Reverse Engineering for Gearbox

By

Fuad Alsaid Sultan Sultan

Supervisor:

Dr. Hussein Amro

Submitted to the College of Engineering

in partial fulfillment of the requirements for the

Bachelor degree in Mechatronics Engineering

Palestine Polytechnic University

December 2019

Palestine Polytechnic University

Collage of Engineering

Mechanical Engineering Department

Hebron – Palestine

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Supervisor Signature

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Department Head Signature

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Jan. 2020

Dedication

For our family,

For those who dare to hope,

For all friends of the environment,

For all enemies of war and destruction,

For whoever is trying to improve rather than worsen,

For whoever is trying to construct rather than destruct,

For whoever is leaving good impacts on their surroundings,

For whoever is working for a better world, better environment and a better future,

We dedicate our work!

Acknowledgment

Our deep gratitude goes for our university, Palestine Polytechnic University, for its continuous support for scientific research and the encouragement for students to find

Solutions for real life problems.

We would like to express our sincere gratitude to our advisor **Dr. Hussein Amro** *for his motivation, guidance and directing of our work progress toward completion.*

We would like to thank ***Eng. Majdee zalloum*** *for his unconditional help, in providing us the data we needed from his previous studies.*

We would also like to thank **Department mechanical Engineering** *who were and will always be our family.*

Abstract

Reverse Engineering in terms of mechanical engineering is the process of extracting knowledge or design of information and re-producing it based on the extracted information. The process usually includes a 3D scanning of the object ,transfer of the 3D scanned data to the computer for measurements, and data sensing, based on 3D high quality analysis. The created model using CAD is used in analyzing, modifying, optimizing, and manufacturing the desired object. This project is aimed to use this technique to find out the dimensions of an existing gearbox components and to reduce the time and the cost of manufacturing. The geometry of the product has been obtained, the 3D model of gearbox has been created using CATIA, and the 3D scanning of gearbox components have been used to find the tolerances by using Geomagic Design X software.

الملخص

الهندسة العكسية من منظور الهندسة الميكانيكية تهتم في استخراج المعرفة أو معلومات التصميم من أي نظام موجود بهدف إعادة تصميمه و إنتاجه لاحقاً إستناداً إلى المعلومات المستخرجة .وعادةً تتضمن هذه العملية القيام بمسح ضوئي ثلاثي الأبعاد للجسم المراد إعادة إنتاجه، ومن ثم يتم نقل البيانات التي تم الحصول عليها الى جهاز الحاسوب لعمل القياسات الخاصة بالنظام من خلال تحليل ثلاثي الأبعاد عالي الدقة عن طريق برمجيات خاصة .يستخدم النموذج الذي تم إنشاؤه باستخدام البرامج المحوسبة (CAD) في تحليل وتعديل وتصميم الجسم المطلوب. ويهدف هذا المشروع إلى استخدام هذه التقنية(الهندسة العكسية) لبناء وتصميم اجزاء نظام ناقل الحركة gearbox و تقليل الوقت وتكلفة الإنتاج. و سيتم ذلك من خلال توفير الابعاد الهنسية و انشاء نموذج 3D لنظام ناقل الحركة باستخدام برنامج CATIA. تم استخراج الابعاد الحقيقية لاجزاء ناقل الحركة باستخدام الماسح الضوئي 3D scanner و معالجة البيانات المستخرجة من خلال برنامج Geomagic Design X و مقارنتها مع النموذج الذي تم انشاءه من اجل ايجاد الفروقات بين النموذج و الاجزاء الحقيقية (Tolerances).

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List of Abbreviations, Symbols and Nomenclature

3D	-	three-dimensional
2D	-	two-dimensional
RE	-	reverse engineering
CAD	-	computer aided design
CAM	-	computer aided manufacturing
CMM	-	coordinate measuring machine
CNC	-	computer numerical control
d	-	Pitch diameter
a	-	addendum
b	-	Dedendum
P	-	Diametric pitch
C	-	Clearance
d_r	-	root diameter(dedendum diameter)
d_o	-	outer diameter(addendum diameter)
Φ	-	Pressure angle
Ψ	-	helix angle

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Chapter One
Introduction

1.1 Introduction

1.2 Problem Definition

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1.1 Introduction

Reverse Engineering can be defined as the process of analyzing a physical object to identify and find the relationship between the particles and components of it, and create representations of the object in another form or at a higher quality. Also, they can say that it is the process of duplicating an existing component by capturing the physical dimensions' components.[1]

The primary goal of reverse engineering is to create high precision models from the object that reflect to geometry, and the intended design behind the geometry. The knowledge of geometric and parametric constraints is not sufficient to optimize the algorithms. Reverse engineering expresses the issue of constrained optimization that produces more accurate models than the knowledge of geometric and parametric constraints.

The original models are available for comparison with the reverse engineered models through successful and powerful developed techniques that will be discussed. They will overcome the error measurements of the original design. [1].

With obtaining the CAD data, the product is still not be able to be produced. This because of missing some of manufacturing information. Usually when some product manufactured, there are a few information that must be considered such as geometric, dimensional and tolerances.[1]

1.2 Problem Definition

In this century, automotive industry has grown up rapidly. In parallel with this development, the demand for cars spare parts is increased. One of these parts is transmission gearbox. In some cases, the original manufacture products no longer exist but the customer still in need of these products. Usually the spare parts required after the car has been in serves for several years. In these days, many spare parts of gearbox are produced from companies that are not design with good quality.

1.3 Project Objectives

The main objective of this project is to apply reverse engineering process on gearbox components to develop CAD model that can be used to manufacture new products that suit the standard of industrial specification. To achieve the main objective of this project, the following should be done:

1) Develop CAD model for gearbox component.
2) Produce 3D model for gearbox that suit the standard of industrial specification.
3) Find the tolerances between the CAD model and real model of gearbox components.
4) Build 3D model for some prats of gearbox using 3D printers.

1.4 Literature Review

In the last decades, many researchers have been interested in application of reverse engineering in the field of design and manufacturing.

Belarifi et. al. [2] the propose of their research is to improve the module of conical spur gear when it became worn or broken, the aid of computer aided design (CAD) its allows to creating a virtual 3D model, using the theoretical geometric to calculate the products volume. The suggested method allows finding out the geometric features of pair conical spur gear after damage. A special AutoCAD software has been develop to accomplish 2D drawing for conical spur gear, the verification of the product assembly and creating the 3D model.

Nichat et. al. [3] explain the application of reverse engineering on the spline shaft of gearbox. An attempt has been made to find out the dimension of an existing spline shaft. The geometric data of the product obtained using coordinate measuring machine (CMM). The 3D model of product has been created using CATIA. The analysis of spline shaft has performed at various loads range from minimum to maximum value. The 3D model allows studying the behavior of spline shaft under loads condition using ANSYS.

Mohammad Shahab [4]presented the application of the reverse engineering on the modeling of pillion step holder for hero Honda CBZ motorbike. The CAD model has been developed using CATIA V5. The stress analysis of pillion step holder was done. Results found that the maximum stress is within the permissible limit and the deflection in the component is less than the permissible value. Again, the stress is performing on the CAD model show that the maximum stress and deflection is still within the permissible value. The CAD model helps to understand the behavior under various load conditions and help to modify it.

Ismail et. al. [5]explained the modeling of four-stroke piston engine using reverse engineering method. The process include digitizing process to capture the point clouds by using layout machine and following by the CAD and CAM stage using unigraphics NX2 software to rebuild the piston engine surface. Then machining the process with CNC to create the piston head engine. Finally, the accuracy of the replicated piston engine checked by using CMM and block gauge.

All of the previously mentioned studies, focused on providing a model using physical measurements and available theories without comparing them with the real model. In this project, a 3D model for gearbox components was developed using CATIA parametric modeling, moreover, a 3D scanner was used to scan these parts; to be compared with the developed model, in order to identify the differences in geometric dimensions between the real model and the model that’s drawn in CATIA, using Geomagic Design X software.

1.5 Task Table

Table 1.1 Task Table

Task ID	Task Description
T1	Project Selection
T2	Collection Information for Project
T3	Literature Review
T4	Learning CATIA Software
T5	Drawing Parts and Assembly
T6	Learning Geomagic Design X Software
T7	Scanning Gearbox Components
T8	Preparing Scanning Data by Geomagic in a Required Shape
T9	Find the tolerances for the objects
T10	Test The Result
T11	Documentation
T12	Writing the Text
T13	Making the Final Adjustments on the Text

1.6 Time Table

Table 1.2 First Semester Time Table

	1st semester															
Task/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Project Selection	■	■	■													
Collection Information				■	■	■	■									
Literature Review							■	■	■							
Learning CATIA Software						■	■	■	■							
Drawing Parts								■	■	■	■	■	■			
Parts Assembly											■	■	■			
Documentation													■	■	■	■

Table 1.3 Second Semester Time Table

	2nd semester															
Task/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Learning Geomagic Design X	■	■	■	■	■	■										
Scanning Gearbox Components			■	■	■	■	■	■	■	■	■	■				
Preparing Scanning Data by Geomagic in a Required Shape				■	■	■	■	■	■	■	■	■	■	■		
Find the tolerances											■	■	■	■		
Writing the Text								■	■	■	■	■	■	■		
Test The Result													■	■	■	
Make the Final Adjustments												■	■	■	■	■

2

Chapter Two
Background

2.1 Reverse Engineering

Engineering is the process involved in designing, manufacturing, constructing, assembling and maintaining products, systems and structure. There are two types of engineering: forward engineering and reverse engineering.

2.1.1 Forward Engineering (FE)

Forward engineering is the traditional process of building from a high-level model and logical design to the physical implementation such as the design, manufacture, machines processes and systems. **Figure 2.1** show engineering flow.

The forward engineering involves the building of product or system by following the general specification using scientific and mathematical principles, where the old system specifications are analyzed, reconstructed and regenerated to get the best quality product [6].

2.1.2 Reverse Engineering (RE)

Reverse engineering is the process of discovering the technical principles of a product, device or system by analyzing its structure, function and operation.

Components or software program analyzing its work and functionality in details to be used in maintenance or try to make a new product that does the same thing without copying anything from the original product[6]

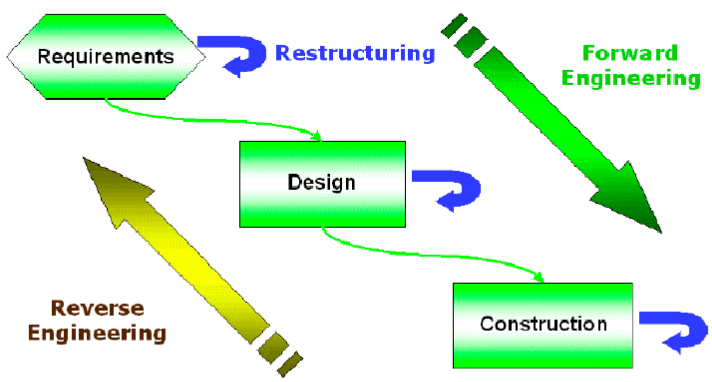


Figure 2.1 Flow of engineering

Reverse engineering carries several benefits in order to make the product last longer. Where there is no information about the dimensions of an object except for the physical item itself, the quickest and most reliable way to reproduce it will be by Reverse Engineering.

2.1.3 Application of Reverse Engineering

Reverse engineering have several benefits in order to make the product last longer. The goal of reverse engineering is to maintain the product if the manufacture no longer produces the product or no longer exist, but the costumer still need the product. Reverse engineering used when the original product has become outdated. In summation of that special function, reverse engineering has made to obtaining data to improve or to manufacture products for which there is no CAD data or data has become obsolete or lost. Moreover, some undesirable features of the products need to be eliminated or reduced. Such as excessive damage that may indicate where the product should be improve. Hereafter, when analyzing the quality features of products, the products features can become stronger based on long-term usage. [7]

Reverse engineering can be used in mechanical industry to achieve the following requirements:

- Create 3D model of a physical product with lost documentation.
- Keep the digital 3D data of a product.
- Analyze the working of a product.
- Measure wear of tools.
- Survey and compare actual geometry with CAD model.
- Estimate competitor's products.

In some situations, designers give a shape to their ideas by using plaster, wood, or rubber, but CAD model is still needed to enable the manufacturing of the product. As products become more essential in shape, designing may be challenging or impossible. There is no warranty that the CAD model will be acceptable close to performed model. This problem can be solve by using reverse engineering because the physical model consider as source of information for the CAD model[7]

Reverse engineering also used to reduce product development times and cost. In the intensely competitive global market, manufacturers are looks for a new ways to reduce lead-times to market new products. Rapid product developments assign to recently developed technologies and techniques that used to assist manufactures and designers to meet the demands of reducing product development times. Where reverse engineering is used, a three dimensional product or model can be captured faster in digital form, remodeled and exported for rapid prototyping or manufacturing. [7]

In general reverse engineering can be viewed as the process of analyzing a system to:

- Identify the systems components and their interrelationships.
- Create representation of the system in another form or higher level of abstraction.
- Create the physical representation of the system.

2.2 Reverse Engineering Processes

Reverse engineering process deals with the detailed study of the product to extract knowledge about the design, material, structure, surface qualities, and working conditions of the product. Reverse engineering process contains four major steps, which are used. The procedural steps are define as data acquisition, preprocessing data, triangulation and CAD geometry creation. The whole process of reverse engineering should be computer aided.

When applying reverse engineering on a product some points must be consider in planning strategy such as the reason for using reverse engineering (documentation lost or the original product no longer exist etc.), product size (small or large), product materials (hard or soft), product geometry and accuracy required[8]

2.2.1 Data Acquisition

This stage intended to gather the data about the geometric aspect of the physical part. There are many ways of gathering valuable dimensional information about the product such as physical measurements and data scanner (3D scanning). In this stage, first prepare the physical part of product, which is to be created for scanning then performing the actual scanning using a suitable scanner (laser or CMM) as shown in **Figure 2.2** in order to size all the information that related to the physical features of the product such as holes, slots, curves, steps etc.

Figure 2.2 3D scanning

2.2.2 Preprocessing/Point Processing

In this stage, the data scanning result is processed in order to remove the noise and to reduce the number of data points. These processes is performed using a range of predefined filters. The advantage of this stage that allow us to merge multiple scan data sets. Sometimes we performing multiple scans process on the product to make sure that all features are scanned. This involves rotating the product; hence, each scan datum become very crucial, so that the merging of data sets can be done successfully which will reduce the effort required in the point processing and avoid the errors from merging multiple scan data. A wide range of commercial software's are available for point processing. The output of the point processing stage is clean, merged, point cloud data sets in the most convenient format.

2.2.3 Triangulation and Feature Extraction

Triangulation is the process of formation and division of point data into triangle. In this stage, triangulation of point data is achieved by creation of triangular mesh using a suitable algorithm as shown in **Figure 2.3**. After triangulation process the feature of the product need to extract, this step is define as the process of defining a set of features, or image characteristic, which will represent the information that is important for analysis and classification.

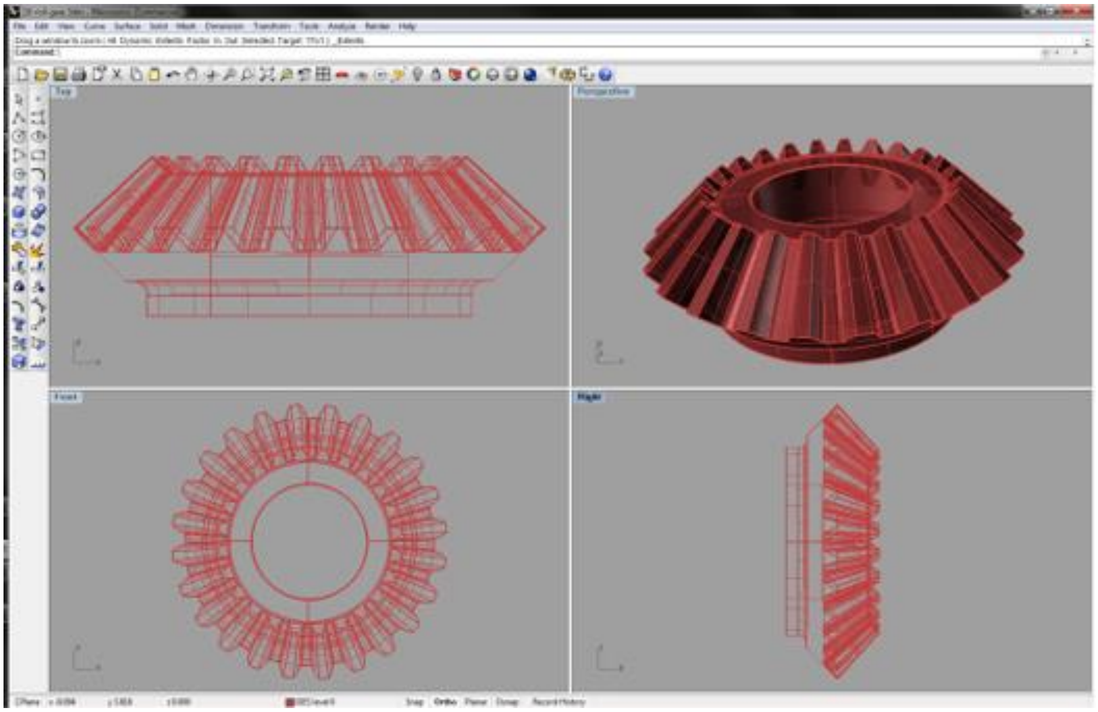


Figure 2.3 Point data division

2.2.4 Segmentation and Surface Fitting

This is known as the most important and critical stage in reverse engineering. Segmentation is a complex process as dynamic fitting algorithms are used for surface generation that flawlessly represent the 3D information depicted point cloud data sets.

Segmentation is the process of splitting a triangular mesh into sub- meshes, which is an appropriate signal surface fitted and affect seriously on the quality of the resulting CAD model. Most CAD system are not designed to display and process large amount of point data for the product, as a result new reverse engineering modules are needed for point processing.

2.3 Reverse Engineering Software's

Reverse engineering process requires software that reconstructs the object as a 3D model. At the beginning, the physical object can be measured using 3D scanning technologies, like a coordinate measuring machine, laser scanner, structured light digitizer or computed tomography, then a high-performance program requires to deal with 3D scanning, and that will be by using a unique software's as Geomagic packages.

Geomagic package "first series were released in nineteen ninety-seven" is a professional computer-aid, which substitutes the requirements of processing and geometric modeling. Selection of Geomagic product depends on the features of this application according to the intended work. The main programs that will use reengineering process are:

Geomagic Design X is one of Scanning Software, which has a feature that doesn't exist in other software's, it owns a combination of an automatic and a guided 3D model extraction, it converts 3D scan data into high quality feature and more accurate CAD models, also it creates a custom component that integrates perfectly with existing products and requires a perfect fit.

Geomagic Studio it's also an integrated program that transforms the 3D scanned data into highly accurate surface and 3D CAD model, in general it has a professional characteristic that can edit point cloud, making mesh analysis, also it has tools that can interstate the internal structure of any object, which help to create a high quality of 3D models, and Optimize for fast data processing

In this project, the process will be done by using Geomagic Design X due to the features that program own, in addition to other software such as CATIA, which assist the program in order to make a modification to final output.

2.4 Gears Geometry

Gears are one of the ancient tools its origins goes to 27th Century B.C in china, Aristotle had the earliest description for the gears in 4th century B.C, and his description is the direction of rotation is reversed when one gear wheel drives another gear wheel, the Greek Inventors use the gears in water wheels and clocks. The major development to the gear happened at 17th century, first attempts were done to provide constant velocity ratios through involute curves in the 19th century was the starts of hopping presses by Whitworth in 1835, and in 1897 Herman Pfauter invented the first hopping machine can cut both spur and helical gears. The next major step came in 1975 when the Pfauter Company in Germany introduced the first NC hopping machine and in 1982 the Full 6 axis machine was introduced[9].

The popular gear types that are used in vehicles gear box are helical and spur as shown in **Figure 2.4.**

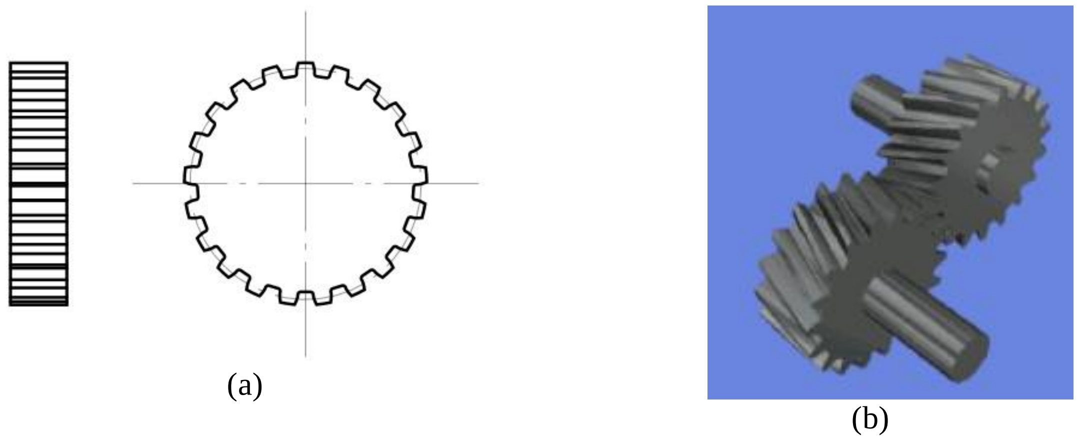


Figure 2.4 Gear types

Spur Gear as in **Figure 2.4(a)** the teeth are parallel to the axis of rotation; they are used to convey the rotation between parallel shafts, compare to other types, spur is the simplest.

Helical gear as in **Figure 2.4 (b)** The different between it and spur is that the helical teeth are inclined to the axis that cause more gradual engagement in the meshing, so less noise and thrust load and bending couples develops over the spur gear, helical gear can be used to transmit motion between non parallel shafts.

Helical and spur are sharing the same profile -parallel to axis view- with the same parameters and the parameters[10]. See **Figure 2.5**

- The pitch circle d as given in **Eq. (2.1)** is theoretical circle upon which all calculations are usually based; its diameter called pitch diameter ‘d’, the pitch circles of a pair of mating gears are tangent to each other.

$$d = \frac{N}{P} \quad 2.1$$

  Where:
  N: number of teeth.
  P: diametric pitch usually in inch

- The addendum ‘a’ as given in **Eq. (2.2)** is the radial distance between the top land and the pitch circle and its value depend mainly on P

$$a = \frac{1}{P} \quad 2.2$$

- The dedendum ‘b’ as given in **Eq. (2.3)** is the radial distance from the bottom land to the pitch circle also as ‘a’ its depend on P

$$b = \frac{1.25}{p} \quad 2.3$$

- The portion of the tooth between the clearance circle and the dedendum circle includes the fillet called the clearance and its value depend in ‘a’ and ‘b’ as given in **Eq. (2.4)**.

$$c = b - a \tag{2.4}$$

- The dedendum circle also called the root circle and its value can be determined for the dedendum and the pitch diameter as given in **Eq. (2.5)**

$$d_r = d - 2b \tag{2.5}$$

- Also the outer diameter is called the addendum circle and it’s the easiest parameter to measure and has relation with the pitch and the pitch diameter as given in **Eq. (2.6)**

$$d_o = d + 2a \tag{2.6}$$

- Using (*2.6*), (2.2), (2.1) a relation between outer diameter ’do’, number of teeth ’N’ and the diametric pitch as given in **Eq. (2.7)**

$$P = \frac{N+2}{D_o} \tag{2.7}$$

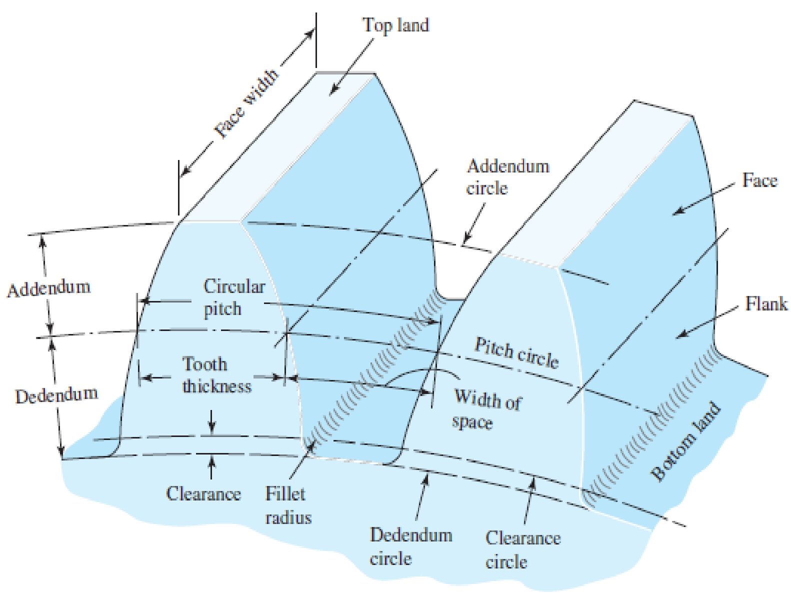


Figure 2.5 Gear profile

The most need of all of the parameter is to define the shape of the tooth, however all of those parameters are not enough to draw the teeth so function for the side of the teeth is needed and that function is the involute and it can be defined as shown in **Figure 2.6**.

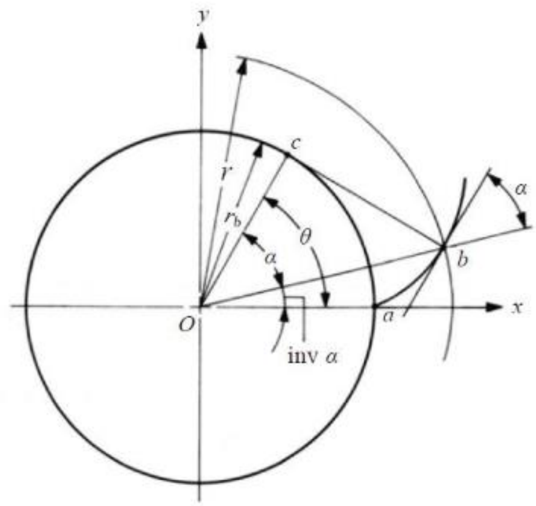


Figure 2.6 Involute curve

First variable in the figure is inv α can be found as given in **Eq. (2.8)**

$$inv\ \alpha = tan\ \alpha - \alpha \quad 2.8$$

Where:

α: is the pressure angle

The values of x and y can be found from **Eq. (2.9)**, **Eq. (2.10)** and **Eq. (2.11)**

$$\alpha = \cos^{-1}\frac{r_b}{r} \quad 2.9$$

$$x = r\cos(inv\ \alpha) \quad 2.10$$

$$y = r\sin(inv\ \alpha) \quad 2.11$$

Where:

rb; is the base circle radius and its value depend on d and the pressure angle at the pitch diameter Φ as in **Eq. (2.12)** [6]

$$d_b = dcoc\phi \quad 2.12$$

3

Chapter Three
3D Modeling

3.1 CATIA

3.1.1 Introduction

CATIA V5 (**C**omputer-**A**ided **T**hree-dimensional **I**nteractive **A**pplication) is the product of the highest technological level and represents standard in the scope of designing. [11]

Instantaneously, it is the most modern integrated CAD/CAM/CAE software system that can be find for commercial use and scientific-research work. The biggest and well-known world companies and their subcontractors use it. It is the most spread in the car industry, airplane industry, and production of machinery. [11]

3.1.2 Why CATIA

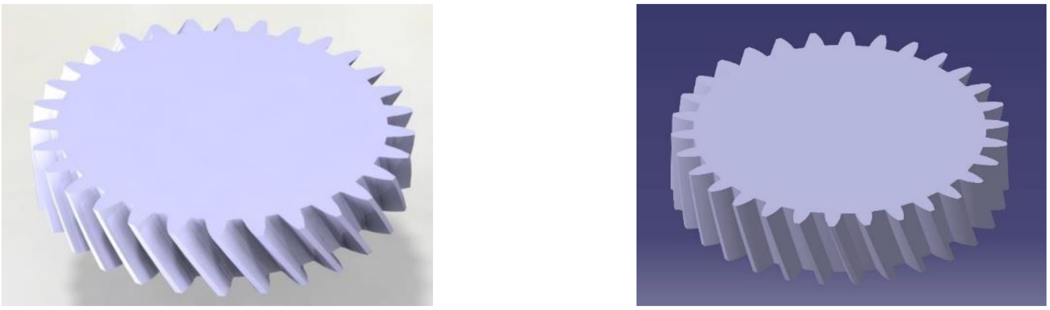
In this project, CATIA V5 will be used to develop 3D CAD model for gearbox components, CATIA has important modules such **Part design, generative shape design and Assembly design** that use in cylindrical gear modeling. The main problem in any gears modeling is to define real gear parameters. This program was choose to work on over other CAD programs such as SOLIDWORKS for the following reasons:

- It is easy to change the parameters to create new part with different parameters without any intervention from the user but in solidworks, it is more complicated because it is need to make a new pattern and trim all the intersection.
- CATIA can deal with the pitch circle particularly when meshing the gears.
- It is easy to make extrapolate for the involute curve.
- CATIA does not need computers with high specification to run.

Though all the above advantages SOLIDWORKS has its own features such as:

- More interactive view that make user access the tools easier.
- Involute curve can be created as continues line without any points.

Figure 3.1 show the different between the same gear in CATIA and SOLIDWORKS.



(a) (b)

Figure 3.1 Different between CATIA and SOLIDWOKS

3.2 Scanning by Using 3D Scanner

3D scanning become more popular in various fields, and its usage for different purpose, this means that there are a different kinds of objects, simple or complex can collect the data of them, and many details to understand[7].

Collecting data about object can be done by two-way hand held or modern scanning. The first way is traditional scanning or manually driven sensors such as calipers and micrometers. These devices fall within the paradigm of touch sensing, requiring sort of probe to physical contact the surface of the part and the second way is modern scanning within use 3D laser scanners[7].

3D laser scanners are a technology that digitally captures the shape of physical objects using a line of laser light as shown in **Figure 3.2**. 3D laser scanners create "point clouds" of data from the surface of an object. In other words, 3D laser scanning is a way to capture a physical object's exact size and shape into the computer world as a digital 3D representation, with high resolution and minimum error data[7].

Figure 3.2 3D scanning

3.2.1 Types of Scanning Technologies

In these days, there are many types of 3D scanning technologies that are available at markets. The most commonly used technologies fall into three class: Displacement, Profile and Snapshot.

1) Displacement devices use a projection for single point beam laser as shown in **Figure 3.3** to measure the height, thickness, or position of a project.

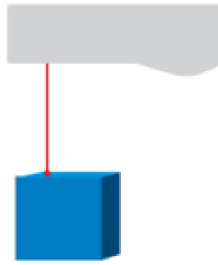


Figure 3.3 Single point laser

2) Line Profile devices typically use a projected laser line to create a cross action profile for measuring aspects of an object contour. Moving an object under the laser line creates many profiles that can be combined into a complete 3D shape. **Figure 3.4** describe the process of this type.

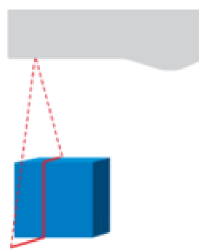


Figure 3.4 Laser profile

3) Snapshot devices use structured light and stereo-vision to generate full 3D volume data. Because Snapshot technology captures so much 3D data at one time, objects need to remain stationary during the scanning process. **Figure 3.5** describe the process of this type

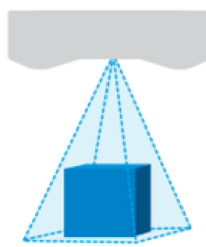


Figure 3.5 Snapshot devices

3.2.2 Powerful of 3D Scanning Data

3D scanner is a device that captures a real object or environment as 3D shape data. The collected 3D information is converted into digital data commonly called 3D scan data or scan data, which is a set of points. The points represent the location on a real object and contains the X, Y, and Z coordinates. Numerous points can be used to describe a real object. For example, a digital photo with a high-resolution pixel count can represent the detailed shape of a real object. **Figure 3.6** describe scan data resolution.

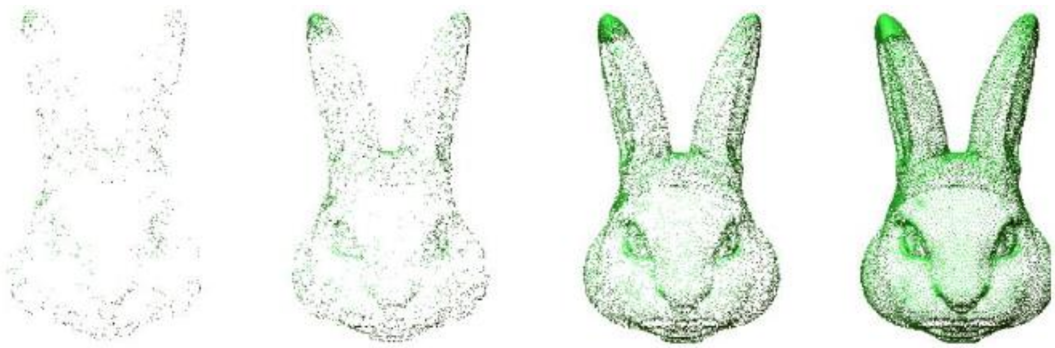


Figure 3.6 3D scan data by resolution

3D scan data has the following strengths:

- Quickly create digital versions of real objects
- Accuracy
- Capture complex freeform surfaces
- Capture small to large scale objects (depending on 3D scanner)
- Obtain color information (depending on 3D scanner)
- Simulate environments and situations
- Change to different scales and measurements easily
- Easily extract length, height, width, volume and position data
- Extract sectional information
- Compatibility in a general PC environment

Since 3D scan data can represent a real object with high accuracy, it is used for various purposes. The usage of scan data is increasing rapidly to custom markets as scan technology becomes easier to use and more intuitive. Currently, 3D scan data used primarily for the various purposes, the following show two types of them:

1. Measurement and inspection

   3D scan data usage become more popular for metrology or inspection. The more accurate points that are used to measure a feature will increase the credibility of an inspection. The scan data of a manufactured part can be aligned with its CAD model, and subsequently compare to check for differences and whether or not they pass/fail within set tolerances.

2. Ultra HD 3D scanner

This is one of scanners, which has an excellent feature such as:

- The dimensional accuracy of this it is about ±100 micron in micro mode.
- The process speed about 50 thousand points per second.
- Twin 5 megapixel CMOS image sensor.
- Scans can be output in different forms such as STL, OBJ, VRML, XYZ, and PLY files, which is available with Geomagic design x program.

This type use Multi Stripe Laser Triangulation (MLT) Technology, this technology is relatively cheaper than other scanners, so it can be available for schools, universities and some people interested with reverse engineering. **Figure 3.7** shows this type of scanner.

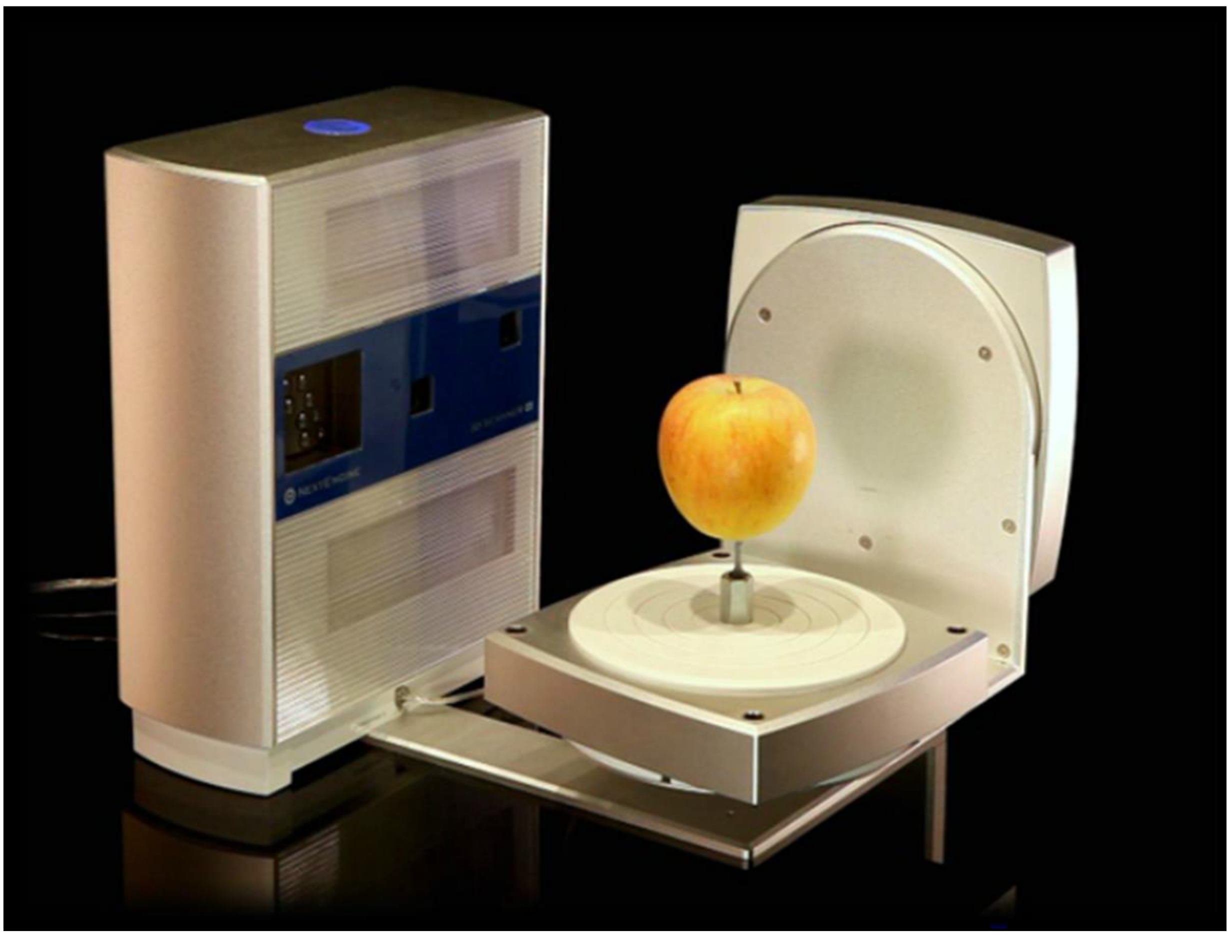


Figure 3.7 Multi Stripe Laser Triangulation (MLT)

3.3 Geomagic Design X

3.3.1 Introduction

Geomagic Design X, the industry's most comprehensive reverse engineering software, combines history-based CAD with 3D scan data processing so the creating feature-based, editable solid models compatible with an existing CAD software, through the experiment of Geomagic design X, there are unique features that this program owned, such as [11]

1) Powerful and Flexible

Geomagic Design X is purpose-built for converting 3D scan data into high-quality feature-based CAD models. It does what no other software can with its combination of automatic and guided solid model extraction, incredibly accurate exact surface fitting to organic 3D scans, mesh editing and point cloud processing. Now, you can scan virtually anything and create manufacturing-ready designs [12].

2) Do the Impossible

Create products that cannot be designed without reverse engineering, customized parts that require a perfect fit with the human body. Create components that integrate perfectly with existing products. Recreate complex geometry that cannot be measured any other way [11].

3) Works Seamlessly with Your Existing CAD

Geomagic Design X connects directly to popular CAD software, including SOLIDWORKS, Siemens NX, Autodesk and CATIA. Using unique Live Transfer technology, Design X transfers complete models, including feature trees, so you can quickly create solid and surface models from 3D scans.

3.3.2 Mesh

Mesh is polyhedron-based 3D digital data which consists of points, edges, and faces usually Triangles. The mesh is frequently used in CAD/CAM/CAE and 3D computer graphics programs. Modern graphic cards are not optimized for rendering point clouds but have advanced technology to display meshes. Therefore, meshes often more advantages in regards to a smooth visualization of complex surfaces and structures of an object. The triangulation, or meshing, is a process to connect 3 points and construct surface as shown in **Figure 3.8** .When connecting points, the edge length can be an important parameter to make a correct object shape.

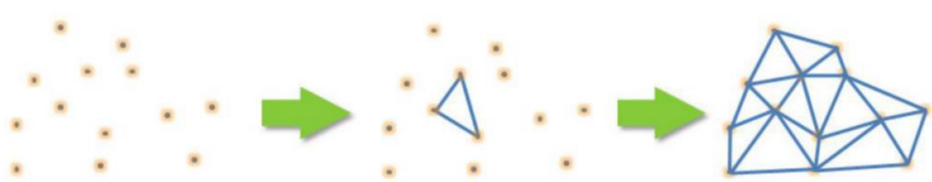


Figure 3.8 Point to mesh sample

3.3.2.1 Healing Wizard

It's one of important commands in Mesh Phase, which detects abnormal poly-faces, such as Non-manifold poly vertices, fold, small, crossing, non-manifold and dangling poly faces, small clusters and small tunnels in a mesh and automatically heals them. **Figure 3.9** describe healing wizard command[12].

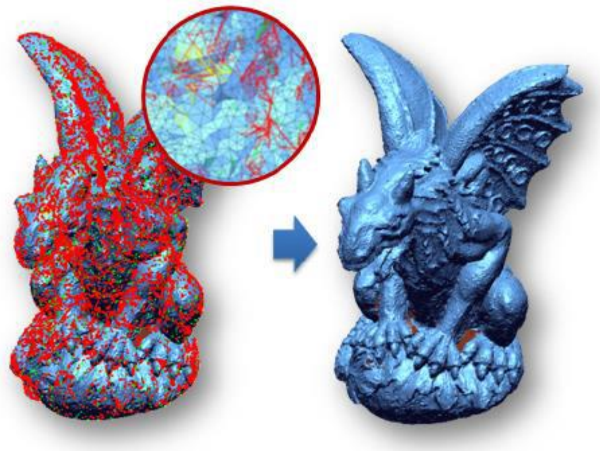


Figure 3.9 Healing wizard detection

3.3.2.2 Edit Boundaries

Command edits the boundaries of mesh as shown in **Figure 3.10** . This command features multiple boundary editing methods such as smooth, shrink, fit, extrude, and fill[12]. (FIG.)

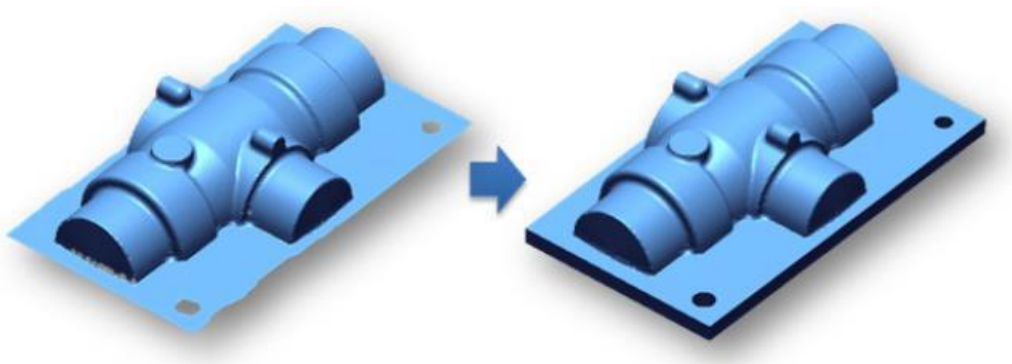


Figure 3.10 Edit boundaries

3.3.2.3 Fill Hole

Command fills in missing holes with poly-faces based on feature shapes of a mesh as shown in **Figure 3.11**

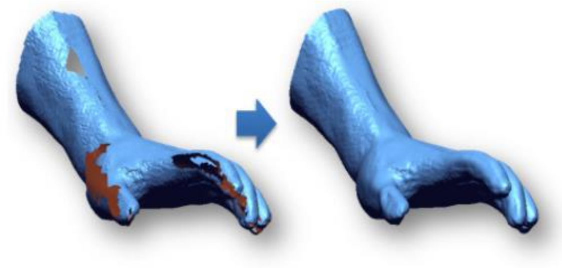


Figure 3.11 Fill hole

3.3.3 Region Group

Region Group classifies areas of mesh based on geometry features. The Region Group mode automatically or manually classify[12].

- Auto Segment

  The Auto Segment command automatically classifies feature regions by recognizing 3D features.

- Merge

  The Merge command merges multiple feature regions as shown in **Figure 3.12** into a single feature region.

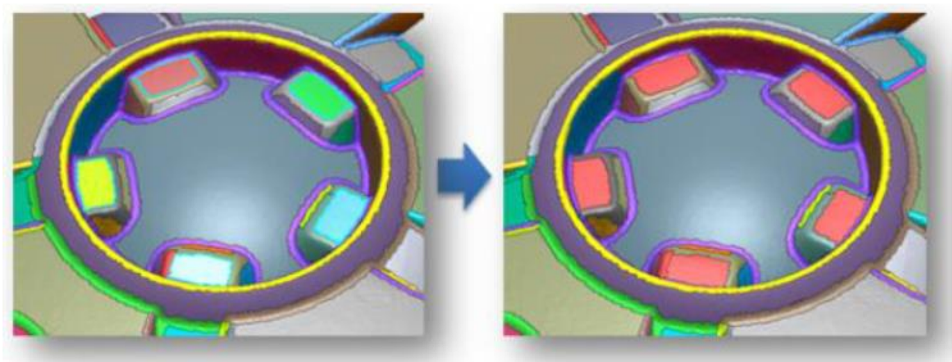


Figure 3.12 Merge in Geomagic Design X

- Split

  The Split command divides a feature region into multiple parts. **Figure 3.13** describe this stage, notice that the color indicated to split command.

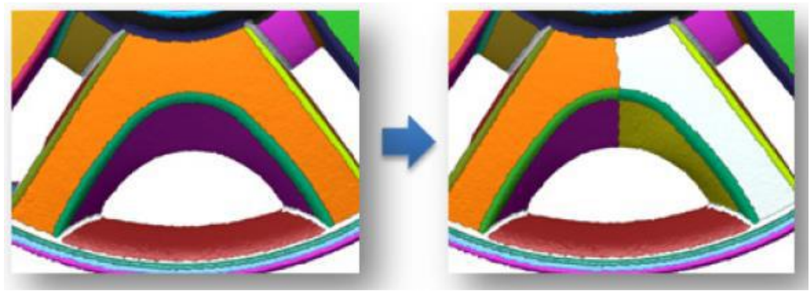


Figure 3.13 Split in Geomagic Design X

3.3.4 Sketching

- Mesh sketch

  This mode extracts sectional polylines or silhouette polylines based on a mesh or a point cloud as well as the creation of 2D geometries such as lines, circles, arcs, rectangles based on Extracted polylines. **Figure 3.14** show mesh sketch for an object.

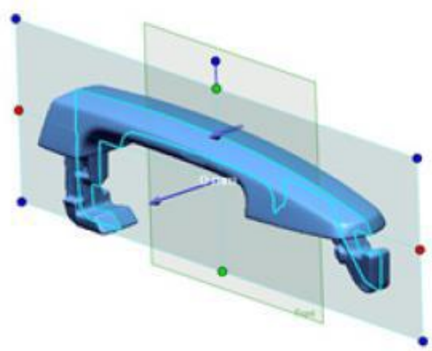


Figure 3.14 Mesh sketch

- Sketch

  The Sketch mode creates 2D geometries such as lines, arcs, and splines, and can edit created 2D geometries.

- 3D Mesh Sketch

  3D Mesh Sketch creates splines on a mesh as shown in **Figure 3.15** using various commands. This splines can be edit by using tools such as the Trim, Offset, and Project commands.

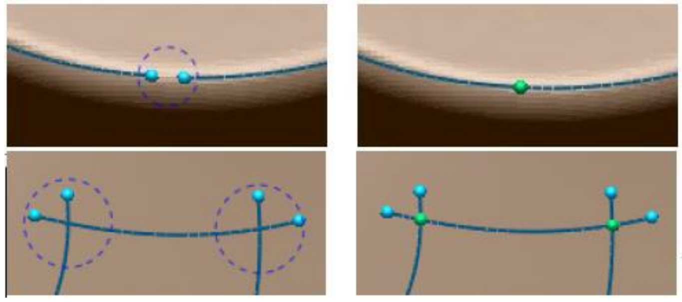


Figure 3.15 3D Mesh Sketch

- 3D Sketch

  3D Sketch mode creates 3D geometries such as splines, sections and boundaries on a mesh. That help to make curves can be edited by using the Trim, Offset, and Project commands.

3.3.5 Auto Surfacing

Auto Surfacing is used in the Modeling phase and is an innovative and automatic tool for fitting surface patches onto a mesh and creating a surface body. **Figure 3.16** shows auto surface process.

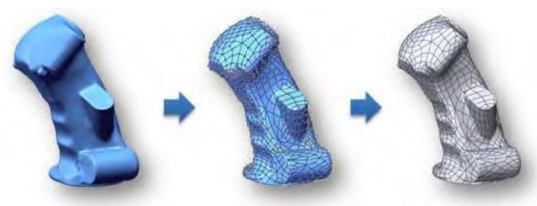


Figure 3.16 Auto surfacing

Solid

The Insert Solid menu features various commands used to generate and edit solid bodies and surfaces.

Generating Solid Bodies from Sketches:

- Extrude

Creates extruded bodies with a sketch, direction and length

- Revolve

Creates revolved bodies with a sketch, axis and revolution angle

- Sweep

Creates swept bodies with a sketch and path

- Loft

Creates lofted bodies with a set of sketches

3.3.6 Accuracy Analyzer

The Accuracy Analyzer is used in the Model Verification phase as shown in **figure 3.17**. Which is a real-time analysis tool for creating the best 3D feature model from 3D scan data within design tolerances. This tool analyzes deviations between the intermediate or final model with the raw 3D scan data and verifies the results. The analyzed results can be used for creating an optimal 3D model within allowable design tolerances

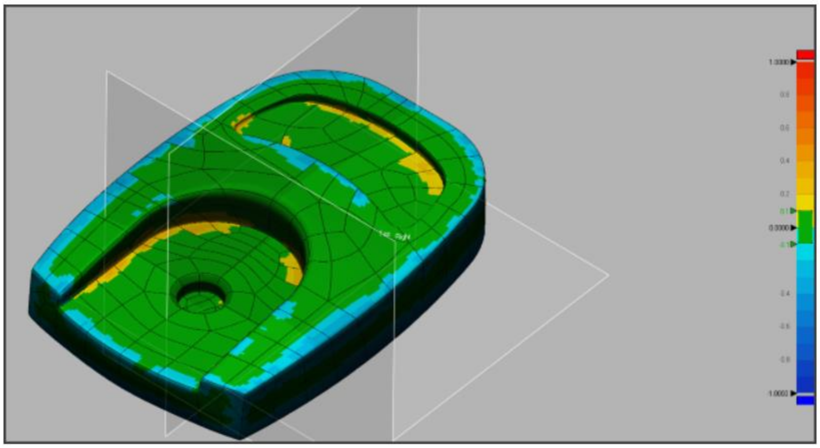


Figure 3.17 Accuracy Analyser

4

Chapter Four
Action Sequence

4.1 Drawing by Using CATIA for an Object

4.2 Processes of Geomagic Design X for an Object

4.1 Drawing by Using CATIA for an Object

4.1.1 Parametric Gear Modeling

The parametric modeling method that developed in this project provides an accurate involute curve creation using formulas and exact geometric equations. In addition, the involute curve by parametric modeling allows using either Cartesian in terms of X and Y, or cylindrical coordinate systems to create the involute curve profile. Since spur and helical gears geometry is controlled by a few basic parameters, a generic gear can be designed by three common parameters namely the pressure angle, pitch diameter and the number of teeth.[11]

When opening CATIA program the interface which shown in **Figure 4.1** will open, so it is easy to start drawing the side surface for the gears to be able to fill it with solid material. Generative shape design is most suitable for this operation. [13]

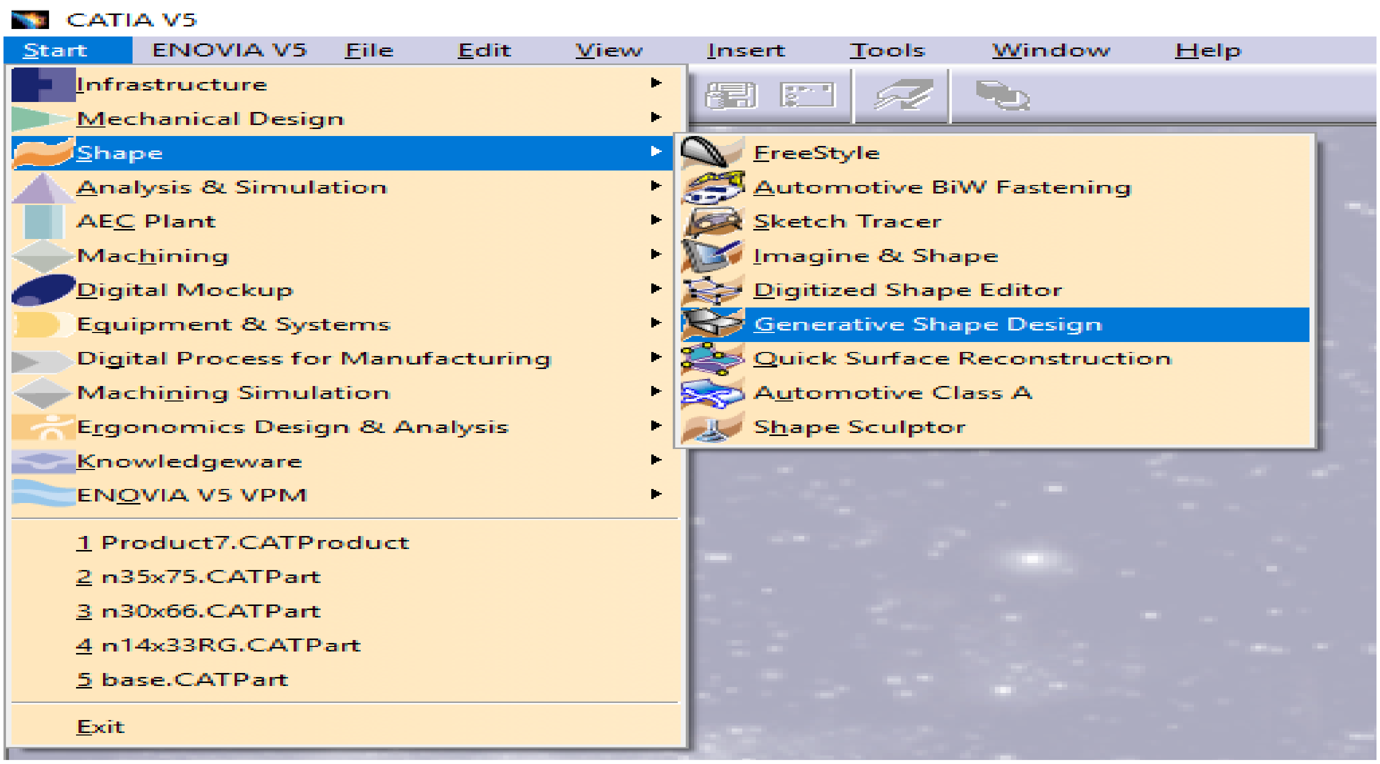


Figure 4.1 CATIA start page

From generative shape design window go to formula from knowledge command as shown in **Figure 4.2** and define the main parameters names values and type.

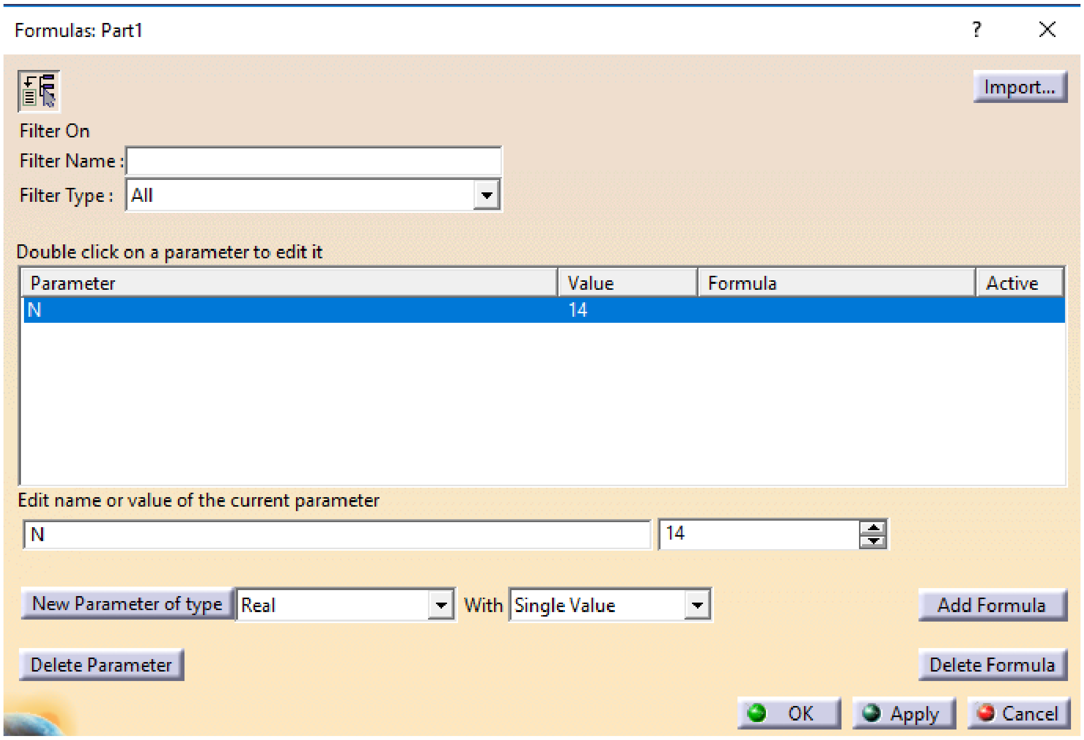


Figure 4.2 Defined gear parameters in CATIA

Now after all parameters are defined it is time to create a relation that implement the X and Y for involute curves as show in **figure 4.3** using fog from knowledge and writing the relations as in Eq. (4.) and Eq. (4.).[13]

$$x = r_b \sin \pi t + r_b t\pi \cos \pi t \qquad 4.1$$

$$y = r_b \cos \pi t + r_b t\pi \sin \pi t \qquad 4.2$$

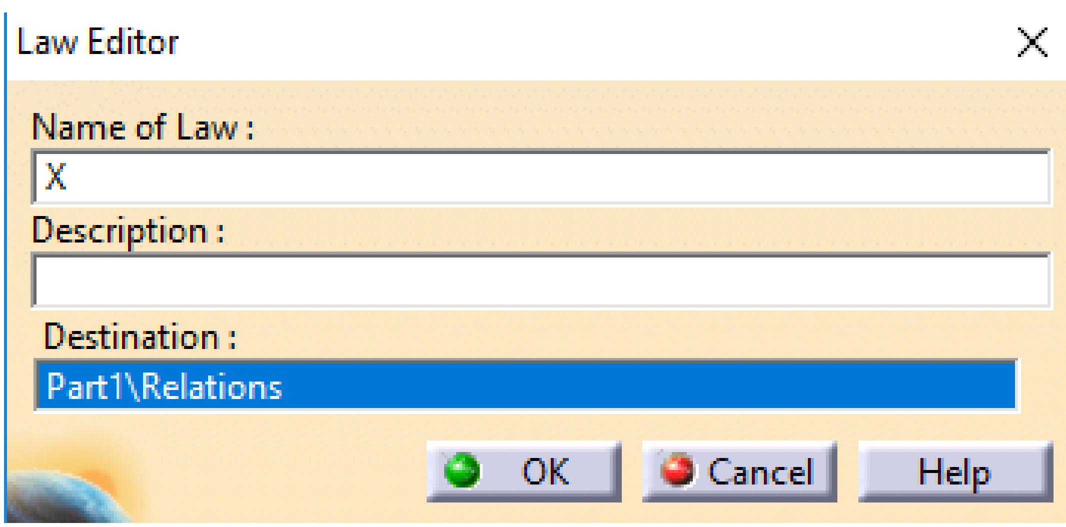


Figure 4.3 Create X low for involute

After creating the low by clicking on ok, window in **Figure 4.4** should appear and defined variable t and create **x** law to be function of **t** as in Eq. (4.1). Then do the same for y law and you can defined t in the y law with the same name in x law by using Eq. (4.2).

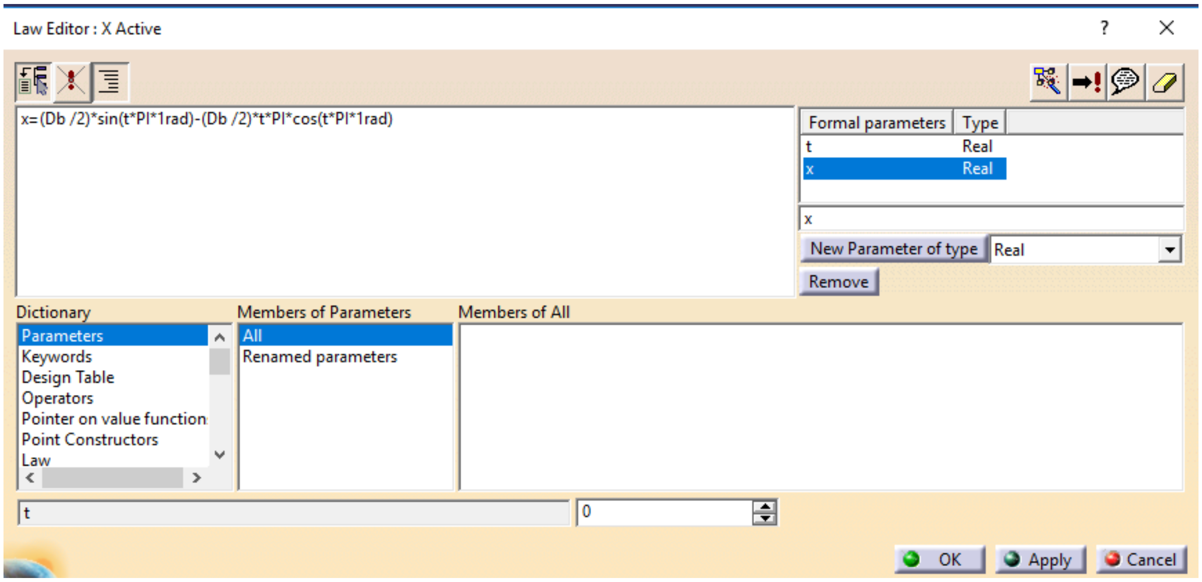


Figure 4.4 Low of x

Now in **Figure 4.5** you need to create points on the involute curve putting them on the XY plane. Then click in H value and chose Edit formula the window as shown in **Figure 4.5** should pop up.

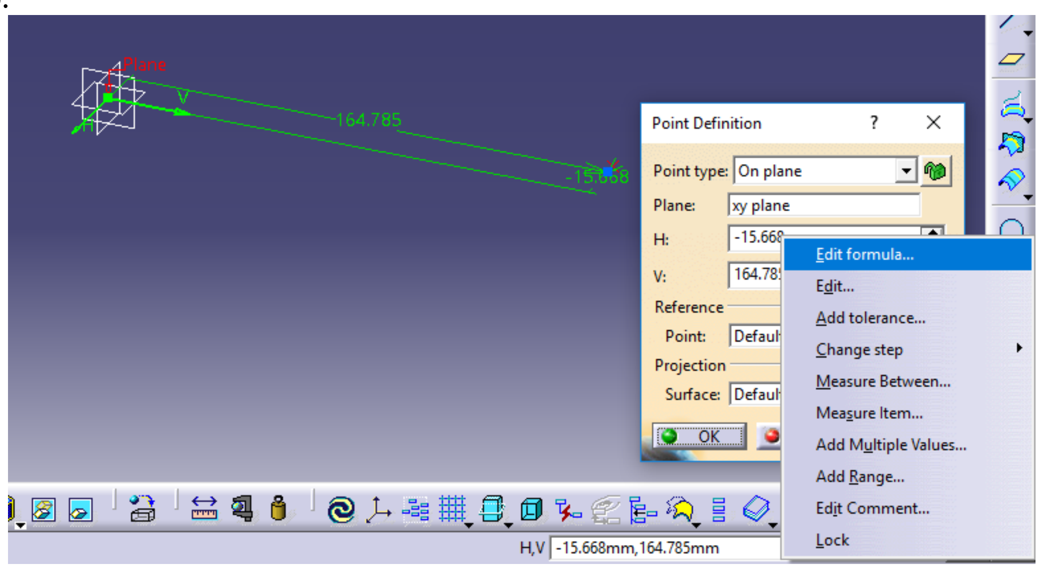


Figure 4.5 Starting involute point

Figure 4.6 show how to create the x coordinate for the point by the x law that defined **figure 4.4**. It can be used by entering the CATIA code Relations\x .Evaluate (t). Then choose the value of t for the first point it would be 0 and repeating the same from **Figure 4.5** by this time for V witch value is determined by code Relations\x .Evaluate (t) and the value of t mast be the same as for x

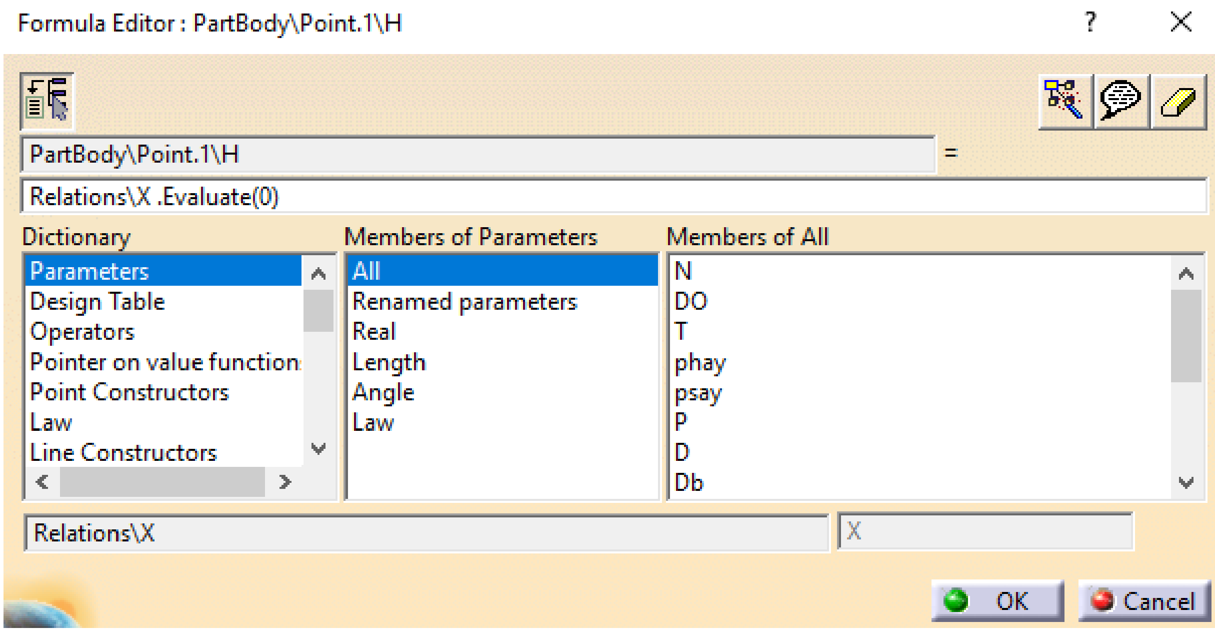


Figure 4.6 Identify x for involute point

By repeating the same process for t = 0.0 to t = 0.5 by step of 0.1 user will get the points as in **Figure 4.7** all of these points are in the involute curve.

Figure 4.7 Involute points

The points alone are not useful so points should be connected together by spline as in **Figure 4.8.**

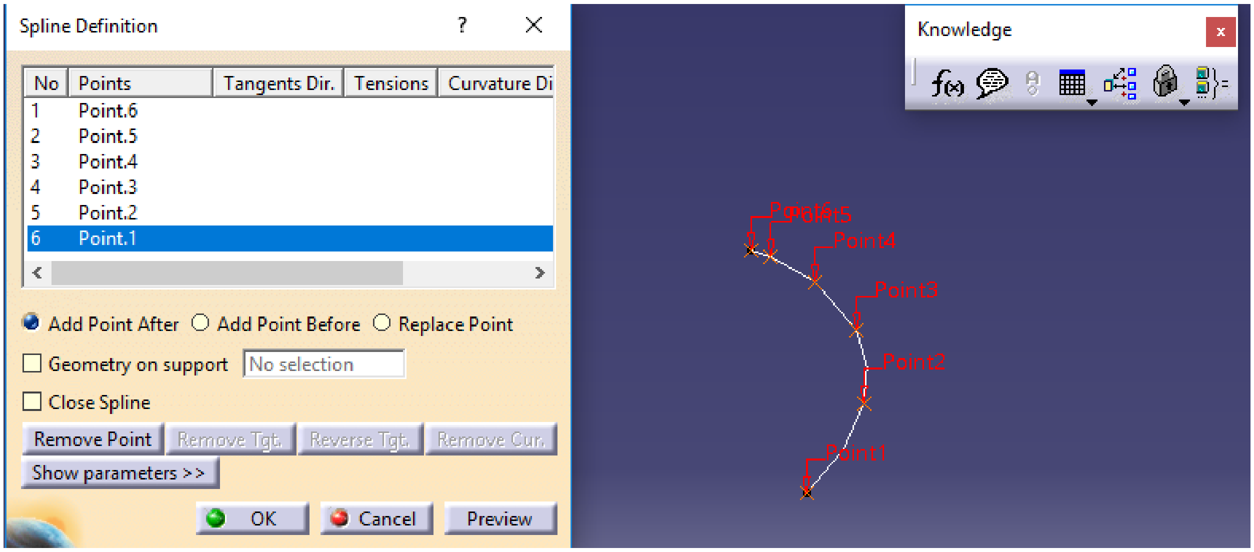


Figure 4.8 Create spline to represent involute curve

In **Figure 4.9** drawing the base circle on XY plane and selecting its center to be at the origin.

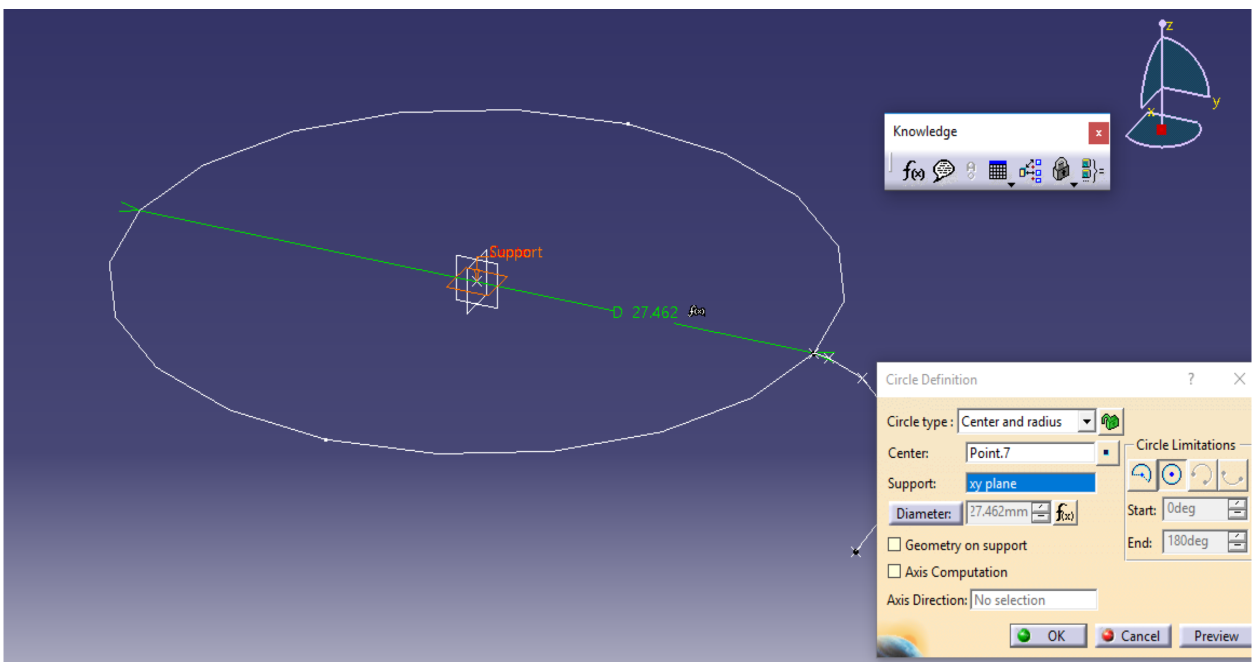


Figure 4.9 Base circle

In **Figure 4.10** extrapolating the spline to make it reach the root radius choosing the boundary to be the first point (t=0) and the Extrapolated to be spline and the light at least $\mathbf{r_b - r_r}$ chose 1.5 of that distances.

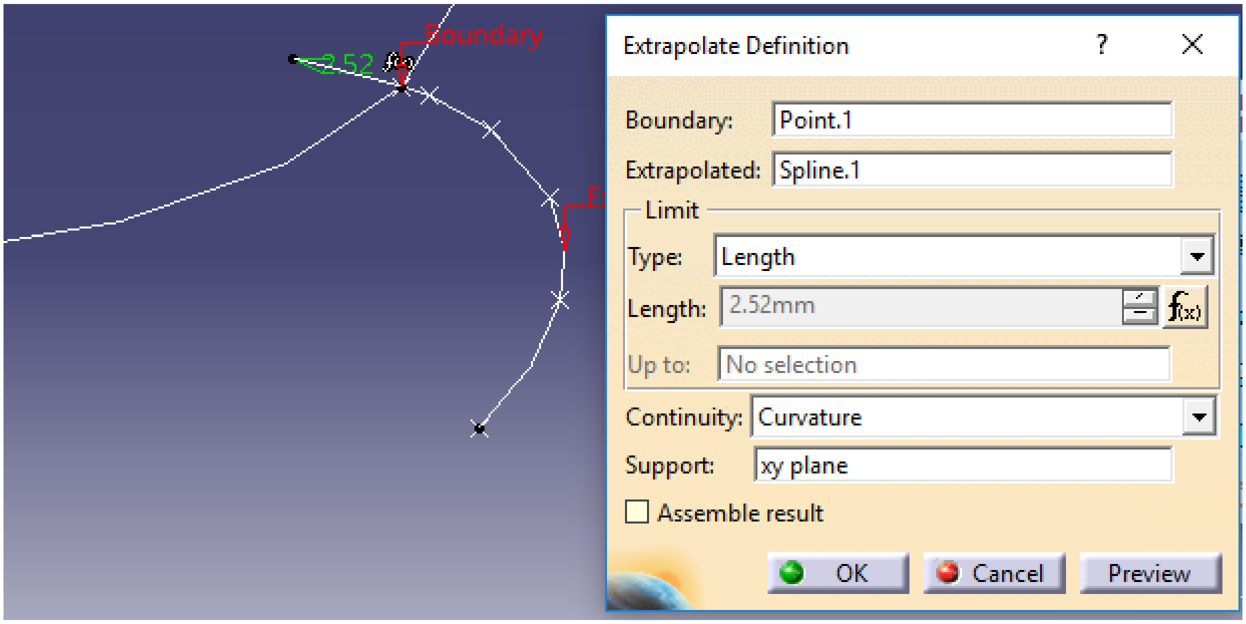


Figure 4.10 Extrapolating the spline

In **Figure 4.11** a plane is created to be the symmetry axis of the tooth and its type is Angle/Normal to plane the rotation axis of it is z-axis with reference YZ plane with angle 90/N.

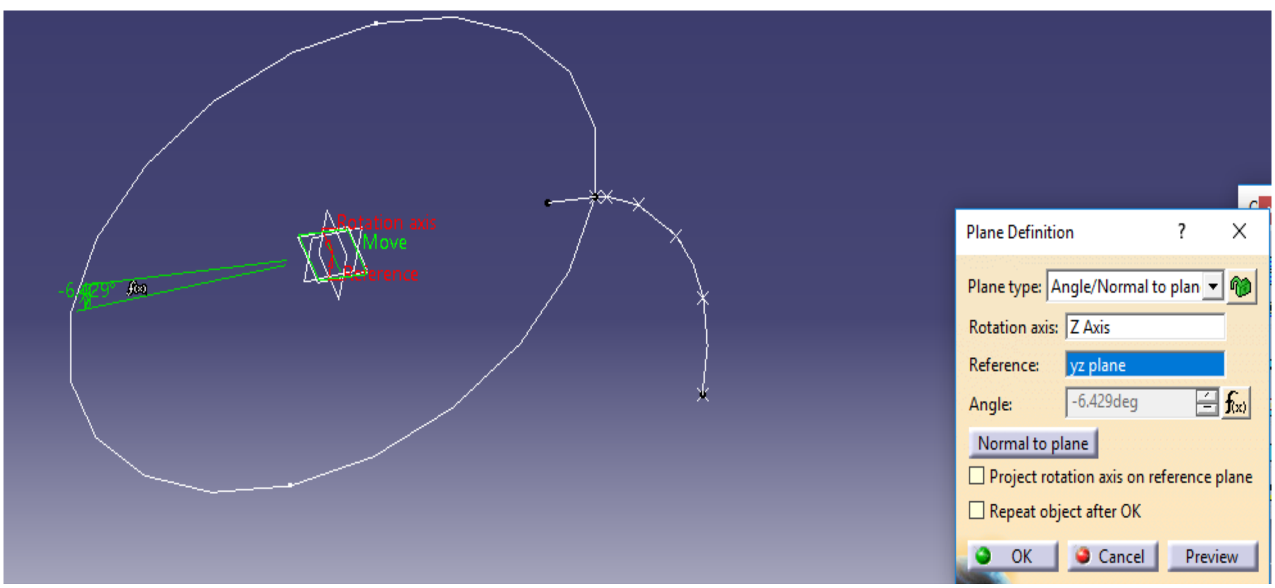


Figure 4.11 Create the involute symmetry plane

In **Figure 4.12**, corner is created between the extrapolate and root circle, the radius of the corner is C so trim the rest of extrapolate.

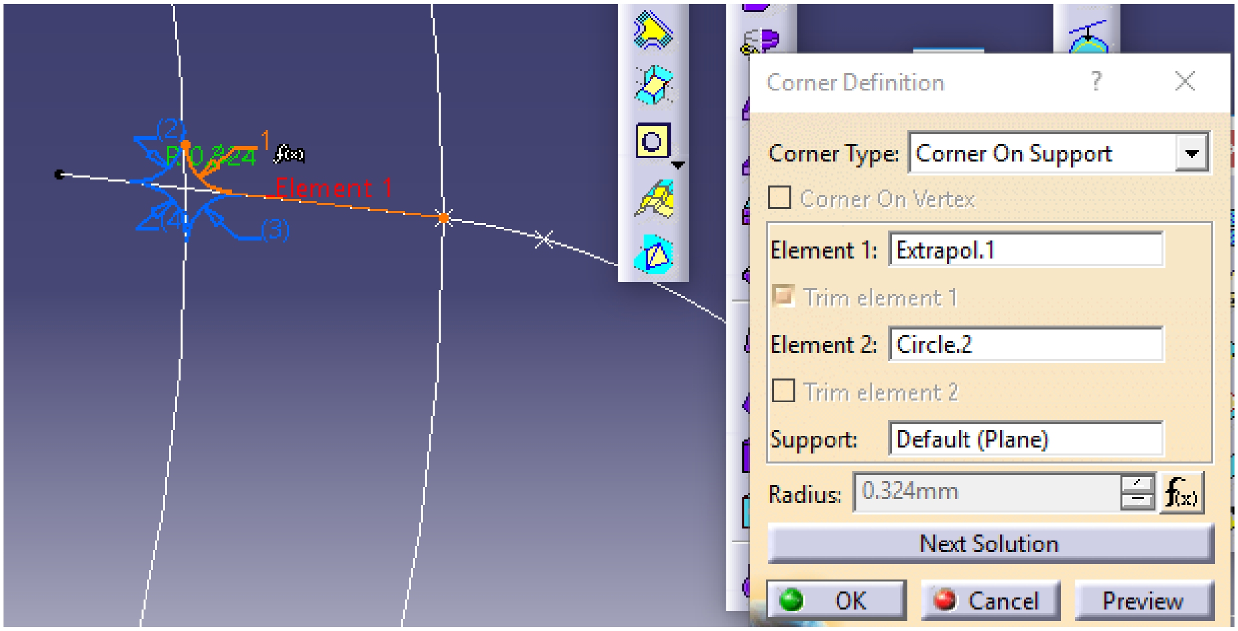


Figure 4.12 Making corner

Now the plane created in **Figure 4.11** is used to make symmetry about it for the corner and extrapolate and the result will be as in **Figure 4.13.**

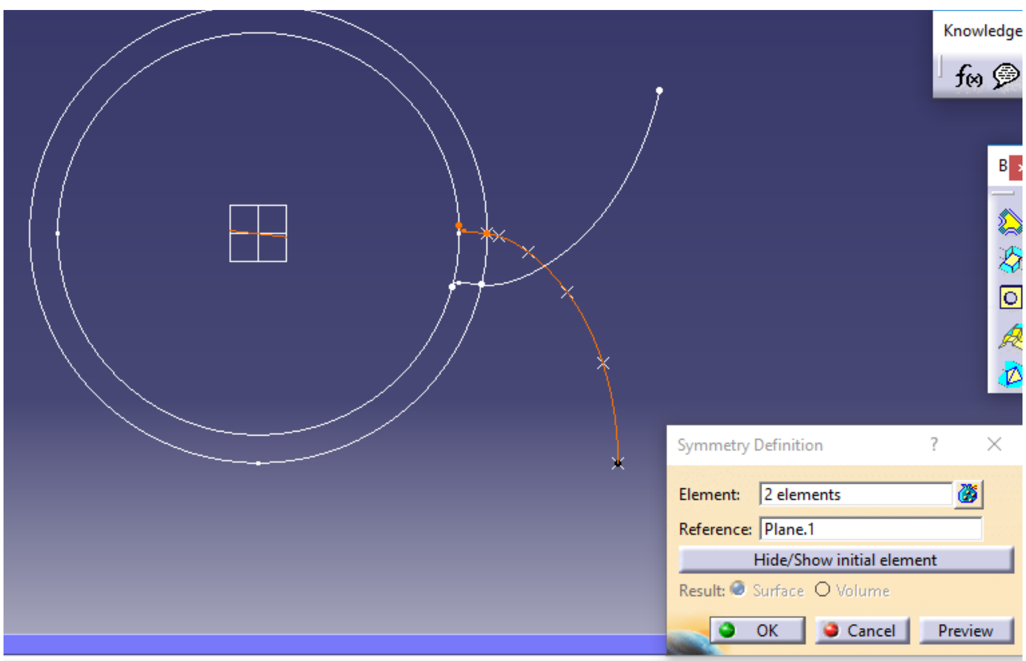


Figure 4.13 Symmetry about plan1

In **Figure 4.14** trim command has been used between the symmetry and the addendum circle to get the tooth shape.

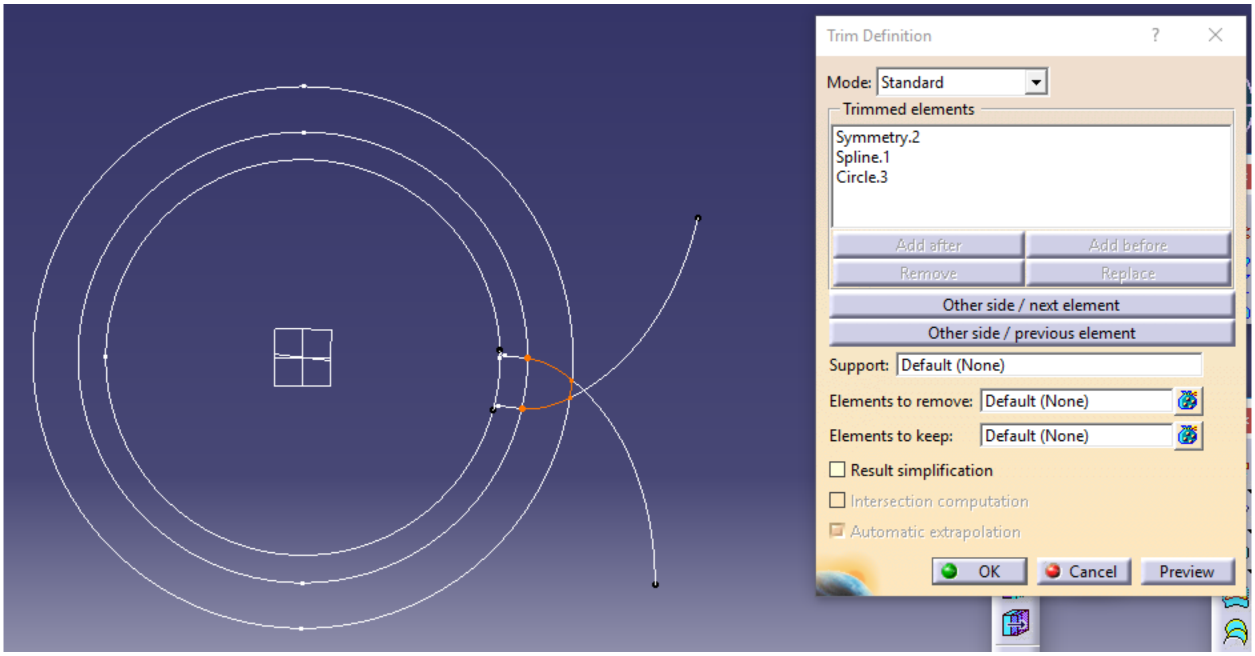


Figure 4.14 Trimming

In **Figure 4.15** joining the tooth lines to be able to deal with it as one line. Note that the points and base circle are hided not deleted.

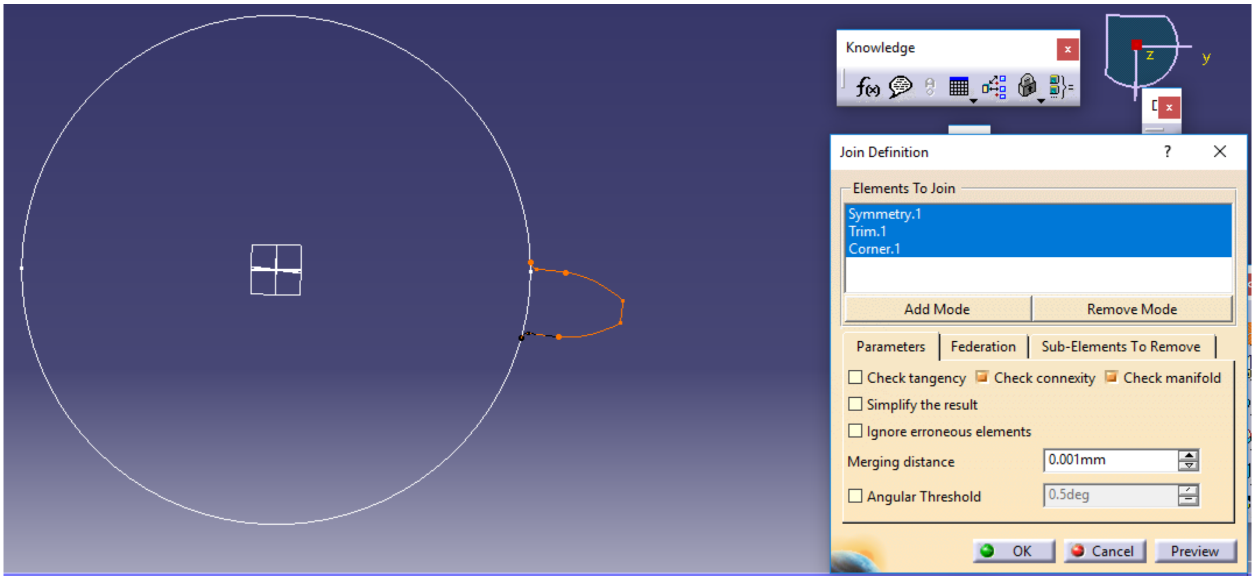


Figure 4.15 Joining the tooth parts

In **Figure 4.16** pattern made for the teeth bout the Z axis with parameter full crown and instances N and the program will automatically choose the angle.

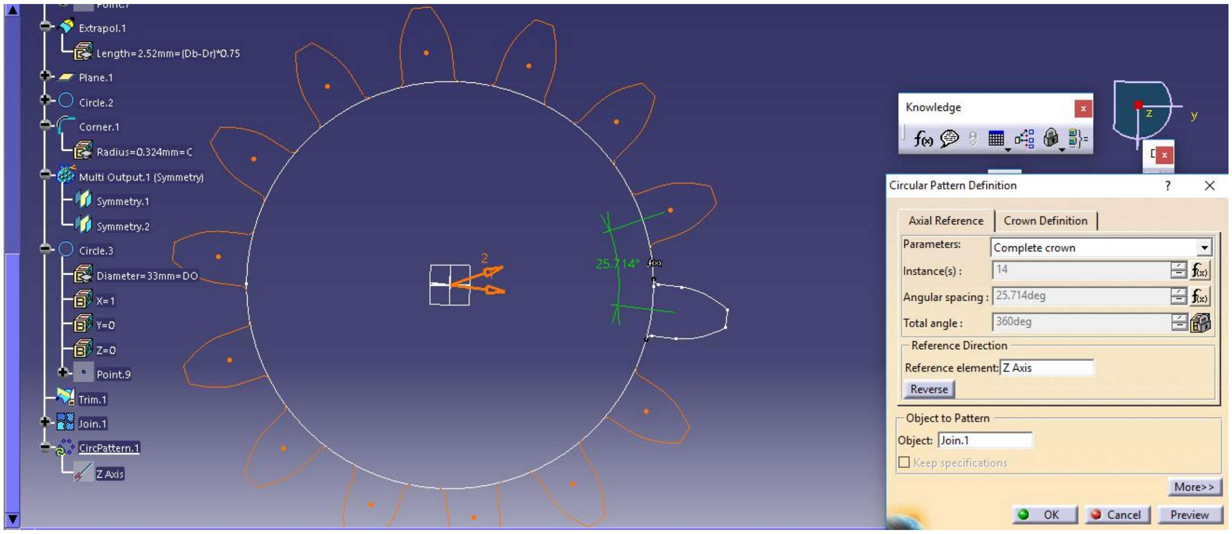


Figure 4.16Pattern of teeth

To create any shape in CATIA the shape should be uninterrupted, so trim is need, in CATIA there is two method to trim the first is trim piece by piece and the second joining the parts then trim ones. **Figure 4.17** show joining method.

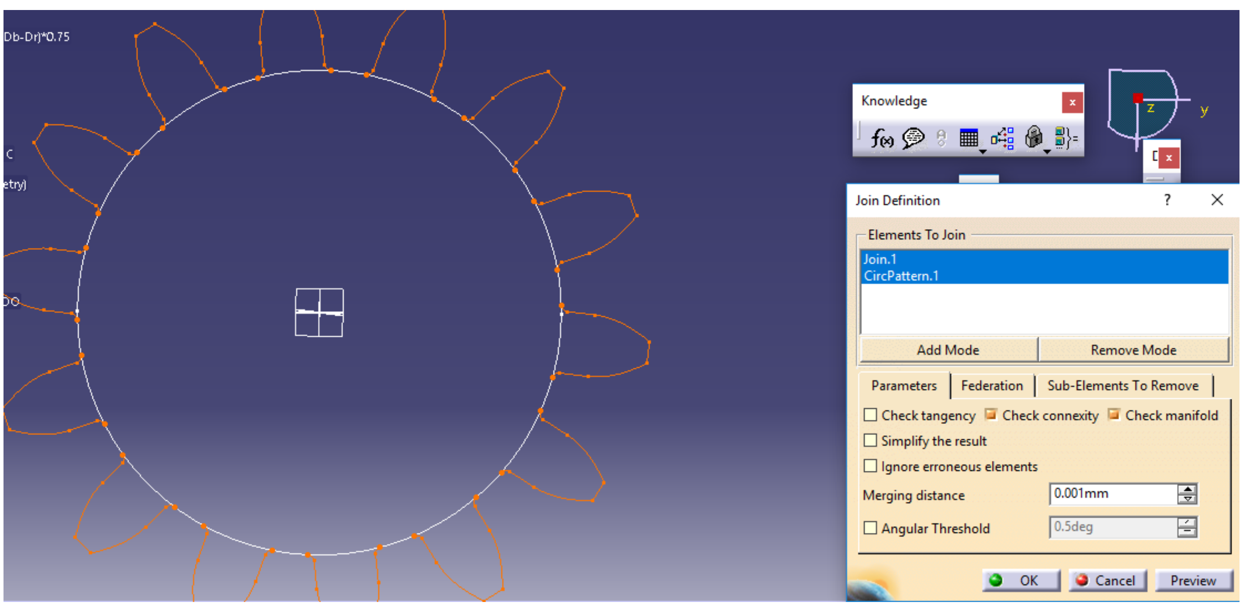


Figure 4.17 Teeth joining

After trimming the shape shown in **Figure 4.18** will be generated and that is the gear profile, spur or helical gear could be made from it depend on the following steps.

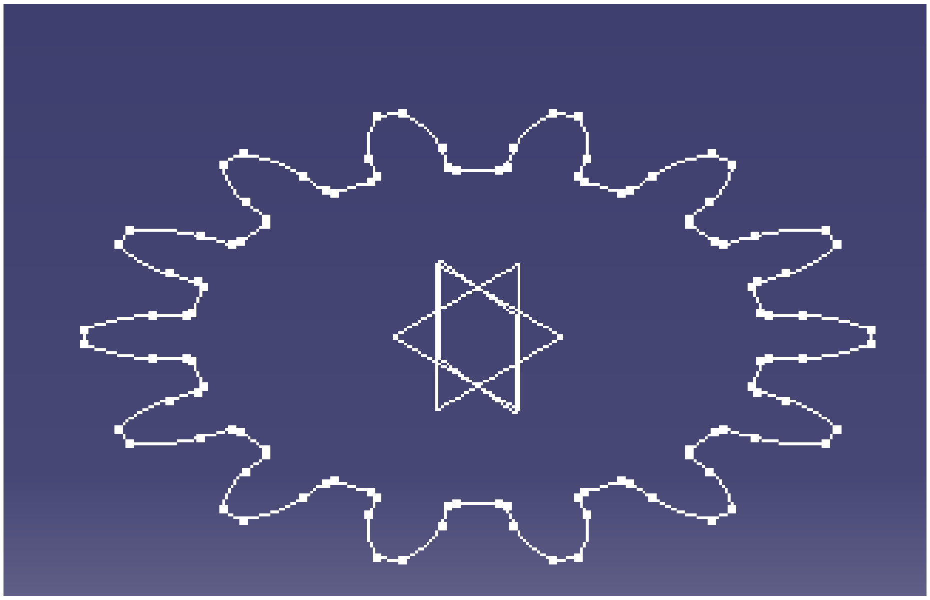


Figure 4.18 Trim the teeth with the base circle

In **figure 4.19** Making z-axis translated copy of the profile with distance T.

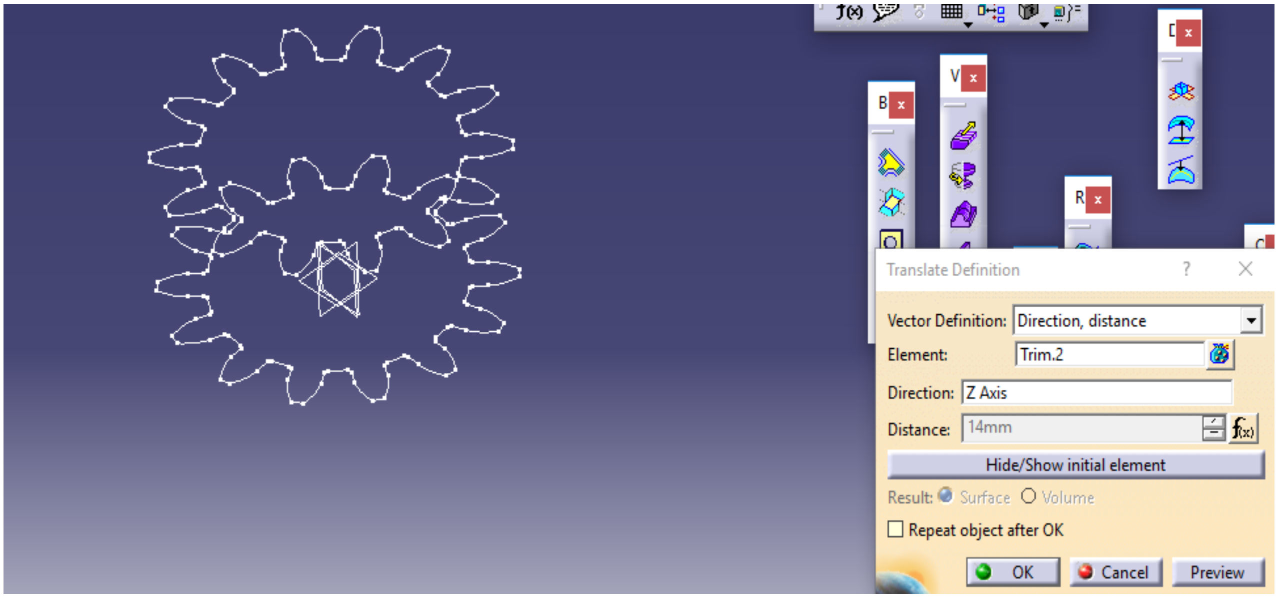


Figure 4.19 Translate

Now the following steps are for the helical gear. Rotating the profile created in the previous step about Z-axis with helix angle Ψ as shown in **figure 4.20**.

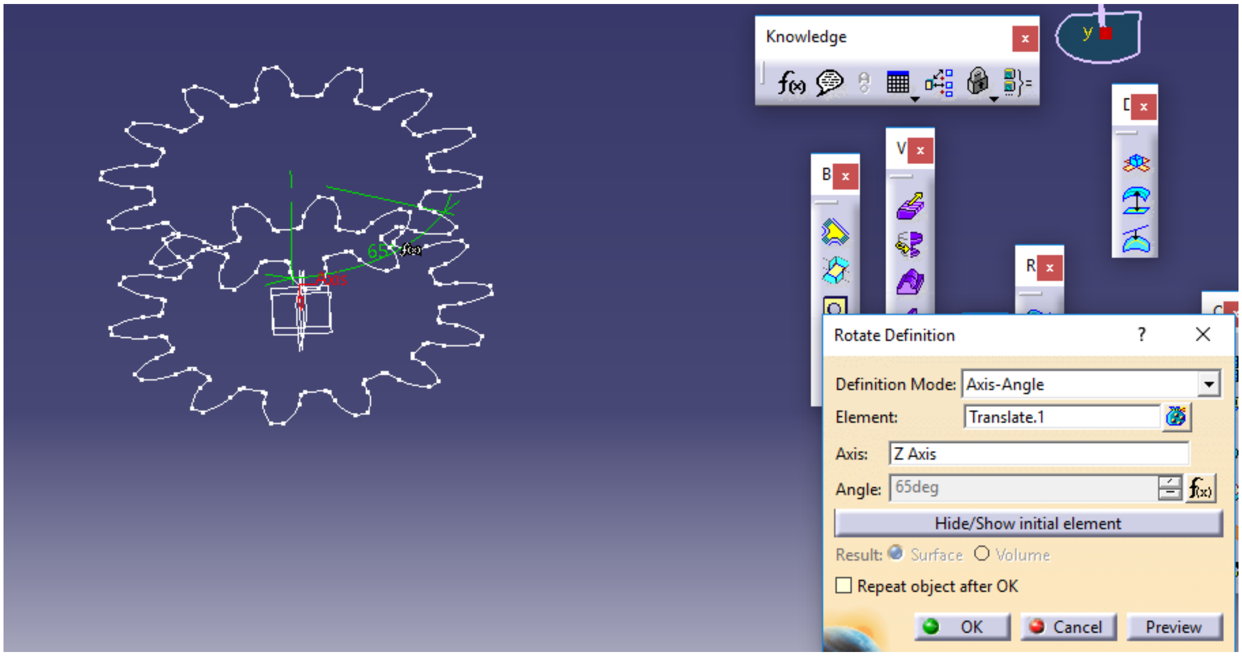


Figure 4.20 Rotate the translated

Now a guide to follow is created for the helix as in **Figure 4.21** with pitch T/Ψ in radian and width T with needed orientation and choosing point in profile it should be closing with the opposite in the rotated profile.

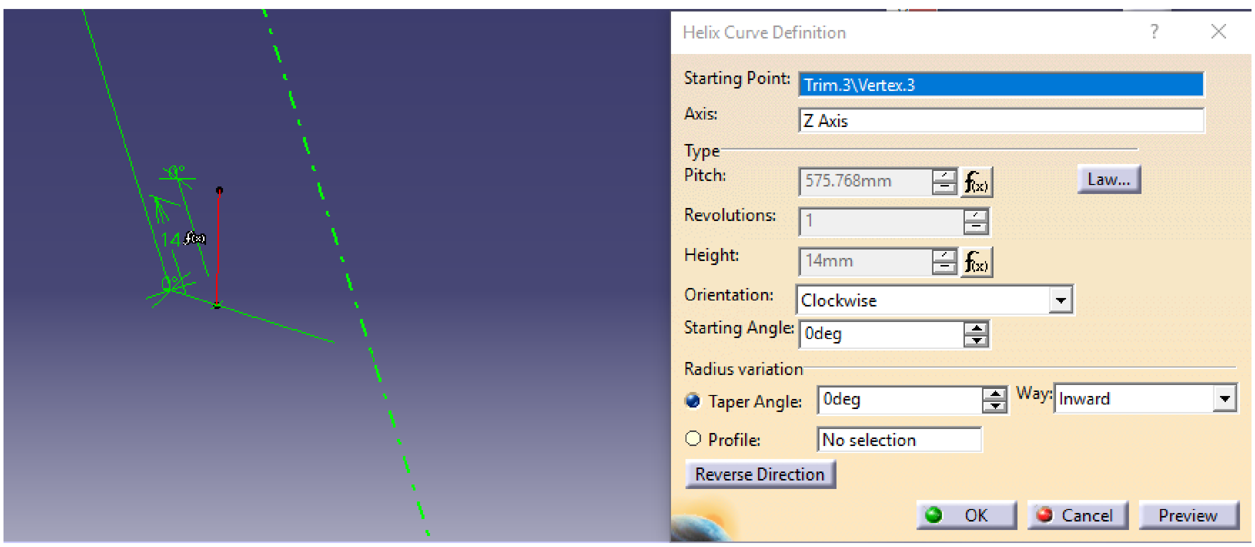


Figure 4.21 Helix guide

After creating the helix now it is possible creating the side surface for the gear by using Multi-sections surface as in **Figure 4.22** putting sections to be the initial profile and the rotated one and guide to be the helix created in **Figure 4.21**.

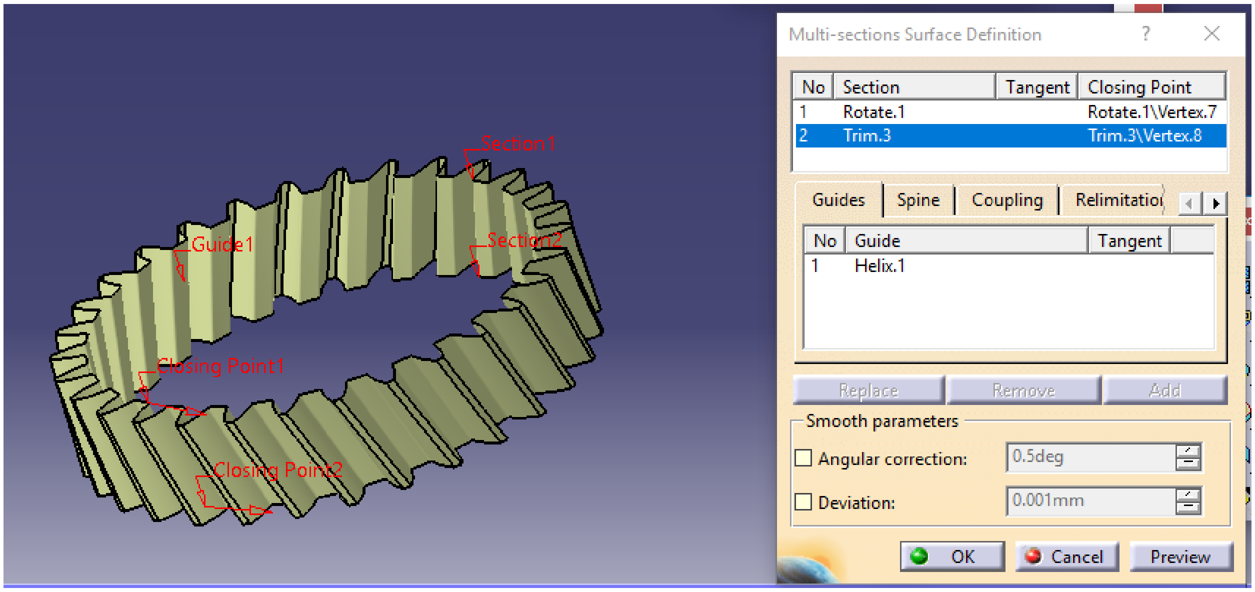


Figure 4.22 Side surface

Now filling the rotated profile as in **Figure 4.23** to be able to fill material in the close profile.

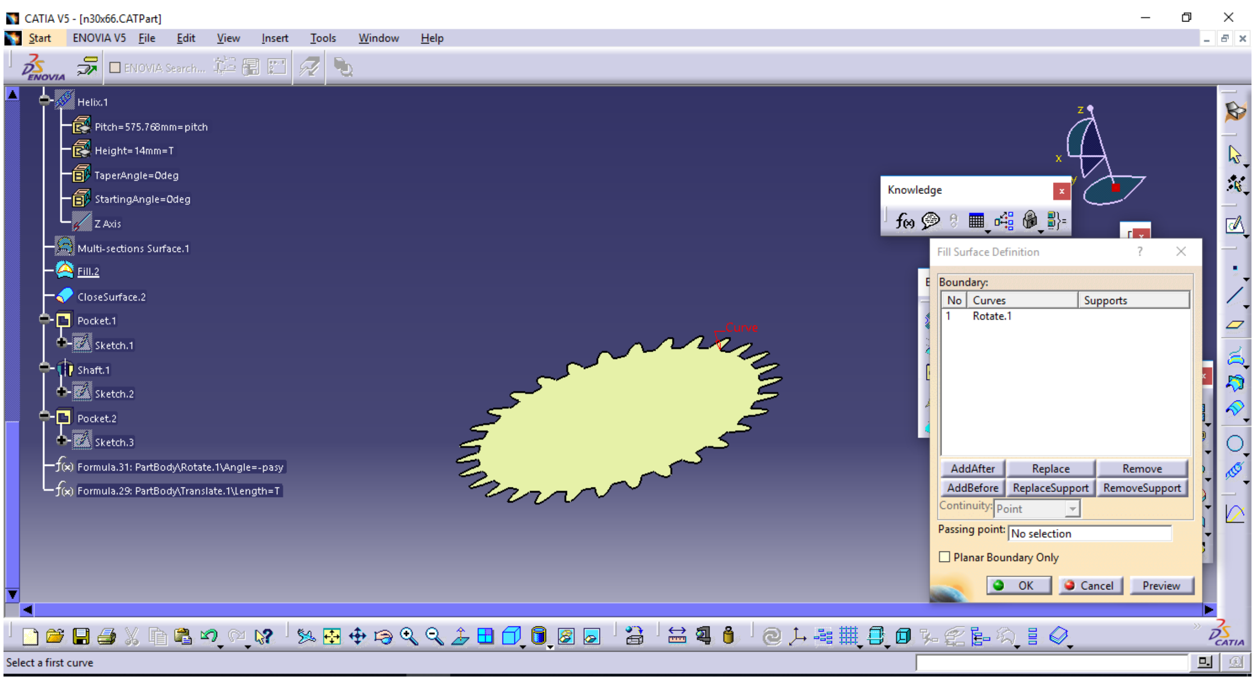


Figure 4.23 Fill surface

Figure 4.24 shows the gear surface after filling the rotated surface. Now convert the surface to part design simple by going to start menu and choosing mechanical design then part design.

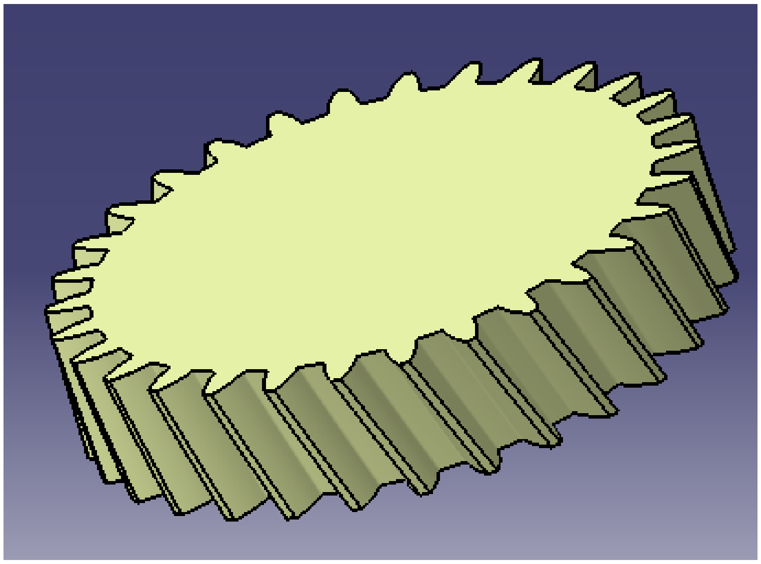


Figure 4.24 Gear surface

By using fill command in part design and choosing the surface will get the shape as in **figure 4.26**.

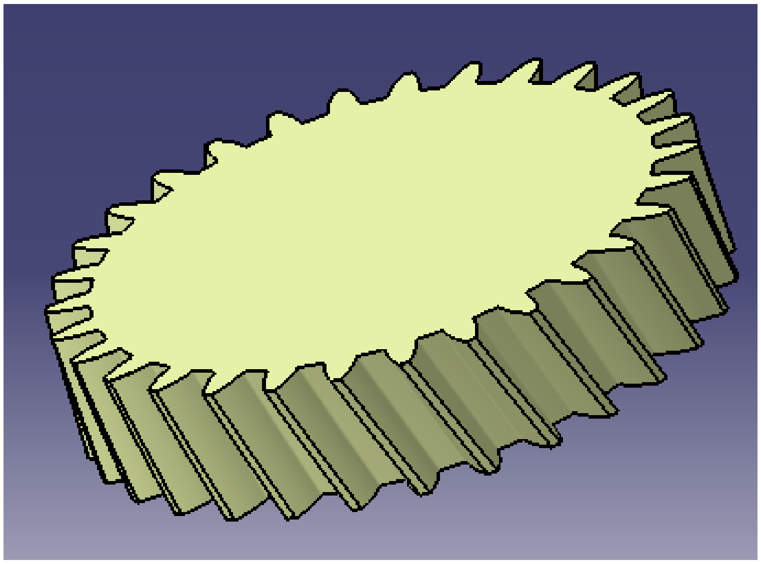


Figure 4.25 Fill the surface

After hiding the outer surface, the final form of the shape as shown in **Figure 4.26**.

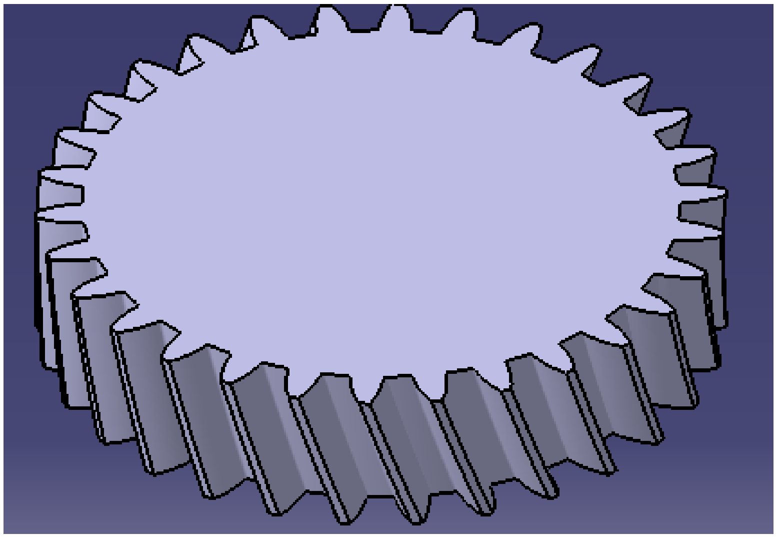


Figure 4.26 Gear model

4.1.2 Other Parts

In the gearbox, there is parts other than gears but we will only talk about three of them shafts, synchronizers and bearing

4.1.2.1 Bearing

Rolling contact bearings are used to minimize the friction associated with the relative motion between the input and output shafts of the gearbox.

The type of bearing that exist in gearbox is ball bearing and its use for keeping the distance between gears and its shape as in **Figure 4.27** its bore 28 and outer the diameter of 68 mm

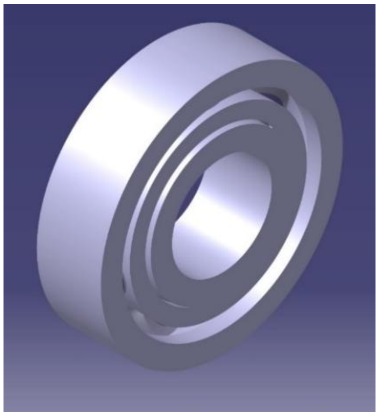


Figure 4.27 Bearing

See appendix A-1

4.1.2.2 The Shafts

In the car gearbox there is two shafts. The first one is the output shaft from gearbox (driven shaft), and the second shaft is the input shaft (driving shaft) that carry the power into the gearbox.

1. The output shaft that shaft contend of 6 gears driving by others and all of the gears are rotating together as one part and 5 of them are fixed and one removable as shown in the **Figure 4.28**.

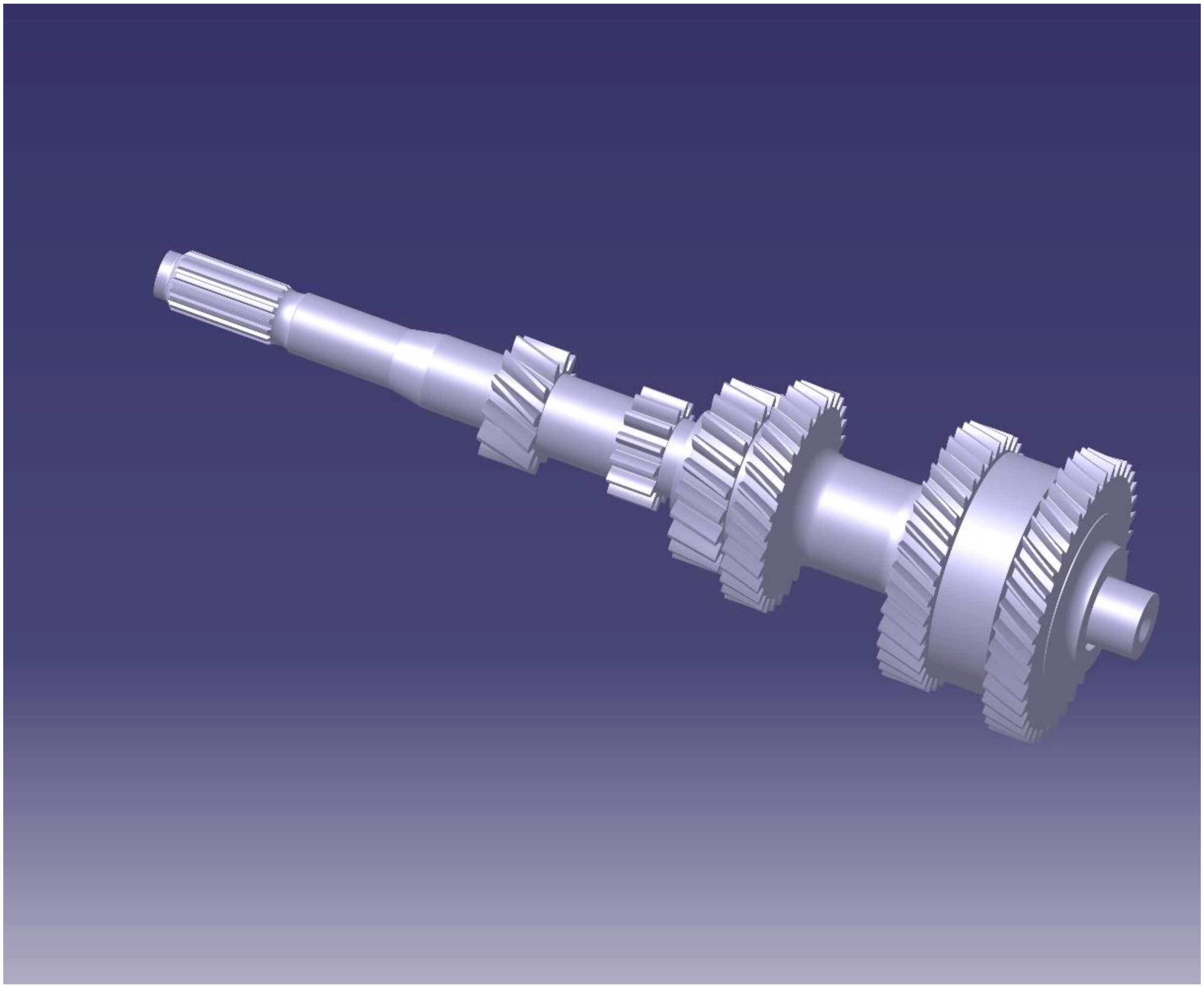


Figure 4.28 Output shaft

See appendix A-2

2. For the output shaft the gears rotate freely on the shaft but can be locked to the shaft by the synchronizer shown in Figure 4.29.

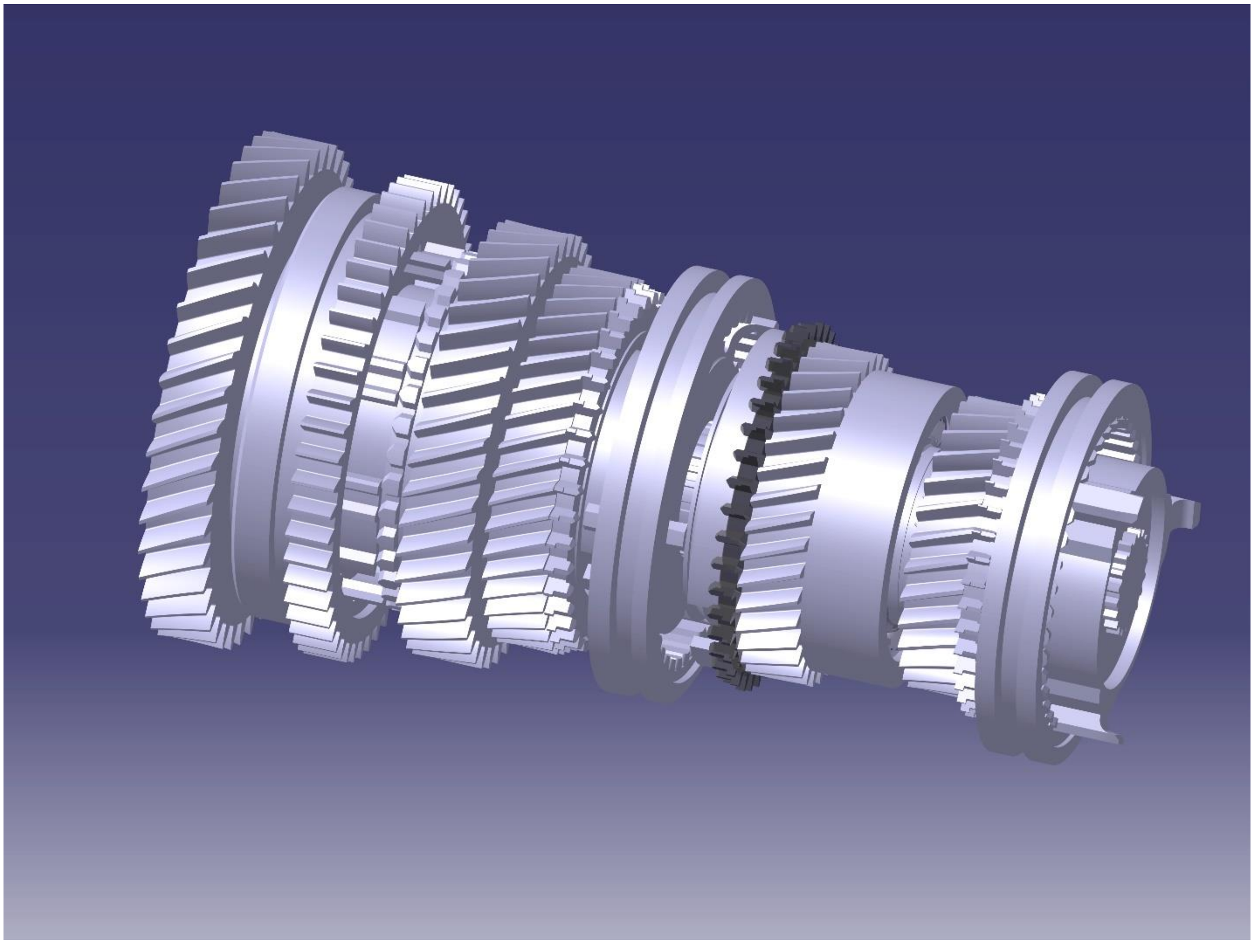


Figure 4.29 Input shafts

See appendix A-3, A-4

4.1.2.3 The Synchronizers

Synchronizers are used to connect a gear in the input shaft with the shaft that connection effect the output shift speed, there is main types of synchronizers but in our gear the type called Porsche type and it’s shown in **Figure 4.30**

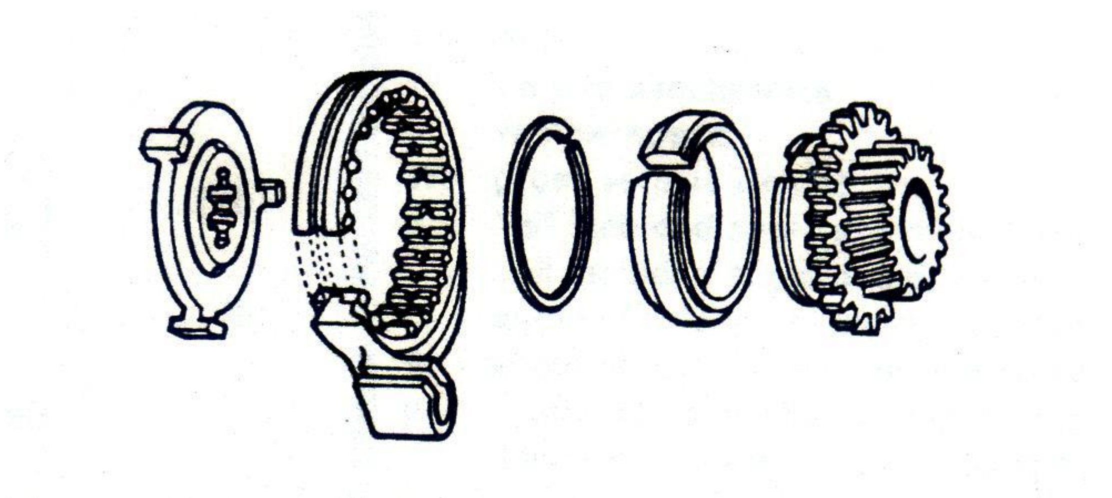


Figure 4.30 Porsche type

The first synchronizers and its gear shown in **figure 4.31** and it's only for one gear its diminutions are taken by measuring by caliber only.

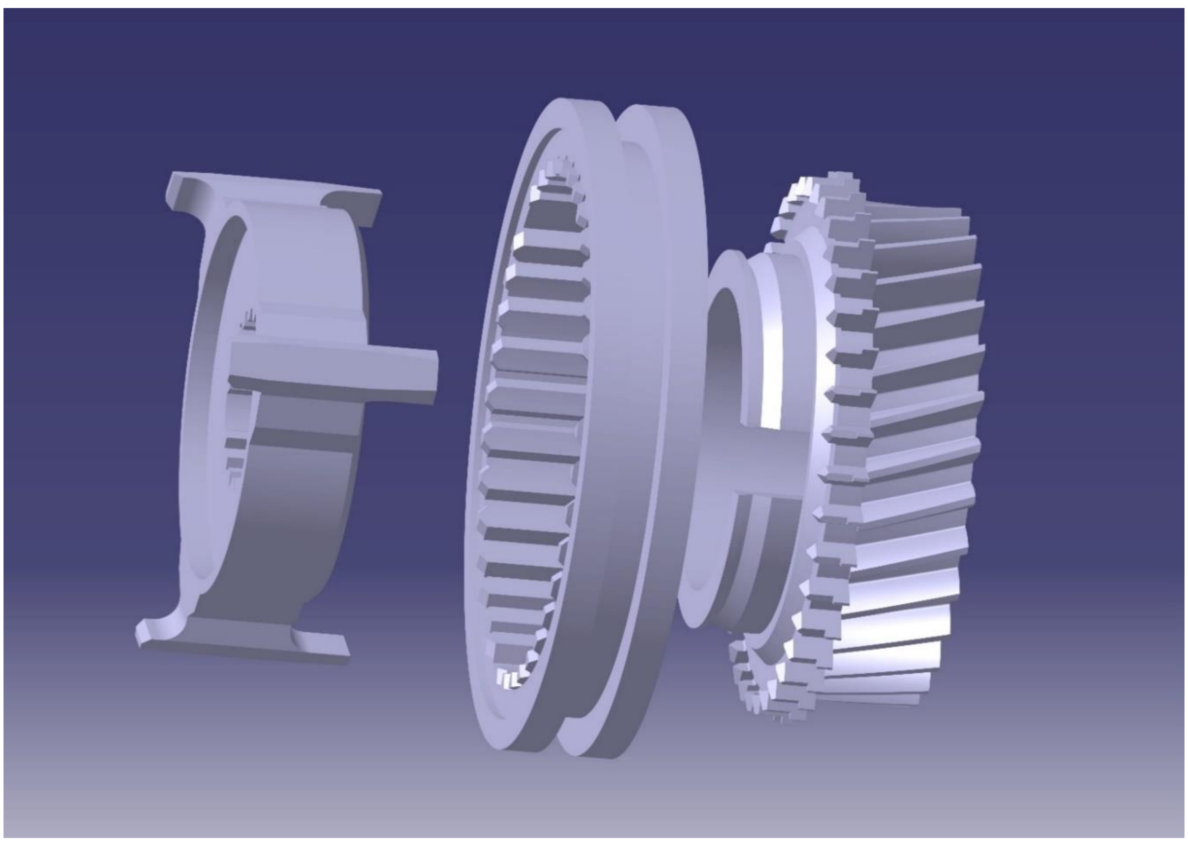


Figure 4.31 First synchronizers

See appendix A-5

The second synchronizers has the same work principle as the first but it's connect two gears and it's shown in **Figure 4.32** Second synchronizers.

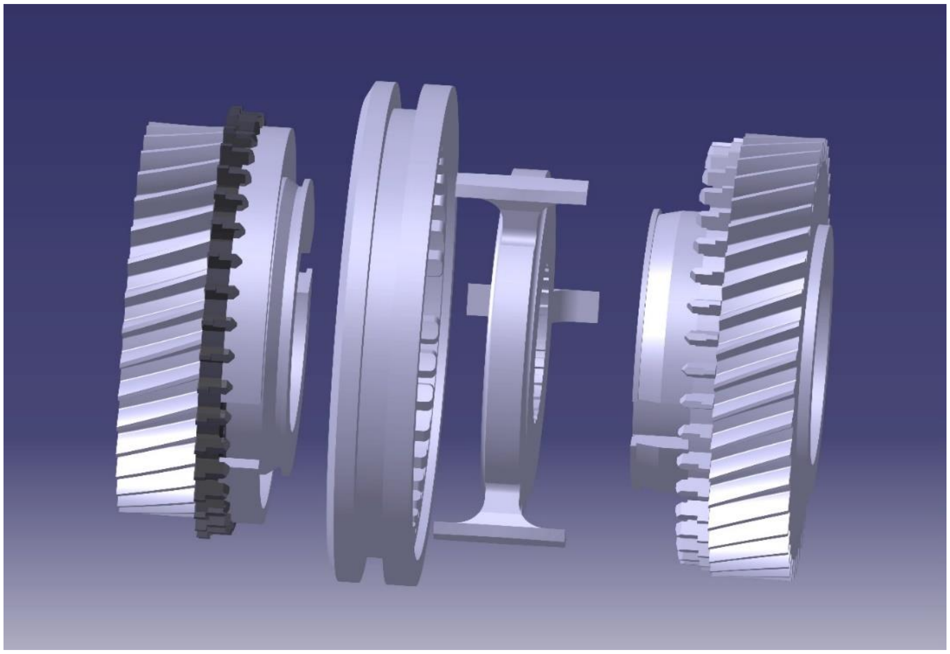


Figure 4.32 Second synchronizers

see appendix A-6

The last type is called double cone and it have different principle and it's shown in **Figure 4.33** Third synchronizers.

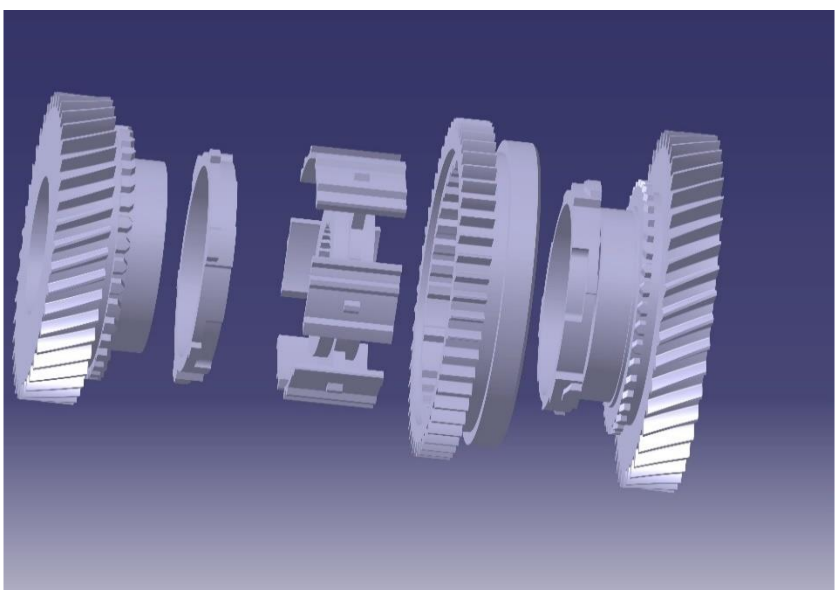


Figure 4.33 Third synchronizers

Figure 4.34 shows gearbox components after assembly.

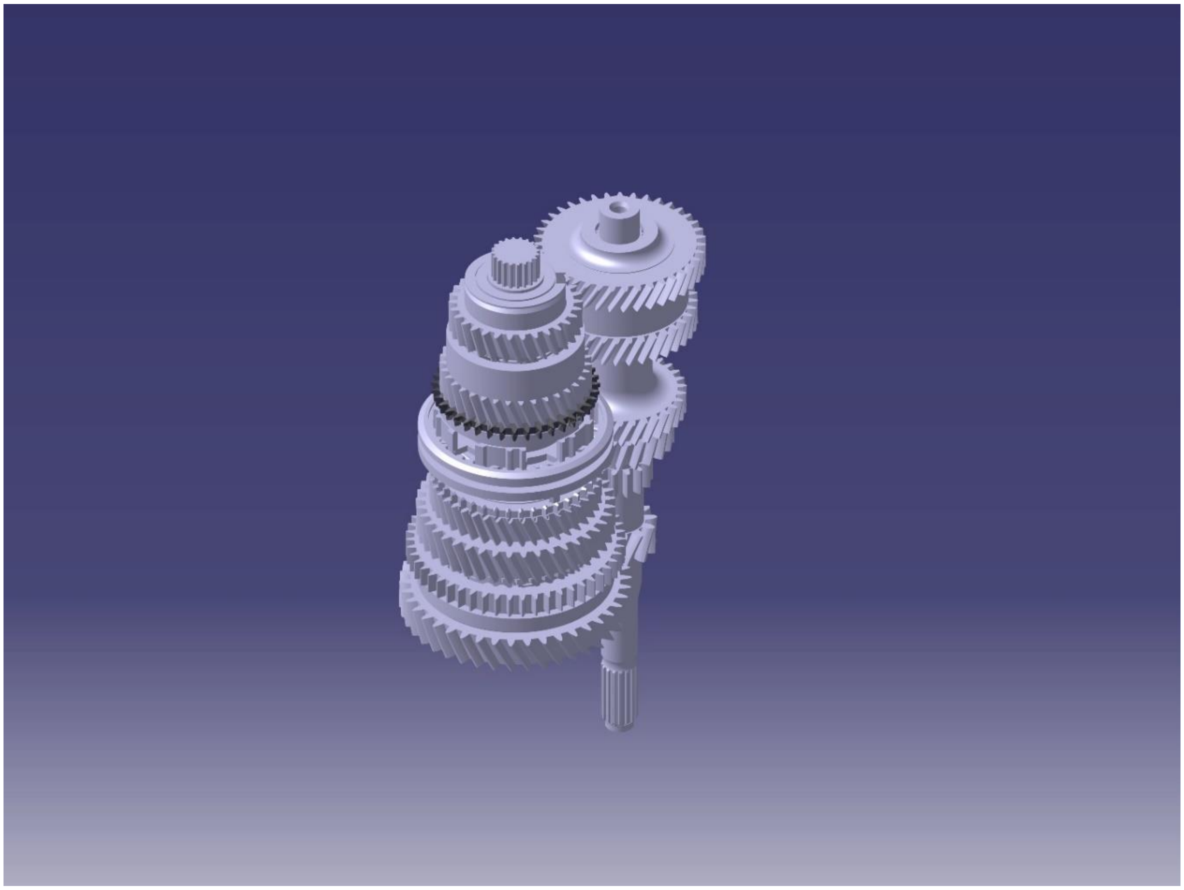


Figure 4.34 Gearbox

4.2 Processes of Geomagic Design X for an Object

The processes in Geomagic Design X depends in general on the structure of the object, some objects can easy make an auto surface for it, others need mesh sketch or 3D mesh sketch, in **figure 4.49** shows in general the processes of the program.

For all cases the process is starting after obtaining a scanning data of sample, first step starts by import the scanning file data as in **Figure 4.35**, usually the type of this files is ".stl or .obj ", also these files are in mesh data, as shown in **Figure 4.36**.

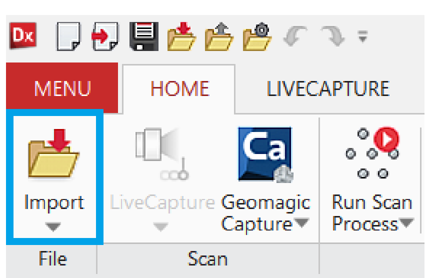


Figure 4.35 Import scanned data file

This chapter will discuss the Auto Surface and mesh sketch because this two process more useful than other processes.

4.2.1 Auto Surface Process

At the beginning, the import file has some disturbances that in the sample and around it, that is because some noises that effect on the original object through the scanning process and it acts on the sensors of scanner, so first step is removing this disturbance.

Figure 4.36 Meshed imported data

4.2.1.1 Optimize the Meshing Data by Mesh Mode

This process will start by select the object from the feature tree, then applying Mesh mode, **Table 4.1** shows an important tools icon that used in this mode to obtain a specific surface

Table 4.1 Tools in mesh phase

Symbol	Tool Name
	Healing Wizard
	Edit Boundaries
	Smooth
	Smart Brush
	Defeature
	Optimize Mesh
	Global Remesh

By using healing wizard, the object automatically detects various defects in the mesh such as Non-manifold poly vertices, fold, small and crossing faces, small clusters and small tunnels. So, can be obtained mesh model without any abnormal poly-faces, **Figure 4.37** Part after healing wizard. shows the part after healing wizard.

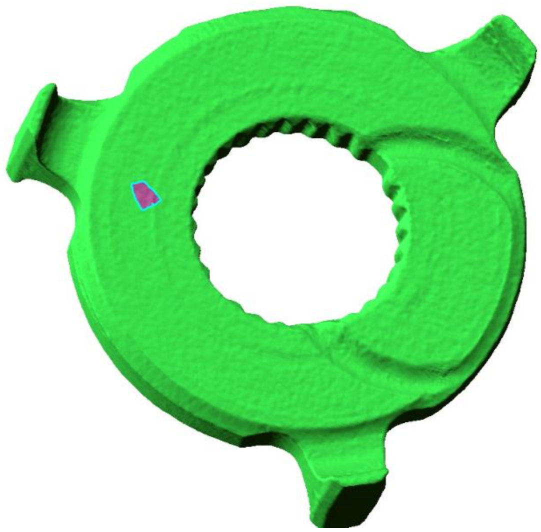


Figure 4.37 Part after healing wizard.

As shown in Figure 4.37 Part after healing wizard. there is a gap in sole, Edit Boundary tool can help to fill this gap also it can reduce the effect of noise and roughness. **Figure 4.38** shows the result after using Edit Boundary command.

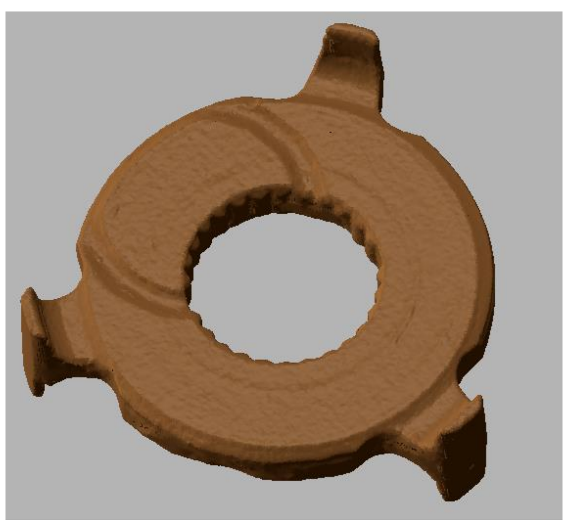
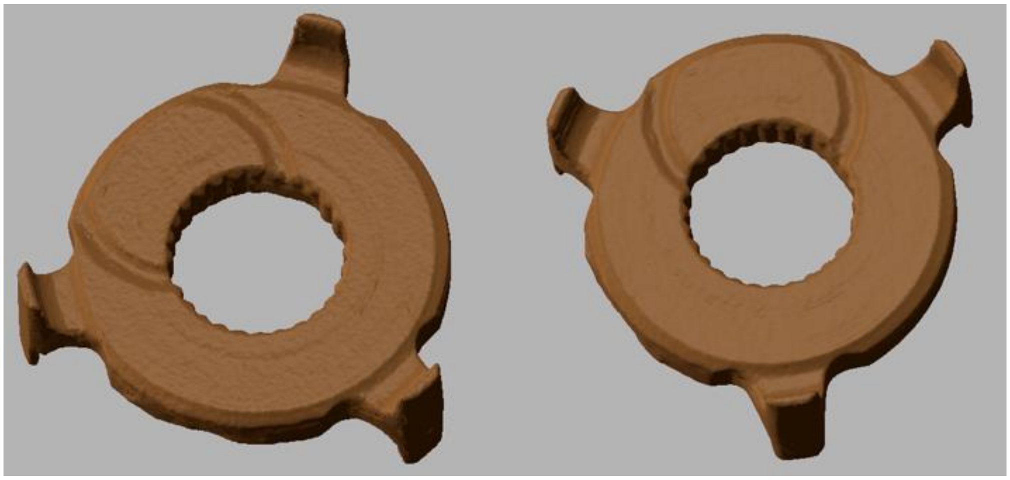


Figure 4.38 After applying Edit Boundary

As shown in **Figure 4.39** (a), mesh seems so rough and contains some of noises, Smooth tool removes spiky points by averaging with surrounding points and improving the quality of a mesh



(a) (b)

Figure 4.39 Before and after Applying smooth sommand

After applying these tools on 3D scanning data, Global Remesh can regenerates poly-faces with uniform poly-edge lengths on a mesh and improves the quality of the mesh. **Figure 4.40** show result after use Global Remesh command

Figure 4.40 After global remesh Command

Global Remesh command is useful for [11]:

1) Optimizing mesh so that it has poly-faces globally with uniform poly-edge lengths.
2) Optimizing mesh to be used for creating a high-quality surface body.
3) Reducing the effects of defects and roughness on a mesh.
4) Creating a high-quality mesh from combined mesh that is complex and overlapped.

The finishing from Mesh mode can be after applying two commands that provides a perfect mesh surface. First command is **Defeature**, which removing feature shapes from a target mesh and removing unnecessary feature shapes. Second is **Optimize Mesh** command, which improves the quality of the mesh.

This commands can be applied more than one time to achieve best quality of surface mesh, **Figure 4.41** show the final shape in Mesh Mode that can be obtained, the surface is smooth and prefect also all holes are filled.

Figure 4.41 Final shape of optimizing mesh phase

4.2.1.2 Auto Surfacing

Auto surfacing technology that automatically converts point clouds into surface model, during this command, scanning data will convert to surface body smooth and quickly. This tool uses when the shape is complex freeform part, it can create a surface body that can envelope an entire geometric shape of a specific mesh, **Figure 4.42** shows the form of helical gear after applying auto surface.

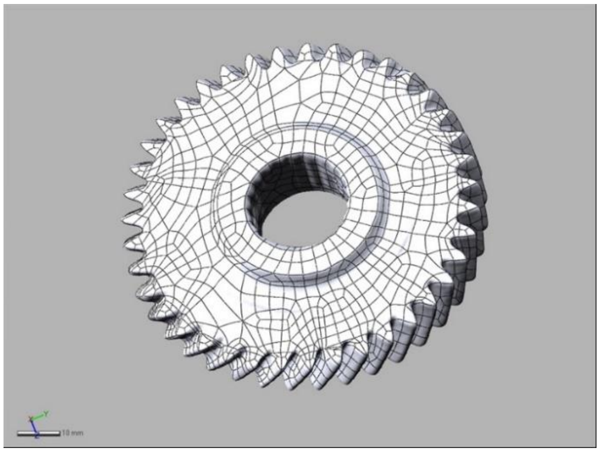


Figure 4.42 Part surface

4.2.1.3 Export File

Through this step can export the object to the CATIA or SOLIDWORKS into Many multi-platform formats are offered, in order to enhancing if required. This command can be done by selecting the command Export from File tape, select the object, press OK, and save it in a specific folder as shown in **Figure 4.43** and select an acceptable type format, that can other software deal with it, for CATIA V5 R17 for example, the suitable format is STEP file " .stp ".

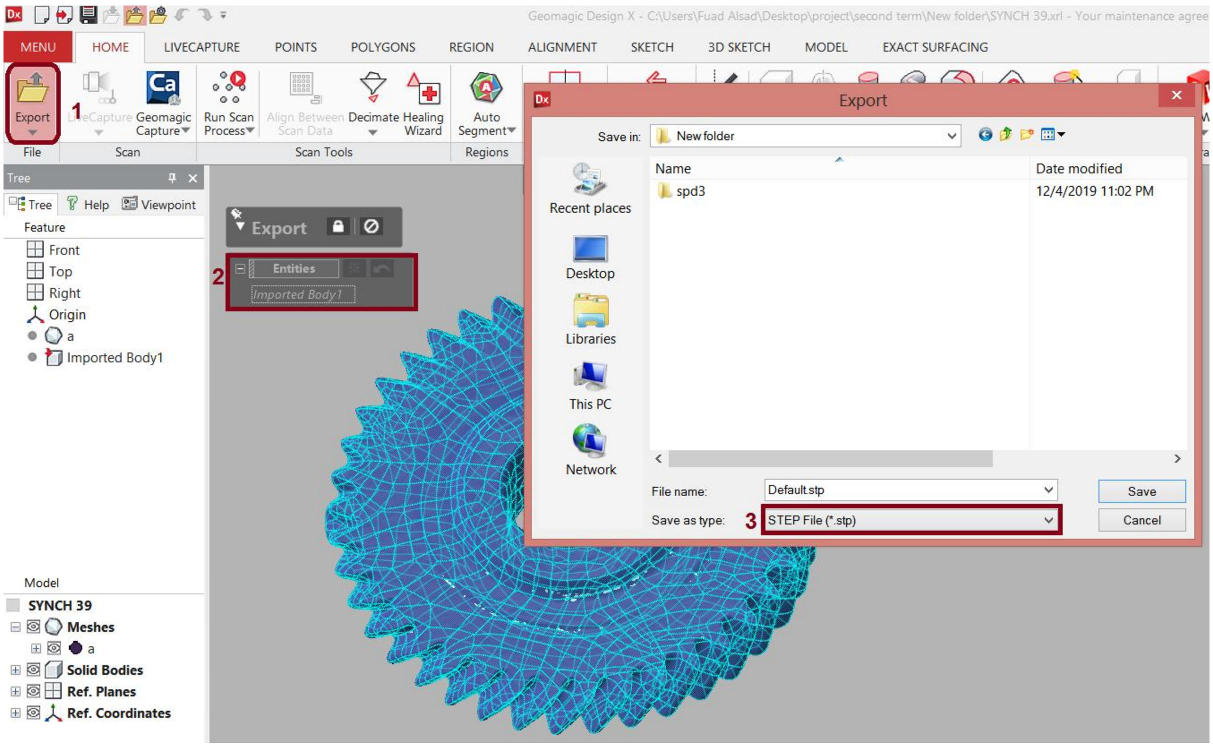


Figure 4.43 steps in Geomagic to export an object

4.2.2 Mesh Sketch Process

The mesh sketch tool provides an easy way to extract cross section profiles from a scanned geometry. These extracted cross sections become sketch profiles in which the user can dimension and select to model solid or surface features (extrusion, revolve, loft, sweep, etc.) Simply select the reference region or plane, an offset value, and direction and the cross section profile is automatically generated.

4.2.2.1 Appling Region Group

Region Group features contains various commands used to generate and edit feature regions on a mesh. It can automatically classify feature regions by recognizing 3D scanning features and detect it quickly, by using geometric feature information for easily and quickly created features. For example, it can detect, define and classify the regions that are a cylinder or plane, **Figure 4.44** shows that each color refers to specific region and that depend on the geometry.

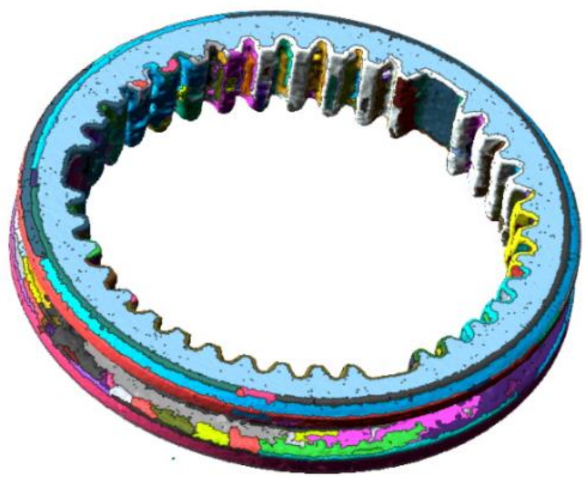


Figure 4.44 Region group

4.2.2.2 Reference Geometry

The Ref. Geometry menu features various commands used to generate primitive geometry and edit other entities. The description of reference geometries are for normal situations. In some situations, the geometries can be used for different purposes such as plane, vectors and points as shown in **Figure 4.45**.

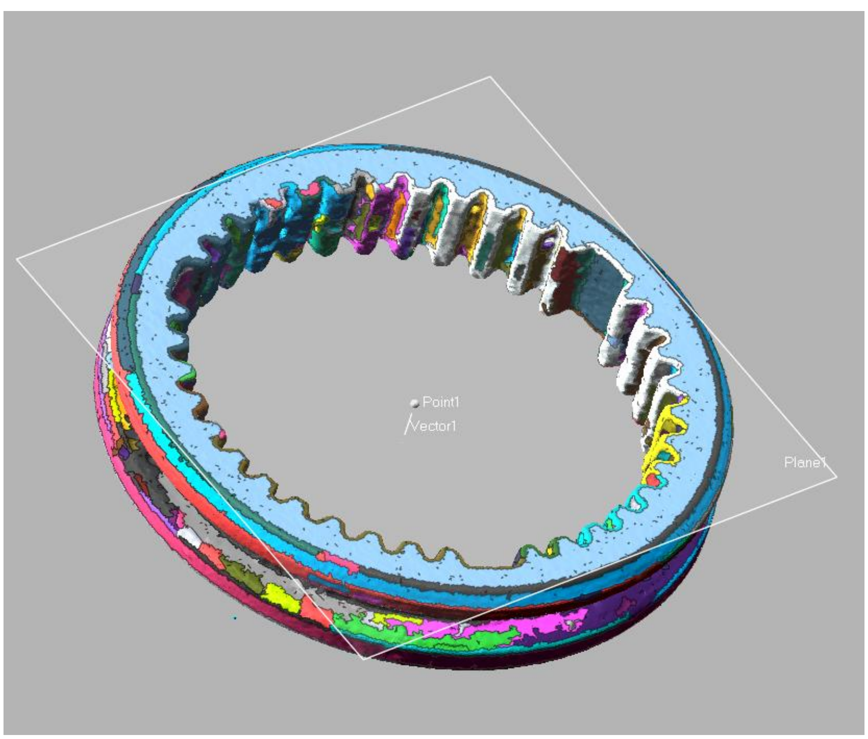


Figure 4.45 Reference geometry

4.2.2.3 CAD Modeling

The main approaches to creating a CAD object from scan data is parametric modeling. Parametric modeling involves creating sketches and features as shown in **Figure 4.46**. Parametric modeling is similar to modeling in most CAD software packages, but the mesh used as a reference for extracting geometry.

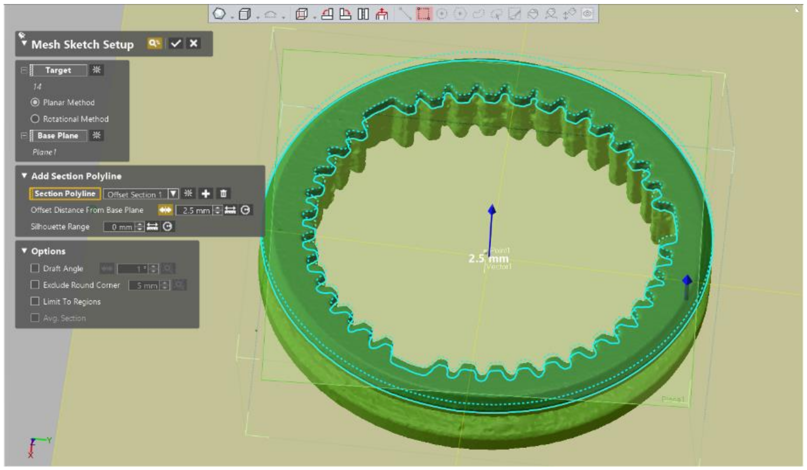


Figure 4.46 Mesh sketch

It is also not necessary to completely finish the model within Geomagic Design X if the user would prefer to use a different CAD software application for the parametric modeling process. The user can first extract and model difficult geometry from the mesh then use Live Transfer to move the feature tree into their preferred CAD application. **Figure 4.47** show CAD model using Geomagic design X.

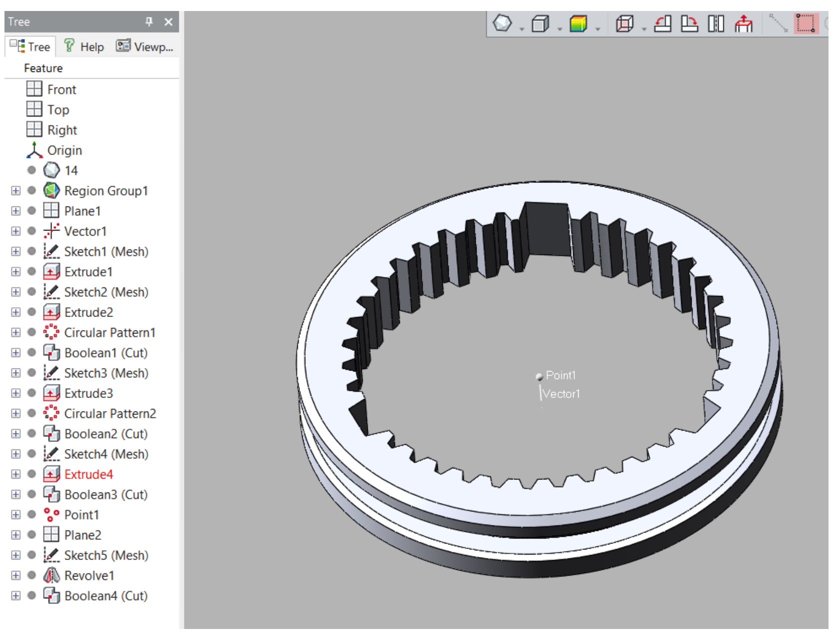


Figure 4.47 CAD model

4.2.2.4 Deviation Analysis

This tool will provide a heat map as shown in **Figure 4.48** that displays the deviation value between the 3D scan data and the parametric CAD model bodies. As a part is converted to a solid through the reverse engineering, deviation analysis provides an easy graphical interface that details which features are most accurate.

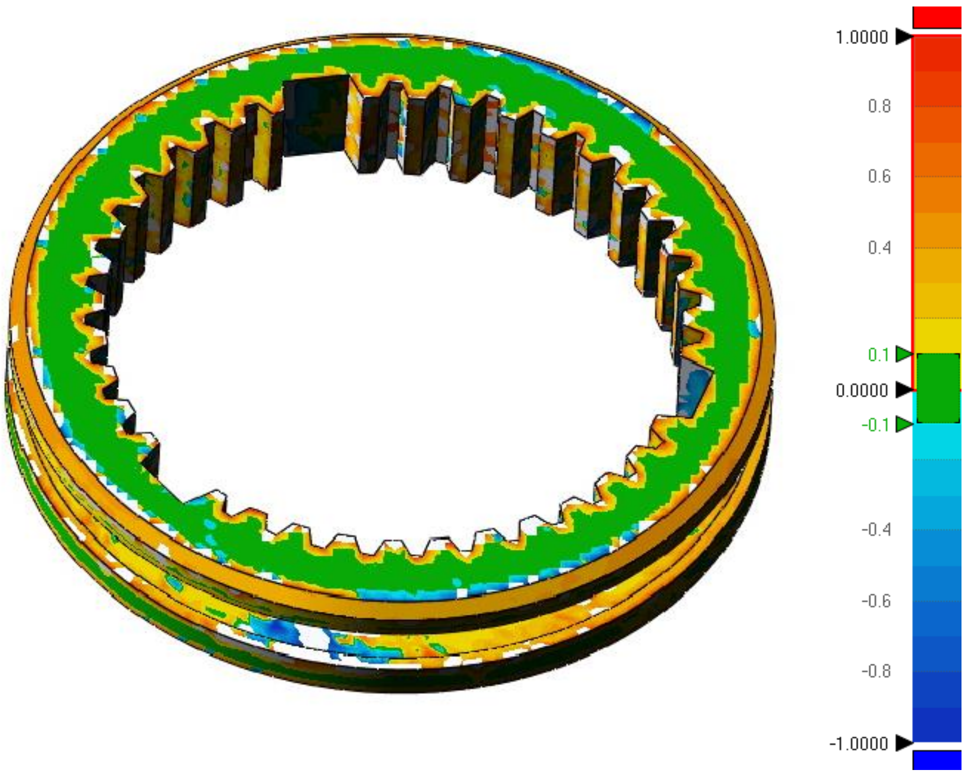


Figure 4.48 Deviation Analysis

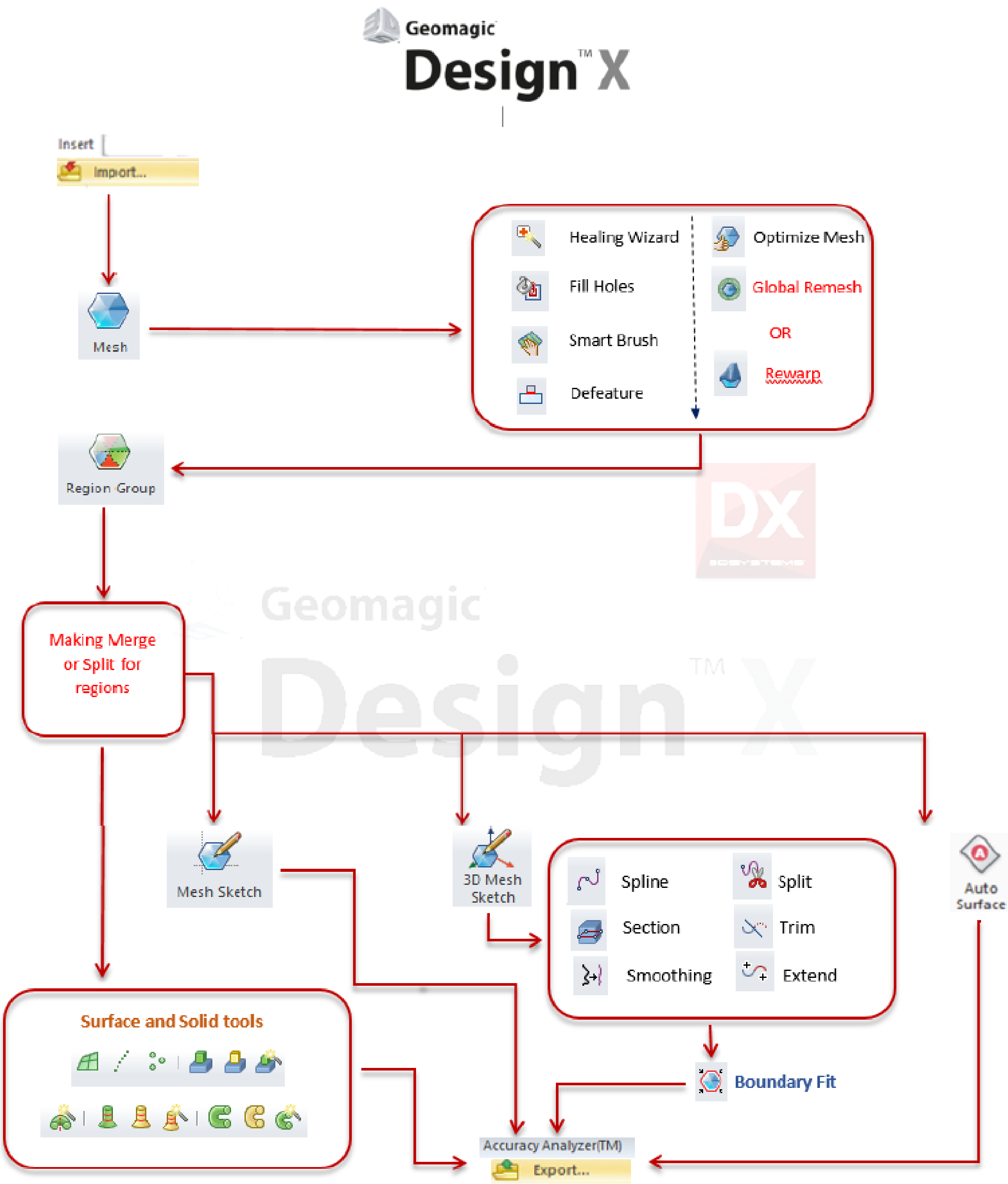


Figure 4.49 Processes in Geomagic Design X

5

Chapter Five
Results

5.3 Results

5.2 Conclusion

5.3 Recommendations

5.1 Results

This Section includes some of parts that are working and optimizing it by using Geomagic Design X, all these objects are using the same procedure that describe previously in **section 4.2**, but some of it required more time to achieve the best result.

- **Gear model**

By comparing between 3D parametric model shown in **figure 5.1** and real model from 3D scanner for the same gear which shown in **figure 5.2** by using accuracy analyzer in Geomagic the results as shown in **figure 5.3** show that both model are similar to each other.

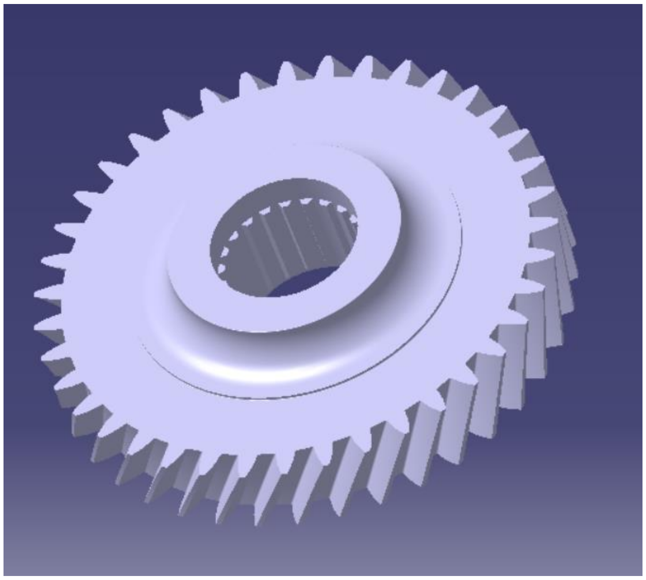


Figure 5.1 3D Parametric model

Figure 5.2 Scanning Data

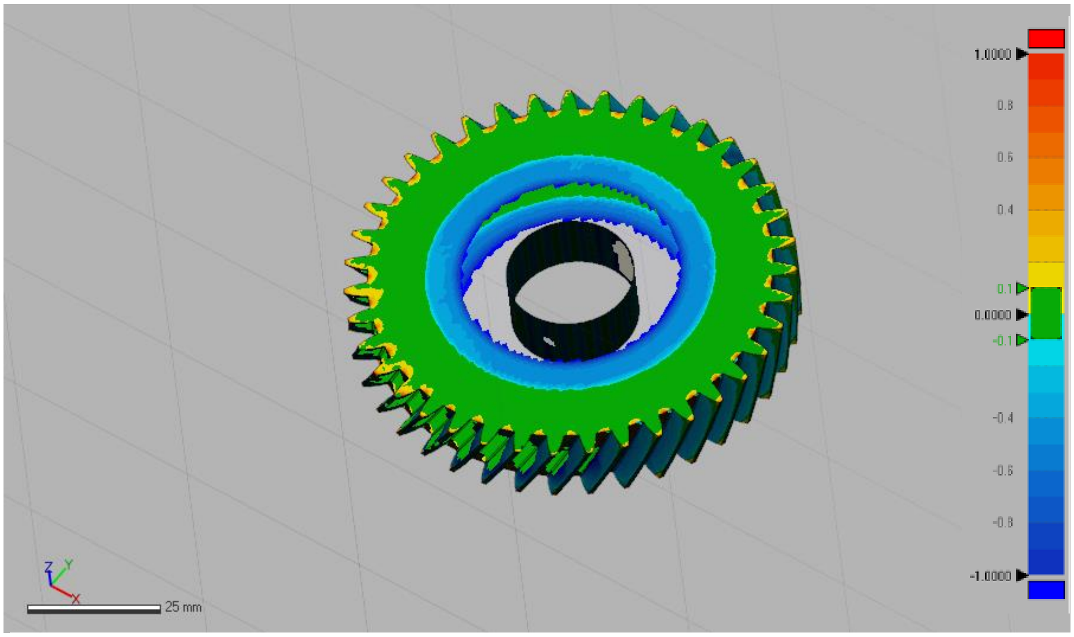


Figure 5.3 Accuracy analyser for gear

- **Synchronizers**

Accuracy analyser done for first Synchronizer part, the CAD model of the part shown in **Figure 5.4** drawn by using calliper to measure its dimensions, and compare it to scanned shown in **Figure 5.5**, the colours in **Figure 5.6** shows the difference between CAD model and real model [see Appendix A-7].

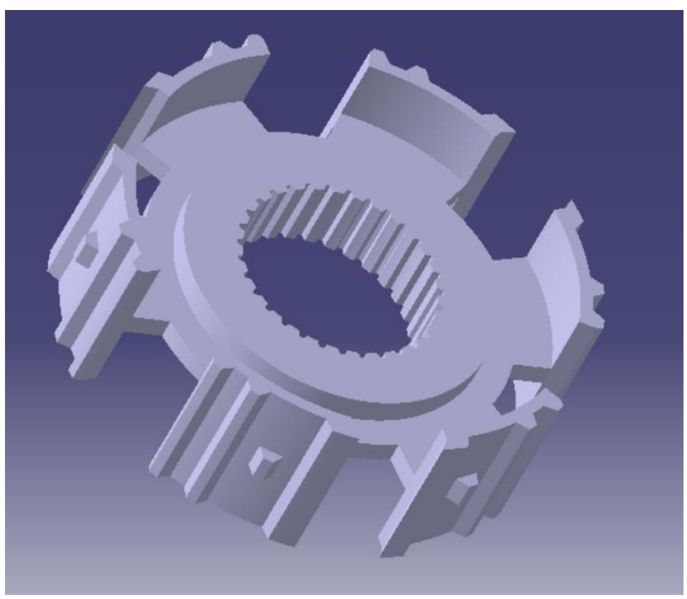


Figure 5.4 3D Synchronizer model

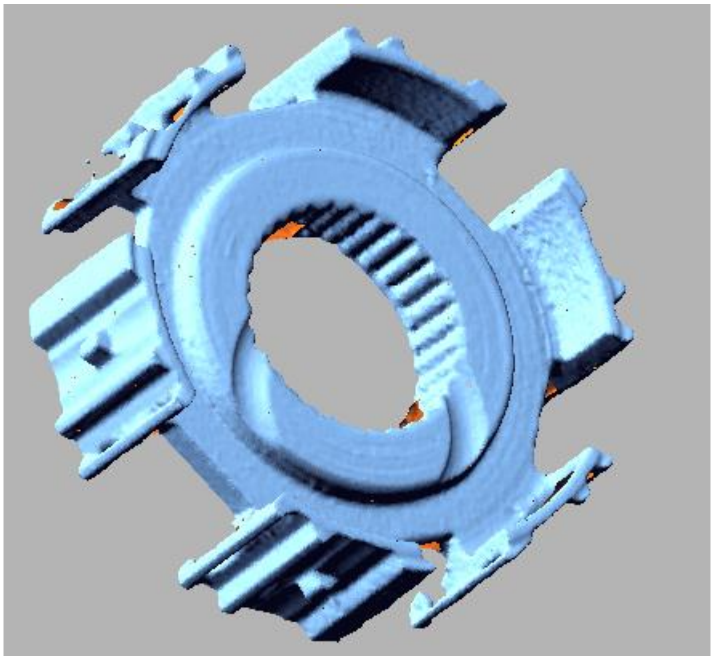


Figure 5.5 Scanning Data

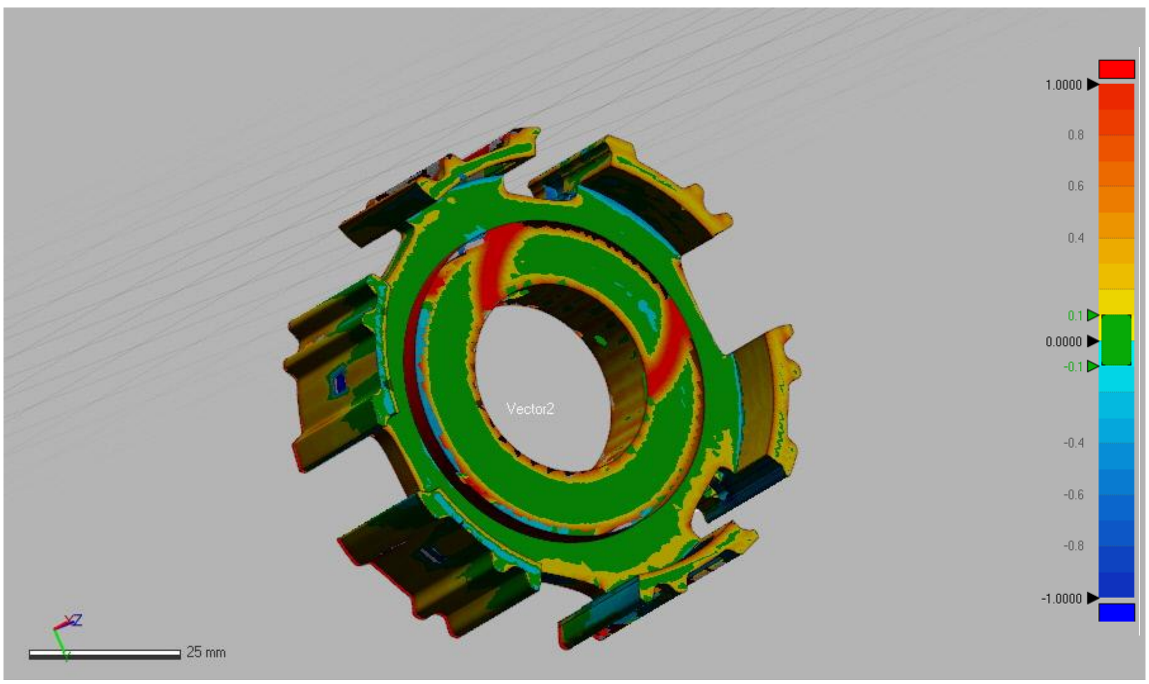


Figure 5.6 Accuracy analyser for synchronizer

Accuracy analyser done for second Synchronizer part, the CAD model in **figure 5.7** that build based on scanning data shown in **figure 5.8** for the real model. **Figure 5.9** shows the difference between CAD model and real model.

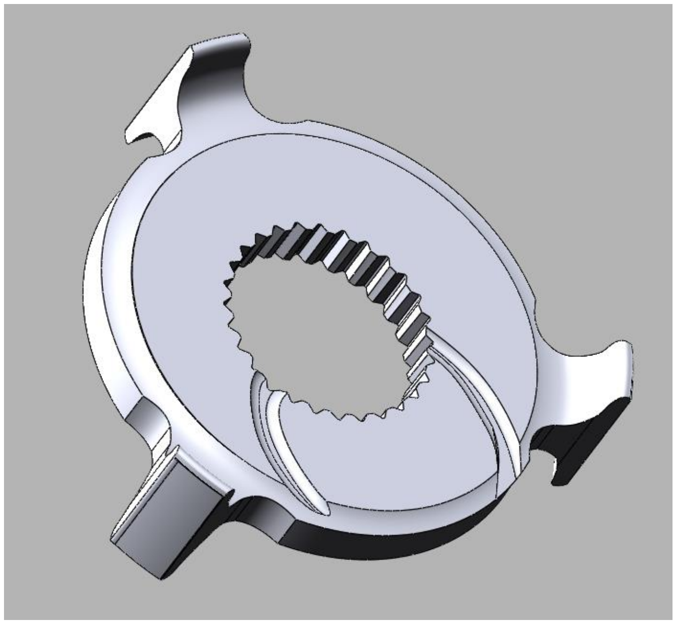


Figure 5.7 CAD model for Synchronizer

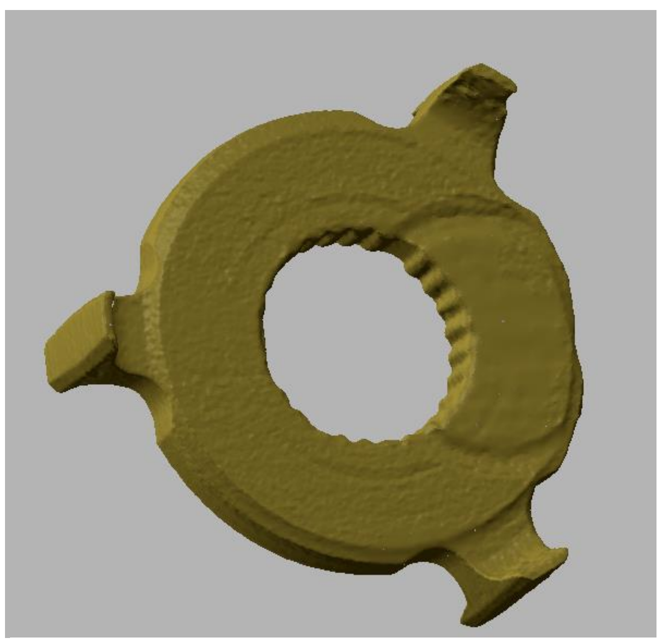


Figure 5.8 Scanning data

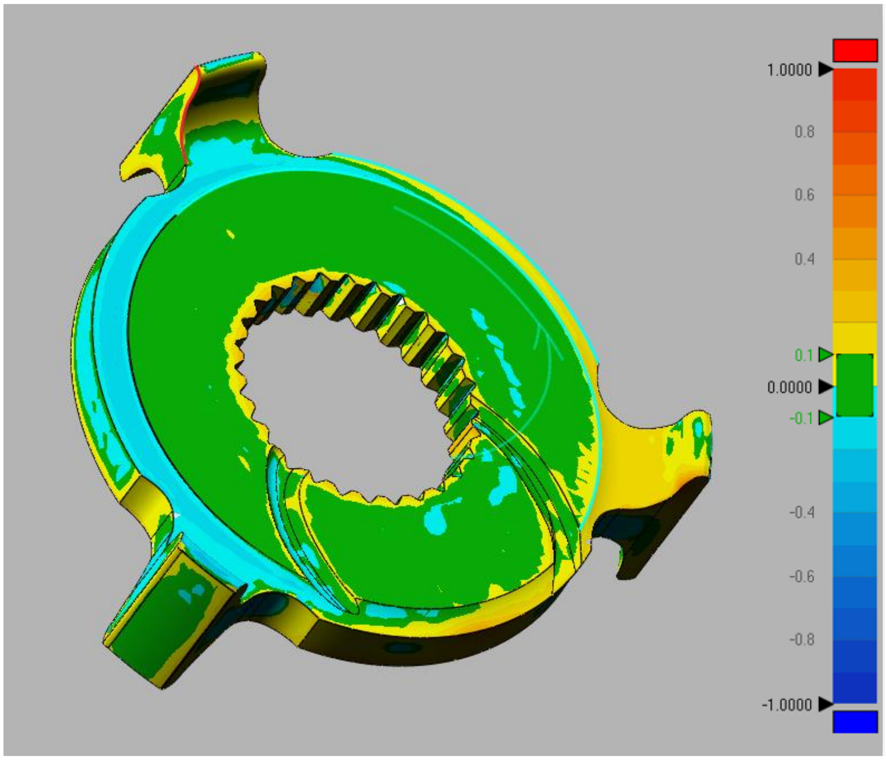


Figure 5.9 Accuracy analyser for synchronizer

Figure 5.10 shows the CAD model for third synchronizer that build based on real model for it, **figure 5.11** show scanning data that used to build the CAD model for this synchronizer, by using accuracy analyser on Geomagic Design X **figure 5.12** shows the different between the two models.

Figure 5.10 CAD model for synchronizer

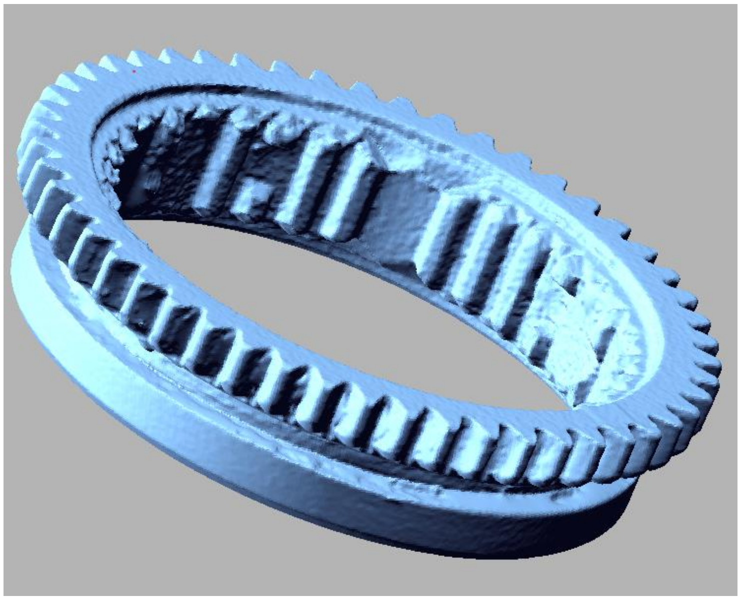


Figure 5.11 Scanning Data

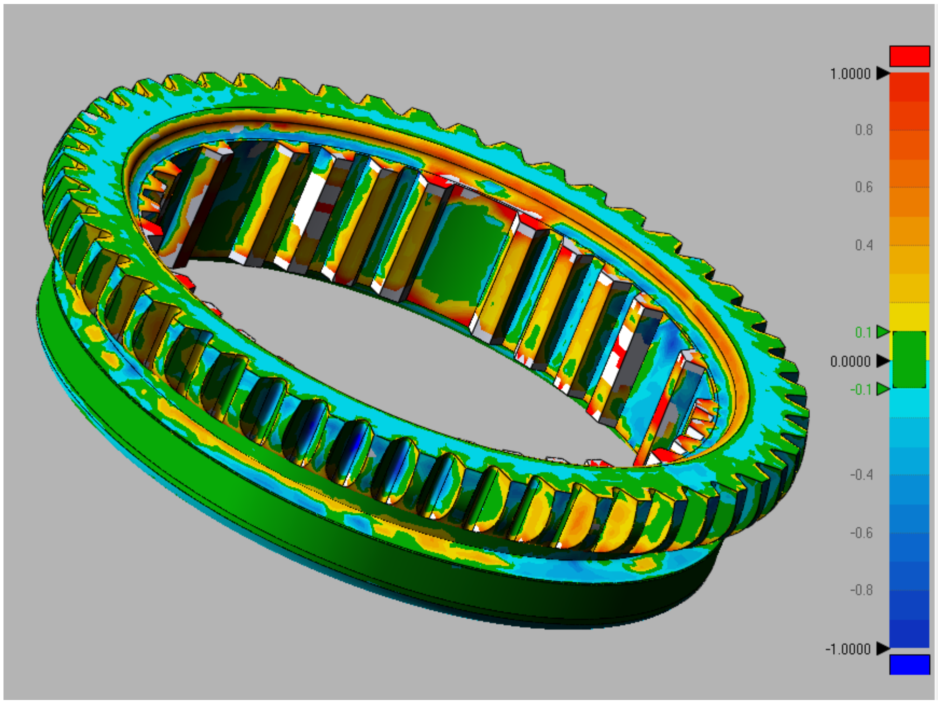


Figure 5.12 Accuracy analyser for synchronizer

5.2 Conclusion

The 3D CAD model can be developed by using CATIA parametric molding which can be consider as a library or templets for gear drawing, so if anyone want to build gear mesh the parametric model can be use with some change in the required parameters such as the outer diameter and number of teeth.

Geomagic design X is a useful tool to find out solid model from point cloud data that obtained by 3D scanner. Tolerances can be found by comparing between the real model and the CAD model for an object and this one of the special features in Geomagic Design X program.

Reverse engineering techniques useful and suitable for complex part and products with unknown or hidden data such as synchronizer in our case, but useless for parts with standard data like gears and bearings.

5.3 Recommendations:

Some recommendations and inferences should be recall during the problems that has been faced in this project:

1. The output quality of objects, which export from the Geomagic Design X depends on the scanning quality. Some parts cannot healing it also cannot making an auto surface or mesh sketch, because the boundaries of object cannot help when applying healing wizard or any process come after this process, also when applying healing wizard or other commands, which use for repairing the sketch and it cannot repair it, the problems will be twice or treble in region group and other processes.

2. The work in this project focuses just on the Geomagic Design X and CATIA, it has been dispensed about Geomagic Studio, because Geomagic Design X has more power, more feature tools and it can export to various formulas, Geomagic Studio works just for simple parts, but in Design X have more powerful and can deal with all objects.

3. Through the various works in this project on various parts, it takes a long time for the processes in programs, for example in the sole Figure 4.37 takes at healing wizard about fifteen minute to finish it, so this work requires a high-performance computers processors and graphics, in order to achieve a good work in short time.

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Appendix