Modeling, analysis, and prototype of an involute spur gear

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ABSTRACT
This project deals with the modeling and analysis of spur gear with involute profile. The involute gear profile is the most commonly used system for gearing today. In an involute gear, the profiles of the teeth are involutes of a circle. In involute gear design contact between a pair of gear teeth occurs at a single instantaneous line. Rotation of the gears causes the location of this contact line to move across the respective tooth surfaces.

The project mainly deals with modelling of spur gear with the help of CATIA, analysis was done in the ANSYS software and prototype was done on the 3D Printer. This model was saved in the form of .stl (stereo lithography) file for 3D Printing; the file was imported in QURA software. QURA will generate the g-code which was used to print the component in 3D.

Keywords: Involute Spur Gears, CATIAV5R20, ANSYS, 3D Printing.

1. INTRODUCTION TO GEARS

Gears (or cogs) are toothed wheels used for the transmission of power in many mechanical systems. When two gears are meshed with each other, a definite velocity ratio is obtained. Velocity ratio (or gear ratio) is the ratio between the angular velocity of driving gear and the angular velocity of driven gear.

Gears are typically used for short distance power transmission. They are compact and have high transmission efficiency when compared to other power transmission systems.

Advantages of Gear Drives:
- Transmission efficiency is high.
- Large power can be easily transmitted.
- Maintenance is easy.
- Gear drives are compact.
- They have zero slip.
- They have good durability and precision

Disadvantages of Gear Drives:
- Installation is difficult.
- Manufacturing of gears is complex and expensive.
- Tooth wear may occur during power transmission.
- Inaccuracies in gear tooth cause noise and vibrations.

Use of Gears:
- To reverse the direction of rotation
To increase or decrease the speed of rotation
To move rotational motion to a different axis
To keep the rotation of two axis synchronized

### 1.1 Classification of Gears:
Classification of gears can be done according to relative position of the axes of revolution into three types. They are:

### 1.2 Gears for Parallel Shafts:
The motion between parallel shafts is same as to the rolling of two cylinders. Gears under this category are the following:

#### 1.2.1 Spur Gears:
Spur gears are the simplest form of gears having teeth parallel to the gear axis. The contact of two teeth takes place over the entire width along a line parallel to the axes of rotation. As gear rotate, the line of contact goes on shifting parallel to the shaft.

#### 1.2.2 Helical Gears:
In helical gear teeth are part of helix instead of straight across the gear parallel to the axis. The mating gears will have same helix angle but in opposite direction for proper mating. As the gear rotates, the contact shifts along the line of contact in involute helicoid across the teeth.

#### 1.2.3 Herringbone Gears:
Herringbone gears are also known as Double Helical Gears. Herringbone gears are made of two helical gears with opposite helix angles, which can be up to 45 degrees.

#### 1.2.4 Rack and Pinion:
In these gears the spur rack can be considered to be spur gear of infinite pitch radius with its axis of rotation placed at infinity parallel to that of pinion. The pinion rotates while the rack translates.

### 1.3 Gears for Intersecting Shafts:
The motion between two intersecting shafts is equivalent to the rolling of two cones. The gears used for intersecting shafts are called bevel gears. Gears under this category are following:

#### 1.3.1 Straight Bevel Gears:
Straight bevel gears are provided with straight teeth, radial to the point of intersection of the shaft axes and vary in cross section through the length inside generator of the cone. Straight Bevel Gears can be seen as modified version of straight spur gears in which teeth are made in conical direction instead of parallel to axis.

#### 1.3.2 Spiral Bevel Gears:
Bevel gears are made with their teeth are inclined at an angle to face of the bevel. Spiral gears are also known as helical bevels.

### 1.4 Gears for Skew Shafts:
The following gears are used to join two non-parallel and non-intersecting shafts.

#### 1.4.1 Hypoid Gears:
The Hypoid Gears are made of the frusta of hyperboloids of revolution. Two matching hypoid gears are made by revolving the same line of contact, these gears are not interchangeable.

#### 1.4.2 Worm Gears:
The Worm Gears are used to connect skewed shafts, but not necessarily at right angles. Teeth on worm gear are cut continuously like the threads on a screw. The gear meshing with the worm gear is known as worm wheel and combination is known as worm and worm wheel.

### 2. INVOLUTE SPUR GEAR
The function of a gear is to work smoothly while transmitting motion or torque. For this the angular velocity ratio at all times should remain constant. This aspect is explained here using various gear terminology that are peculiar to gears. Understanding of the definition of these terminologies helps to grasp the functioning of gears and the design of gears.
2.1 NOMENCLATURE OF INVOLUTE SPUR GEARS

Refer to the Figs.2.1 and 2.2 which show a portion of a pair of involute gears in mesh.

• Pitch surface: The surface of the imaginary rolling cylinder (cone, etc.) that replaces the toothed gear.
• Pitch surface: The surface of the imaginary rolling cylinder (cone, etc.) that replaces the toothed gear.
• Pitch circle: A normal section of the pitch surface.
• Addendum circle: A circle bounding the ends of the teeth, in a normal section of the gear.
• Dedendum circle or Root circle: The circle bounding the spaces between the teeth, in a normal section of the gear.
• Addendum: The radial distance between the pitch circle and the addendum circle.
• Dedendum: The radial distance between the pitch circle and the root circle.
• Clearance: The difference between the Dedendum of one gear and the addendum of the mating gear.
• Face of a tooth: That part of the tooth surface lying outside the pitch surface.
• Flank of a tooth: The part of the tooth surface lying inside the pitch surface.
• Top land: The top surface of a gear tooth.
• Bottom land: The bottom surface of the tooth space.
• Circular thickness (tooth thickness): The thickness of the tooth measured on the pitch circle. It is the length of an arc and not the length of a straight line.
• Tooth space: The space between successive teeth.
• Width of space: The distance between adjacent teeth measured on the pitch circle.
• Backlash: The difference between the tooth thickness of one gear and the tooth space of the mating gear.
Circular pitch $p$: The width of a tooth and a space, measured on the pitch circle. It is equal to the pitch circumference divided by the number of teeth. If,

$$p = \frac{\pi d}{z}$$

- $p$ - circular pitch, $z$ - number of teeth, $P$ - diametral pitch, $d$ - pitch diameter
- Diametral pitch $P$: The number of teeth of a gear per unit pitch diameter. The diametral pitch is hence the number of teeth divided by the pitch diameter.

$$P = \frac{z}{d}$$

- The product of the diametral pitch and the circular pitch equals $\pi$.

$$pP = \pi$$

- The effect of diametral pitch on the size of the gear tooth is shown in Fig. 2.3

Fig. 2.3 Variation of tooth size with diametral pitch

- Actual tooth size for various diametral pitches is shown in Fig. 2.4. The diametral pitches are standardized and these values are given Table 2.1.

<table>
<thead>
<tr>
<th>$p$</th>
<th>0.3</th>
<th>0.4</th>
<th>0.5</th>
<th>0.6</th>
<th>0.7</th>
<th>0.8</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.25</td>
<td>1.5</td>
<td>1.75</td>
<td>2</td>
<td>2.25</td>
<td>2.5</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>3.5</td>
<td>4</td>
<td>4.5</td>
<td>5</td>
<td>5.5</td>
<td>6</td>
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<td></td>
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<tr>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
<td>13</td>
<td></td>
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<tr>
<td>14</td>
<td>15</td>
<td>16</td>
<td>18</td>
<td>20</td>
<td>22</td>
<td>24</td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>28</td>
<td>30</td>
<td>33</td>
<td>36</td>
<td>39</td>
<td>42</td>
<td></td>
</tr>
<tr>
<td>45</td>
<td>50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Further increase is in terms of 5 mm</td>
</tr>
</tbody>
</table>

Table 2.1 Standard diametral pitches
Fig. 2.4 Actual size of the gear tooth for different diametral pitches

In metric system, module is used instead of diametral pitch. It is nothing but the inverse of diametral pitch. The standard modules for which cutters are readily available in the market are given in Table 2.2

Module m: Pitch diameter divided by number of teeth. The pitch diameter is usually specified in millimeters.

\[ m = \frac{d}{z} \]  

Table 2.2 Standard modules in mm

- Fillet Radius: The small radius that connects the profile of a tooth to the root circle.
- Crowning: Grinding of tooth edges to prevent edge loading is known as crowning.

This is shown in Fig. 2.5

Fig 2.5 Crowning of tooth to overcome edging

3. INTRODUCTION CATIAV5R20

CATIA is the leading solution for product success. It addresses all manufacturing organizations. CATIA can be applied to a wide variety of industries, from aerospace, automotive, and industrial machinery, to electronics, shipbuilding, plant design, and consumer goods. Today, CATIA is used to design anything from an airplane to jewelry and clothing. With the power and functional range to address the complete product development process, CATIA supports product engineering, from initial specification to product-in-service, in a fully-integrated manner. It facilitates reuse of product design knowledge and shortens development cycles, helping enterprises to accelerate their response to market needs.

CatiaV5R20 is an interactive Computer- Aided Design and Computer Aided Manufacturing system. The CAD functions automate the normal engineering, design and drafting capabilities found in today’s manufacturing companies. The CAM functions provide NC programming for modern machine tools using the CatiaV5 R16 design model to describe the finished part. CatiaV5R20 functions are divided into “applications” of common capabilities. These applications are supported by a prerequisite application called “CatiaV5R20 Gateway”.
CatiaV5R20 is fully three dimensional, double precision system that allows to accurately describing almost any geometric shape. By combining these shapes, one can design, analyze, and create drawings of products.

4. BASIC PROCEDURE FOR CREATING A 3-D MODEL IN CATIAV5R20:

Creation of a 3-D model in CatiaV5R20 can be performed using three workbenches i.e., sketcher, modeling and assembly.

**Sketcher:**

Sketcher is used to create two-dimensional representations of profiles associated within the part. We can create a rough outline of curves, and then specify conditions called constraints to define the shapes more precisely and capture our design intent. Each curve is referred to as a sketch object.

**Creating a new sketch:**

To create a new sketch, chose Start ➔ Mechanical Design ➔ Sketcher then select the reference plane or sketch plane in which the sketch is to be created.

**SKETCH PLANE**

The sketch plane is the plane that the sketch is located on. The sketch plane menu has the following options:

- **Face/Plane:** With this option, we can use the attachment face/plane icon to select a planar face or existing datum plane. If we select a datum plane, we can use the reverse direction button to reverse the direction of the normal to the plane.

- **XC-YC, YC-ZC, and ZC-XC:** With these options, we can create a sketch on one of the WCS planes. If we use this method, a datum plane and two datum axes are created as below.

**SPUR GEAR MODELING WITH INVOLUTE PROFILE**

Involute is a special case gear design profile for creating tooth. In this modeling we use some relations based on the gear terminology.

**Involute Gear Relations**

\[ m = \text{Module} \]

\[ \text{Angle1} = 20 \text{ Pressure Angle} \]

\[ \text{rp} = m \times N / 2 \quad \text{Pitch Circle} \]

\[ \text{rb} = \text{rp} \times \cos(a) \quad \text{Base circle} \]

\[ \text{rd} = \text{rp} - 1.25 \times m \quad \text{Dedendum Circle} \]

\[ \text{ra} = \text{rp} + m \quad \text{Addendum Circle} \]

\[ \text{Angle2} = 90 / N \times 1 \text{deg} \]

Fillet Radius = .30m to .40m

![Fig 3.1 Gear Terminology](image)
When you start CATIA, go to TOOLS->OPTIONS-infrastructure- part infrastructure and in Display select Parameters and Relations.

Then in Options-General in Parameters and Measures select with value and with formula in Parameters Tree View.

Open Part Design through Start Menu - We get the following window

Add the Knowledge toolbar to add the above relations
Select \( f(x) \) to add the relations for gear design:

Now it is time to enter some basic parameters that define gear. This is done by clicking at \( f(x) \) icon: And then when you see dialog box: Formulas. Part first select Parameter type (real, length or angle) click new parameter of type and then edit value. You can do this until all parameters are entered.

![Fig 3.4 Entering and editing formula](image)

When you enter parameters it is time to enter some formulas.

For, \( r_p \), \( r_b \), \( r_a \) and \( r_d \) we enter formulas by naming them and by clicking Add Formula. Formula editor will appear:

![Fig 3.5 Expanding Specification tree](image)

After typing all formulas and expanding specification tree you will see the following specifications tree as below.
Create two circles on the top plane and add the dimensions and double click the dimension to edit and put the radius in Dimension tab, right click on the Radius tab select Edit formula pitch circle for one and for other circle base circle from the relations we get the circles as shown in second fig.

Fig 3.6 Drawing and adding three axis line

Draw three axis lines as shown below. One is left side of vertical line and one is right side and the other is on the vertical plane. Now add the angle dimension between the axes lines one is between two end axes lines and the other is between vertical and left axis line. Between end axes line add 20° it is a pressure angle and other is 90/N°1deg as shown below.

We get the angles as shown below.
Fig 3.7 Drawing a circle

After applying all the relations we draw a circle by taking center as a intersection of right side inside circle and point on the above circle and left axis line intersection, we get the circle as shown below.

Fig 3.8 Continuing the profile creation

Now convert the circles into reference and hide them to continue the profile creation.

Fig 3.9 Creating addendum and dedendum circles

After hiding the previous circles, create two circles, namely addendum and dedendum circles for these circles also add the relations same as above, we get the circles with required dimensions.
After creating the circles Trim the unnecessary portion to get the involute profile as shown below and mirror the same other side to get the total teeth.

Adding of material to the tooth use PAD tool we get the following tooth with involute profile: It is selected from the sketch based feature. This feature is used to add the material normal to sketch or through a reference. In the part design use pad tool from the sketch based features and select the sketch then give the required height (Length).

To create the remaining teeth select Pattern tool: This feature is used to create the copy of number of instances of the existing part body in specified direction as shown below.
Adding of the material for supporting all teeth. Create a circle as shown below with the reference and create a shaft circle with key hole and pad it to add the material.

Pad: It is selected from the sketch based feature. This feature is used to add the material normal to sketch or through a reference. In the part design use pad tool from the sketch based features and select the sketch then give the required height (Length).

Fillets at roots: This is to provide strength to the gear tooth. We use that value in between 0.3m to 0.4m (m is module). For this also use formulaw and select all the roots as shown below and apply we get the fillets for all the teeth.

To display only gear hide the unnecessary features from the window and we get the final model as shown below with material property.
INTRODUCTION TO ANSYS

The company was founded in 1970 by Dr. John A. Swanson as Swanson Analysis Systems, Inc. SASI. Its primary purpose was to develop and market finite element analysis software for structural physics that could simulate static (stationary), dynamic (moving) and heat transfer (thermal) problems. SASI developed its business in parallel with the growth in computer technology and engineering needs. The company grew by 10 percent to 20 percent each year, and in 1994 it was sold to TA Associates. The new owners took SASI’s leading software, called ANSYS®, as their flagship product and designated ANSYS, Inc. as the new company name.

ANSYS, Inc. is an engineering simulation software (computer-aided engineering, or CAE) developer that is headquartered south of Pittsburgh in the Southpointe business park in Cecil Township, Pennsylvania, United States.

ANSYS was listed on the NASDAQ stock exchange in 1996. In late 2011, ANSYS received the highest possible score on its Smart Select Composite Ratings according to Investor's Business Daily.[5] The organization reinvests 15 percent of its revenues each year into research to continually refine the software.

ANSYS offers engineering simulation solution sets in engineering simulation that a design process requires. Companies in a wide variety of industries use ANSYS software. The tools put a virtual product through a rigorous testing procedure (such as crashing a car into a brick wall, or running for several years on a tarmac road) before it becomes a physical object.

ANSYS PRODUCTS

A. ANSYS MULTIPHYSICS

ANSYS Multiphysics software offers a comprehensive product solution for both multiphysics and single-physics analysis. The product includes structural, thermal, fluid and both high- and low-frequency electromagnetic analysis. The product also contains solutions for both direct and sequentially coupled physics problems including direct coupled-field elements and the ANSYS multi-field solver.

B. ANSYS WORKBENCH

The ANSYS Workbench platform is the framework upon which the industry’s broadest and deepest suite of advanced engineering simulation technology is built. An innovative project schematic view ties together the entire simulation process, guiding the user...
through even complex multiphysics analyses with drag-and-drop simplicity. With bi-directional CAD connectivity, powerful highly-automated meshing, a project-level update mechanism, pervasive parameter management and integrated optimization tools, the ANSYS Workbench platform delivers unprecedented productivity, enabling Simulation Driven Product Development.

C. ANSYS CFX

ANSYS CFX is a commercial Computational Fluid Dynamics (CFD) program, used to simulate fluid flow in a variety of applications. The ANSYS CFX product allows engineers to test systems in a virtual environment. The scalable program has been applied to the simulation of water flowing past ship hulls, gas turbine engines (including the compressors, combustion chamber, turbines and afterburners), aircraft aerodynamics, pumps, fans, HVAC systems, mixing vessels, hydro cyclones, vacuum cleaners, and more.

D. ANSYS FLUENT

ANSYS Fluent software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants. Special models that give the software the ability to model in-cylinder combustion, aero acoustics, turbo machinery, and multiphase systems have served to broaden its reach.

The ANSYS program is self-contained general purpose finite element program developed and maintained by Swanson Analysis Systems Inc. The program contains many routines, all interrelated and all for main purpose of achieving a solution to an engineering problem by Finite Element Method.

ANSYS provides a complete solution to design problems. It consists of powerful design capabilities like full parametric solid modeling, design optimization and auto meshing, which gives engineers full control over their analysis.

The following are the special features of ANSYS software:

- It includes bilinear elements.
- Heat flow analysis, fluid flow and element flow analysis can be done.
- Graphic package and extensive preprocessing and post processing.

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

<table>
<thead>
<tr>
<th>Table 4.2. Various stages of ANSYS</th>
</tr>
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<tbody>
<tr>
<td><strong>PRE-PROCESSOR PHASE</strong></td>
</tr>
<tr>
<td>Geometry definitions</td>
</tr>
<tr>
<td>Mesh generation</td>
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<tr>
<td>Materials definition</td>
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<tr>
<td>Constraint definition</td>
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<td>Load definition</td>
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</table>

MESHING

Manual Meshing:

In manual meshing the elements are smaller at joint. This is known as mesh refinement, and it enables the stress to be captured at the geometric discontinuity. Manual meshing is long and tedious process for models with any degree of geometric complication, but with useful tool emerging in pre-processes, the task is becoming easier.

Meshing controls:

The default meshing controls that the program uses may produce a mesh that is adequate for the model we are analyzing. In this case, we need not specify any meshing controls. However if we do use meshing controls we must set them before meshing the solid model.

Meshing controls allow us to establish the element shape, midside node placement and element size to be used in meshing the solid model, this step is one of the most important of the entire analysis for the decisions we make at this stage in the model development will profoundly affect the accuracy and economy of the analysis.
SMART SIZING OF ELEMENT:

Smart element sizing (Smart sizing) is a meshing feature that creates initial element sizes for free meshing operations. Smart sizing gives the mesher a better chance of creating reasonably shaped elements during automatic mesh generation.

FREE AND MAPPED MESH:

A free mesh is one that has no restrictions in terms of element shapes, and no specific pattern applied to it. Compared to a free mesh, a mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped mesh contains only quadrilateral (area) or only hexahedron (volume) elements. If this type of mesh is desired, the user must build the geometry as series of fairly regular volumes and/or areas that can accept a mapped mesh.

PRE-PROCESSOR

The pre-processor stage in ANSYS package involves the following:

- Specify the title, which is the name of the problem.
- Set the type of the analysis to be used, i.e., structural, thermal, fluid, or electro-magnetic, etc.
- Create the model – The model is drawn in 1D, 2D, or 3D space in the appropriate units (m, mm, in, etc). The model may be created in pre-processor, or it can be imported from another CAD drafting package through a neutral file format like IGES, STEP, ACIS, Para solid, DFX, ETC. The same units should be applied in all directions, otherwise results will be difficult to interpret, or in extreme cases the result will not show up mistakes made during loading and restraining of the model.
- Define the element type, this may be 1D, 2D or 3D, and specify the analysis type being carried out.
- Apply mesh – Mesh generation is the process of dividing the analysis continuum in to number of discrete parts or finite elements. The finer the mesh, the better the result, but the longer the analysis time. Therefore, the compromise between accuracy and solution speed is usually made.
- Assign the properties – Material properties (Young’s Modulus, Poisson’s ratio, density, and if applicable coefficient of expansion, friction, thermal conductivity, damping effect, specific heat, etc.) have to be defined.

SOLUTION:

- Apply the loads. Some type of load is actually applied to the analysis model. The loading may be in the form of a point load, pressure or a displacement in a stress analysis, a temperature or heat flux in a thermal analysis and a fluid pressure or velocity in a fluid analysis. The loads may be applied to a point, an edge, a surface or even to a complete body.
- Applying the boundary conditions. After applying load to the model in order to stop it accelerating infinitely through the computer virtually either at least one boundary condition must be applied.
- FE solver can be logically divided in to three main parts, the pre-solver, the mathematical-engine and the post-solver. The pre-solver creates the model defined by the pre-processor and formulates the mathematical representation of the model and calls the mathematical-engine, which calculates the results. The result returned to the solver and the post-solver is used to calculate the strains, stresses, etc., for each node within the component or continuum.

POST-PROCESSOR:

In this module, the results of the analysis are read and interpreted. All post-processor include the calculation of stress and strain in all of the X, Y, or Z directions, or indeed in the direction at an angle to the coordinate axes. The principle stress and strain may also be plotted.

MODAL ANALYSIS

Definition: We use Modal Analysis to determine the vibration characteristics (Natural frequencies and mode shapes) of a structure of a machine component while it is being designed. It also can be a starting point for another, more detailed, Dynamic Analysis, such as a transient dynamic, a harmonic response analysis, or a spectrum analysis.

Uses for Modal Analysis: The Natural frequencies and mode shapes are important parameters in the design of a structure for Dynamic loading conditions. They are also required if you want to do a spectrum analysis or a mode superposition harmonic or transient analysis.

We can do modal analysis on a pre stressed structure, such as a spinning turbine blade. Another useful feature is modal cyclic symmetry, which allows you to review the mode shapes of a cyclically symmetry structure by modeling just a sector of it.

Modal Analysis in the ANSYS family of products is a linear analysis. Any nonlinearity, such as plasticity and contact (gap) elements, are ignored even if they are defined. You can choose from several mode extraction methods: subspace, Block Lanczos, Power Dynamics, reduced, unsymmetrical, and damped. The damped method allows you to include damping in the structure. Details about mode extraction methods are covered later in this section.

Open Ansys Workbench through Start All programs – Ansys - Workbench

Model Analysis after modification and before modification. Select the Geometry tab and Model tab and link as shown below.
Project Schematic Window

Right Click on the mesh – Insert Sizing, Select Units from Units Menu as Metric (mm, kg, N, S etc.),

Put the cursor on body sizing – select all objects by using body and box selection method, select all by dragging a window – apply – Ok and keep the cursor on body sizing and enter element sizing 10mm in bottom details window.

Right Click on the mesh – Generate Mesh. The mesh will be generated as shown below.

Mesh

To add the material properties select all parts in the geometry – select required material in the bottom details window.

Apply Boundary Conditions:

Fixed Support: Right Click on analysis settings – insert – fixed support – select the bottom of spring - Apply as shown below.

Right Click on the solution – Solve.

Right click on the graph window – select all – Right click again Create Mode Shape Results – Right Click on solution – Evaluate all results

We get the results for 6 mode shapes at different frequencies as shown below
4.7 Deformations

1st Mode of Deformation

The deformation obtained in first mode is 230.02 mm at 33616 Hz natural frequency.

2nd Mode of Deformation

The deformation obtained in second mode is 413.47 mm at 41868 Hz natural frequency.

3rd Mode of Deformation

The deformation obtained in third mode is 293.76 mm at 45647 Hz natural frequency.

4th Mode of Deformation

The deformation obtained in fourth mode is 330.86 mm at 48983 Hz natural frequency.
5th Mode of Deformation

The deformation obtained in fifth mode is 352.25mm at 55437 Hz natural frequency.

6th Mode of Deformation

The deformation obtained in sixth mode is 339.29 mm at 55551 Hz natural frequency.

Table 4.3 deformations

<table>
<thead>
<tr>
<th>S.no</th>
<th>Frequency in hz</th>
<th>Deformation in mm</th>
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<td>1</td>
<td>33616</td>
<td>230.02</td>
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<tr>
<td>2</td>
<td>41868</td>
<td>413.47</td>
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<tr>
<td>3</td>
<td>45647</td>
<td>293.76</td>
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<tr>
<td>4</td>
<td>48983</td>
<td>330.86</td>
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<tr>
<td>5</td>
<td>55437</td>
<td>352.25</td>
</tr>
<tr>
<td>6</td>
<td>55551</td>
<td>339.29</td>
</tr>
</tbody>
</table>

STATIC STRUCTURAL ANALYSIS

Definition: A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

Structural analysis is probably the most common-application of the FEM. The term structural implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical, and mechanical components such as pistons, machine parts and tools. The primary unknowns (nodal degree of freedom) calculated in a structural analysis are displacements other qualities, such as strains, stresses and reaction forces are derived from the nodal displacements.

Loads in a Static Analysis: - Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not include significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure’s response are assumed to vary slowly with respect to time. The kinds of loading that can be applied in a static analysis

Include: -
- Externally applied forces pressures
- Steady-state internal forces (such as gravity or rotational velocity)
- Imposed (non-zero) displacements
- Temperatures (for thermal strain)
- Fluencies (for nuclear swelling)

Linear vs. Nonlinear Static Analysis: A static analysis can be either linear or non linear. All types of nonlinearities are allowed—large deformations, plasticity, creep, stress stiffening, contact (gap) elements, hyper elastic elements etc.

Overview of steps in a static analysis:

The process for static analysis consists of three main steps:

1. Build the Model: To build the model, specify the job name, analysis title and then define the element types, element real constants, material properties, and the model geometry. The structural elements can be linear or nonlinear. Material properties can be linear or nonlinear, isotropic or orthographic, and constant or temperature-dependent. The Young’s modules should also be defined.

2. Apply the loads obtain the solution: In this step, define the analysis type and options, apply loads, specify load step options, and begin the finite element solution. The loads that can be applied are:

   - Displacements—degree of freedom, constraints usually specified at modal boundaries to define rigid support points.
   - Forces—concentrated loads usually specified on the model exterior, moments.
   - Pressures—surface loads usually applied on the model exterior, temperatures.
   - Fluencies—applied to stuffy the effects of swelling or creep.

3. Review the results: Results from a static analysis include the nodal displacements, nodal and element stresses, nodal and element strains, element forces, nodal reaction forces etc.

MAIN WINDOW OF ANSYS WORKBENCH 14.5

Importing of the SPUR GEAR will be done after opening the workbench. For the supporting purpose of the geometry, the file format of CATIA will be changed to STEP format. This is to match up the graphical properties of the CATIA V5 to ANSYS WORKBENCH 14.5.

The full form of the STEP is Standard for the Exchange of Product model data which itself states that will exchange the graphical properties of models to other graphical user interfaces.

MATERIAL PROPERTIES

The material properties are the important factor which will be considered as the second preference after importing or creating the geometry. The procedure of material application, double click on the engineering data which will appear on the top of the analysis system. The analysis system which we are using in this project is Fatigue ANALYSIS. After opening the window of engineering data the material application will be done by selecting the add symbol in the general materials. In this project we are working on steel.

After importing the model into project schematic window drag and drop the Static Structural tab on to the screen from the toolbox window and link the geometry to geometry both will be linked together. Double click on the model it opens the mechanical window with object.

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Assigning of material:

From the outline tab – select the geometry – Part – from the bottom detailed window - material –Assignment – Select required material.

Mesh: To generate the meshing, there are two methods one is automatic mesh generation and the other is with required size meshing. In this we used size meshing with 5mm size meshing.

Meshed Geometry

To apply the displacement: Right click on the Analysis Settings – Insert – displacement – as shown above.

Fixed Support

To apply rotational velocity: Right click on the Analysis Settings – Insert – rotational velocity – Select the geometry as shown above.
Solution:

Right Click on solution – Solve. We get the required results.

Results Window

VELOCITY AT 4000 Rad/S

Equivalent Stress

The stress obtained is 27.399 MPa which is very low as compared to the yield strength of material (330MPa).

Total Deformation
The deformation of the body is very low as compared to the body size, so it will not be taken into the consideration.

**LOAD AT 5000 rad/s**

**Equivalent Stress**

The stress obtained is 42.81 which is very low as compared to the yield strength of material (330MPa).

**Total Deformation**

The deformation of the body is very low as compared to the body size, so it will not be taken into the consideration.

**Table 4.4 Deformation due to stress and velocity**

<table>
<thead>
<tr>
<th>S.NO</th>
<th>Velocity (rad/s)</th>
<th>STRESS(MPa)</th>
<th>DEFORMATION (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4000</td>
<td>27.339</td>
<td>0.00033302</td>
</tr>
<tr>
<td>2</td>
<td>5000</td>
<td>42.81</td>
<td>0.00052034</td>
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</tbody>
</table>

**5. RESULTS AND CONCLUSION**

**Results:**

**Static Structural Analysis**

<table>
<thead>
<tr>
<th>S.NO</th>
<th>Velocity (rad/s)</th>
<th>STRESS(MPa)</th>
<th>DEFORMATION (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4000</td>
<td>27.339</td>
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<tr>
<td>2</td>
<td>5000</td>
<td>42.81</td>
<td>0.00052034</td>
</tr>
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Modal analysis:

Table 4.6

<table>
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<th>S.no</th>
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</thead>
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<td>1</td>
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<td>339.29</td>
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</table>

Conclusion

The modeling of the component was done by using the advanced modeling software CATIA V5. The static structural analysis and modal analysis was done by using one of the most important numerical methods is FEA and the software used is ANSYS 14.0. According to the results the spur gear with involute profile will be having a good life and the structural steel was best material for the manufacturing.

6. REFERENCES