POLITECNICO DI TORINO

Faculty of Mechanical Engineering

Master of Mechanical Engineering

Master’s Thesis

Academic Year 2021-2022

Estimate an analytical expression that can predict the stiffness of the internal Spur gears tooth by means of parametric FE model

Supervisor:

Carlo Rosso

Co-supervisors:

Fabio Bruzzone

Candidate:

Marikallara Rajashekarappa Kenchappa

Table of Contents

Figures Index

Tables Index

ACKNOWLEDGEMENT

First and foremost, I offer my sincerest gratitude to my supervisor, professor Carlo rosso, who has supported me throughout my thesis with his patience and knowledge whilst allowing me the room to work in my own way. I attribute the level of my master’s degree to his encouragement and effort and without him this thesis, too, would not have been completed or written. One simply could not wish for a better or friendlier supervisor.

I would like to thank Department of Mechanical Engineering for providing me the support and assistance.

Finally, I would like to thank my parents and my friends in Torino for their continuous support and encouragement.

ABSTRACT

Gear is the one of the important machine elements in the mechanical power transmission system. Spur gear is most basic gear used to transmit power between parallel shafts. Spur gear generally fails by bending failure or contact failure. This project analyses the stiffness characteristics of an internal spur gear tooth under nominal or normal loading conditions. The tooth profile is generated using Solid-Works and the analysis is carried out by Finite Element Method (FEM) using same software. The stresses and displacements at the tooth root are evaluated by static simulation using FEM. A stiffness is calculated using force and displacement used in the previous simulation. The produced findings were then correlated with the original specifications for the internal spur gears to formulate an equation for stiffness using MATLAB curve fitting tool.

1. Introduction

Gears are machine parts used to transmit rotational motion, or torque, between two axes. Each gear is typically circular and has teeth that operate in mesh with the teeth of a complementary gear. Teeth in mesh must have a similar shape to mesh properly. Gears are generally used as a simple machine to produce a change in torque, which is achieved through their gear ratio. A gear assembly with a unitary ratio can be replaced with a universal joint, or a pair of such joints if axes are to be parallel.

Two or more pairs of meshing gears are referred to as a gear train or transmission. The same meshing mechanism can combine a circular gear with a linear toothed part, a rack, to combine rotation with translation.

History

Gears have been in use as early as the 4th century BC in China during the late East Zhou dynasty. The oldest gears found in Europe are those of the Antikythera mechanism, designed to calculate astronomical positions, dating back to 150-100 BC. Other use of gears can be linked to Hero of Alexandria, in Roman Egypt in 50 BC, which can be traced back to the development of mechanics in the Alexandrian school in the 3rd century BC in Ptolemaic Egypt, primarily done by Archimedes (287 – 212 BC) the Greek philosopher. Examples of old gear applications include but are not limited to:

- The Antikythera mechanism (2nd century BC)
- The South-pointing chariot of Ma Jun (200 – 265 AD)
- The mechanical clocks built in China (725 AD)
- The water-lifting device invented by Al-Jazari (1206 AD)
- The cotton gin in the Indian subcontinent (13th -14th centuries) which saw the invention of worm gears.
- The Salisbury Cathedral clock (1386 AD) rumoured to have the oldest still functional geared mechanical clock.

Replacing other drive mechanisms

Gears have a similar function to belt pulley systems, as far as torque transmission goes. The advantages gears offer is the definite transmission ratio due to the absence of slippage and the reduced number of parts in the mechanism. The downsides are the high manufacturing cost and the lubrication requirements which result in a high operating cost per unit of time.

Material

The earliest gear material was wood, which was later replaced by nonferrous alloys, powder metallurgy, cast irons and plastics post-industrialization. Nowadays, the most common material used is steel due to its high strength-to-weight ratio. Plastics are used in low torque applications where cost and weight can be a concern. Plastics also have the advantage of high dirt tolerance compared to steels as well as the possibility of operating without lubrication, and the reduction of repair costs. Plastic gears have been used in applications such as copy machines, printers, servo motors, and radios.

Types of gear

There are nine types of Gear. They can be classified according to the placement of the teeth (external – internal), the orientation of the leading edge of the teeth, or the layout of the gear axes.

The following is a list of the most common gear types found in the modern industry.

External gears: The teeth are formed on the outer surface of a cylinder or cone.

Internal gears: The teeth are formed on the inner surface of a cylinder of cone.

Spur gears: Straight-cut gears, which are the simplest gears of all, and commonly have an involute tooth profile.

Helical gears: An upgrade of spur gears, with the addition of a helix angle. They allow smoother meshing and can be found in parallel or crossed orientations.

Double helical gears: Also known as Herringbone gears. They consist of two helical gears mounted together with opposite directions to solve the axial thrust problem of helical gears.

Bevel gears: They have a conical shape and are used to transmit rotational motion between non-parallel axes.

Face gears: A particular type of bevel gears where the pinion is a spur gear.

Spiral bevel gears: They have similar advantages as helical gears do to straight-cut spur gears.

Hypoid gears: Similar to spiral bevel gears but whose axes do not intersect.

Worm gears: They resemble screws and mesh with a worm wheel that resembles a spur gear. They typically have high gear ratios. Axes are crossed but do not intersect.

Non-circular gears: They're characterized by a variable transmission ratio and are used for well-defined special purposes.

Rack and pinion: It convert rotational motion to translational motion and can be thought of a gear with infinite radius.

Epicyclic Gears: Gear assemblies where one or more gear axes move. Examples are mechanical differentials and sun/planet gearing

1.1. Internal Spur Gear

Figure 1. Isometric view of internal spur gear

An internal spur gear, in combination with a standard spur-gear pinion, **provides a compact drive mechanism for transmitting motion between parallel shafts that rotate in the same direction**. The internal gear is a wheel that has teeth cut on the inside of its rim and the pinion is housed inside the wheel.

Internal gears are primarily used for **planetary gear drives**. Spur gears are generally seen as best for applications that require speed reduction and torque multiplication, such as ball mills and crushing equipment.

Applications

Spur gears are used for a wide range of speed ratios in a variety of mechanical applications, such as **clocks, electric screwdrivers, pumps, watering systems, material handling equipment, power plant machinery, and clothes washing and drying machines**.

2. Nomenclature

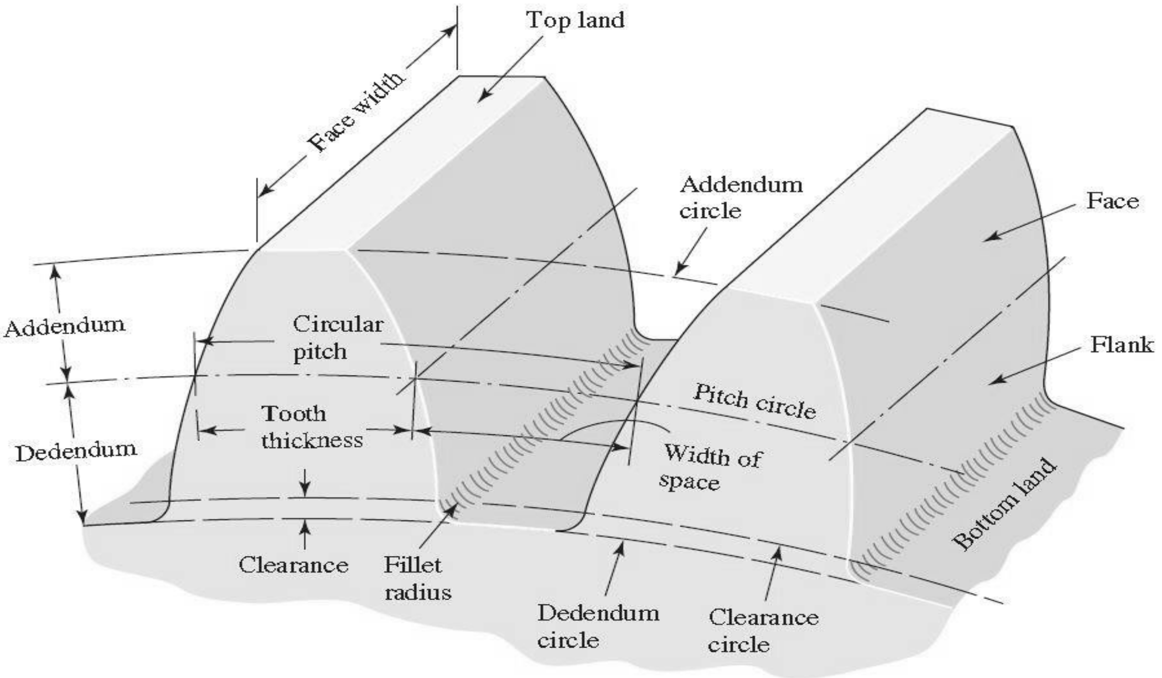


Figure 2. Nomenclature of spur-gear teeth

The terminology of spur gear teeth is shown in figure 2. The ***pitch circle*** is a theoretical circle upon which are based all calculations; its diameter is the ***pitch diameter***. The pitch circles of two mating gears are tangent one to the other.

The ***circular pitch p*** is the distance, from a point on one tooth to a corresponding point on an adjacent tooth. Therefore, the circular pitch is equal to the sum of the ***tooth thickness*** and the ***width of space.***

The ***module m*** is the ratio of the pitch diameter to the number of teeth. Generally, the adopted unit of length is the millimetre. The module is the index of the tooth size in SI.

The ***diametral pitch P*** is the ratio of the number of teeth on the gear to the pitch diameter. Therefore, it is the inverse of the module. It is expressed as teeth per inch.

The ***addendum a*** is the radial distance from the ***top land*** to the pitch circle.

The ***dedendum b*** is the radial distance from the bottom land to the pitch circle. The whole ***depth*** h_t is the sum of the addendum and the dedendum.

The ***clearance circle*** is a circle that is tangent to the addendum circle of the mating gear.

The ***clearance c*** is the amount by which the dedendum in each gear exceeds the addendum of its mating gear. The ***backlash*** is the amount by which the width of space of a tooth exceeds the tooth thickness of the engaging tooth measured on the pitch circle.

Rim thickness ratio is considered with the variation of the module.

3. Contact stress and stiffness for gear

3.1. Hertz Contact Stress: -

The study of deformation of solids which are in contact at one or more points is called Contact Mechanics. Pressures and Adhesion acting perpendicular to the contact surfaces are its central aspects. The present study deals with the cylinders with two parallel axes which are in contact. The contact in between is a non-Adhesive contact.

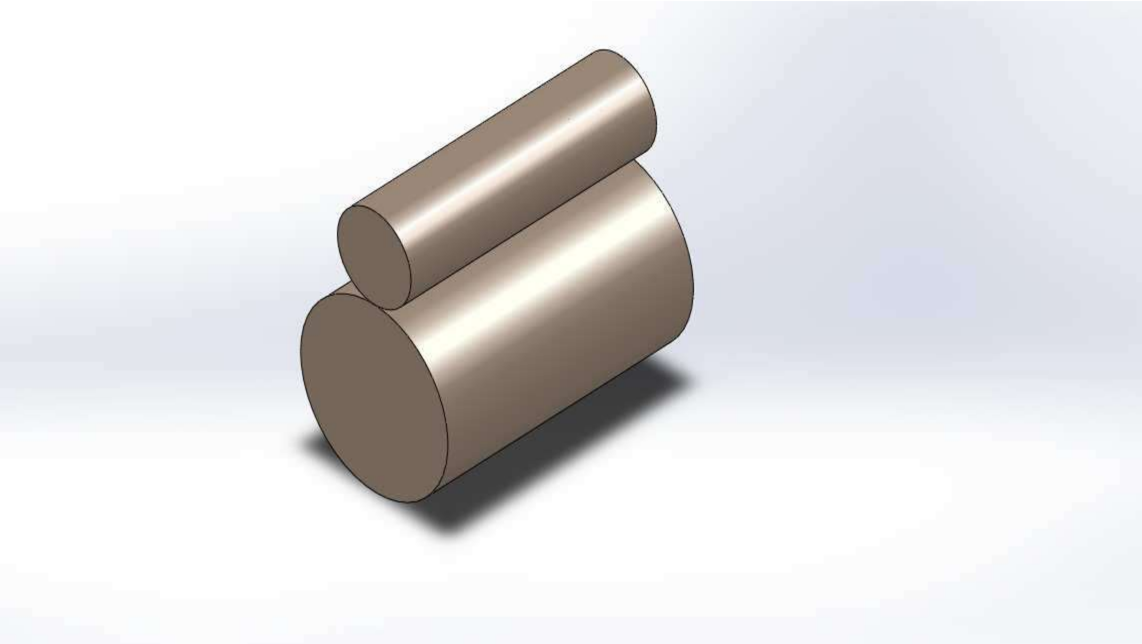


Figure 3. Cylinders

The Cylinders used in this study are having a length L, d_1 and d_2 as diameters. On application of Force F, on the surfaces of the cylinders a narrow rectangle of width 2b and length L is generated at the contact surfaces. A pressure is developed in the rectangle area in an elliptical shape. The half width of the ellipse is given by

$$C = \sqrt{\frac{2F}{\pi L}\frac{\frac{1-v_1^2}{E_1}+\frac{1-v_2^2}{E_2}}{1/d_1 + 1/d_2}} \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\ equation\ (1)$$

Where

- C is ellipse half width
- F is Force acting on surface of Cylinder
- L is length of Cylinder
- d_1 and d_2 are diameters of Cylinder
- ν1, ν2 represent the Poisson's ratio of cylinder material
- E1, E2 represent Young's Modulus of the materials.

The maximum contact pressure in between the cylinders acts along a longitudinal line at the centre of the rectangular contact area, which is given by

$$P_{max} = \frac{2.F}{\pi.C.L} \text{ …………………… equation (2)}$$

Where

- F is force acting on the Cylinders
- C is half width of the ellipse
- L is length of Cylinder
- P_{max} is the maximum pressure generated

3.2. Static Analysis: -

The Lewis Formula (Stress Calculation) The analysis of bending stress in gear tooth was done by Mr. Wilfred Lewis in his paper, 'The investigation of the strength of gear tooth' submitted at the Engineers club of Philadelphia in 1892. Even today, the Lewis equation is considered as the basic equation in the design of gears. Wilfred Lewis was the first person to give the formula for bending stress in gear teeth using the bending of a cantilevered beam to simulate stresses acting on a gear tooth shown in Cross-section =b*t, height = h, Load=Ft uniform across the face.

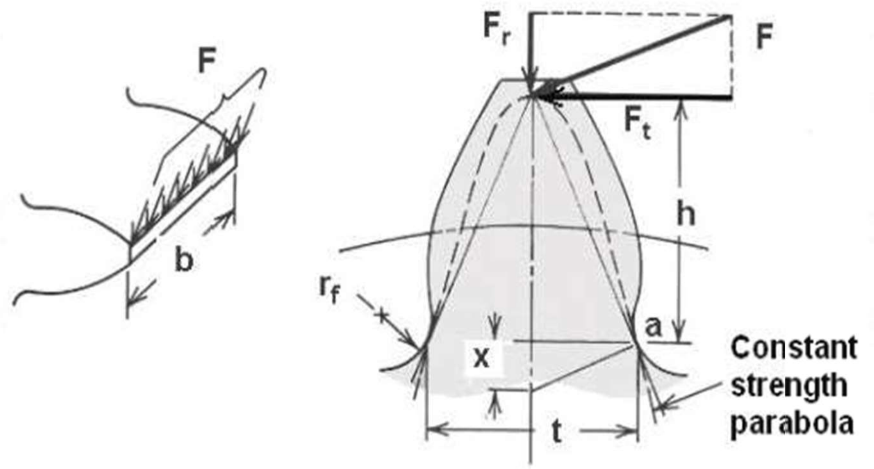


Figure 4. Lewis Equation

3.3. Stiffness: -

Stiffness in mechanical engineering is **an indicator of the tendency for an element to return to its original form after being subjected to a force**.

$$k = \frac{F}{\delta} \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots . \text{equation (3)}$$

Where

- k stiffness *(N/mm)*
- *F is force or load (N/kg)*
- δ is a displacement(mm)

The calculation of gear stiffness is important **for determining the load distribution between the gear teeth when two sets of teeth are in contact**. In the literature, several techniques and theoretical equations were developed to identify the single tooth deformation of gears. These methods generally depend on numerical approaches and theory of elasticity.

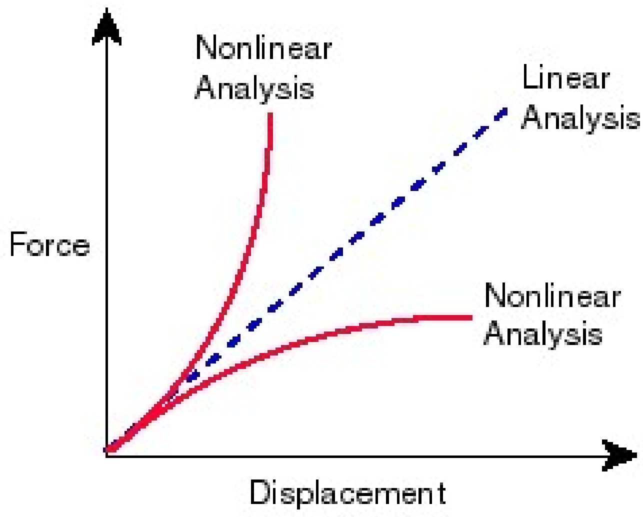


Figure 5. Stiffness curve

3.4. F.E. A Modelling and Design of Gear Tooth for finding stiffness: -

Internal spur gears are the most common form of gears which are used to transfer the motion between the parallel shafts. The main concerns while designing an internal spur gear include generation. In earlier days to design an involute spur gear there are many theoretical procedures to draw an approximate internal, but no procedure was present to draw a perfect involute for performing Analysis. In the present day with the 3-D modeling software's it is easy to generate the internal spur gear with exact internal teeth. For the current project, the internal spur gear with perfect involute is generated from Solid-Works.

4. Design procedure

The internal spur gear used for the current analysis has the following specifications

1. Number of tooth **Z** (50 to 250)

2. Module **m** (1 to 5 mm)

3. Pressure Angle ***α*** (20)

4. Rim thickness (3.5 to 5)

Parameters	Symbols	Formulas
Root Diameter	RD	“z" * "m" + 2 * 1.25 * "m"
Pitch Circle Diameter	PCD	"z" * "m"
Outer Circle Diameter	OCD	"z" * "m" + 2 * 1.25 * "m" + 2 * "rim" * "m"
Internal Diameter	ID	"z" * "m" - 2 * "m"
Fillet		0.38 * "m"
Width of tooth		("m" * PI) / 2

Table 1.Gear Parameters

The gear is generated using above equations as expressions in Solid-Works. To model the first configuration, we will choose the approximate values as mentioned in table 2.

Input Data	
Module m (mm)	1
Number of teeth (z)	60
Rim thickness	3.5
Tooth width (mm)	10
alpha (degree)	20

Table 2. Input data

Using the above equations, a 2-D sketch of the spur gear is generated, it is now extruded to length equal to Face Width to obtain 3-D Internal Spur Gear.

Design Specifications	
Outer Diameter	69.5
Root diameter	62.5
Pitch diameter	60
Inside diameter	58
alpha (degree)	20
Width of tooth	1.6
Fillet	0.38
tooth Extrude	10
Rim Extrude	10

Table 3. Specifications of gear

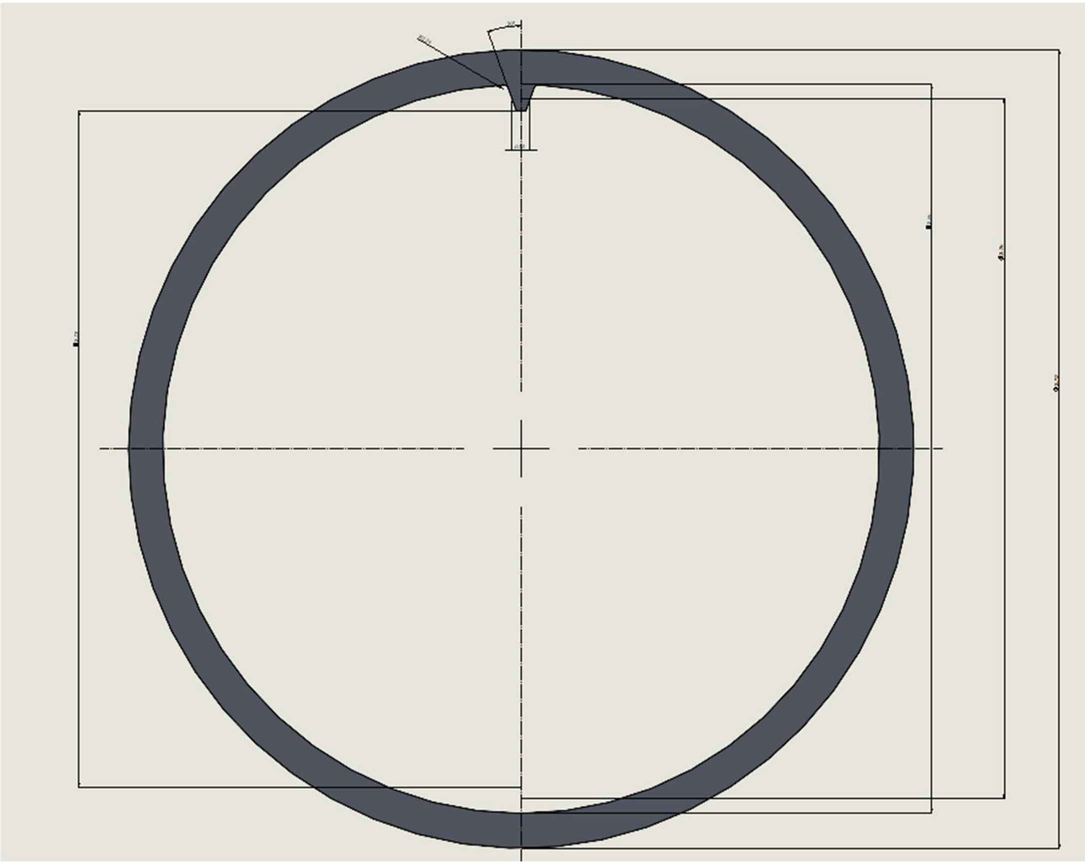


Figure 6. Internal spur gear 2 D drawing

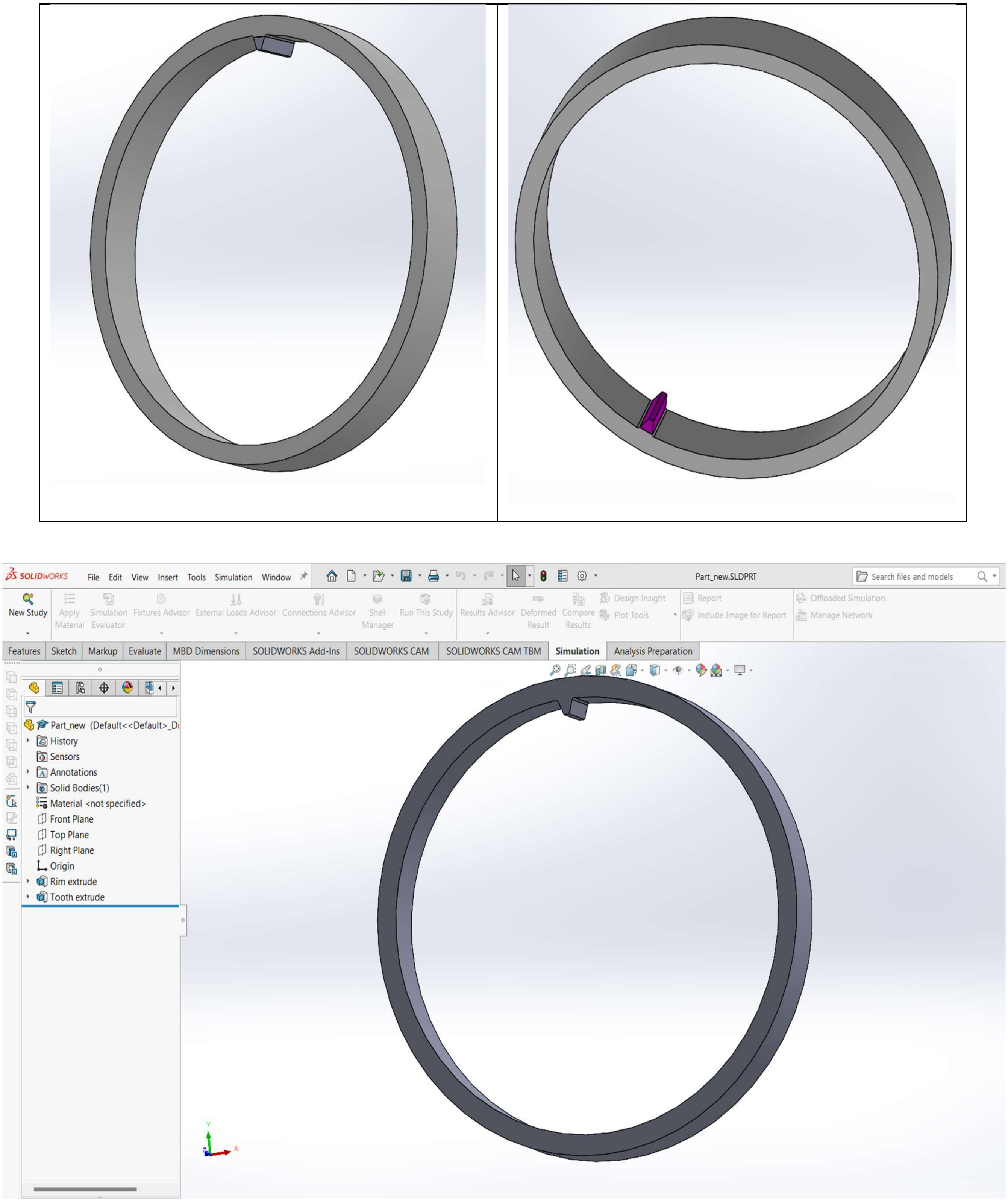


Figure 7. Internal spur gear 3 D drawing

5. Analysis of Gear Tooth

Finite Element analysis for any imported 3-D model is performed in three main steps

- Pre-Processing
- Processing or solution
- Post-Processing

5.1. Pre-Processing: -

The Pre-Processing mainly involves the modelling of the 3-D part. The following are the main steps in Pre-Processing

- Engineering Data
- Geometry
- Discretization

Engineering Data

In an analysis system the main resource for material properties is engineering data, they can either be experimental or user defined. In this analysis the linear elastic properties like Young's Modulus are defined slightly different for Rim and gear tooth.

Geometry:

The gear geometry generated from solid works is used for simulation Workbench in the same software.

Discretization:

Discretization is the method of converting continuous models to discrete parts. The goal is to select and locate finite element nodes and element types so that the associated analysis is sufficiently accurate. Element Aspect ratio must be near unity to obtain accurate results. For the current analysis average aspect ratio is obtained as **1** by setting the mesh relevance to fine and smoothing too medium.

Tetrahedral elements are used since stress is present all alone the thickness of the part. Patch independent algorithm is used since a finer mesh is required around edges and corner. For rest of the body a normal mesh is sufficient and in advanced meshing features the proximity and curvature feature needs to turn on since the curvature size function examines the curvature on the faces and edges and computes the element sizes so that the element size doesn't exceed the maximum size of curvature angle which are either defined by the user or taken automatically. The proximity size function allows defining the minimum number of element layers in the region that constitute gaps. The minimum size limit is defined as 0.1 mm

Modelling of gear tooth: -

We create a model using above gear parameters in accordance with our needs in Solid-Works. I also decided to create many configurations. I require a lot of time to model various configurations, thus I used the rebuilt option to set up 7 configurations with various specifications. I linked the excel document with equations and values to rebuild the model and give it autonomy. The time I saved was considerable. You can easily set up any configuration using an excel file, and it also saves time.

	Input Data							
	Module **m**	Number of tooth **Z**	Rim thickness				Tooth width	alpha (degree)
Model_01	1	60	3.5	4	4.5	5	10	20
Model_02	1	90	3.5	4	4.5	5	10	20
Model_03	1	120	3.5	4	4.5	5	10	20
Model_04	1	150	3.5	4	4.5	5	10	20
Model_05	1	180	3.5	4	4.5	5	10	20
Model_06	1	210	3.5	4	4.5	5	10	20
Model_07	1	240	3.5	4	4.5	5	10	20

Table 4. Input Data for sample model

After modelling, we must prepare the model for analysis using the simulation option in Solid-Works software.

We must first set up the model before running any simulations; to do this, we must take the procedures listed below.

- applying materials and properties
- Connections
- Boundary condition,
- Load or Force
- Generate a Mesh

Applying materials and properties: -

In this project, we choose appropriate materials for our FEM simulation to achieve the best outcomes. I decided to choose a different material for the gear rim with alloy steels and the gear tooth with Young modulus higher than alloy steels. If higher the Young modulus higher will be the resistance for the given force or load better the stiffness. Analysing what happens when you apply force to a gear tooth is our major goal. For this reason, we intended to provide a material that has a Young modulus for the gear teeth higher than the rim.

In the figure 8 the colour light green represents gear rim and colour pink represents for Gear tooth.

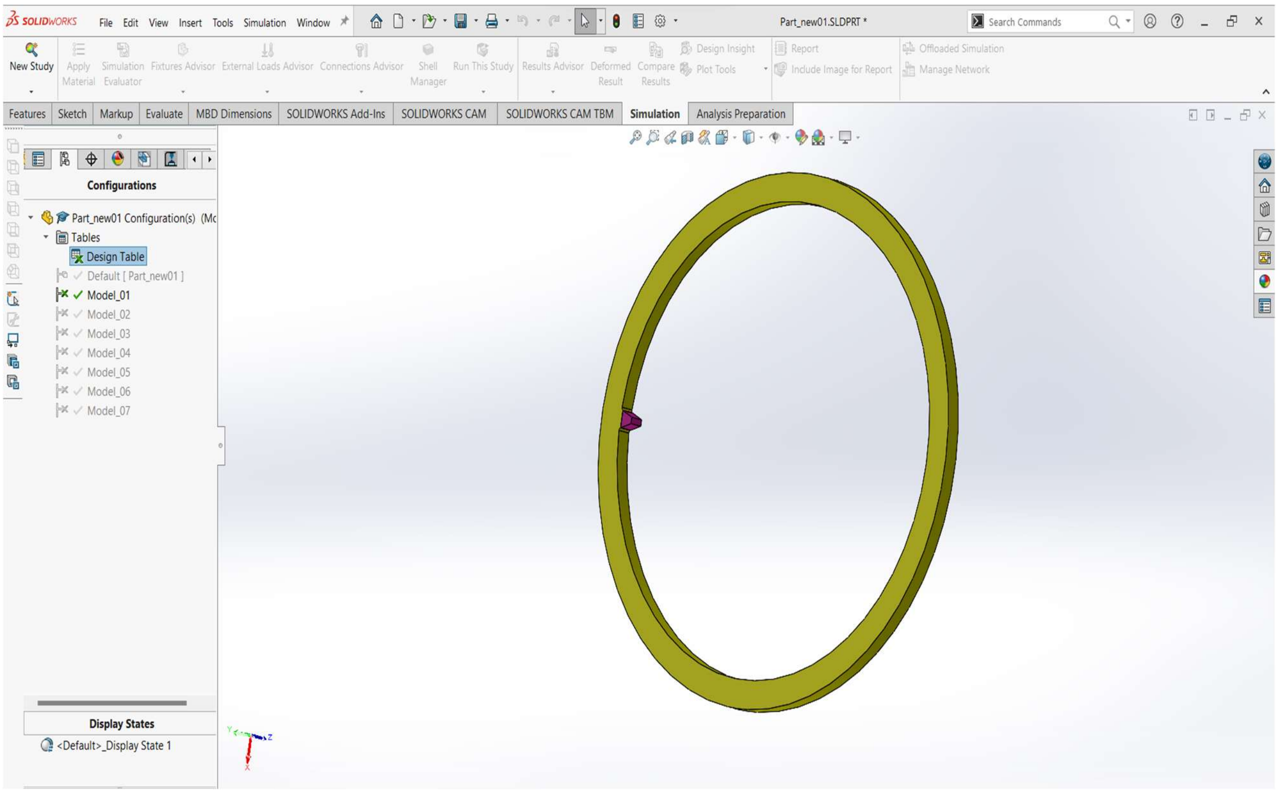


Figure 8. Applying materials and properties

Connections: -

Our next step after applying materials is connections. Connections are used to join assemblies or various elements. In FEM simulation, a variety of connections are employed to interconnect the components or parts. Our project is straightforward; there are only two distinct elements or parts. Gear tooth and Gear rim. We use the bond option under component interactions in the Solid-Works simulation software to unite these elements.

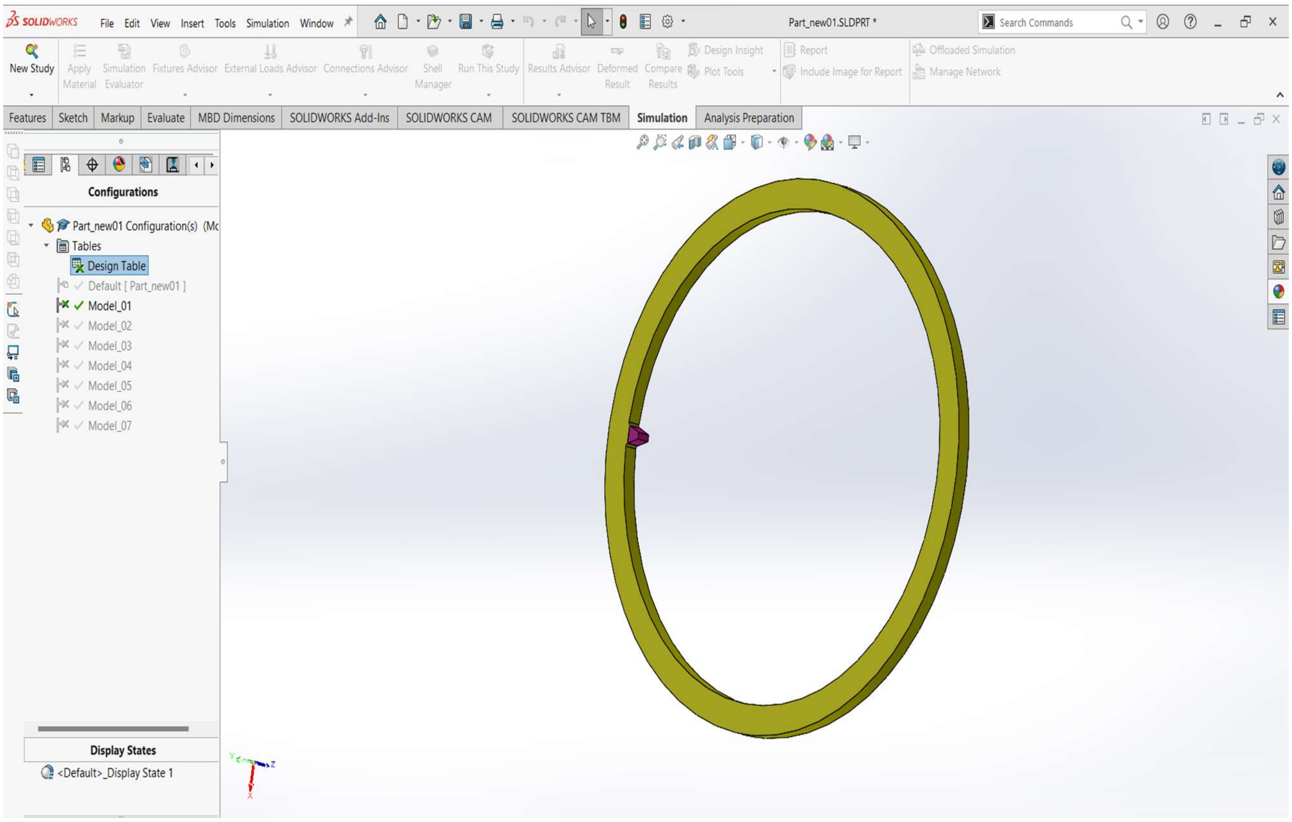


Figure 9. Connections

Boundary condition: -

The main types of loading available in FEA include force, pressure, and temperature. These can be applied to points, surfaces, edges, nodes, and elements or remotely offset from a feature. The way that the model is constrained can significantly affect the results and requires special consideration. Over or under constrained models can give stress that is so inaccurate that it is worthless to the engineer. In an ideal world we could have massive assemblies of components all connected to each other with contact elements, but this is beyond the budget and resource of most people. We can, however, use the computing hardware we have available to its full potential, and this means understanding how to apply realistic boundary conditions.

In our project, we are performing a static simulation to adjust the exterior geometry of the internal spur gear. We must apply fixed boundary condition to fix the geometry because we are performing static analysis.

For solids this restraint type sets all translational degrees of freedom to zero. For shells and beams, it sets the translational and the rotational degrees of freedom to zero. For truss joints, it sets the translational degrees of freedom to zero. When using this restraint type, no reference geometry is needed.

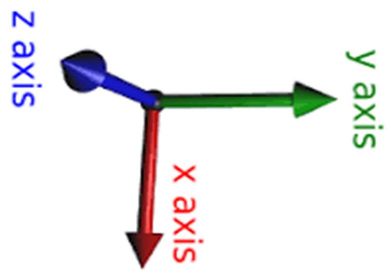


Figure 10. Constraints applied to the model

In the figure 11 the green constraints represent the fixed boundary condition.

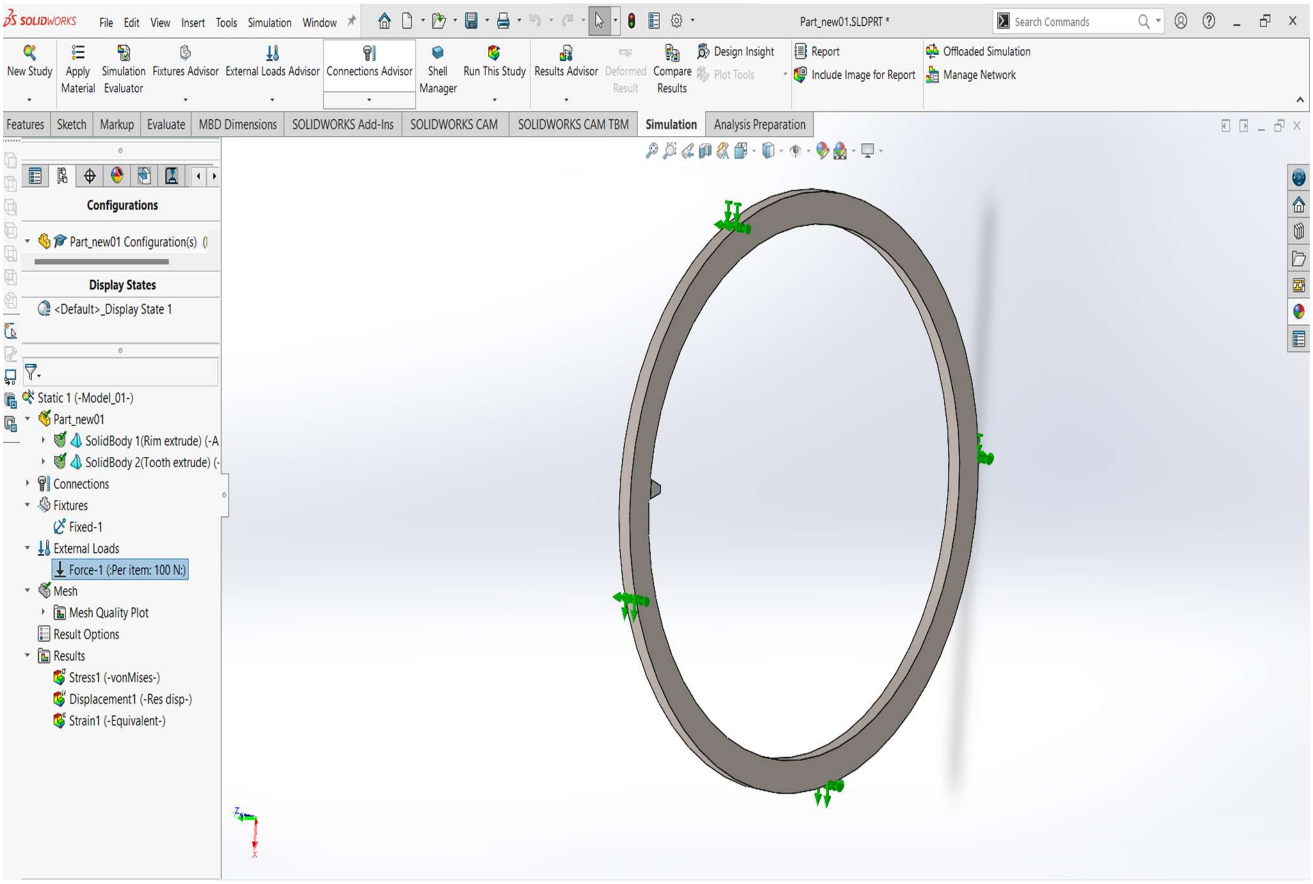


Figure 11. Boundary condition

Applying load or force: -

When loads are applied to a body, the body deforms, and the effect of loads is transmitted throughout the body. The external loads induce internal forces and reactions to render the body into a state of equilibrium.

Linear Static analysis calculates displacements, strains, stresses, and reaction forces under the effect of applied loads.

Linear static analysis makes the following assumptions:

Static Assumption. All loads are applied slowly and gradually until they reach their full magnitudes. After reaching their full magnitudes, loads remain constant (time-invariant). This assumption allows us to neglect inertial and damping forces due to negligibly small accelerations and velocities. Time-variant loads that induce considerable inertial and/or damping forces may warrant dynamic analysis. Dynamic loads change with time and in many cases induce considerable inertial and damping forces that cannot be neglected.

Linearity Assumption. The relationship between loads and induced responses is linear. For example, if you double the loads, the response of the model (displacements, strains, and stresses), will also double. You can make the linearity assumption if:

- all materials in the model comply with Hooke's law, that is stress is directly proportional to strain.
- the induced displacements are small enough to ignore the change in stiffness caused by loading.
- boundary conditions do not vary during the application of loads. Loads must be constant in magnitude, direction, and distribution. They should not change while the model is deforming.

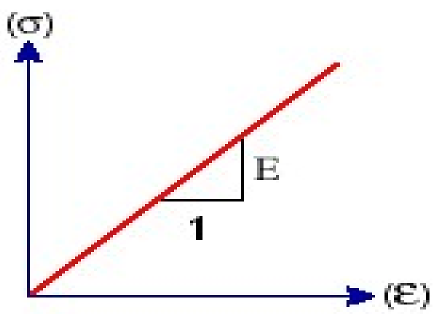


Figure 12. Stress- Strain curve

In the figure13 the load or force acting in the X-direction.

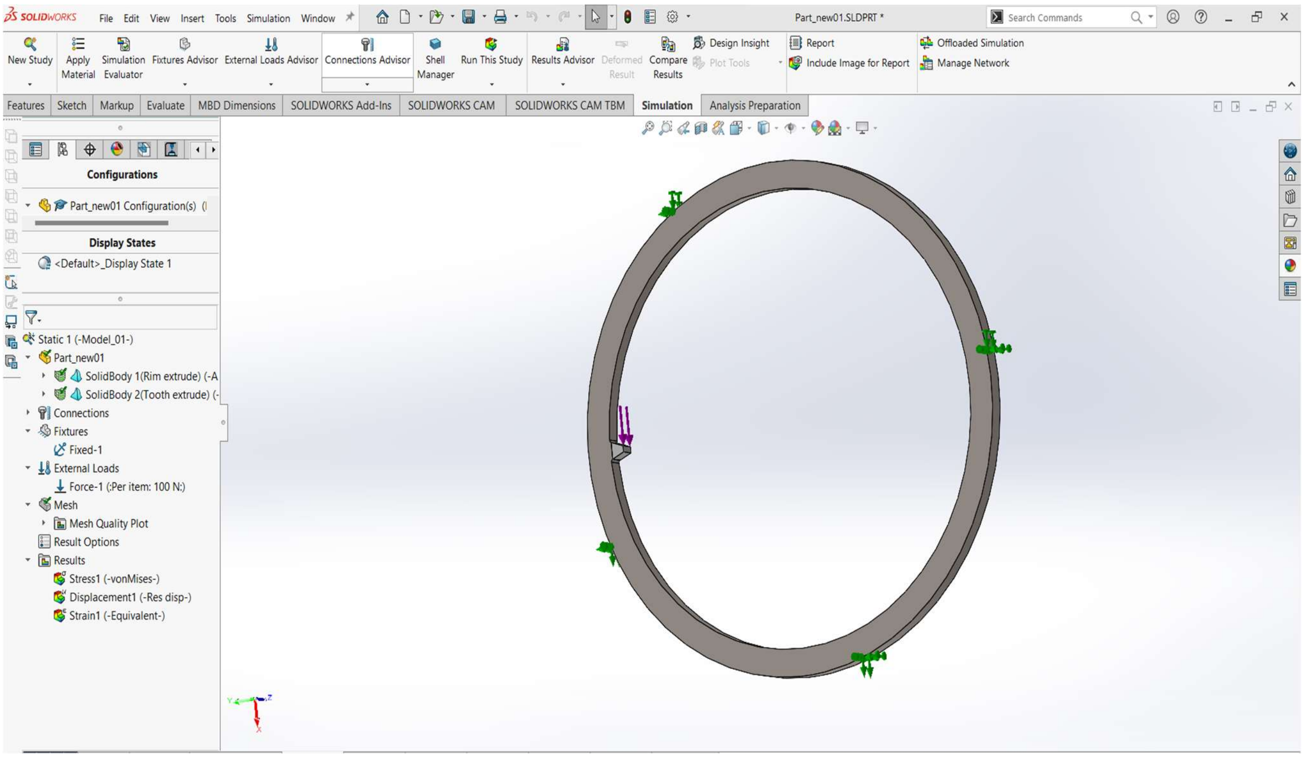


Figure 13. Loading condition

Generate a Mesh: -

When you mesh a model, the software generates a mixture of **solid, shell**, spring, and contact elements based on the created geometry. The program subdivides the model into small pieces of simple shapes (elements) connected at common points (nodes). Meshing is a very crucial step in design analysis. The automatic masher in the software generates a mesh based on a global element size, tolerance, and local mesh control specifications. The program automatically creates the following meshes:

- **Solid Mesh**: The program creates a solid mesh with **tetrahedral 3D solid elements** for all solid components in the Parts folder. Tetrahedral elements are suitable for bulky objects.
- **Shell Mesh**: The program automatically creates a shell mesh for **sheet-metals** with uniform thicknesses (except drop test study) and surface geometries. For

sheet metals the mesh is automatically created at the mid-surface. The program extracts the shell thickness from the thickness of the sheet metal.

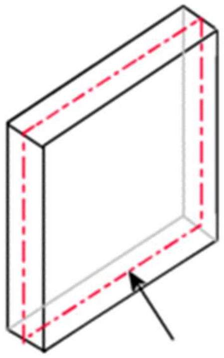


Figure 14. The mid-surface of a sheet metal is highlighted.

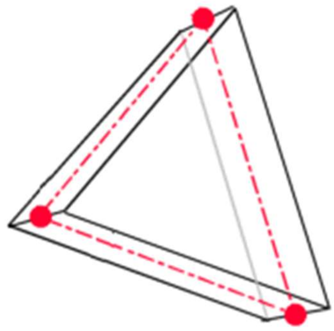


Figure 15. A shell element created at the mid-surface with the nodes shown

For surfaces, the program locates the mesh on the surface (mid-plane of shell). You assign the thickness for the shell in the **Shell Definition** Property Manager.

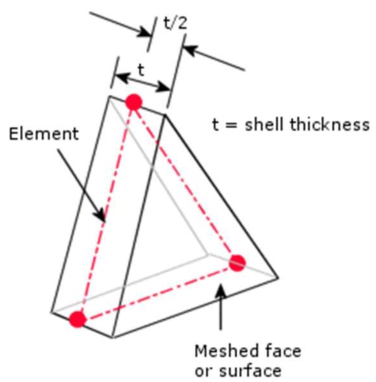
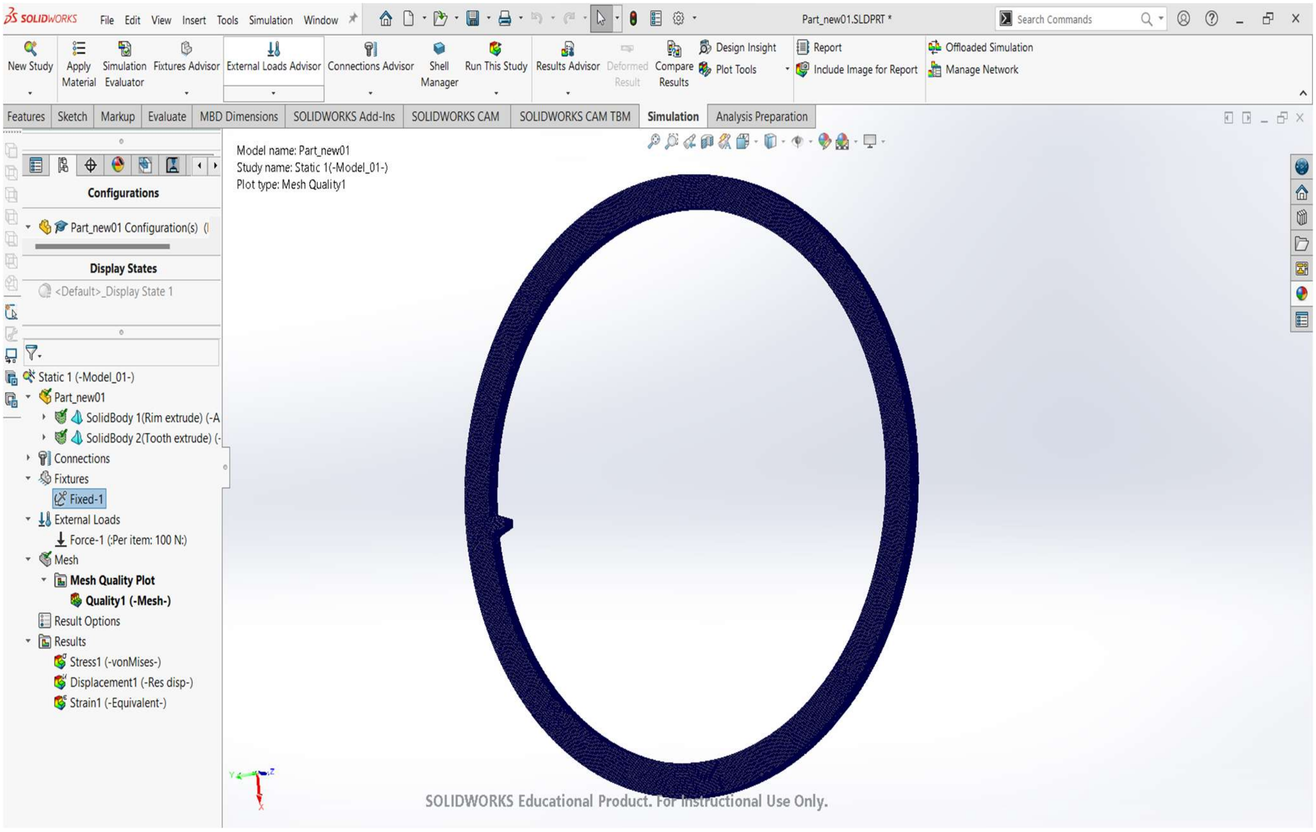


Figure 16. Solid tetra mesh

- **Beam Mesh**: The program automatically uses beam mesh and identifies joints for touching or interfering structural members and non-touching structural members within a certain distance (tolerance). A beam element is a line element defined by two end points and a cross-section. Beam elements can resist axial, bending, shear, and torsional loads. Trusses resist axial loads only. When used with weldments, the software defines cross-sectional properties and detects joints.

- **Mixed mesh**: The program automatically uses a mixed mesh when different geometries are present in the model.



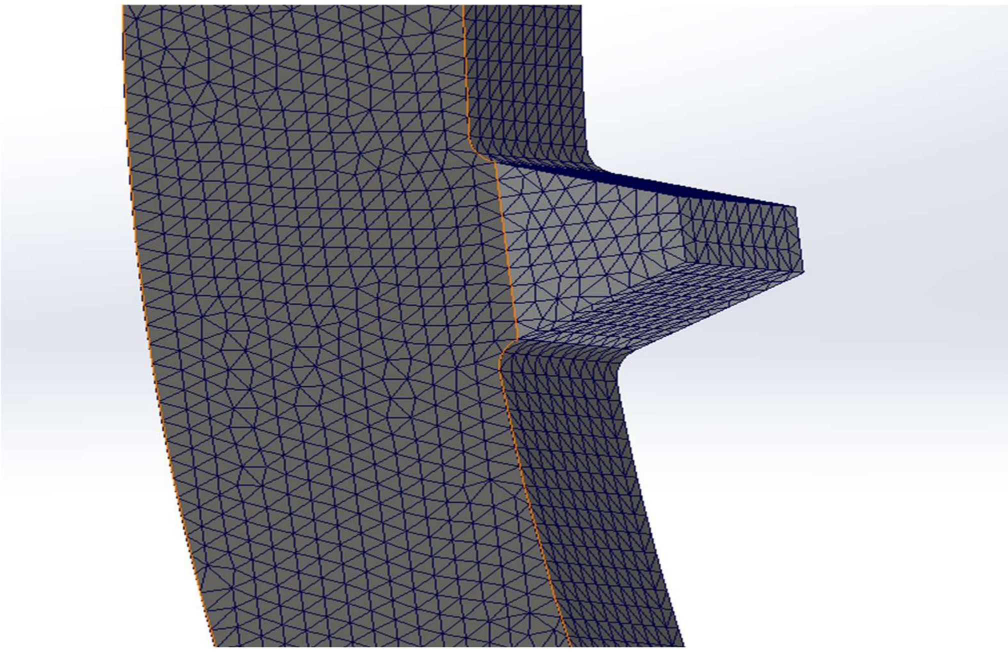


Figure 17. Solid tetrahedral mesh

Parameters		
Materials	Gear Rim Alloy steel	Gear Tooth New Material
Elastic modulus (N/mm^2)	210000	2000000
Load (N) Mesh size (mm)	100 0.1 to 1 mm	

Table 5. Input data for processing setup

5.2. Processing or solution: -

During processing user must work hard while solution step is to assign the computer to do the job. User must just click on solve icon and enjoy cup of tea! Internally the software carries out matrix formations, inversions, multiplications Solution for unknown example displacement and then find strain and stress for static analysis.

Today we are using FEA just because of availability of computers. FEM has been known to mathematicians and engineers right from late 50s but since solving so many equations manually was not possible, in true sense FEA got recognition only after emergence of high-capacity computers.

Static structural analysis determines the stress, strain and displacements in a body that are caused due to loads. In this analysis steady loading is used. Solid-Works supports the following types of loading for the static analysis.

1. External loads
2. Steady state inertial forces
3. Imposed displacement’s

5.3. Post Processing: -

Post processing is viewing results, verifications, conclusions and thinking about what steps could be taken to improve the design. Stiffness of the gear can be obtained from the Normal load or forces and displacement menu by choosing the normal stress and displacement in X-direction. The following below figures 18,19&20 displays simulation results of the stress, Strain and Displacement contour of the gear tooth.

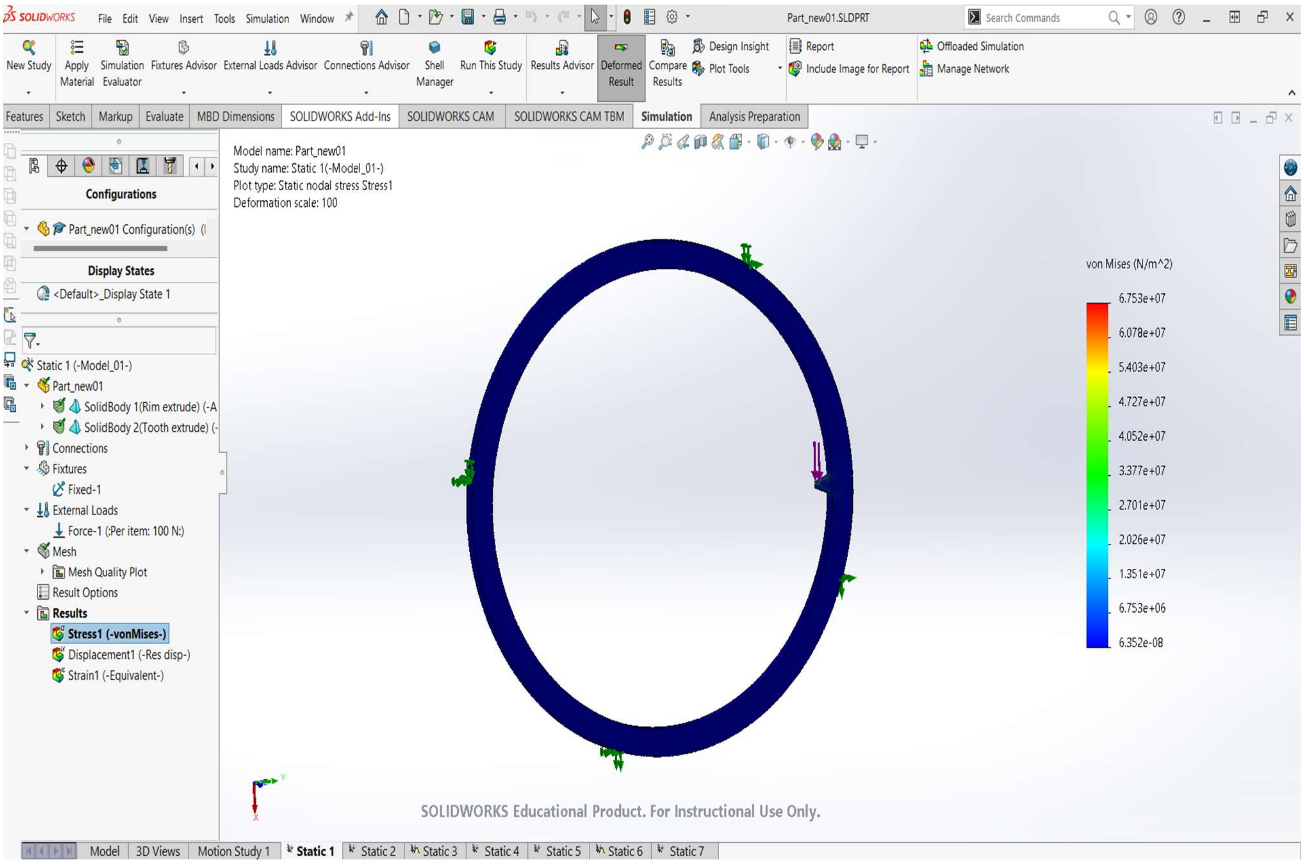


Figure 18. Simulation of stress results

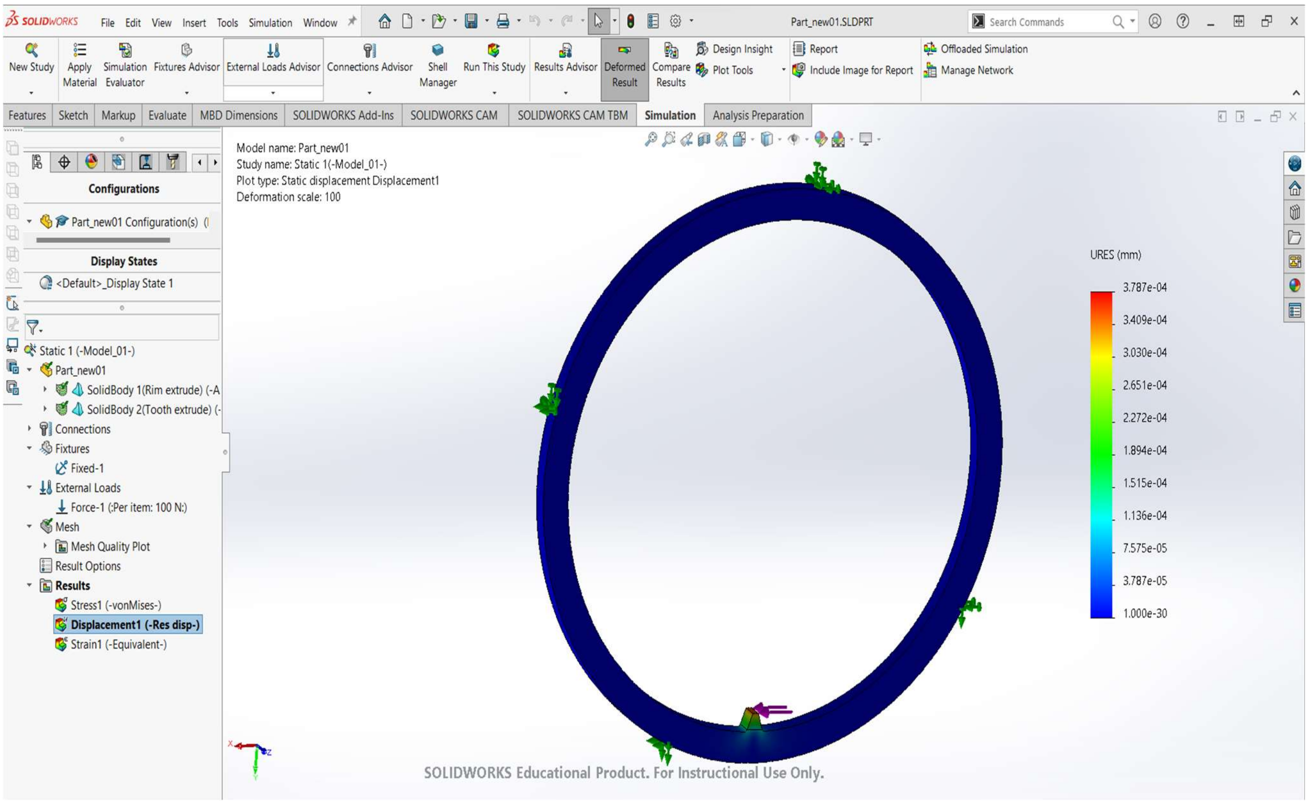


Figure 19. Simulation of Displacement results

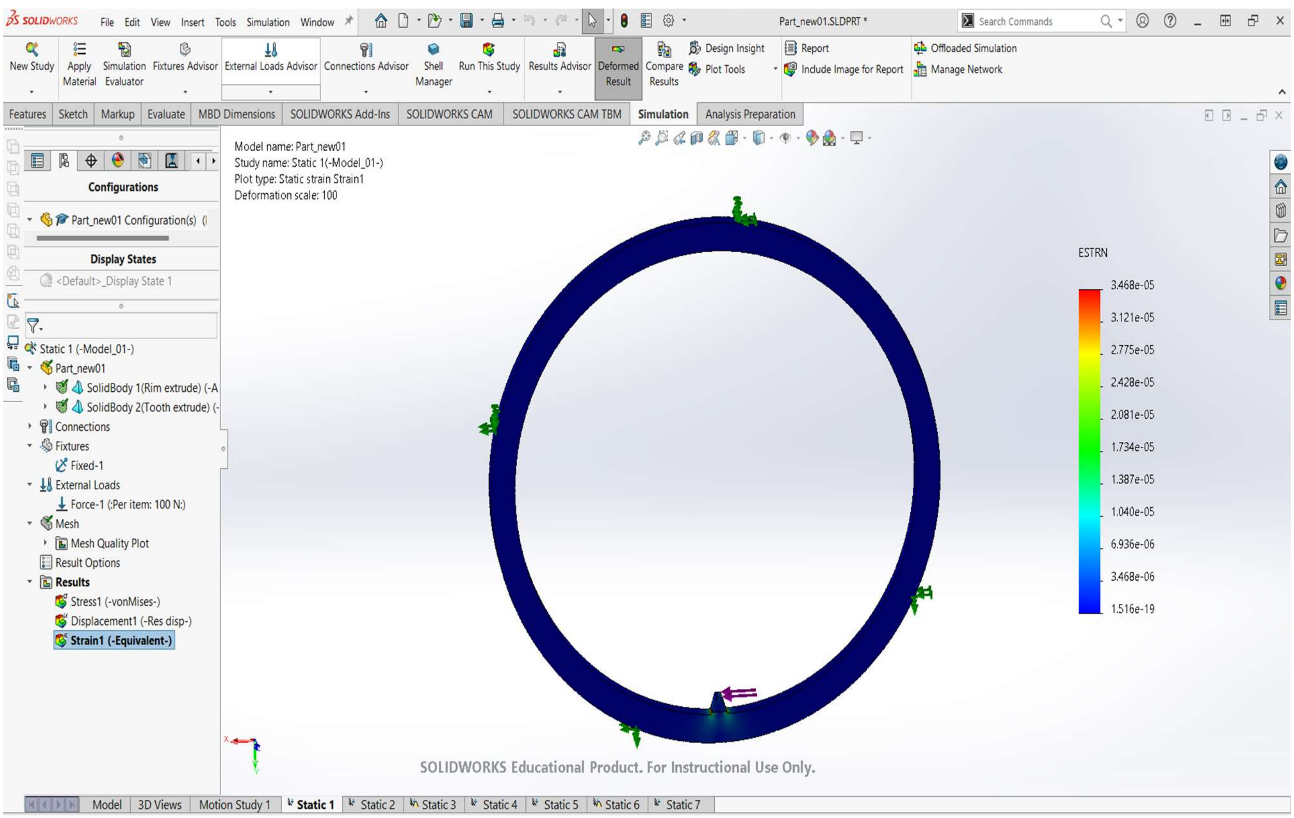


Figure 20. Simulation of strain results

By altering some internal gear characteristics, we have chosen to perform **three iterations.** We decided to alter the module, rim thickness, and number of tooth and other parameters remain constant for all the iterations.

Now we start with the first iteration, and we are set upping 7 configuration for static simulation in Solid-Works software.

There are some inputs for first iteration in the below table 6.

	Input Data for 1st iteration				
	Module **m**	Number of tooth **Z**	Rim thickness	Tooth width	alpha (degree)
Model_01	1	60	3.5 4 4.5 5	10	20
Model_02	1	90	3.5 4 4.5 5	10	20
Model_03	1	120	3.5 4 4.5 5	10	20
Model_04	1	150	3.5 4 4.5 5	10	20
Model_05	1	180	3.5 4 4.5 5	10	20
Model_06	1	210	3.5 4 4.5 5	10	20
Model_07	1	240	3.5 4 4.5 5	10	20

Table 6. Input Data for 1st iteration

The number of teeth and rim thickness are listed in the table 6. We shall record the displacement for applied force during postprocessing evaluation. The stiffness in the last column of the table 7 is determined.

	Number of teeth(Z)	Rim thickness				Displacement(mm)				Load (N)	Stiffness(N/mm)			
						3.5	4	4.5	5		3.5	4	4.5	5
Model_01	60	3.5	4	4.5	5	3.10E-04	3.19E-04	3.28E-04	3.22E-04	100	3.23E+05	3.13E+05	3.05E+05	3.11E+05
Model_02	90	3.5	4	4.5	5	3.14E-04	3.24E-04	3.31E-04	3.36E-04	100	3.19E+05	3.09E+05	3.02E+05	2.98E+05
Model_03	120	3.5	4	4.5	5	3.17E-04	3.25E-04	3.32E-04	3.39E-04	100	3.16E+05	3.08E+05	3.01E+05	2.95E+05
Model_04	150	3.5	4	4.5	5	3.14E-04	3.24E-04	3.31E-04	3.36E-04	100	3.18E+05	3.09E+05	3.02E+05	2.98E+05
Model_05	180	3.5	4	4.5	5	3.13E-04	3.23E-04	3.33E-04	3.36E-04	100	3.20E+05	3.09E+05	3.00E+05	2.97E+05
Model_06	210	3.5	4	4.5	5	3.18E-04	3.26E-04	3.35E-04	3.40E-04	100	3.14E+05	3.07E+05	2.98E+05	2.94E+05
Model_07	240	3.5	4	4.5	5	3.16E-04	3.25E-04	3.32E-04	3.38E-04	100	3.16E+05	3.08E+05	3.01E+05	2.96E+05

Table 7. Output data for 1st iteration

The inputs for the second iteration are listed in the table 8. For this iteration, we decide to increase the module to 2 mm and repeat the processing and postprocessing steps.

	Input Data for 2nd iteration				
	Module **m**	Number of tooth **Z**	Rim thickness	Tooth width	alpha (degree)
Model_01	2	60	3.5 4 4.5 5	10	20
Model_02	2	90	3.5 4 4.5 5	10	20
Model_03	2	120	3.5 4 4.5 5	10	20
Model_04	2	150	3.5 4 4.5 5	10	20
Model_05	2	180	3.5 4 4.5 5	10	20
Model_06	2	210	3.5 4 4.5 5	10	20
Model_07	2	240	3.5 4 4.5 5	10	20

Table 8. Input Data for 2nd iteration

We repeat the same procedure for recording the displacement for applied force during postprocessing evaluation. The stiffness in the last column of the table 9 is determined.

	Number of teeth(Z)	Rim thickness				Displacement(mm)				Load (N)	Stiffness(N/mm)			
						3.5	4	4.5	5		3.5	4	4.5	5
Model_01	60	3.5	4	4.5	5	3.31E-04	3.40E-04	3.47E-04	3.52E-04	100	3.02E+05	2.94E+05	2.88E+05	2.84E+05
Model_02	90	3.5	4	4.5	5	3.34E-04	3.42E-04	3.50E-04	3.55E-04	100	3.00E+05	2.92E+05	2.86E+05	2.81E+05
Model_03	120	3.5	4	4.5	5	3.35E-04	3.44E-04	3.51E-04	3.57E-04	100	2.99E+05	2.91E+05	2.85E+05	2.80E+05
Model_04	150	3.5	4	4.5	5	3.35E-04	3.45E-04	3.51E-04	3.58E-04	100	2.98E+05	2.90E+05	2.85E+05	2.80E+05
Model_05	180	3.5	4	4.5	5	3.36E-04	3.45E-04	3.52E-04	3.58E-04	100	2.98E+05	2.90E+05	2.84E+05	2.79E+05
Model_06	210	3.5	4	4.5	5	3.36E-04	3.45E-04	3.52E-04	3.58E-04	100	2.98E+05	2.90E+05	2.84E+05	2.79E+05
Model_07	240	3.5	4	4.5	5	3.37E-04	3.46E-04	3.53E-04	3.59E-04	100	2.97E+05	2.89E+05	2.84E+05	2.78E+05

Table 9. output data for 2nd iteration

Here you can find the Third iteration inputs in the below table 10. We choose to increase module to 5 mm and other parameters remain constant.

	Input Data for 3rd iteration							
	Module **m**	Number of tooth **Z**	Rim thickness				Tooth width	alpha (degree)
Model_01	5	60	3.5	4	4.5	5	10	20
Model_02	5	90	3.5	4	4.5	5	10	20
Model_03	5	120	3.5	4	4.5	5	10	20
Model_04	5	150	3.5	4	4.5	5	10	20
Model_05	5	180	3.5	4	4.5	5	10	20
Model_06	5	210	3.5	4	4.5	5	10	20
Model_07	5	240	3.5	4	4.5	5	10	20

Table 10. Input Data for 3rd iteration

We repeat the same procedure for recording the displacement for applied force during postprocessing evaluation. The stiffness in the last column of the table 11 is determined.

	Number of teeth(Z)	Rim thickness				Displacement(mm)				Load (N)	Stiffness(N/mm)			
						3.5	4	4.5	5		3.5	4	4.5	5
Model_01	60	3.5	4	4.5	5	3.42E-04	3.51E-04	3.58E-04	3.63E-04	100	2.92E+05	2.85E+05	2.79E+05	2.75E+05
Model_02	90	3.5	4	4.5	5	3.45E-04	3.53E-04	3.60E-04	3.65E-04	100	2.90E+05	2.83E+05	2.77E+05	2.74E+05
Model_03	120	3.5	4	4.5	5	3.46E-04	3.55E-04	3.62E-04	3.67E-04	100	2.89E+05	2.82E+05	2.76E+05	2.72E+05
Model_04	150	3.5	4	4.5	5	3.47E-04	3.55E-04	3.63E-04	3.68E-04	100	2.89E+05	2.81E+05	2.76E+05	2.72E+05
Model_05	180	3.5	4	4.5	5	3.47E-04	3.56E-04	3.63E-04	3.68E-04	100	2.89E+05	2.81E+05	2.75E+05	2.72E+05
Model_06	210	3.5	4	4.5	5	3.47E-04	3.56E-04	3.64E-04	3.69E-04	100	2.88E+05	2.81E+05	2.75E+05	2.71E+05
Model_07	240	3.5	4	4.5	5	3.48E-04	3.57E-04	3.64E-04	3.69E-04	100	2.88E+05	2.81E+05	2.75E+05	2.71E+05

Table 11. Output Data for 3rd iteration

Now using all the above iterations, we can correlate these results using MATLAB software. There are many software’s that are used to interpolate the results to obtain or to formulate any formula. We decided to use curve fitting toolbox. To use curve fitting tool, we need the values in the form of matrix to create a surface.

5.4. Curve Fitting Toolbox: -

The Curve Fitting Toolbox is a collection of graphical user interfaces (GUIs) and M-file functions built on the MATLAB® technical computing environment. The toolbox provides you with these main features:

• Data pre-processing such as sectioning and smoothing

• Parametric and nonparametric data fitting: -

You can perform a parametric fit using a toolbox library equation or using a custom equation. Library equations include polynomials, exponentials, rational, sums of Gaussians, and so on. Custom equations are equations that you define to suit your specific curve fitting needs.

You can perform a nonparametric fit using a smoothing spline or various interpolants.

• Standard linear least squares, nonlinear least squares, weighted least squares, constrained least squares, and robust fitting procedures.

• Fit statistics to assist you in determining the goodness of fit.

• Analysis capabilities such as extrapolation, differentiation, and integration.

• A graphical environment that allows you to: -

Explore and analyse data sets and fits visually and numerically

Save your work in various formats including M-files, binary files, and workspace variables

The Curve Fitting Toolbox consists of two different environments:

• The Curve Fitting Tool, which is a graphical user interface (GUI) environment.

• The MATLAB command line environment You can explore the Curve Fitting Tool by typing cf tool.

It provides an app and functions for **fitting curves and surfaces to data**. The toolbox lets you perform exploratory data analysis, preprocess, and post-process data, compare candidate models, and remove outliers.

Using this curve fitting toolbox, we create a surface to represent all the data’s which is obtained from the simulation.

We require a value in the form of a square matrix to generate a surface. The four displacements for each configuration are shown in the table 6,8&10 and were determined through simulation by changing the gear's rim thickness ratio from 3.5 to 5.
The curve fitting toolbox includes many interpolation methods, including polynomial, exponential, Fourier, gaussian, and custom equations, among others.

We decided to utilize a custom equation because we must develop an equation for stiffness for static analysis. By interpolating three parameters Rim thickness, Number of teeth, and stiffness from the iterations, we create formula to test the outcomes of our simulations. Now, by interpolating in three axes, the x-axis denotes the number of teeth, the y-axis denotes the thickness of the rim, and the z-axis denotes the stiffness outcomes.

Run the model using a custom equation after interpolating. The coefficients a, b,c and d values can be seen in the right column of figure 21. The R-square value should be one or slightly less than one, according to our final inspection. It's crucial for us to have an R-square of between 0.8 and 1.

The figures 21,22&23 represents the interpolation of 1^{st}, 2^{nd} and 3^{rd} iterations.

From the 1^{st} iteration we got an R-square value equal to 0.87 and is shown in the figure 21.

From the 2^{nd} iteration we got an R-square value equal to 0.98 and is shown in the figure 22.

From the 3^{rd} iteration we got an R-square value equal to 0.97 and is shown in the figure 23.

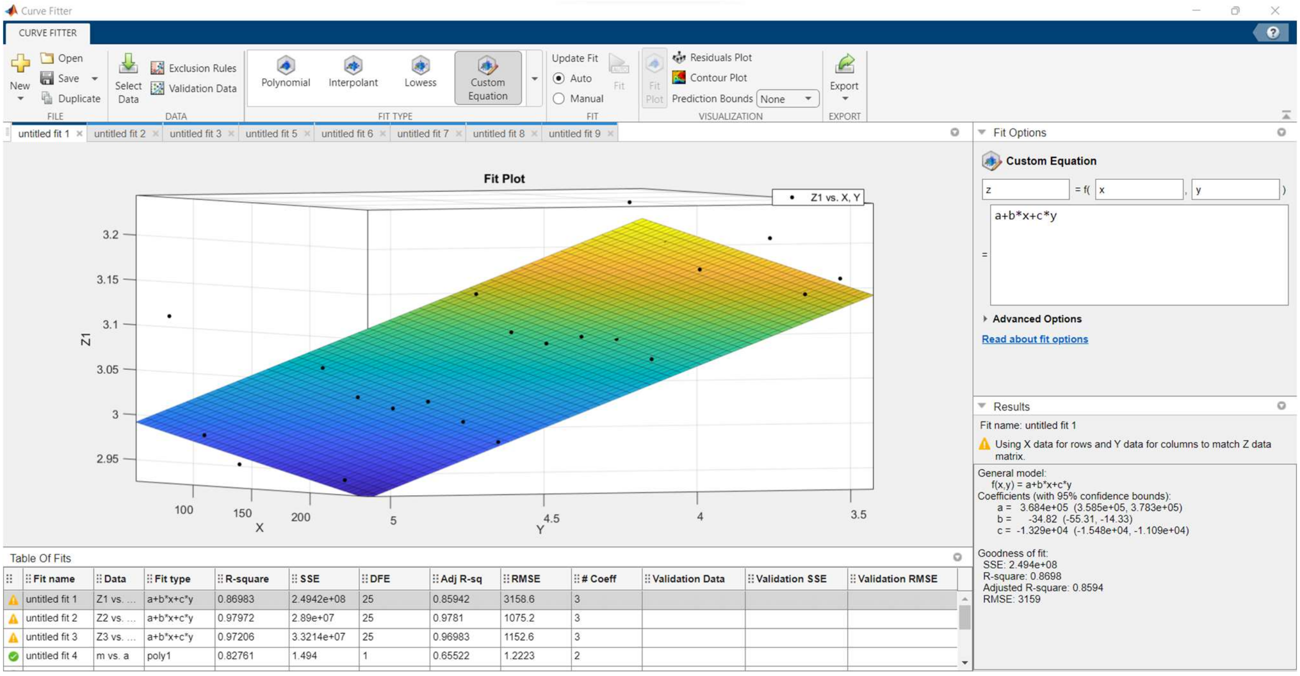


Figure 21. 1st iteration surface

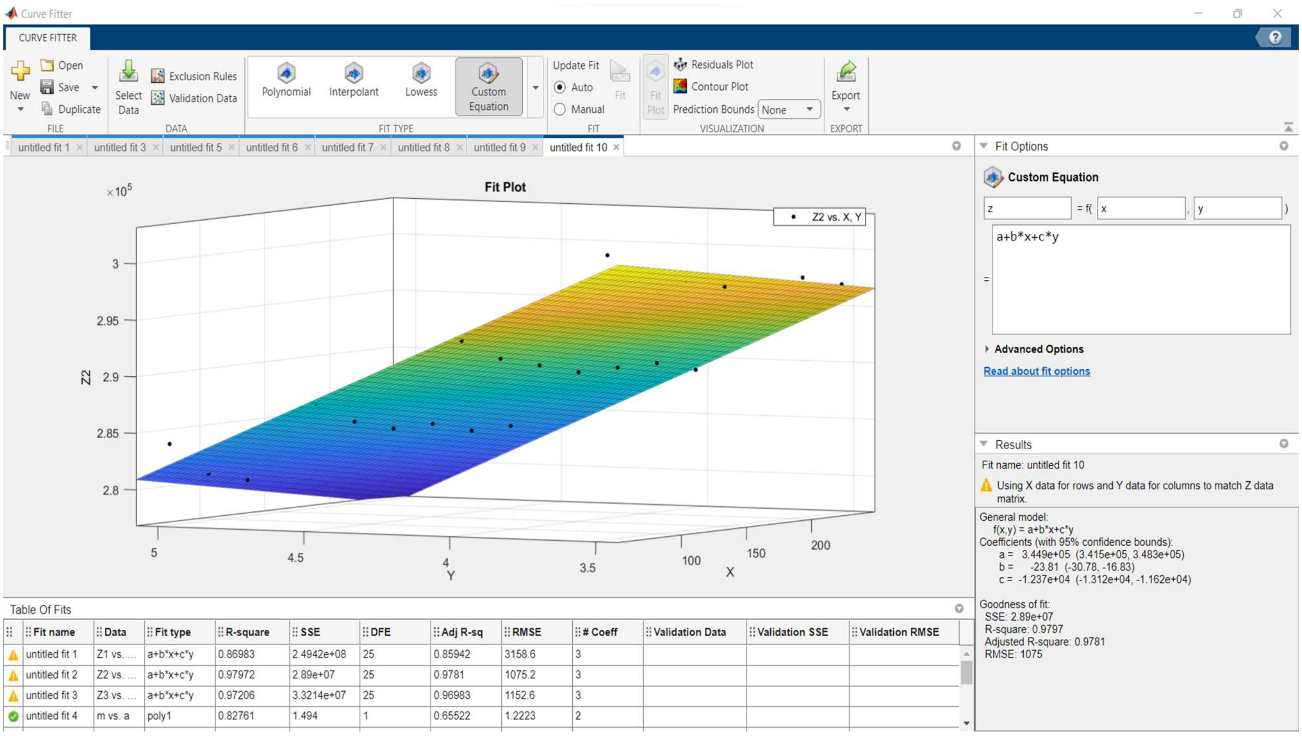


Figure 22. 2nd iteration surface

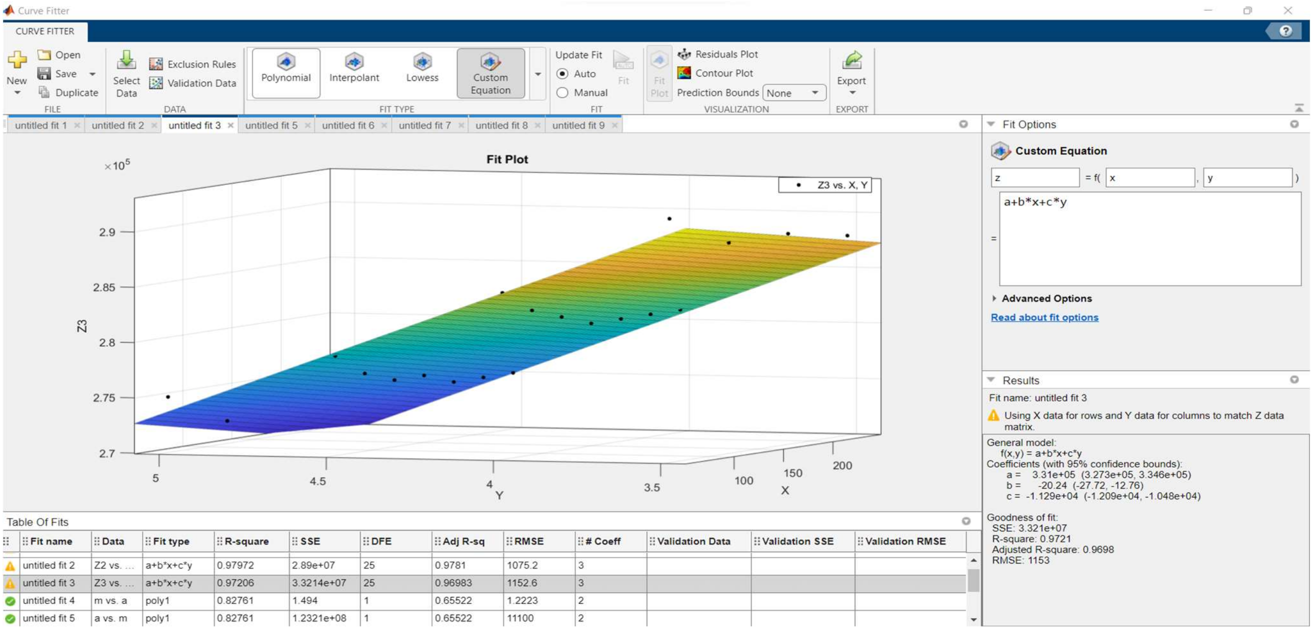


Figure 23. 3rd iteration surface

The next step is to create a graph using the **module** "**m**" values and coefficient "**a,b,c**" values once the correlation has been completed.

The relationship between coefficient "**a** "and **module** "**m**" is depicted in figure 24.

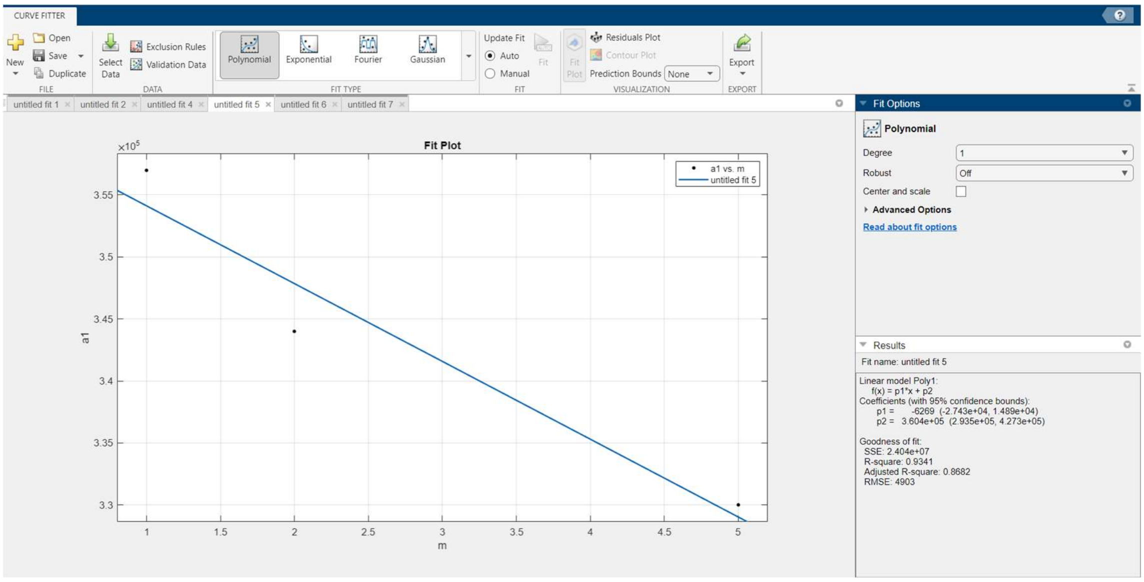


Figure 24. Interpolation between a and m

We can note the values of p_1 *and* p_2 from figure 24, which you can see on the right side.

$$a = f(x, y) = p_1 * x + p_2 = (-6269 * m) + 3.604 * 10^5$$

Where

$$p_1 = -6269$$

$$p_2 = 3.604 * 10^5$$

$$a = (-6269 * m) + 3.604 * 10^5$$

The module m points in the interpolation between **a** and **m** are not in the line we wish to attempt; therefore, I simply interpolated the results of the any two iterations.

Now, in Figure 25, the results with coefficient **b** and module **m** are shown with two and five modules by removing one iteration with module one.

The relation between module “**m**” and coefficient “**b**” value are represented in the figure 25.

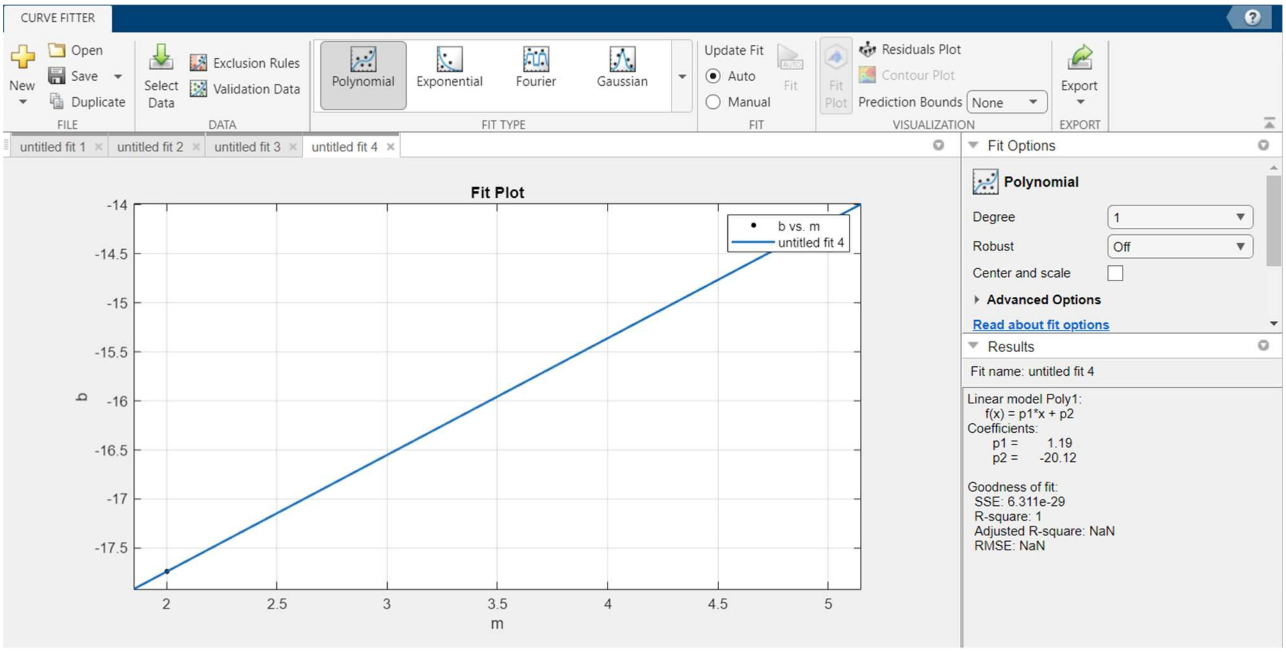


Figure 25. Interpolation between b and m

We can note the values of p_1 *and* p_2 from figure 25, which you can see on the right side.

$$b = f(x,y) = p_1 * x + p_2 = (1.19 * m) - 20.12$$

Where

$$p_1 = 1.19$$

$$p_2 = -20.12$$

$$b = (1.19 * m) - 20.12$$

The module m points in the interpolation between b and m are in the line we wish to attempt; therefore, I simply interpolated the results of the second and third iterations in the figure 25.

The relation between module “**m**” and coefficient “**c**” value are represented in the figure 26.

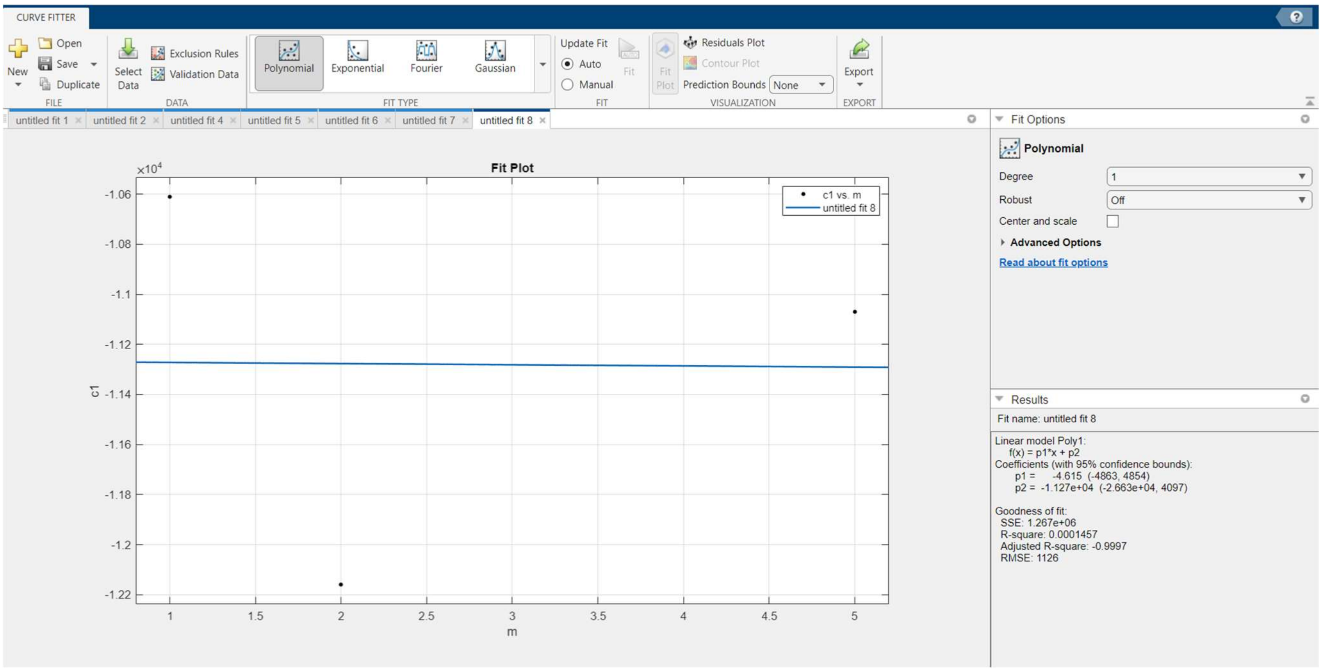


Figure 26. Interpolation between c and m

We can note the values of p_1 *and* p_2 from figure 26, which you can see on the right side.

$$c = f(x, y) = p_1 * x + p_2$$

Where

$$p_1 = -4.615$$

$$p_2 = -1.217 * 10^4$$

$$c = -1.127 * 10^4$$

Because p 1 has such a little value, I don't include it in the final formula. The constant value is our choice. The interpolation in figure 26 is constant, which suggests its line. I only consider the value of p2 in the final formula.

The relation between module "**m**" and coefficient "**d**" value are represented in the figure 27.

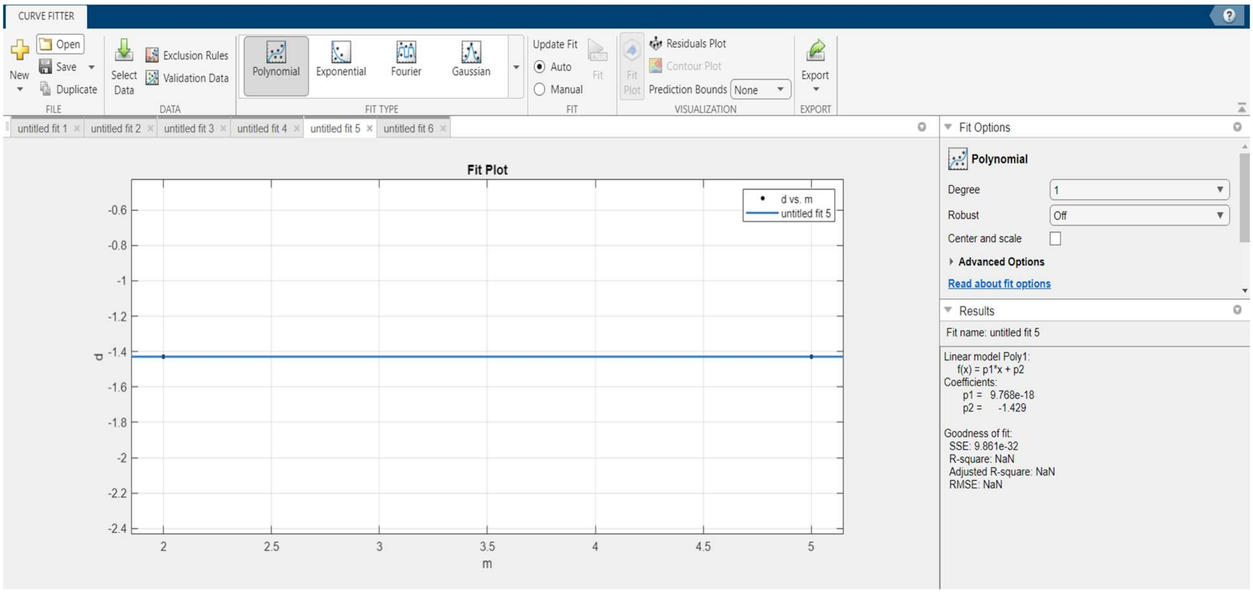


Figure 27. Interpolation between d and m

We can note the values of p_1 *and* p_2 from figure 27, which you can see on the right side.

$$d = f(x, y) = p_1 * x + p_2$$

Where

$$p_1 = 9.768 * 10^{-18}$$

$$p_2 = -1.429$$

$$d = -1.429$$

The same rationale that was given in figure 26 is also the reason here. I disregard the value p1.I only consider the value of p2 in the final formula also in figure 27.

From the above graphs we understood that if I increase the module, the values of coefficients a,b,c and d are decreasing. But the R-square value more or else remains constant.

6. Validation of results: -

Now our final step is to validate our results. By Considering the results from Simulation and MATLAB curve fitting toolbox. The average values a, b, c&d are noted from the above figures 24,25,26&27. The final equation is obtained is shown below.

$$k_m = a_m + b_m * x + c_m * y + d_m * x * y \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots..equation(4)$$

Where

a,b,c&d are coefficients and m is a integer

x is number of tooth(z)

y is the gear rim thickness(mm)

The equation is shown below for various module m

$$k_1(m = 1) = a_1 + b_1 * x + c_1 * y + d_1 * x * y \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots..equation(5)$$

$$k_2(m = 2) = a_2 + b_2 * x + c_2 * y + d_2 * x * y \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots..equation(6)$$

.

.

$$k_3(m = 5) = a_5 + b_5 * x + c_5 * y + d_5 * x * y \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots..equation(7)$$

Where

$$a_{1,2,5} = -\mathrm{d} * \mathrm{m} + \mathrm{e} = p_1 * x + p_2$$

$$b_{1,2,5} = \mathrm{f} * \mathrm{m} + \mathrm{g} = p_1 * x + p_2$$

$$c_{1,2,5} = \mathrm{h} * \mathrm{m} + \mathrm{i} = p_1 * x + p_2$$

$$d_{1,2,5} = \mathrm{l} * \mathrm{m} + \mathrm{n} = p_1 * x + p_2$$

$$k = (-\mathrm{d} * \mathrm{m} + \mathrm{e}) + (\mathrm{f} * \mathrm{m} + \mathrm{g}) * x + (\mathrm{h} * \mathrm{m} + \mathrm{i}) * y + (\mathrm{l} * \mathrm{m} + \mathrm{n}) * \mathrm{x} * \mathrm{y} \dots.equation(8)$$

Now by using the results from the graph shown above, the final stiffness equation can be created

$$k_m = a_m + b_m * x + c_m * y + d_m * x * y \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots..equation(9)$$

$$k_m = \left((-6269 * m) + 3.604 * 10^5\right) + \left((1.19 * m) - 20.12\right) * x + (-1.127 * 10^4) * y + (-1.429) * x * y \text{.........equation (10)}$$

Now, using a separate simulation result, we may cross-check our final analytical formula.

We choose the module m=4&3 and rim thickness 3.8&4.2, substitute in final analytical stiffness formula (10)

$$k(m = 4) = 3.35 * 10^5 * 4 - 11.28 * 10^3 * 3.8 - 2.69 * 4 * 3.8$$

$$k = 2.94 * 10^5 \quad \text{N/mm}$$

$$k(m = 3) = 3.45 * 10^5 - 1.2 * 3 - 11.28 * 10^3 * 4.2 - 2.69 * 3 * 4.2$$

$$k = 2.97 * 10^5 \quad \text{N/mm}$$

These are the simulation results for different module 3& 4.

	Number of teeth(Z)	Rim thickness		Displacement(mm)		Load (N)	Stiffness(N/mm)	
				3.8	4.2		3.8	4.2
Model_01	60	3.5	4.2	3.46E-04	3.48E-04	100	2.89E+05	2.87E+05
Model_02	90	3.5	4.2	3.49E-04	3.51E-04	100	2.87E+05	2.85E+05
Model_03	120	3.5	4.2	3.50E-04	3.53E-04	100	2.86E+05	2.84E+05
Model_04	150	3.5	4.2	3.50E-04	3.53E-04	100	2.85E+05	2.83E+05
Model_05	180	3.5	4.2	3.51E-04	3.54E-04	100	2.85E+05	2.83E+05
Model_06	210	3.5	4.2	3.51E-04	3.54E-04	100	2.85E+05	2.82E+05
Model_07	240	3.5	4.2	3.52E-04	3.54E-04	100	2.84E+05	2.82E+05

Figure 12. Parameters for verification

By contrasting the analytical formula with the outcomes of the simulation. We are aware that the differences are minimal and not very significant.

We can also calculate the percentage difference

$$\%\text{ change in stiffness (k)} = \frac{(\text{Analytical results } - \text{simulation results})}{\text{simulation results}} * 100$$

$$\text{stiffness (k)} = \frac{(2.97 * 10^5 \; - 2.89 * 10^5)}{2.89 * 10^5} * 100 \approx 1.7\ \%$$

The % change between these approaches is 1.7 or 2 %. It is very negligible.

7. Conclusion: -

In this thesis an attempt is made to understand the behavior of internal spur gear tooth under a normal loading condition. The forces and displacement of an internal spur gear tooth which are obtained by FEM simulation. Stiffness of the gear tooth can be calculated by using results of FEM. The static analysis of internal spur gear tooth helps to determine maximum displacement, maximum induced stress.

The following conclusions can be drawn from the results obtained.

- A new analytical formula has been set up which allows fast and accurate calculations of stiffness for internal gear tooth.
- The proposed analytical approach has a real advantage the get solutions but still finite element models which are more general and flexible and can directly account for calculating stiffness.

8. References:

- On the Origin of Clockwork, Perpetual Motion Devices, and the Compass Derek J. de Solla Price Page 84 Design of Machinery

Third Edition Norton, Robert L.

Gearing in the Ancient World.

Endeavour Lewis, M. J. T. The Book of Knowledge of Ingenious Mechanical Devices

Donald Hill Economic of Medieval India, Irfan Habib Science and Civilization in China:

- Shigley's Mechanical Engineering Design

Ninth Edition Richard G. Budynas and Keith Nisbett

- Weber, C., 1949, ''The Deflection of Loaded Gears and the Effects on their Load Carrying Capacity,'' Dept. of Scientific and Industrial Research, Sponsored Research, Germany
- Weber, C., and Banaschek, K., 1953, ''Forma¨nderung und Profilru¨cknahme bei Gerad-und Schra¨gverzahnten Antriebstechnik,'' Heft 11, F. Vieweg und Sohn, Braunschweig, Germany
- Finite element Method – its basics and fundamentals by Zienkiewicz, O.C.; Taylor, R.L.; Zhu, J.Z.
- Finite Element Analysis by S.S Bhavikatti