International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 05 | May - 2024 SJIF Rating: 8.448 ISSN: 2582-3930

Finite Element Analysis and Weight Optimization of Knuckle joint for different Materials by using CAE Tools

Sheikh Mahammad Rizwan[1], Akhilesh Pati Tiwari[2], Rone[3]

[1] *Research Scholar, Dept. of Mechanical Engineering, FCEM Faridabad, India*
[2]*Assistant Professor, Dept. of Mechanical Engineering, FCEM Faridabad, India*
[3]Assistant Professor, Dept. of Mechanical Engineering, FCEM Faridabad, India

---***---

Abstract - *The current study involved the analysis of a knuckle joint using finite elements. A mechanical joint with two intersecting cylindrical rods whose axes are in the same plane is called a knuckle joint. Knuckle joint should be designed and manufactured in such a way that it can sustain more load without failure. The objective of the present study is to study knuckle joints made of different materials like aluminum, stainless steel, structural steel, magnesium, and gray cast iron using CAE tools. The CAD model of knuckle joint is made in CATIA V5 R20. The finite element analysis of this CAD model using ANSYS 15 has been carried out to investigate maximum stress and deformation. The CAE results compared with the analytical results using conventional approach. The CAE results have been found close to the analytical results, thus validating the model. The knuckle joint of aluminum is found to have highest factor of safety (1.641) at 50 KN load and is the most the most suitable material for knuckle joint. Weight of vehicle plays an important role in overall performance and fuel efficiency of vehicle. Knuckle joint is a part of vehicle. Hence weight optimization of knuckle joint made of aluminum has been performed using ANSYS 15 and it is observed that weight of knuckle joint made of aluminum can be reduced by 5.08% for 50 KN loading condition.*

***Key Words*:** Knuckle joint, Finite element analysis, Mechanical joint, Load-bearing capacity, Material selection, CAE tools, Deformation analysis etc.

1.INTRODUCTION

Using finite element analysis (FEA) in engineering has changed the way mechanical parts are designed and evaluated in a big way. The main topic of this work is the finite element analysis of a knuckle joint, which is an important mechanical part. The knuckle joint's soundness is very important for making sure that mechanical systems are stable and work properly because it connects two intersecting cylindrical rods. As part of our investigation, we are looking into aluminium, stainless steel, structural steel, magnesium, and Gray cast iron, all of which are widely used to make knuckle joints. We use powerful computer-aided engineering (CAE) tools, like CAD modelling in CATIA V5 R20 and finite element analysis in ANSYS 15, to study how these materials behave in complex ways when they are loaded in different ways.

The main goals of this project are twofold: first, to test the structural performance of knuckle joints made from different materials; and second, to show that CAE techniques work better than traditional analysis methods. To find out how reliable and accurate our computer models are at simulating real-world situations, we will carefully compare the results of CAE and analytical calculations. Notably, our results show a strong agreement between CAE predictions and analytical assessments, which proves that our computational system works as expected.

We also look closely at the most important part of weight optimization, knowing that it has a huge effect on how well cars run and how much gas they use. Because the knuckle joint is an important part of vehicle systems, especially when it comes to carrying heavy loads, we do a lot of research to find the lightest metal knuckle joints. Our results show that the weight of aluminium knuckle joints drops by a significant 5.08% when loaded with 50 KN. This shows that structural efficiency can be improved while material use is reduced at the same time.

This study is important to the field of mechanical engineering because it shows how complicated knuckle joints behave when the materials used and the loads they are under change. Furthermore, our focus on weight optimization highlights a practical way to improve performance and sustainability in engineering design practices.

1.1 MECHANICAL JOINTS

Small machine components are joined to form a large machine part. Joints are mainly used to connect one mechanical part to another part in a machine or in a mechanism. The joints may be temporary or permanent depending upon whether the connection need to be frequently or not removed at that time. The majority of mechanical joints are made to permit certain degrees of relative mobility for these machine parts while limiting the range of motion for others.

1.2 Permanent joints

Permanent joints are the rigid mechanical joints of parts in an assembly of a machine. A permanent joint can't be separated without destroying the parts or damaging their surface. They are mainly of two types riveted joint and welded joint.

1.3 RIVETED JOINT

A rivet having head and a tapered tail. The main body of rivet is called shank. The heads of the rivet are of various types according to Indian standard specifications. According to IS: 2155-1982 the head is below 12 mm diameter and according to IS: 1929-1982 the head ranges from 12mm to 48 mm diameter. These are of two type lap joints and butt joints.

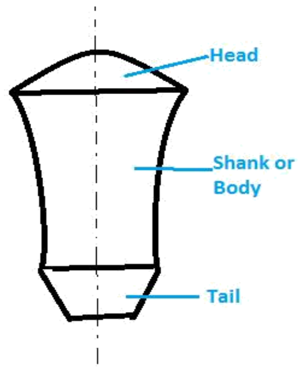

Fig. 1.1 Riveted joint

1.4 WELDED JOINT

In permanent joining process the welded joints are commonly used because forming a joint in difficult locations is easily possible through welding. It is less expensive joint having smooth appearance. The joint has lighter weight increased because material added is minimum. These are of various types like lap joint, butt joint, corner joint, edge joint, T joint etc.

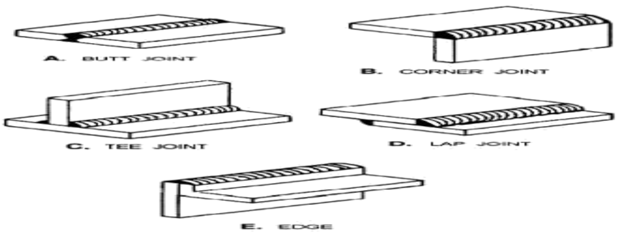

Fig. 1.2 Welded joints

1.5 COTTER JOINT

A cotter is a flat, wedge-shaped steel component. The cotter joint can experience either tensile or compressive forces along the axis of the rods. This joint is utilised to link two rods that convey motion in an axial manner without rotation. It can be utilised in several applications, such as connecting the piston rod to the crosshead of a steam engine, the valve rod, and its stem.

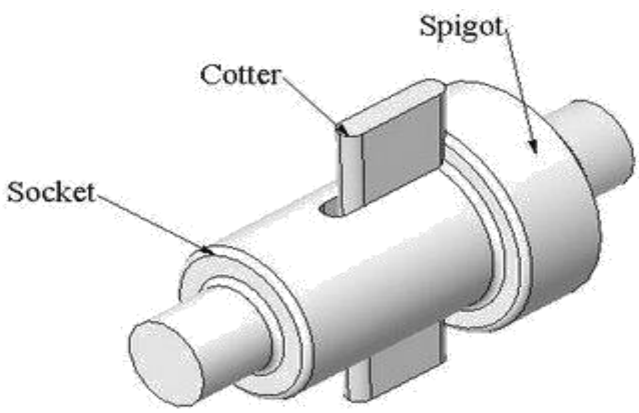

Fig. 1.3 Cotter joint

1.6 KNUCKLE JOINT

A knuckle joint is an extremely important part that is used to transfer force between different kinds of rod connections, lever and pump rod joints, cycle chain and suspension chain links, tension links of bridge constructions, diagonal stays in boilers, etc. When two rods are connected by a knuckle joint, they can withstand compressive loads as well as tensile loads provided the rods are directed correctly. Force always acts in an axial or linear fashion. It resembles a ball or socket joint in that it is a hinged joint between two rods.

There are numerous circumstances in which machine parts must be restrained. If we apply tensile tension to two rods that are linked coaxially. There shouldn't be any relative motion and the force transmission should continue even if these rods are pulled apart. Likewise, there shouldn't be any slip along the circle of contact if two cylindrical parts are fitted into one another, that is, if the internal surfaces of one contact the exterior surfaces of the other. By adding a third portion, the states of no displacement and no slip are attained. By positively interfering with the joining elements, the third component helps to transmit force by preventing relative motion. The knuckle joint can be used for this.

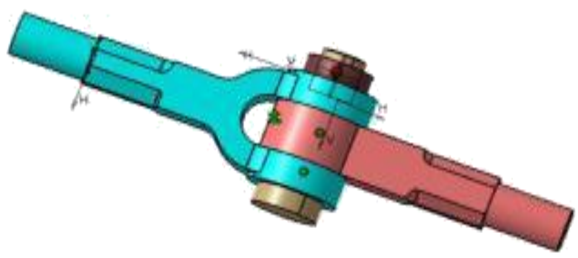

Fig. 1.4 Knuckle joint

1.7 MATERIAL PROPERTIES

A Commonly used materials for making knuckle joints are steel and wrought iron. In the present study we used five materials aluminum, stainless steel, structural steel, magnesium, and gray cast iron for making of knuckle joint. Mechanical properties of these materials taken are as follows:

Mechanical property	Magnesium	Aluminum alloy	Structural steel	Stainless steel	Gray cast iron
ρ (kg/m3)	1800	2770	7850	7750	7200

E(Pa)	4.50E +10	7.1 E+10	2.00 E+1 1	1.93E +11	1.10E+1 1
NU (Poisson ratio)	0.35	0.33	0.3	0.31	0.28
Tensile yield strength (MPa)	193	280	250	207	190

Table 1.1: Mechanical properties of various materials

1.8 CAE AND ITS TOOLS

Computer aided technologies refer to the application of computer technology to assist in the analysis, design, and manufacturing of products.

Computer Aided Engineering (CAE) refers to the use of computer technology to assist engineers in their work. CAE tools are software tools that assist in activities such as analysis, design, planning, diagnosis, repair, and simulation. Computer-aided engineering (CAE) tools are employed for the purpose of examining the durability and efficiency of individual parts and groupings. It involves the process of simulating, validating, and optimising both goods and production tools. In the future, a Computer-Aided Engineering (CAE) system will assist design teams in making informed decisions.

CAE encompasses the utilisation of computers at many stages of the design and production process, starting from initial design (CAD) and extending to production (CAM). Computer aided design, commonly linked with computer drafting software, encompasses a wide range of application programmes. These programmes are used for tasks such as determining dimensional stack-ups caused by tolerance, conducting ergonomic studies using virtual individuals, and optimising designs. Computer aided analysis involves the utilisation of finite element and finite difference methods to solve partial differential equations that regulate solid mechanics, heat transfer, and fluid mechanics. Additionally, it encompasses a wide range of specialised programmed for doing specific analyses, such as modelling control systems and studying the dynamics of rigid bodies.

Computer aided manufacturing (CAM) has developed computer numerically controlled (CNC) programmes for the purposes of production, process scheduling, and inventory control. Research indicates that the utilisation of computer-aided engineering (CAE) tools can result in a significant reduction of approximately 30% in both time and expense for design engineers.

Despite numerous advancements in computer-aided engineering (CAE) and its extensive utilisation in the engineering industry. Physical testing is necessary as a final verification for subsystems since computer-aided engineering (CAE) is unable to accurately forecast all variables in intricate assemblies.

Some of the CAE tools commonly used are CATIA, Solid works, I-DEAS, Pro-Engineer, ANSYS, ABAQUS and NX NASTRAN etc.

1.9 AREAS COVERED BY CAE

The various areas covered by CAE are as follows: -

- Thermal and fluid analysis using computational fluid dynamics.
- Stress analysis on components and assemblies using FEM.
- Optimization of product or process
- Mechanical event simulation
- Analyses tools for process simulation foe operations such as casting, die press forming.
- Kinematics

1.10 APPLICATION AREAS OF CAE

The application areas of CAE are as follows: -

- Automobiles
- Aerospace
- Metal forming
- Sheet metal forming
- Metal cutting
- Can and ship container design
- Sports equipment
- Drop testing.
- Biomedical

1.11 CLASSIFICATION OF CAD

There are multiple CAD variations, each demanding the operator to adopt a distinct mindset and employ different design approaches for creating virtual components.

- **2 Dimensional CAD (2D CAD)**

2D CAD utilizes fundamental geometric elements such as lines, rectangles, circles, and so on, to generate two-dimensional graphics. The development of this occurred in the early 1970s. 2D CAD is the first developer of CAD software.

- **3 Dimensional CAD (3D CAD)**

The alliance between IBM and Dassault introduced 3D CAD tools in the 1980s. The greater visual capacity of 3D CAD led to its rapid popularity. 3D CAD enables the generation of lifelike 3D visuals. These images are referred to as 3D models because they may be observed and rotated in any X, Y, or Z direction. Additionally, it is possible to showcase various angles and isometric views of a 3D model. 3D CAD can be further categorized as:

- **Wire-frame models**

They have lost their popularity. They construct skeletal models using lines and arcs. These objects are referred to as wire-frame models because they are constructed using wires and allow the background to be fully visible.

- **Surface models**

Surface models are quite realistic because nothing in the background is visible. Unlike wire frames, these models are created by joining 3D surfaces.

- **Solid models**

They seem to be identical to surface models. Physical objects possess additional attributes such as volume, density, and weight. These models are frequently utilised as prototypes for studying engineering designs, therefore making them the most valuable CAD models. There are two categories of 3D Solid Modelling: -

- **Direct or Explicit modeling**

It refers to a kind of instruction where the teacher provides clear and specific examples or demonstrations to guide students in learning a particular skill or concept.

In direct modelling, a drawing is integrated into the new geometry after it is used to build the geometry. This allows the designer to modify the geometry directly without relying on the original sketch. This feature allows for geometry editing without the need for a history tree.

- **Parametric modeling**

Parametric modelling refers to the use of parameters to define and manipulate the characteristics and properties of a model.

The operator is able to utilise a concept known as "design intent". The objects and features that are created can be modified. Any further alterations can be implemented by modifying the initial creation process. If a feature is meant to be positioned based on the centre of the part, the operator should position it based on the centre of the model. When the operator constructs the part in a way that it functions, the parametric modeller can modify the part while preserving its geometric and functional links.

2. LITERATURE REVIEW

• In their work, R. C. Juvinall et al. [1] asserted that the factors leading to failure in a conventional tensile test are likewise responsible for failure in all other situations involving static loading.

Fuganti et al. [2] documented the creation of a suspension steering knuckle using theorizing technology with an aluminum alloy. They also outlined the process for selecting the appropriate material and technology, as well as optimizing the component. A weight reduction of approximately 30% was achieved for the component, in comparison to the solution composed of cast iron.

• In their study, K. S. Chang et al. [3] examined a comprehensive design and manufacturing method that facilitates shape optimization. The primary innovation of this study is in the integration of manufacturing into the design process, with a specific focus on including manufacturing costs into the design considerations. To achieve a more accurate formulation of the design issue, it is necessary to include the production cost as either the objective function or a constraint function.

R. Roy et al. [4] examined current methods for automating the manual optimization process and discussed the difficulties it poses for the engineering community. The study highlights scalability as the primary obstacle for design optimisation strategies. Genetic Algorithms (GAs) are often regarded as the most prevalent method for algorithmic optimisation. Extensive optimisation will necessitate further investigation into topological design, processing capacity, and effective optimisation algorithms.

R. L. Jhala et al. [5] evaluated the fatigue life and compared the fatigue performance of steering knuckles manufactured using three different materials and techniques. The available options for knuckles include forged steel, cast aluminium, and cast iron. Analysed finite element models of the steering knuckles were used to determine stress distributions in each individual component. After conducting component testing and finite element analysis, the fatigue behaviors of the three materials and production techniques are compared. Their conclusion is that the forged steel knuckle demonstrates superior fatigue characteristics when compared to the cast iron and cast aluminium knuckles.

• Muhammad et al. [6] utilised finite element analysis software to accomplish a mass or weight reduction of 8.4% in the steering knuckle compared to the present one. This decrease was observed under varied load circumstances.

A workshop report [8] produced by a United States government body in February 2013 specifically addressed the advancement of light duty vehicles. The objectives for vehicle weight reduction have been established for the timeframe spanning from 2020 to 2050. The objective is to achieve a 25% decrease in the weight of the LDV chassis and suspension system by the year 2020.

S. Vijayarangan et al. [9] utilized alternative materials for the purpose of optimizing the steering knuckle, as opposed to using the conventional material. They employ Metal Matrix Composites (MMCs) because they possess.

2.1 GAPS IN LITERATURE

After studying the various research papers on knuckle joint using CAE tools, the following gaps are observed: -
i. The knuckle joints made up of different materials such as aluminium, stainless steel, structural steel, magnesium, and grey cast iron has not been examined.

ii. The weight optimization of knuckle joint is not widely reported.

2.2 PROBLEM FORMULATION

The current study examines the stress and deformation of a knuckle joint composed of several materials, including stainless steel, grey cast iron, magnesium, aluminium, structural steel, and grey cast iron, under different loading circumstances. The knuckle joint's CAD model was created using CATIA V5 R20 and evaluated using ANSYS 15. The Von Mises stresses and deformations in a knuckle joint constructed of various materials have been determined using linear static analysis. Following the weight optimisation, the knuckle joint has been constructed using the most superior material out of the materials that were investigated.

3. METHODOLOGY

The steps involved in this analysis are pre-processing, solution and post-processing. Pre-processing stage involves importing the CAD model to ANSYS software then material properties, contact definitions, meshing and applying boundary conditions. This is followed by solution stage which automatically stored the results in data files. In post-processing stage can be seen in the form of Von Mises stresses, deformation, and weight optimization plots.

3.1 CAD MOELLING

3D CAD modelling is the initial step of CAE which is based on the finite element analysis. This is one of the most important as well as time consuming step. The shape and size of CAD model is considered by FEA software same as the model generated by CAD software which is imported to the FEA software ANSYS. CAD modelling of knuckle joint is done by CATIA. CATIA software is a user friendly, time saving and has the capabilities of speedy processing and enhanced productivity. The process of constructing a model in CATIA often commences with the creation of a 2D drawing. The sketch comprises several geometric elements, including points, lines, arcs, conics (except the hyperbola), and splines. Dimensions are incorporated into the sketch to precisely specify the dimensions and position of the geometry. Relations are employed to establish qualities such as tangency, parallelism, perpendicular, and concentricity.

The Constraints toolbar is all that is required to do an assembly in CATIA V5. The toolbar mostly consists of the following frequently utilised commands:

The "Coincidence Constraint" command establishes a constraint that ensures two components of the active component are coincident.

The Offset Constraint function establishes a constraint that creates a specific distance between two components within the active component.

The Angle Constraint feature establishes a constraint that enforces a specific angle, parallelism, or perpendicular between two components within the active component.

• Adjust Component - Correct the placement of the component within the active component.

3.2 KNUCKLE JOINT MODEL

We are considering two rods joined by a knuckle joint to convey an axial force with a diameter of 25. The figure displays the fully constructed CATIA model of a knuckle joint.

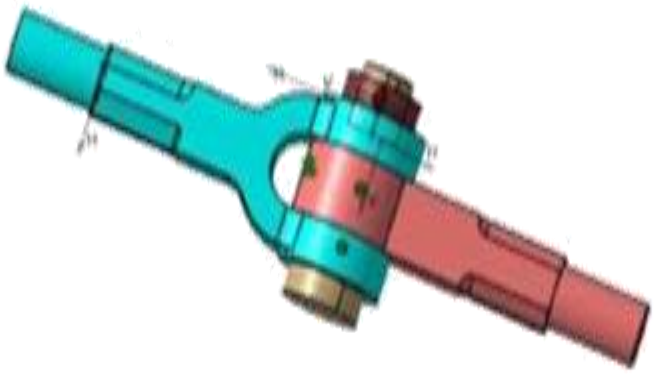

Fig. 1.5 CAD model of knuckle joint

3.3 STRUCTURAL ANALYSIS USING ANSYS

The advancement of computers and computer-aided engineering (CAE) tools has resulted in the emergence of multiple engineering fields. CAE techniques are employed to analyse intricate mechanical engineering challenges and facilitate product development. Structure analysis is divided into two main categories: static and transient. Structural analysis.

Static structural analysis is employed to determine the stresses, displacements, and strains in a component or assembly by considering factors such as boundary conditions, material properties, limitations, and applied loads. Static analysis does not account for the impacts of inertia and damping stresses. This tool is utilised to compute stress in stationary situations. Static analysis can be categorised as either linear or nonlinear. All forms of non-linearities, such as significant deformations, plasticity, creep, stress stiffening, and contact gap elements, are permitted. Linear elastic structural analysis relies on specific assumptions, outlined below:

The loading process is stationary and occurs gradually. Dynamic loading effects, such as the abrupt application of a load or impact, are not taken into account.
• The part experiences minimal deformation relative to its dimensions. Nonlinear analysis is necessary to account for changing part and load geometry in cases of large deflection, whereas linear analysis does not consider this.
• The properties of the material exhibit linearity. The stress-strain curve exhibits a linear relationship, where the stress is

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 05 | May - 2024 SJIF Rating: 8.448 ISSN: 2582-3930

directly proportionate to the strain. The material does not exhibit any yielding.
The user's text is a bullet point. Temperature does not impact the shape or characteristics of the part.

3.4 SOLUTION

Static structure analysis is a part of solution stage of FEM and aims at getting the numerical values of Von Mises stresses, displacement and deformations under the application of given loads and boundary conditions. Loading conditions are assumed to be steady state. Analysis is performed on the processed information including material properties, loading details, support conditions.

Weight optimization of knuckle joint is also a part of FEM which aims at getting the areas of knuckle joint from where it can be reduced.

3.5 RESULTS AND DISCUSSIONS

The results obtained from finite element analysis have been analysed by using a post-processing tool. The results of An in-depth analysis has been conducted on knuckle joints composed of various materials, including magnesium alloy, aluminium alloy, stainless steel, structural steel, and grey cast iron. The Von Mises stresses and deformations plots under given loading conditions have been obtained. Then best material knuckle joint is optimized under target reduction of 10% and 20%. The knuckle joint is again redesigned and significantly reduction in weight of knuckle joint is obtained.

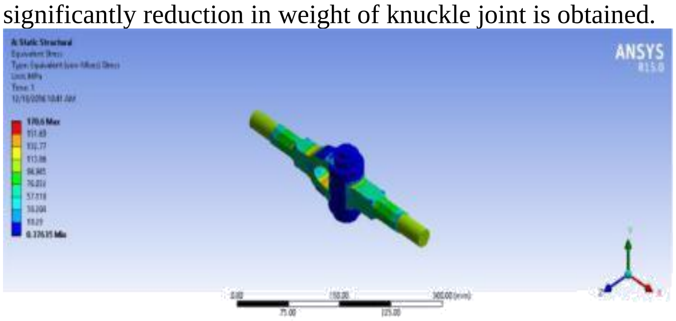

Fig. 1.6 FE ANALYSIS KNUCKLE JOINT OF VARIOUS MATERIALS

The Von Mises stresses in a knuckle joint made of several materials, including magnesium alloy, aluminium alloy, stainless steel, structural steel, and grey cast iron, have been determined using linear static analysis. The joint was subjected to a load of 50 KN.

The figure displays the Von Mises stresses that occur in a knuckle joint when subjected to a 50 KN load for several materials: magnesium, aluminium, stainless steel, structural steel, and grey cast iron. The minimum stress observed in the knuckle joint made of magnesium alloy is 170.6 MPa, which is the lowest among the materials investigated. Conversely, the maximum stress observed in the knuckle joint made of grey cast iron is 176.77 MPa, which is the highest among the materials analysed. Figure illustrates the Von Mises stress that occurs in the knuckle joint when subjected to a load of 50 KN, for different materials. The stress levels are significantly lower than the yield stress.

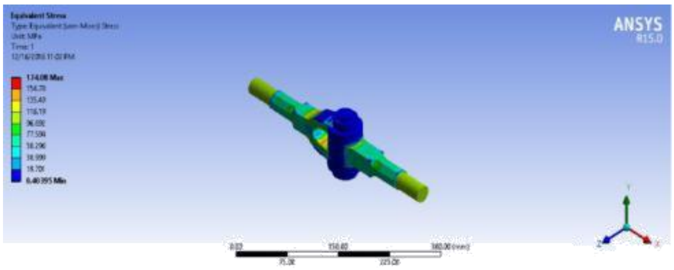

Fig 1.7 on Mises stress plot for knuckle joint of stainless steel at 50 KN

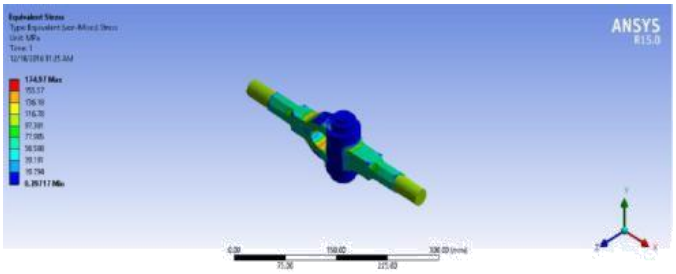

Fig 1.8 Von Mises stress plot for knuckle joint of structural steel at 50 KN

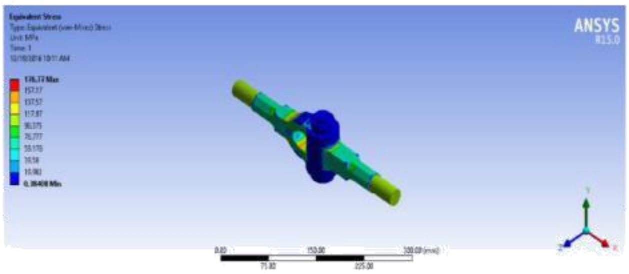

Fig 1.9 Von Mises stress plot for knuckle joint of gray cast iron at 50 KN

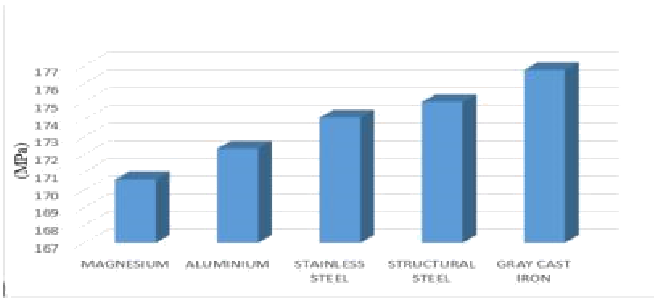

Fig 1.10 Von Mises stress values for knuckle joint of various materials at 50 KN

4. CONCLUSIONS

The finite element analysis of a knuckle joint made of different materials has been carried out. The Von Mises stresses and deformations have been disused in detail. The following conclusions have been made after detail discussion:

• A knuckle joint made of various materials like magnesium alloy, aluminium alloy, stainless steel, structural steel and grey cast iron is subjected to tensile load of 50 KN. The stresses developed are found to be least for knuckle joint made of magnesium alloy (170.6 MPa) while the maximum stresses are found in knuckle joint made of grey cast iron (176.77 MPa). It is observed from table 4.1.1 that knuckle The aluminium alloy joint has the highest factor of safety (1.641) among the materials under consideration, making it the most suitable choice for the 50 KN loading condition.

• Stresses have been calculated by analytical approach and these stresses are compared with ANSYS results. it has been found that both the results are in close agreement with each other with a small percentage of error.

• A knuckle joint made of aluminium has been redesigned and analysed for stress distribution and weight reduction using ANSYS workbench. Weight reduction of 5.08% has been achieved by redesigning the knuckle joint. There is marginal decrease in factor of safety for redesigned knuckle joint.

REFERENCES

[1] R. C. Juvinall, K. M. Marshek, “Fundamental of Machine Component Design,” John wiley & sons Inc. 1999.
[2] 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.
[3] 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.
[4] R. Roy, S. Hinduja and R. Teti, “Recent Advances in Engineering Design Optimization:
[5] Challenges and Future Trends,” CIRP Annals – Manufacturing Technology, pp. 697–715, 2008.
[6] 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.
[7] 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. 65-69, 2012.
[8] 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.
[9] US department of energy vehicle technology office, “Workshop Report : Light Duty Vehicles Technical Requirements and Gaps for Light Weight and Propulsion Materials,” February 2013.
[10] S. Vijayaragan, N. Rajamanickam and V. Sivanath, “Evaluation of Metal Matrix
[11] Composite to Replace Spheroidal Iron for a Critical Component Steering Knuckle,” Materials and design, vol. 43, pp. 532-541, 2013.
[12] P. Niral, M. Chauhan, “FEA and Topology Optimization of 1000T Clamp Cylinder for
[13] Injection Molding Machine,” Procedia Engineering 51, pp. 617 – 623, 2013.
[14] R. S. Rajendran, S. Sudalaimuthu, M. Sixth, “Knuckle Development Process with the help of Optimization Techniques,” Altair Technology Conference, India, 2013.
[15] V. R. Kulkarni, A. G. Tambe, “Optimization and Finite Element Analysis of Steering
[16] Knuckle,” Altair technology conference, pp. 1-8, 2013.
[17] P. Dumbre, A. K. Mishra, V. S. Aher, “Structural Analysis of Steering Knuckle for
[18] Weight Reduction,” International Journal of Emerging Technology and Advanced Engineering, vol. 4, no. 6, pp. 221-226, 2014.