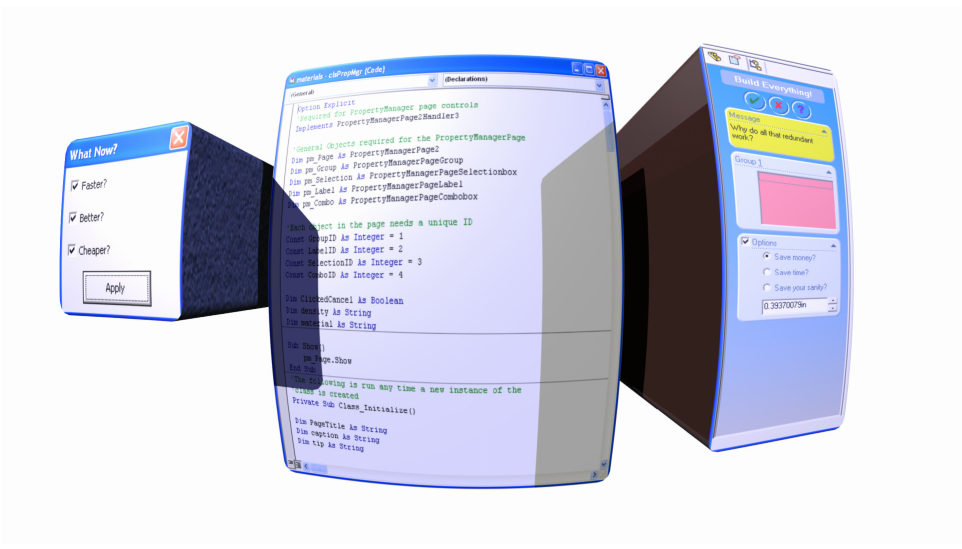
Automating SolidWorks 2006 using Macros



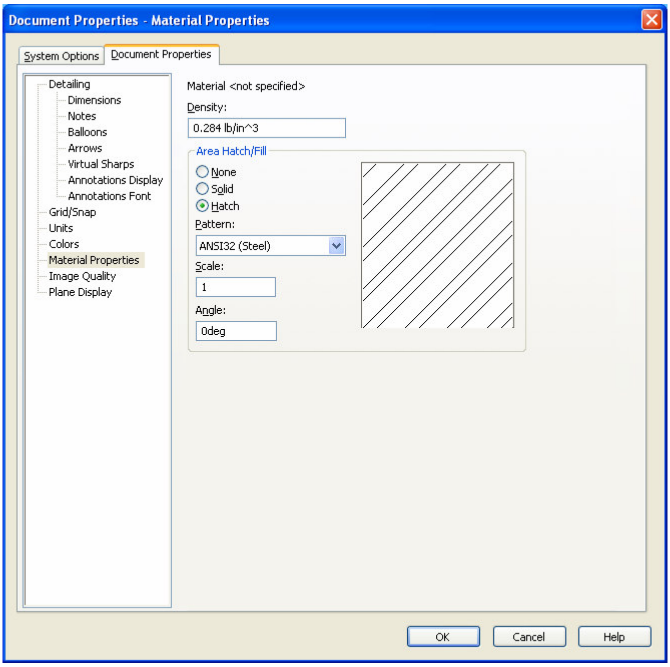
```
Option Explicit
'Required for PropertyManager page controls
Implements PropertyManagerPage2Handler3

'General Objects required for the PropertyManagerPage
Dim pm_Page As PropertyManagerPage2
Dim pm_Group As PropertyManagerPageGroup
Dim pm_Selection As PropertyManagerPageSelectionbox
Dim pm_Label As PropertyManagerPageLabel
Dim pm_Combo As PropertyManagerPageCombobox

'Each object in the page needs a unique ID
Const GroupID As Integer = 1
Const LabelID As Integer = 2
Const SelectionID As Integer = 3
Const ComboID As Integer = 4

Dim ClickedCancel As Boolean
Dim density As String
Dim material As String

Sub Show()
    pm_Page.Show
End Sub
'The following is run any time a new instance of the
'class is created
Private Sub Class_Initialize()

Dim PageTitle As String
Dim caption As String
Dim tip As String
```

A Visual Basic for Applications tutorial for SolidWorks users

Now Includes PDMWorks

Written by Mike Spens

Material Properties



Material <not specified>

Density: 0.284 lb/in^3

Area Hatch/Fill

None [✓ Hatch] Solid

Pattern: ANSI32 (Steel)

Scale: 1

Angle: 0deg

- **Basic Material Properties**
- **User Forms**
- **Arrays**
- **Working with Assemblies**
- **Selection Manager**
- **Verification and Error Handling**

Introduction

This exercise is designed to go through the process of changing document properties such as density. It will also review several standard VB programming methods. This exercise could easily be extended to copying document properties to other SolidWorks files. It will also examine a method for parsing through an assembly tree to change some or all of the parts in the assembly.

Part 1: Basic Material Properties

The initial goal will be to make a tool that allows the user to select a material from a pull-down list or combo box. When he clicks an OK button, the macro should apply the appropriate density to the active part. The user should be able to cancel the operation without applying a material density.

Macro Limitations

One of the limitations of recorded macros in SolidWorks is that some dialog box actions are not recorded. At the time of publication, material option settings were not recorded. To automate selections in dialogs you will need to find the appropriate method or property from the API help. This can take some searching, so I will point out several throughout the example.

Initial Code

1. Create a new macro by selecting **Tools, Macro, New** or clicking 📄.

2. Save it as *materials.swp.*

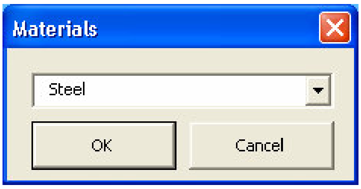
3. Declare `swApp` and `Part` publicly so they can be used in other modules and add the code to capture the active document.

```
Option Explicit
Public swApp As SldWorks.SldWorks
Public Part As SldWorks.ModelDoc2
```

```
Sub main()

Set swApp = Application.SldWorks
Set Part = swApp.ActiveDoc
End Sub
```

User Forms

Through the first exercises, two different methods for gathering user input have been introduced. One method was the input box. That is fine for simple text input. The other method was Excel. This improved things by adding command buttons and multiple input values. Many times it is better to be able to organize user input in a dialog box or form. This is a standard with practically any Windows software. These allow users to input text, click buttons to activate procedures, and select options. After this example you will have created a form that looks like the one shown using a pull-down list or ComboBox and two command buttons.

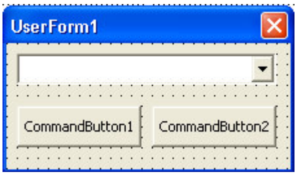


4. Add a user form to your macro by clicking or select **Insert, UserForm**.

A new user form window will show on the screen along with the same **Controls Toolbox** we saw in Excel. We need to add a **ComboBox** control and two **CommandButton** controls to our form.

5. Drag and drop each control from the Control Toolbox onto your form.

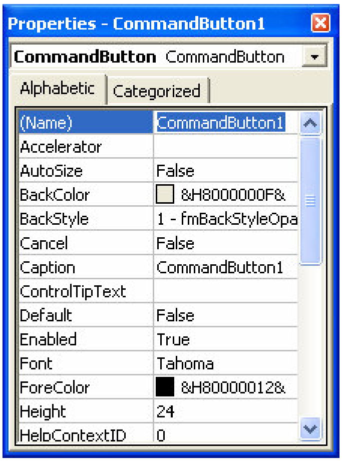
After adding the controls your form should look something like the following. The controls can be moved around and resized for visibility as desired. A good form is one that is compact enough to not be intrusive while still being easy to read and use.



Object Properties

Each of these controls has a list of properties that you can change. It would be impractical to have to remember that CommandButton1 is for OK and that CommandButton2 is for Close. So you can change their caption property to be OK and Close respectively.

6. Select **CommandButton1**. The Property list will show all of the object's properties. This is the same idea as the SolidWorks Property Manager.



(Name)	CommandButton1
Accelerator	
AutoSize	False
BackColor	☐ &H8000000F&
BackStyle	1 - fmBackStyleOpa
Cancel	False
Caption	CommandButton1
ControlTipText	
Default	False
Enabled	True
Font	Tahoma
ForeColor	■ &H80000012&
Height	24
HelpContextID	0

7. Change the text next to **Caption** to **OK** and change the **Name** to **cmdOK**. Change the **Default** property to **True**.

The Default property will make the OK button the default of the available command buttons. If the user selects a material from the list and hits the enter key, it will be the same as clicking the button.

8. Select **CommandButton2** and change its **Caption** to **Close** and its **Name** to **cmdClose**.

9. Select the form itself and change its **Name** to **frmMaterials,** its **Caption** to **Materials** and **ShowModal** to **False**.

Setting the ShowModal property to False allows the SolidWorks user to interact with SolidWorks while the dialog box displays. This can be helpful if the user forgot to select some parts before running the macro.

10. Select the **ComboBox** and set its **Name** to **cboMaterials**, its **ColumnCount** property to **2** and its **ColumnWidths** property to **80 pt**.

The ColumnCount property will allow us to see both the material name and its density in the same combo box. The ColumnWidths property sets the width of each column in our combo box control.

Populating and Showing the Form

The macro needs a line of code that will display the form at the right time. If you run the macro right now, it will only attach to the SolidWorks application and active document. But it would never show the form.

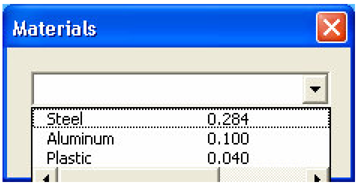
11. Change the code in your main procedure as follows.

```
Sub main()

Set swApp = Application.SldWorks
Set Part = swApp.ActiveDoc

'Set up materials list in the form: (column, row)
Dim MyProps(1, 2) As String
```

```
MyProps(0, 0) = "Steel"
MyProps(1, 0) = "0.284"
MyProps(0, 1) = "Aluminum"
MyProps(1, 1) = "0.100"
MyProps(0, 2) = "ABS Plastic"
MyProps(1, 2) = "0.036"

'Populate pull-down list with values
frmMaterials.cboMaterials.Column = MyProps

frmMaterials.Show

End Sub
```

12. Go ahead and run the main procedure as a test.

You should see your new dialog box show up. This is a result of the **frmMaterials.Show**. Every user form object has a show method that makes the form visible to the user.



Steel	0.284
Aluminum	0.100
Plastic	0.040

If you click on the combo box, you should see Steel, Aluminum and Plastic listed with their corresponding densities in lbs/in^3. The OK and Close buttons are not yet defined however.

13. To close the running macro, click the close button in the upper-right corner or click the stop button in the VBA editor. Do not close the VBA interface yet.

It was determined that the combo box control would have two columns for data when its properties were set. These properties were set at **design time** since they were set before the code was running. These two columns were filled with an array of values at **run time** or while the program was running.

Arrays

An array is simply an m by n matrix of values. To declare an array you use the form **Dim variablename(m,n) As type**.

`MyProps(1, 2)` was declared as a string type variable. In other words, you made room for two columns and three rows of string information in that one variable. "Wait! I thought you declared one column and two rows"! If you haven't learned this already, programmers count from zero. If you stick to this practice, you will avoid confusion in most cases.

ComboBox.Column Property

To populate the combo box with the array, you must tell the macro where to put things. By typing `frmMaterials.cboMaterials.Column = MyProps` you have told the procedure that you want to populate the column of the cboMaterials object in the frmMaterials object with the values in the MyProps array. The combo box control automatically creates a row for each row in the array. There is no need to assign the combo box the number of rows required, just the number of columns.

Command Button Code

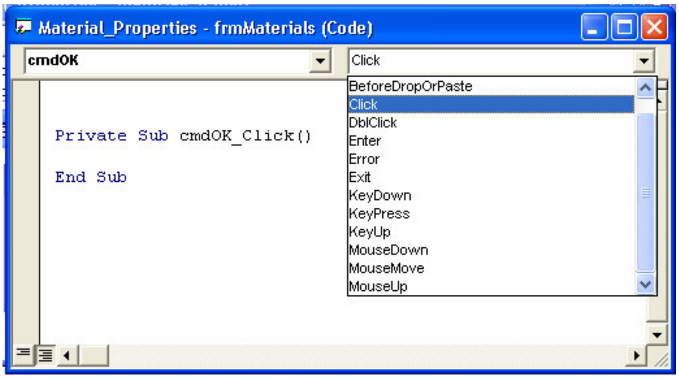
In the Model Parameters exercise you learned how to add code to a command button. You will use the same method for the OK and Close buttons. You can start with the easy one.

14. In the VBA editor, switch to the user form window and **double-click on the Close button**. This will open a code module window and will create a name for the procedure.

The name of the procedure is directly connected to the name of the object. For example, the Close button is named cmdClose. So the procedure is as follows.

```
Private Sub cmdClose_Click()

End Sub
```

The _Click portion of the name is related to the action performed on the button that causes this code to run. The default action for command buttons is **click**. That is what most people do with buttons isn't it? In the code module window you will see a list of actions that can be performed on each object. This is a good way to learn how to make your user interfaces more complicated. You can create code for any action in the list.



```
Private Sub cmdOK_Click()

End Sub
```

End Statement

All the Close button should do is end the macro without applying any material properties. That is easy.

15. Add the **End** statement as shown.

```
Private Sub cmdClose_Click()
End
End Sub
```

Now you will get your macro to do something real!

16. Switch back to the user form window and **double-click on the OK button**.

17. Add the code in the newly created procedure as follows.

```
Private Sub cmdOK_Click()
Dim density As Double

'The required density is in kg/m^3
'so multiply by a scale factor

density = frmMaterials.cboMaterials.Column(1, _
      frmMaterials.cboMaterials.ListIndex) * 27680


Part.SetUserPreferenceDoubleValue
swMaterialPropertyDensity, density

End
End Sub
```

Code Description

At this point the macro is fully functional. Try it out on a part. It will not work on assemblies at this point.

You first declared a new variable `density`. This variable is declared as a **Double** since it will be used in the call **SetUserPreferenceDoubleValue**.

Working with ComboBoxes

The procedure also sets the variable `density` to the value stored in the combo box object. You used a couple properties of the combo box object to get the density value that corresponds to the material selected. The goal is to extract the value from the combo box's second column and the current row.

combobox.Column(*column, row*) [= *Variant*]

combobox.ListIndex [= *Variant*]

The **Column** property should be 1 (remember to count from 0) and the current row number is represented by the **ListIndex** property of the same combo box.

The **Column property** requires (*column*, *row*) input, so you have listed column 1 and then the current row. Adding a multiplier of 27680 converts the combo box value from lbs/in^3 to the required kg/m^3 (specified in the API help).

The last thing added to the code was the `End` statement prior to `End Sub`. This automatically terminates the program so the user does not have to click OK and then Close to finish.

SolidWorks Materials Library

You could now go in several different directions from your base macro. You can add additional columns to your combo box control for hatch type, shaded color and lighting options. This can be useful if you wish to select a material from the list and have the density, hatch, color, texture and lighting set accordingly. Alternatively, you can simply set the SolidWorks Material settings based on the material library. The remainder of the exercise will assume the SolidWorks Material Library will be used so the density setting can be removed.

18. Add the following code changes to set the material settings from the SolidWorks Material Library.

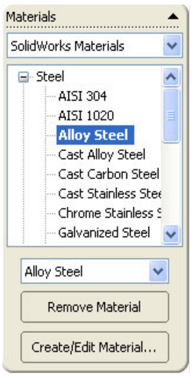
```
Sub main()

Set swApp = Application.SldWorks
Set Part = swApp.ActiveDoc

'Set up materials list in the form: (column, row)
Dim MyProps(2, 2) As String

MyProps(0, 0) = "Steel"
MyProps(1, 0) = "0.284"
MyProps(2, 0) = "Alloy Steel"
MyProps(0, 1) = "Aluminum"
MyProps(1, 1) = "0.100"
MyProps(2, 1) = "6061 Alloy"
MyProps(0, 2) = "ABS Plastic"
MyProps(1, 2) = "0.036"
MyProps(2, 2) = "ABS"
```

```
'Populate pull-down list with values
frmMaterials.cboMaterials.Column = MyProps

frmMaterials.Show

End Sub

---------------------------------------------

Private Sub cmdOK_Click()
'Dim density As Double

Dim material As String

'The required density is in kg/m^3
'so multiply by a scale factor

'density = frmMaterials.cboMaterials.Column(1, _

    frmMaterials.cboMaterials.ListIndex) * 27680


material = frmMaterials.cboMaterials.Column(2, _
       frmMaterials.cboMaterials.ListIndex)


'Part.SetUserPreferenceDoubleValue _

    swMaterialPropertyDensity, density

Part.SetMaterialPropertyName2 "", _
    "solidworks materials.sldmat",  material

End
End Sub
```

SetMaterialPropertyName2 is a method of the special ModelDoc2 object called PartDoc. The ModelDoc2 object can be broken into three additional derived objects that have functionality specific to the document type. For example, AddMate3 is actually a method of the AssemblyDoc object. However, the methods from these derived objects still work with the ModelDoc2 object.

void = PartDoc.SetMaterialPropertyName2(configName, database, name)

- **configName** is passed a string representing the configuration to change. If it is passed an empty string, all configurations are changed.
- **database** defines where the material comes from. This can be a full path to a library or an empty string. If it is an empty string, the default material library is assumed.
- **name** is a string representing the material name as listed in the material library. For example, "Alloy Steel" would assign the material as shown below.



Part 2: Working with Assemblies

You can now extend the functionality of this macro to assemblies. When completed, you will have a tool that allows the user to assign material properties to selected components in an assembly. After all, it is usually when trying to figure out the mass of an assembly when you realize that you have forgotten to set the mass properties for your parts. And who wants to open up every part and set the properties individually?

Selection Manager

In the Model Parameters exercise we discussed selections using the SelectByID2 method. However, the code had to be specific to the item being selected. We had to pass the name of the component. Instead, you can employ a method that is similar to several functions in SolidWorks. You can require the user to pre-select the components he wishes to change prior to running the macro. The only trick is to program your macro to change the settings for each item the user selects. The Selection Manager object will be the key tool for this part of the macro.

SelectionManager Object

Connecting to the SelectionManager object is similar to setting the ModelDoc2 object (called `Part`). The SelectionManager object is a child of the ModelDoc2 object.

19. Add the following code to your main procedure to declare the selection manager and to attach to it.

```
Option Explicit
Public swApp As SldWorks.SldWorks
Public Part As SldWorks.ModelDoc2
Public SelMgr As SldWorks.SelectionMgr
Sub main()

Set swApp = Application.SldWorks
Set Part = swApp.ActiveDoc
Set SelMgr = Part.SelectionManager
...
```

Some of the things you can access from the **SelectionManager** object are the selected object **count**, **type**, or even the **xyz point** where the object was selected in space. In this macro you will need to access the selected object count, or number of items selected, and the component object that was selected. Remember that components in SolidWorks can be either parts or assemblies. Since we can only set density for parts, we will need to make sure the item selected is a part. For each item in the selection manager, you must get the ModelDoc2 object and then set its material settings.

20. Add the following code under the OK button’s Click event as shown.

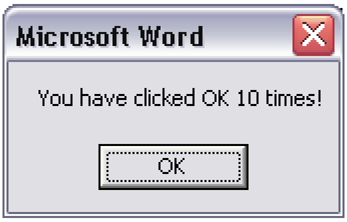
```
Private Sub cmdOK_Click()
'Dim density As Double
Dim material As String
Dim Component As SldWorks.Component2
Dim Model As SldWorks.ModelDoc2
Dim i as Integer

'The required density is in kg/m^3 so we're multiplying _
by a scale factor

'density = frmMaterials.cboMaterials.Column(1, _
       frmMaterials.cboMaterials.ListIndex) * 27680


material = frmMaterials.cboMaterials.Column(2, _
       frmMaterials.cboMaterials.ListIndex)


For i = 1 To SelMgr.GetSelectedObjectCount2(-1)
    Set Component = _
        SelMgr.GetSelectedObjectsComponent3(i, -1)
    Set Model = Component.GetModelDoc
    Model.SetMaterialPropertyName2 "", _
    "solidworks materials.sldmat",  material

Next i

Part.ClearSelection2 True
'End
End Sub
```

For … Next Statements and Loops

As was mentioned earlier, you want to set the material properties for each part that was selected by the user. What if the user has selected 500 parts? You certainly do not want to write 500 lines of code for each item selected. In many cases you will want to apply the same action to a variable number of items.

For … Next statements allow you to repeat a section of code over as many iterations as you want. Computers are great at this! You just have to know how many times to loop through the code if you use a For … Next statement.

```
For I = 0 To 10
    MsgBox "You have clicked OK " & I & " times!"
Next I
```



You have clicked OK 10 times!

Add this sample code to a procedure and then run the procedure. You get a VB MessageBox stating exactly how many times you have clicked OK. That is great if you know exactly how many times the loop needs to process. In the macro, you do not know how many times to repeat the loop because you do not know how many parts the user might select. So you can make the procedure figure it out for you. You can use the SelectionManager object to help.

Using Selection Manager Methods

You need to determine how many times to run through the loop. This should correspond to the number of items selected. This number can be extracted from the Selection Manager through its **GetSelectedObjectCount2** method. The argument passed is related to a selection Mark. A value of -1 indicates all selections will be counted, regardless of Mark. See the SolidWorks API Help for more information.

```
For i = 1 To SelMgr.GetSelectedObjectCount2(-1)
    (loop code here)
Next i
```

The For loop starts with an initial value of 1 so that it will only loop if the number of selected items is greater than zero.

GetSelectedObjectsComponent3

The next thing you need to do is get the ModelDoc2 object for each of the selected items. You will need this object so you can set its material properties. This requires a two-step process. The first thing you need to access will be the Component object. To get the component object you use the SelectionManager's **GetSelectedObjectsComponent3(*item number, Mark*)** method. The Mark argument is again -1 to get the component regardless of selection Mark. You must use the Component object to access its underlying ModelDoc2 object. Notice the declarations for `Model` and `Component`. They are specific to the type of SolidWorks object we are accessing (early binding).

GetModelDoc

The **Component.GetModelDoc** method allows you to access the underlying ModelDoc object.

Now that you have the ModelDoc2 object assigned to variable `Model`, you can use the same code to set the material properties.

Component vs. ModelDoc Objects

If you have been wondering why we have to take the extra time to dig down to the ModelDoc2 object of the Component object, this discussion is for you. If you have not and it all makes perfect sense, move on to the next subject.

Think of it this way – a Component object understands specific information about the ModelDoc2 it contains. It knows which configuration is showing, which instance it is, if it is hidden or suppressed and even the component's rotational and translational position in the assembly. These are all things that can be changed in the Component object. However, if you want to change something specific to the underlying part such as its material density, or to the underlying assembly such as its custom properties, then you must take the extra step of getting to the actual ModelDoc2 object.

Verification and Error Handling

At times you may want to check the user's interactions to make sure they have done what you expected. After all, your macro may not have a user's guide. And even if it does, how many people really read that stuff? If you're reading this, you probably would. What about the other 90% of the population?

You can make sure the user is doing what you think they should. First, define some criteria.

- Is the user in an assembly? The user must be in an assembly to use the **GetSelectedObjectsComponent3** method.

- If the active document is an assembly, has the user pre-selected at least one part? If not, they will think they are applying material properties while nothing happens.

- Has the user selected items other than parts? If they select a plane, the macro may generate an error or crash because there is no ModelDoc2 object.

21. Add the following additions to check for any of these possible user errors.

```
Private Sub cmdOK_Click()
'Dim density As Double
Dim material As String
Dim Component As SldWorks.Component2
Dim Model As SldWorks.ModelDoc2
Dim i as Integer

'The required density is in kg/m^3
'so multiply by a scale factor

'density = frmMaterials.cboMaterials.Column(1, _
    frmMaterials.cboMaterials.ListIndex) * 27680

material = frmMaterials.cboMaterials.Column(2, _
    frmMaterials.cboMaterials.ListIndex)

If Part.GetType = swDocASSEMBLY Then
    'check for selected components
    If SelMgr.GetSelectedObjectCount2(-1) < 1 Then
        MsgBox "Please select one or more " _
            & "components first.", vbCritical
    End
End If
    For i = 1 To SelMgr.GetSelectedObjectCount2(-1)
        Set Component = _
            SelMgr.GetSelectedObjectsComponent3(i, -1)
        Set Model = Component.GetModelDoc
        'if the model is a part
        If Model.GetType = swDocPART Then
            Model.SetMaterialPropertyName2 "", _
                "solidworks materials.sldmat",  material
        End If
    Next i
ElseIf Part.GetType = swDocPART Then
    Part.SetMaterialPropertyName2 "", _
        "solidworks materials.sldmat",  material
Else
    MsgBox "You can only use this tool " _
        & "in parts or assemblies.", vbExclamation
End If
'End
End Sub
```

GetType Method

The first If statement is used to determine the active document type. This could be either a part or an assembly. If the type is the SolidWorks constant **swDocPart** then the selected object is a part and we can apply its material properties. The **GetType** method allows us to find the type of model we have accessed. The constant **swDocAssembly** refers to an assembly (in case you had not guessed).

If … Then…Else Statements

If the active document is an assembly, you should check if the user has selected at least one component before continuing. Checking the GetSelectedObjectCount property of the Selection Manager object for a value less than one will accomplish this. If it is less than one the user has failed to select anything.

MessageBox

You can make use of the VB **MessageBox** function to give the user feedback. The MessageBox function allows you to tell the user anything in a small dialog box. This dialog box has an OK button by default, but it can have Yes and No buttons, OK and Cancel, or other combinations. If you use anything besides the default you can use the return value to determine which button the user selected.

The macro will now provide feedback to the user to make sure they are using the macro correctly. It makes good programming sense to build good error trapping into your macros. Users tend to quickly get frustrated when a tool crashes or generates undesired results.

Conclusion

This macro now adds some additional value to the SolidWorks interface. In a simple format you have given the user a quick way to change material settings in any selected part. This functionality can easily be extended to changing or listing any setting related to a model.