Secondary Development of Solid works for Standard Components Based on Database

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Abstract: *This work employs Visual Basic, SolidWorks and Access to develop a database management system for standard components. Two technologies namely coding and parameter design have been used. The database management system for standard components has been built. The secondary development of Solid Works has been performed by Visual Basic 6 and the system is interconnected with SolidWorks. This system has various advantages such as simple operation, auto coding, easy maintenance and greater flexibility. By the use of both SolidWorks and Visual Basic, you can quickly create similar parts library, to improve design efficiency and correctness of great help for the enterprise in product parametric design process. This can also be used as a reference.*

Keywords: SolidWork, Visual Basic, Coding, Secondary Development.

1. Introduction

Modern design has made life so easier by reducing time wastage and maintaining proper usage of resources. By using SolidWorks and Visual Basic, products having the same mechanical characteristics and shapes, the corresponding entity model library can be created through parametric design. This can improve design efficiency and simplify design work.

1.1 SolidWorks

SolidWorks is a CAD/CAE/CAM/PDM desktop system, and the first 3D mechanical CAD software in Windows developed by the SolidWorks Company. It provides product-level automated design tools (Liu and Ren 2005) [1], [2]. SolidWorks is a tool that helps design engineers to harness their imagination and add creativity to their designs. It is a 3D mechanical CAD (computer-aided design) program that is used to maximize the productivity of your design and engineering resources, to create products that are better, faster, and more cost-effectively. Solidworks runs on Microsoft Windows. It has been observed that SolidWorks is one of the easiest programs to use in designing and up to now it is projected that it is currently used by over 2 million engineers and designers at more than 165,000 companies worldwide.

1.2 Visual Basic

Visual Basic is easy to learn programming language. It is much easier to learn than other language (like C++) and yet it is powerful programming language. The most fundamental part of a Visual Basic program is the form. The form object is the most fundamental building block of the interface upon which controls can be placed and behind which code can be created. A program can contain many forms. Controls are objects to be contained within a form. There are virtually thousands of controls available and include: textboxes, labels, timers, dialog boxes, buttons, and many more. Most students will use controls that allow for entering and displaying data, accessing other applications, and the processing of data. Each object in Visual Basic has its own set of: properties, methods, and events. Due to its popularity there are many good resources that can help you learn the language. You can much more easily find the answers to your programming problems than other programming languages. Visual Basic code can be added to a program in two ways: (1) behind a form (2) into a module. A form is graphical but behind it resides a coding area. This area is available for general coding of subroutines as well as code that respond to an object's events. A module is another coding area available for subroutines, but cannot be used for event code. 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 [3].

2. The basic principle of Solidwork Secondary Development

2.1 Introduction To Solidworks API

An Application Programming Interface (API) is an interface which specifies how some software components should interact with each other. As a result, a direct integration between different applications can be developed [4]. Communication between the different packages can be simplified and enhanced by the use of the Visual Basic Application Programmer Interface (API). 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 modeling. SolidWorks API is a tree like hierarchical structure. SolidWorksAPI covers all the functions of the software
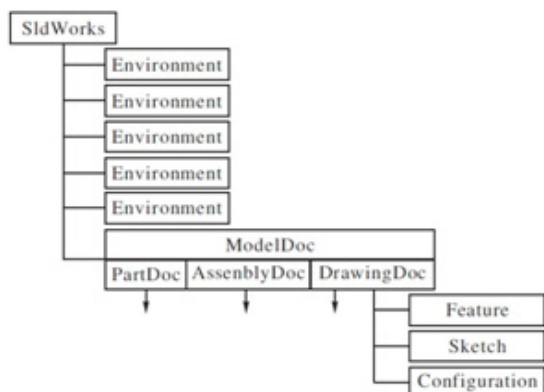
Figure 1: SolidWorks API objectives

2.2 The Basic Idea behind SolidWork Secondary Development

Currently, the SolidWorks software development is done in two different ways. One of them is the one based on OLE (Object Linking and Embedding) automation technology. The other one is that based on COM (Component Object Model) specification. SolidWorks program provides a 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, VB supports both OLE and COM technologies and hence this paper has used VB.

3. Creating Solidworks Add-In and User Interfaces

3.1 Basic Introduction

Here below are some basic commonly used API functions. They are more useful in the process of creating user interfaces. SolidWorks connection we need to have some commands which will allow us to open SolidWorks interface and here below is its syntax:

```
Dim swApp As Object
Set swApp = CreateObject ("SldWorks.Application")
```

With these two commands above, a SolidWorks interface will be open.

The establishment of a new part and assembly drawings

To create a parts diagram:
```
Dim Part As Object
Set Part = swApp.newpart
```
Create assembly drawings:
```
Dim Assem As Object
Set Assem = swapp.NewAssembly
```

These two functions create a new workspace, to enable us to carry out modeling and assembly.

3.2 Code Generation

In this paper, add-in code was created in order to build menus in the SolidWorks environment for automating the process of drawing a bolt and nut. The menus were built by using VB 6 integrating with SolidWorks API (Application Programming Interface). An ActiveX DLL project was developed in VB including SolidWorks libraries as references. These include SldWorks Type Library and SolidWorks Exposed Type library. These libraries allow control of SolidWorks commands and functions. The developed add-in code in this study is:

```
Private Sub Command1_Click()
Dim swApp As Object
Dim Part As Object
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long
Dim FeatureData As Object
Dim Feature As Object
Dim Component As Object
Set swApp = CreateObject("sldworks.application")
Set Part = swApp.NewPart(App.Path & "\Model\BOLTGBT5782.prtdot", 0, 0#, 0#)
Set Part = swApp.ActiveDoc
Part.Parameter("D1@sketch2").SystemValue = (TextsLS.Text / 1000)
Part.Parameter("D1@stretch1").SystemValue = (TextsLS.Text / 1000)
Part.Parameter("D1@sketch3").SystemValue = (TextsLS.Text / 1000)
Part.Parameter("D1@stretch2").SystemValue = (TextsLS.Text / 1000)
Part.Parameter("D2@sketch4").SystemValue = (TextsLS.Text / 1000)
Part.ClearSelection
Part.EditRebuild
Part.ShowNamedView2 "isometric", 7
Part.ViewZoomtofit2
End Sub
```


Figure 2: A bolt drawn through generated codes

4. Results

Through this paper, a bolt has been able to be drawn by the use of VB commands and then being able to be interconnected

with SolidWorks. VB 6 was employed in this paper due to its capabilities for building the macros in all Microsoft software and, therefore, accesses the functions of SolidWorks API for developing the graphical user interfaces and new menus. VB 6 is also considered as a flexible programming language that can be integrated with different applications. Therefore, the secondary development approach presented in this paper enhances and simplifies the automation process of drawing standard documents.

5. Conclusion

SolidWorks API was employed for automating the process of drawing a bolt. SolidWorks plays a big role in this project through the use of macros commands. For this purpose, an ActiveX DLL project was created in VB 6 and a plu-in file in .dll format was generated for developing new menus into SolidWorks environment. The developed approach results in saving time and cost when drawing standard components. The future work of this research can be directed towards extending the knowledge base for adding more standard components, as it is obvious that there are so many standard components. In addition, a comprehensive automated process can be achieved by extending the VB codes and macros. By so doing, the errors can be overcome in order to make the process more efficient and in one step rather than many steps. The problem of the interface between SolidWorks and Visual Basic is resolved in this study, realizing an effective combination of the advantages of both SolidWorks and Visual Basic.

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