A

Dissertation Report

on

AUTOMATION USING CAD FOR GEAR, SPROCKET AND SPLINES HOB CUTTER MANUFACTURE

Submitted in partial fulfillment of the requirement for the award of degree of

MASTER OF ENGINEERING
IN
PRODUCTION & INDUSTRIAL ENGG

Submitted by:
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JULY- 2013

DECLARATION

I hereby declare that the Dissertation Report entitled "**Automation using CAD for gear, sprocket and splines hob cutter manufacture**" submitted towards partial fulfillment of requirement for award of **Master of Engineering** degree in **Production & Industrial Engineering** in **Mechanical Engineering Department** of **Thapar University, Patiala,** is an authentic record of work carried out by me under the supervision and guidance of **Mr. A.S. JAWANDA, Associate Professor, Mechanical Engineering Department, Thapar University, Patiala.**

This matter embodied in this report has not been submitted in part or full to any other university or institute for the award of any degree.

Dated 15/07/2013

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(801182007)

This is to certify that above declaration made by the student concerned is correct to the best of my knowledge & belief.

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ABSTRACT

Gears are widely used product in manufacture of machines. Their application is everywhere we need to transmit power with high efficiency from one point to other. The most common methods for gear cutting are hobbing, milling, rack cutting.

The latest trend of the high scale industries is the knowledge base design automation. Using the CAD Automation any user can generate desired gear profile more accurately through the automated design software by defining the proper relation between the parameters.

This automation is attempt to do elimination of the manual drawing along with the conventional machines for generation of gear to cutter profiles used by the optical profile grinder if the profile matches then grind the form tool cutter profile using which the hob cutter will be made used in manufacturing tool industry.

Thus the industry requires automation in generation of gear to cutter profile which is normally done manually. Incorporation of parametric CAD softwares is used for the generation of gear to cutter profile with relating the various parameters of gear terminology. This technique gives high accuracy and possible to produce high quality cutters.

Further this CAD software automation is attempt to make the macros for generating the required profiles i.e cutter profiles and tracing them with the use of API (Application Program Interface) for the actual use in manufacturing process.

Work has been done on various kind of profiles like gear cutter profiles, sprocket cutter profiles, spline cutter profiles, chamfered spline cutter profiles and rack to gear profile generation.

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CHAPTER-1

INTRODUCTION

1.1 Introduction

Gears are toothed wheels used to transmitting motion and power from one shaft to another. In comparison with belt, chain drives and friction drives, gear drives are more compact, can operate at high speeds and can be used where precise timing is required.

1.2 Gear manufacturing

Gear manufacturing can be divided into two categories namely forming and machining.

1.2.1 Forming Gear teeth

Forming consists of direct casting, molding, drawing or extrusion of tooth forms in molten, in powdered, or heat softened materials and machining involves roughing and finishing operations.

In all tooth-forming operations, the teeth on the gear are formed all at once from a mold or die into which the tooth shapes have been machined. The accuracy of the teeth is entirely dependent on the quality of the die or mold and in general is much less than that can be obtained from roughing or finishing methods. The various forming techniques are Sand casting, die casting are the casting processes that are best suited for gears.

Sand casting

Sand casting is used for gear manufacture which are used in variety of applications such as for toys, small appliances, hand operated crane etc. The materials that can be sand cast are C I, cast steel, bronzes, brass and ceramics.

Die cast

Gears that are die cast are used in instruments, cameras, washing machines, gear pumps, small speed reducers, and lawn movers. Materials used to manufacture these gears are zinc, aluminum and brass. The gears made from this process are not used for high speeds and heavy tooth loading.

Injection molding

Injection molding is used to make nonmetallic gears in various thermoplastics such as nylon and acetal. These are low precision gears in small sizes but have the advantages of low cost and the ability to be run without lubricant at light loads. Injection molded gears are used in cameras, projectors, wind shield wipers, speedometer. The materials for injection molding components are Nylon, cellulose acetate, polystyrene, polyimide, phenolics.

Extruding

Extruding is used to form teeth on long rods, which are then cut into usable lengths and machined for bores and keyways etc. Nonferrous materials such as aluminum and copper alloys are commonly extruded rather than steels. This result in good surface finishes with clean edges and pore free dense structure with higher strength. Aluminum, copper, naval brass, architect-ural bronze and phosphor bronze are the materials that are commonly extruded.

Cold drawing

Cold drawing forms teeth on steel rods by drawing them through hardened dies. The cold working increases strength and reduces ductility. The rods are then cut into usable lengths and machined for bores and keyways, etc.

1.2.2 Machining

The bulk of power transmitting metal gears of machinery are produced by machining process. For classification of machining processes. Roughing processes include milling the tooth shape with formed cutters a shaping cutter or a hob cutter.

Roughing Process

Roughing process consists of forming, generation, shaping and hobbing processes. By this method gears are made to an accuracy which is more than adequate for the slow speed operations. Forming is sub-divided into milling by disc cutters and milling by end mill cutter which are having the shape of tooth space.

Form milling by disc cutter

The disc cutter shape conforms to the gear tooth space. Each gear needs a separate cutter. However, with 8 to 10 standard cutters, gears from 12 to 120 teeth can be cut with fair accuracy. Tooth is cut one by one by plunging the rotating cutter into the blank as shown in Fig.1.1.

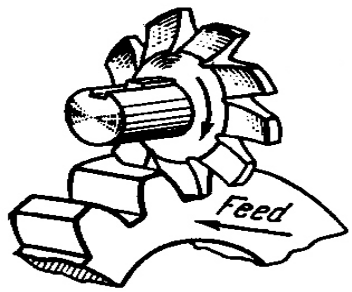

Figure 1-1.Form milling by disc cutter [11]

Form milling by end mill cutter

The end mill cutter shape conforms to tooth spacing. Each tooth is cut at a time and then indexed for next tooth space for cutting. A set of 10 cutters will do for 12 to 120 teeth gears. It is suited for a small volume production of low precision gears. The form milling by end mill cutter is shown in Fig.1.2.

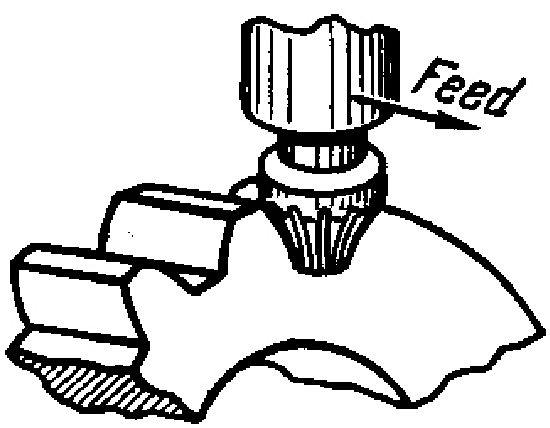

Figure 1-2.Form milling by end mill cutter [11]

Rack generation

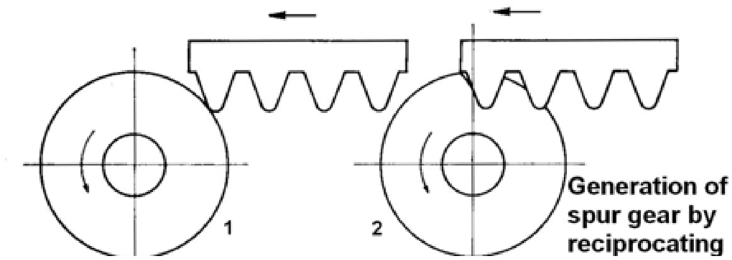

Figure 1-3 Generation of spur gear by planning [11]

Hobbing

Hob teeth are shaped to match the tooth space and are interrupted with grooves to provide cutting surfaces. It rotates about an axis normal to that of the gear blank, cutting into the rotating blank to generate the teeth as shown in Fig. 1.4.

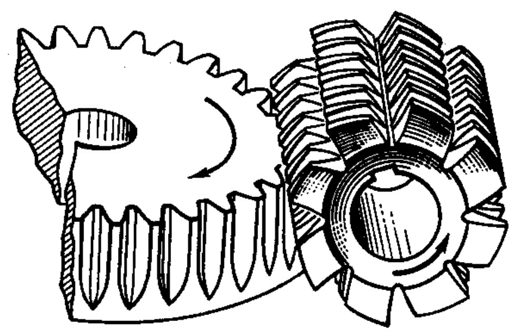

Figure 1-4 Hobbing [11]

It is the most accurate of the roughing processes since no repositioning of tool or blank is required and each tooth is cut by multiple hob-teeth, averaging out any tool errors. Excellent surface finish is achieved by this method and it is widely used for production of gears.

Finishing process

When high precision is required secondary operation can be performed to gears made by any of the above roughing methods. Finishing operations typically removes little or no material but improves dimensional accuracy, surface finish, and or hardness.

Shaving

Shaving is similar to gear shaping, but uses accurate shaving tools to remove small amounts of material from a roughed gear to correct profile errors and improve surface finish. Shaving operation is shown in fig.1.5.

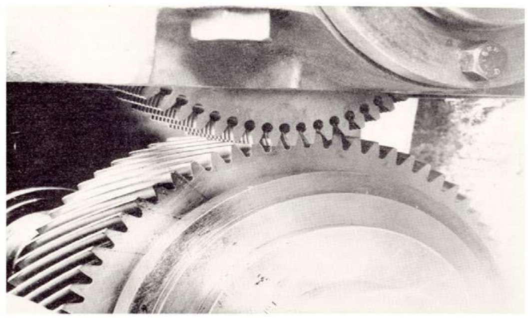

Figure 1-5 External gear being shaved [11]

Lapping and Honing

Lapping and honing both employ an abrasive-impregnated gear or gear-shaped tool that is run against the gear to abrade the surface. In both cases, the abrasive tool drives the gear in what amounts to an accelerated and controlled run-in to improve surface finish and the accuracy.

1.3 Types of Toothed wheels

Spur Gears

The two parallel and co-planar shafts connected by the gears as shown in Fig. 1.6.These gears are called spur gears. These gears have teeth parallel to the axis of the wheel.

Figure 1-6 Spur Gears [12]

Helical Gears

These gears have teeth inclined to the axis of the wheel as shown in Fig.1.7.The double helical gears are known as herringbone gears. As shown in Fig.1.8.

Figure 1-7 Helical Gears [12]

Figure 1-8 Herringbone Gears [12]

Bevel Gears

The two non-parallels or intersecting, but coplanar shaft connected by gears is shown in Fig.1.9.These gears are called bevel gears.

Figure 1-9 Bevel Gears [12]

1.4 Law of Gearing

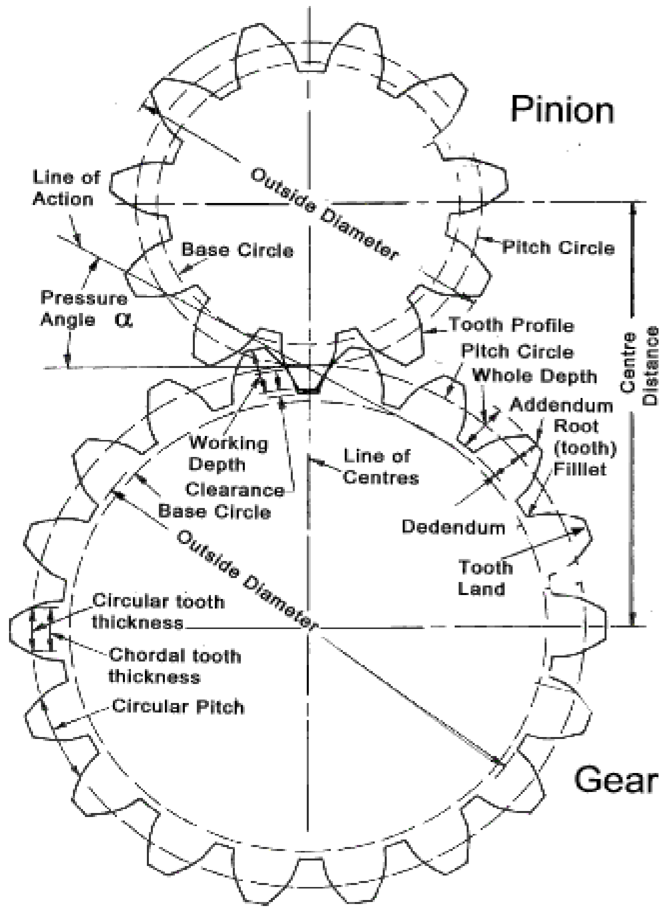

Figure 1-10 Nomenclature of gear [12]

The law of gearing states that the conditions which must be fulfilled by the gear tooth profiles to maintain a constant angular velocity ratio between two gears. It states that the common normal at the point of contact of the two teeth should always pass through a common pitch point.

1.5 Terms Used in Gear

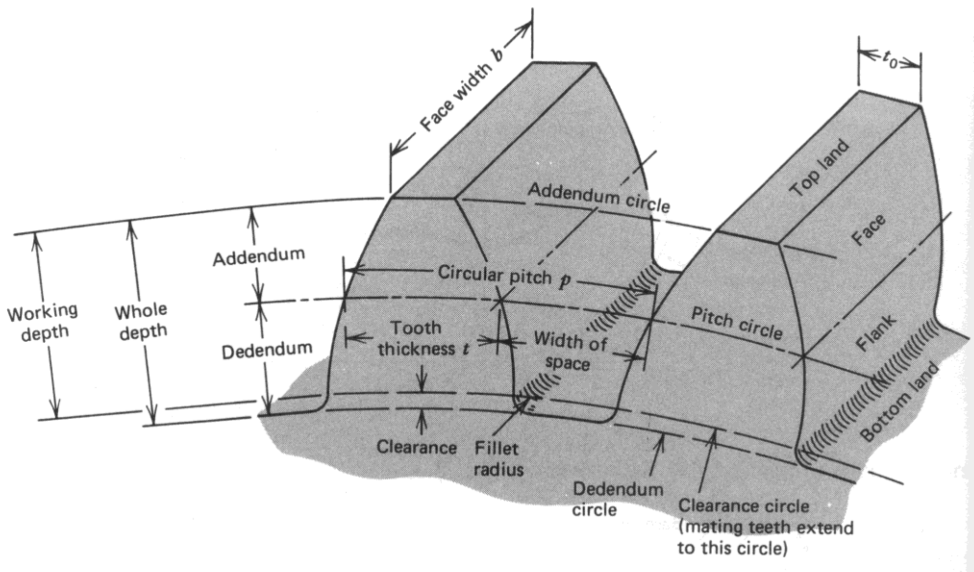

Figure 1-11 Nomenclature of gear [12]

Pitch circle

It is an imaginary circle which by pure rolling action would give the same motion as actual gear.

Pitch circle diameter

It is the diameter of the pitch circle. The size of the gear is usually specified by the pitch circle diameter. It is also known as pitch diameter.

Pitch point

It is a common point of contact between two pitch circles.

Pitch surface

It is the surface of the rolling discs which the meshing gears have replaced at the pitch circle.

Pressure angle

It is the angle between the common normal to two gear teeth at the point of contact and the common tangent at the pitch point. It is usually denoted by φ.The standard pressure angles are 14 $^{1/2°}$ and 20°.

Addendum

It is the radial distance of a tooth from the pitch circle to the top of the tooth.

Dedendum

It is the radial distance of a tooth from the pitch circle to the bottom of the tooth.

Addendum circle

It is the circle drawn through the top of the teeth and is concentric with the pitch circle.

Dedendum circle

It is the circle drawn through the bottom of the teeth. It is also called root circle.

Circular pitch

It is the distance measured on the circumference of the pitch circle from a point of one tooth to the corresponding point on the next tooth. It is usually denoted by p_c.

Mathematically,

Circular pitch, $p_c = \pi D/T$

Where D= Diameter of the pitch circle, and
T = Number of teeth on the wheel.

Diametral pitch

It is the ratio of number of teeth to the pitch circle diameter in millimeters. It is denoted by P_d.

Mathematically, Diametral pitch, $p_d = T/D$

Where T = Number of teeth, and
D= Pitch circle diameter.

Module

It is the ratio of the pitch circle diameter in millimetres to the number of teeth. It is usually denoted by m.Mathematically,

Module,m=D/T

Note:The recommended series of modules in Indian Standard are 1,1.25,1.5,2,2.5,3,4,5,6,8,10,12,16 and 20

Clearance

It is the radial distance from the top of the tooth to the bottom of the tooth, in meshing gear. A circle passing through the top of the meshing gear is known as clearance circle.

Total depth

It is the radial distance between the addendum and the dedendum circles of a gear. It is equal to the sum of the addendum and dedendum.

Working depth

It is the radial distance from the addendum circle to the clearance circle. It is equal to the sum of the addendum of the two meshing gears.

Tooth thickness

It is the width of the tooth measured along the pitch circle.

Tooth space

It is the width of space between the two adjacent teeth measured along the pitch circle.

Face of tooth

It is the surface of the gear tooth above the pitch surface.

Flank of tooth

It is the surface of the gear tooth below the pitch surface.

Top land

It is the surface of the top of the tooth.

Face width

It is the width of the gear tooth measured parallel to its axis.

Fillet radius

It is the radius that connects the root circle to the profile of the tooth.

1.6 Gear Materials

The material used for the manufacture of gears depends upon the strength and service conditions like wear, noise etc. The gears may be manufactured from metallic or non-metallic materials. The metallic gears with cut teeth are commercially obtainable in cast iron, steel and bronze. The non-metallic materials like wood, raw hide, compressed paper and synthetic resins like nylon are used for gears, especially for reducing noise. The cast iron is widely used for the manufacture of gears due to its good wearing properties, excellent machinability and ease of producing complicated shapes by casting method. The cast iron gears with cut teeth may be employed, where smooth action is not important. The steel is used for high strength gears and steel may be plain carbon steel or alloy steel. The steel gears are usually heat treated in order to combine properly the toughness and tooth hardness.

The phosphor bronze is widely used for worm gears in order to reduce wear of the worms which will be excessive with cast iron or steel.

1.7 CURRENT METHODS OF GENERATING HOB TOOL PROFILE

1.7.1 Manual Method

Drawing or specimen of the gear from the customer.

↓

3D model of the gear generated using manual drafting or CADD.

↓

Tool drawing(HOB tool profile) created with the help of optical profile projectors(shown in Fig.1.13.) it magnifies the drawing of gear. The scale is high i.e. 25 times the original for easily tracing the HOB profile in small incremental steps similar to manufacturing of gear.

↓

After the HOB tool drawing it is used to trace the shape of grinding tool. Which is created then for use on a grinding machine, to physically create the single point cutting tool (shown in Fig.1.14.) for use on a lathe machine to machine the HOB tool

↓

The HOB tool is used on a HOBBING machine which cut the blank into Gear.

Figure 1-12 Gear Hob

(Courtesy: Basal Tools Ltd, Patiala)

Figure 1-13 Optical profile projector

(Courtesy: Basal Tools Ltd, Patiala)

Figure 1-14 Gear hob forming tools

(Courtesy: Basal Tools Ltd, Patiala)

CHAPTER-2

LITERATURE REVIEW

Antoniadis A. *et al.* **[4]**

In this paper study the authors has done work on the gear quality through three main stages i.e. the rough cutting, the heat treatment and the finishing process. the gear skiving or hard hobbing are adopted for the gear finishing. This paper involves the algorithm that simulates rigorously the hard hobbing. This algorithm is supported by a computer code that offers the parameters. The developed initially performs the simulation of gear cutting within to determine the cutting boundary conditions. The aim of this research work is to optimizing the cutting process and to predict the component before the actual or physically used.

Masataka Yoneekura.*et al.* [5]

This paper deals with the study of improvement in gear accuracy whish has obtained through the practical use of a newly developed hobbing machine. There is a use of a carbide hob for finishing the hardened gears. The hob was characterized by its performance of finishing only tooth surface of a gear without chipping the cutting edge on a universal hobbing machine. The new CNC hobbing machine with higher rigidity and the backlash eliminator has been developed for large gears. From the experimental results, the new machine was found to be effective in finishing of hardened gears.

In comparison to the universal hobbing machine, the newly developed NC hobbing machine improved the tooth profile errors that occur in the work of the cutting edge etc.When finishing the small sized gears. This improvement seemed to have been realized by the improved accuracy and rigidity of the hobbing machine as well as the installation of the backlash eliminator. The new type of hobbing machine made the tooth profile to be identical in its shape even when there were tooth profile errors in between the neighboring grooves caused by hardening and when difference in hardness.

Mohan L.V.*et al.* [6]

This paper described the study of inherent errors in the gear hobbing process by geometrical analysis and by simulation. An attempt has been made to give guideline to the manufacturers as well as the designers, in the proper selection of cutter and design of the gear.

After analysis the following results are:

Hobbing can be safely used to cut external gears more than 22 teeth without undercutting. When less than 22 teeth is required addendum modification is to be done to avoid undercutting .Otherwise, we can go gear shaping or form cutting. As the number of flutes of hob is fixed the profile accuracy cannot be improved still further with the same hob. The inherent nature of hobbing process forms crest on the flank surface of the gear.

Chung-Yunn Lin.*et al.* **[7]**

This paper presents the complete set of mathematical model of helical set with small number of teeth. The unavoidable tooth profile undercutting of the gear with small number of gear is examined by using developed mathematical model.The proposed methods and developed mathematical models of the modified helical gear canbe helpful to facilitate to design and manufacture of spur and helical gears with small number of teeth.

Michalski J.*et al.* [8]

This paper deals with the simulation methodology of modeling tooth flanks of cylindrical gears with the help of computer. A model of tooth flanks is an envelope curve of a family of envelopes that originate from the rolling motion of a solid tool model in relation to a solid model of the cylindrical gear.

This method of generating cylindrical gear flanks which provide the precise definition of the stereometric shape and surface topography.

Stephen P. *et al*. [9]

This paper deals with the impact of the major design parameters of an involute hob on the deviation of hob tooth profile in which the author considered the desired and actual hob tooth profile. Using an analytical method for computation of two deviation is developed. Based on the Based on the research results, the author has proposed the great ways for increasing the accuracy of involute hob in actual practice. With the help of commercial software's MathCAD/Scientific.

Michalski J.*et al.* [10]

This paper deals with the methodology of modeling tooth flanks of cylindrical gears with the help of CAD environment. Computer simulation of gear generation consists of rolling motion of a solid tool model with respect to the solid model of the cylindrical gears. The surface stereometry and topography of the tooth flanks by Fellows method used to comparing the numerical models. Physically metrological measurements of gears have been done with the help of profilometer and CMM.

2.1 Summary of literature surveyed

Gear manufacturing using hob is standardized manufacturing technique. The manufacturing of hub cutter requires a good quality production of gears. The survey of literature and practical industries in the manufacturing of hob cutter, the method used by most of the manufacturers is a conventional method of manual drawing/tracing along with the use of conventional machines from the observation point of view it takes a lot of manpower and time as well as resulting in production of lower accuracy. Thus there is a need of automation will help increase efficiency and productivity with the incorporation CAD tools with higher accuracy. This method is to adopt to eliminate the manual inaccuracies by incorporation use of automated CAD desired tool profile.

CHAPTER-3

WORK DONE

3.1 Work done gear to cutter profile generation using CAD softwares

By understanding the whole work related to the CAD automation generation of gear to cutter profile from thesis [3] work. Implementation of Methodology by own firstly done in Pro/E then in solidworks to observe the method how it works when it relate with the parameters and/or equations. To see the change in the shape of profile if there is any change in the relation between the various parameters. The Different types of hob cutters whose profile have been automated are: Gear hob cutter, Spline hob cutter(required to cut spline on shaft),Sprocket hob cutter(required to cut a chain drive sprocket).

Replicate the methodology in the Pro/E:

Replication of methodology generation of gear to cutter profile with module 6mm. The gear to cutter generation is done in Pro/E as shown in fig.3.1 below gear profile by relating the various parameters of the gear.

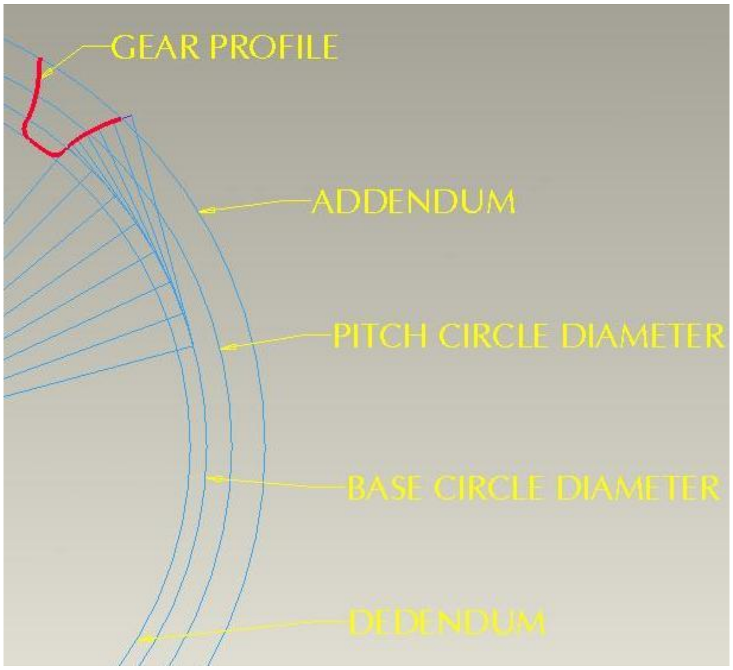

Figure 3-1 Sketch of Gear Profile (module 6)

Fig.3.2 shows the steps of making tool profile by rotating the gear profile at certain angle with translate motion.

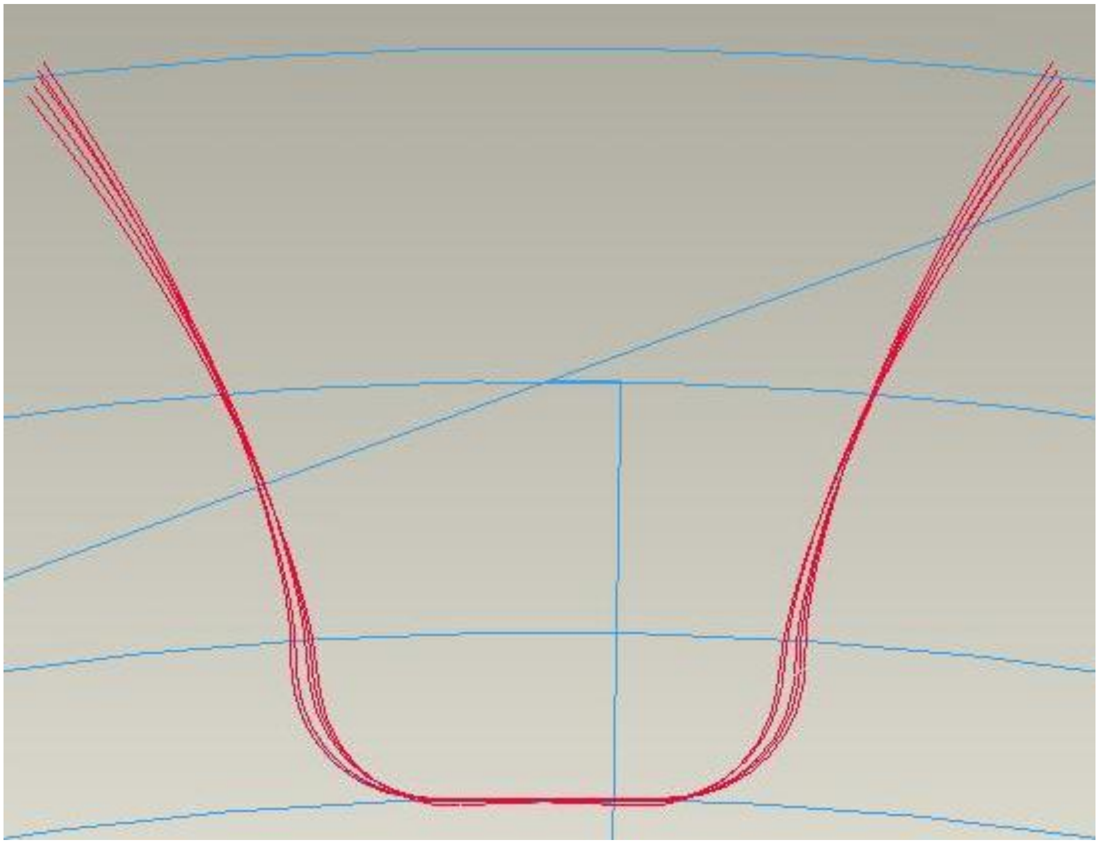

Figure 3-2 Steps of making tool profile sketch

Fig.3.3 shows the outline of the tool profile generated from gear profile.

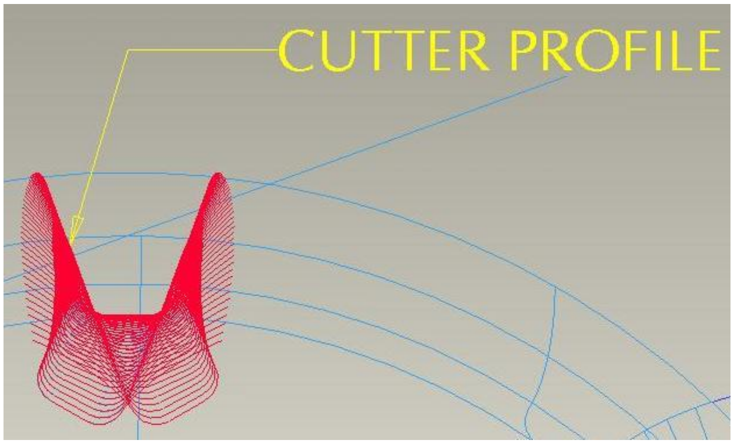

Figure 3-3 Outline of tool profile

The automation is done with by defining the proper relationships between the parameters. These all shown in the fig.3.4 for changing the profile of the gear as required.

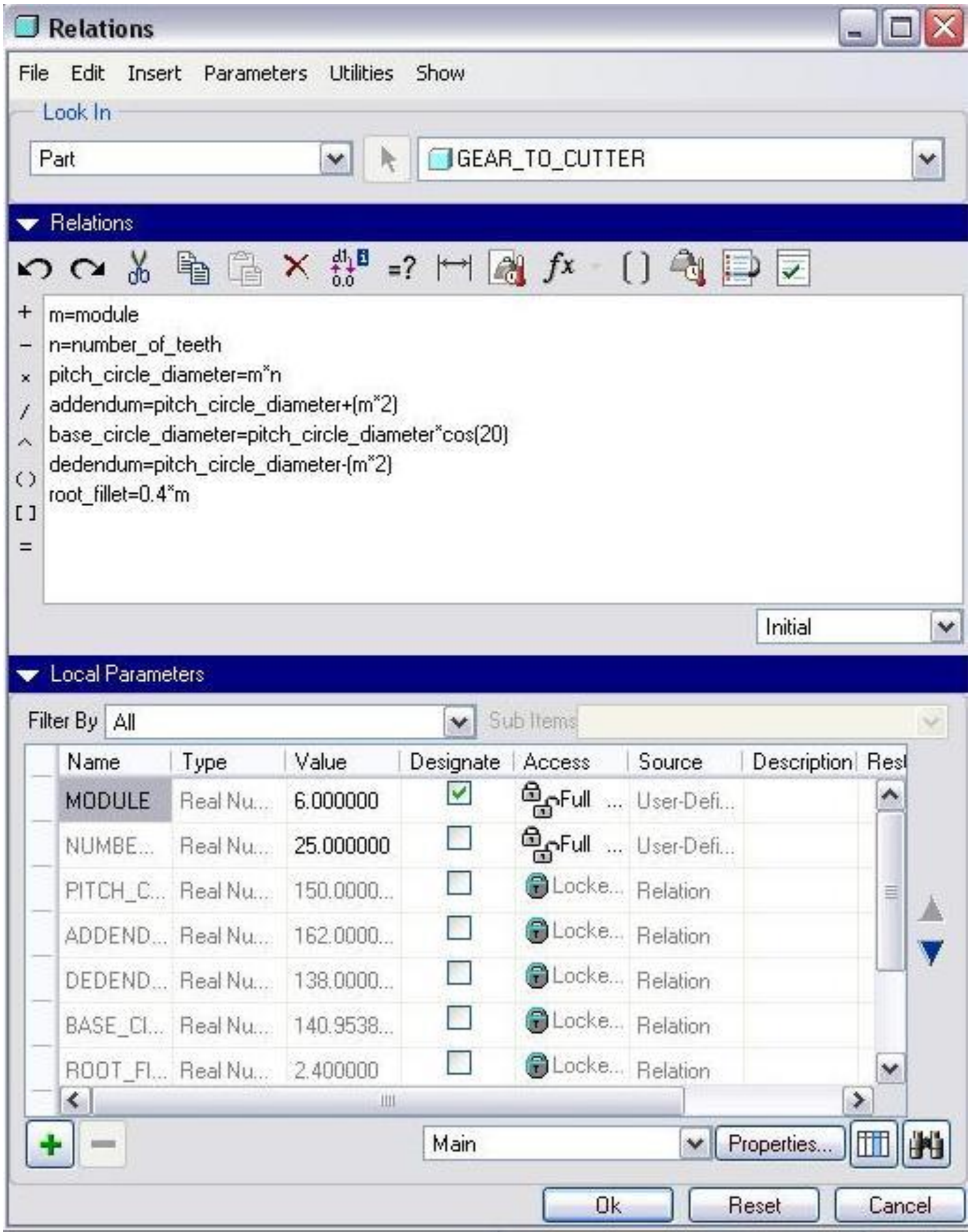

Figure 3-4 Parametric automation

3.2 Application Programming Interface (API)

The quickest and easiest way to start programming with the SolidWorks API is to record a SolidWorks macro, which contains the SolidWorks API, calls that correspond to the actions performed in the user interface. You can modify the macro in Microsoft Visual Basic for Applications (VBA).

Microsoft VBA is a toolset based on Microsoft Visual Basic for Applications (VBA) and is embedded in the SolidWorks software.

Solidworks macro toolbar consists of various option like record/pause, run macro, edit macro, new macro, run macro. As shown below in fig.3.5.

Figure 3-5 Macro toolbar

Visual Basic for Applications (VBA) is the engine that records, runs, or edits macros in SolidWorks. Recorded macros are saved as .swp VBA project files. We can read and edit .swb and .swp (VBA) files with the VBA editor. When you edit an existing .swb file, the file automatically converts into a .swp file. We can use VBA to create forms and to provide more user interaction with recorded macros.

3.3 Methodology Implementation in Solidworks for various profiles:

3.3.1 Gear to cutter Profile

Developing of gear to cutter profile generation in Solidworks software with gear (module= 10 mm) are done by relating the equation between the various parameters of gear e.g. module, number of teeth, pitch circle diameter,addendum,dedendum,pressure angle, base circle etc.As shown in fig.3.6.

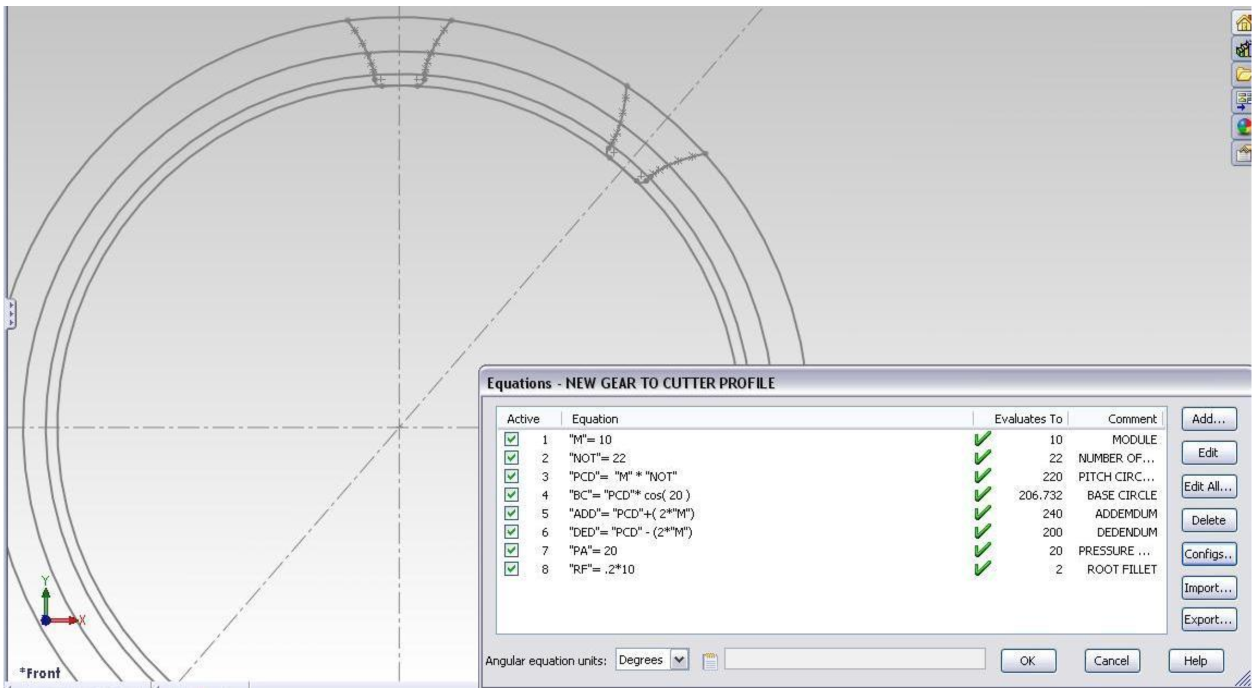

Figure 3-6 Sketch of Gear Profile (module 6)

After simulating the respective gear profile as shown in fig.3.6 by rotating about z – axis at some angle and translating from right plane at some distance will give the cutter profile from the gear profile showing the reverse simulation of machining process as shown in fig.3.7.has been done with the help of API (Application Program Interface) using the visual basics coding as shown in fig.3.8.

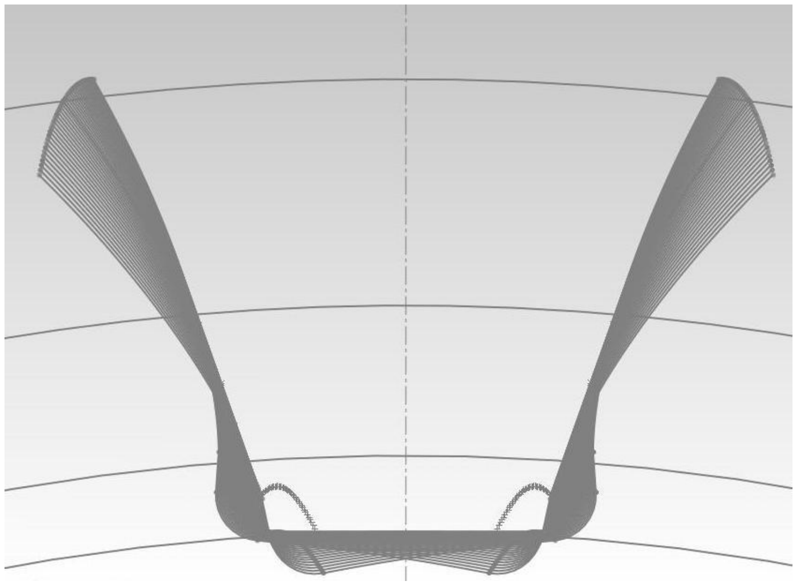

Figure 3-7 Shows the steps of making cutter profile by rotating the gear profile at certain angle with translate motion

```
Dim swApp As Object

Dim Part As Object
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long
Dim i As Integer

Sub main()
i = 0
Set swApp = _
Application.SldWorks

Set Part = swApp.ActiveDoc
Do
boolstatus = Part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
Part.SketchManager.InsertSketch True
boolstatus = Part.Extension.SelectByID2("Chain", "EXTSKETCHSEGMENT", -0.01181971257091, 0.1141614877112, 0, True, 0, No
boolstatus = Part.Extension.SelectByID2("Spline3@Sketch1", "EXTSKETCHSEGMENT", -0.1643134722763, 0.1515211603133, 0, Tr
boolstatus = Part.Extension.SelectByID2("Line26@Sketch1", "EXTSKETCHSEGMENT", -0.1643134722763, 0.1515211603133, 0, Tru
boolstatus = Part.Extension.SelectByID2("Arc8@Sketch1", "EXTSKETCHSEGMENT", -0.1643134722763, 0.1515211603133, 0, True,
boolstatus = Part.Extension.SelectByID2("Arc9@Sketch1", "EXTSKETCHSEGMENT", -0.1643134722763, 0.1515211603133, 0, True,
boolstatus = Part.Extension.SelectByID2("Arc10@Sketch1", "EXTSKETCHSEGMENT", -0.1643134722763, 0.1515211603133, 0, True
boolstatus = Part.Extension.SelectByID2("Line27@Sketch1", "EXTSKETCHSEGMENT", -0.1643134722763, 0.1515211603133, 0, Tru
boolstatus = Part.Extension.SelectByID2("Spline4@Sketch1", "EXTSKETCHSEGMENT", -0.1643134722763, 0.1515211603133, 0, Tr
boolstatus = Part.SketchManager.SketchUseEdge2(False)
Part.ClearSelection2 True
Part.ClearSelection2 True
boolstatus = Part.Extension.SketchBoxSelect("-0.024889", "0.127395", "0.000000", "0.031236", "0.085999", "0.000000")
Part.Extension.RotateOrCopy False, 1, False, 0, 0, 0, 0, 0, 1, 0.006981317008 * i
Part.ClearSelection2 True
boolstatus = Part.Extension.SketchBoxSelect("-0.031746", "0.128411", "0.000000", "0.023871", "0.087777", "0.000000")
Part.Extension.MoveOrCopy False, 1, False, 0, 0, 0, 0.000768 * i, 0, 0
Part.ClearSelection2 True
Part.SketchManager.InsertSketch True
i = i + 1
Loop While i <= 20
End Sub

'Set Part = swApp.ActiveDoc
```

Figure 3-8 Visual Basic coding for generating the gear cutter profile

The fig.3.9 shows the respective simulated gear to cutter profile with line extending macro has been generated for locating the end points on the outline of the simulated gear to cutter profile with the help of API using the visual basics coding .After getting the end points on the simulated profile then curve through reference points has been used for generating the outline final profile, ready to use for manufacturing. as shown in fig.3.10.

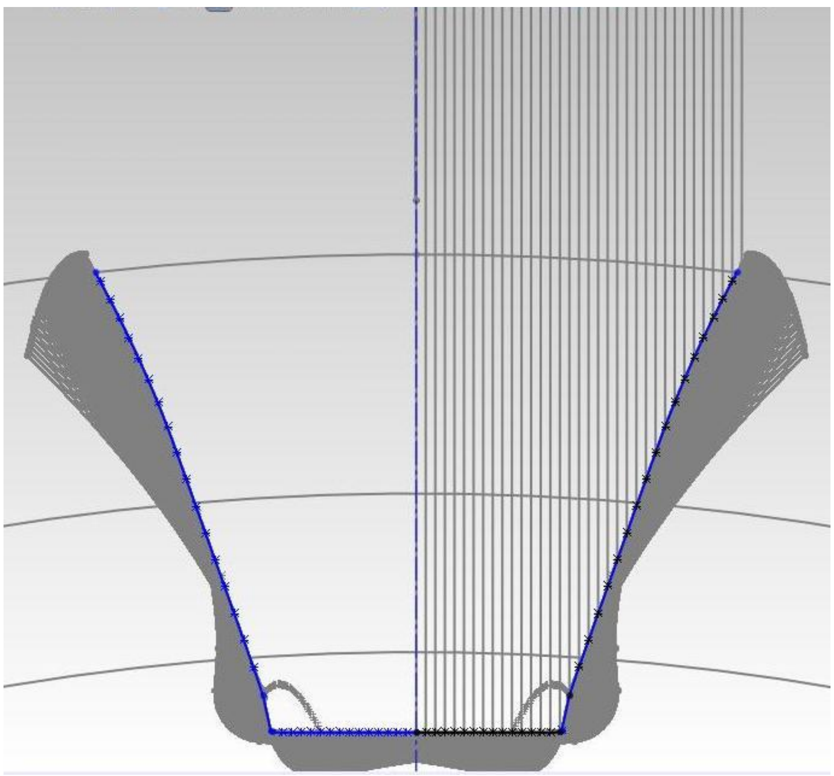

Figure 3-9 Tracing the gear cutter profile

Drawing format:

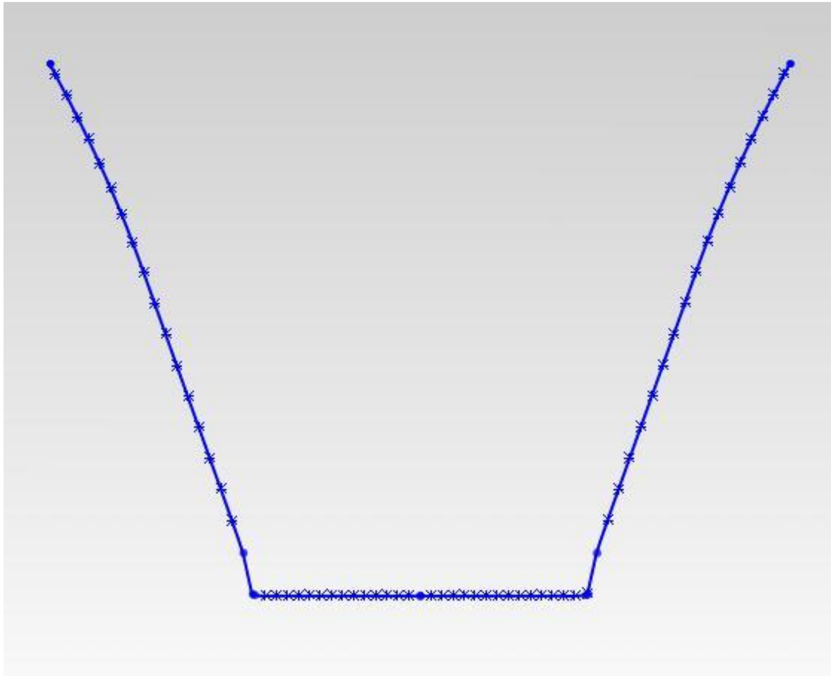

Figure 3-10 Drawing of gear cutter profile ready to use for manufacturing

3.3.2 Rack to gear Profile

Developing of rack to gear profile generation in Solidworks software with (module= 10 mm) are done by relating the equation between the various parameters of rack e.g. module, number of teeth, pitch circle diameter,addendum,dedendum,pressure angle,circular pitch,tooth thickness etc.As shown in fig.3.11.

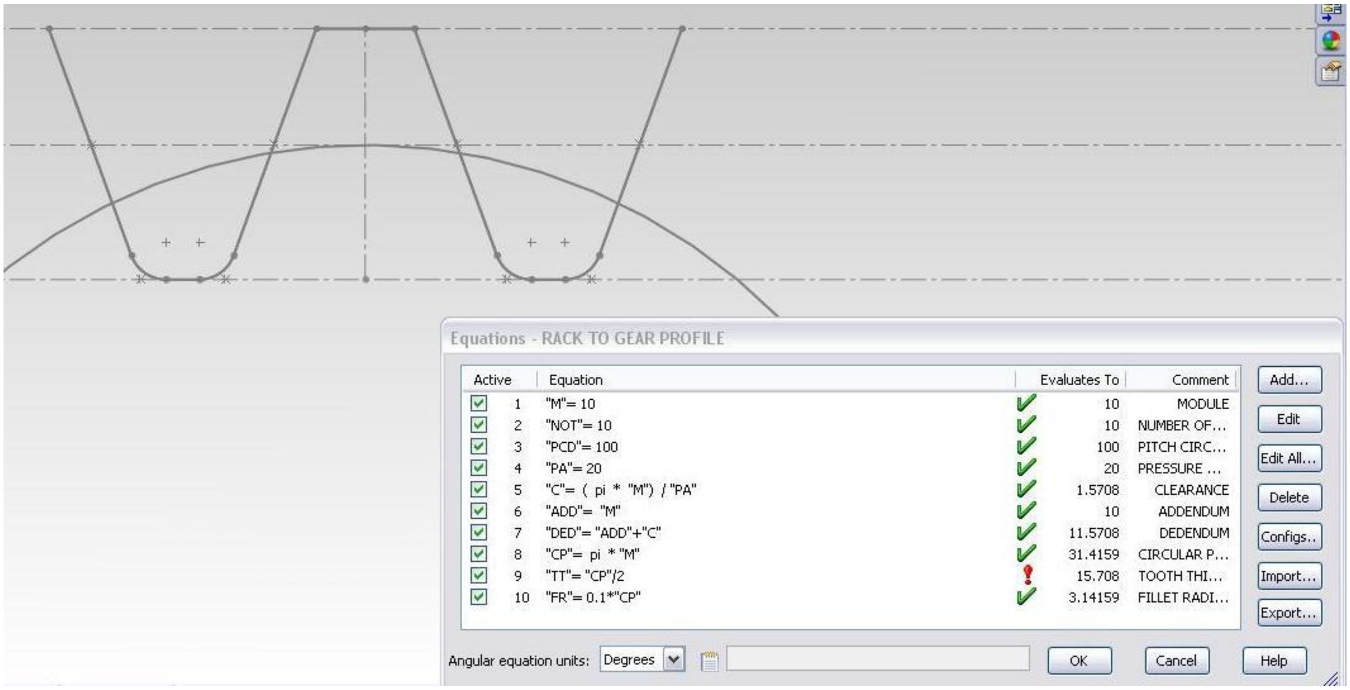

Figure 3-11 Sketch of Rack profile

After simulating the respective rack profile as shown in fig.3.11 by rotating about z – axis at some angle and translating from right plane at some distance will give the gear profile from the rack profile showing the reverse simulation of machining process as shown in fig.3.12 has been done with the help of API (Application Program Interface) using the visual basics coding as shown in fig.3.13.

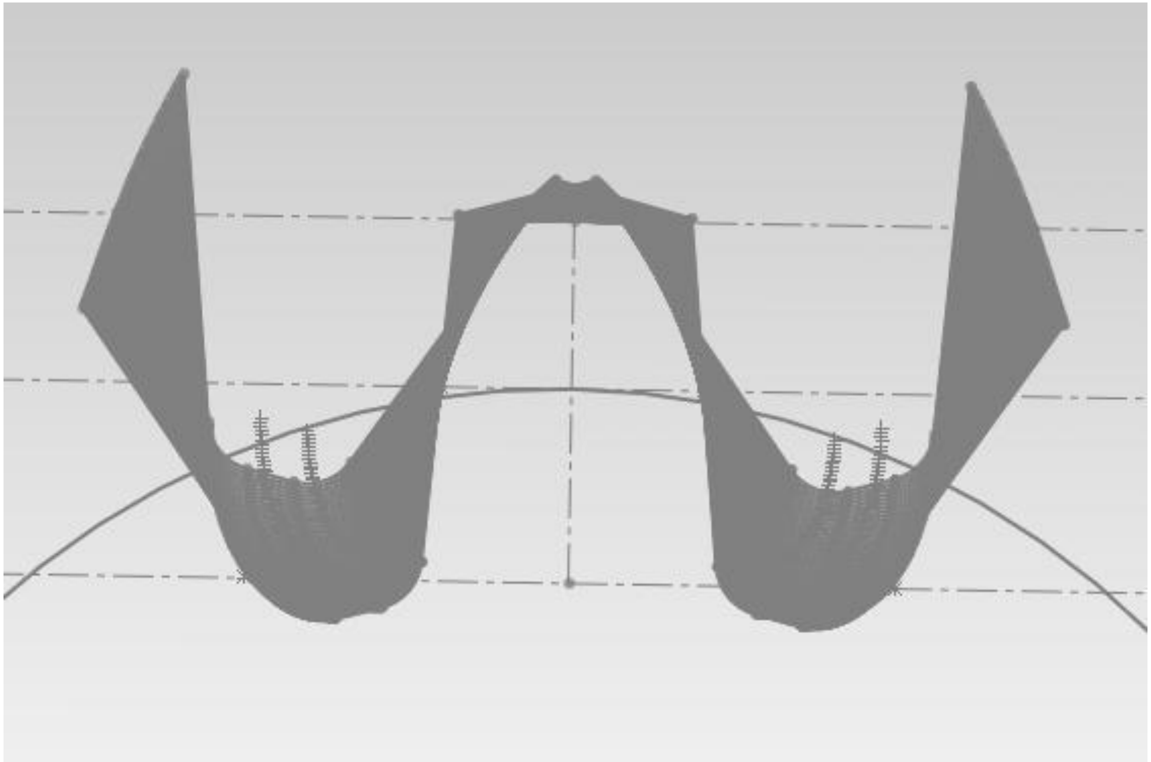

Figure 3-12 Shows the steps of making gear profile by rotating the rack profile at certain angle with translate motion

```
Dim swApp As Object

Dim Part As Object
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long
Dim i As Integer

Sub main()
i = 0

Set swApp = _
Application.SldWorks

Set Part = swApp.ActiveDoc
Do
boolstatus = Part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
Part.SketchManager.InsertSketch True
boolstatus = Part.Extension.SelectByID2("Line41@RACK PROFILE", "EXTSKETCHSEGMENT", -0.02445147765871, 0.05245491061253, 0, True, 0,
boolstatus = Part.Extension.SelectByID2("Arc11@RACK PROFILE", "EXTSKETCHSEGMENT", -0.01846378798276, 0.03871823850997, 0, True, 0, N
boolstatus = Part.Extension.SelectByID2("Line36@RACK PROFILE", "EXTSKETCHSEGMENT", -0.01605761416511, 0.0384292036732, 0, True, 0, N
boolstatus = Part.Extension.SelectByID2("Arc10@RACK PROFILE", "EXTSKETCHSEGMENT", -0.01223032827078, 0.03917343247996, 0, True, 0, N
boolstatus = Part.Extension.SelectByID2("Line42@RACK PROFILE", "EXTSKETCHSEGMENT", -0.01097447329269, 0.04141558018064, 0, True, 0,
boolstatus = Part.Extension.SelectByID2("Line35@RACK PROFILE", "EXTSKETCHSEGMENT", -0.001897815597829, 0.06, 0, True, 0, Nothing, 0)
boolstatus = Part.Extension.SelectByID2("Line39@RACK PROFILE", "EXTSKETCHSEGMENT", 0.006868883789162, 0.05271747353396, 0, True, 0,
boolstatus = Part.Extension.SelectByID2("Arc9@RACK PROFILE", "EXTSKETCHSEGMENT", 0.01222918491891, 0.03918117763111, 0, True, 0, Not
boolstatus = Part.Extension.SelectByID2("Line37@RACK PROFILE", "EXTSKETCHSEGMENT", 0.01526480052587, 0.03842920367322, 0, True, 0, N
boolstatus = Part.Extension.SelectByID2("Arc8@RACK PROFILE", "EXTSKETCHSEGMENT", 0.01891657043849, 0.0389693072365, 0, True, 0, Noth
boolstatus = Part.Extension.SelectByID2("Line38@RACK PROFILE", "EXTSKETCHSEGMENT", 0.02110480730823, 0.04323813049545, 0, True, 0, N
boolstatus = Part.SketchOffsetEntities2(0, False, True)
Part.ClearSelection2 True
Part.ClearSelection2 True
boolstatus = Part.Extension.SketchBoxSelect("-0.034256", "0.070226", "0.000000", "0.028313", "0.030848", "0.000000")
Part.Extension.MoveOrCopy False, 1, False, 0, 0, 0, 0.00043635 * i, 0, 0
Part.Extension.RotateOrCopy False, 1, False, 0, 0, 0, 0, 0, 1, 0.00872664626 * i
Part.SketchManager.InsertSketch True
i = i + 1
Loop While i <= 30
End Sub

'Set Part = swApp.ActiveDoc
```

Figure 3-13 Visual Basic coding for generating the gear profile

The fig.3.14 shows the respective simulated rack to gear profile with line extending macro has been generated for locating the end points on the outline of the simulated rack to gear profile with the help of API using the visual basics coding .After getting the end points on the simulated profile then curve through reference points has been used for generating the outline final profile as shown in fig.3.15.

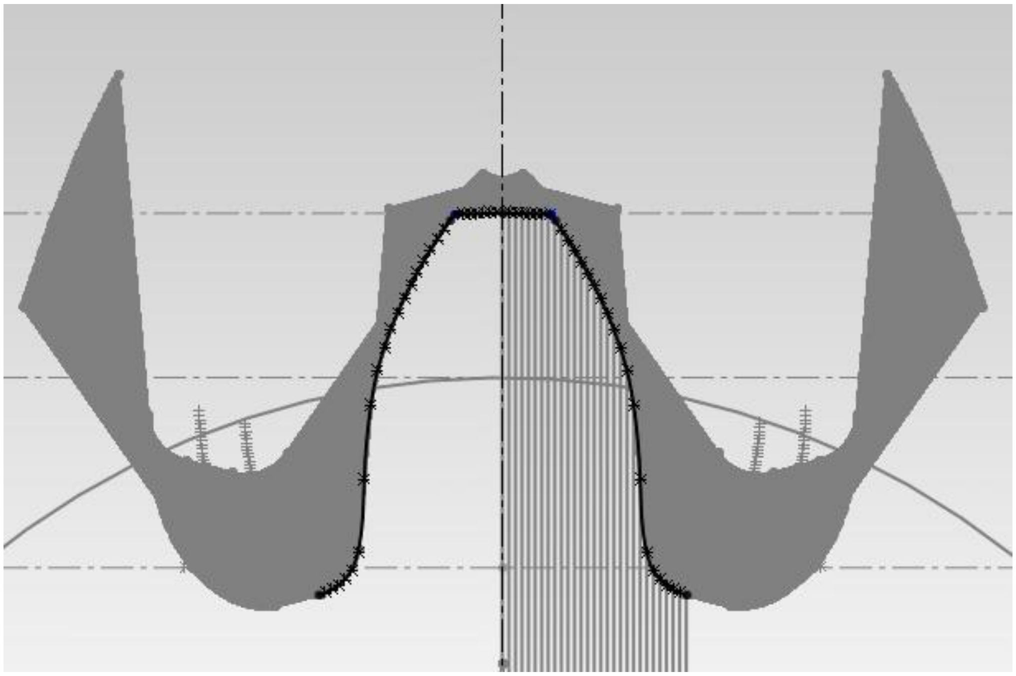

Figure 3-14 Tracing the gear profile

Drawing format:

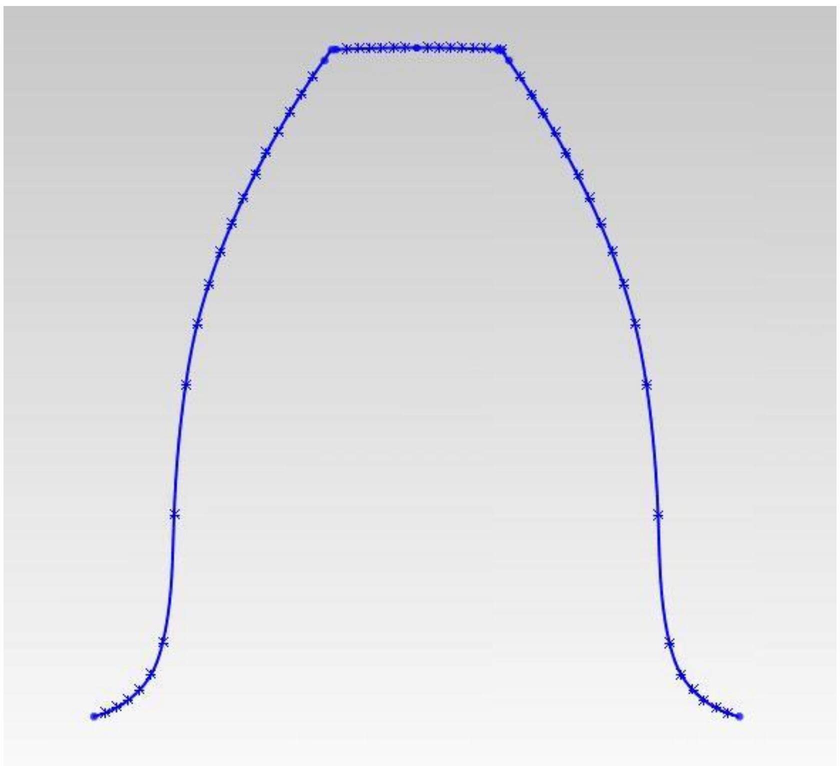

Figure 3-15 Drawing of actual gear profile

3.3.3 Sprocket hob cutter profile:

Generating of sprocket hob cutter in Solidworks software are done by relating the equation between the various parameters of sprocket e.g. pitch of chain, number of teeth, roller diameter, pitch circle etc.As shown in fig.3.16.

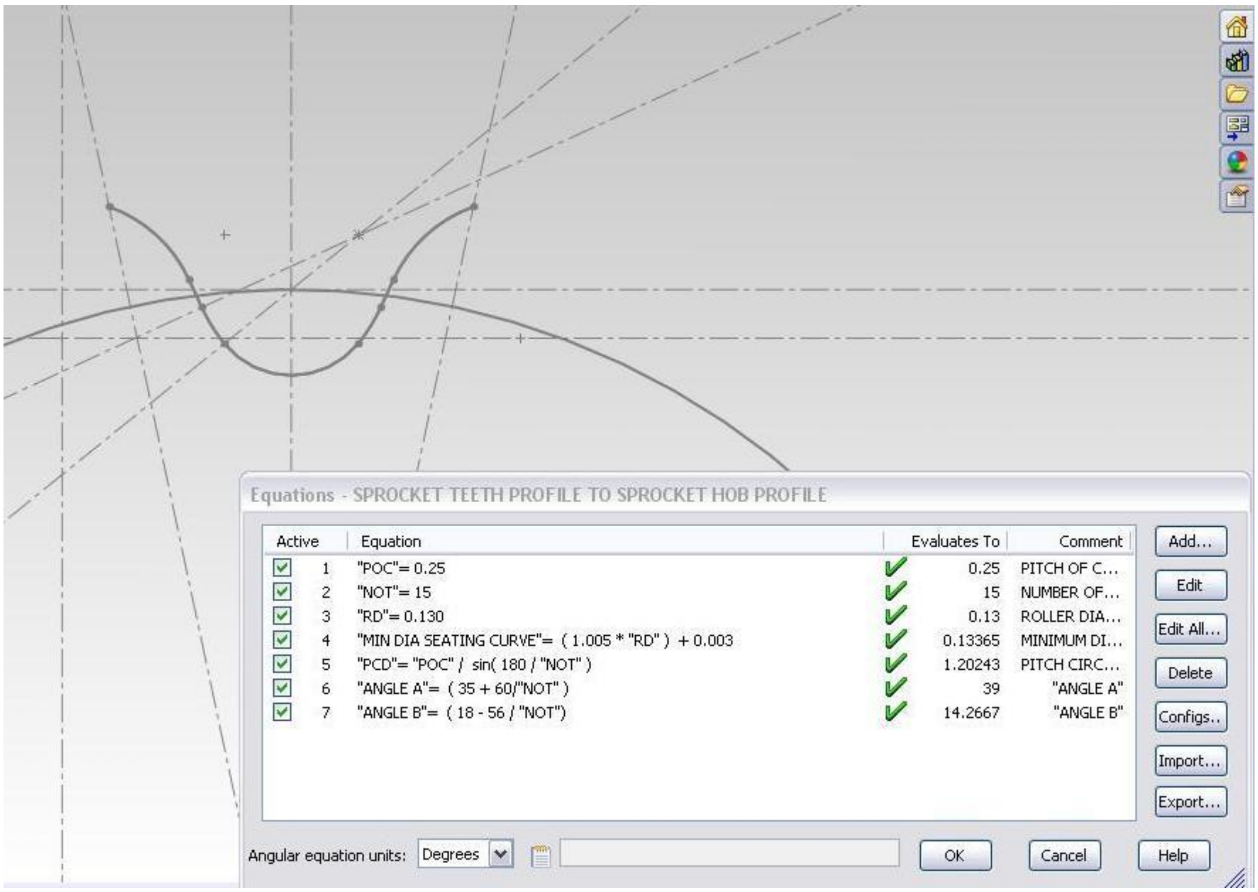

Figure 3-16 Sketch of Sprocket Profile

After simulating the respective sprocket profile as shown in fig.3.16 by rotating about z – axis at some angle and translating from right plane at some distance will give the cutter profile from the sprocket profile showing the reverse simulation of machining process as shown in fig.3.17 has been done with the help of API (Application Program Interface) using the visual basics coding as shown in fig.3.18.

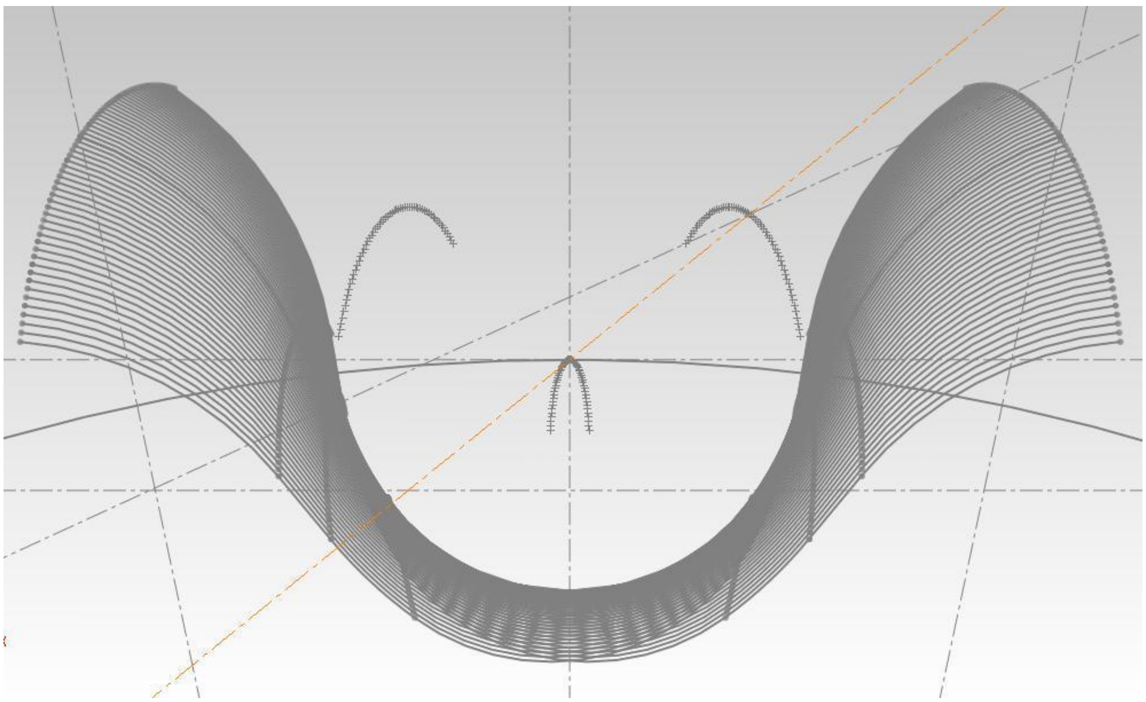

Figure 3-17 Shows the steps of making sprocket cutter profile by rotating the sprocket profile at certain angle with translate motion

```
Dim swApp As Object

Dim Part As Object
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long
Dim i As Integer

Sub main()
i = 0

Set swApp = _
Application.SldWorks

Set Part = swApp.ActiveDoc
Do
boolstatus = Part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
Part.SketchManager.InsertSketch True
boolstatus = Part.Extension.SelectByID2("Arc13@SPROCKET TOOTH PROFILE", "EXTSKETCHSEGMENT", -1.222901216615E-04, 6.566
boolstatus = Part.Extension.SelectByID2("Line11@SPROCKET TOOTH PROFILE", "EXTSKETCHSEGMENT", -7.324635091192E-05, 5.95
boolstatus = Part.Extension.SelectByID2("Arc10@SPROCKET TOOTH PROFILE", "EXTSKETCHSEGMENT", -5.710520042922E-05, 5.659
boolstatus = Part.Extension.SelectByID2("Arc7@SPROCKET TOOTH PROFILE", "EXTSKETCHSEGMENT", -3.381847269918E-05, 5.4358
boolstatus = Part.Extension.SelectByID2("Arc18@SPROCKET TOOTH PROFILE", "EXTSKETCHSEGMENT", 6.281489724336E-05, 5.7474
boolstatus = Part.Extension.SelectByID2("Line15@SPROCKET TOOTH PROFILE", "EXTSKETCHSEGMENT", 7.387178873684E-05, 5.969
boolstatus = Part.Extension.SelectByID2("Arc19@SPROCKET TOOTH PROFILE", "EXTSKETCHSEGMENT", 9.120075303178E-05, 6.2868
boolstatus = Part.SketchOffsetEntities2(0, False, True)
Part.ClearSelection2 True
Part.ClearSelection2 True
boolstatus = Part.Extension.SketchBoxSelect("-0.000217", "0.000747", "0.000000", "0.000240", "0.000516", "0.000000")
Part.Extension.RotateOrCopy False, 1, False, 0, 0, 0, 0, 0, 1, 0.00872664626 * i
Part.Extension.MoveOrCopy False, 1, False, 0, 0, 0, 0.000005 * i, 0, 0
Part.SketchManager.InsertSketch True
i = i + 1
Loop While i <= 30
End Sub

'Set Part = swApp.ActiveDoc
```

Figure 3-18 Visual Basic coding for generating the sprocket cutter profile

The fig.3.19 shows the respective simulated sprocket to cutter profile with line extending macro has been generated for locating the end points on the outline of the simulated sprocket to cutter profile with the help of API using the visual basics coding .After getting the end points on the simulated profile then curve through reference points has been used for generating the outline final profile as shown in fig.3.20.

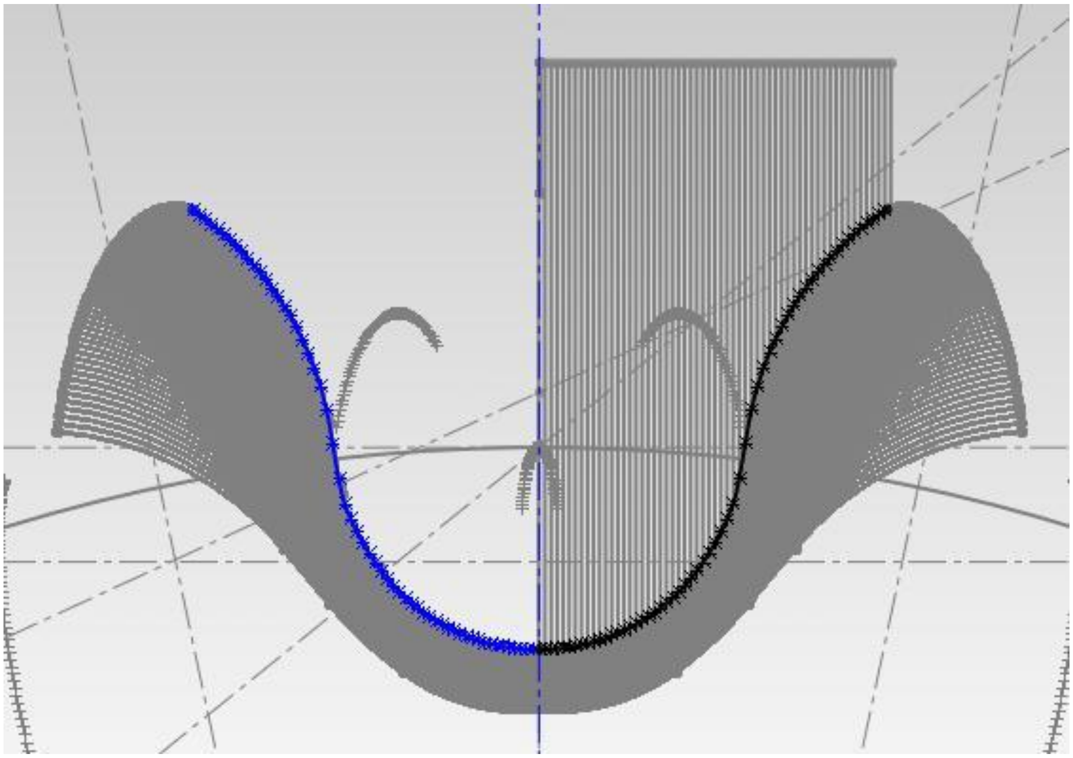

Figure 3-19 Tracing the sprocket cutter profile

Drawing format:

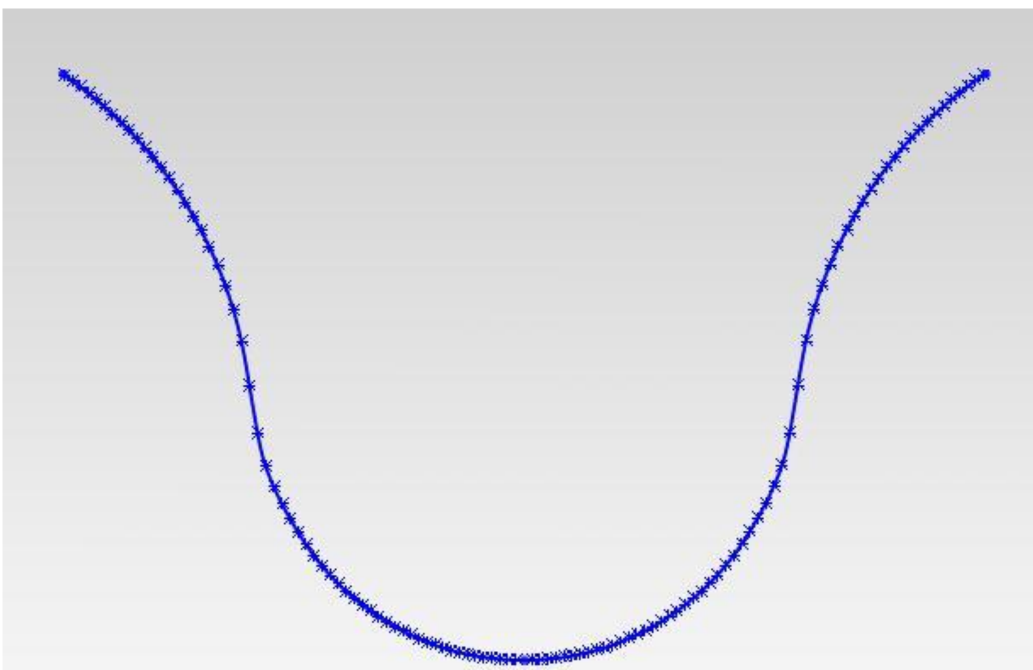

Figure 3-20 Drawing of sprocket cutter profile

3.3.4 Spline hob cutter profile:

Generating of spline hob cutter in Solidworks software are done by relating the equation between the various parameters of sprocket e.g.outside diameter,root diameter,width of spline,number of spline,pitch circle diameter.As shown in fig.3.21.

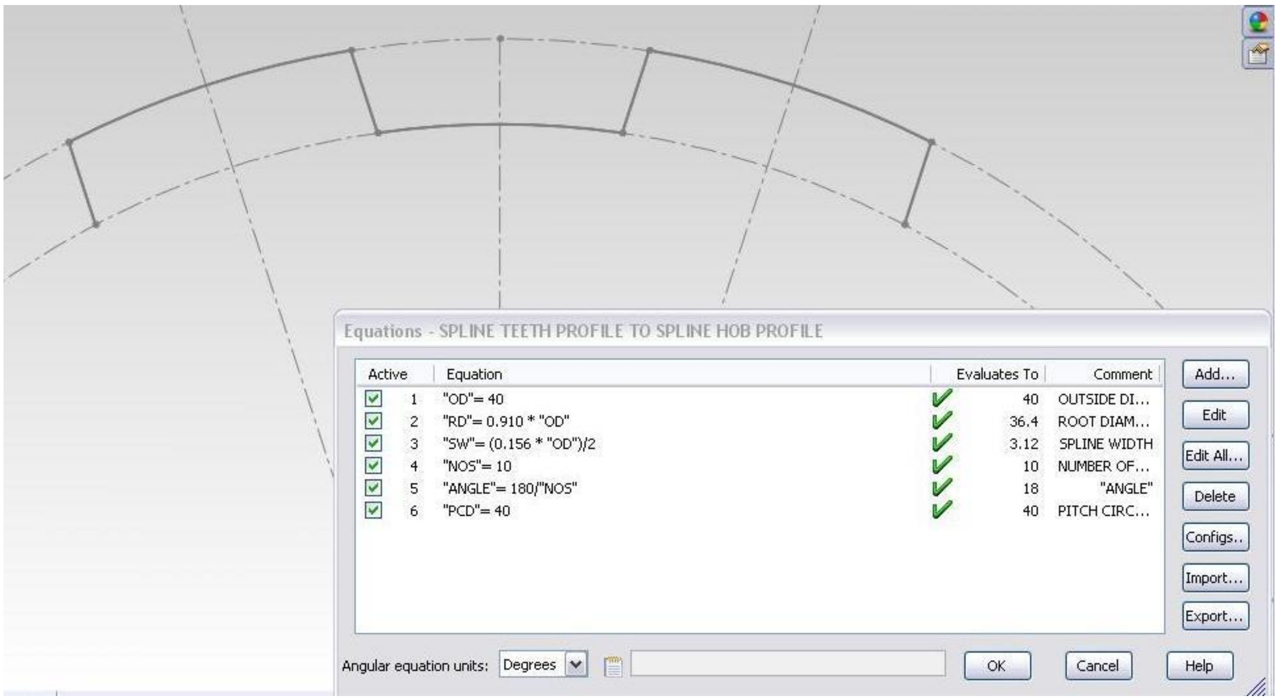

Figure 3-21 Sketch of Spline Profile

After simulating the respective spline profile as shown in fig.3.21 by rotating about z – axis at some angle and translating from right plane at some distance will give the cutter profile from the spline profile showing the reverse simulation of machining process as shown in fig.3.22 has been done with the help of API (Application Program Interface) using the visual basics coding as shown in fig.3.23.

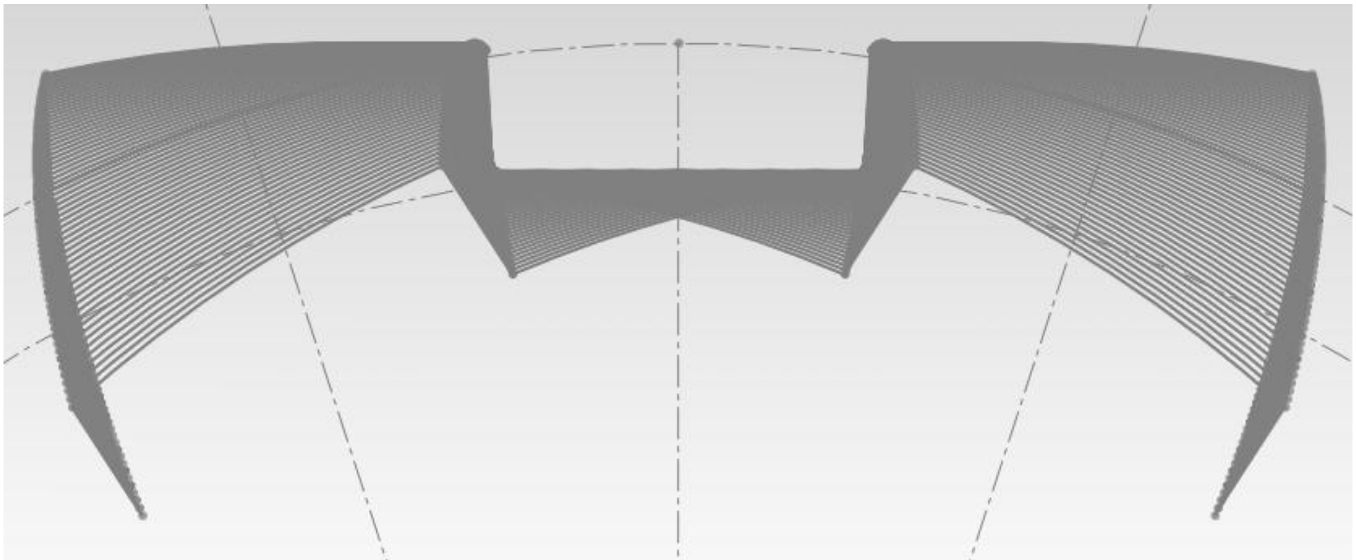

Figure 3-22 Shows the steps of making spline cutter profile by rotating the spline profile at certain angle with translate motion

```vb
Dim swApp As Object

Dim Part As Object
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long
Dim i As Integer

Sub main()
i = 0

Set swApp = _
Application.SldWorks

Set Part = swApp.ActiveDoc
Do
boolstatus = Part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
Part.SketchManager.InsertSketch True
boolstatus = Part.Extension.SelectByID2("Line28@SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", -0.008774690139243, 0.01690
boolstatus = Part.Extension.SelectByID2("Arc7@SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", -0.00798955988175, 0.01833485
boolstatus = Part.Extension.SelectByID2("Line29@SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", -0.002867235511944, 0.01892
boolstatus = Part.Extension.SelectByID2("Arc8@SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", -0.001151533864132, 0.0181635
boolstatus = Part.Extension.SelectByID2("Line25@SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.002735080118192, 0.018514
boolstatus = Part.Extension.SelectByID2("Arc6@SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.004079950014996, 0.01957942
boolstatus = Part.Extension.SelectByID2("Line27@SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.008750751636181, 0.016835
boolstatus = Part.SketchOffsetEntities2(0, False, True)
Part.ClearSelection2 True
Part.ClearSelection2 True
boolstatus = Part.Extension.SketchBoxSelect("-0.011523", "0.020449", "0.000000", "0.009743", "0.015543", "0.000000")
Part.Extension.RotateOrCopy False, 1, False, 0, 0, 0, 0, 0, 1, 0.00872664626 * i
Part.Extension.MoveOrCopy False, 1, False, 0, 0, 0, 0.00016 * i, 0, 0
Part.SketchManager.InsertSketch True
i = i + 1
Loop While i <= 30
End Sub

'Set Part = swApp.ActiveDoc
```

Figure 3-23 Visual Basic coding for generating the spline cutter profile

The fig.3.24 shows the respective simulated spline to cutter profile with line extending macro has been generated for locating the end points on the outline of the simulated spline to cutter profile with the help of API using the visual basics coding .After getting the end points on the simulated profile then curve through reference points has been used for generating the outline final profile as shown in fig.3.25.

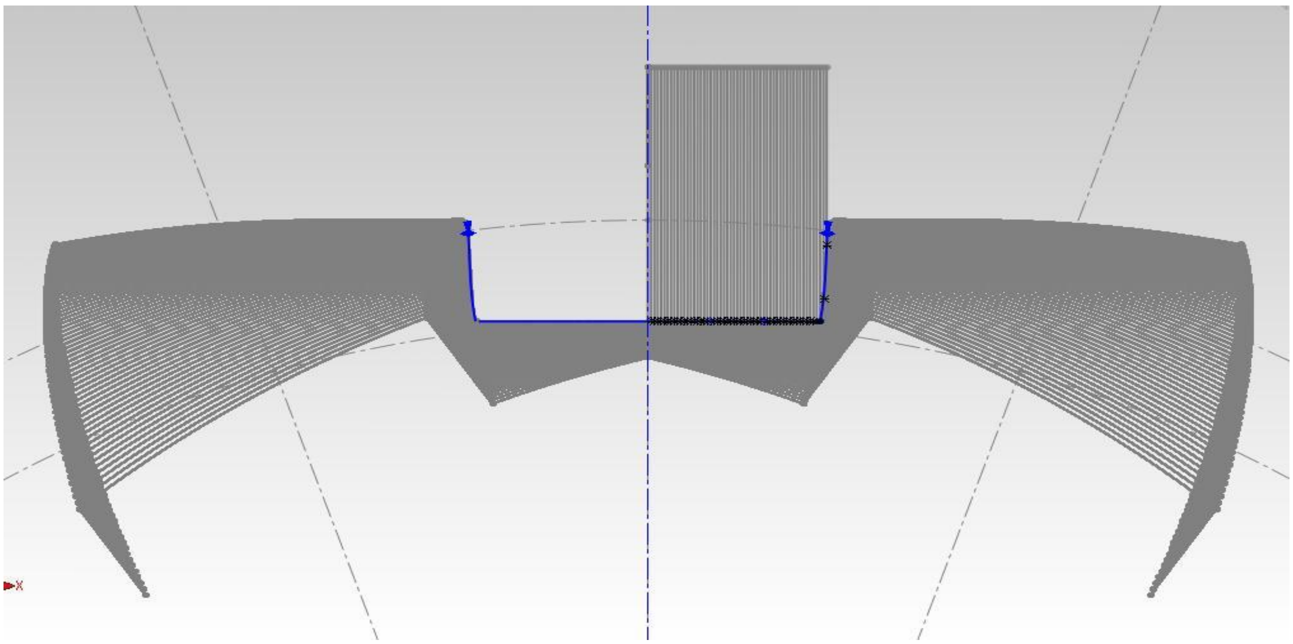

Figure 3-24 Tracing the spline cutter profile

Drawing format:

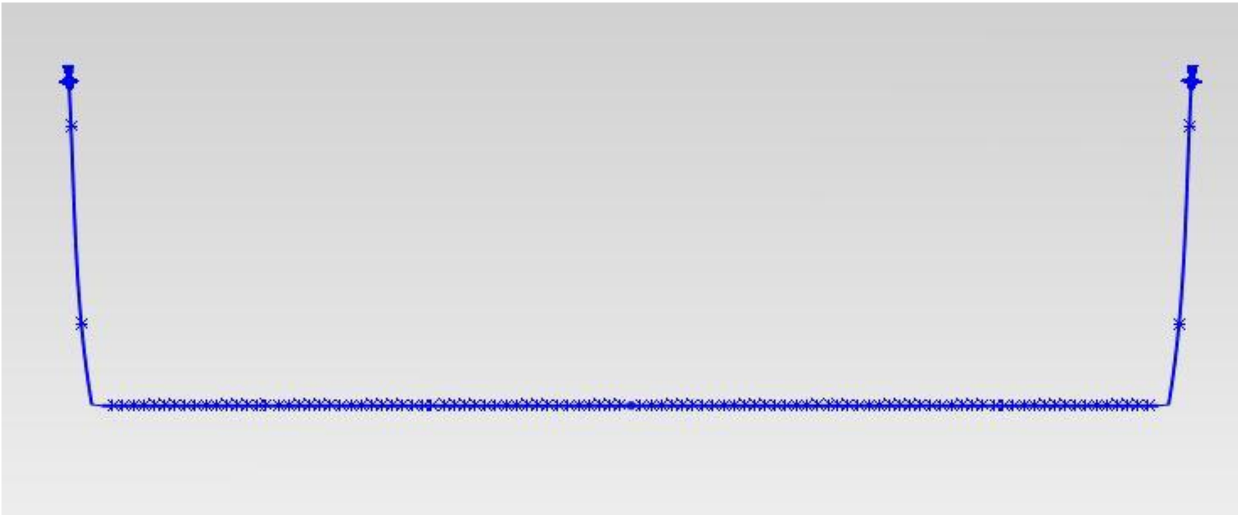

Figure 3-25 Drawing of spline cutter profile

3.3.4 Chamfered Spline hob cutter profile:

Generating of spline hob cutter in Solidworks software are done by relating the equation between the various parameters of sprocket e.g.outside diameter,root diameter,width of spline,number of spline,pitch circle diameter.As shown in fig.3.26.

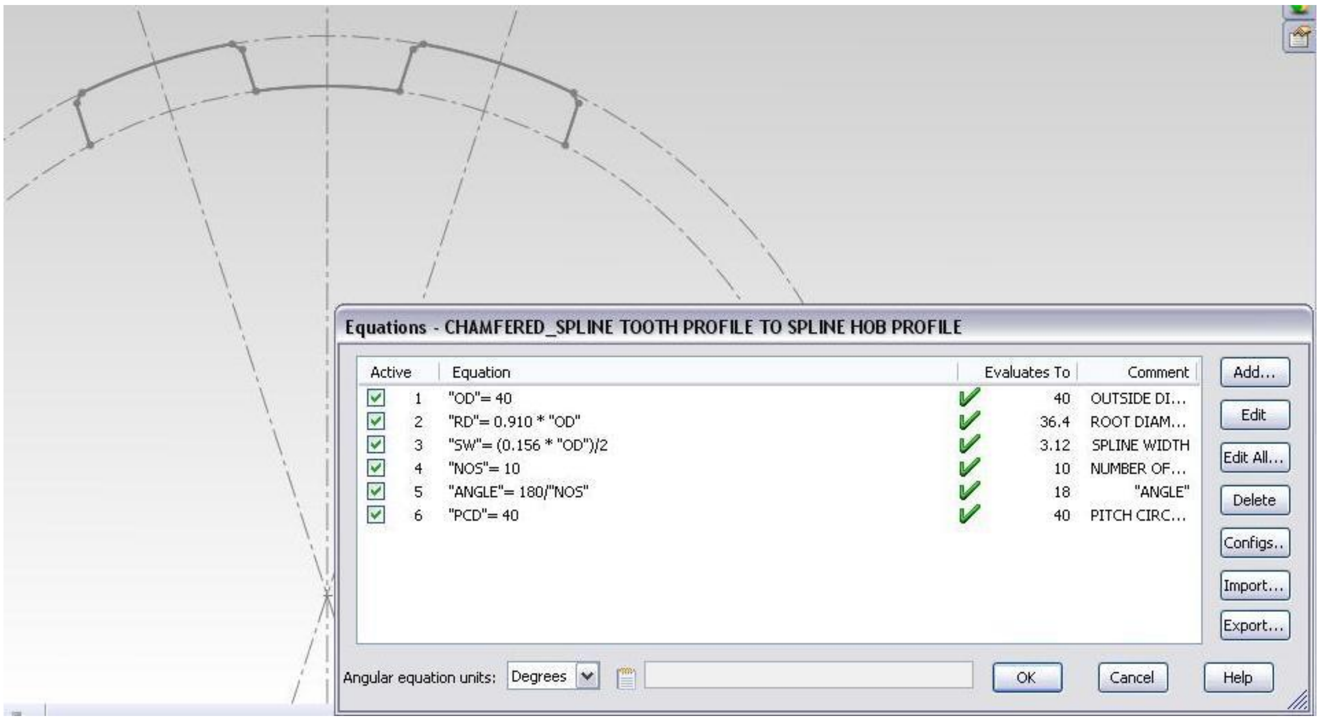

Figure 3-26 Sketch of chamfered spline Profile

After simulating the respective chamfered spline profile as shown in fig.3.26 by rotating about z – axis at some angle and translating from right plane at some distance will give the cutter profile from the chamfered spline profile showing the reverse simulation of machining process as shown in fig.3.27 has been done with the help of API (Application Program Interface) using the visual basics coding as shown in fig.3.28.

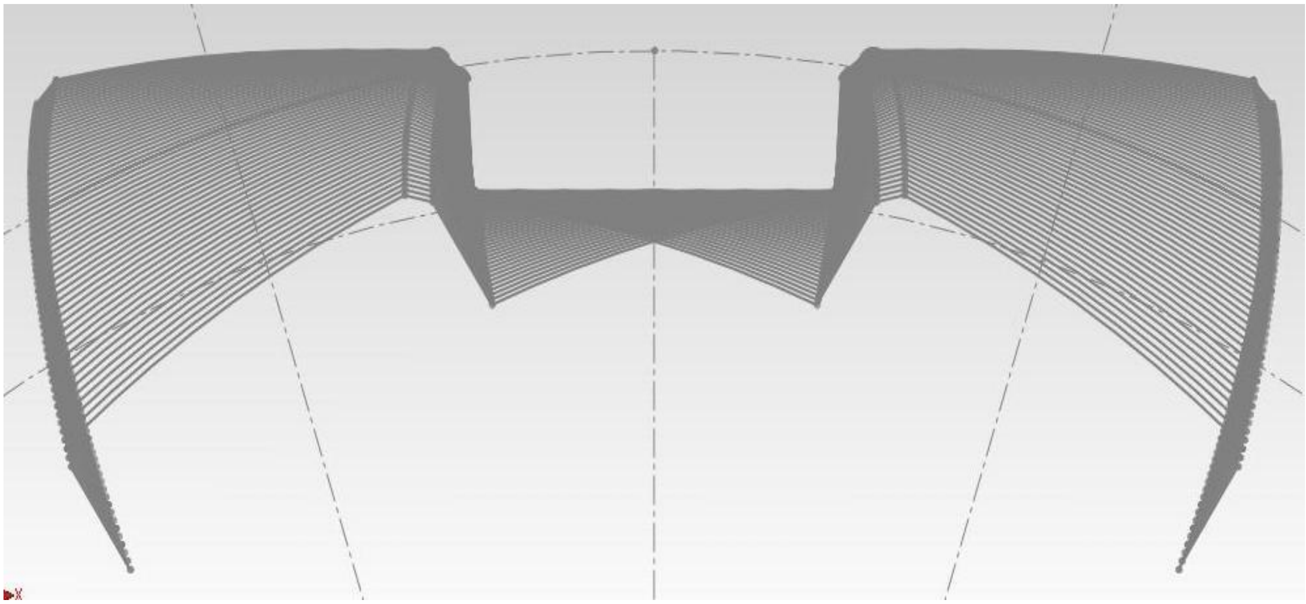

Figure 3-27 Shows the steps of making chamfered spline cutter profile by rotating the chamfered spline profile at certain angle with translate motion

```
Dim swApp As Object

Dim Part As Object
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long
Dim i As Integer

Sub main()
i = 0

Set swApp = _
Application.SldWorks

Set Part = swApp.ActiveDoc
Do
boolstatus = Part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
Part.SketchManager.InsertSketch True
boolstatus = Part.Extension.SelectByID2("Line34@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", -0.00872950542859
boolstatus = Part.Extension.SelectByID2("Line32@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", -0.00888423342481
boolstatus = Part.Extension.SelectByID2("Arc9@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", -0.007185542209053,
boolstatus = Part.Extension.SelectByID2("Line33@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", -0.00341103509565
boolstatus = Part.Extension.SelectByID2("Line35@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", -0.00286818232072
boolstatus = Part.Extension.SelectByID2("Arc8@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", -0.001286293269392,
boolstatus = Part.Extension.SelectByID2("Line25@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.0026776692623,
boolstatus = Part.DeSelectByID("Line25@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.003060123747672, 0.01951
boolstatus = Part.Extension.SelectByID2("Line25@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.00291533771013З
boolstatus = Part.DeSelectByID("Line25@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.003060123747672, 0.01951
boolstatus = Part.Extension.SelectByID2("Line25@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.002882022097011
boolstatus = Part.DeSelectByID("Line25@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.003060123747672, 0.01951
boolstatus = Part.Extension.SelectByID2("Arc6@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.004885421785779,
boolstatus = Part.Extension.SelectByID2("Line25@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.00291087855413б
boolstatus = Part.Extension.SelectByID2("Line30@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.003430872497947
boolstatus = Part.Extension.SelectByID2("Line27@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.0088123438823б,
boolstatus = Part.Extension.SelectByID2("Line31@CHAMFERED SPLINE TOOTH PROFILE", "EXTSKETCHSEGMENT", 0.00884282288395б
boolstatus = Part.SketchOffsetEntities2(0, False, True)
Part.ClearSelection2 True
Part.ClearSelection2 True
boolstatus = Part.Extension.SketchBoxSelect("-0.014950", "0.022143", "0.000000", "0.013622", "0.013684", "0.000000")
Part.Extension.RotateOrCopy False, 1, False, 0, 0, 0, 0, 0, 1, 0.00872664626 * i
Part.Extension.MoveOrCopy False, 1, False, 0, 0, 0, 0.00016 * i, 0, 0
Part.SketchManager.InsertSketch True
i = i + 1
Loop While i <= 30
End Sub
```

Figure 3-28 Visual Basic coding for generating the chamfered spline cutter profile

The fig.3.29 shows the respective simulated chamfered spline to cutter profile with line extending macro has been generated for locating the end points on the outline of the simulated chamfered spline to cutter profile with the help of API using the visual basics coding .After getting the end points on the simulated profile then curve through reference points has been used for generating the outline final profile as shown in fig.3.30.

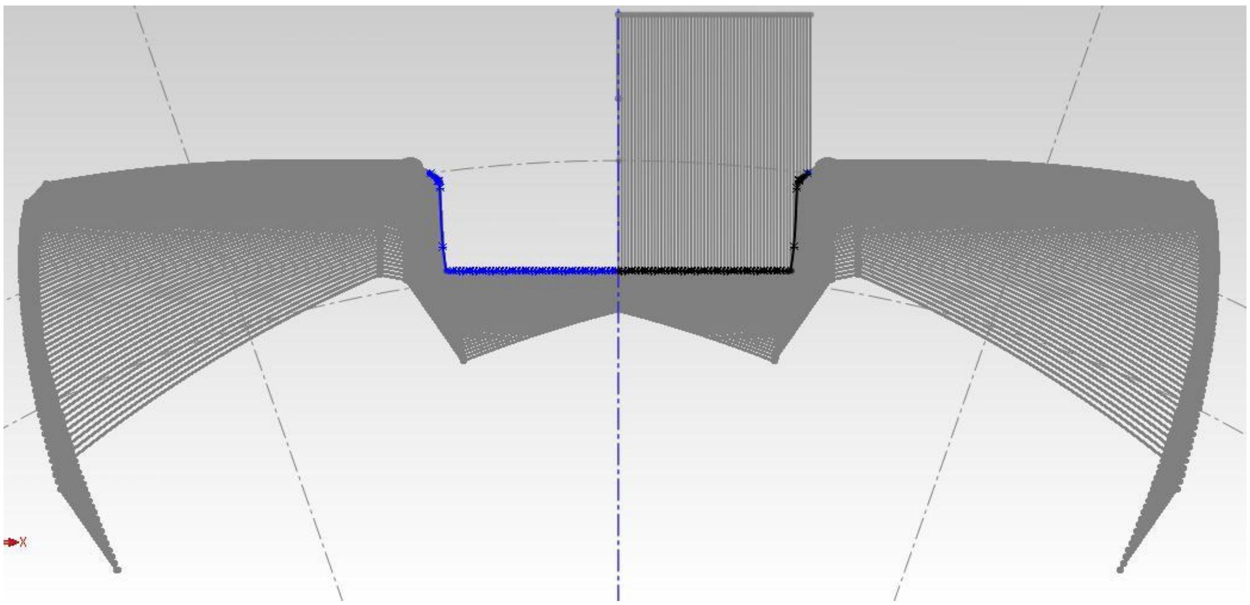

Figure 3-29 Tracing the chamfered spline cutter profile

Drawing format:

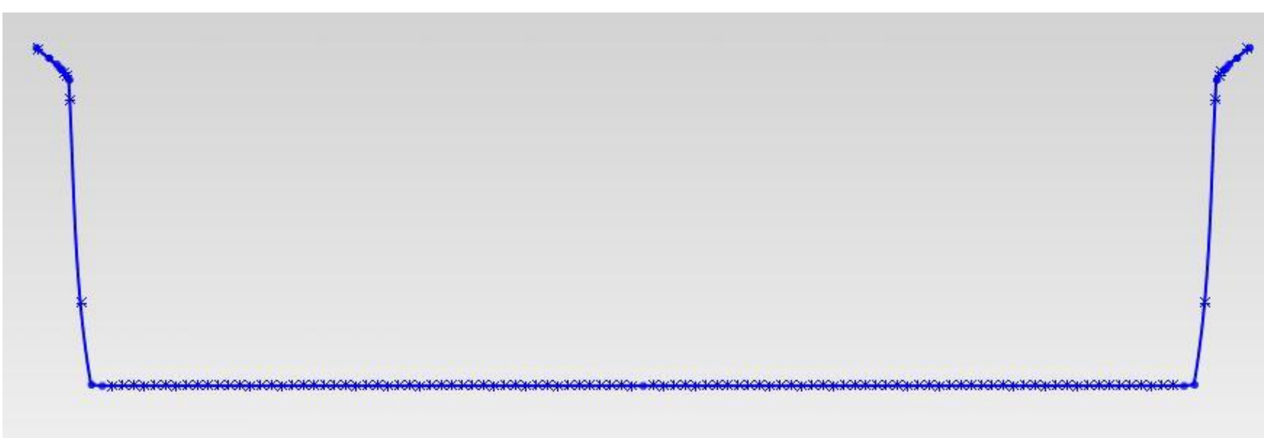

Figure 3-30 Drawing of chamfered spline cutter profile

CHAPTER-4

CONCLUSION AND FURTHER SCOPE OF WORK

4.1 Conclusion

This method is automated using Parametric CAD software by defining proper relative relations during the generation of profile which provides accuracy without any human errors with the help of API (Application Program Interface) by generating the macros for shifting the profiles which actual shows the reverse simulation of the machining process. Further another macros has been created for tracing the outline of the respective profiles which shows the final cutter profiles. This method replaces a designer, as only the parameters defining the job need to input, the generation of the profile is automated which leads to reduces the human inaccuracies, increases productivity and saves time also.

4.2 Further Scope of Work

1. Further this method can be extended by making the use of API (Application Program Interface) in solidworks which develop the window dialog box from which the user can directly input the parameters for the generation of cutter profiles.
2. The drawing format of various existing cutter profiles can be get in the form of PDF (Portable Document Format) with the use of API (Application Program Interface) which can be use for the manufacturing point of view.
3. Validation and comparison of CAD generated existing cutter profiles with the use of API (Application Program Interface) can be done for comparing the accuracy between the existing cutter profiles like gear cutter, sprocket cutter and spline cutter and the manual method which is presently used in industries for the generation of cutter profiles. To achieve and enhance the productivity with the help of CAD automation.

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