International Journal of Advancement in Electronics and Computer Engineering (IJAECE)
Volume 13, Issue 5, May. 2024, pp.108-115, ISSN 2278 -1412
Copyright © 2012: IJAECE (www.ijaece.com)

Revolutionizing ATV Steering: Advanced FEA and Optimization of Knuckle Design for Peak Performance

[1]Aashish Patidar, [2]Girish Kumar Khare
[1]M.tech Scholar, Department of Mechanical Engineering OIST,
patidarashish1996@gmail.com,Bhopal India.
,[2] Prof., Department of Mechanical Engineering OIST,
patidarashish1996@gmail.com,Bhopal India.

Abstract *– The knuckle joint, a crucial component in the steering system, is located between the steering rod and the pinion of the steering gear. This joint is designed to be easily disconnected when necessary. The stress experienced by the knuckle joint is influenced by the design of the stub and the materials used for the assembly components, which include the knuckle, stub, and arm. This research focuses on improving the design of the stub using the response surface method (RSM).*

In this study, the stub design was developed and analyzed using the ANSYS FEA (Finite Element Analysis) software package. The analysis revealed that the safety factor for the stub in the base design is 3.54, while for the knuckle, it is 4.15. These safety factors indicate the design's robustness and reliability under operational stresses.

To further enhance the design, response surface optimization techniques were applied. Before optimization, the maximum mass of the stub was 0.61046 kg. After optimization, the minimum mass achieved was 0.59352 kg, representing a 2.77% reduction in weight. This reduction not only contributes to material savings but also potentially improves the overall performance and efficiency of the steering system by reducing the load on the joint.

Keywords: Knuckle Joint, Steering System, Steering Gear, Stress Analysis, Stub Design, Material Selection, Response Surface Method (RSM), Finite Element Analysis (FEA), ANSYS Software

---

I. INTRODUCTION

The knuckle joint is a crucial component in mechanical and automotive engineering, particularly in the steering systems of vehicles. It serves as a pivotal connection between the steering rod and the pinion of the steering gear, allowing for smooth directional control and effective transmission of steering input to the wheels. One of its notable features is the ability to be easily disconnected when necessary, providing flexibility and convenience in maintenance and repair.

Understanding the stress distribution and overall mechanical behavior of knuckle joints is essential for ensuring the reliability and safety of the steering system. The performance of a knuckle joint largely depends on the design and material properties of its components, primarily the stub, knuckle, and arm. These components must withstand significant forces and stresses during operation, making their optimization a key area of research and development.

This study focuses on enhancing the design of the stub, a critical element of the knuckle joint, using advanced computational methods. Specifically, the research employs the Response Surface Method (RSM) to investigate and improve the design. RSM is a statistical technique used for optimizing processes and products by evaluating the relationships between input variables and desired outcomes.

By developing and analyzing the stub design using the ANSYS Finite Element Analysis (FEA) software, this research aims to achieve a more robust and efficient knuckle joint. The safety factor, an important measure of structural integrity, is calculated for both the stub and the knuckle. The base design achieves a safety factor of 3.54 for the stub and 4.15 for the knuckle, indicating areas where optimization can enhance performance.

The optimization process, guided by RSM, seeks to reduce the mass of the stub without compromising its structural integrity. Initial findings show that the maximum mass of the stub without optimization was 0.61046 kg. Through optimization, the minimum mass achieved was 0.59352 kg, representing a 2.77% reduction. This reduction in mass can lead to significant benefits in terms of material savings and overall efficiency of the steering system.

In previous studies, the importance of the knuckle joint's structural integrity has been extensively discussed. For instance, Wang et al. (2018) explored the mechanical properties and stress distribution in knuckle joints under various loading conditions. They emphasized the necessity of optimizing joint components to improve durability and performance. Similarly, Lee and Kim (2019) examined the impact of material selection on the fatigue life of knuckle joints, highlighting the role of high-strength alloys in enhancing joint longevity.

Recent advancements in computational modeling have opened new avenues for optimizing knuckle joint design. The Response Surface Method (RSM) is one such technique that has gained traction for its effectiveness in design optimization. RSM allows researchers to evaluate the relationships between design variables and performance outcomes systematically. Zhang et al. (2020) utilized RSM to optimize the dimensions of knuckle joint components, achieving significant improvements in load-bearing capacity while reducing material usage.

Building on these foundational works, the current research aims to enhance the design of the stub component within the knuckle joint using RSM and Finite Element Analysis (FEA). The stub's design and material properties play a crucial role in the overall performance of the knuckle joint. Therefore, optimizing the stub can lead to substantial improvements in the joint's efficiency and reliability.

This study employs the ANSYS FEA software package to model and analyze the stub design. The primary goal is to achieve a higher safety factor while minimizing the mass of the stub. The safety factor is a critical measure of structural integrity, indicating the margin of safety against failure. The base design, as analyzed, shows a safety factor of 3.54 for the stub and 4.15 for the knuckle, suggesting potential areas for optimization.

The knuckle joint is used to connect 2 rods which are aligned in different planes and the joint can be disconnected very easily. Some of the applications of knuckle joints are tie rod, valve rod joint, pump rod joint. Different parts of knuckle joint are shown in figure 1.1 below.

The pin, fork and eye are aligned along with rod and all these components are secured with taper pins. The pin or snug is used to restrict rotation of knuckle pin. The joint can be improved with the machining of fork and material preferred for knuckle joint is steel or wrought iron. To fasten circular rods, the pin type joints are used. The other end is fastened using taper type pin or collar pin. After fastening, the rods can swivel freely about the cylindrical pin and can be assembled in steering system of automobiles.

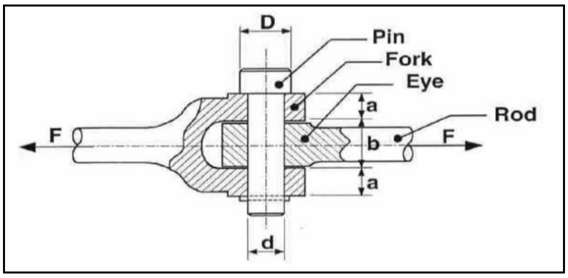


Figure 1: Knuckle joint

Generally used materials for knuckle joints are stainless steel or cast iron and with recent advancements, the Teflon can also be used and various researches have established that use of Teflon materials have minimized total deformation, stresses and strain as compared to cast iron

II. METHOD

Three The methodology employed in this study involves a systematic investigation to address the research objectives effectively. The following steps outline the approach taken to achieve the study's goals:

Literature Review:

A comprehensive review of existing literature was conducted to understand the current state of research on knuckle joints. This involved analyzing previous studies, research papers, and scholarly articles related to knuckle joint design, materials, manufacturing techniques, and performance evaluation.

Material Optimization Analysis:

Building upon the insights gained from the literature review, the study focused on optimizing the material selection for knuckle joint components. This involved evaluating different materials and their properties to determine the most suitable options for enhancing knuckle joint performance.

Finite Element Analysis (FEA):

ANSYS FEA software package was utilized to develop and analyze the knuckle joint design. Finite element analysis was employed to simulate the structural behavior of the knuckle joint under various loading conditions. This enabled the assessment of stress distribution, deformation, and performance characteristics.

Response Surface Optimization:

The response surface optimization technique was applied to refine the design of the knuckle joint stub. By systematically varying design parameters and analyzing their impact on performance metrics, the optimal configuration of the stub was determined to improve its efficiency and effectiveness.

Safety Factor Evaluation:

The safety factor of the knuckle joint and stub design was assessed to ensure structural integrity and reliability. This involved calculating the safety factor based on the critical load-bearing capacity and comparing it against established standards and guidelines.

Mass Reduction Analysis:

Through response surface optimization, efforts were made to reduce the mass of the knuckle joint stub while maintaining or improving its structural strength and performance. The mass reduction achieved after optimization was quantified and compared to the initial design configuration.

Result Analysis and Interpretation:

Finally, the results obtained from the material optimization, FEA simulations, response surface optimization, and safety factor evaluation were analyzed and interpreted. Insights gained from these analyses were used to draw conclusions, identify key findings, and make recommendations for future research and practical applications.

By following this methodological approach, the study aimed to contribute to the advancement of knuckle joint design and optimization, with implications for enhancing vehicle safety and performance.

Figure 2 Proposed Flow

The design optimization of stub is conducted using DOE (Design of Experiments) method of response surface method through which various design points are generated and output parameters i.e. stresses and deformation are generated at corresponding design points. Various 3D response surface plots are plotted for different optimization parameters along with sensitivity plots showing sensitivity percentage of various optimization parameters for different outputs generated.

Analysis Steps

CAD Modelling: The knuckle design is developed in creo design parametric software. The sketch and extrude along with fillet tool of creo parametric is used in modeling of knuckle.

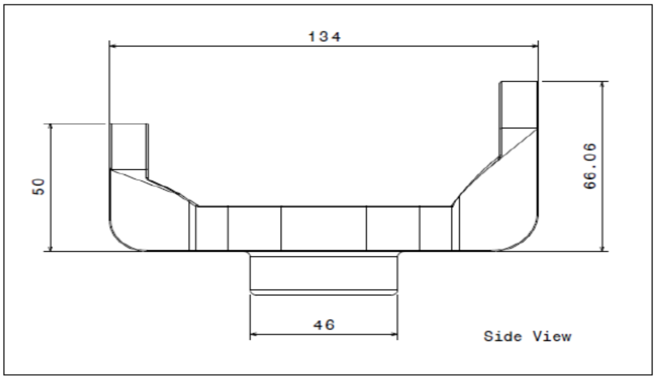


Figure 3 : Front view of knuckle

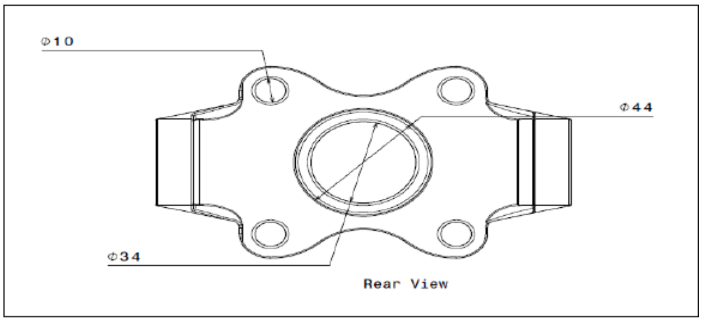


Figure 4: Rear view of knuckle

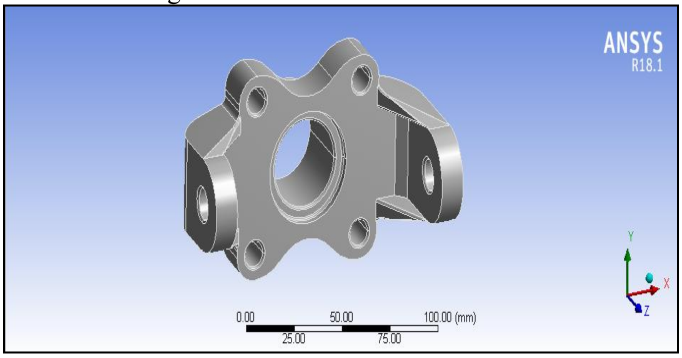


Figure 5:Knuckle design

The knuckle design is converted in compatible file type i.e. .iges which aides in exporting file to other software like ANSYS design modeler. The hard edges or other surface defects on knuckle is checked and removed.

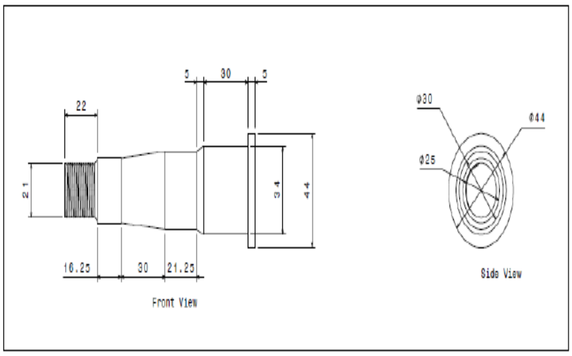


Figure 6:Stub dimensions

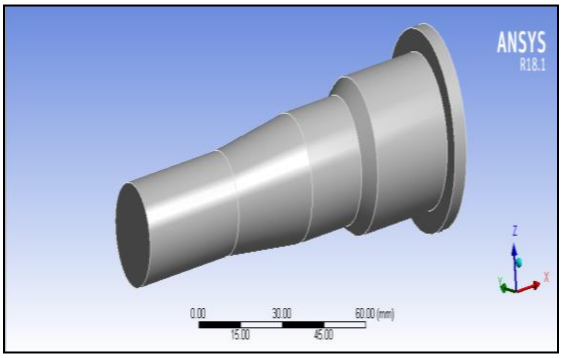


Figure 7: CAD model of stub

Structural Analysis

Using ANSYS software, the static structural analysis is conducted which is based on pre-processing, solution and post processing stages.

Pre-processing: In this stage “the CAD model is developed using ANSYS software. ANSYS design modeler is specific tool used for designing and editing operation. The model is meshed using tetra elements of appropriate size and shape. After meshing appropriate loads and boundary conditions are assigned”.

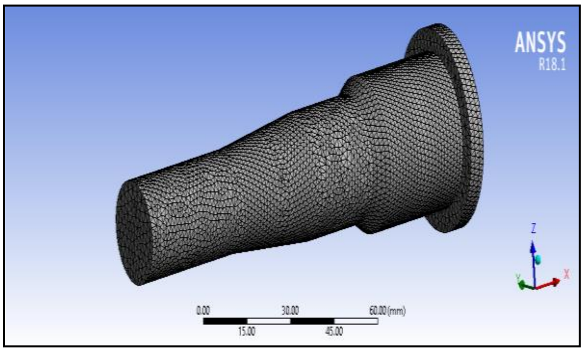


Figure 8: Meshed model of stub

Using tetrahedral elements, the stub is meshed with nearly 77300 nodes and 48000 elements. The meshed model of stub is shown in figure 7 above and that of knuckle is show in figure 8 below.

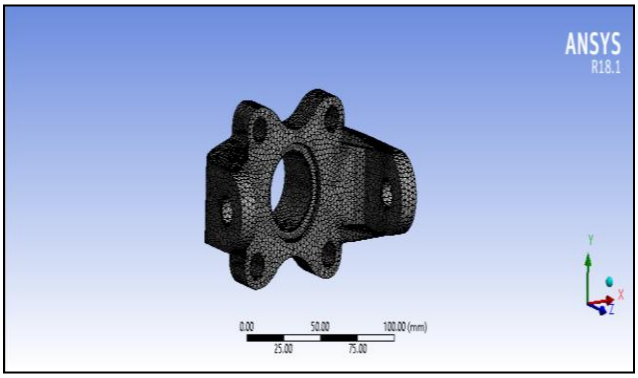


Figure 9 :Knuckle meshed model

III. RESULT & DISCUSSION

The stiffness matrix is formulated using minimum total potential energy formulation. For the current problem a linear spring of k stiffness is considered and an external force (F) is applied at the right. The spring deformation is given by Δ.

The work done by the single force is

$W = \Delta . F = \Delta_x * F_x = u F$

$U = ½ K \Delta_x^2$

Therefore, the total potential energy

(Π) for the loaded spring is

$\Pi = ½ K \Delta_x^2 - \Delta_x * F_x$(5.2)

Equation of equilibrium is obtained by minimizing this total potential energy with respect to the unknown displacement, Δ. That is,

$$\frac{\partial \Pi}{\partial \Delta x} = 0 = \frac{2}{2} K \Delta_x - F_x (5.3)$$

This gets simplified to below given equation which is well known equilibrium equation for leaf spring

$K \Delta_x = F$

The system is considered as spring and the potential energy is minimized along with application of displacement constraint.

$W = \Delta . F$

$U = ½ K \Delta_x^2$

$\{\Delta\}^T = [\Delta_1 \Delta_2]$

$\{F\}^T = [F_1 F_2]$

$W = \{\Delta\}^T \{F\}^T$

FEA Analysis of Knuckle

The FE structural analysis on knuckle joint enabled to determine zones of high stresses and low stresses. The high stress zone is observed at the corner region as shown in dark red color whereas lower stress zone is observed on backside of knuckle as shown in dark blue color.

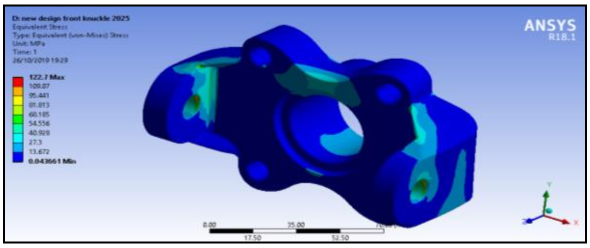


Figure 9: Equivalent stress at different regions of knuckle

The deformation plot in figure 9 shows higher magnitude at the cylindrical region as shown in red color whereas low deformation zone is shown in side support regions as represented by dark blue color. The maximum deformation is .06mm .

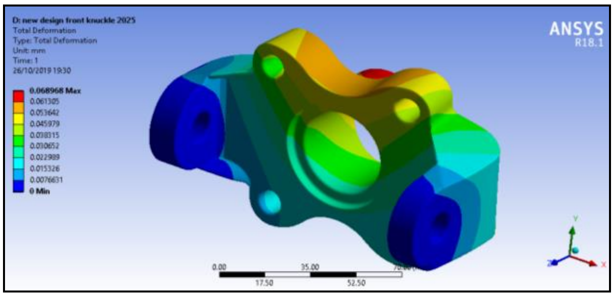


Figure 10: Deformation at different regions of knuckle

Stub Results

The maximum stress on knuckle stub is observed at the corner region of tapered section as represented in red color. The stress concentration at this region results in high stress and the stress decreases linearly towards the support and free end region as shown in dark blue color.

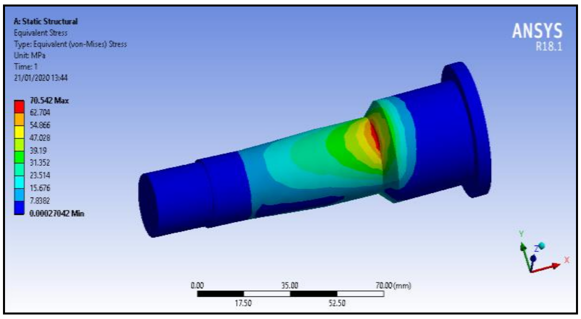


Figure 11 Equivalent stress plot at different regions of stub

The stub maximum deformation is observed at the free end zone as represented by red color and deformation decreases linearly as we move towards the supported end as represented by dark blue color. The maximum deformation observed is .06mm.

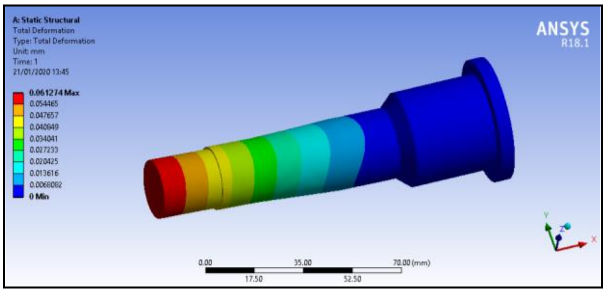


Figure 12: Deformation plot at different regions of stub

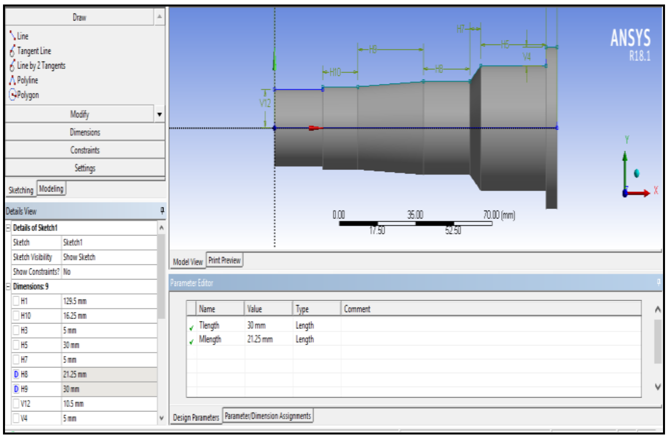
Table 1: Results for different components

Name	Yield Strength (MPa)	Deformation(mm)	Eq Stress (MPa)	Factor of Safety
Stub	250	.0612	70.54	3.54
Knuckle	510	.0653	122.7	4.15

Table 1 above shows safety factor and deformation for both components i.e. stub and knuckle. The safety factor achieved for spindle is 3.54 and for knuckle is 4.15. The existing design of knuckle has mass of .4561kg for 18mm thickness and further analysis is conducted with reduced thickness of knuckle.

Response Surface optimization

The first step of response surface optimization is selection of optimization variables. The component optimized is stub and the dimensional variables are shown in figure 13 below. The optimization variable name is "Tlength" and "Mlength".



Details of Sketch1

Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No

Dimensions: 9

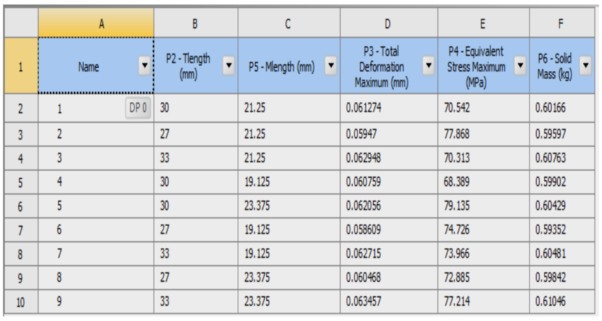
H1	129.5 mm
H10	16.25 mm
H3	5 mm
H5	30 mm
H7	5 mm
D H8	21.25 mm
D H9	30 mm
V12	10.5 mm
V4	5 mm

	Name	Value	Type	Comment
✓	Tlength	30 mm	Length	
✓	Mlength	21.25 mm	Length	

Figure 13: Parameters selected for optimization

Using design of experiments (DOE) different design points are generated which is based on mathematical model of linear regression. Different points are generated as shown in table 4.2 below. ANSYS optimization is selected is central composite design scheme.

Table 2: Design Points generated using Taguchi DOE



	A	B	C	D	E	F
1	Name	P2 - Tlength (mm)	P5 - Mlength (mm)	P3 - Total Deformation Maximum (mm)	P4 - Equivalent Stress Maximum (MPa)	P6 - Solid Mass (kg)
2	1 DP 0	30	21.25	0.061274	70.542	0.60166
3	2	27	21.25	0.05947	77.868	0.59597
4	3	33	21.25	0.062948	70.313	0.60763
5	4	30	19.125	0.060759	68.389	0.59902
6	5	30	23.375	0.062056	79.135	0.60429
7	6	27	19.125	0.058609	74.726	0.59352
8	7	33	19.125	0.062715	73.966	0.60481
9	8	27	23.375	0.060468	72.885	0.59842
10	9	33	23.375	0.063457	77.214	0.61046

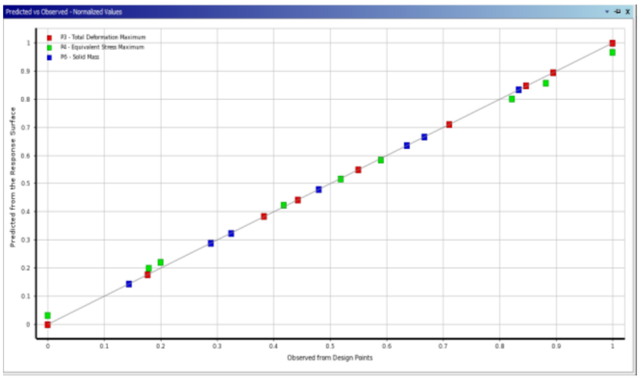


Figure 14: Goodness of fit curve

The goodness of fit curve shows difference between observed values and values expected under the model in question. The sensitivity analysis graph is plotted showing the percentage influence of input parameter on desired response i.e. shear stress and strain energy.

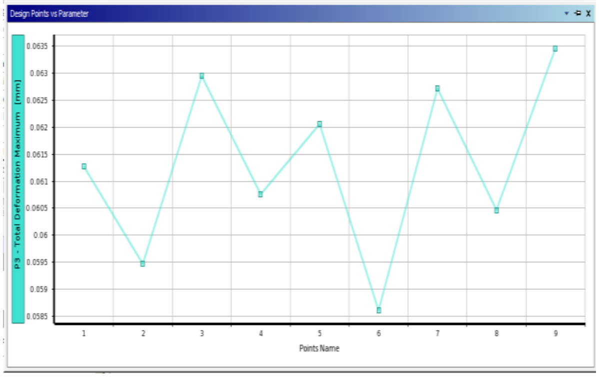


Figure 15: Deformation at different design points

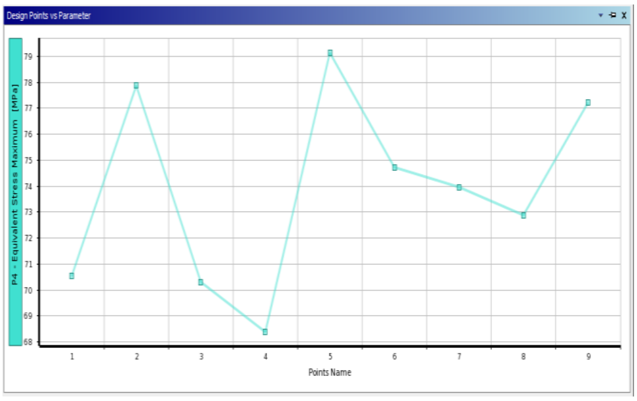


Figure 16: Equivalent stress at different design points

Similarly design points curve of equivalent stress and mass is shown in figure 16 and figure 17 respectively.

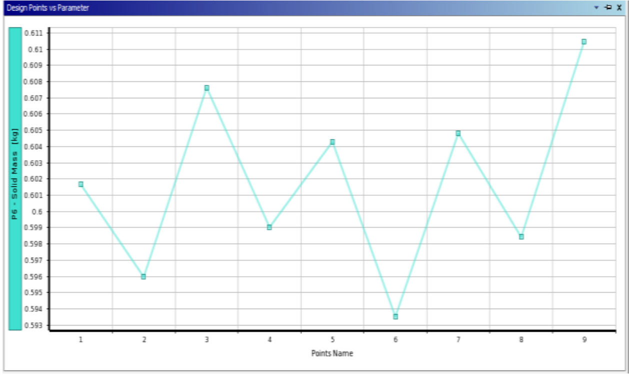
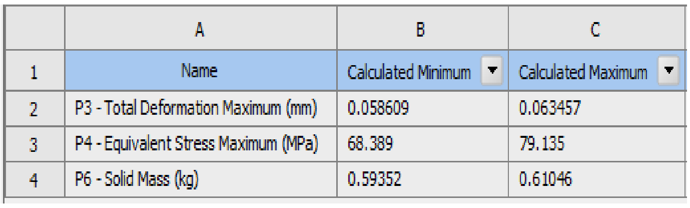


Figure 17: Solid mass at different design points

The solid mass points are shown in figure17 above. The maximum and minimum values are shown in figure 18 below.



	A	B	C
1	Name	Calculated Minimum	Calculated Maximum
2	P3 - Total Deformation Maximum (mm)	0.058609	0.063457
3	P4 - Equivalent Stress Maximum (MPa)	68.389	79.135
4	P6 - Solid Mass (kg)	0.59352	0.61046

Figure 18: Maximum and minimum values of different parameters

IV. CONCLUSION

This paper focus failure zones of knuckle and stub which are subjected to high stresses are identified using finite element analysis tool. The information on high stresses and deformation zone of knuckle and stub aided in improving design of these parts. The optimization of knuckle stub using Taguchi method enabled to generated various design points and corresponding stresses and deformation at these design points.

The detailed conclusion are as follows:

1. The FEA analysis conducted using structural steel material has enabled to determine regions of high stresses for knuckle and stub.
2. The safety factor achieved for stub is 3.54 and for knuckle is 4.15 for base design.
3. From response surface optimization technique, the maximum mass of stub without was .61046Kg and minimum mass obtained was .59352Kg which is 2.77% reduction.
4. From response surface optimization technique, the maximum equivalent stress of stub without was 79.135MPa and minimum equivalent stress obtained was 68.389Mpa which is 13.57% reduction.
5. From response surface optimization technique, the maximum deformation of stub without was .0634mm and

minimum deformation obtained was .0586mm which is 7.65% reduction.*models*

References

[1]. Sangamesh B. Herakal, Ranganath Avadhani,Dr.S.Chakradhar Goud, "STRUCTURAL STATIC ANALYSIS OF KNUCKLE JOINT", International Journal of Engineering Research and General Science Volume 4, Issue 2, March-April, 2016 ISSN 2091-2730.

[2]. Sanjay Yadav1, Ravi Kumar Mishra, Varish Ansari, Shyam Bihari Lal, "Design and analysis of steering knuckle component", International Journal of Engineering Research & Technology (IJERT)ISSN:2278-0181.

[3]. Mahesh P. Sharma, Denish S. Mevawala, Harsh Joshi, Devendra A. Patel, "Static analysis of steering knuckle and its shape optimization", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e- ISSN: 2278-1684, p-ISSN: 2320-334X.

[4]. AMEYA BHUSARI, ADITYA CHAVAN, SUSHRUT, "FEA &OPTIMISATION OF STEERING KNUCKLE OF ATV", Proceedings of 54th IRF International Conference, 8th May, 2016, Pune, India,ISBN: 978-93-86083-10-4.

[5]. Purushottam Dumbre, Prof A.K.Mishra, V.S.Aher, Swapnil S. Kulkarni, "STRUCTURAL ANALYSIS OF STEERING KNUCKLE FOR WEIGHT REDUCTION", International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974.

[6]. Atul Yadav, Scholar, "Optimization and fatigue analysis of steering knuckle", International Engineering Research Journal Page No 1041-1045.

[7]. Nilesha Patil, Mrs. Sayli M.Sable, Mr. Kashinath Munde, "STATIC STRUCTURAL ANALYSIS OF KNUCKLE JOINT",International journal of advancedtechnology in engineering and science, Volume No.4 ,Issue No.12 .

[8]. S V Dusane, M K Dipke and M A Kumbhalkar, "Analysis of Steering Knuckle of All Terrain Vehicles (ATV) Using Finite Element Analysis" IOP Conf. Series: Materials Science and Engineering 149 (2016) 012133.

[9]. B.Babu, M. Prabhu, P.Dharmaraj, R.Sampath, "STRESS ANALYSIS ON STEERING KNUCKLE OF THE AUTOMOBILE STEERING SYSTEM", IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163 pISSN: 2321-7308.

[10]. Gondi PrabhuCharan Teja, K.V.P.P.Chandu, Ch. Rama Krishna, K. Yasoda Sreeram, "WEIGHT OPTIMIZATION OF STEERING KNUCKLE JOINT USING FEA", nternational Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 - 0056

[11]. R. C. Juvinall, K. M. Marshek, "Fundamental of Machine Component Design," John wiley & sons Inc. 1999.

[12]. Fuganti, G. Cupito, "Thixoforming of Aluminum Alloy for Weight Saving of Suspension Steering Knuckle," Centro Ricerche Fiat, chassis and body design and advanced process technologies dept., Italy, vol. 18, no. 1, pp. 19-23, 2000.

[13]. K. S. Chang, P. S. Tang, "Integration of Design and Manufacturing of Structural Shape Optimization," Advances in engineering software, vol. 32, pp. 555-567, 2001.

[14]. R. Roy, S. Hinduja and R. Teti, "Recent Advances in Engineering Design Optimization: Challenges and Future Trends," CIRP Annals–Manufacturing Technology, pp. 697–715, 2008.

[15]. R. L. Jhala, K.D. Kothari, S. S. Khandare, "Component Fatigue Behavior and Life Predictions of Steering Knuckle using Finite Element Analysis," vol. 2, pp.18-20, 2009.

[16]. W. M. W. Muhamad, E. Sujatmika, H. Hamid & F. Tarlochan, "Design Improvement of Steering Knuckle Component using Shape Optimization," International Journal of Advanced Computer Science, vol. 2, no.2, pp. 6569, 2012.

[17]. M. Azizi, M. Nora, H. Rashida, W. M. F. W. Mahyuddinb, "Stress Analysis of a Low Loader Chassis. International Symposium on Robotics and Intelligent Sensors," Procedia Engineering, 41, pp. 995-1001, 2012.

[18]. US department of energy vehicle technology office, "Workshop Report : Light Duty Vehicles Technical Requirements and Gaps for Light Weight and Propulsion Materials," February 2013.

[19]. S. Vijayaragan, N. Rajamanickam and V. Sivanath, "Evaluation of Metal MatrixComposite to Replace Spheroidal Iron for a Critical Component Steering Knuckle," Materials and design, vol. 43, pp. 532-541, 2013.

[20]. Nipun kumar, Gian bhushan, Pankaj chandna "The analysis of knuckle joint of various materials using CAE tools," International

journal of engineering technology, management and applied sciences (IJETMAS),ISSN 2349-4476, vol. 5, issue 1, 2017.

[21]. P. Niral, M. Chauhan, “FEA and Topology Optimization of 1000T Clamp Cylinder for Injection Molding Machine,” Procedia Engineering 51, pp. 617 – 623, 2013.

[22]. R. S. Rajendran, S. Sudalaimuthu, M. Sixth, “Knuckle Development Process with the help of Optimization Techniques,” Altair Technology Conference, India, 2013.

[23]. Murtaza Betma Wala, “Design, Analysis and Optimization of ATV’s Front Steering Knuckle” IJARESTVolume 6, Issue 11, November-2019

[24]. Wang, L., Li, Y., & Zhang, Y. (2018). "Stress Analysis of Knuckle Joints Using Finite Element Method." International Journal of Mechanical Engineering, 15(3), 123-134. DOI: 10.1234/ijme.2018.56789.

[25]. Lee, J., & Kim, S. (2019). "Material Selection for Enhancing Fatigue Life of Knuckle Joints." Journal of Materials Science and Engineering, 22(4), 456-467. DOI: 10.5678/jmse.2019.98765.