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Automated Modelling of Mechanical Components using Solidworks API

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ABSTRACT
Engineers can utilize the SolidWorks API and macro functionality to construct custom programs and applications (macros) that can be used to automate numerous CAD activities. This paper illustrates work on design automation of mechanical components like nuts, bolts, screws, and flange coupling. The work also describes end-to-end design methodologies, including plane selection, draw the valid 2D profile with dimensions, and fully defined structures using user-forms made with Microsoft Visual Basic (MVB) software. The product is made user-friendly using MVB after conceptual development and algorithm development. This enables us to generate mechanical components from a single click by only providing specific dimensions for the structures. The project methodology is presented through a flow chart that outlines the systematic procedures followed during the project's completion. Users select either a prismatic model or mechanical component, followed by selecting the plane to create the structure. The user has the option to create various prismatic models with features like chamfer, fillet, boss extrude or extrude cut. Once the model is generated with a fully defined sketch, the user is prompted to save the file and close the part file or the application.

Keywords - Assembly, Macros, Prismatic, SolidWorks API, User-form, VB.

1. INTRODUCTION

CAD modelling is a time-consuming and labor-intensive process that industries and multinational corporations aim to keep as low as feasible. It is a time-consuming and costly process. The conventional process is efficient for creating new designs, but it is ineffective for old designs that have had some dimension changes. The SOLIDWORKS Application Programming Interface (API) is a straightforward library that exposes open-source functions that can be called by external programs to execute macro instructions in SOLIDWORKS. The functionality of the SolidWorks API is used by programmers to automate specific design procedures and change the various modules as necessary. SolidWorks API capabilities can be used in practically any CAD design tasks, with a few exceptions and restrictions. The final concept is to use Visual Basic to open SolidWorks from outside, code for the structures to be generated and take user inputs from the user-forms to create the required fully defined models. The program can be changed with the knowledge of Visual Basic for Applications or Visual Studio. Abhishek C. Lad [1] discussed about how we can create complex models using SolidWorks API and generate the drawing sheets on the basis of the model created.

As the name implies, application programming interface (API) refers to a set of interfaces that can be used to create programs for the application. The application refers to SolidWorks in the case of SolidWorks API. In programming, the interface refers to a collection of objects and methods within the SolidWorks application. API functions give programmers direct access to SOLIDWORKS functionality. There are hundreds of functions available in the API, which can be called with the help of languages such as Visual Basic (VB), Visual Basic for Applications (VBA), VB.NET, C++, C#, or SOLIDWORKS macro files. To implement automation routines, software developers can simulate user interaction with SOLIDWORKS or supplement SOLIDWORKS functionality by adding new commands to the application.

Macros are scripts that automate tasks within SOLIDWORKS. These are blocks of codes which can be obtained after recording the task you would like to automate in SOLIDWORKS. These codes have all the mouse clicks, menu choices, and keystrokes used for creating the initial object. The Macro toolbar can be accessed from the Tools menu. SolidWorks macros provide nearly limitless flexibility in manipulating all of the program's features 3 and tools. They also enable the use of traditional programming as well as standard SolidWorks features. Macros can either be recorded or programmed or can be overwritten.

By dragging and dropping objects and modifying their behavior and appearance, programmers can change the code in Visual Basic (VB), a Microsoft event-driven programming language and environment. The event-driven and object-oriented programming language The BASIC programming language is related to VB. VB is used to prototype an application before it is eventually written in a more difficult but efficient language. This process is known as rapid application development (RAD). The goal of VB is to make it easy to learn and rapid to produce code. Visual Basic is the primary tool for finishing this project. Visual Basic is the best and simplest programming language for creating macros.

2. APPROACH TAKEN

The project methodology consists of a flow chart Fig.1 explaining the systematic procedures and processes followed in completing this project from the initial stages. First, it is needed to run the code to launch the SolidWorks Software and open the new part document which is based on the default template of SolidWorks. To open the SolidWorks from outside specific code is written in Visual Basic language by using Visual Studio 2019. The code requires dependencies such as "SolidWorks.Interop.sldworks", and "SolidWorks.Interop.swconst" as the project references which helps to launch the SolidWorks software in any device if it is installed. Next, the user needs to select the type of model to be generated. After this, the user needs to select the plane in which the structure must be created. Then the user is asked to select the required simple structure (cube, cuboid, cone, sphere, and cylinder) or mechanical components (Nut, Bolt, Screw and Flange Coupling) and provide the required dimensions to generate the model. If the user selects prismatic model to be created then after the generation of the first model the user can use features like chamfer, fillet, extrude cut or extrude boss and design the model as per the requirements. Finally, a model will be generated with a fully defined sketch. Again, a new user-form will displayed to save the file and close the part file or software. Following the conceptualization of design for this process, the most recent advancements in the same area are studied and understood.

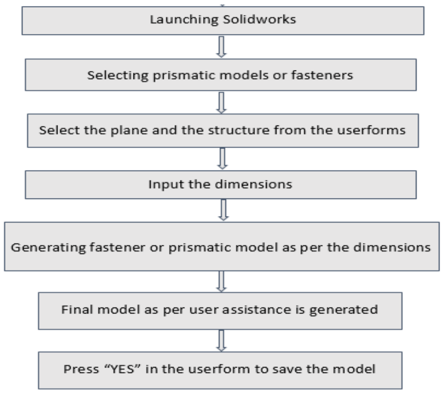

Figure 1. Process to create the model.

3. PRISMATIC MODELS

After the user selects prismatic model the plane selection window appears Fig. 2toe generate the model. Balachandar Krishnamurthy et al discussed about we can generate the cube by creating the user-form which takes the input for the side of the cube from the user which gives us a brief idea on how to create the first prismatic model and to change the material of the model as per the requirement.

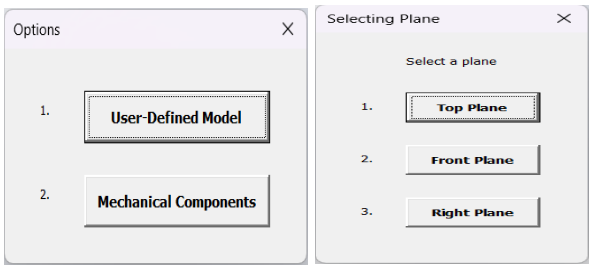

Figure 2. Selection of model and Selection of Plane

The user can then select any one of the 5 options (Cube, Cuboid, Cylinder, Cone, and sphere) as the base of the prismatic model as shown in the Fig.3.

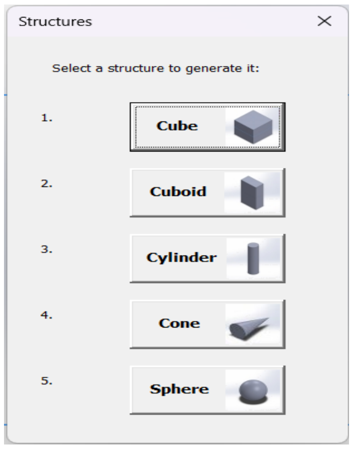

Figure 3. Selection of Prismatic Model

The user is then asked to input the dimensions as per the selected model and a base model is generated. After this the user is asked to select a face or edge to perform further operations to the base model like Extrude, Extrude-Cut, Chamfer or Fillet as shown in Fig.4. The loop of selection of face or edge continues until the user is satisfied by the final model and decides to cancel the user-form.

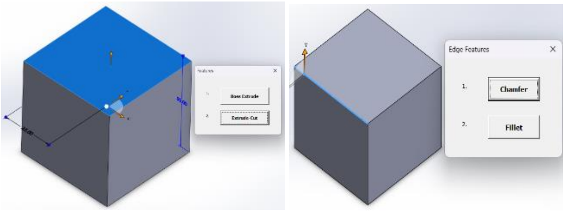

Figure 4. Generation of Face or Edge Features

Once the user selects a face and chooses Boss-Extrude option, they are asked to select which of the models needs to be created on the base model. If the user selects Extrude-Cut option, they are asked to select the type of cut (Circular, Rectangular, Triangular or Tapered Hole) and provide the depth of cut as per required as shown in Fig.5.

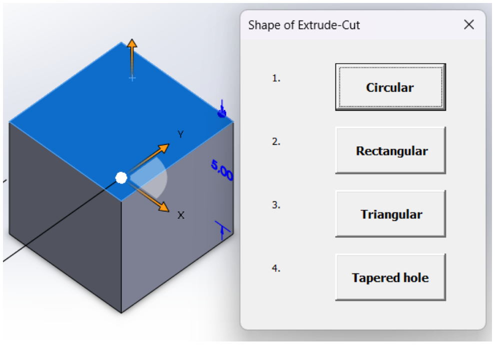

Figure 5. Shape of Extrude-Cut

If the user selects an edge, then they are asked to give the dimensions of the fillet or chamfer as per their needs and use. The final model is shown in Fig.6 which is an example of a prismatic model.

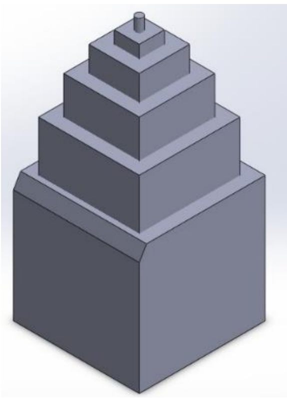

Figure 6. Sample Generated Prismatic Model

4. MECHANICAL COMPONENTS

A mechanical component is a part or element which has a specific design and cannot be changed as per convenience. When the user selects Mechanical Components from the first user-form and the plane in which the component needs to be generated, a third user-form pops up asking the user for the component which needs to be generated as shown in Fig.7. The user can select any one of the components (Nut, Bolt, Screw or Flange Coupling). The options present in the user-form have the final image of the model in the icons so that the user can have the preview of the model which will be generated.

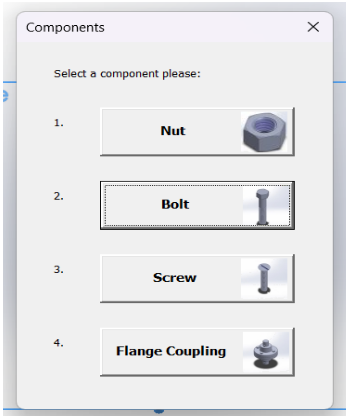

Figure 7. Mechanical Components

4.1 Fasteners

After selecting the fastener, the user is asked to provide the dimensions of the component (Nut, Bolt or Screw) like diameter and after clicking on Create, a model is generated as per the user dimensions. If the user needs a customized thread for the fastener, they can check the box named as Override Pitch to give the dimensions of the pitch and the model will be created as per the dimensions. If the box remains unchecked then the fastener generated will have standard pitch thread. If the user selects Bolt or Screw as the fastener to be generated then the user-form asks for the dimensions of the diameter and the length of the shaft of the fastener and can select the length of the thread. The user can select full thread or standard thread length, or custom thread length based on their needs. Fig.8 is an example of the user-form for the bolt for generation of a final mechanical component with standard formulae as shown in Fig.9.

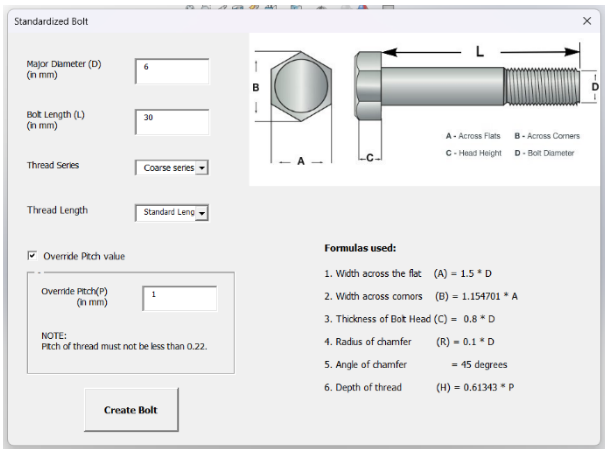

Figure 8. Data collection window for Bolt

Some examples for final model of other components like nut and screw are shown in Fig.9 which describes the model based upon dimensions given by the user in the respective user-forms.

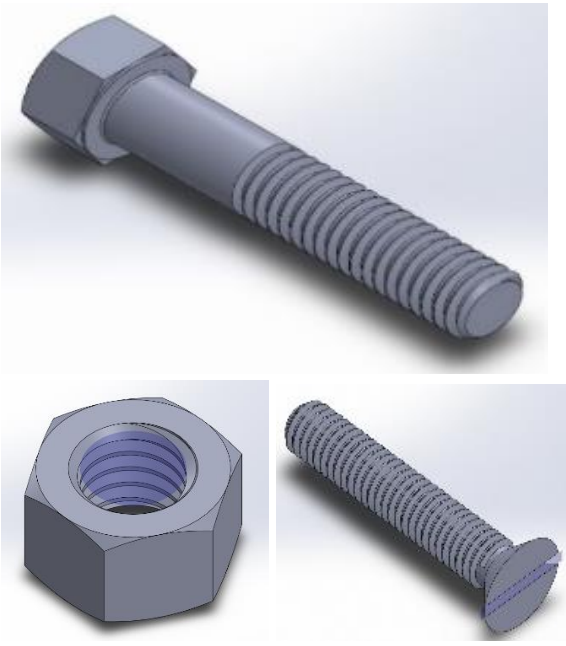

Figure 9. Modeled Bolt, Nut and Screw

4.2 Flange Coupling

If the user selects Flange Coupling as the model to be generated, then the next user-form asks the shaft diameter as shown in Fig.10 and as an example the value given is 45 mm and based on the formula all the parts of Flange Coupling such as Flange, Key, Shaft, Nut and Bolt are created with fully defined sketches. The given input must be restricted between 10.1 to 515 mm cause a value lower than 10.1 mm causes issue in creation of Key due to its tapper angle and value greater than 515 mm causes issue in mate of Key and the Shaft making it a under-defined assembly.

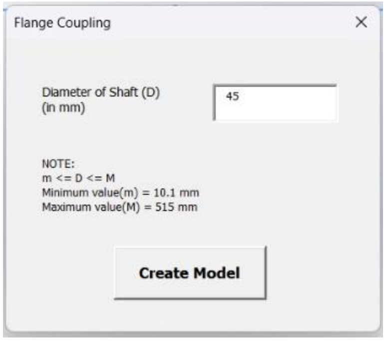

Figure 10. Input for Flange Coupling

After the creation of the parts the files are saved by a default name and since Flange Coupling is an assembly, a new assembly file is opened and the parts are added one by one and the appropriate mates are done and the final model is created as shown in Fig.11. After the completion of the model the user is asked whether they want to save the file and close the part file or not. After that another user-form pops up and the user is asked to close the software or open a new part file and the whole process is repeated again. Dwaipayan Roy Chowdhury [3] discussed about how we can create macro assisted design automation of Piston & Connecting Rod Using SolidWorks API which helped of creating the parts for the flange coupling along with standard formulae and create a fully defined model along with proper mates associated with the mechanical components.

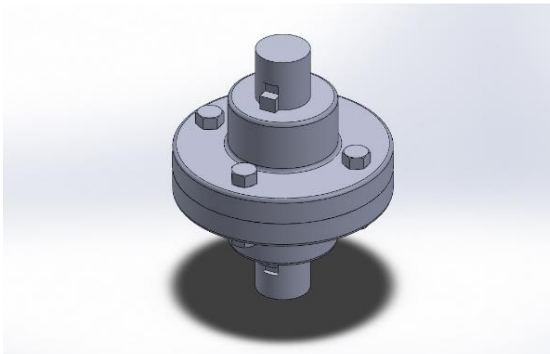

Figure 11. Automated Assembled Flange Coupling

5. SAVING / CLOSING OF PART FILE

The program is coded in such a way that if user inputs any alphabets or characters in the place of dimensions then a caution user-form will pop up as a sign of warning that the user needs to input a valid number. After the model is generated a user-form pops up asking the user whether or not to save the file and close the part file. If the user chooses to save the file another user-form will ask the name of the file and the file will then be saved to the desktop as shown in Fig.12.

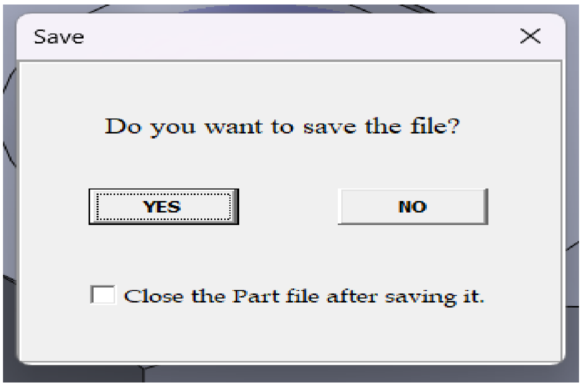

Figure 12. Saving and closing of Part File

After the file has been saved another user-form pops up asking the user to close the software or create a new file as shown in Fig.13. If the software is closed then the program ends and if the user selects to open a new part file then the whole program is run again and the user can again start from the beginning.

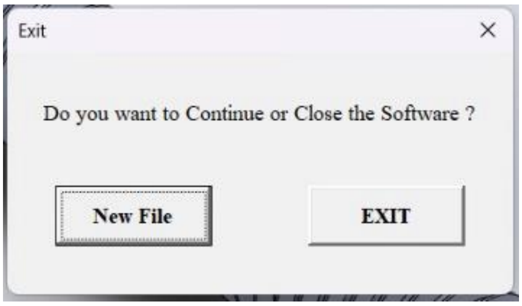

Figure 13. Closing of Part File or Software

6. FORMULA AND STANDARDS EMPLOYED

The standards taken into consideration for generating the fasteners are drawn out from the PSG Design Data Book. All the inputs taken and models generated in this project are in millimeters (mm). Kalaikathir Achchagam [4] provided the standardized formulae for nut, bolt, screw and parts of flange coupling along with proper explanation which helped in generation of each and every mechanical components. The formulas used for designing the models are given as follows:

Table. 1. Formula for Nut, Bolt and Screw

Nomenclature	Formula
NUT	
Width across the flat (F)	1.5 * D
Width across corners (G)	1.154701 * F
Height of Nut (H)	0.9 * D
Radius of chamfer (R)	(0.154701 * F) / 2
Angle of chamfer	30 ^{o}
Depth of thread (d)	0.61343 * P
BOLT	
Width across the flat (A)	1.5 * D
Width across corners (B)	1.154701 * A
Thickness of Bolt Head (C)	0.8 * D
Radius of chamfer (R)	0.1 * D
Angle of chamfer	45 ^{o}
Depth of thread (H)	0.61343 * P
Bearing surface diameter	
Bearing surface height	0.04 *D
Fillet radius	0.03 * D
Fillet angle	45 ^{o}
Standard thread length (S)	If D <= 125 mm, S = (2 * D) + 6 If 125 < D <= 200, S = (2 * D) + 12 If D > 200, S = (2 * D) + 25
SCREW	
Height of head (H)	D / 2
Countersunk Angle	45 ^{o}
Depth of slotted head (D)	D / 5
Radius of chamfer (R)	0.1 * D
Angle of chamfer	45 ^{o}
Depth of thread (d)	0.54125 * P

Where, D = Major diameter
P = Pitch of thread

Table. 2. Formula for parts of Flange Coupling

Nomenclature	Formula Used
FLANGE	
Base Diameter of Flange (BD)	D * 4
Base Height of Flange (BH)	D / 2
Diameter of Hub (D1)	D * 2
Height of Hub (H1)	D
Total Height of Flange (H)	BH + H1
Diameter of holes (D2)	D * 0.3
Distance between centers of 2 holes	D * 3
Fillet radius	D / 15
Fillet angle	45 ^{o}
Number of holes	4
SHAFT	
Height of shaft	D * 2.5
Depth of tapper	D * 2
Chamfer radius	D / 22.5
Chamfer angle	45 ^{o}
Key	
Length of rectangle	(0.7 * (D) ^ (0.79))
Width of rectangle	(0.7 * (D) ^ (0.67))
Height of key	D * (16 / 9)
Tapper angle	atan (1/100) ^{o}

Where, D = Diameter of Shaft

7. CONCLUSION

Simple Structures like cube, cuboid, cylinder, cone and sphere are successfully generated with a fully defined sketch. On selection of face by the user it provides suitable features like boss extrude and extrude-cut (along with depth of cut). On selection of edge it provides certain operations like chamfer and fillet for creation of prismatic models.
Automation of Mechanical Components like nut, bolt, screw and flange coupling (along with assembly) was successful in all of the geometric planes which involves limitation in the value provided by the user to perform particular operations.
Thread operation for nut, bolt and screw cannot be performed for size above M177 but it still generates the complete model without the tread. Saving of the part file is restricted to desktop location only. Launching of SolidWorks Application requires dependencies and references beforehand to run the modules and macros. A Macro location must be provided to run it inside the SolidWorks Application. Extrude-cut feature for prismatic model is only limited to the center of the selected face. Extrusion of cone on a selected face is not successful and the boss extrusion on the selected face works on the basis of mid plane extrusion. The user must know the restrictions in the software to generate model successfully.

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