ISSN – 2348-2397

APPROVED UGC CARE

SHODH SARITA

Vol. 4, Issue 29 January to March 2021

Page Nos. 241-248

AN INTERNATIONAL BILINGUAL PEER REVIEWED REFEREED RESEARCH

DESIGN AND ANALYSIS OF UNIVERSAL JOINT FOR EFFECTIVE TRANSMISSION AND DECIDE MATERIAL TYPE

Sake Vijayanirmala

*Assistant Professor, Department of Mechanical Engineering, St. Martin's Engineering College, Hyderabad, Telangana, India

ABSTRACT:

The universal joint is considered to be one of the oldest of all flexible couplings. It is generally known for its utilization on vehicles and trucks. An all universal joint in its least complex structure comprises of two shaft yokes at right angle to each other and a four point cross which interfaces the yokes. The cross rides inside the bearing cap assemblies , which are squeezed into the yoke eyes. Industrial applications work continuously and with high torque loads. This requests most extreme quality and long existence of the all universal joint segments. The advanced all universal joint has turned out to be significantly more mind boggling than its straightforward predecessor.

The aim of project is to design and perform structural analysis on universal joint to find out stresses induced in it.

INTRODUCTION

A universal joint or Hooke's joint is a joint or coupling in a rigid rod that allows the rod to bend in any direction, and is commonly used in shafts that transmit rotary motion. It generally consists of two hinges located close together, oriented at 900 to each other, connected by a cross shaft. It is widely used in industrial applications and vehicle drivelines to connect misaligned shafts. A major problem with the use of a Hooke's joint is that it transforms a constant input speed to a periodically fluctuating one. The kinematical consequences of this property of this joint can be remedied, as long as rigid body rotations are concerned, by using two converse Hooke's joint .But if torsion vibrations of the propeller shaft are concerned, there is no way of removing the dynamical consequences of an introduced Hooke's joint in a rear wheel drive vehicle. In a widely used single piece drive shaft, two universal joints are used Universal joint and drive shaft assembly two universal joints are preferred in order to avoid the transformation of constant input speed into a fluctuating speed which is encountered when a single universal joint is used. Now a day's more emphasis is being given on reducing the weight of the drive shaft. It is being tried to replace the existing Steel shafts with Aluminium alloy shafts as it has a higher strength to weight ratio. Attempts are also being made to replace conventional steel shafts with hybrid aluminum composite drive shafts.

The main concept of the universal joint is based on the design of gimbals, which have been in use since antiquity. One anticipation of the universal joint was its use by the Ancient Greeks on ballistae. The first person known to have suggested its use for transmitting motive power was Gerolamo Cardano, an Italian mathematician, in 1545, although it is unclear whether he produced a working model. In Europe, the device is often called the Cardan joint or Cardan shaft. Christopher Polhem of Sweden later reinvented it, giving rise to the name Polhemsknut in Swedish.

The mechanism was later described in Technical curiosa sive mirabilis artist (1664) by Gaspar Schott, who called it the paradoxum, but mistakenly claimed that it was a constant-velocity joint. Shortly afterwards, between 1667 and 1675, Robert Hooke analysed the joint and found that its speed of rotation was no uniform, but that this property could be used to track the motion of the shadow on the face of a sundial.[2] In fact, the component of the equation of time which accounts for the tilt of the equatorial plane relative to the ecliptic is entirely analogous to the mathematical description of the universal joint. The first recorded use of the term universal joint for this device was by Hooke in 1676, in his book Helios copes. He published a description in 1678,]resulting in the use of the term Hooke's joint in the English-speaking world. In 1683, Hooke proposed a solution to the no uniform rotary speed of the universal joint: a pair of

Hooke's joints 90° out of phase at either end of an intermediate shaft, an arrangement that is now known as a type of constant-velocity joint.

The term universal joint was used in the 18th century and was in common use in the 19th century. Edmund More wood's 1844 patent for a metal coating machine called for a universal joint, by that name, to accommodate small alignment errors between the engine and rolling mill shafts. Lardner's 1877 Handbook described both simple and double universal joints, and noted that they were much used in the line shaft systems of cotton mills.[6] Jules Weisbach described the mathematics of the universal joint and double universal joint in his treatise on mechanics published in English in 1883.

19th century uses of universal joints spanned a wide range of applications. Numerous universal joints were used to link the control shafts of the Northumberland telescope at Cambridge University in 1843. Ephraim Shay's locomotive patent of 1881, for example, used double universal joints in the locomotive's drive shaft.] Charles Amid on used a much smaller universal joint in his bit-brace patented 1884. Beauchamp Tower's spherical, rotary, high speed steam engine used an adaptation of the universal joint circa 1885.

The term Cardin joint appears to be a latecomer to the English language. Many early uses in the 19th century appear in translations from French or are strongly influenced by French usage. Examples include an 1868 report on the Exposition Universal of 1867 and an article on the dynamometer translated from French in 1881.

LITERATURE REVIEW

Early articles on universal joints made of rigid links address various aspects of these mechanisms. Basically a universal joint is a spherical four bar linkage. In literature there are lots of studies about this type of mechanism addressing its analysis, synthesis, applications and type determination. For example Mohan et al. (1973) introduced closed form synthesis of a spatial function generation mechanism which consists of a spherical four bar linkage. Freudenstein (1965) proposed a new type of a spherical mechanism. Yang (1965) worked on static force and torque analysis of a spherical four bar mechanism. Dynamic analysis of a universal joint and its manufacturing tolerances are introduced by Chen and Freudenstein (1986). Freudenstein and Macey (1990) worked on the inertia torques of the Hooke joint. Moment transmission by a universal joint is studied by Porat (1980). Homokinetic joint allows to transmit power through a variable angle, at constant rotational speed. For a double cardanhomokinetic joint Wagner and Cooney (1979) developed a new approach to increase its dynamic mechanical efficiency. Universal joint has the advantage of easiness in manufacturing. On the other hand traditional universal joints consist of many parts which are assembled and therefore manufacturing tolerances on these parts must be complied with. Tolerances of a universal joint are studied by Fischer and Freudenstein (1984).

Compliant mechanisms are flexible mechanisms, which gain some or all of their motion through the deflection of members. They can be fully or partially compliant. Generally compliant mechanisms have lower number of parts which reduce manufacturing and assembly time. Some of them may even be made of a single piece. They are lighter and they have fewer number of movable joints, which cause wear and need lubrication. The main disadvantage of compliant mechanisms is that, their analyzes and design is difficult to accomplish. The pseudo rigid body model is used to simplify the analysis and design of compliant mechanisms. In Figure 1.1 first compliant spatial four bar mechanism is shown which is designed by Tanık and Parlaktaş (2011). Salamon (1989) introduced a methodology which uses a pseudo rigid body model of the compliant mechanisms with compliance modelled as torsional and linear springs. Howell and Midha (1994) and (1998) used closed form elliptic integral solutions to develop deflection approximations for an initially straight flexible segment subjected to bending. A spherical four bar mechanism which is a special case of spatial four bar mechanism that possesses out of plane motions is studied by Tanık and Parlaktaş (2012). Recently, Tanık and Parlaktaş (2012) proposed a new design for a compliant cardan universal joint which is shown in Figure 1.2. The design consists of two identical parts assembled at right angles with respect to each other. In that study, dimensions of the mechanism are designed in order to satisfy the Cardan joint theory and to avoid an undesired contact between the identical parts for proper functioning of the mechanism. This prototype is made of polypropylene and manufactured and operated under specified loading conditions to verify the theoretical approaches.

OBJECTIVE AND SCOPE:

The successful implementation of a compliant universal joint in real life applications depends not only on its kinematic design but its strength as well. The purpose of this thesis is therefore to analyze the stresses and fatigue strength of a compliant universal joint, whose flexible parts are made of blue polished spring steel by analytical, numerical and experimental methods. Hence theoretical approaches will be experimentally verified. To the best of Author's knowledge there are not any studies in the literature which address the strength issues of compliant universal joints except the study of Tanık and Parlaktaş (2012) , where only a preliminary 5 finite element analysis has been done to determine the torque capacity of the mechanism.

Here, a design is proposed according to the dimensional constraints that satisfy the theory of universal joints and thereby avoid undesired contact between the parts. The stress analysis is done analytically and numerically by finite element method using ANSYS software.

Fig1: Fork of Universal Joint in CATIA

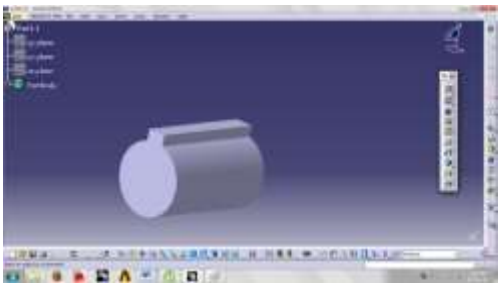

Fig2: Shaft of Universal Joint in CATIA

Fig3: Center block of Universal Joint in CATIA

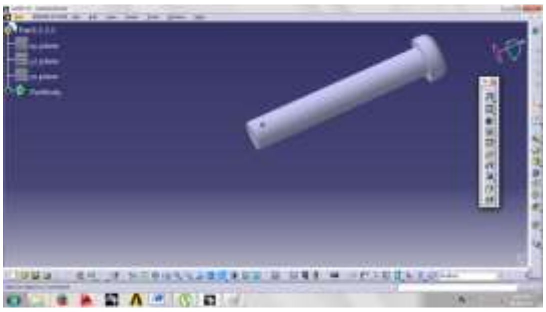

Fig4: Pin of the Universal Joint in CATIA

Fig5: Taper Pin of Universal Joint in CATIA

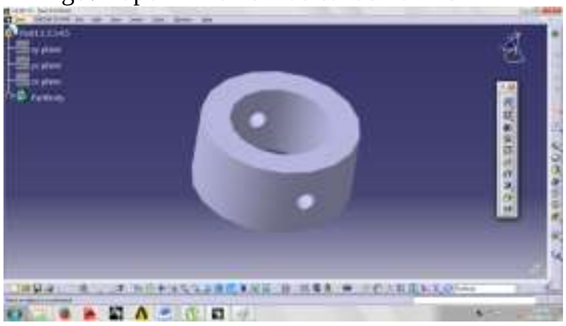

Fig 6: Coller of Universal Joint in CATIA

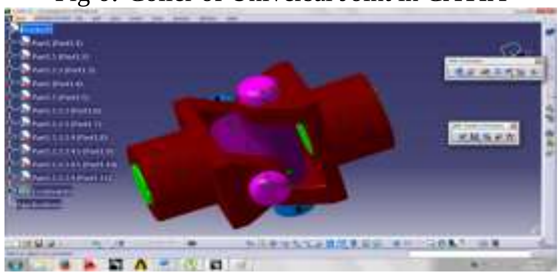

Fig 7: Basic Universal Joint in CATIA

FINITE ELEMENT ANALYSIS

The essential idea in fem is that the body or structure may be separated into littler components of limited measurements called "Finite Elements". The first body or the structure is then considered as a gathering of these components associated at a limited number of joints called "nodes" or "nodal points"

Basic capacities are approximated the relocations over each limited component. Such accepted capacities are called "shape capacities". This will speak to the uprooting within the component as far as the remit Element technique is a scientific apparatus for illuminating common and fractional at the hubs of the components. The Final differential comparison in light of the fact that it is a numerical instrument, it can take care of the unpredictable issue that can be spoken to in differential mathematical statement from. The use of FEM is boundless as respects the arrangement of commonsense configuration issues.

FEM has good efficiency to solve problems and cost critical problems as the cost for computing power is high.

The finite element method can be utilized to solve problems in the following areas:

- Structural analysis
- Thermal analysis
- Vibrations and dynamics
- Buckling analysis
- Acoustics
- Fluid flow simulations
- Crash simulations
- Mould flow simulations

To calculate the area of circle without using conventional formula, one of the approach could be dividing the area into number of equal segments. The area of each triangle multiplied by the number of such segments gives the total area of the circle.

MORE ABOUT FEA:

Finite Element Analysis was first initially produced for use in the nuclear and aerospace industries where the safety of the structures is critical. Today, the growth in usage of the method is directly attributable to the rapid advances in computer technology in recent years. As a result, not just structural analysis most sophisticated problems can also be solved. Be that as it may, utilized for a wide variety of uses for example, consistent state and transient temperature appropriations, liquid stream reenactments furthermore recreation of assembling procedures, for example, infusion mounding and metal framing.

FEA comprises a computer model of a material or design that is analyzed by applying the loads for specified results. It is utilized as a part of new item plan, and existing item refinement. An outline specialist should have the capacity to confirm the proposed plan, which is planned to meet the client prerequisites preceding the assembling. For example adjusting the outline of a current item or structure so as to qualify the item or structure for another administration condition.

Can likewise be proficient if there should arise an occurrence of auxiliary disappointment, FEA may be utilized to decide the configuration adjustments to meet the new condition.

THE BASIC STEPS INVOLVED IN FEA:

Numerically, the structure to be examined is subdivided into a cross section of limited estimated components of straightforward shape. Inside of every component, the variety of dislodging is thought to be dictated by basic polynomial shape capacities and nodal relocations.

Comparisons for the strains and hassles are created as far as the obscure nodal relocations. From this, the mathematical statements of the balance are amassed in a grid from which can be effortlessly being customized and illuminated on a PC. In the wake of applying the proper limit conditions, the nodal relocations are found by understanding the framework firmness mathematical statement.

Once the nodal relocations are known, component hassles and strains can be figured

BASIC STEPS IN FEA:

- Discretization of the domain.
- Applying the boundary conditions.
- Assembling the system equations.
- Solution for system equations.
- Post processing the results.

DISCRITIZATION OF THE DOMAIN:

The task is to divide the continuum under study into a number of subdivisions called element. Based on the continuum it can be categorised into line or area or volume elements.

APPLICATION OF BOUNDARY CONDITIONS:

From the physics of the problem we have to apply the field conditions i.e. loads and constraints, which will help us in solving for the unknowns.

SYSTEM EQUATIONS ASSEMBLING:

The formulation of respective characteristic (Stiffness in case of structural) equation of matrices and assembly is involved in this.

SOLUTION FOR SYSTEM EQUATIONS:

Solve the equations to know the unknowns. This is basically the system of matrices which are nothing but a set of simulations equations are solved.

VIEWING THE RESULTS:

After the completion of the solution we have to review the required results. The first two steps of the above said process is known as pre-processing stage, 3rd and 4th steps are the processing stage and the final step is known as post-processing stage.

ANSYS:

The ANSYS program is self-contained general purpose finite element program. This is developed and maintained by Swason analysis systems Inc.

ANSYS finite element analysis software enables following tasks:

- Apply design performance conditions or other operating loads.
- Build computer model or transfer models of structures, components, products, or system.
- Testing prototype in environments where it otherwise would be impossible or undesirable.
- Studying physical responses such as temperature distributions, stress levels or electromagnetic fields.
- Reducing the productions cost by optimizing design early in the development process.

The ANSYS project has a compressive graphical client interface (GUI) that gives clients simple, intelligent access to program capacities, orders, documentation and reference material. A natural menu framework offers clients some assistance with navigating through the ANSYS program. Clients can enter information utilizing a mouse, a console, or a blend of both.

A graphical client interface all through the project, to direct new clients through the learning process and furnish more experienced clients with different windows, draw down menus, dialog boxes, apparatus bar and online documentation.

ORGANIZATION OF THE ANSYS PROGRAM

The ANSYS program is organized into two basic levels:

- begin level (Start level)
- Processor (or routine) level

Begin level acts as a gateway into and out of the ANSYS program. Changing the name of job, database clearing, and binary files copying are program controls used. When we first enter the program, we at the begin level.

At the processor level, several processors are available; each processor is a set of functions that specific analysis task perform. For instance, the general preprocessor (PREP7) is the place we fabricate the model, the arrangement processor (SOLUTION) is the place we apply stacks and get the arrangement, and the general postprocessor(POST1) is the place we assess the outcomes and acquire the arrangement. An extra postprocessor (POST26), empowers we to assess arrangements results at particular focuses in the model as an element of time.

PERFORMING A TYPICAL ANSYS ANALYSIS

The ANSYS system has numerous limited component investigation capacities, extending from a straightforward, direct, static examination to a nonlinear, transient element investigation. The investigation guide manuals in the ANSYS documentation set portray particular systems for performing examination for diverse building controls. A typical ANSYS analysis has three distinct steps:

- construct the model
- Apply loads and boundaries
- Obtain the solution
- Review the results

The following table shows the brief description of steps followed in each phase.

Pre-processor	**Solution processor**	**Post processor**
Assigning element type	Analysis definition	Read results
Geometry definition	Constant definition	Plot results on graphs
Assigning real constants	load definition	view animated results
Material definition	Solve	
Mesh generation		
Model display		

4.4.1 PRE-PROCESSOR

Preprocessor prepares the input data for ANSYS analysis. The general preprocessor (PREP 7) contains solid modeling and mesh generation capabilities, and is also used to define all other analysis data with the benefit of data base definition and manipulation of analysis data. Parametric input, user files, macros and extensive online documentation are also available, providing more tolls and flexibility for the analyst to define the problem. Extensive graphics ability is available throughout the ANSYS program, including isometric, perceptive, section, edge and hidden-line displays of three-dimensional structures-y graphs of input quantities and results, and contour displays of solution results.

THE PREPROCESSOR STAGE INVOLVES THE FOLLOWING:

- Specify the title, which is the name of the issue. This is discretionary yet exceptionally valuable, particularly if various configuration cycles are to be finished on the same base mode.
- Analysis types thermal analysis, modal analysis, Harmonic analysis etc.
- Creating the model: The model may be made in pre-processor, or it can be imported from other design software by changing the file format.
- Defining element type: these chosen from element library.
- Assigning real constants and material properties like young`s modules, Poisson’s ratio, density, thermal conductivity, damping effect, specific heat, etc.
- Apply mesh: Meshing is nothing but dividing the whole area into discrete number of particles.

SOLUTION PROCESSOR

Here we create the environment to the model, i.e. applying constraints & loads. This is the main phase of the analysis, where the problem can be solved by using different solution techniques. Here three major steps involved:

- Solution type required, i.e. static, modal, or transient etc. is selected.
- Defining loads: The loads may be surface loads, point loads; thermal loads like temperature, or fluid pressure, velocity are applied.
- Solve FE solver can be logically divided into three main steps, the pre-solver, the solution and post-solver. Model read by pre

solver which is created by the pre-processor and makes the arithmetical representation of the model and calls the mathematical engine, which calculates the result. The result return to the solver and the strains, stresses, etc. for each node within the component or continuum are calculated by post solver.

POST PROCESSOR

Post processing means the results of an analysis. It is probably the most important step in the analysis, because we are trying to understand how the applied loads affects the design, how the meshing is done.

Post processor analyzes results, which display stress and strain contours, distorted geometries, flow fields, safety factor contours, contours of potential field results; vector field displays shapes of mode and graphs related to time history. The post processor can also be used for algebraic operations, database manipulators, differentiation and integration of calculated results.

REVIEW THE RESULTS

Once the solution has been calculated, results can be reviewed in post processor. Two post processors are available: POST 1 and POST 26. We use POST 1, the general post processor to review the results at one sub step over the entire model or selected portion of the model. We can obtain contour displays, deform shapes and tabular listings to review and interpret the results of the analysis. POST 1 offers many other capabilities, including error estimation, load case combination, calculation among results data and path operations.

We use POST 26, the time history post processor, to review results at specific points in the model over all tome steps. We can obtain graph plots of results, data vs. time and tabular listings. Other POST 26 capabilities include arithmetic calculations and complex algebra.

The simultaneous set of equations that the finite element method generates the solution taken by the computer, the results of the solution are:

- Nodal degree of freedom values, which form the primary solution.
- Derived values which frame the component arrangement

MESHING

Before lattice the model and even before building the model, it is essential to consider whether a free work or a mapped cross section is proper for the examination. A free work has no limitations as far as component shapes and has no predefined example connected to it. Contrast with a free work, a mapped cross section is confined as long as the component shape it contains and the pattern of mesh. Mapped area mesh contains either quadrilateral or just triangular components, while a mapped volume cross section contains just hexahedron components. In the event that we need this kind of lattice, we must form the geometry as arrangement of genuinely normal volumes and/or regions that can acknowledge a mapped network.

STRUCTURAL STATIC ANALYSIS

The load effects can be calculated on a structure by ignoring the damping and inertia effects, such as those caused by time varying loads can be calculated by structural static analysis. Steady equivalent loads like steady inertia loads and time varying loads are included in Static analysis. Static analysis is utilized to decide the removals, burdens, strains and powers in structures or segments brought about by burdens that don't instigate noteworthy dormancy and damping impacts. Enduring stacking and reaction conditions are accepted, i.e. the stress and the structure's reactions are expected to differ gradually as for time. The kinds of load can be applied in static analysis include:

- Force and pressure application on body.
- Steady state inertial forces.
- Displacement.
- Thermal behavior.

ANALYSIS STEPS WITH DIFFERENT MATERIALS:

The steps needed to perform an analysis depend on the study type. You complete a study by performing the following steps:

- Create a study defining its analysis type and options.
- If needed, define parameters of your study. A parameter can be a model dimension, material property in this project we take two type of materials were selected which are Structural Steel, Gray Cast Iron.
- Define material properties.
- Specify restraints and loads.
- The program automatically creates a mixed mesh when different geometries (solid, shell, structural members etc.) exist in the model.
- Define component contact and contact sets.
- Mesh the model to divide the model into many small pieces called elements. Fatigue and optimization studies use the meshes in referenced studies.
- Run the study.
- View results.

STRUCTURAL STEEL

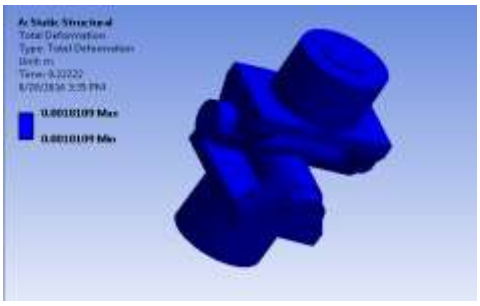

Fig8: Total Deformation on Universal Joint

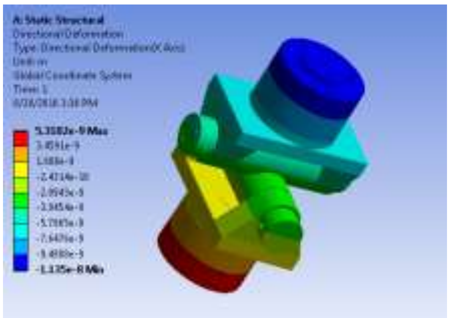

Fig9: Directional(X-axis) Deformation on Universal Joint

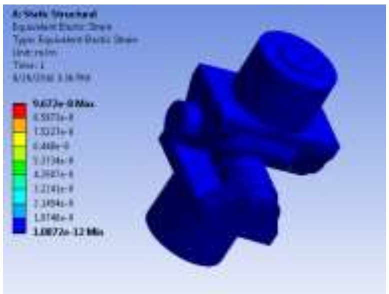

Fig10: Elastic Strain of Universal Joint

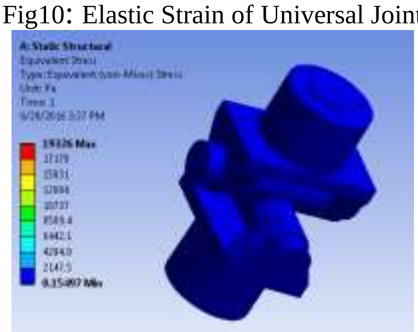

Fig11: Equivalent Stresses in Universal Joint

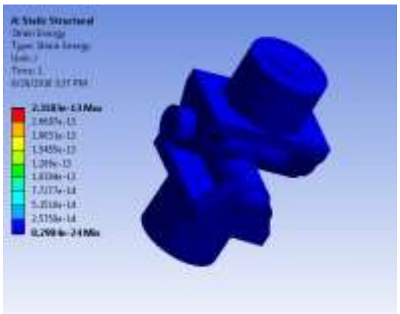

Fig12: Strain Energies Universal Joint

GREY CAST IRON

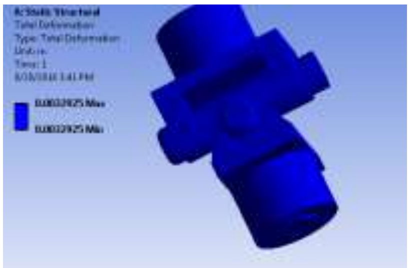

Fig13: Total Deformation on Universal Joint

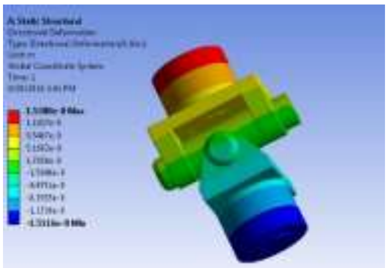

Fig14: Directional(X-axis) Deformation on Universal Joint

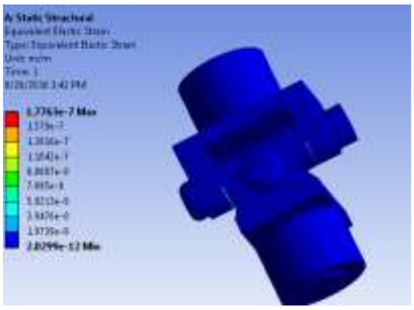

Fig15: Equivalent Elastic Strain of Universal Joint

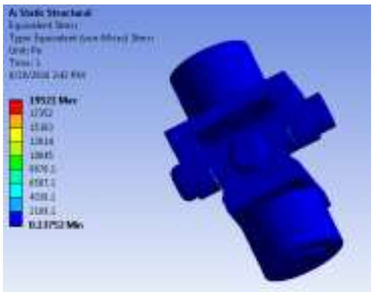

Fig16: Equivalent Stresses in Universal Joint

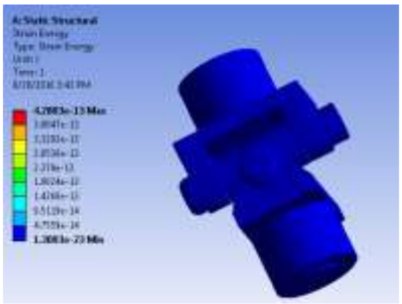

Fig17: Strain Energy in Universal Joint

CONCLUSIONS

- The universal joint has become recognized as the most precise and reliable joint on the global market today.
- Study on different materials which are suitable for the improvement of Universal joint. And the best material has been suggested for universal joint by analysis on different materials.
- By observing the all properties of Structural steel and Gray Cast Iron like Deformation, Stress, Strain and Strain Energy, Structural steel is the best material for Universal joint compare to the other materials.
- This joints are coupling devices that transmit torque or rotary motion from one shaft to another at fixed or variable angle of intersection and also for effective transmission.
- Result and Discussion:
  Thus the 3D assembly of the universal coupling has been created on the software CATIA V5 with accurate dimension and with all respects and the structural steel has the best material for the Universal joint for effective transmission.

REFERENCE:

I have completed this project by the following websites and books.

- International Journal of Advanced Technology in Engineering and Science, Vol. No. 4, Issue No. 07, July 2016
- Advanced Materials Manufacturing & Characterization Vol3 Issue 1 (2013), http://www.ijammc-griet.com/
- http://www.efunda.com/ for study of materials and its Properties.
- www.universaljoint.co.in/products.php about universal joint.