

# Profile Modification for Increasing the Tooth Strength in Spur Gear using CAD & CAE

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**Abstract-** This project examines the tooth failure in spur gears. In this project we are optimizing the gear profile geometry by using CAD &CAE and improving the gear tooth strength. Corrective measure is taken to avoid tooth damages by introducing profile modification in root fillet using cad. In general, spur gear with less than 17 numbers of teeth had the problem of undercutting during gear manufacturing process which minimizes the strength of gear at root. In this study a novel design method, namely circular root instead standard trochoidal root fillet is introduced in spur gear and analyzed using CAE software. The strength of these modified teeth is studied in comparison with the standard design. The analysis demonstrates that the novel design exhibit higher bending strength over standard trochoidal root fillet gear. The result reveals that the circular root fillet design is particularly suitable for lesser number of pinions and where as the trochoidal root fillet is more opt for higher number of teeth.

**Keywords –**SPUR GEAR, CAD& CAE, PRO-E

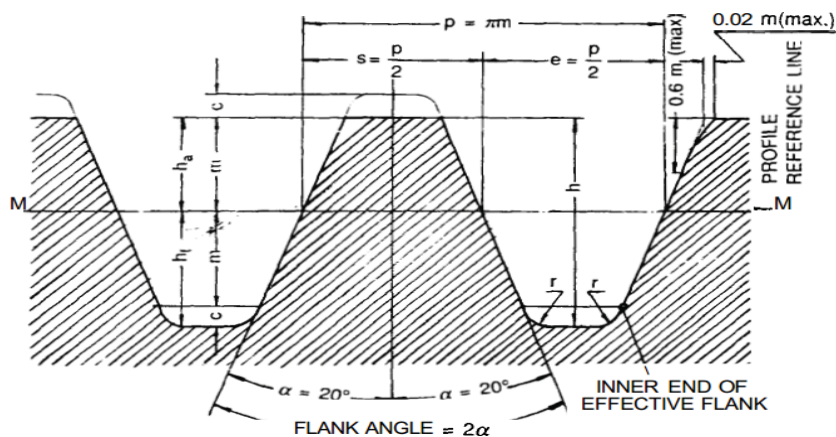
## I. INTRODUCTION

Spur gears are the simplest and most common type of gear. Their general form is a cylinder or disk. The teeth project radially, and with these straight-cut gears, the leading edges of the teeth are aligned parallel to the axis of rotation. These gears can only mesh correctly if they are fitted to parallel axles. The torque ratio can be determined by considering the force that a tooth of one gear exerts on a tooth of the other gear. Consider two teeth in contact at a point on the line joining the shaft axes of the two gears. The force will have both a radial and a circumferential component. Gears are a very useful simple machine. The torque ratio can be determined by considering the force that a tooth of one gear exerts on a tooth of the other gear. Consider two teeth in contact at a point on the line joining the shaft axes of the two gears.

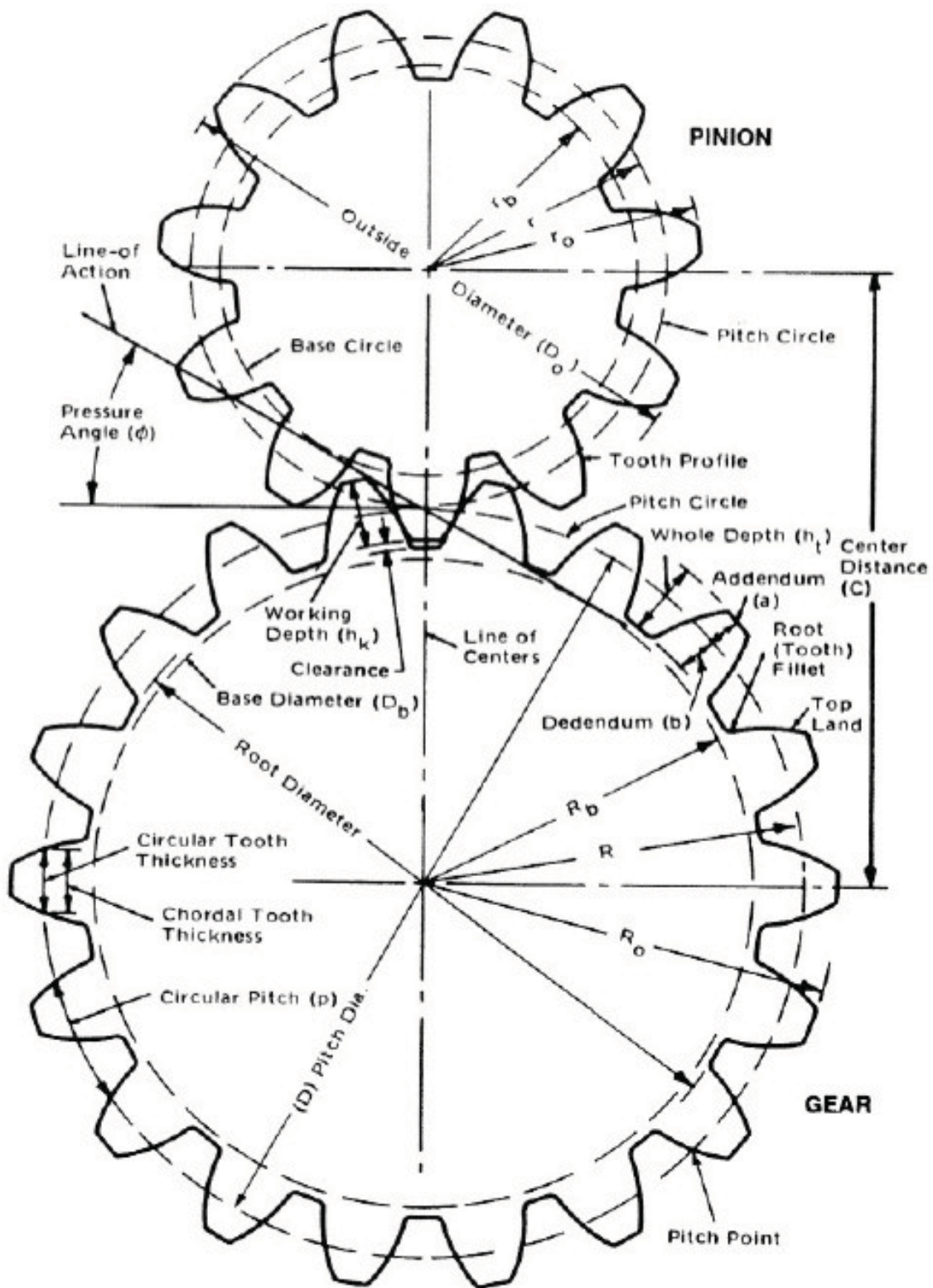


A gear is component within a transmission device. Transmit rotational force to another gear or device. A gear is different from a pulley in that a gear is a round wheel. Mesh with other gear teeth, allowing force to be fully transferred without slippage. Depending on their construction and arrangement, geared devices can transmit forces at different speeds, torques, or in a different direction, from the power source. Gears are a very useful simple machine. The most common situation is for a gear to mesh with another gear, but a gear can mesh with any device having compatible teeth, such as linear moving racks

## II. GEAR TOOTH TERMINOLOGY & GEOMETRY



Description	Pinion	Gear
Number of teeth	$z_1$	$z_2$
Pitch circle diameter	$d_1 = z_1 m$	$d_2 = z_2 m$
Tip circle diameter	$d_{a1} = d_1 + 2m$	$d_{a2} = d_2 + 2m$
Root circle diameter	$d_{r1} = d_1 - 2 \times 1.25m$	$d_{r2} = d_2 - 2 \times 1.25m$
Base circle diameter	$d_{b1} = d_1 \cos \alpha$	$d_{b2} = d_2 \cos \alpha$
Tooth thickness on pitch circle	$s = e = \frac{p}{2} = \frac{\pi m}{2}$	
Centredistance	$a = \frac{d_1 + d_2}{2} = m \frac{z_1 + z_2}{2}$	



### III. MODELLING SOFTWARES

There are different software's available for modeling some of them are:

1. Solid works
2. Pro-E
3. Ideas
4. Inventor
5. Mechanical desktop
6. Unigraphics
7. Catia v5

Pro-E (computer aided three dimensional interactive application) a multi platform CAD/CAM/CAE is used as the modeling tool in this paper

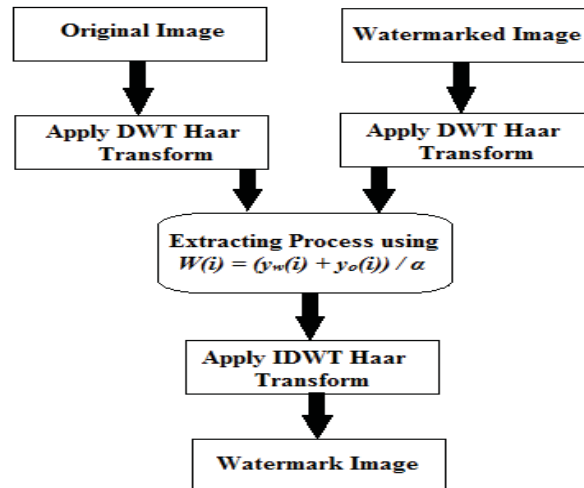


Figure 1. Watermark Extraction algorithm Block Diagram

Pro/ENGINEER, PTC's parametric, integrated 3D CAD/CAM/CAE solution, is used by discrete manufacturers for mechanical engineering, design and manufacturing. Created by Dr. Samuel P. Geisberg in the mid-1980s, Pro/ENGINEER was the industry's first successful parametric, 3D CAD modeling system. The parametric modeling approach uses parameters, dimensions, features, and relationships to capture intended product behavior and create a recipe which enables design automation and the optimization of design and product development processes. This powerful and rich design approach is used by companies whose product strategy is family-based or platform-driven, where a prescriptive design strategy is critical to the success of the design process by embedding engineering constraints and relationships to quickly optimize the design, or where the resulting geometry may be complex or based upon equations. Pro/ENGINEER provides a complete set of design, analysis and manufacturing capabilities on one, integral, scalable platform. These capabilities, include Solid Modeling, Surfacing, Rendering, Data Interoperability, Routed Systems Design, Simulation, Tolerance Analysis, and NC and Tooling Design.

#### 3.1 THE STEPS TO CREATE THE MODEL AND ASSEMBLY SECTION IS GIVEN BELOW.

CREATE THE WORKING DIRECTORY-First create the working directory to save the all model in one folder

File – set working directory – selects the required folder – ok.

SKETCH- This command is used to create the new sketch like circle, line, rectangle, ellipse, etc.,.

The pro-e window select the sketch icon and select the plane or surface want to sketch.

CIRCLE- This command is used to create the circle. Create circle by picking the center point and a point on the circle from Right Tool chest.

Pick the origin for the circle's center - pick a point on the circle's edge- click the middle mouse button – ok

ELLIPSE- This command is used to create the ellipse. Create ellipse by picking the center point and a minor radius point and major radius point, the minor and major radius of the ellipse is vertical and horizontal direction depend upon the shape of ellipse we want.

Select the ellipse icon from right tool chest- Pick the center for the ellipse – pick the minor radius of ellipse point and pick the major radius of the ellipse- click the middle mouse button – ok

LINE- This command is used to create the line. Create the line by start point and end point.

Select the line icon from the right Tool chest – click the start point of the line – click the end point of the circle - enter

ARC- This command is used to create the arc. Create the arc by using three points. The three points are start point, end point and center point of the arc.

Select the arc icon from the right Tool chest – click the start point of the arc – click the end point of the arc and click the middle point of the arc –enter.

The dimension of the arc is modified by double click on the arc then the dimension will appear in the pop up box, then provide the value of the arc.

CREATE THE HEXAGON – The hexagon is created by insert foreign data icon in the Right Tool chest.

Insert foreign data from Palette into active object - scroll down to see the hexagon - double-click hexagon- Place the hexagon on the sketch by picking a position - with the left mouse button, drag and drop the center of the hexagon at the origin - modify Scale value to the required size – click enter

RECTANGLE- This command is used to create the rectangle and square.

Click the rectangle icon in the right Tool chest – click the lower left point of the rectangle and higher right corner of the rectangle we want to draw.

After drawing the rectangle the dimension of the rectangle is provided by the pick the dimension command from the dimension icon in the right tool chest of the pro-e software.

DIMENSION- This command is used to provide the dimension of the sketched entities the entities may be circle, line, rectangle, ellipse, etc,...

The dimension is provide to the sketch by select the dimension icon from the right tool chest then select the sketched entities and press the middle mouse button to finish the dimensioning. To change the dimension of the sketched entities by just double clicks the dimension line of created sketch.

EXTRUDE – This command is used to create the material (to make 3D object from 2D sketch) from the sketched entities. The entities may be circle, line or rectangle, etc,...

Select the extrude icon from the right tool chest then select the sketched part in the window, enter the extrude length and press the middle mouse button to finish the extrude command. There is a provision for removing material in pro –e which is called cut. The main condition to create the solid model is the sketched section must be closed.

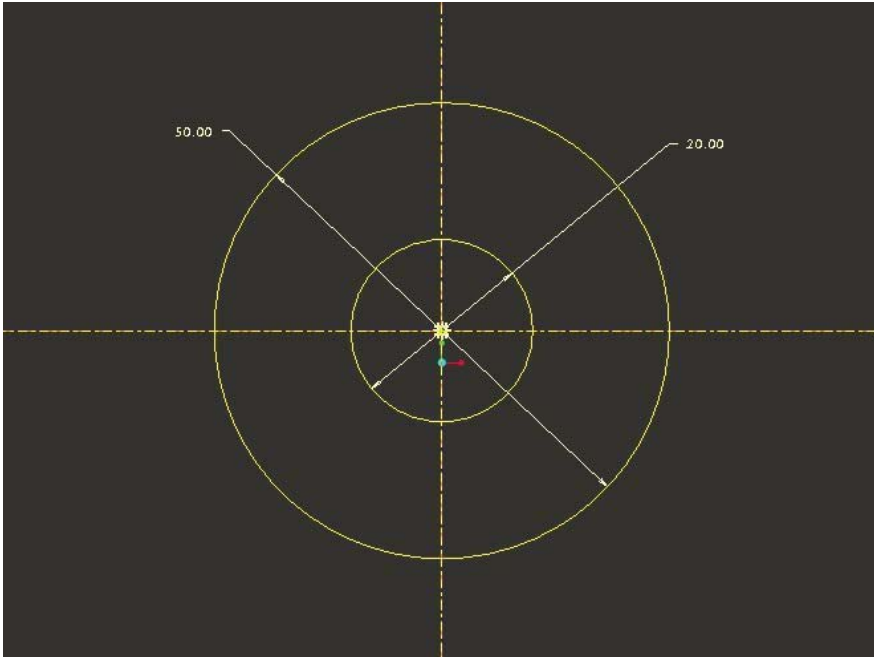
REVOLVE- This command is used to create the material from taking the one axis and sketched entities. The axis is the center of the revolved part. The revolve angle should between 0 degree to 360 degree.

Select the revolve icon from the right tool chest then select the sketched part and axis of the object in the graphical window, enter the revolve angle and press the middle mouse button to finish the extrude command.

#### SWEEP FEATURES

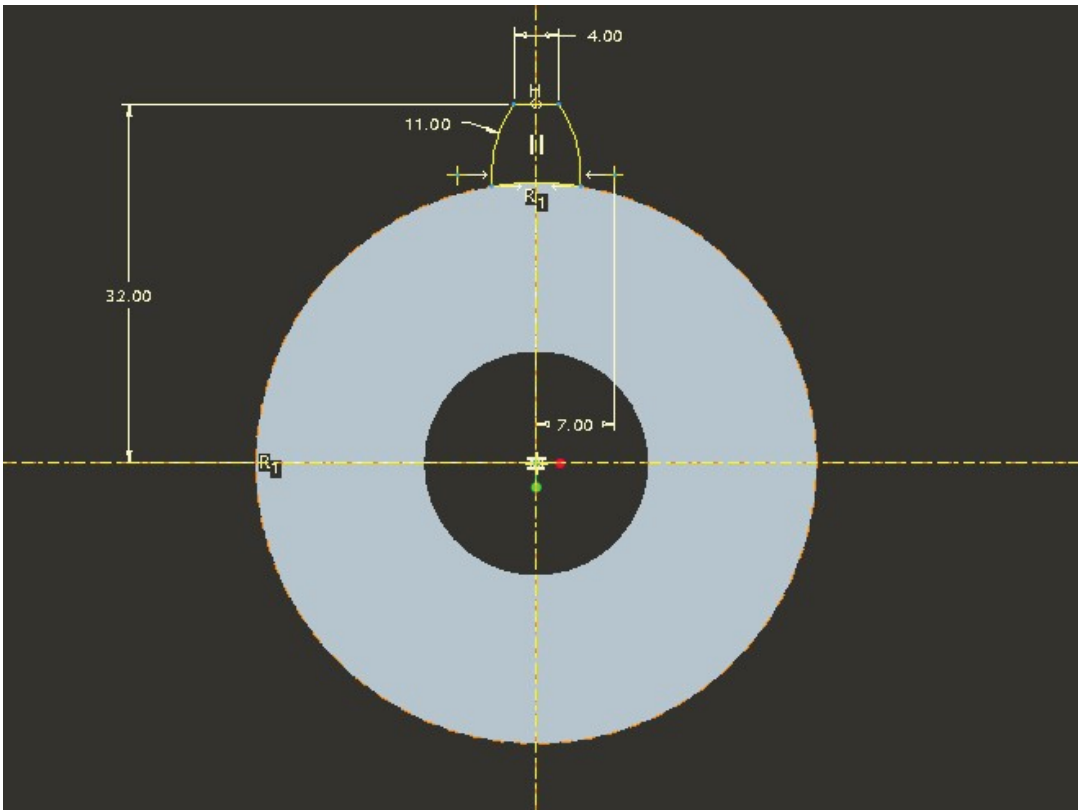
The Sweep option extrudes a section along a defined trajectory. The order of operation is to first create a trajectory and then a section. A trajectory is a path along which a section is swept. The trajectory for a sweep feature can be sketched or selected. The Sweep option of protrusion is similar to the Extrude option. The only difference is that in the case of the Extrude option, the feature is extruded in a direction normal to the sketching plane, but in the case of the Sweep option, the section is swept along the sketched or selected trajectory. The trajectory can be open or closed. Normal sketching tools are used for sketching the trajectory. The cross-section of the swept feature remains constant throughout the sweep

#### *2D SKETCH FOR SPUR GEAR*



The 2d sketch for creating the spur gear is sketched in the sketcher window using circle command

*2D SKETCH FOR SPUR GEAR TEETH*



The teeth profile which is to be formed on the spur gear is drawn in the sketcher window using line and arc command.

## IV. FEA & FEA SOFTWARES

### 4.1 FINITE ELEMENT ANALYSIS

The finite element method is numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in almost every industry.

In more and more engineering situations today, we find that it is necessary to obtain approximate solutions to problem rather than exact closed form solution. It is not possible to obtain analytical mathematical solutions for many engineering problems. An analytical solutions is a mathematical expression that gives the values of the desired unknown quantity at any location in the body, as consequence it is valid for infinite number of location in the body. For problems involving complex material properties and boundary conditions, the engineer resorts to numerical methods that provide approximate, but acceptable solutions. The finite element method has become a powerful tool for the numerical solutions of a wide range of engineering problems. It has been developed simultaneously with the increasing use of the high- speed electronic digital computers and with the growing emphasis on numerical methods for engineering analysis. This method started as a generalization of the structural idea to some problems of elastic continuum problem, started in terms of different equations

### 4.2 FEA SOFTWARES

There are different software's available for FEA some of them are:

- ALTAIR HYPER WORKS
- ANSYS
- NASTRAN
- COSMOS
- LS –DYNA

ANSYS is used as the FEA tool

#### A.ANSYS

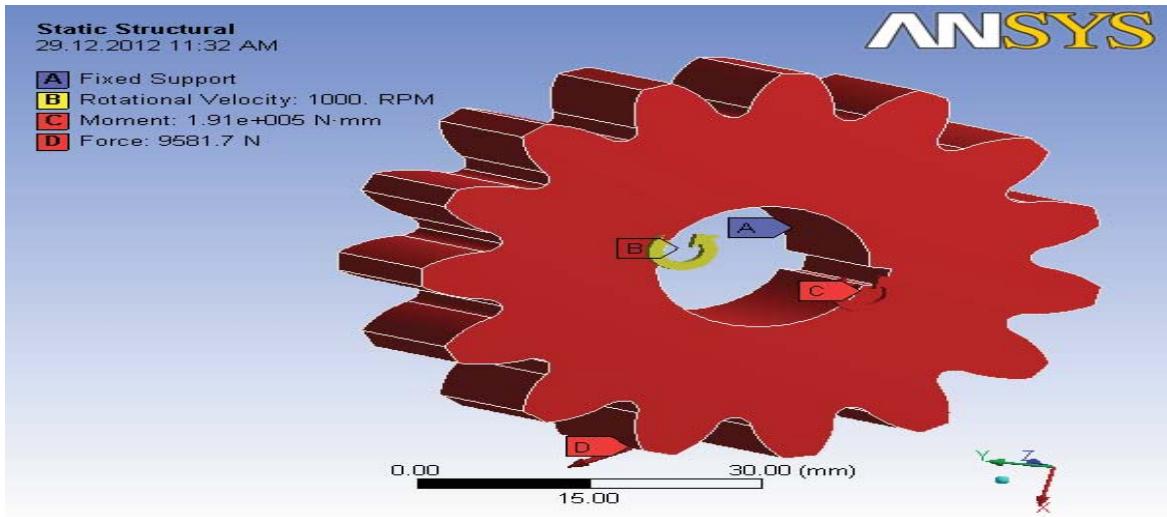
ANSYS is an engineering simulation software provider founded by software engineer John Swanson. It develops general-purpose finite element analysis and computational fluid dynamics software. While ANSYS has developed a range of computer-aided engineering (CAE) products, it is perhaps best known for its ANSYS Mechanical and ANSYS Multiphysics products. ANSYS Mechanical and ANSYS Multiphysics software are non exportable analysis tools incorporating pre-processing (geometry creation, meshing), solver and post-processing modules in a graphical user interface. These are general-purpose finite element modeling packages for numerically solving mechanical problems, including static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

#### B.ANALYZING THE MODEL – STEP BY STEP PROCEDURE

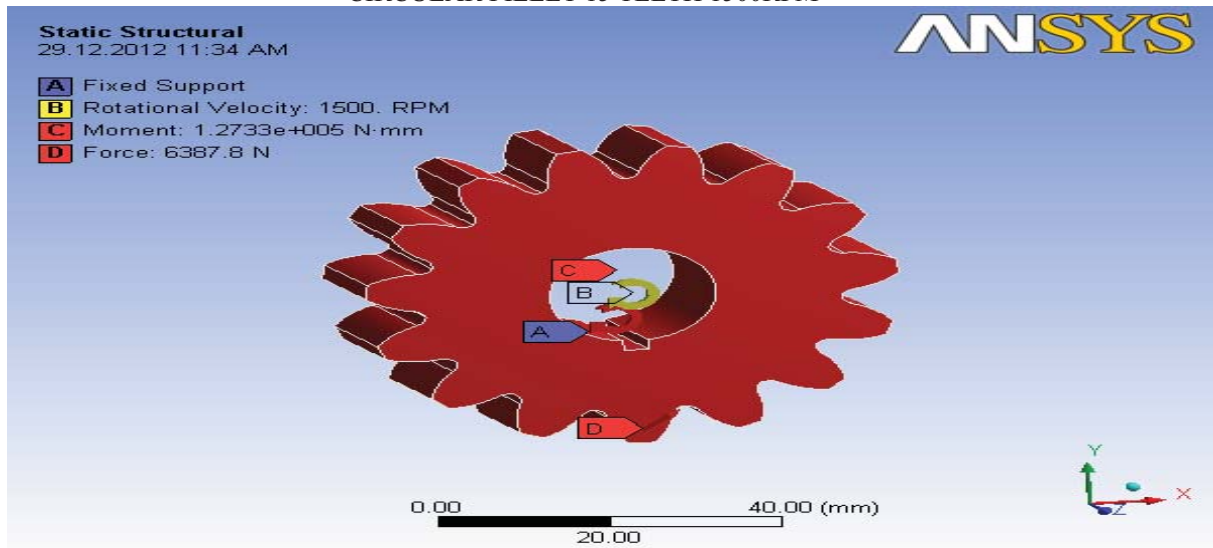
- The 3D model of the gear is converted as **iges** format through the PRO-E software
- The **IGES (Initial Graphic Exchange System)** format is suitable to import in the ANSYS Workbench for analyzing
- open the ANSYS workbench
- Create new geometry
- File – import external geometry file – generate
- Project – new mesh
- Defaults – physical preference – mechanical
- Advanced – relevance center – fine
- Advanced – element size – 0 mm
- Right click the mesh in tree view – generate mesh
- Project – convert to simulation – yes
- Select the solid in geometry tree
- Definition – material – import – work bench samples – select carbon steel and stainless steel – ok
- New analysis – static structural
- Static structural – right click – insert – fixed support
- Select the face
- Geometry – apply
- Static structural – right click – insert – force
- Geometry – apply
- Magnitude – N
- Static structural – right click – insert – moment

- Geometry – apply
- Magnitude –N/mm
- Static structural – right click – insert –rotational velocity
- Geometry – apply
- Magnitude –Rpm
- Solution – insert the total deformation and equivalent stress.
- Right click the solution icon in the tree – solve
- After solve the analysis – take the reading of above mentioned items.
- The all results are taken in a picture – and save it to the required folder in the system
- The all readings are tabulated

*C.CIRCULAR FILLET 15 TEETH 1000RPM*

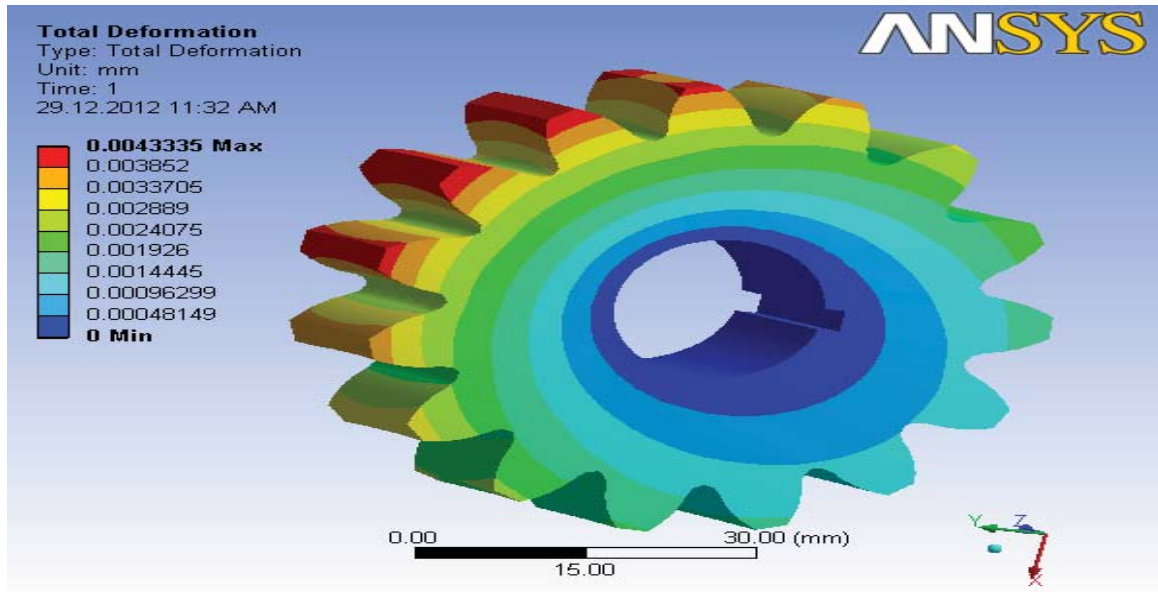


*CIRCULAR FILLET 15 TEETH 1500RPM*

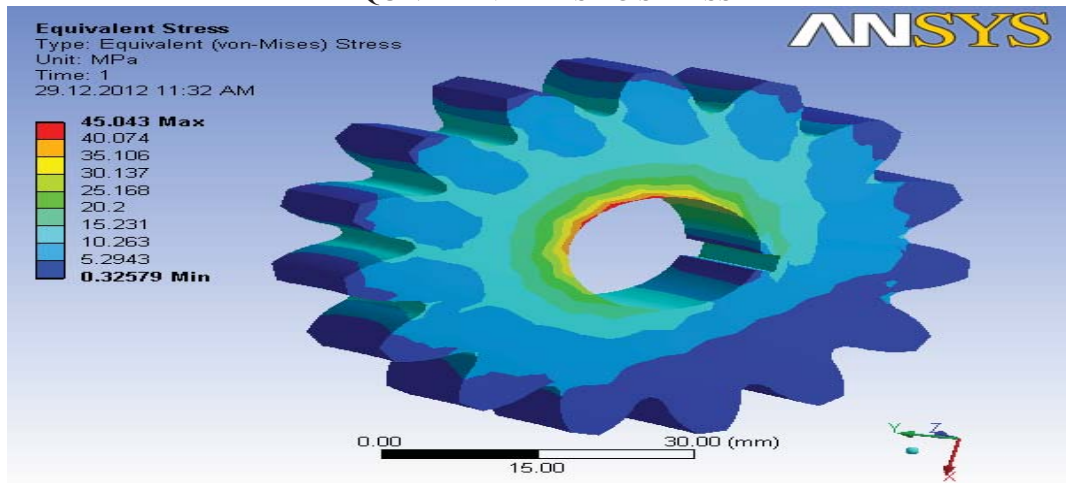


STATIC STRUCTRUL DEFINED ON SPUR GEAR IN ANSYS WORKBENCH CIRCULAR FILLET 15 TEETH 1000RPM  
 RESULTS FOR MODEL  
 TOTAL DEFORMATION

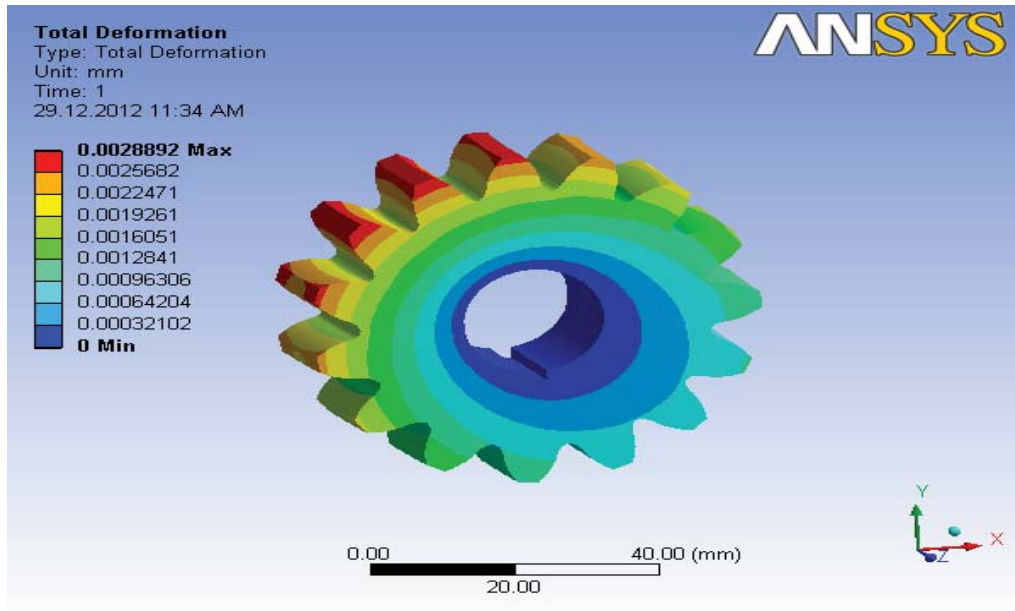




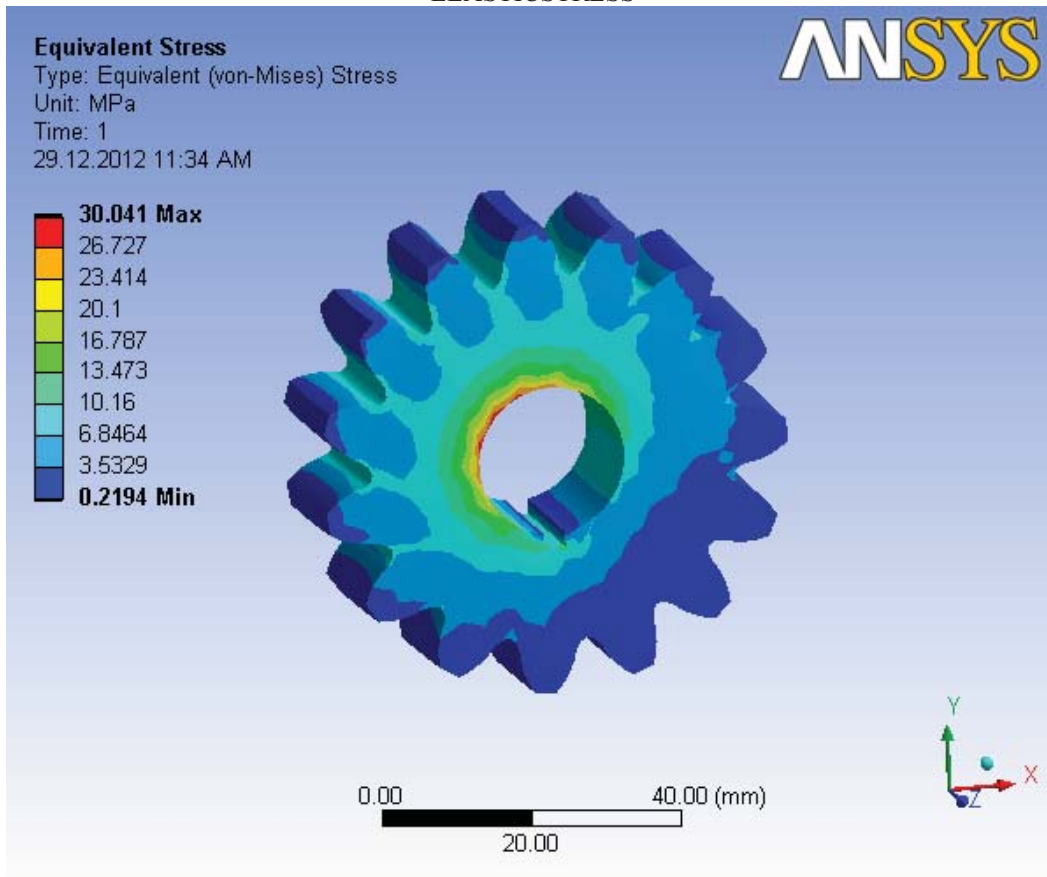
EQUIVALENT ELASTIC STRESS



CIRCULAR FILLET 15 TEETH 1500RPM  
RESULTS FOR MODEL  
TOTAL DEFORMATION



EQUIVALENT  
ELASTICSTRESS



RESULTS OF ABOVE MATERIALS  
ALLOY STRUCTURAL STEEL

CIRCULAR FILLET 15 TEETH 1000RPM

<b>ALLOY STRUCTURAL STEEL</b>	<b>MIN</b>	<b>MAX</b>
<b>Total deformation (mm)</b>	0	0.043335
<b>Equivalent Elastic Stress(Mpa)</b>	0.32579	45.043

**CIRCULAR FILLET 15 TEETH 1500RPM**

<b>ALLOY STRUCTURAL STEEL</b>	<b>MIN</b>	<b>MAX</b>
<b>Total deformation (mm)</b>	0	0.0028892
<b>Equivalent Elastic Stress(Mpa)</b>	0.2194	30.041

### V. CONCLUSION

The paper carried out by us will make an impressive mark in the field of automobile. This project we are study about the spur gear. While carrying out this project we are able to study about the 3D modelling software (PRO-E) and Study about the analyzing software (ansys) to develop our basic knowledge to know about the industrial design.

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