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MODELLING AND ANALYSIS OF DRIVEN SHAFT WITH COMPOSITE MATERIALS

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ABSTRACT
This study deals with the review of the optimization of drive shafts using ANSYS. The substitution of composite material over regular steel material for the drive shaft has expanded the upsides of the outline because of its high particular firmness and quality. The drive shaft is the primary part of the drive arrangement of a vehicle. The utilization of traditional steel for assembling drive shafts has many impediments, for example, low particular firmness and quality. Regular drive shaft is made up of two sections to build its major common twisting recurrence.
A two-piece drive shaft expands the heaviness of the drive shaft which is not alluring in the present market. Numerous strategies are accessible at display for the plan advancement of basic frameworks and these strategies are in view of scientific programming procedures including slope look and direct inquiry. These techniques expect that the plan factors are persistent. Be that as it may, in useful basic building improvement, all the plan factors are discrete. This is because of the accessibility of segments in standard sizes and imperatives because of development and assembling rehearses. This paper talks about the past work done on composite drive shafts utilizing ANSYS

Keywords:
Parts,assembly,drawing,gray Cast Iron,epoxy E-glass Ud

INTRODUCTION

A drive shaft is a mechanical section for transmitting torque and turn, by and large used to relate distinctive parts of a drive setup that can't be related particularly in the perspective of partition or the need to consider relative advancement between them.
As torque bearers, drive shafts are obligated to torsion and shear extend, equivalent to the difference between the data torque and the pile. They ought to this way be adequately strong to persevere through the uneasiness while keeping up a vital separation from an overabundance of additional weight as that would therefore fabricate their inaction.
To mull over assortments in the course of action and partition between the driving and driven fragments, drive shafts regularly combine no less than one comprehensive joints, jaw couplings, or material joints, and as a less than dependable rule a splined joint or vivid joint.

1.1. Front-Wheel Drive

In British English, the articulation "drive shaft" is bound to a transverse shaft that transmits vitality to the wheels, especially the front wheels. A drive shaft interfacing the gearbox to a back differential is known as a propeller shaft or prop shaft. A prop-shaft get-together includes a propeller shaft, a slip joint, and no less than one comprehensive joints. Where the engine and axles are separated from each other, as on four-wheel drive and back-wheel drive vehicles, it is the propeller shaft that serves to transmit the driving urge created by the engine to the axles.
A couple of particular sorts of drive shafts are used as a piece of the auto business:

- One-piece drive shaft
- Two-piece drive shaft
- Slip-in-tube drive shaft

The slip-in-tube drive shaft is another sort that improves crash prosperity. It can be compacted to ingest imperativeness in the event of a crash, so it is generally called a collapsible drive shaft.

1.1.1. Four-Wheel and All-Wheel Drive

These progressed from the front-engine back-wheel drive outline. Another sort of transmission called the trade case was placed among the transmission and last drives in the two axles. This Split the drive to the two axles and may in like manner have included diminishment prepares, a pooch handle, or differential. No under-two drive shafts were used, one from the trade case to each rotation. In some greater vehicles, the trade box was halfway mounted and was itself driven by a short drive shaft. In vehicles the traverse of a Land Rover, the drive shaft to the front rotation is conspicuously shorter and more steeply verbalized than the back shaft, making it a more troublesome planning issue to manufacture a tried and true drive shaft, which may incorporate a more current kind of boundless joint.

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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.

3.1 Solid Works Components - Parts

Before we begin looking at the software, it is important to understand the different components that make up a solid works model.

1. Part
2. Assembly
3. Drawing

3.2 Solid Works – Let's Begin

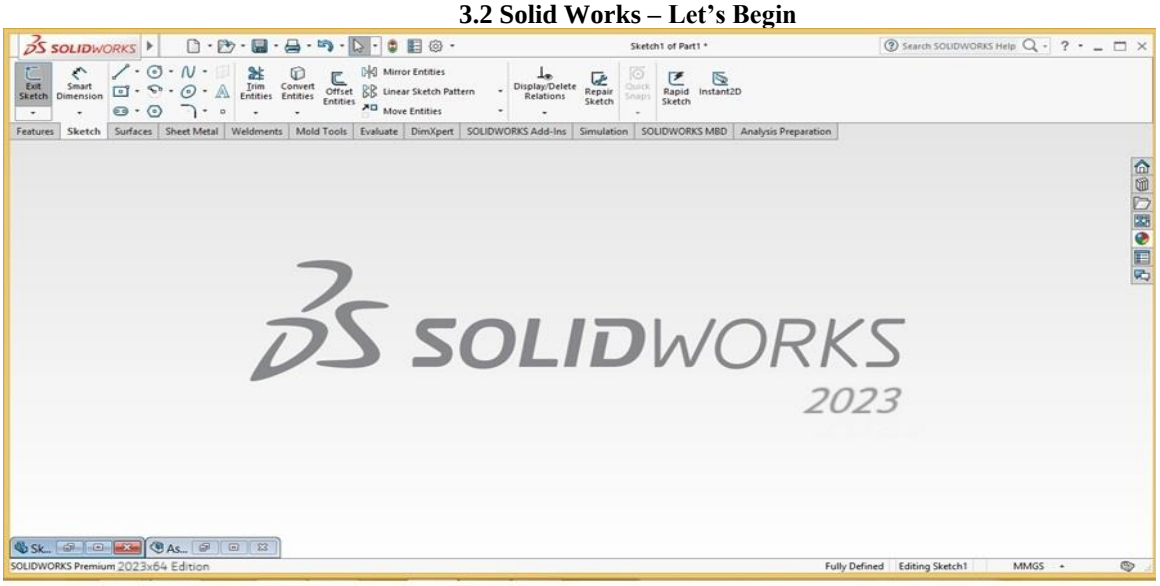

Figure 3.1 Solid Works Default Page

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

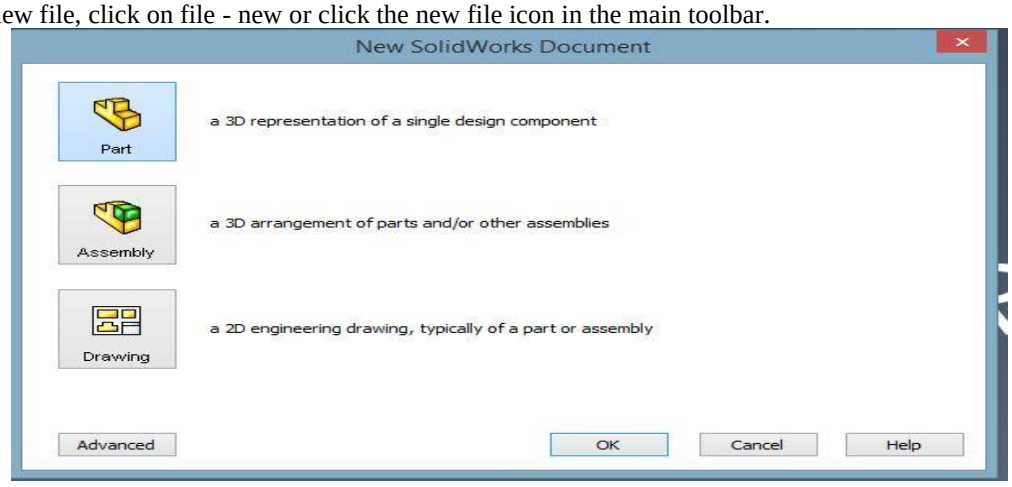

Figure 3.2 Parts of Solid Works

- Let's begin by creating a new part.

3.3. Feature Manager /Design Tree

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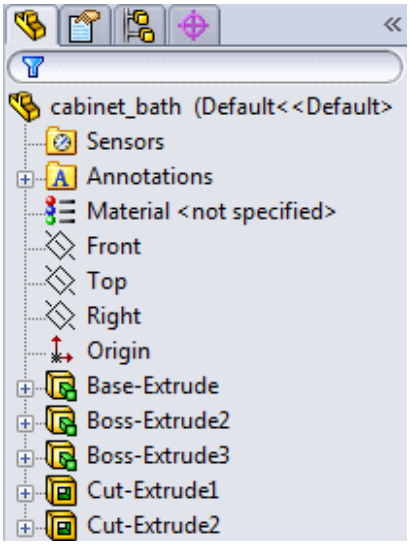

Figure 3.3 Specification Tree

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 un suppress the feature or component, for example.

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

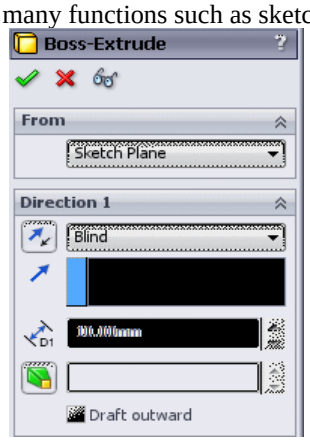

Figure 3.4 Extrude Tool Box

- **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.

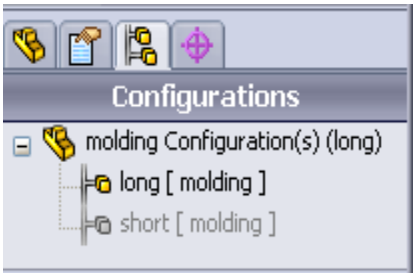

Figure 3.5 Configuration Box

3.4. Modeling Tools

3.4.1. 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.

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6. Under direction1:
7. Set the end condition to daze.
8. Set profundity
9. Click to make the expulsion.

3.4.2. 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.

3.4.3. 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
7. Click to make the expulsion.

3.4.4. Revolute Cut:

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

3.4.5. Circular Pattern:

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

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

3.4.6. Filets:

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's appeared as beneath

3.4.7. Opening:

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

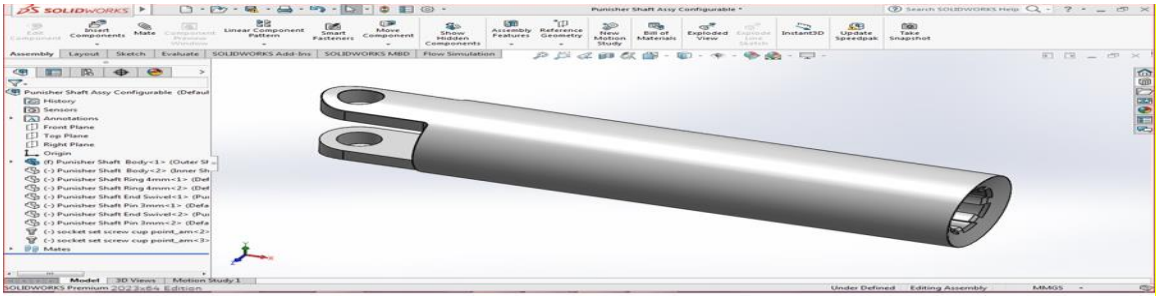

Figure 3.6 Male Shaft

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METHODOLOGY

4.1 Project Objective

In this chapter, we will be able to define:

- Understand the types of system
- Understand different types of cells
- Understand the graphic user interface of the workbench window
- Start a new project in ansys workbench windows
- Add the first and subsequent analysis system to a project
- Set units for the project

We can perform a variety of tasks ranging from design assessment to finite element analysis to complete product optimization analysis by using ansys workbench. Ansys also enables the combination of a standalone analysis system into a project and to management of the project workflow.

In ansys workbench, this is the list of analyses that can be determined:

- Modal analysis
- Static structural analysis
- Transient structural analysis
- Steady-state thermal analysis
- Transient thermal analysis
- Fluid flow (CFD)

4.2. Starting Ansys Workbench 16.0

To start ansys workbench 16.0, choose start- programs/ all programs- ansys 16.0 - workbench 16.0 from the taskbar. Alternatively, we can start the ansys workbench by double-clicking on the workbench 16.0.

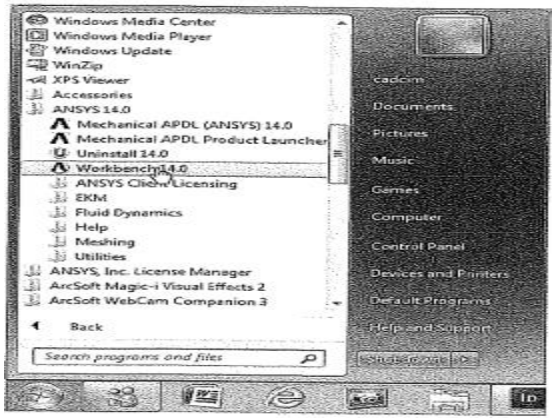

Figure 4.1 Starting of Ansys Workbench Using Taskbar

The workbench windows help streamline an entire project to be carried out in Ansys workbench 16.0. In this window, one can create, manage, and view the workflow of the entire project created by using a standard analysis system. The workbench windows mainly consist of the menu bar, standard toolbar, the toolbar windows, project schematic windows, and the status bar.

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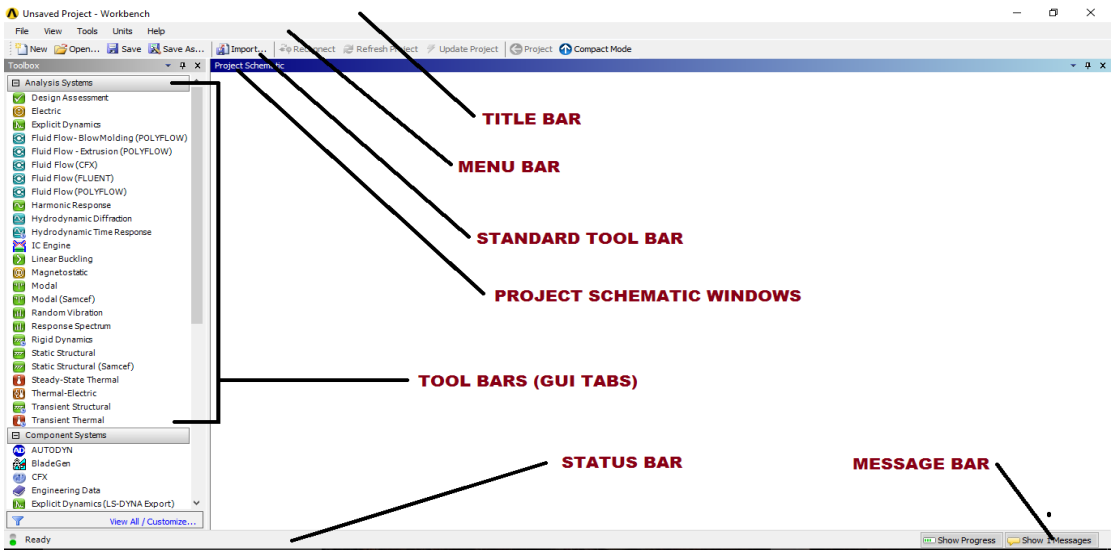

Figure 4.2 The Component of The Workbench Windows

4.3 Toolbox Windows

The toolbox windows are located on the left in the workbench windows. The toolbox windows list the standard and customized templates or the individual analysis components that are used to create a project. To create a project, drag a particular analysis or component system from the toolbox window and drop it into the project schematic

CONCLUSION

In this investigation, To assess a few tube-shaped composite shafts via carbon overlay approach. The work has focused on extreme quality, unwavering quality with the least material utilization, Change of cover stacking succession and sheet winding generation process result in a definitive quality going 20% higher than regular idea shaft.

The transient analysis has done on the given driven shaft by changing with different materials like Grey Cast Iron, structural steel and Epoxy E-Glass fiber. The analysis is carried out by taking different loading conditions when the body in dynamic nature and stacking process is also helpful for enhancing the lifetime survival of driven shaft .

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