in the impact load, installing a rigid ring over the cardan joint arm to act as the inner ring [9].

3. Methodology

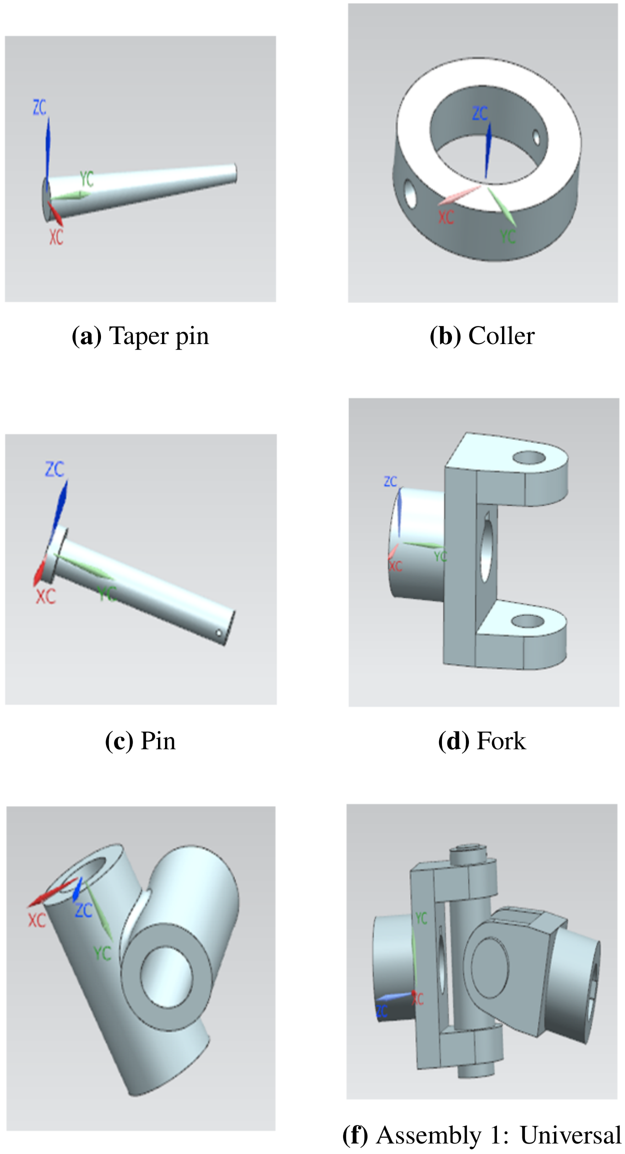
In the methodology a review of existing material and existing design of universal joint is done. The FEA analysis is conducted on the universal joint which involves 3 different stages i.e. preprocessing, solution and post processing. In the preprocessing stage, the CAD model of universal joint is developed in CREO parametric design software and the design is converted in para solid file format. The model is imported in ANSYS design modeler where it is checked for geometric errors or surface imperfections. The model is meshed which involves selection of element type, mesh type and element sizing.

After meshing, appropriate loads and boundary conditions are applied on the structure to determine stresses. The material of the universal joint is then changed to Al 7075 alloy and the FEA analysis is reconducted. A comparison study is then conducted between the results obtained from Al 7075 alloy, cast iron and stainless steel to determine which is the best material in terms of good strength to weight ratio.

The primary design is modeled in NX 12 software by separately modeling the components in separate part design windows. Then all components are called in the assembly window to make a complete assembled universal joint. Different constraints like touch and align, concentric, parallel, fix etc were applied.

The components of universal joint is modeled separately in NX software and then assembled together to form a strong joint. This joint will not have any transverse motion but only rotate about horizontal axis. To transfer the torque generated from the engine to differential their will be two universal joints attach to the propeller shaft ends.

The Finite element analysis is carried out in ANSYS software for the assembly 1 universal joint .The first step was to select the material of the assembly and center block1 and it was structural steel. Then the meshing of the assembly is done with number of nodes 41873 and elements 21294 with mesh command. The output came is shown below: The meshing of center block 1 mesh was refined by doing the refinement of



(a) Taper pin **(b)** Coller

(c) Pin **(d)** Fork

(e) Center block 1 **(f)** Assembly 1: Universal joint

Figure 3: Key components of the mechanical assembly

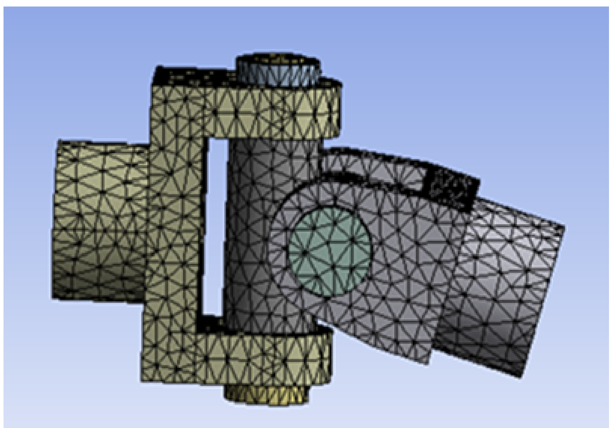


Figure 4: Meshing of assembly1

mesh using commands shown below:

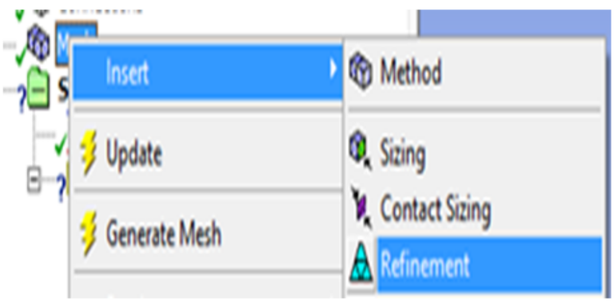


Figure 5: Step for mesh refinement

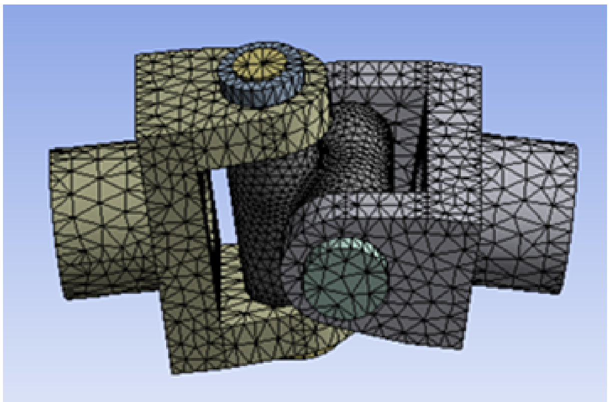
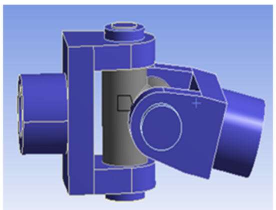


Figure 6: Refined mesh

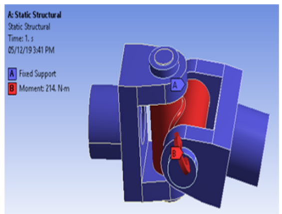
4. Boundary Conditions

The boundary condition were applied such as fixing the both forks, coolers, needle pins and pins also applied the moment of 214 Nm on the central block design 1 and 2 with respect to z axis by keeping its value zero in the x and y axis. This input is provided to the Ansys software and after solving the mathematical equations by the software working behind it we get the following results.

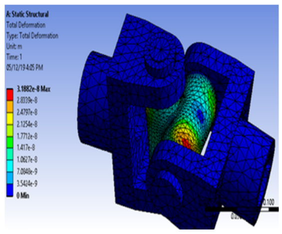
The stress concentration is appeared the ends of the center block due to bending effect at ends that can cause wear and fracture at those sensitive areas. To avoid those possible failure there is a need to modify the center block design so the outer diameter of center block ends are increased from 56 mm to 65 mm and extrude to a length of 8mm at all four ends the edges were also given filet of 2 mm radius to avoid stress concentration. The new modified design is given name as center block 2. All the other components dimensions of the universal joint were maintained the same. New assembly is named assembly 2. The center block 2 and assembly 2 is shown in fig. below. The same way the assembly 2 of the universal joint is meshed and



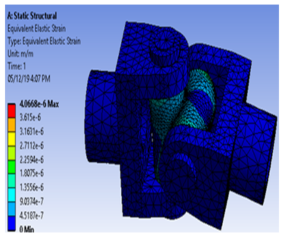
(a) Fixing of fork ends



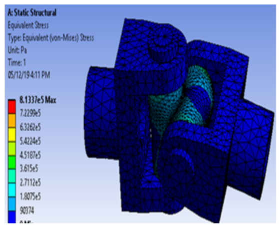
(b) Moment applied



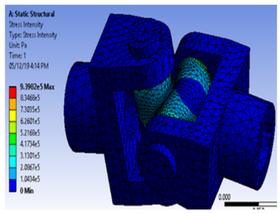
(c) Total deformation



(d) Equivalent strain diagram

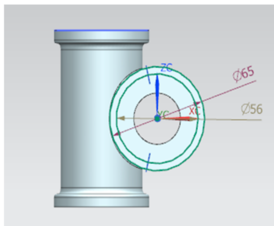


(e) Equivalent (von – mises) stress

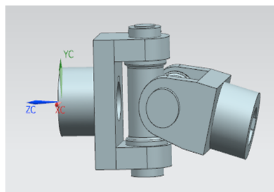


(f) Stress intensity

Figure 7: Ansys software Output



(a) Center block 2



(b) Assembly 2: Universal joint

Figure 8: Assembly 2

analyzed by ansys software. Different materials were selected for center block 2 for analyzing the effect and suitability with respect to strength improvement and weight reduction.

5. Discussion of Results

The FEA analysis is conducted on universal joints with structural steel to determine equivalent stress and deformation plot. From

the FEA analysis the critical regions of high stresses and deformation are obtained. The equivalent stress

Table 1: Comparison of Parameters for Center Block Design 1 and 2

Parameters	Designs	Structural Steel	Stainless Steel	Aluminium Alloy	Grey Cast Iron
Total deformation (in 10^{-8} m)	Center block 1	3.18	3.33	9.22	5.68
	Center block 2	2.68	2.81	7.78	4.79
	Difference (%)	15.72	15.61	15.61	15.66
Equivalent elastic strain (in 10^{-6} m)	Center block 1	4.06	4.22	1.16	7.42
	Center block 2	3.84	3.99	1.09	6.99
	Difference (%)	16.52	5.45	6	5.79
Equivalent (Von-mises) stress	Center block 1	8.13	8.14	8.22	8.16
	Center block 2	7.67	7.69	7.74	7.67
	Difference (%)	5.65	5.52	5.83	6
Max.Shear Stress(in 10^5 Pa)	Center block 1	4.09	4.10	4.17	4.13
	Center block 2	3.84	3.85	3.94	3.89
	Difference (%)	6.2	6.1	5.51	5.81
Volume (in m^3)	Center block 1	3.43	3.43	3.43	3.43
	Center block 2	3.71	3.71	3.71	3.71
	Difference (%)	7.65	7.65	7.65	7.65
Mass (In kg)	Center block 1	2.68	2.65	0.949	2.46
	Center block 2	2.91	2.87	1.02	2.67
	Difference (%)	7.9	7.7	6.9	7.86

distribution plot is obtained from the FEA analysis. The maximum equivalent stress obtained from the analysis is 466.28MPa and the total deformation plot obtained from the FEA analysis is 0.85034. Which is inclosing agreement with results shown in literature [10].

The two designs of universal joints were prepared in NX software with the design of assembly 2 was prepared with minor modifications on the ends of the center block model to make it a safer design .The ANSYS software is used to compare the effect of materials selected for the center block. The following table describes the different parameters compared for center block 1 and center block 2 with same boundary conditions.

After thoroughly studying the above results we found that the minor change in mass and volume (about 7%) of center block design 1 we got remarkable reduction in all parameters and that with all types of material selected. We also found that the red region came during stress analysis of center block 1 got removed in case of center block 2. The value of total deformation was reduced by about 15% in the new design.

6. Conclusion

The results obtained are quite favorable which was expected. From the outcomes acquired from FE Analysis, numerous dialogs have been made. This work is carried out to study the effect of change in designed dimensions which were done according to failure behavior found in the component due to many external reasons. In future this work will help one to emphasize more work on design features rather than simply selecting the materials according to strength, cost and availability.

The FEA is a viable tool in investigating the strength of universal joint which saves time and cost as compared to experimental testing methods. From the FEA analysis the critical regions of high stresses and deformation are obtained. The FEA analysis is conducted on universal joint using structural steel, stainless steel, cast iron and aluminium7075 alloy. The detailed findings are summarized below:

i The maximum equivalent stress is obtained at the corner regions of the universal joint for all the materials.

ii The maximum equivalent stress is obtained for cast iron and minimum equivalent stress is obtained for aluminium 7075 alloy.

iii The maximum weight is obtained for structural steel and minimum weight is obtained for aluminium 7075 alloy. From this research, Aluminium 7075 alloy has minimum weight and minimum equivalent stress. We can conclude that the best material in terms of strength to weight ratio is aluminium 7075 alloy.

Authorship contribution: Akash Singh is Student of M.Tech, and Santosh Tripathy,Assistant professor in Machine Design.

Funding: No funding was used to support this research and the preparation of the manuscript.

Conflict of interest: The authors have no conflict of interest to report.

Declaration: The study has been conducted based on the method of document review in accordance with the qualitative approach of research and has been done on the basis of the secondary sources of data.

References

[1] Sanghani, D., & Gosvami, N. D. (2014). Optimization and finite element analysis of single joint universal coupling in automobile truck. *International Journal for Innovative Research in Science and Technology*, 1(7). ISSN: 2349-6010.

[2] Tathe, P. G., & Bajaj, D. S. (2013). Failure analysis of yoke assembly of a transmission drive shaft subjected to torsion and shear. *International Journal of Engineering Research and Technology*, 2(10). ISSN: 2278-0181.

[3] Mahajan, S., & Khamankar, S. D. (n.d.). Finite element analysis of universal joint. *Journal of Emerging Technologies and Innovative Research*. ISSN: 2349-5162.

[4] Chandole, S. K., Shende, M. D., & Bhavsar, M. K. (2014). Structural analysis of steering yoke of an automobile for withstanding torsion/shear loads. *International Journal of Research in Engineering and Technology*, 3(3), ISSN: 2319-1163.

[5] Shah, P., & Patil, A. (2016). Modeling and FEA of universal coupling of an automobile truck. *International Journal for Innovative Research in Science and Technology*, 3(2), 131–136.

[6] Bajaj, D. S., Tathe, P. G., & Kulkarni, S. S. (2014). Failure analysis and optimization in yoke assembly subjected by torsion and shear. *International Journal of Advanced Engineering Research Studies*, 3(4), 13–17.

[7] Solanke, S. G., & Bharule, A. S. (2014). An investigation on stress distribution for optimization of yoke in universal joint under variable torque condition: A review. *International Journal of Mechanical Engineering and Technology*, 3(2). ISSN: 2278–0149.

[8] Schmelz, F. (2011). Failure investigation and stress analysis of a longitudinal stringer of an automobile chassis. *Engineering Failure Analysis*, 16, 1696–1702.

[9] Ritchie, R. O. (1999). Mechanisms of fatigue-crack propagation in ductile and brittle solids. *International Journal of Fracture*, 100, 55–83.

[10] Singh, G., & Singh, S. (2017). Finite element methods-based structural analysis of universal joint using structural steel material. *Indian Journal of Science and Technology*, 10(31). https://doi.org/10.17485/ijst/2017/v10i31/113918

Copyright and License

How to Cite?

Akash Singh & Santosh Tripathy (2025). Design and Analysis of Universal Joint Center Block. *Graduate Journal of Interdisciplinary Research, Reports and Reviews*, 3(01), 16-23. Retrieved from https://jpr.vyomhansjournals.com/index.php/gjir/article/view/45