



JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)

An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Static Analysis of Driven Shaft with Composite Material

Mr. M. Sarath Chandra¹, Kummari Krishna Teja^{2*}, Nenavath Ganesh^{2*}

Lavudiya Sandeep^{2*}, Palthya Sai Kumar^{2*}

¹Assistant Professor, Mechanical Engineering,

Guru Nanak Institutions Technical Campus, Hyderabad, India

²Students, Mechanical Engineering,

Guru Nanak Institutions Technical Campus, Hyderabad, India

Abstract: A driven shaft, also known as a driven shaft or propeller shaft, is a mechanical component crucial for transmitting rotational power within machines. It comprises a cylindrical shaft with universal joints at each end, facilitating flexibility and accommodating changes in angle or alignment. Found in automotive vehicles, industrial machinery, marine propulsion systems, and power transmission equipment, driven shafts transfer power from engines or motors to various components, enabling functionality. Their design considers factors such as torque, speed, alignment, material properties, and environmental conditions to ensure optimal performance, reliability, and safety.

In this project, the modeling of the Driven Shaft was done in the Solid Works software, and the Static Analysis was done using the Ansys software. Three different materials Structural Steel, Gray Cast Iron, and Epoxy E-Glass UD were considered for analysis of the driven shaft. To analyze for better Structural characteristics the total deformation, Equivalent elastic strain, and Equivalent stress of the driven shaft were determined and compared. It was found that Structural Steel was best best-suited material from this analysis.

LITERATURE REVIEW

Vogwell J (1998) Analysis of a vehicle wheel shaft failure. Eng Fail Anal, vol 5, 4 ed, pp 271–277

This paper describes an investigation that was carried out on a failed wheel drive shaft component used on an unmanned, remotely operated vehicle for maneuvering military targets. As many vehicles had been manufactured and delivered to customers it was necessary to establish whether it was thought likely that more failures might occur. A study of the broken shaft shows how vulnerable such a rotating component can be to failure by fatigue, even when operating under steady conditions if basic preventative design actions are not taken. The analysis considers the effects of both transmission torque and weight (thus bending) upon stress levels and assesses their individual effects on the breakage and upon any subsequent modifications needed to improve the design. The drive shaft arrangement is compared with the feasible alternative of using a driven wheel arrangement rotating on a stationary axle. Findings confirm the importance of recognizing in advance the salient factors leading to fatigue and the necessity of paying adequate attention to detail

during design and manufacture if long service life is to be achieved.[1]

Bhaumik SK, Rangaragu R, Venkataswamy MA, Bhaskaran TA, Parameswara MA (2001) Fatigue fracture of the crankshaft of an aircraft engine. Eng Fail Anal 9:255–263

The crankshaft of a piston engine of a transport aircraft failed during climb. Cracks were found to initiate at the web radius region of journals 2 and 3 and progressed in the transverse direction of the shaft axis causing it to fracture. The initiation of cracks was by surface contact fatigue due to the constant rubbing of bearings against the journal webs. The movement of the bearing was caused by some axial load on the crankshaft. An analysis of the failure is presented in this paper.[2]

Rastogi N (2004) Design of Composite Driveshafts for Automotive Applications. SAE Technical Paper 1:485

This work presents a comprehensive approach to designing driveshafts for automotive applications. The two important aspects of driveshaft design, viz., (i) design of composite shaft

tube, and (ii) design of adhesively bonded tubular joint between the yoke and the tube are discussed. Based on the closed-form analytical solutions preliminary design tools are developed to aid quick analysis and design of composite driveshaft for automotive applications. Furthermore, detailed finite element analyses are performed to validate the preliminary design tools.[3]

Shenoy P, Ali F (2005) Connecting rod optimization for weight and cost reduction, SAE-2005-01-0987

An optimization study was performed on a steel-forged connecting rod with consideration for improvement in weight and production cost. Since the weight of the connecting rod has little influence on its total production cost, the cost and the weight were dealt with separately. Reduction in machining operations, achieved by change in material, was a significant factor in manufacturing cost reduction. Weight reduction was achieved by using an iterative procedure. A literature survey suggests cyclic loads comprised of static tensile and compressive loads are often used for the design and optimization of connecting rods. However, in this study weight optimization is performed under a cyclic load comprising dynamic tensile load and static compressive load as the two extreme loads. Constraints of fatigue strength, static strength, buckling resistance, and manufacturability were also imposed. The fatigue strength was the most significant factor in the optimization of the connecting rod. An estimate of the cost savings is also made. The study results in an optimized connecting rod that is 10% lighter and 25% less expensive, as compared to the existing connecting rod.[4]

Montazersadgh FH, Fatemi A (2007) Stress analysis and optimization of crankshafts subjected to dynamic loading, AISI

In this study, a dynamic simulation was conducted on a forged steel crankshaft from a single cylinder four stroke engine. Finite element analysis was performed to obtain the variation of the stress magnitude at critical locations. The dynamic analysis resulted in the development of the load spectrum applied to the crankpin bearing. This load was then applied to the FE model and boundary conditions were applied according to the engine mounting conditions. Results obtained from the aforementioned analysis were then used in the optimization of the forged steel crankshaft. Geometry, material, and manufacturing processes were optimized using different geometric constraints, manufacturing feasibility, and cost. The first step in the optimization process was weight reduction of the component considering dynamic loading. This required the stress range under dynamic loading not to exceed the magnitude of the stress range in the original crankshaft. Possible weight reduction options and their combinations were considered. The optimization and weight reduction were considered in an interactive manner and evaluated by manufacturing feasibility and cost. The optimization process resulted in an 18% weight

reduction, increased fatigue strength, and reduced cost of the crankshaft.[5]

Farrahi GH, Hemmati F, H-Gangaraj SM, Sakhaei M (2011) Failure analysis of four-cylinder diesel crankshaft engine crankshaft made from nodular cast iron, J Eng Res, 22 Spring

The premature breakage in some four-cylinder diesel engine crankshafts was reported. All crankshafts were failed from the same region. Failures had occurred in the first crankpin, the nearest crankpin to the flywheel. Dynamic analysis and finite element modeling were carried out to determine the state of stress in the crankshaft. FEM results revealed that the first crankpin fillet is the most vulnerable point to fracture. Soderburg's diagram of the studied crankshaft showed that the service operation point, which stands for mean and alternating stresses of the critical region (first crankpin fillet) was located in the safe region. Therefore, it can be concluded that fatigue fracture has not occurred in the crankshaft. SEM images of the fractured surface also showed cleavage fracture and put evidence that the failure was a brittle fracture. No sign of fatigue failure was observed. The fracture may have been caused by an overload. However, the results suggest a re-evaluation of the design and manufacturing. The fillet rolling may play an important role in this matter. Optimization of the fillet rolling process by changing process parameters has been recommended to the manufacturer. This recommendation has been adopted by the manufacturer and no further fracture has been reported since.[6]

Yingkui G, Zhibo Z (2011) Strength analysis of diesel engine crankshaft based on PRO/E and ANSYS, third international conference on measuring technology and mechatronics automation, Shanghai

The crankshaft is an important component of a diesel engine. In this paper, a three-dimensional model of a diesel engine crankshaft was established corresponding to the practical conditions by Using PRO/E software. Using the ANSYS analysis tool, the finite element analysis for the crankshaft was conducted under extreme operation conditions, and the stress distribution of the crankshaft was presented. The static rigidity and intensity of the crankshaft were analyzed in detail. The crank stress change model and the crank stress biggest hazard point were found by using finite element analysis, and the improvement method for the crankshaft structure design was given.[7]

Gibson R (2012) Principles of composite material mechanics, 3 edn, CRC Press

Extensively updated and maintaining the high standard of the popular original, Principles of Composite Material Mechanics, Second Edition reflects many of the recent developments in the mechanics of composite materials. It draws on the decades of teaching and research experience of

the author and the course material of the senior undergraduate.[8]

Novo FMF, Souza MMS, Savoy J, Silva MAC (2012) Analysis of the vibration modes of an automotive propeller shaft using FEM and analytical models. SAE 36:224

Traditional propeller shafts using universal joints have been replaced by sophisticated and complex solutions that not only reduce weight but also increase the performance of such systems in modern automotive vehicles. Due to its complexity that nowadays even may combine plastic and metallic components, traditional analytical models reach their limits to support engineers during their design phase.

Particularly, in the case of their analysis under vibration, it becomes critical, as the lifetime of a propeller shaft and its components (bushes and joints) have to work far away from their natural frequency values. Analytical solutions seem not to be helpful anymore when one needs to reach a mostly precise value of a natural frequency of complex shafts. Although the FEM analysis nowadays is so far highly developed, they are still not responding to the increasing demand for highly accurate results in a short period of development time.

This work focuses on the development of a reliable method to simulate complex propeller shafts under vibration, including the components bush and joints, and assessing its limits when compared to available analytical solutions. The paper presents a positive and efficient tool for the design phase of such powertrain products.[9]

Jayanaidu P, Hibbatullah M, Baskar P (2013) Analysis of a drive shaft for automobile applications, IOSR J Mech Civil Eng, e-ISSN: 2278-1684, p-ISSN: 2320-334X, 10(2):43–46

This study deals with the optimization of the drive shaft using ANSYS. The substitution of Titanium drive shafts over the conventional steel material drive shafts has increased the advantages of design due to its high specific stiffness, strength, and low weight. The drive shaft is the main component of the drive system of an automobile. The use of conventional steel for manufacturing drive shafts has many disadvantages such as low specific stiffness and strength. Many methods are available at present for the design optimization of structural systems. This paper discusses the past work done on drive shafts using ANSYS and the design and modal analysis of shafts made of Titanium alloy (Ti-6Al-7Nb).[10]

INTRODUCTION OF SOLIDWORKS

Solid Works is a 3d solid displaying bundle that enables clients to grow full solid models in a reproduced situation for both plan and examination. In solid works; you portray thoughts and try different things with various plans to make 3d models. Solid works are utilized by understudies, originators, engineers, and different experts to deliver

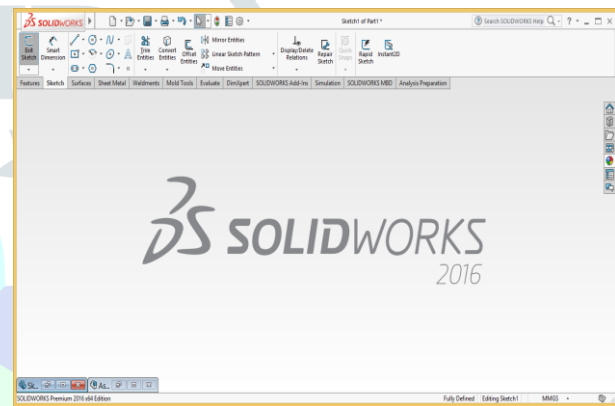
straightforward and complex parts, congregations, and illustrations. Outlining in a displaying bundle, for example, solid works is gainful because it spares time, exertion, and cash that would somehow or another be spent prototyping the plan.

solid Works Components - Parts

Before we begin looking at the software, it is important to understand the differences.

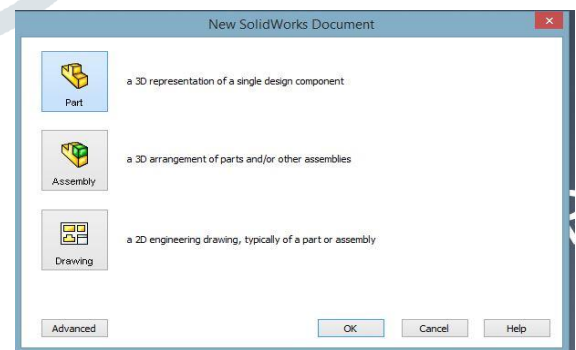
1. Part:
2. Assembly:
3. Drawing:

solid Works – Let's Begin



Solid Works Default Page

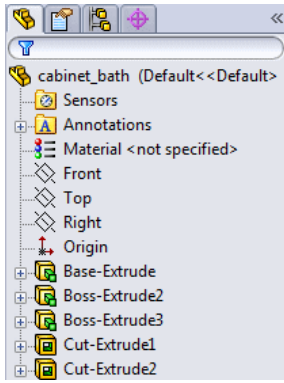
- To create a new file, click on file - new or click the new file icon in the main toolbar.



Parts of Solid Works

- Let's begin by creating a new part.

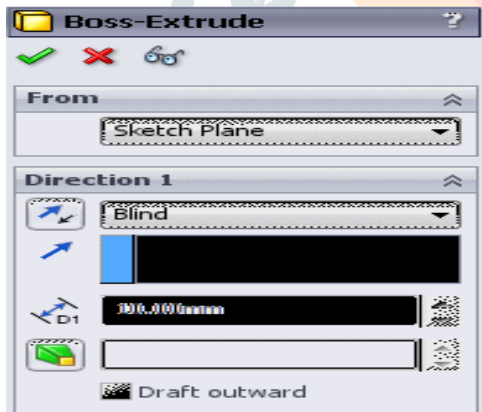
Feature Manager /Design Tree



Specification Tree

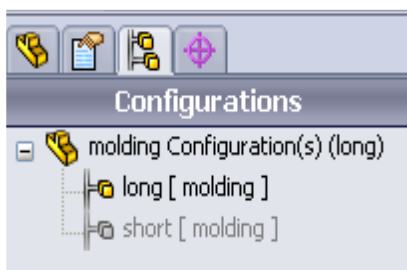
Displays the structure of the part, assembly, or drawing. Select an item from the feature manager design tree to edit the underlying sketch, edit the feature, and suppress and unsuppress the feature or component, for example.

- **Property manager:** provides settings for many functions such as sketches, fillet features, and assembly mates.



Extrude Tool Box

- **Configuration manager:** lets you create, select, and view multiple configurations of parts and assemblies in



a document. Configurations are variations of a part or assembly within a single document. For example, you can use configurations of a bolt to specify different lengths and diameters.

Configuration Box

Modeling Tools

Extrude:

You can utilize similar techniques you learned in lesson 1 to make the base for another part.

1. Click new (standard toolbar) and open another part.
2. Click extruded supervisor/base (highlights toolbar) and select the front plane.
3. Sketch a corner profile starting at the birthplace.
4. Click brilliant measurement (measurements/relations toolbar) and measure the profile
5. Click leave draw (outline toolbar) to leave the portrait.
6. Under direction1:
7. Set the end condition to daze.
8. Set profundity
9. Click to make the expulsion.

Extrude Cut:

1. Click extruded supervisor/base (highlights toolbar) and select the front plane.
2. Sketch a corner profile starting at the root.
3. Click shrewd measurement (measurements/relations toolbar) and measure the profile
4. Click leave outline (portray toolbar) to leave the draw.
5. Under direction1: set end condition to daze.
6. Set profundity
7. Click to make the expulsion.

Revolute:

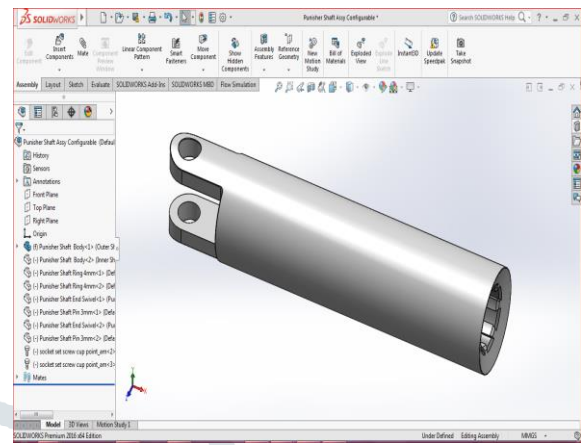
1. Click Revolute Supervisor/base (highlights toolbar) and select the front plane.
2. Sketch a corner profile starting at the root.
3. Click shrewd measurement and measure the profile.
4. Click leave portray (draw toolbar) to leave the outline.
5. Under direction1: set end condition to daze.
6. Set profundity

- Click to make the expulsion.

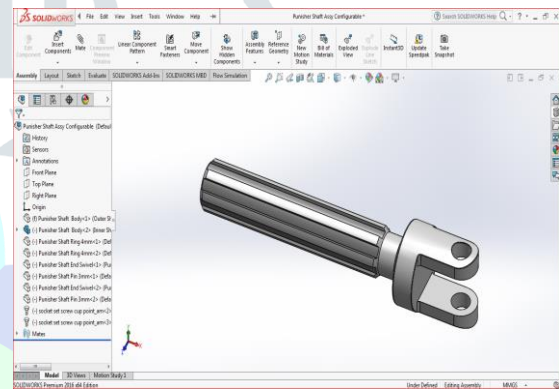
Revolute Cut:

- Click Revolute cut (highlights toolbar) and select the front plane.
- Sketch a corner profile starting at the birthplace.
- Click brilliant measurement and measure the profile to 99.54 mm.
- Click leave draw (portray toolbar) to leave the outline.
- Under direction1: set end condition to daze.
- Set profundity
- Click to make the expulsion.

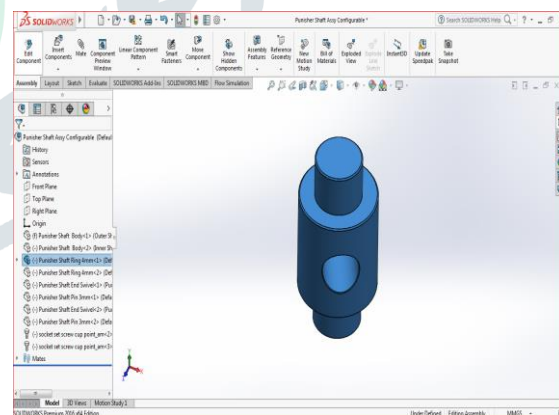
Parts



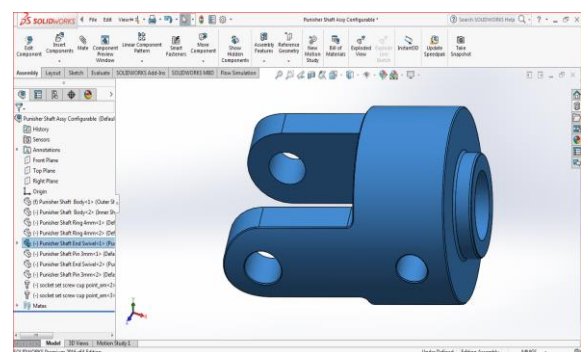
Male Shaft



Female Shaft



King Shaft



Tail Shaft Holder

Circular Pattern:

Make 7 gap chamber expulsions with cuts and fillets equally divided about the focal pivot of the part utilizing the round example instrument.

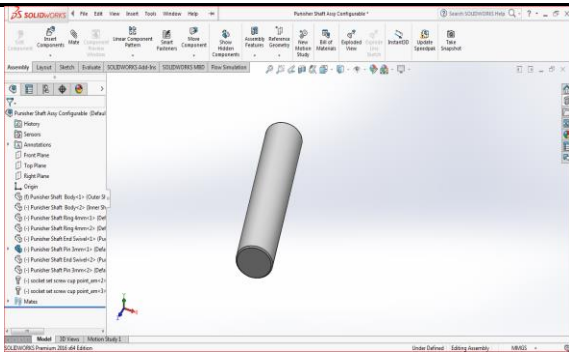
- On the highlighted toolbar, extend the straight example fly-out toolbar and snap the roundabout example.
- In the property director, under parameters:
- Select the edge in the focal point of the part for the design pivot.
- Select equivalent separating to design the number of occurrences consistently around the hub inside 360°.
- Set the number of cases to 7.
- Click in highlights to design.
- In the fly-out component administrator configuration tree in the designs range, select the last highlights (cut-extrude2).

Files:

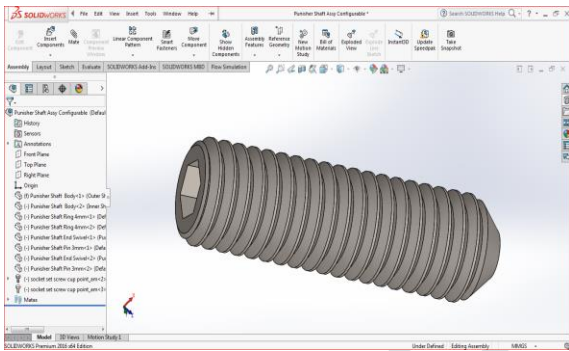
To choose the edges for including the round shapes with required thickness as appeared underneath by utilizing extrude, for adding the material to make the shape with a thickness of 5mm circle it appeared as beneath

Opening:

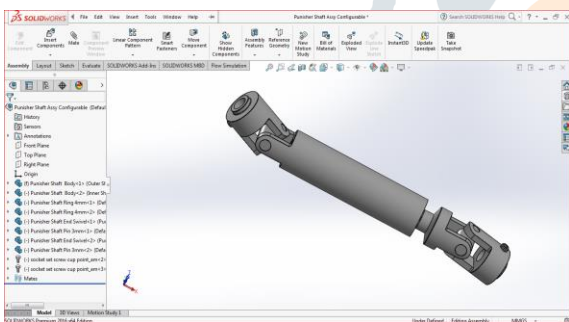
Opening Apparatus we can apply the expel the material on the surface of the brake rotor for the required measurement as appeared underneath



Shaft Pin



Socket Set Screw

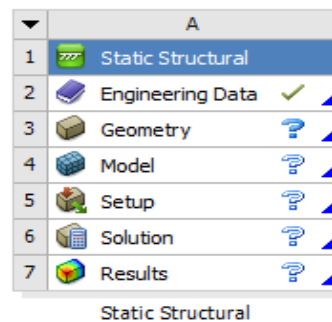


Assembly

deformation, equivalent stress, maximum principal stress, and minimum principal stress.

Static structural analysis is one of the important analyses in Ansys workbench. It is available as a static structural analysis system under the analysis system toolbox in the toolbox window, this system analyses the structural components for displacements (deformation), stresses, strains, and forces under different loading conditions. The loads in this analysis system are assumed not to have damping characteristics (time-dependent). Steady loading and damping conditions are assumed in this type of analysis system.

To start a new static structural analysis system, double-click on static structural in the analysis systems toolbox in the toolbox window; the static structural analysis system will be added to the project schematic window. To start an analysis, first, you need to specify the geometry on which the analysis is to be done. To do so, you can import the geometry from an external CAD package, or you can create the geometry in Ansys's design modeler software. After the model is specified for analysis, you need to double-click on the model cell of the static structural analysis system to open the mechanical window. In this window, you can specify the parameters and run the analysis.



The Static Structural Analysis Added To The Project Schematic

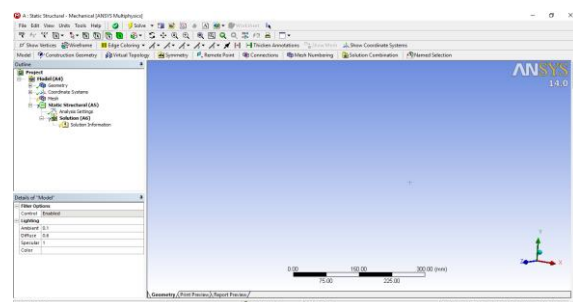
STATIC STRUCTURAL ANALYSIS

Project Objective

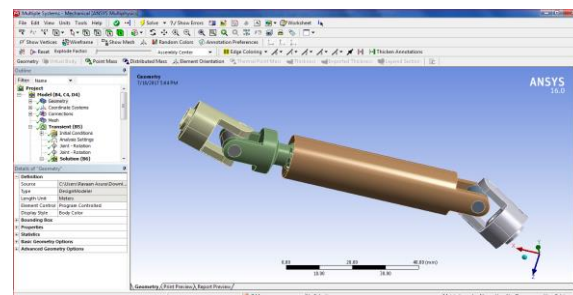
In this project, we will be able to define total deformation stress, etc.

- Create the static structural analysis system
- Apply different types of materials
- Applying boundary conditions
- Apply a different type of constraints
- Apply different loads
- Generate the results as required
- Generate project reports

In this project, we imported the geometry of the component to show the dimensions of the component concerning the load applications. The material to be applied to the model is stainless steel. Next, you will run the analysis under two conditions and evaluate the total deformation, directional



The Mechanical Window



Modal

As discussed in previous chapters, analysis can be carried out in three major steps: pre-processing, solution, and post-processing. The tools required to carry out these steps are discussed next.

Pre-Processing

The pre-processing of an analysis system involves specifying the material, generating a mesh, and defining boundary conditions.

In ansys workbench, the various tools related to boundary conditions are available in the environment contextual toolbar, which is displayed when you select the static structural node in the tree outline



The Environment Contextual Toolbar

To provide support to the model, you need to choose the required tool from the supports drop-down. Similarly, to add a load, choose the desired tool from the loads drop-down in the environment contextual toolbar. Also, when you choose any tool from the environment contextual toolbar; the corresponding entity is placed under the static structural node in the tree outline.

The main purpose of an analysis is to evaluate the results. After the boundary condition is set and loads are applied, you need to specify the desired outcomes of the analysis. In ansys workbench, you can analyze various parameters such as deformation, stresses, strains, and so on. To do so, you need to specify the results required and then evaluate them. You can use the tools available in the solution contextual toolbar to specify results. Alternatively, right-click on the solution node in the tree outline and then use the desired option from the shortcut menu displayed.



The Solution Contextual Toolbar

To evaluate deformations, stresses, strains, and so on, choose the desired options from the drop-downs available in the solution contextual toolbar.

Solution

In an analysis, the next step is to solve the analysis after pre-processing (meshing, specifying material, and specifying boundary condition). In ansys workbench, you will use the solve tool from the standard toolbar to run the solver. The solver runs in the software background and acquires analysis results, based on the specified boundary conditions.

Post-Processing

After completing the analysis, you need to generate the report in the mechanical window. To do so, choose the report preview tab from the bottom of the graphics screen; the ansys report generation in progress message is displayed on the screen. After some time, this message vanishes and the report is generated.



The Report Generation in Progress

Project Overview

The dimensions to create the model and its boundary and loading conditions are also given in the same figure. Run a static structural analysis on the model and evaluate the total deformation and the directional deformation. Determine directional deformation along the x, y, and z axes. After evaluating the results, interpret them. (expected time: 3hr)

- A. Start a new project and create the model.
- B. Generate the mesh.
- C. Set the boundary and loading conditions.
- D. Solve the model.
- E. Duplicate the existing analysis system.
- F. Interpret results.
- G. Save the project.

Starting A New Project and Creating the Model

The first step is to start a new project in the workbench window.

1. Start ansys workbench.
2. Choose the save button from the standard toolbar; the save as dialog box is displayed.
3. Double-click on static structural in the toolbox window; the static structural analysis system is added in the project schematic window.
4. Rename the static structural analysis system (if).
5. In the cantilever analysis system, double-click on the geometry cell; the design modeler window along with the ansys workbench dialog box is displayed.
6. In the ansys workbench dialog box, set the unit to millimeter. Now, create the model on the xy plane
7. Exit the design modeler window to display the workbench window.

Adding The Material to The Engineering Data Figure Workspace

After creating the holes in the model, you now need to apply the material to it. The material to be applied is stainless steel.

1. Double-click on the engineering data cell of the spring plate analysis system; the engineering data workspace is displayed in the workbench window.
2. Choose the engineering data sources toggle button from the standard toolbar; the engineering data sources window is added to the engineering data workspace.

- In the engineering data sources window, select the general materials library to display the outline of the general materials window.
- In the outline of the general materials window, choose the plus symbol corresponding to aluminum alloy; the material is added to the engineering data in the outline window of the engineering data workspace.
- Again, choose the engineering data sources toggle button from the standard toolbar to switch to the default view of the engineering data workspace.

Choose the return to project button from the standard toolbar to display the project schematic window.

Specifying A Material

Generating The Mesh

After the model is created in the design modeler window, you need to generate the mesh.

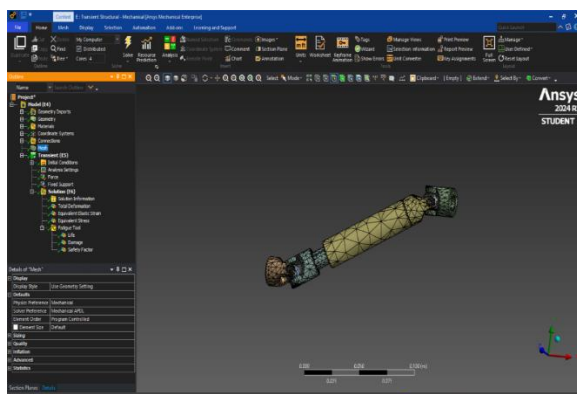
- double-click on the model cell in the static structural analysis system
- Select mesh in the tree outline to display the details of the "mesh" window.

The element size edit box is used to specify the size of an element. The element size specified in this edit box is according to the size of the geometry. However, this edit box will not be visible when the

- On: proximity and
- On: proximity and curvature

Options are selected from the use advanced size function drop-down list. When default is displayed in the element size edit box, it indicates that a default value, based on the size of the geometry, is already specified by the software.

- Choose the generate mesh tool from the mesh drop-down in the mesh contextual toolbar; the mesh is generated.



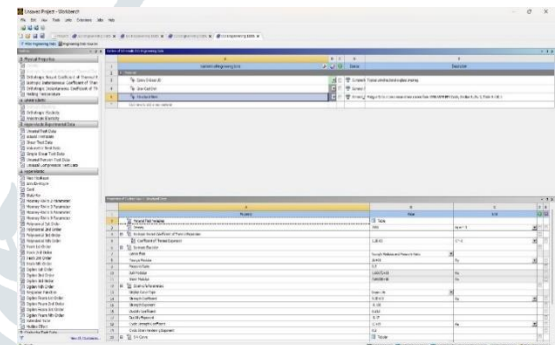
Mesh Generated with Default Mesh Controls

- Expand the statistics node in the details of the "mesh" window to display the total number of elements created. On doing so, you will find that the total number of elements.

Specifying The Boundary Conditions

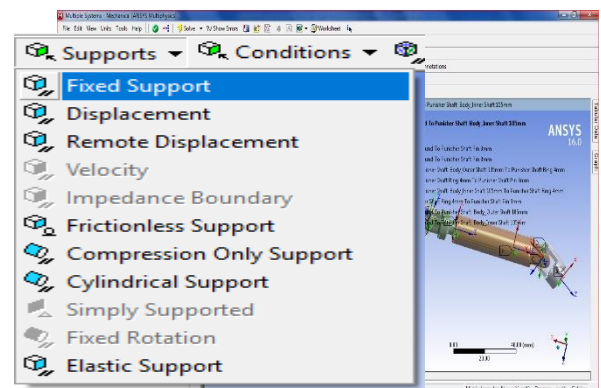
After you mesh the model, it is required to specify the boundary and loading conditions.

- In the mechanical window, select the static structural node from the tree outline; the details of the "static structural" window are displayed along with the environment contextual toolbar.
- Go to the environment contextual toolbar- select the connection in outline- click on the contact region and select the required suitable face for friction contact



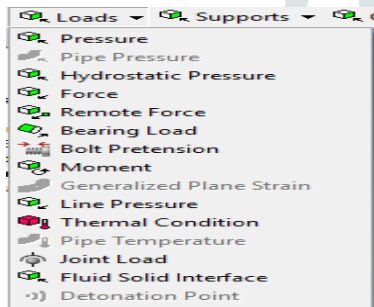
- Go to the environment contextual toolbar select body to ground then go to details of the body to ground connection select revolute connection then select face as shown below - click on a mobile reference
- Go to the environment contextual toolbar select body to ground then go to details of the body to ground connection select fixed connection then select face as shown below - click on mobile reference.

Connection and Force



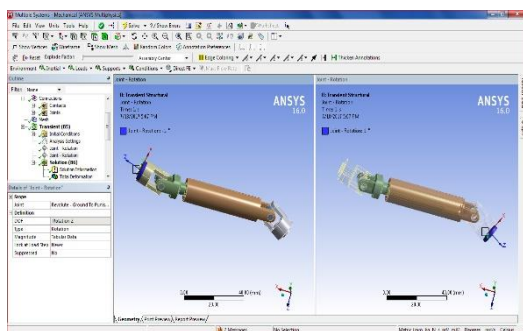
Choosing the Fixed Support Tool from The Supports Drop-Down

1. Select faces on the model. Next, choose the apply button in the geometry selection box; the selected faces turn purple indicating that fixed support is applied
2. Choose the joint load tool from the loads drop-down in the environment contextual toolbar; the joint load is added under the static structural node in the tree outline. Also, the details of the "joint load" window are displayed
3. Click on the geometry selection box to display the apply and cancel buttons, if they are not already displayed.
4. Choose the apply button from the geometry selection box; the cylindrical face turns red indicating that the joint load is applied.
5. In the details of the "joint load" window, expand the Definition node, if it is not already expanded. Specify magnitude 10°C



The Loads Drop-Down

6. Select vector from the define by drop-down list, if it is not already selected.



Fixed

Supported

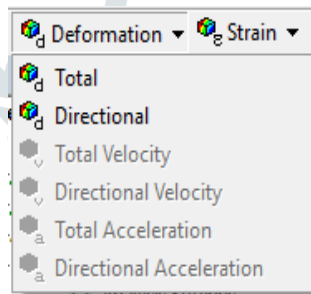
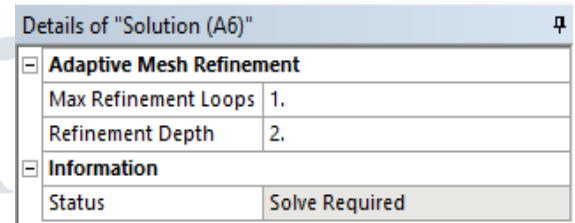
Solving The Fe Model and Analyzing the Results

After the boundary and load conditions are specified for the model, you need to solve the analysis. After solving, you will get the total and directional deformations due to the given condition. Also, you will get equivalent stress, maximum principal, and minimum principal stresses.

1. Select the solution node in the tree outline; the solution contextual toolbar is displayed. Also, the details of the "solution" window are displayed.

The Details of the Solution Window

2. Choose the total tool from the deformation drop-down of the solution contextual toolbar; total deformation is added under the solution node.



Choosing the Total Tool from The Deformation Drop-Down

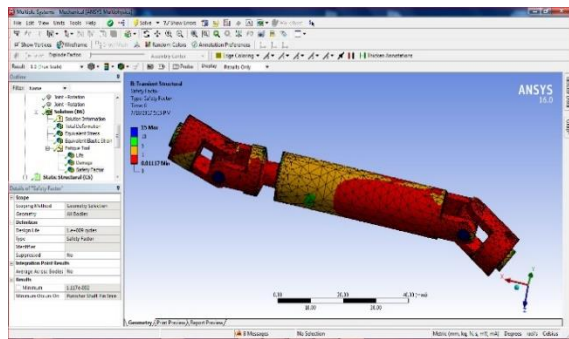
3. Now, choose the directional tool from the deformation drop-down; directional deformation is added under the solution node.
4. Choose the equivalent (von-misses) tool from the stress drop-down in the solution contextual toolbar;

The equivalent or von-misses stress is the criteria by which the effect of all the directional stresses acting at a point is considered 1" This helps in finding out whether the model will fail or bear the stress at that particular point.

5. Choose the contact tools option from the stress drop-down in the solution contextual toolbar; contact tools are added under the solution node.
6. Choose the contact tools and right-click on the contact tool to select frictional stress
7. Choose the contact tools and right-click on the contact tool select pressure

8. In the details of the “total deformation” window, expand the results node, if it is not already expanded. Note that the maxi and mini deformations are displayed, respectively.

The Scaling Factor



10. Select all other

parameters from the solution node; the respective view is displayed on the graphics screen. The table given next lists all the results obtained from the analysis. Show the corresponding graphical representation of values obtained.

11. Close the existing mechanical window; the workbench window is displayed. A body is called to be deformed if its shape is changed temporarily or permanently. The temporary change of shape is known as elastic deformation and a permanent change of shape is known as plastic deformation. In ansys workbench, you can determine deformation in terms of total and directional deformations. Total deformation is the total change of shape in a given working condition. You can view the total deformation induced in any component by using the total tool from the deformation drop-down in the solution contextual toolbar. Directional deformation is the total change of shape in a particular axis, due to given working conditions. You can view directional deformation by using the directional tool from the deformation drop-down in the solution contextual toolbar.

Total deformation is the summation of all directional deformations produced in a certain region of the model. The following equation describes the total deformation:

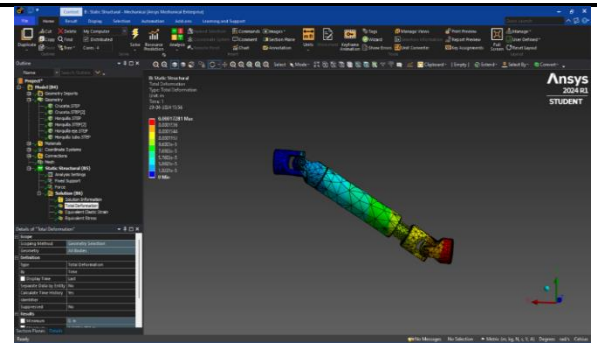
$$U = \sqrt{U_x^2 + U_y^2 + U_z^2}$$

If deformation in the x-axis is U_x
 Deformation in the y-axis is U_y
 Deformation in the z-axis is U_z

The total deformation, U will be given as follows:

$$U = (u_x^2 + u_y^2 + u_z^2)^{1/2}$$

The legend has colors arranged in a band from top to bottom. Depending upon the type of analysis and the parameters evaluated, each color will indicate a different value. A typical legend is displayed when the total deformation of the tree outline. The blue color in the legend is the value of total deformation. In this case, it displays 0 which means there is no deformation at that region. The value displayed next to each color is the total deformation in the region which is depicted by that particular color in the model. The blue color in the model represents the lowest value of total deformation,



whereas the red color denotes the maximum value of the total deformation. To view the transparent mode of the model, select the show undeformed model option from the edges drop-down list. Similarly to show the elements created after meshing in the deformed shape of the model, select the show elements option from the edges drop-down list, and show the undeformed model when the show undeformed model option is selected. Displays the elements when the show elements option is selected. Exit the static structural analysis - mechanical window.

RESULTS

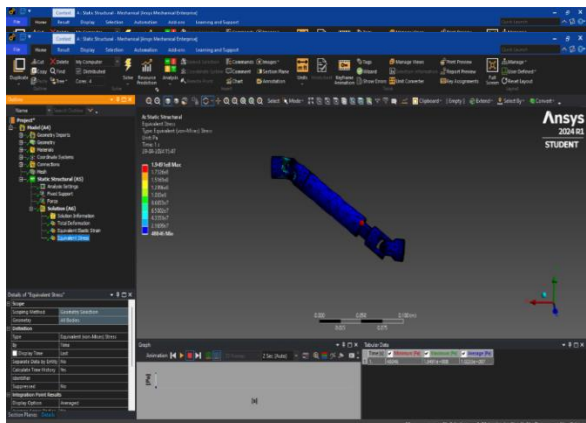
By using Ansys software total deformation, equivalent elastic strain, and equivalent stress, are done for the designed crash guard. Analysis was done by using different materials Structural Steel, Gray Cast Iron, And Epoxy E-Glass UD. By applying a 1500N load the structural steel has less deformation than stainless steel.

Total deformation results for structure steel and gray Cast iron are given in the below table.

Equivalent (Von Mises) Stress results for structure steel and gray Cast iron are given in the below table.

Total Deformation

Materials	Minimum	Maximum
Structural Steel	0	0.00017281
Gray Cast Iron	0	0.00017281
Ebony E-Glass UD	0	0.00078765



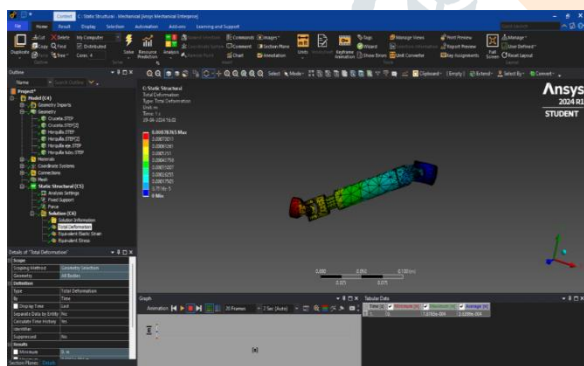
Value of Total Deformation of Structural Steel

The Value of Total Deformation of Gray Cast Iron

The Value of Total Deformation of Epoxy E-Glass UD

Equivalent Elastic Strain results for structure steel and gray Cast iron are given in the below table

Equivalent Stress

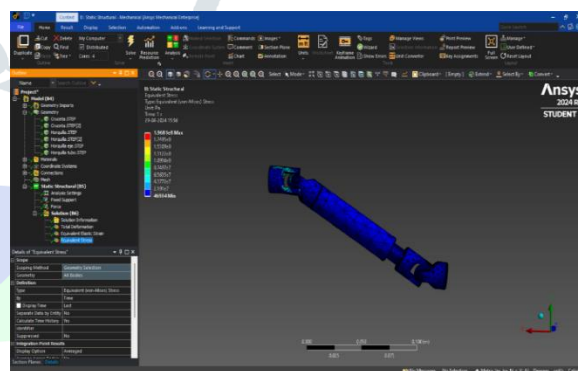


Materials	Minimum	Maximum
Structural Steel	48046	1.9491e8
Gray Cast Iron	50423	1.9297e8
Ebony E-Glass UD	75648	2.8462e8

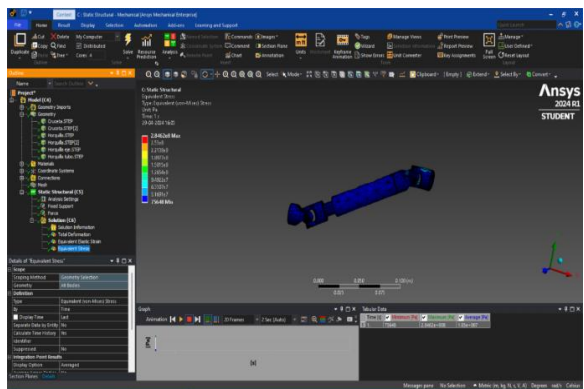
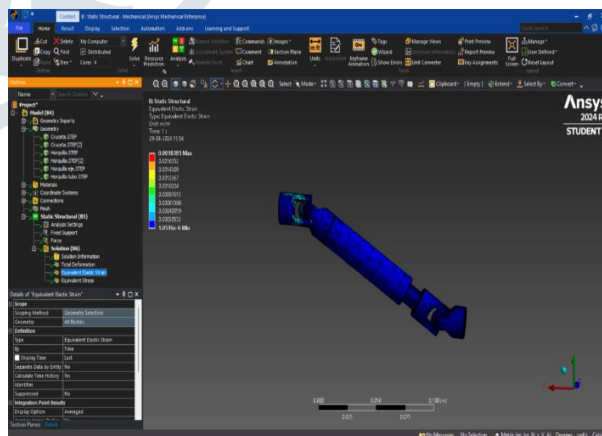
- The Value of the Equivalent Stress of Structural Steel
- The Value of Equivalent Stress of Gray Cast Iron.
- The Value of Equivalent Stress of Epoxy E-Glass UD

Equivalent Elastic Strain

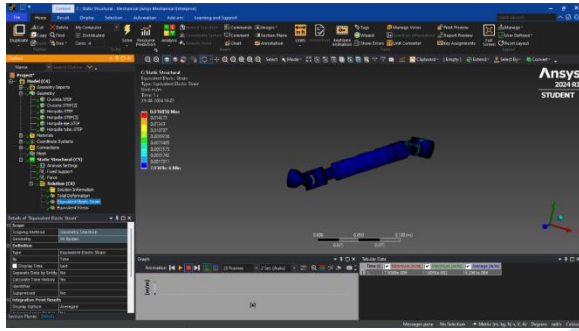
Materials	Minimum	Maximum
Structural Steel	5.7957e-7	0.0010009
Gray Cast Iron	1.0516e-6	0.0018395
Ebony E-Glass UD	7.9369e-6	0.016056



The Value of Equivalent Elastic Strain of Structural Steel



The Value of Equivalent Elastic Strain of Gray Cast Iron



The Value of Equivalent Elastic Strain of Epoxy E-Glass UD

CONCLUSION

In this project, the modeling of the Driven Shaft by using one of the most advanced 3-dimensional software mostly known as Solidworks software designed the Driven Shaft by using different types of tools and features that can be seen in the modeling of Driven Shaft. Later file is saved in the format as an STP or IGES file to analyze the component. The 3d model had been taken from the 3d model's library grab cad which has the suitable design specification which is required for this project.

The analysis is done by ANSYS software one of the most practical meshing accurate analysis software to find the result over the component. Structural Steel has less deformation than Gray Cast Iron and Epoxy E-glass UD. Structural Steel also has less stress and strain induced compared to Gray Cast Iron and Epoxy E-glass UD. Structural Steel provides more efficiency than Gray Cast Iron and Epoxy E-Glass UD.

In this project, a comparison analysis between Gray Cast Iron and Epoxy E-Glass UD and Structural Steel. Compared to Gray Cast Iron and Epoxy E-Glass UD, Structural Steel has less Deformation.

In future work, we will attempt to demonstrate numerically that is hostile to clasp impact by SMPW's winding cover structure. Furthermore, we will assess the impact on the weak point of confinement of this innovation. This SMPW innovation ought to be affirming significantly add to decrease weight, fortify and spare carbon fiber material to meet cost prerequisite for car parts; ordinarily for light-weight torque shaft like a propeller or drive pivot to accomplish extreme execution.

REFERENCES

1. Vogwell J (1998) Analysis of a vehicle wheel shaft failure. Eng Fail Anal, vol 5, 4 edn, pp 271–277
2. Bhaumik SK, Rangaragu R, Venkataswamy MA, Bhaskaran TA, Parameswara MA (2001) Fatigue fracture of the crankshaft of an aircraft engine. Eng Fail Anal 9:255–263

3. Rastogi N (2004) Design of Composite Driveshafts for Automotive Applications. SAE Technical Paper 1:485

4. Shenoy P, Ali F (2005) Connecting rod optimization for weight and cost reduction, SAE-2005-01-0987

5. Montazersadgh FH, Fatemi A (2007) Stress analysis and optimization of crankshafts subjected to dynamic loading, AISI

6. Farrahi GH, Hemmati F, H-Gangaraj SM, Sakhaei M (2011) Failure analysis of four-cylinder diesel crankshaft engine crankshaft made from nodular cast iron, J Eng Res, 22 Spring

7. Yingkui G, Zhibo Z (2011) Strength analysis of diesel engine crankshaft based on PRO/E and ANSYS, third international conference on measuring technology and mechatronics automation, Shangai

8. Gibson R (2012) Principles of composite material mechanics, 3 edn, CRC Press

9. Novo FMF, Souza MMS, Savoy J, Silva MAC (2012) Analysis of the vibration modes of an automotive propeller shaft using FEM and analytical models. SAE 36:224

10. Jayanaidu P, Hibbatullah M, Baskar P (2013) Analysis of a drive shaft for automobile applications, IOSR J Mech Civil Eng, e-ISSN: 2278-1684, p-ISSN: 2320-334X, 10(2):43–46