

Tolerance Based Assembly of Mechanical Joints using Visual Basic

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Abstract

The goal of this project is to performing assembly of mechanical joints based on tolerance. To obtain the proper functioning at the product the tolerance are important role. In this project the assembly of the component based on it tolerance. The required type of fit information gathered through form. This project is to reduce the time taken for assembly process while manual assembly, to improve the best assembly sequence. In this project to do the assembly of machine elements and its dimension with tolerance values are tabulated while reading the component through VB, and database to be created with the available components. Depends upon the user requirement or the application the appropriate components are identified and assembled in the modelling software. A mechanical assembly is a composition of interrelated parts. Assembly data base stores the geometric models of individual parts, the spatial positions and orientations of the parts in the assembly, and the relationships between parts. The manual assembly is a sophisticated tasks which consume time and requires skilled labour and cost. The work employs Visual Basic, SolidWorks and Excel to develop a database management system for standard components. The secondary development of SolidWorks has been performed by Visual Basic and the system is interconnected with SolidWorks. The outcomes were creating a application form to create a tolerance based assembly application which results in increase in speed of part assembly saving time and cost.

Keyword- SolidWorks, Visual Basic, Graphical User Interface, Ms-Excel database, Fits and Tolerances

I. INTRODUCTION

As a result of the rapid development of the manufacturing systems, productivity has become one of the goal that need to be achieved as well as reducing production time. Three factors should be considered by the designer, the database of the fixture elements, the proper design approach, and powerful 3D CAD software for modelling, design and assembly process. SolidWorks was used as CAD software due to its excellent 3D capabilities for performing the modeling process; the use of secondary development of SolidWorks with VB for automating design and assembly. This approach includes generating StandradEXE project in VB and integrating with SolidWorks API for creating part in SolidWorks environment. A 3D model of the specific workpiece was designed in SolidWorks.

A. Tolerance and Fits

1) Tolerance

The permissible variation of a size is called tolerance. It is the difference between the maximum and minimum permissible limits of the given size.

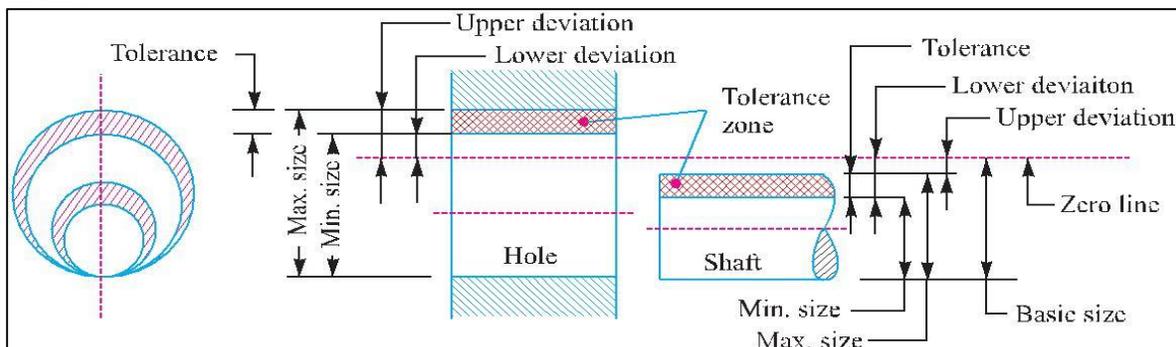


Fig. 1: Tolerance zone

2) *Unilateral*

It is a tolerance in which variation permitted only in one direction from the design size.

3) *Bilateral*

The variation is permitted in both the directions from the design size.



Fig. 2: Method of assigning tolerances

B. *Fit*

The degree of tightness or looseness between the two mating parts is known as a fit of the parts. The nature of fit is characterized by the presence and size of clearance and interference. The clearance is the amount by which the actual size of the shaft is less than the actual size of the mating hole in an assembly. In other words, the clearance is the difference between the sizes of the hole and the shaft before assembly. The difference must be positive. The interference is the amount by which the actual size of a shaft is larger than the actual finished size of the mating hole in an assembly. In other words, the interference is the arithmetical difference between the sizes of the hole and the shaft, before assembly. The difference must be negative.

1) *Type of Fits*

a) *Clearance fit*

In this type of fit, the size limits for mating parts are so selected that clearance between them always occur. It may be noted that in a clearance fit, the tolerance zone of the hole is entirely above the tolerance zone of the shaft. In a clearance fit, the difference between the minimum size of the hole and the maximum size of the shaft is known as minimum clearance whereas the difference between the maximum size of the hole and minimum size of the shaft is called maximum clearance. The clearance fits may be slide fit, easy sliding fit, and running fit, slack running fit and loose Running fit.

b) *Interference fit*

In this type of fit, the size limits for the mating parts are so selected that interference between them always occurs. It may be noted that in an interference fit, the tolerance zone of the hole is entirely below the tolerance zone of the shaft. In an interference fit, the difference between the maximum size of the hole and the minimum size of the shaft is known as minimum interference, whereas the difference between the minimum size of the hole and the maximum size of the shaft is called maximum interference. The interference fits may be shrink fit, heavy drive fit and light drive fit.

c) *Transition fit*

In this type of fit, the size limits for the mating parts are so selected that either a clearance or interference may occur depending upon the actual size of the mating parts. It may be noted that in a transition fit, the tolerance zones of hole and shaft overlap. The transition fits may be force fit, tight fit and push fit.

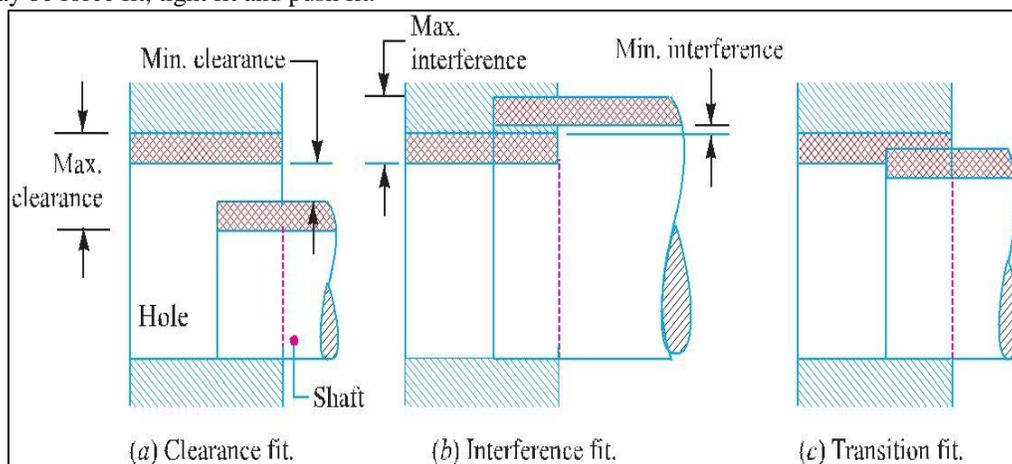


Fig. 3: Types of fits

C. *Basis of Limit System*

1) *Hole Basis System*

When the hole is kept as a constant member (i.e. when the lower deviation of the hole is zero) and different fits are obtained by varying the shaft size then the limit system is said to be on a hole basis.

2) Shaft Basis System

When the shaft is kept as a constant member (i.e. when the upper deviation of the shaft is zero) and different fits are obtained by varying the hole size, then the limit system is said to be on a shaft basis.

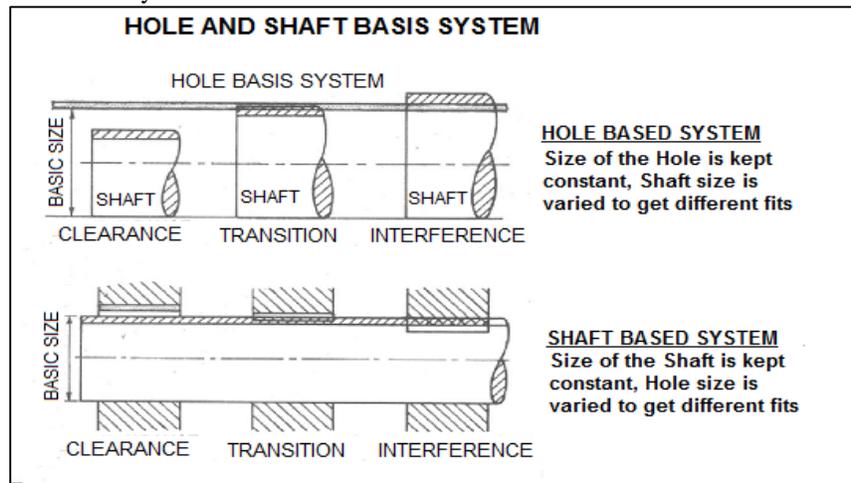


Fig. 4: Basis of limit system

3) Visual Basic

Visual basic is a programming language. It can be developed by Microsoft. This language widely used among a high level of the developers as "primary development tool" compared to other programming languages. Visual basic uses integrated development environment with application program interface and graphical user interface. Visual basic is considered as engine for building the macros in all Microsoft Software. VB is the advanced version of the Beginners All-Purpose Symbolic Instruction Code (BASIC) language with "Integrated Development Environment (IDE)" which API accessing and graphical user interfaces. There are different types of projects can be generated in VB. For simple programming purposes, Standard EXE is used more common by the programmers. For more advanced programming functions, ActiveX projects are used. In this project Standard EXE, has been created. Also, this project controls the features and the operations of other applications by creating new menus and toolbars into the application environment. For database management purposes, VB is the engine of Microsoft Access for building the database and this gives the programmer the opportunity for controlling the database efficiently.

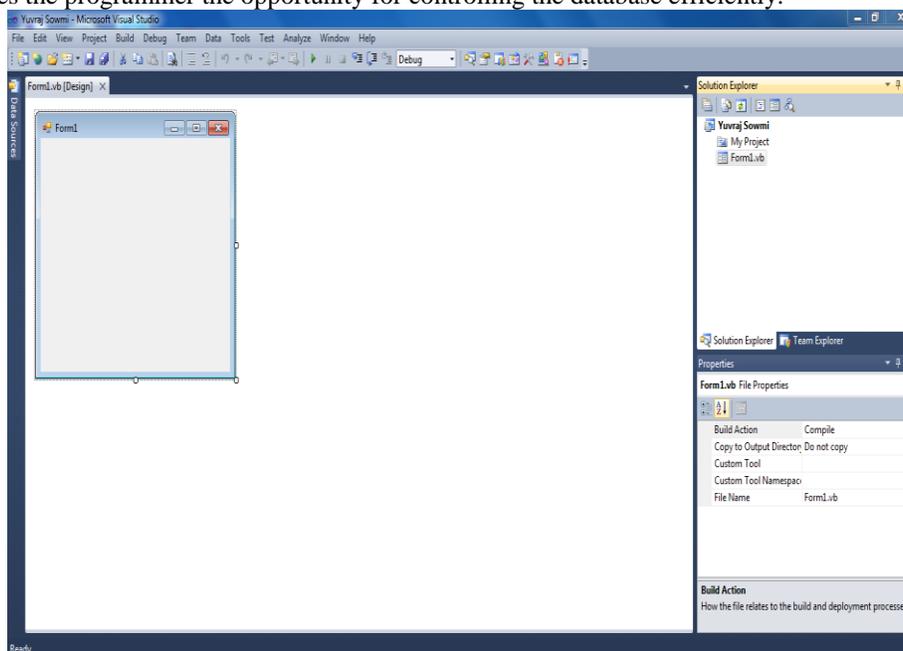


Fig. 5: Project window and form 1

4) SolidWorks

SolidWorks is a 3D mechanical CAD (computer-aided design) program that runs on Microsoft Windows and is being developed by Dassault Systems SolidWorks Corp.,. SolidWorks works on Microsoft Windows which gives Visual Basic Integration and development of application program interface. 3D modelling produces higher accuracy in designs because the nature of "virtual

mock up" allows errors such as interference to be identified easily and human error is minimized as design variations are automatically updated by the computer. A major advantage of 3D modelling is its flexibility.

II. SOLID WORKS SECONDARY DEVELOPMENT

A. SolidWorks API (Application Program Interface)

Application programming interface (API) is a tool to write a code in a programming language within another application. As a result, a direct integration between different applications can be developed. SolidWorks is one of the applications which support API with different programming languages such as C++, VB, and Visual Studio. SolidWorks API automates the design and the assembly operations by creating codes in a specific programming language and it has been applied for different design methods. An API defines functionalities that are independent of their respective implementations, which allows definitions and implementations to vary without compromising the interface. A good API makes it easier to develop a program by providing all the building blocks. A programmer then puts the blocks together. Using the API, model files created in SolidWorks can be manipulated directly inside of a control program such as Excel and the resulting altered parameters can be returned for further review. It makes the work of programming the graphical user components become easier. This is the most important part which completes the SolidWorks based in the two development of parametric design. SolidWorks API includes SolidWorks events, methods, properties and correlation functions so as to complete the whole process of parts modelling. SolidWorks API is a tree like hierarchical structure. SolidWorks API covers all the functions of the software.

B. SolidWorks Secondary Developments

The SolidWorks software development is done in two different ways. One of them is the one based on Object linking and embedding (OLE) automation technology. The other one is that based on Component Object Model (COM) specification. SolidWorks program provides an OLE automation interface, so that other applications can operate on it. OLE automation technology allows the programmer to control other applications of the project especially access to these objects properties and methods. COM is the development of OLE technology in higher level; it is a kind of binary communication specification of cover and contains everything (that is everyone should comply with the same). The Object Oriented Programming (OOP's), VB supports both VB and COM technologies and hence this project has used VB.

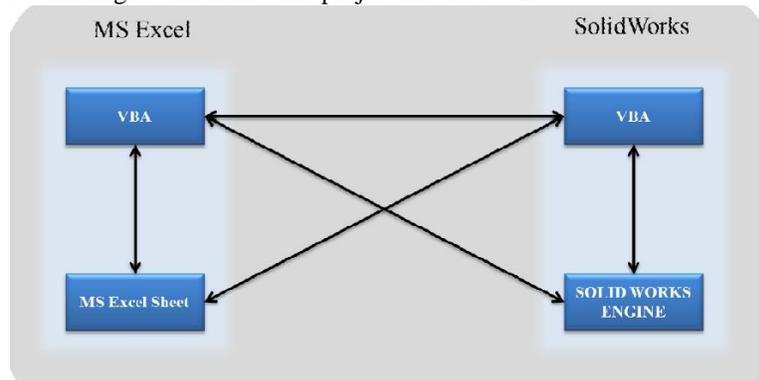


Fig. 6: Methodology flowchart

III. AUTOMATED ASSEMBLY APPROACH

The 3D model is an important factor that can provide the appropriate specification of the work piece. SolidWorks is used the CAD environment for undertaking the 3D modelling in this study. The expert system is applied as a tool to make right decisions about the CAFD planning process, including locating and clamping methods and their related element selection. This helps achieve a feasible fixture design layout. Solid-Works API and Visual Basic programming language are employed for the automating and simulation of the assembly process of MFs. A feature library of modular fixture elements is constructed as a means to simplify the fixture design process. However, the macros can be applied only for the master SolidWorks document that they have been created.

The solution of this problem was to make global macros by modifying the recording macros. It can be completed by changing VB methods and classes of the recorded macros and adding the swConst modules and SW Assembly or swPart class modules. This makes the macros available for any SolidWorks documents. This global macros are still not in the format for the developed Active X Direct Linking Library (DLL) project. The important function of the add-in project is how to make the created menus calling the global macros which perform SolidWorks design and assembly. This was achieved by importing the global macros into the Active X DLL project as modules with .bas format and then writing a subroutine code for each macro and finally calling this subroutine by the specific menu's icon.

IV. DATA EXTRACTION

SolidWorks (.sldprt) file the one of the component from the assembly parts. The .sldprt files have to be read by the hierarchy which is provided in SW help.

A. Feature Manager Tree

The Feature Manager Design tree on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing. This makes it easy to see how the model or assembly was constructed or to examine the various sheets and views in a drawing. The Feature Manager Design Tree makes many selection and filtering operations simple, and provides access to many folders and tools useful when working with models. The Feature Manager Design tree and the graphics area are dynamically linked. User can select features, sketches, drawing views, and construction geometry in either pane. User can split the Feature Manager Design tree and either display two Feature Manager instances, or combine the Feature Manager Design tree with the Configuration Manager or Property Manager.

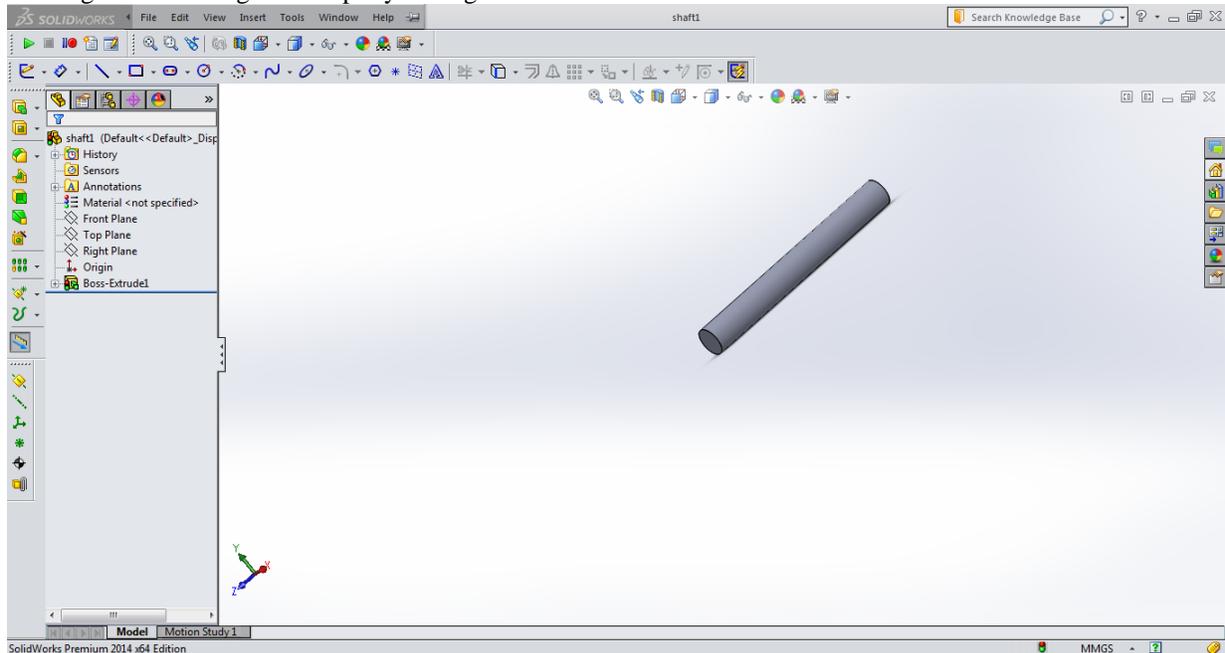


Fig. 7: Feature Manager Tree for the part

V. CODE GENERATION

In this paper add in code was created in order to build menus in SolidWorks environment for Tolerance based assembly of Mechanical joints. The menus were built by using visual basic 2010 integrating with SolidWorks API (application program interface). SolidWorks reference were linked in the visual basic. The reference includes SldWorks Type Library, SolidWorks Exposed type Libraries and SldWorks Const Type Libraries.

```
Public Class YuvrajSowmi
Public swapp As Sldworks.Sldworks
Dim xlapp As Excel.Application = CreateObject ("excel.application", "")
Dim xlwbk As Excel.Workbook = xlapp.workbook.add()
Dim xlsh1 As Excel.worksheet = xlwbk.worksheets(1)
Private Sub YuvrajSowmi_Load()
Xlapp.Visible = True
End Sub
Public Subswopen()
Dim swapp As SldWorks.SldWorks = CreateObject ("sldworks.application", "")
swapp.Visible = True
Dim ModelDoc2 As SldWorks.ModelDoc2 = swapp.OpenDoc ("E:\Solidworks\SuntharaPandiSowmiya\nut1.SLDPRT", "1")
End Sub
End Class
```

VI. ASSEMBLY

In this project an Application window have been created using VB to read the Solid Work part file [.sldprt]. It reads the data from the feature manager tree and the form extracts the data in solid work part, and its dimension with tolerance values are tabulated while reading the component through VB, and database to be created with the available components using MS Excel. The created database interfacing with modelling software using VBA form and detects the mating components. Depends upon the user requirement the appropriate component are identified and assembled in the modelling software based on its fits and tolerances.

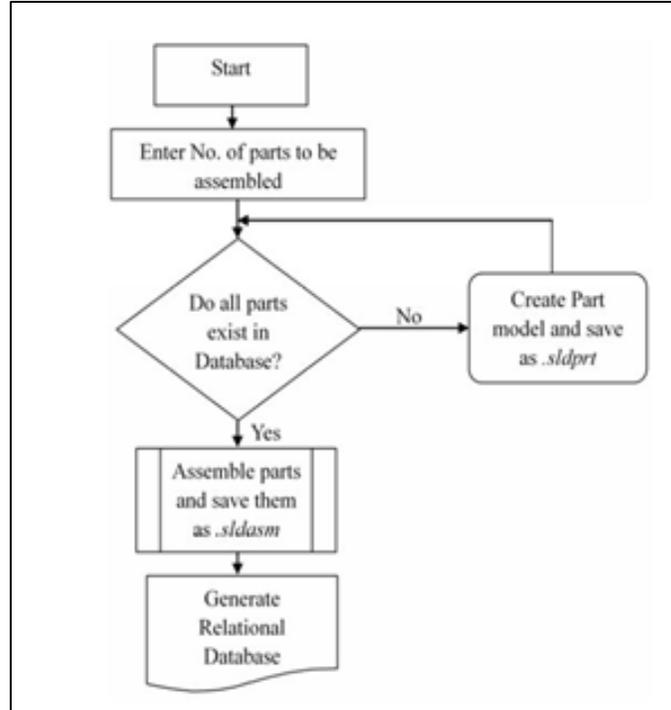


Fig. 8: Process flow to generate relational database.

A. Tolerance Calculation

Hole Basis					
Type of fit	Hole tolerance	Shaft tolerance	Hole diameter In(mm)	Shaft diameter in(mm)	
				Min	Max
Clearance	H6	g7	15	14.994	14.976
Transition	H6	k7	15	15.001	15.019
Interference	H6	p7	15	15.018	15.036

Table 1: Whole basis measured value

Shaft Basis					
Type of fit	Shaft tolerance	Hole tolerance	Shaft diameter In(mm)	Hole diameter in(mm)	
				Min	Max
Clearance f	G7	h6	15	15.006	15.024
Transition	K7	h6	15	14.998	15.006
Interference	P7	h6	15	14.971	14.989

Table 2: Shaft basis measured value

Part Model List				
Part name	Base diameter in(mm)	Tolerance		Measured value in(mm)
		Bi lateral	Uni lateral	
Shaft.1	15			14.974
Shaft.2	15			14.99
Shaft.3	15			15.019
Shaft.4	15			15.05

Shaft.5	15			15.08
Shaft.6	15			15.034

Table 3: Part Model List for Shaft

Part Model List				
Part name	Base diameter in(mm)	Tolerance		Measured value in(mm)
		Bi lateral	Uni lateral	
Hole.1	15		✓	14.971
Hole.2	15	✓		14.949
Hole.3	15	✓		15.004
Hole.4	15		✓	15.023
Hole.5	15	✓		14.989
Hole.6	15		✓	14.998

Table 4: Part Model List for Hole

Assembly Matrix		
Type of fit	Part name	Mate Part name
Clearance	Shaft.1	Hole.4
Transition	Shaft.3	Hole.6
Interference	Shaft.6	Hole.2

Table 5: Assembly Matrix

B. Assembly

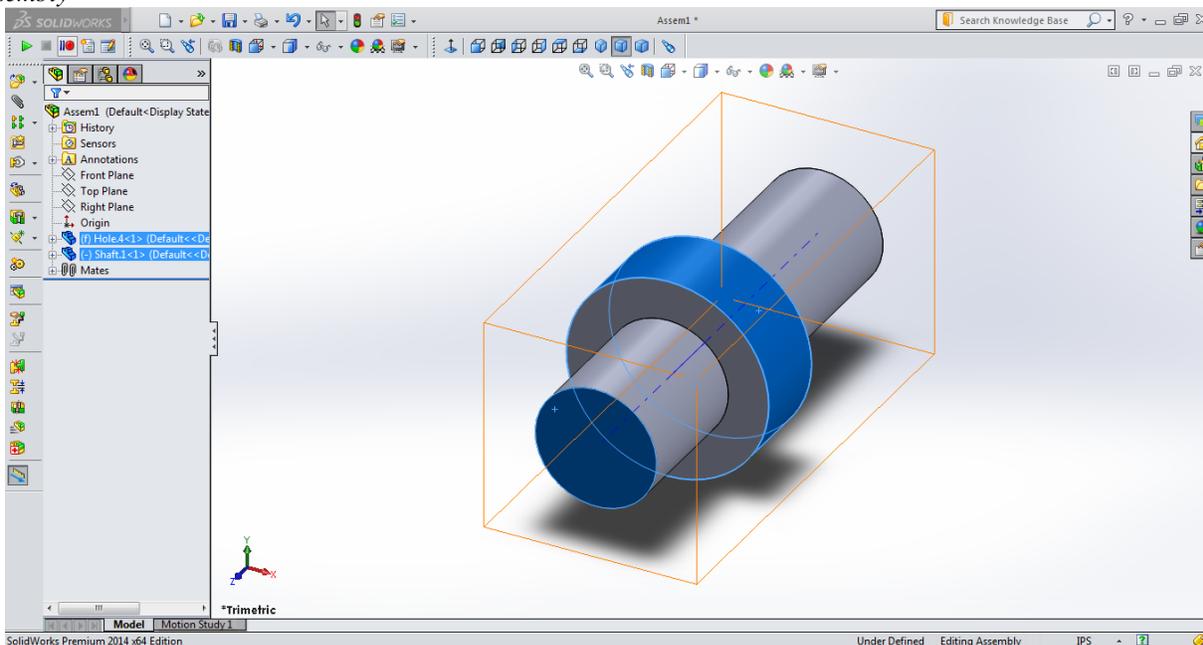


Fig. 9: Assembly File

VII. CONCLUSION

The basic information needed for assembly sequence planning. There are many modelling packages provide information about the solid models, the information regarding to the relationships between parts in the assembly. The mating feature of two contacting parts in the assembly is established. In this project an Application window have been created using VB to read the Solid-Work part file [.sldprt]. It reads the data from the feature manager tree and the form extracts the data in solid work part, and its dimension with tolerance values are tabulated while reading the component through VB, and database to be created with the available components using MS Excel. The created database interfacing with modelling software using VBA form and detects the mating components. Depends upon the user requirement the appropriate component are identified and assembled in the modelling software based on its fits and tolerances.

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