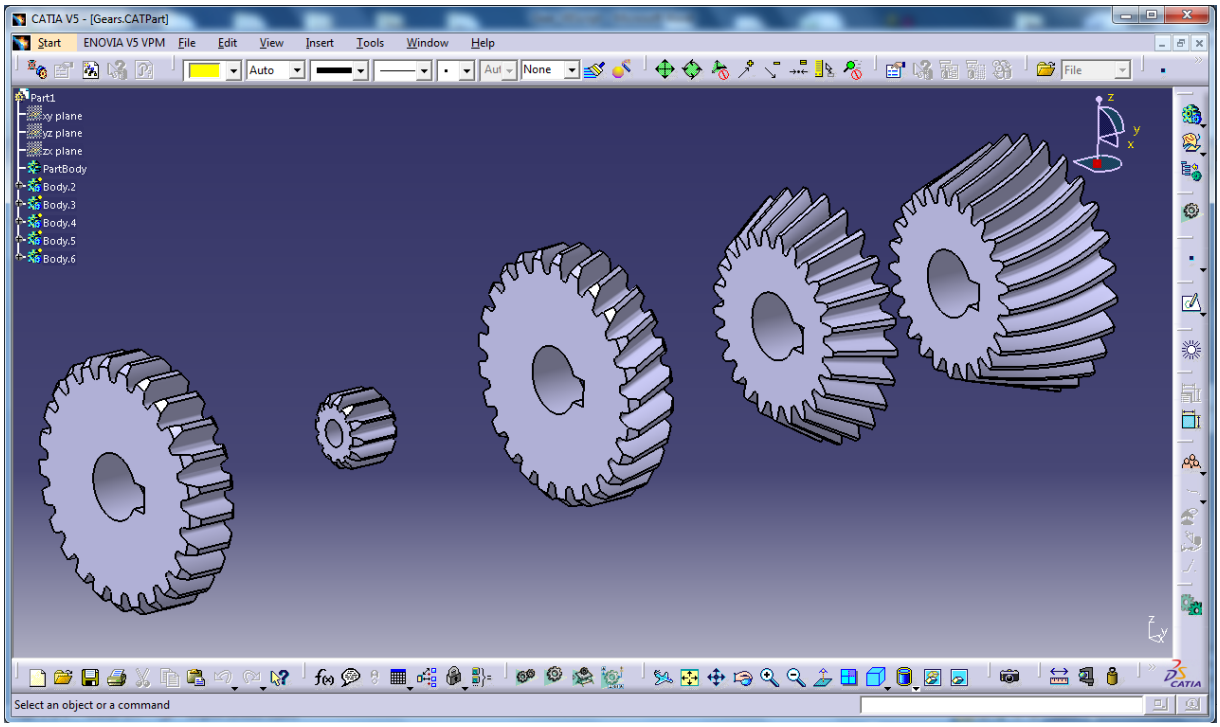
VB Scripting for CATIA V5:
How to Model Gears with VBA User Form

PREVIEW VERSION



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VB Scripting for CATIA V5: How to Model Gears with VBA User Form **PREVIEW VERSION**

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A prerequisite for this guide is to know the basics of CATIA, programming by Visual Basic, and VBScript for CATIA. To start learning programming for CATIA V5 from scratch, please read **VB SCRIPTING FOR CATIA V5** by ***Emmett Ross.***

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Introduction

A good practice to take your CATIA programming skills to the next level is to model a complex example like a gear. In this guide, you'll learn how to develop a CATVBA program to model different types of gears in CATIA V5.

This guide is divided into two main steps:

1. Drawing gears in CATIA V5 using sketcher
2. Programming in CATIA V5 step-by-step including VBA user form

In this case, step one is as important as step two, because there are many ways to model a gear but some of them are not good and are inefficient. This is especially evident in Helical Gears where sometimes they create errors in CATIA or the file size and/or time of creation is very high.

Before starting these two steps, let's briefly review the structure of this program.

Structure of this VB Program

This program consists of one Module (Module 1) and one Form (FrmGear). Module 1 has two subroutines: *CATMain()* to show Form and *HidePlanes* to Hide three main planes (X, Y and Z). Subroutine *HidePlanes* is not necessary to draw gears, it's just for better viewing of the gears.

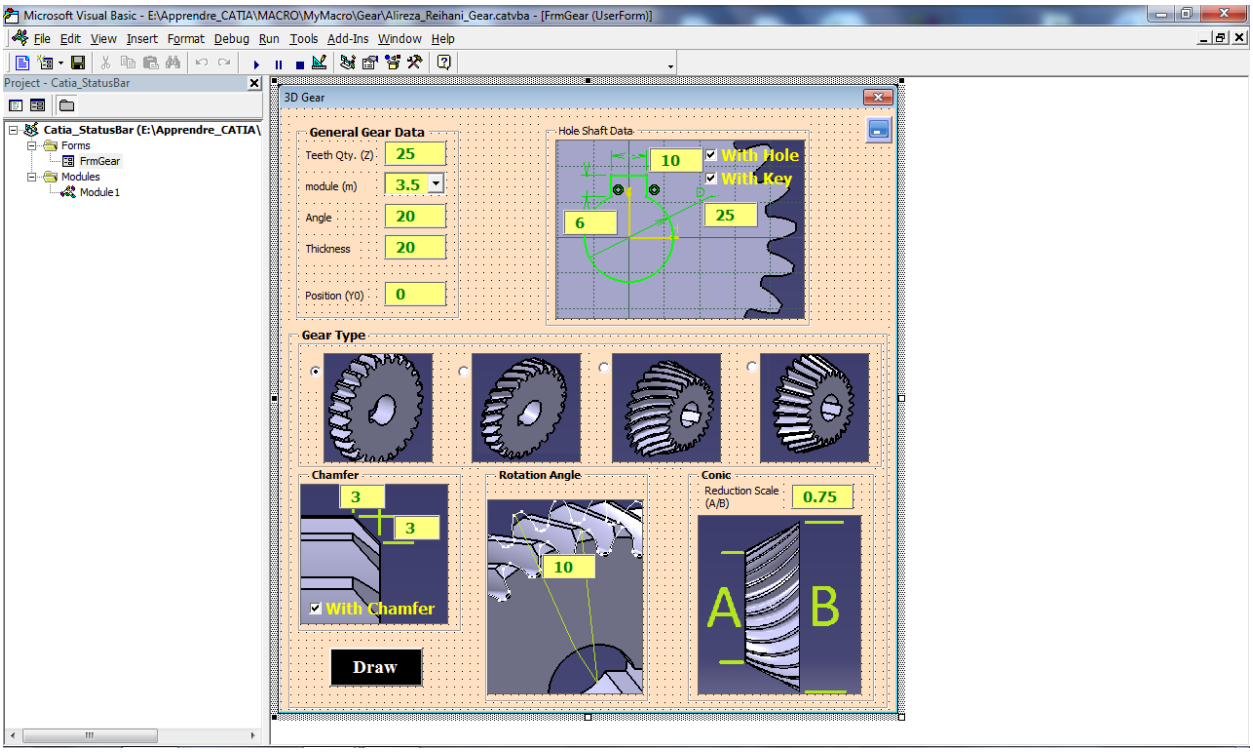


Fig 1: Project explorer and Program window in design mode

The UserForm consists of seven subroutine and two functions. At first, Subroutine common creates Full Gear Profile, and then, depending on the Gear Type, one of four subroutines that draws the desired gear type:

- **Gear 1 Type: Spur**
- **Gear 2 Type: Helical**
- **Gear 3 Type: Straight Bevel**
- **Gear 4 Type: Spiral Bevel**

Two functions **Xn** and **Yp** are used to calculate coordinates of intersection of two circles (to draw Gear profile). Subroutines **HoleKey** and **Chamfer** draw Hole of Shaft and Chamfer.

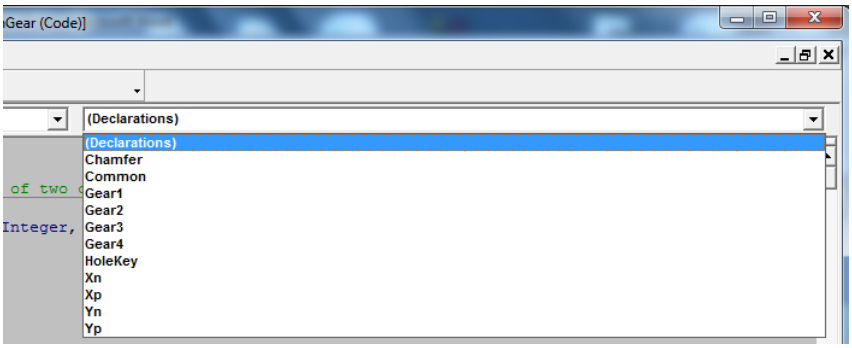


Fig 2: Subroutines of FrmGear

Step 1: Drawing Gear Types in CATIA

Before starting a Sketch of the Tooth Profile (manually or by programming), it's necessary to have equations for drawing a gear tooth profile.

- Ro = m * Z / 2 where m=module and Z=number of gear teeth
- Rt = 0.94 * Ro
- Rd = Ro - 1.25 * m ; Dedendum
- Ru = Ro + m ; Addendum
- Rf = 0.35 * m ; (fillet; you can get the value from the user)

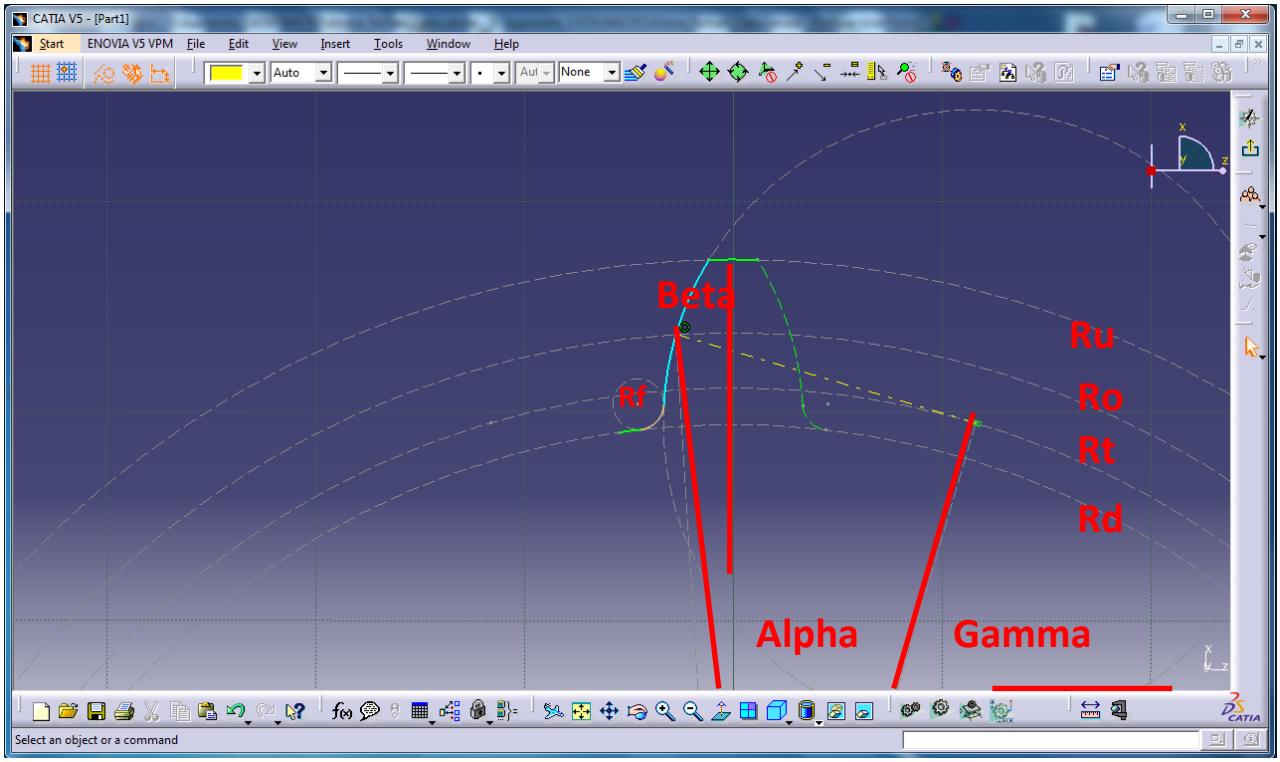


Fig 1-1: Gear profile

Note 1: Normally to draw a tooth profile, you must draw circles and lines, then trim them. This can be very difficult to do by programming so use intersection points of two circles by these equations:

$$\begin{cases} x = \dfrac{L*(X_2-X_1)}{d} \pm \dfrac{h*(Y_2-Y_1)}{d} + X_1 \\ y = \dfrac{L*(Y_2-Y_1)}{d} \mp \dfrac{h*(X_2-X_1)}{d} + Y_1 \end{cases}$$

$$\begin{cases} d = \sqrt{(X_1-X_2)^2+(Y_1-Y_2)^2} \\ L = \dfrac{R_1^2-R_2^2+d^2}{2*d} \\ h = \sqrt{R_1^2-L^2} \end{cases}$$

where x,y are coordinates of intersection points, X1,Y1 are coordinates of center point of first circle with radius=R1 and X2,Y2 are coordinates of center point of second circle with radius=R2.

The fillets will also be drawn in sketch rather than using the fillet tool because it will be easier to do programmatically.

In this guide, you will draw the Gear Sketch in a plane parallel to XZ plane. As a practice, you can develop this program and then draw a gear in each plane.

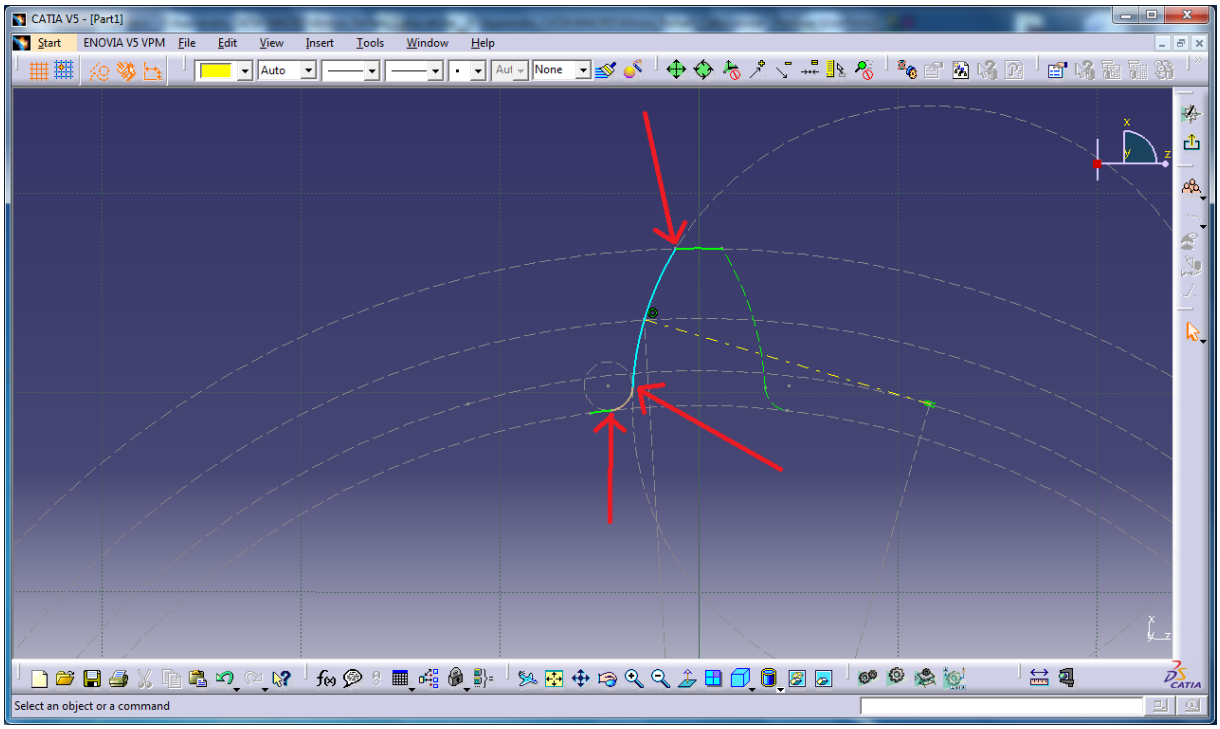


Fig 1-2: Tooth Profile drawn by finding intersection points

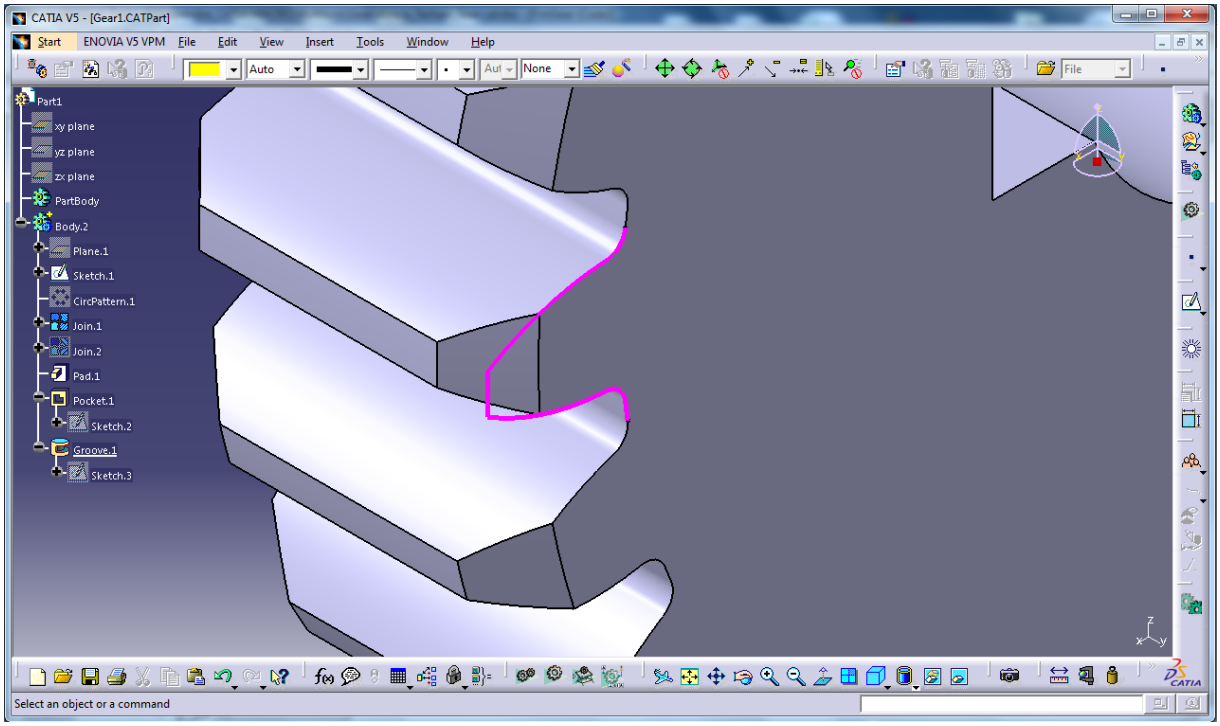


Fig 1-3: First Tooth Profile (Sketch)

Note 2: Drawing Full Gear Profile

For drawing full teeth profile, it doesn't appear there is a VB function to have **Circular Pattern** in Sketch ( ). Therefore, we will use Circular pattern in GSD ( ).

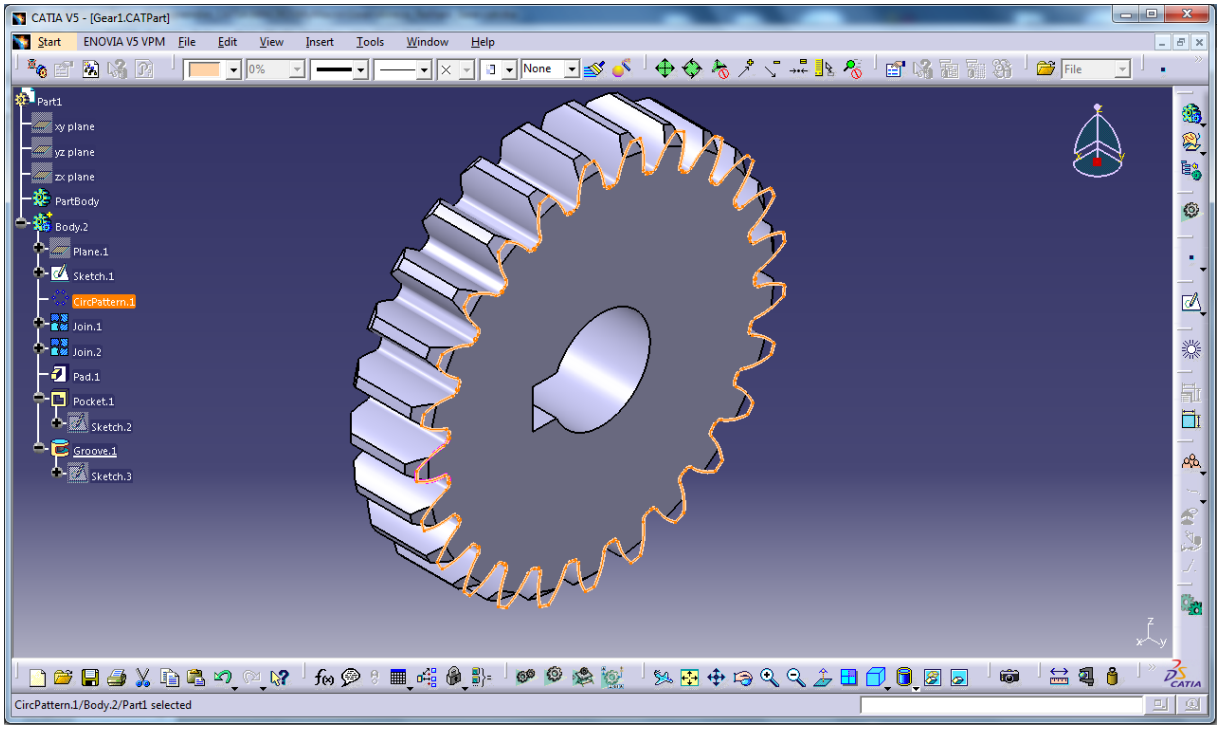


Fig 1-4: Full Teeth Profile and Create Pad for Gear Type 1

In gear types 2,3 and 4, Multi-section Solid creates the volume.

- **Gear 1 Type: Spur = PAD**
- **Gear 2 Type: Helical = Multi-section**
- **Gear 3 Type: Straight Bevel = Multi-section**
- **Gear 4 Type: Spiral Bevel = Multi-section**

In type 2 (helical gear); after Translating of full teeth profile; it must be rotated (Fig. 1-5).

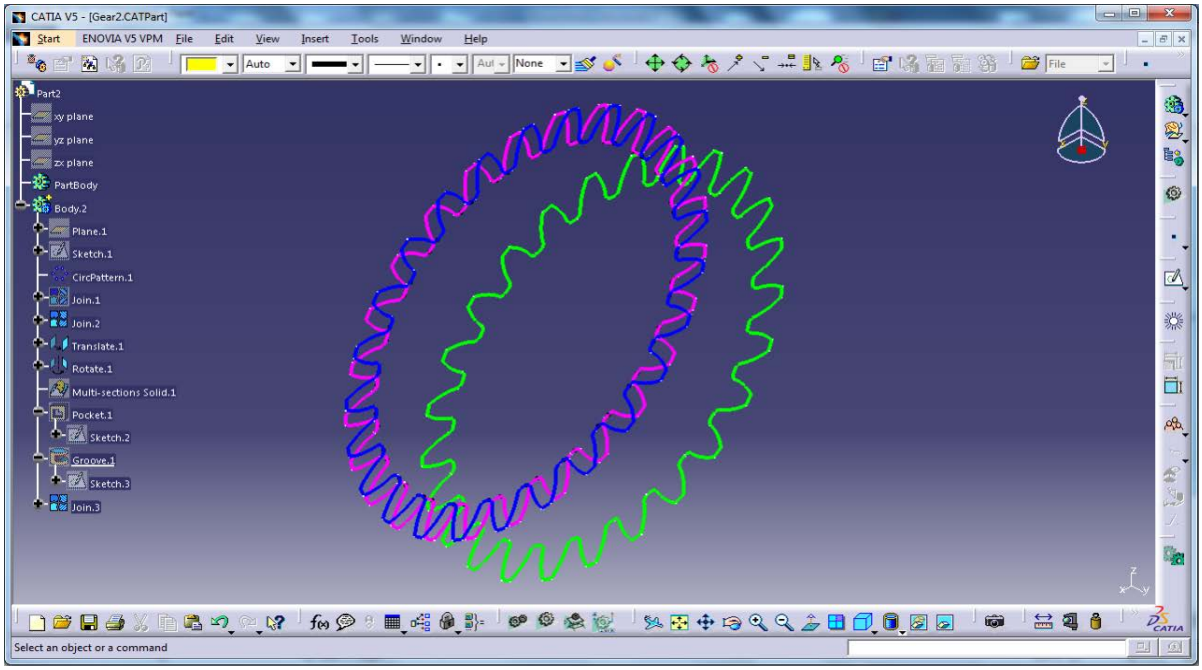


Fig 1-5: Profile Translated and Rotated

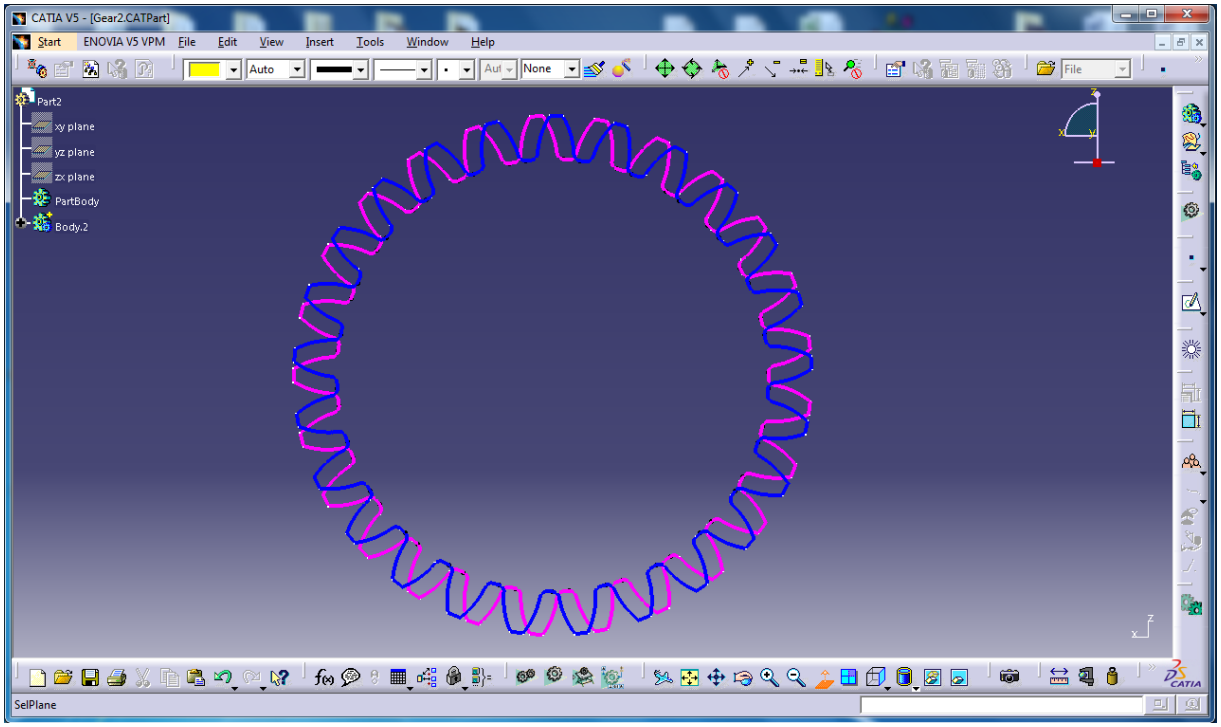


Fig 1-6: Profile Translated and Rotated in normal view

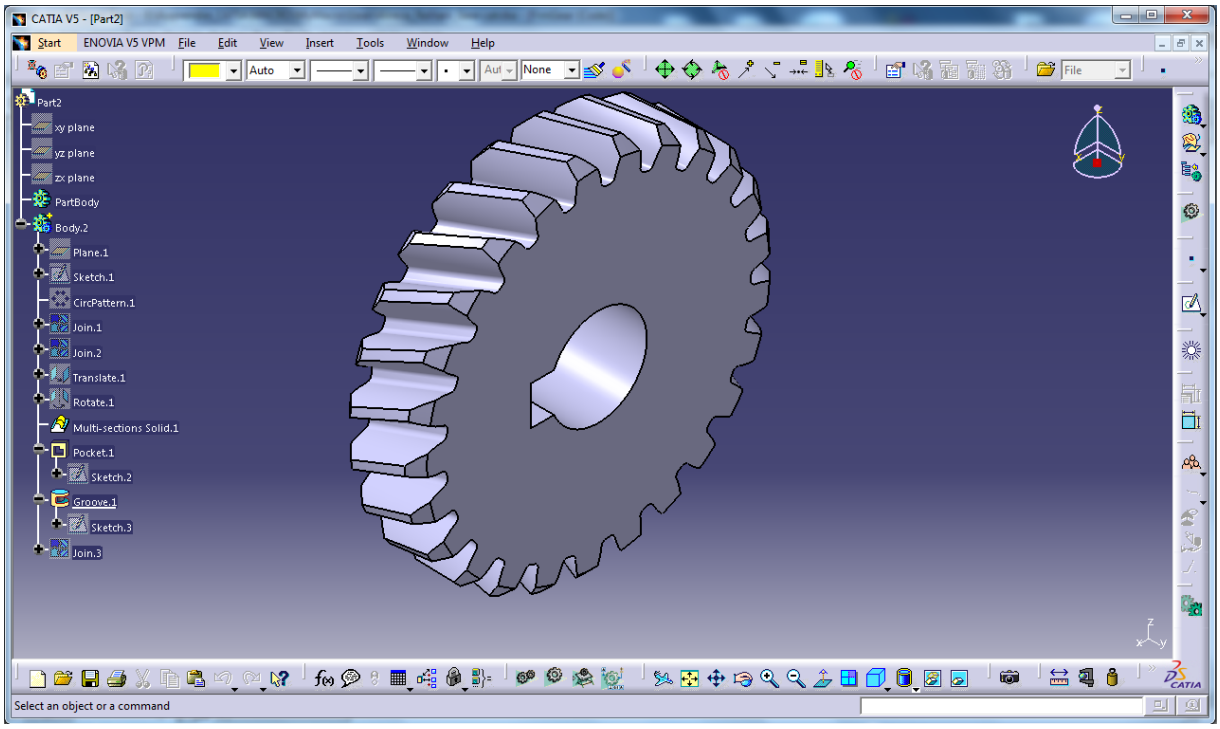


Fig 1-7: Gear Type 2

In type 3 (straight bevel); after Translating of full teeth profile; it must be scaled (Fig. 1-8).

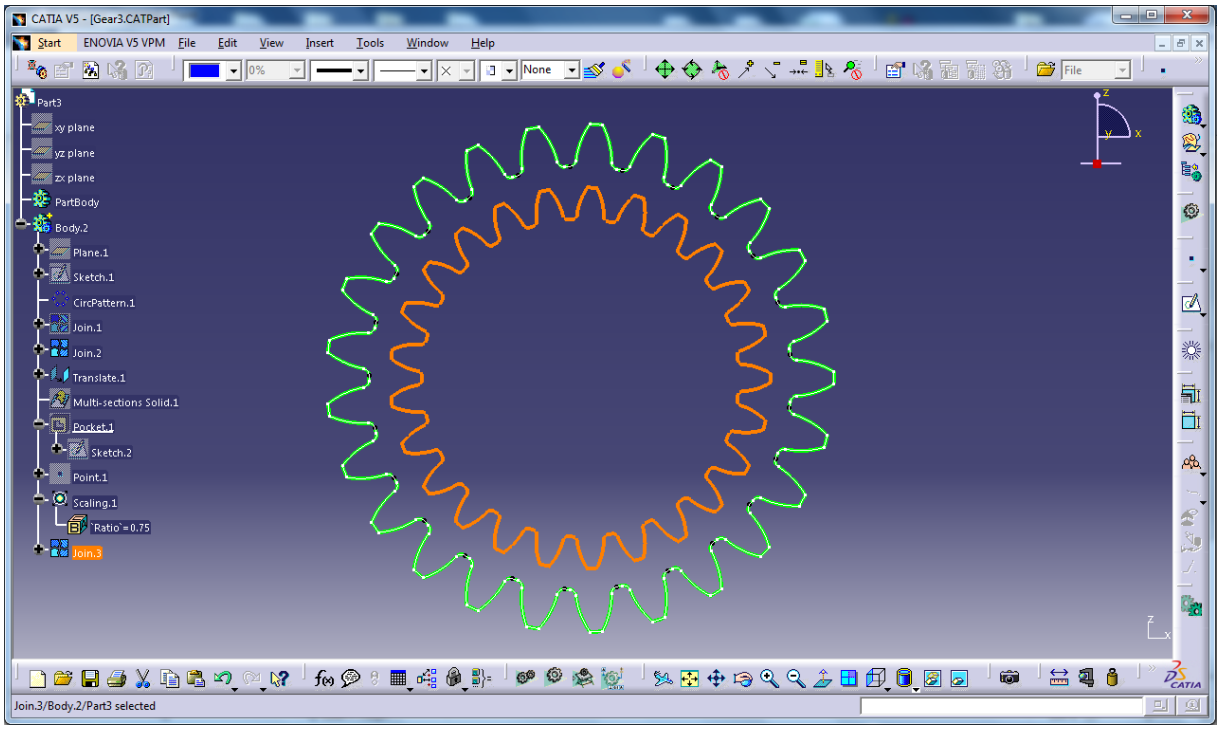


Fig 1-8: Profile Translated and Scaled in normal view

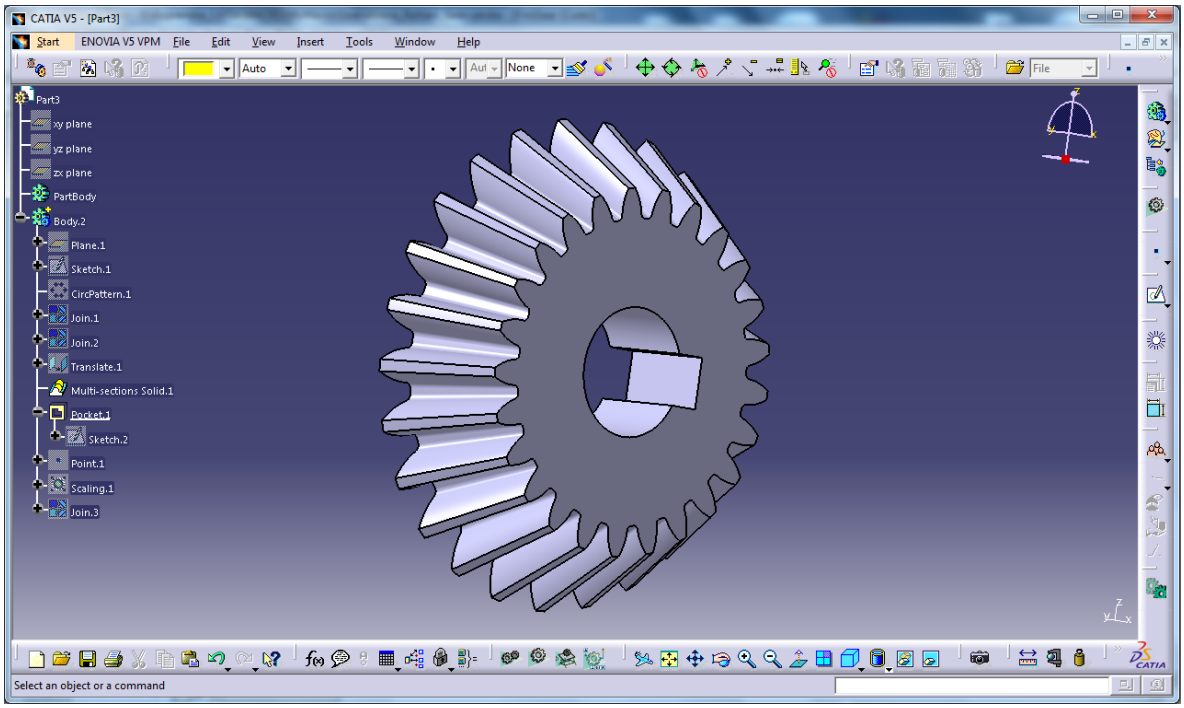


Fig 1-9: Gear Type 3 (straight bevel)

In type 4 (spiral bevel); after Translating of full teeth profile; it must be rotated and be scaled. Do it two times (Fig. 1-10).

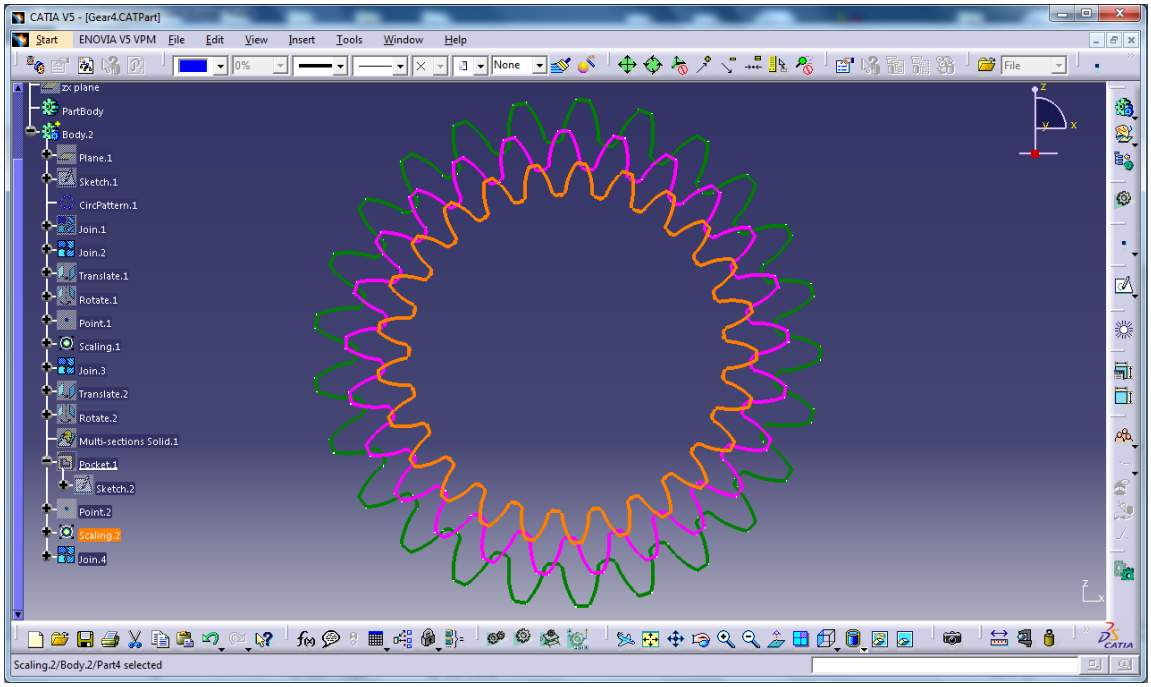


Fig 1-10: Profile Translated, Rotated and Scaled in normal view (two times)

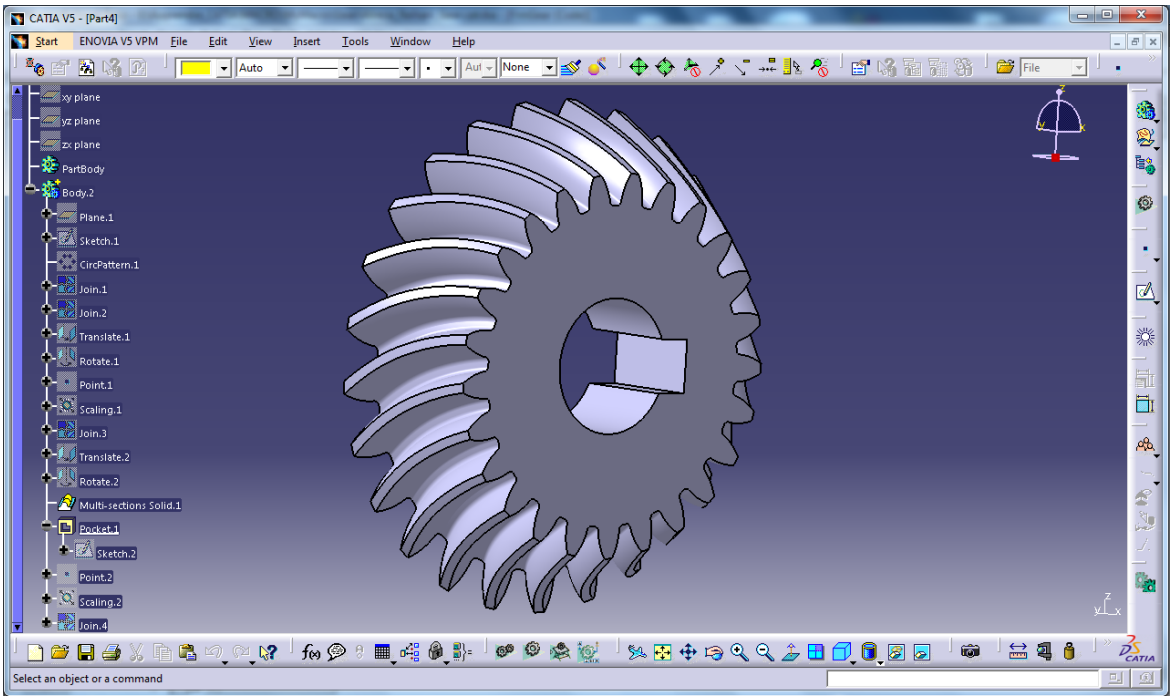


Fig 1-11: Gear Type 4 (spiral bevel)

We have just created four different types of gear. Now it's time to add automation so a user can change the size of gears through a user form.

Step 2: Programming in CATIA step by step

2-1) Getting Data and controlling them (sub: common)

2-1-1) Create Form in a new VBA project

You can get the data needed by using an InputBox but the more user friendly way is to use a TextBox, ComboBox, CheckBox combination in a Form like Fig 2-1-1

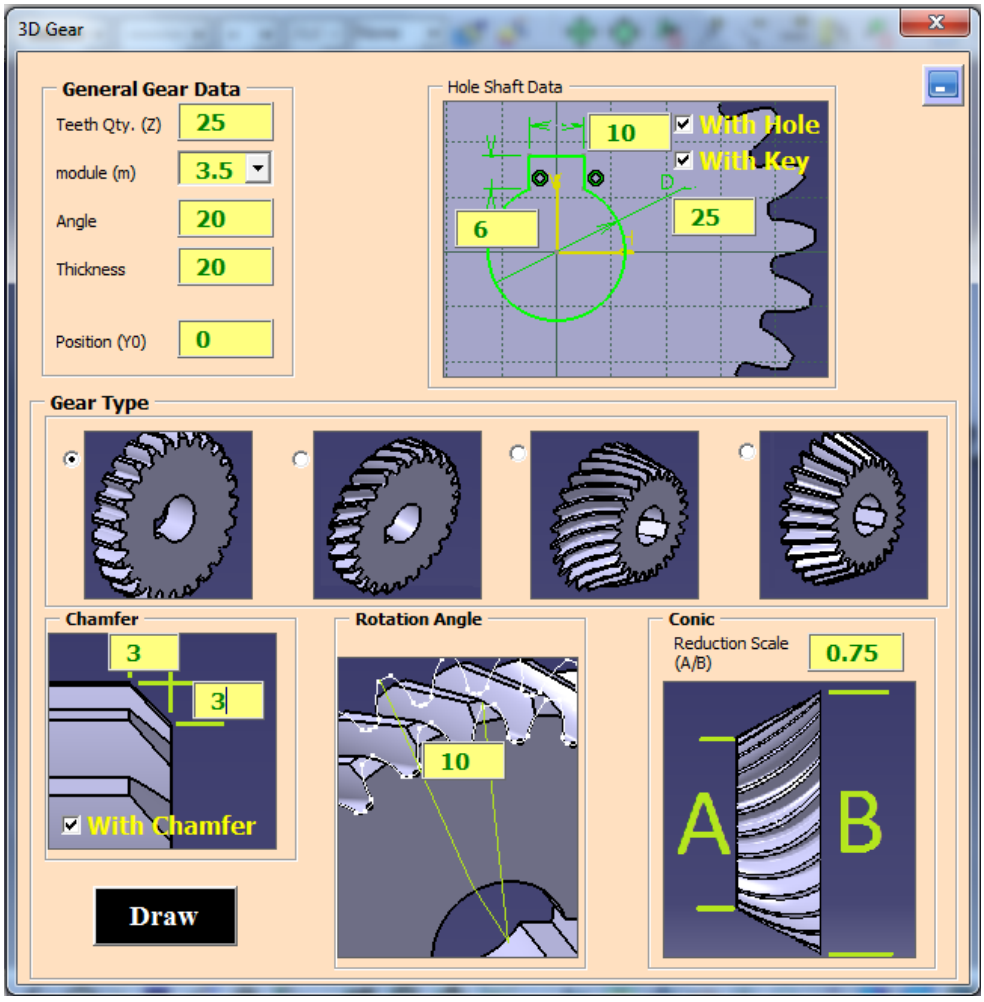


Fig 2-1-1: Form to get data for creating the gears to the specified size

For creating this User Form, we need to create a new program file. In CATIA, click on the Tools>Macro>Visual Basic Editor... or press Alt+F11 (Fig 2-1-2).

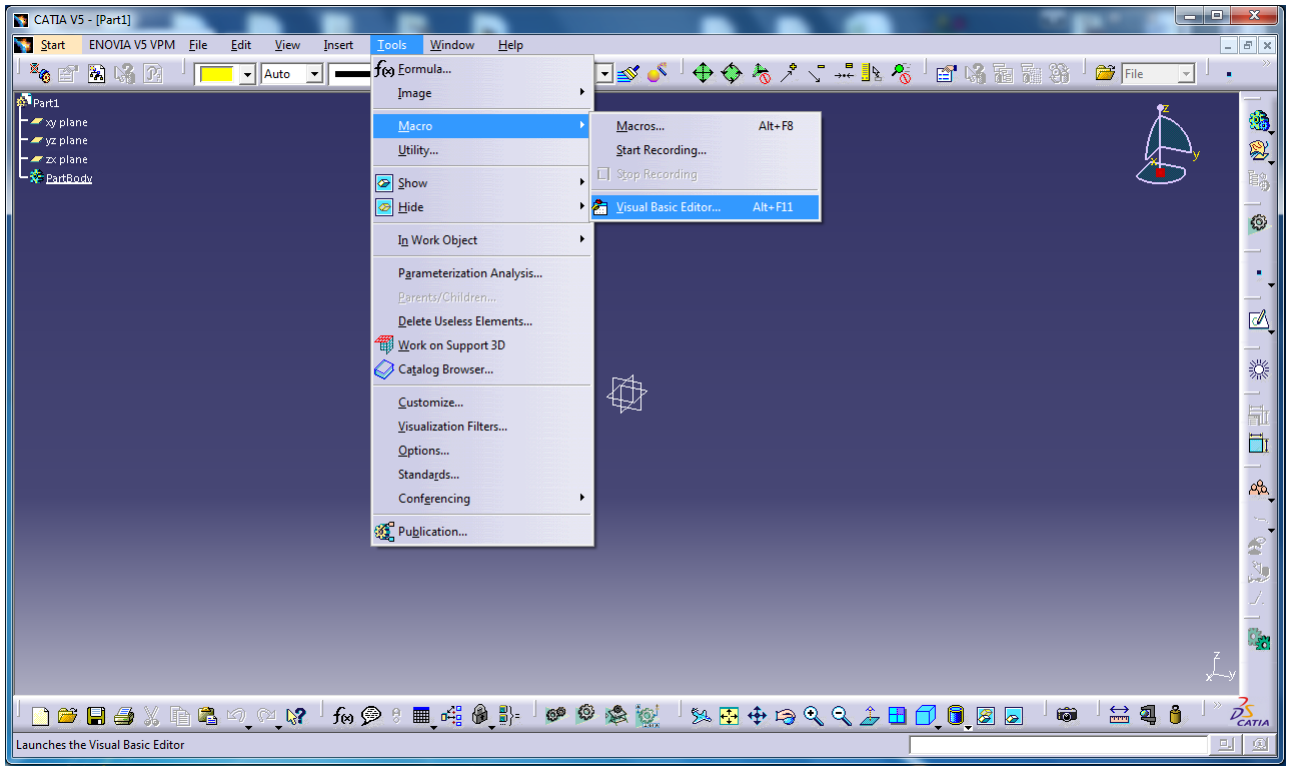


Fig 2-1-2: Enter the Visual Basic Editor

If there is not a loaded CATVBA file, one message appears (Fig 2-1-3).

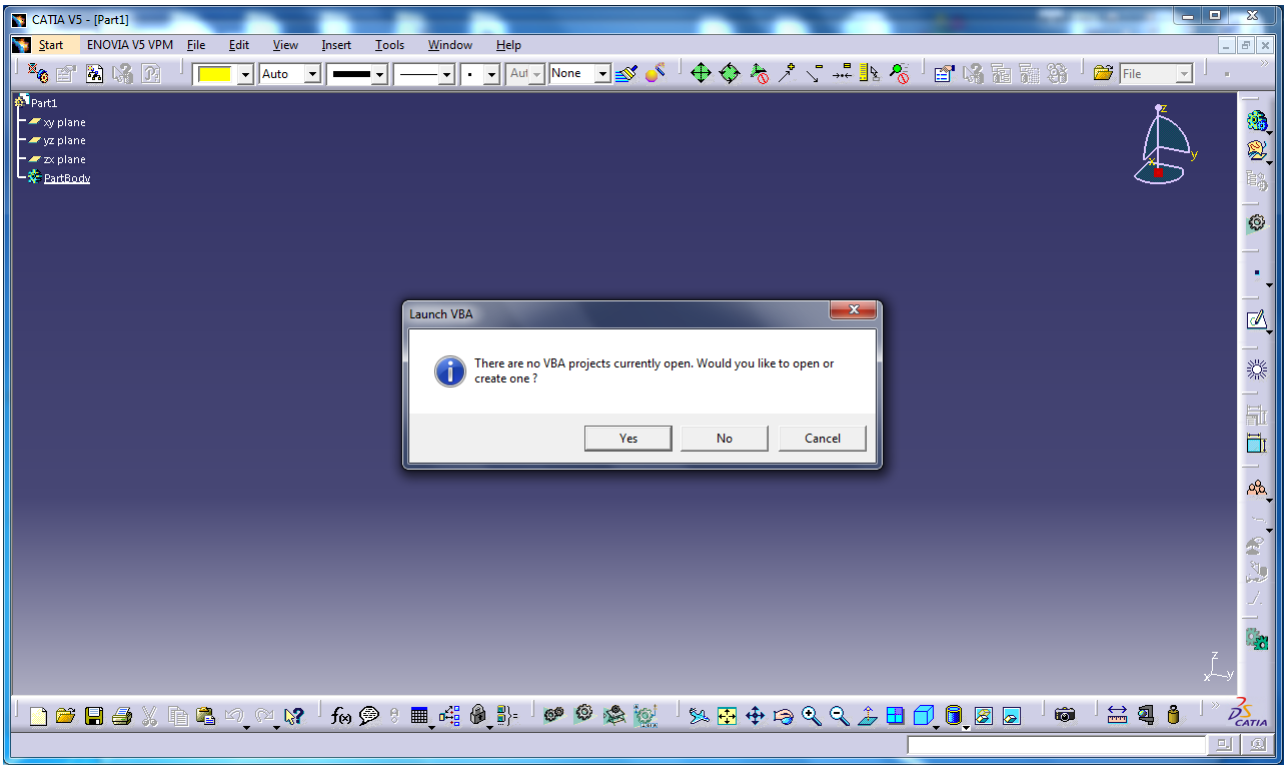


Fig 2-1-3: Launch VBA message

Press *Yes* to create one VBA project. Now *Macro libraries* window appears (Fig 2-1-4)

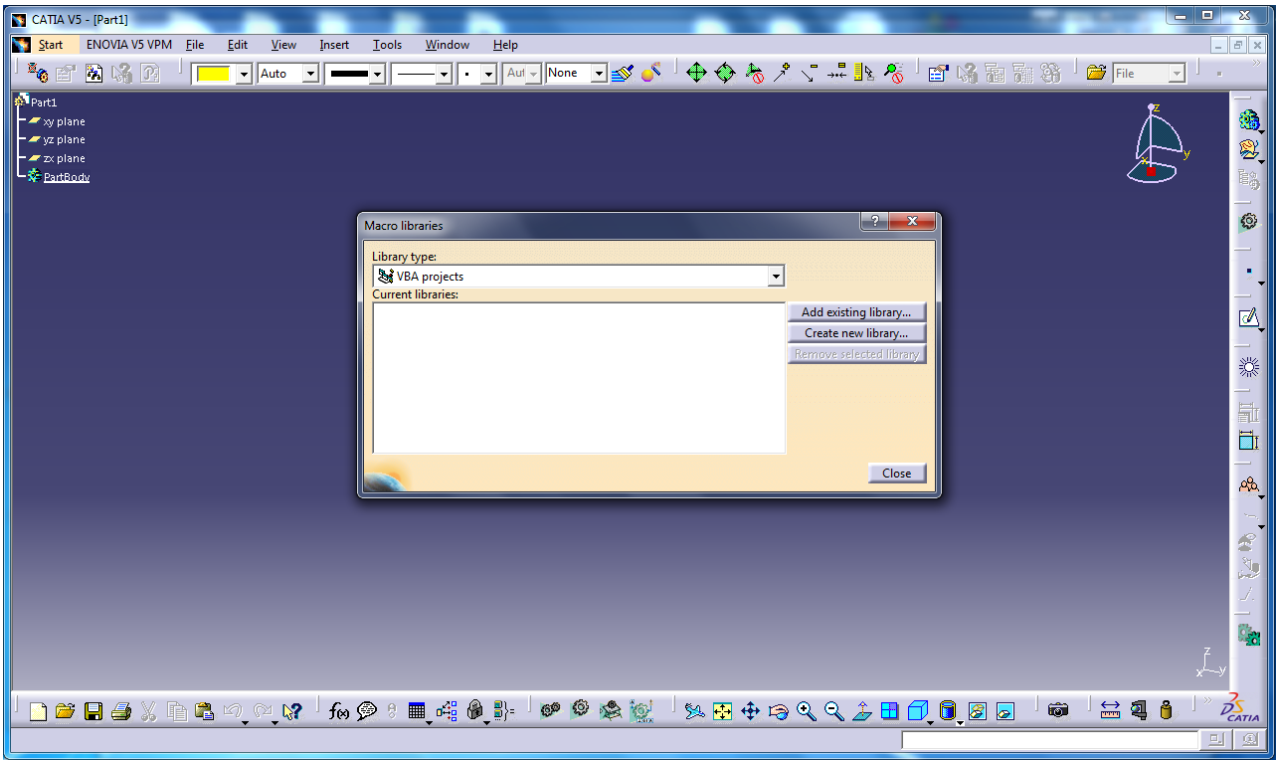


Fig 2-1-4) *Macro libraries* window

Click on *Create new library...* button (Library type must be *VBA Project* Fig 2-1-5). Now *Create a new VBA project* window appears (Fig. 2-1-5). Enter a path and name for your VBA project. (This path must be valid, otherwise an error message appears and CATIA doesn't help you to create these Folders.)

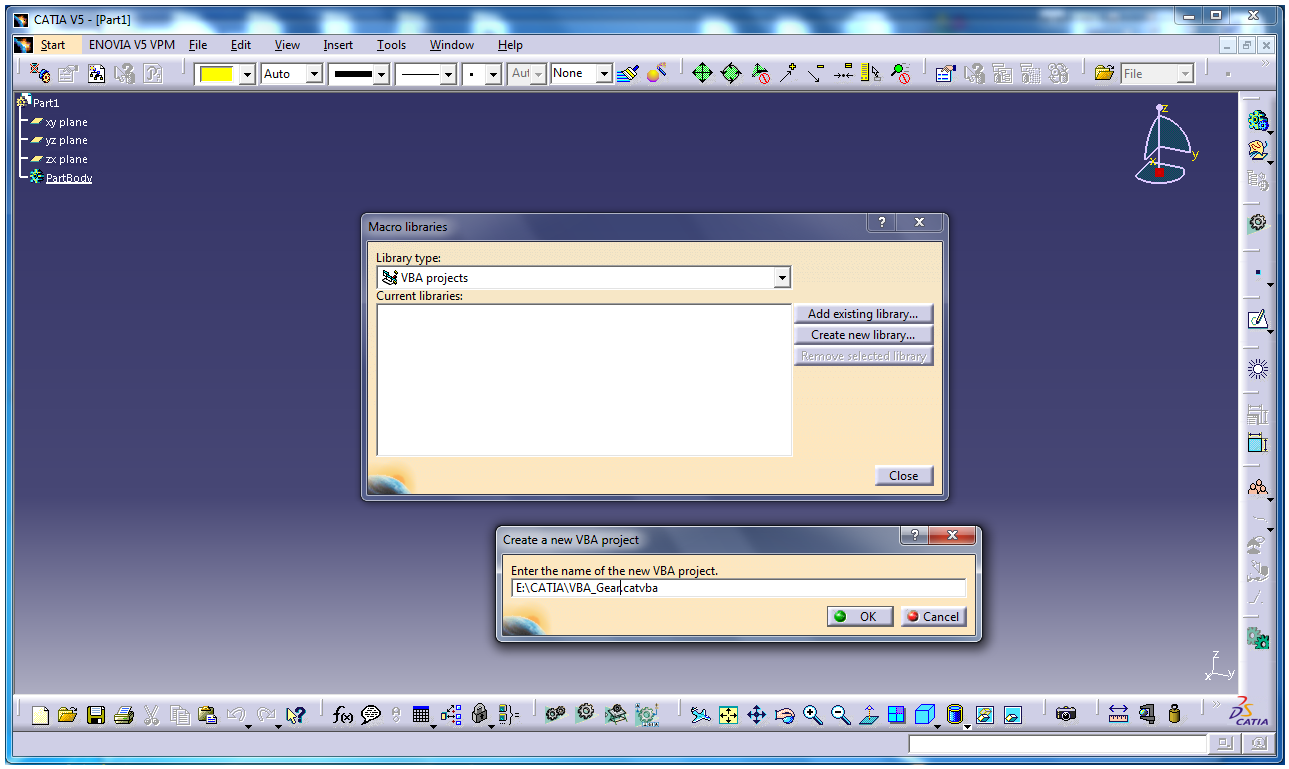


Fig 2-1-5) *Create a new VBA project* window

By clicking on the *Ok* button you will come back to *Macro libraries* window and you will see your new full path of new VBA project in this window (Fig 2-1-6).

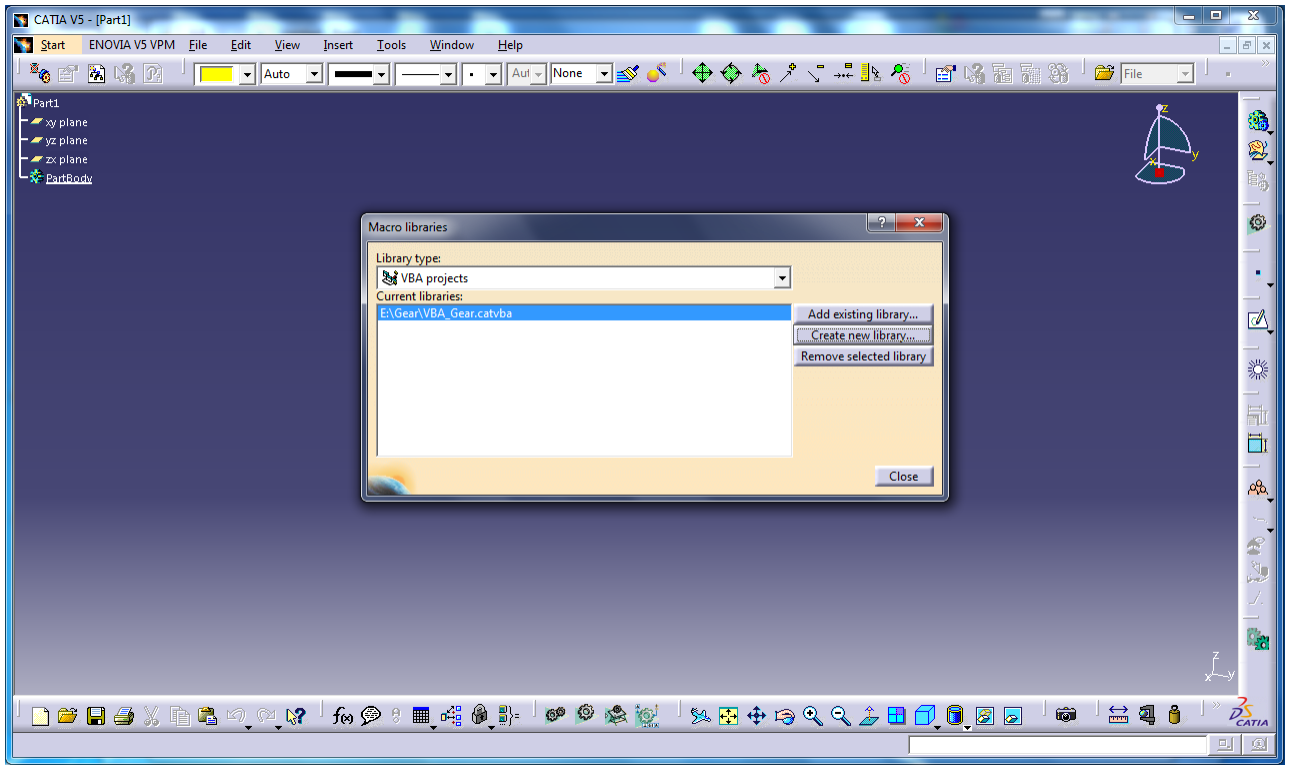


Fig 2-1-6: new VBA project in *Macro libraries* window

By clicking on *Close* button you will enter into the VBA environment (Fig 2-1-7)

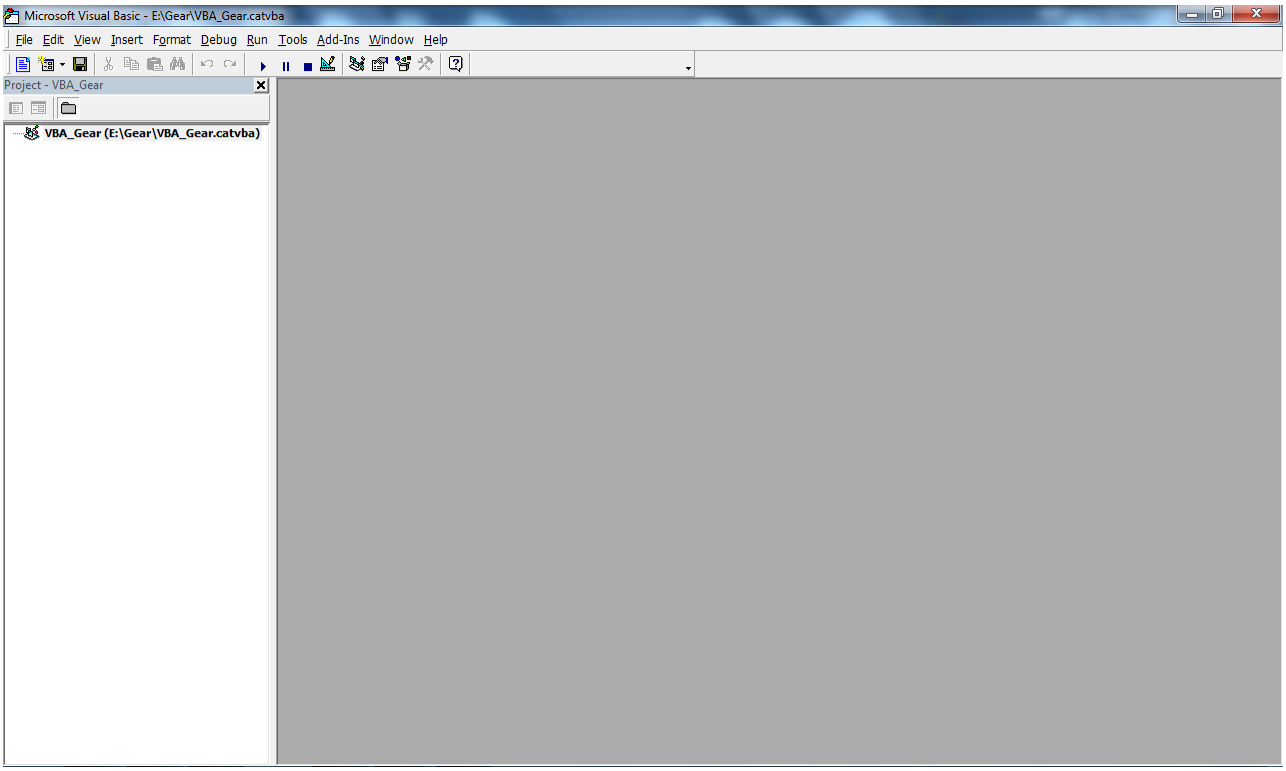


Fig 2-1-7) VBA environment

To create a Form, Right click on the Project window (left side in Fig 2-1-7) then click on *Insert>UserForm* (Fig 2-1-8) or use menu *Insert>UserForm* (Fig 2-1-9). New Form appears (Fig 2-1-10).

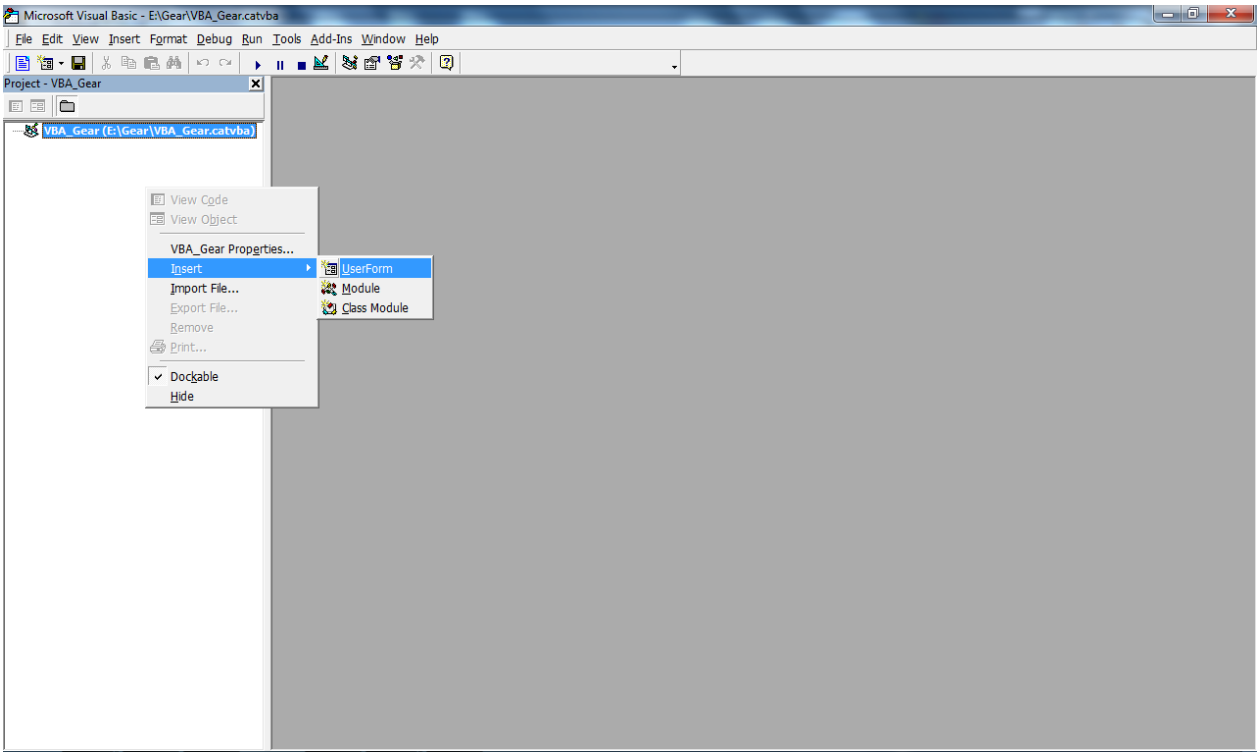


Fig 2-1-8: Add a new Form in your VBA project by right click

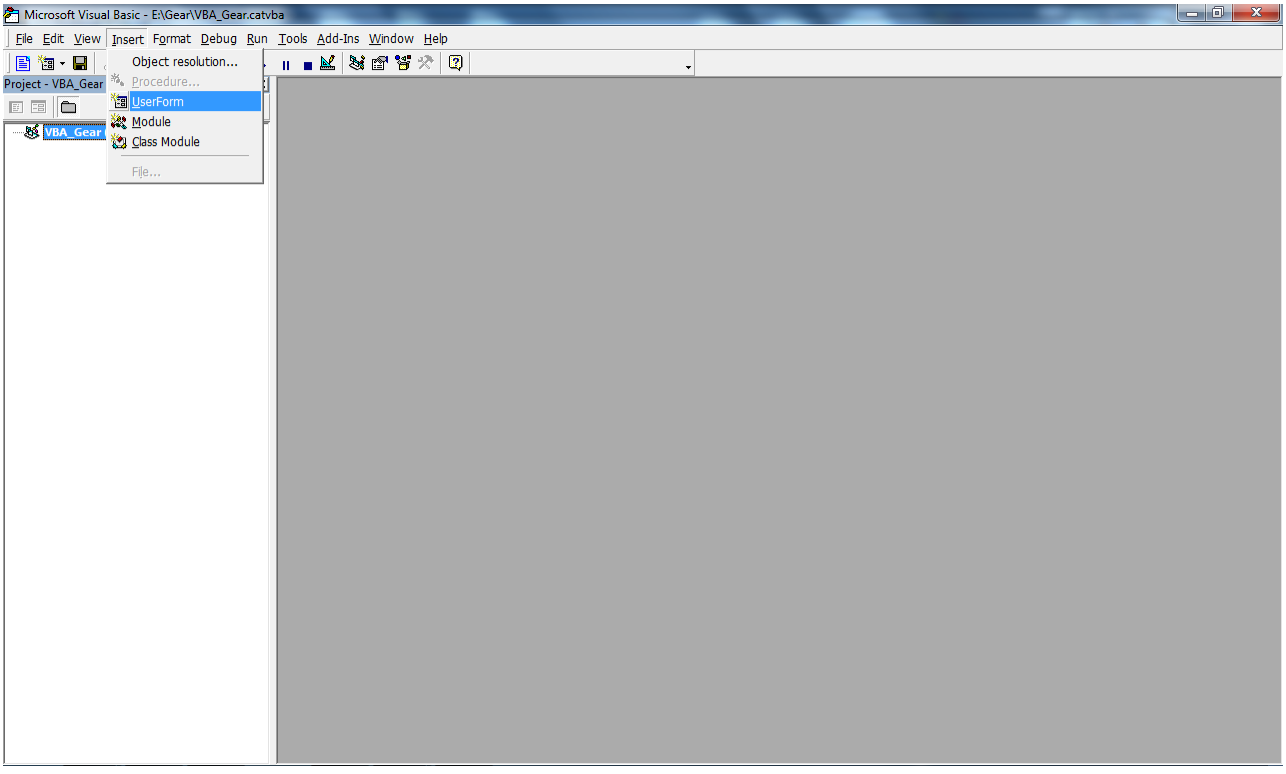


Fig 2-1-9: Add a new Form in your VBA project by menu

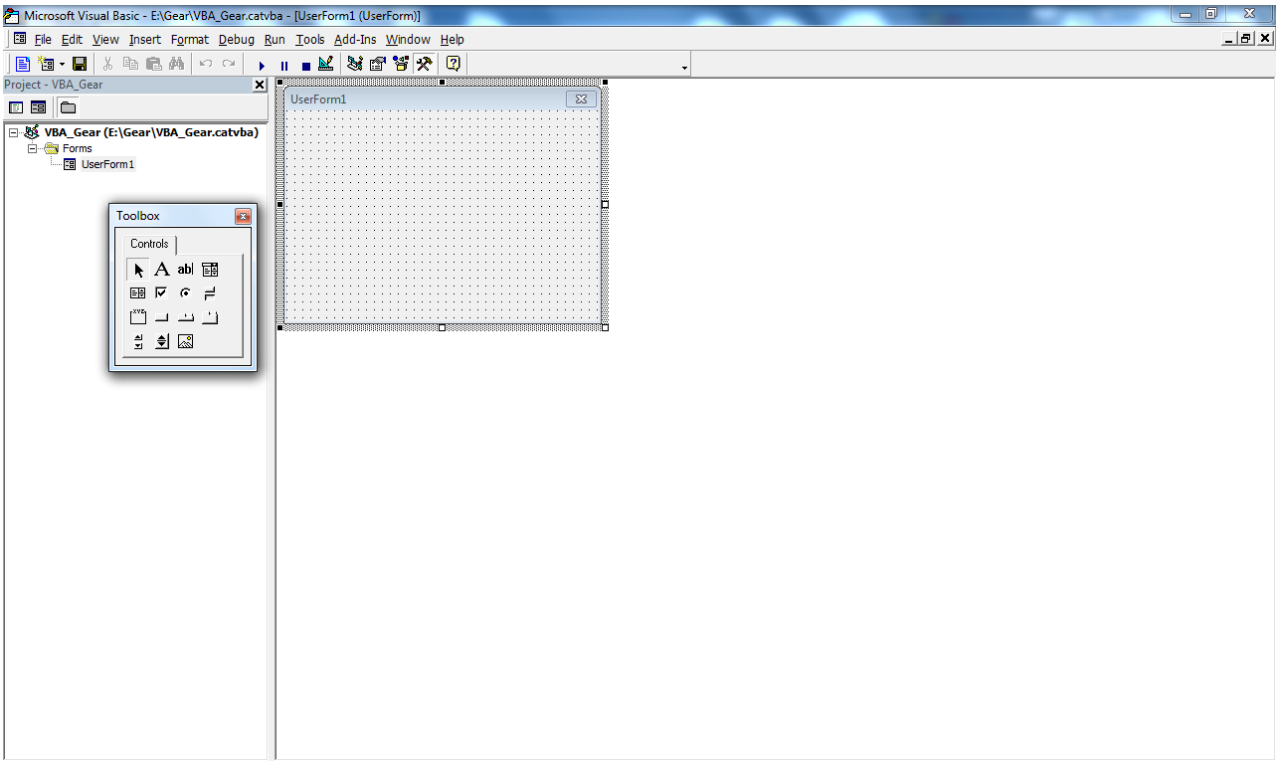


Fig 2-1-10: new Form

To access to properties of this Form, click on menu *View>Properties Window* or press *F4* or click on *Properties window* icon ( ) or right-click on the form and select properties (Fig 2-1-11) . Now properties window of Form appears (Fig 2-1-12). Change the *Name* property of form to *FrmGear*.

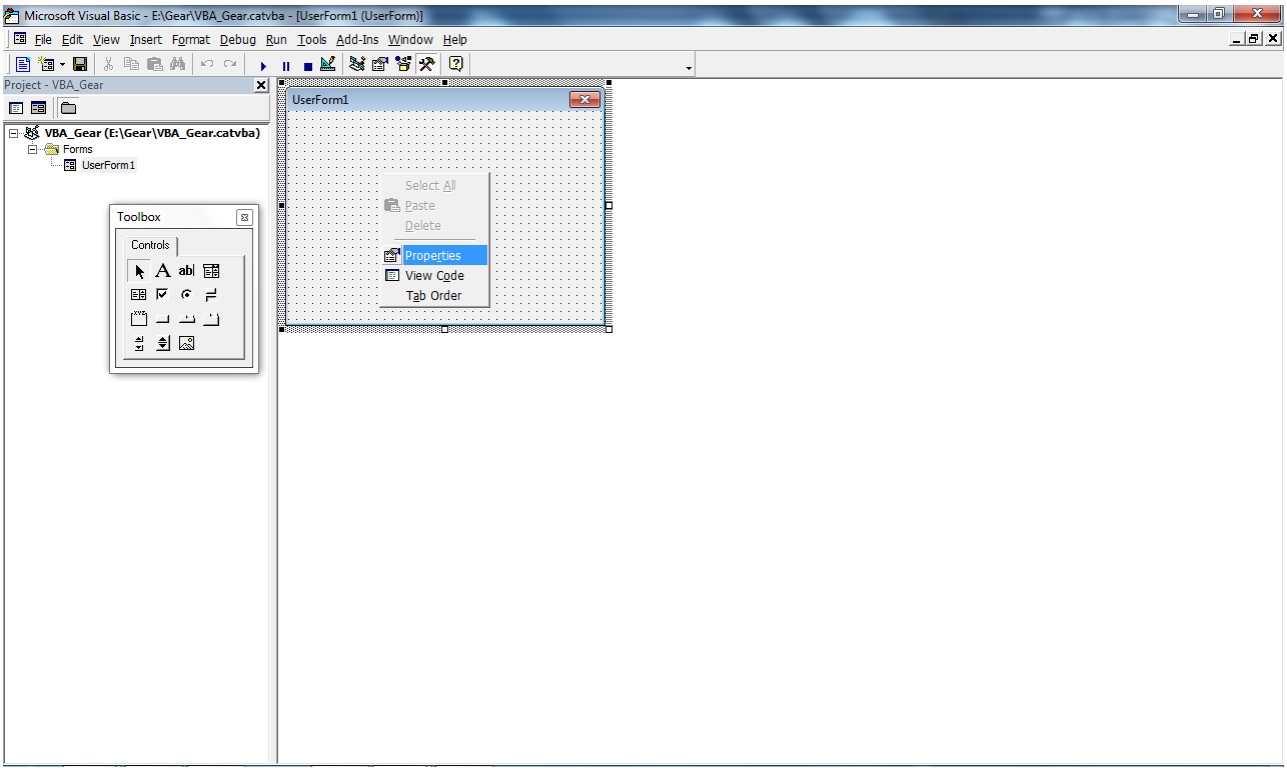


Fig 2-1-11: Access to properties of Form by right-click

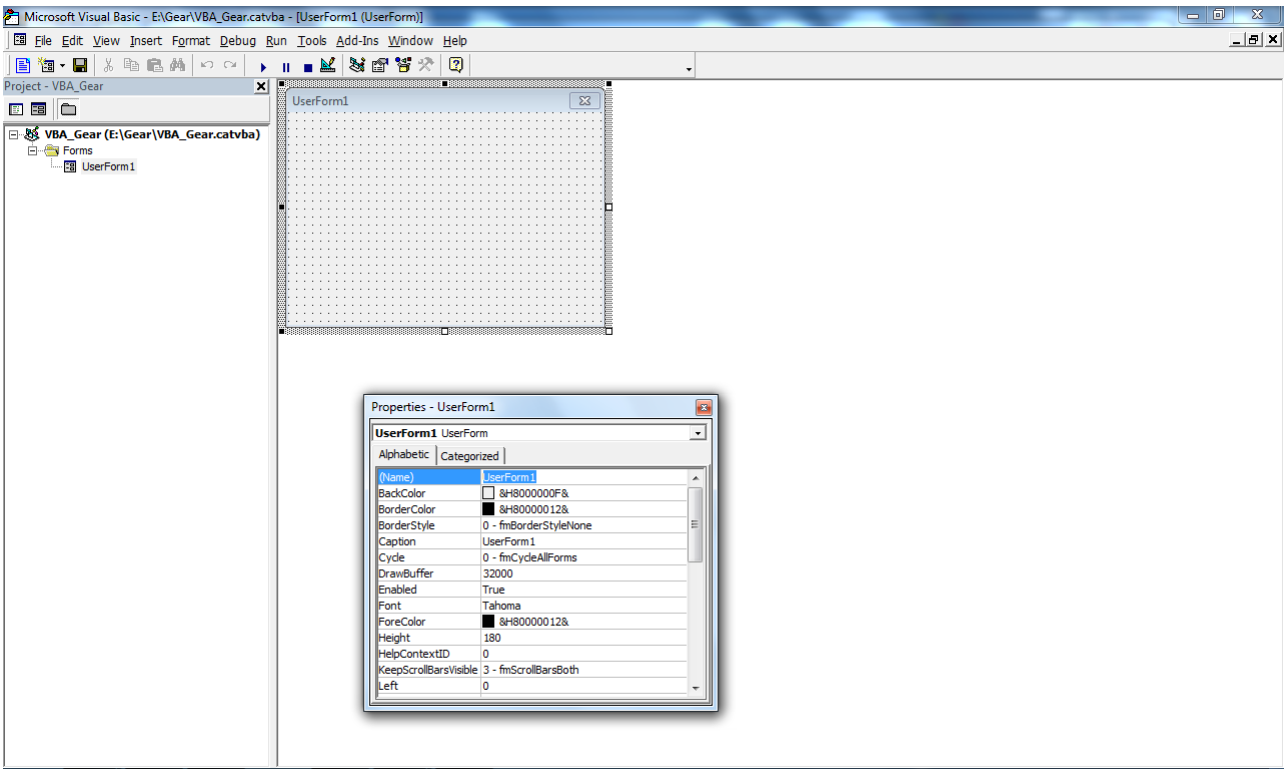


Fig 2-1-12: properties of Form

To access CATIA when the Gear program is running, then set *ShowModal* property to *False* (Fig 2-1-13). Forms are displayed by the Show method of the form object. This method has an optional argument to specify whether the form should be displayed as either modal or modeless. A modal form will not let you interact with any other part of the application until that form is closed. This means that you cannot interactively use other CATIA commands or interact with any other forms that might be displayed at the time. Forms are, by default, modal unless otherwise specified. We want to interact with CATIA so set *ShowModal* property to *False.*

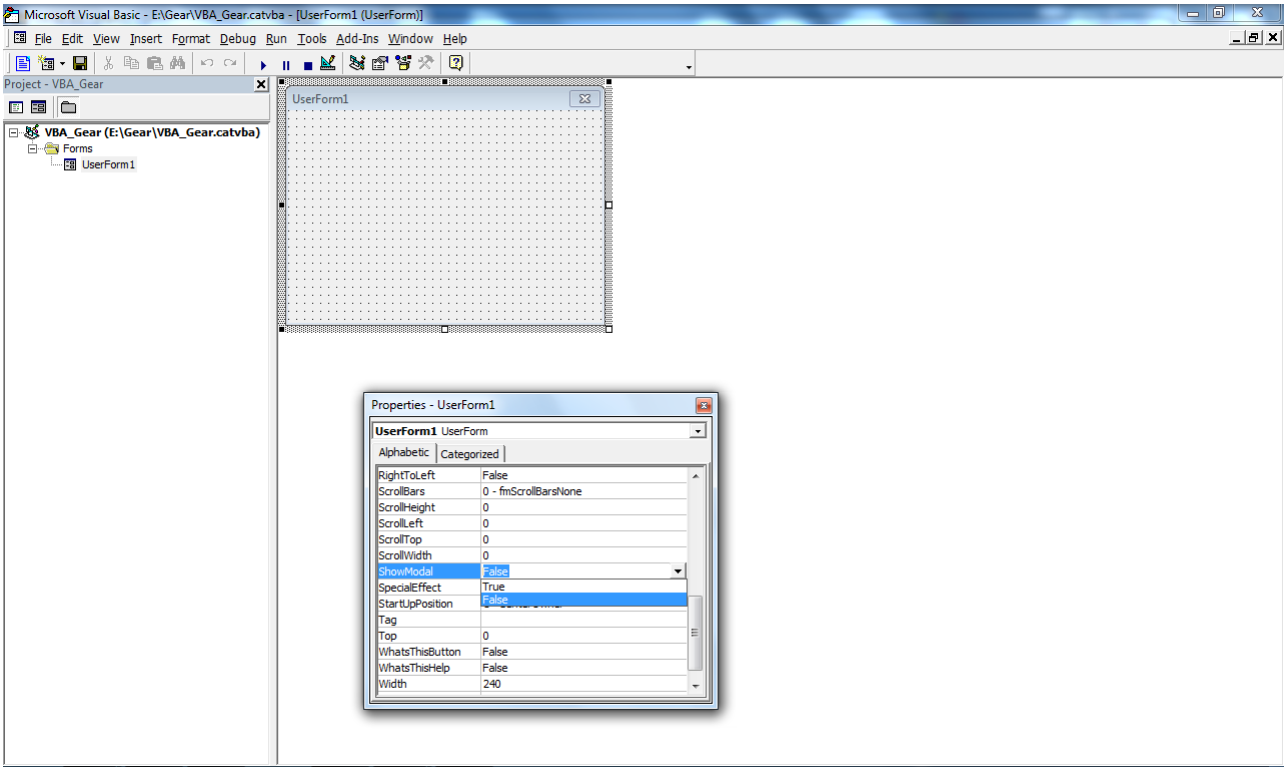


Fig 2-1-13: *ShowModal* property of Form

Change the *Caption* property to *3D Gear* (Fig 2-1-14). Finally, change the size of the Form (by dragging of corners of Form or by setting *width* and *Height* property).

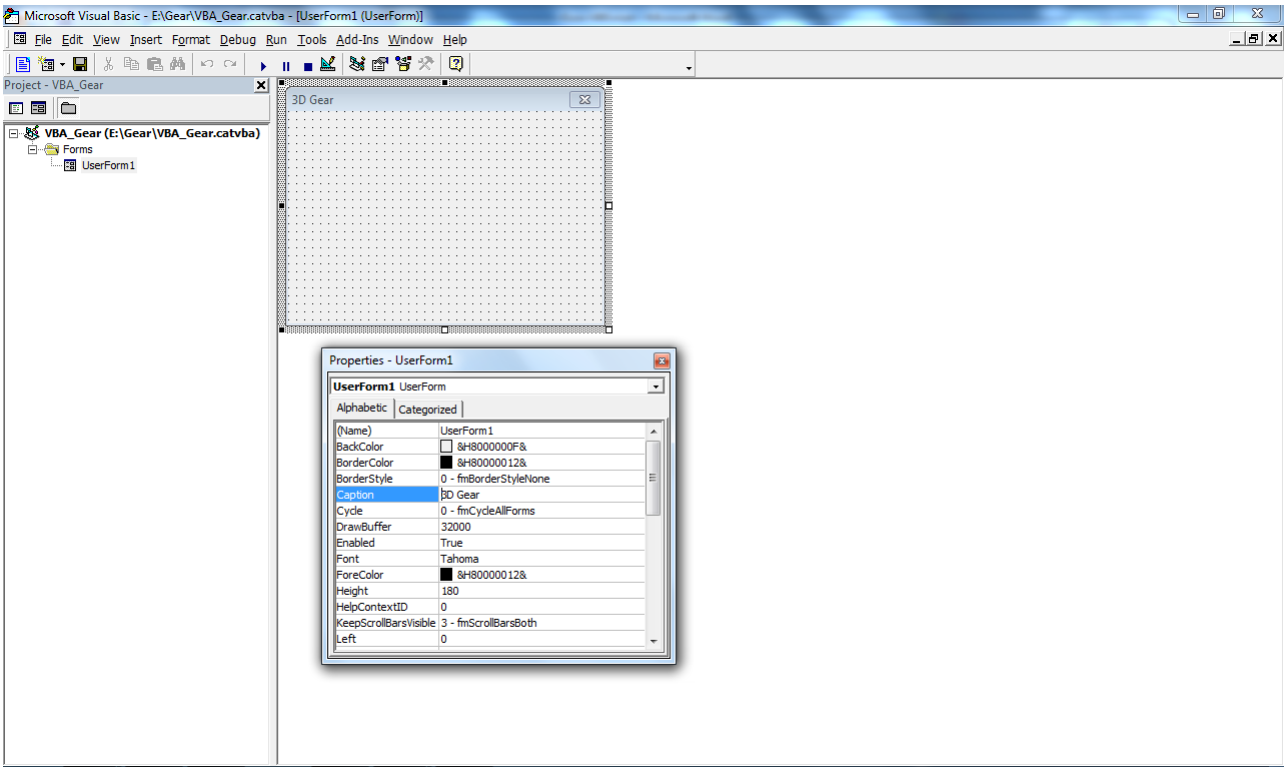


Fig 2-1-14: *Caption* property of Form

To classify the controls such as TextBoxes, Labels,... , insert a Frame (Fig 2-1-15) by dragging its icon on Toolbox window (Fig 2-1-16) to Form or by clicking on its icon and click a point on Form. (If you don't see this window click on menu: *View>Toolbox* or click on its icon in Standard Toolbars)

2-1-9) Get the Scale of Diameter and check its value

```
Dim Sc As Double                'Scale in conic Gear types
Sc = Val(TxtScale.Text)
    If (Op3.Value = True Or Op4.Value = True) And Sc = 0 Then
        MsgBox "Please Increase Conic Scale.", , "Data Review"
        FrmGear.Show
        TxtScale.SetFocus
        Exit Sub
    End If
```

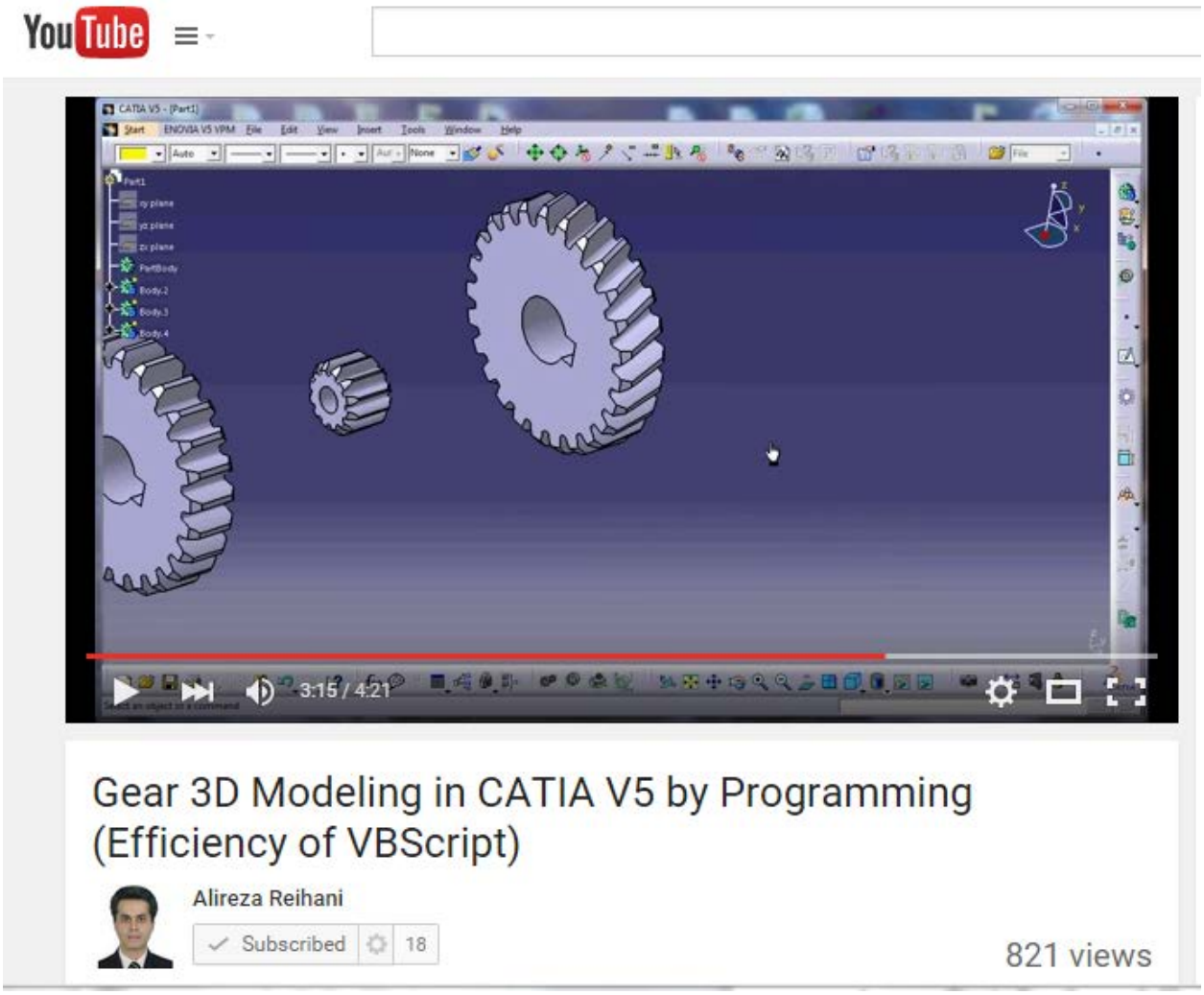
2-1-10) Calculate the five Radiuses necessary to draw Gear Tooth

```
Dim Rt As Double
Dim Ro As Double
Dim Rd As Double    'Gear minimum Radius (Dedendum)
Dim Ru As Double    'Gear Maximum Radius (Addendum)
Dim Rf As Double    'Fillet Radius

Ro = m * Z / 2#
Rt = 0.94 * Ro
Rd = Ro - 1.25 * m
Ru = Ro + m
Rf = 0.35 * m

If Rd <= 0 Then
    MsgBox "Please Increase Teeth Qty.", , "Data Review"
    FrmGear.Show
    TxtZ.SetFocus
    Exit Sub
End If
```

Video Demo

Watch this program in action.

https://www.youtube.com/watch?v=Q7pERp0PfvI



Learn how to create an impressive CATVBA project you can use in your next job interview.

Learn how to automatically model four different types of gears in CATIA V5 and create a complex user form to create new gears with your desired properties on-the-fly. Along the way of this step-by-step tutorial you'll also learn how to:

• Sketch and model different types of gears in CATIA V5

• Create a detailed userform with command buttons, combo boxes, etc.

• Break up a complicated project into smaller, more manageable components

• Use subroutines and functions

• Anticipate and check for user errors in your program

After going through the guide you'll have a fully functional program that you'll be able to show off to potential employers. This tutorial will show you one way you can unlock the full power of CATIA V5, a skill that can be translated to countless other tasks and systems.

What's Included:

• 120 page step-by-step guide (PDF)

• Gear 1 and Gear 4 CATPart Examples

• CATVBA Code

• 60-day money back guarantee

Click here to get started: http://www.scripting4v5.com/gear-tutorial

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Questions? Contact us.

We wish you nothing less than success. Thanks again, good luck, and happy programming!

- Alireza Reihani and Emmett Ross

Start modeling gears today!

Appendix I: Keyboard Shortcuts

Default CATIA and VBA editor shortcuts:

CATIA V5:

- F1: Open the CATIA V5 online contextual help file
- Alt+F8: Macro shortcut
- Alt+F11: Open the macro editor

VBA Editor:

- F1: Visual Basic help
- F2: Open the Object Browser
- F4: Properties Window
- F5: Run macro
- F7: Code window ➢ F8: Step Into
- Crtl + Break: Break
- Crtl + J: List properties and methods
- Alt+F11: Go back to CATIA
- End: Quit a running macro

Appendix II: Resources

The following is a list of resources used when creating this tutorial and other recommended tools.

HOW TO: Création Roue Dentée Paramétrée CATIA V5 R20 By madriver30

https://www.youtube.com/watch?v=dvyLuOPvMV8

Helical gear part by Priyam Bajpai

https://www.youtube.com/watch?v=H989ooFHrxo

Reference for finding intersection points of two circles:

http://math.stackexchange.com/questions/256100/how-can-i-find-the-points-at-which-two-circles-intersect

Types of gears:

http://sdp-si.com/resources/gears/pdf/gear_types_manufacturing.pdf

Confused? Post your question in the forum if you get stuck or have any suggestions for improving the tutorial.

http://www.scripting4v5.com/forums/forum/catia-macro-programming/