



Tutorial 250320-01	Autodesk Inventor	Addin	Vb.net	Arthur Knoors
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## Creating an Autodesk Inventor Add-in using Visual Studio Vb.net 2022 Part 02.

### Get the Inventor application from the Addin.

First, we need to get the Inventor Application from the Addin to communicate with.

```
Imports System.Runtime.InteropServices
Imports Inventor
<ProgIdAttribute("MyInventorAddIn.StandardAddInServer"),
GuidAttribute("ad0a3353-0c3b-42a6-87ba-e1a86f4529e7")>
Public Class StandardAddInServer

    Implements Inventor.ApplicationAddInServer

    Private m_inventorApp As Inventor.Application

    Public Sub Activate(AddInSiteObject As ApplicationAddInSite, FirstTime As Boolean)
        Implements ApplicationAddInServer.Activate

        MsgBox("My First Add In Activated")

        m_inventorApp = AddInSiteObject.Application
        MsgBox(m_inventorApp.UserName)

    Public Sub Deactivate() Implements ApplicationAddInServer.Deactivate
        Throw New NotImplementedException()
    End Sub

    Public Sub ExecuteCommand(CommandID As Integer) Implements
ApplicationAddInServer.ExecuteCommand
        Throw New NotImplementedException()
    End Sub

    Public ReadOnly Property Automation As Object Implements ApplicationAddInServer.Automation
        Get
            Throw New NotImplementedException()
        End Get
    End Property
End Class
```



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## Create a button definition:

First, we need to create two global properties as below.

```
Imports System.Runtime.InteropServices
```

```
Imports Inventor
```

```
<ProgIdAttribute("MyInventorAddIn.StandardAddInServer"),
```

```
GuidAttribute("ad0a3353-0c3b-42a6-87ba-e1a86f4529e7")>
```

```
Public Class StandardAddInServer
```

```
    Implements Inventor.ApplicationAddInServer
```

```
    Private m_inventorApp As Inventor.Application
```

```
    Private WithEvents m_sampleButton As ButtonDefinition
```

```
    Private guid As String = Nothing
```

```
    Public Sub Activate(AddInSiteObject As ApplicationAddInSite, FirstTime As Boolean) Implements  
ApplicationAddInServer.Activate
```

```
        MsgBox("My First Add In Activated")
```

```
        m_inventorApp = AddInSiteObject.Application
```

```
        MsgBox(m_inventorApp.UserName)
```



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Create the GUID for the Button definition:

*Try*

```
Dim t As Type = GetType(MyInventorAddin.StandardAddInServer)
```

```
Dim customAttributes() As Object = t.GetCustomAttributes(GetType(GuidAttribute), False)
```

```
Dim guidAttribute As GuidAttribute = CType(customAttributes(0), GuidAttribute)
```

```
guid = "(" + guidAttribute.Value.ToString() + ")"
```

*Catch*

*End Try*



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## Create the Button Definition selves (KISS):

We create a simple button without bells (KISS) with the code below. We need to use the above create GUID.

```
m_sampleButton = m_inventorApp.CommandManager.ControlDefinitions.AddButtonDefinition("My First  
Button", "MyFirstButton", Inventor.CommandTypeEnum.kShapeEditCmdType, guid, "My Description", "My Tool  
Tip Text",,, ButtonDisplayEnum.kDisplayTextInLearningMode)
```

```
    If FirstTime Then
```

```
        AddToUserInterface()
```

```
    End If
```

```
End Sub
```



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## Add this button to the Ribbon of Autodesk Inventor (ZeroDoc and ToolsTab)

We create a button by adding this code to the class in the Ribbon of Inventor In the Tools tab of ZeroDoc.

You can also choose instead of ZeroDoc, (Part Assembly Drawing, IFeature, Presentation).

```
Private Sub AddToUserInterface()
```

```
Dim r As Ribbon = m_inventorApp.UserInterfaceManager.Ribbon.Item("ZeroDoc") 'Part Assembly Drawing, Ifeature, Presentation
```

```
Dim rt As RibbonTab = r.RibbonTabs.Item("id_TabTools")
```

```
Dim rp As RibbonPanel = rt.RibbonPanels.Add("My First Ribbon Panel", "MyFirstRibbonPanel", guid)
```

```
Dim cmd As CommandControl = rp.CommandControls.AddButton(m_sampleButton)
```

```
End Sub
```



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Add the Button Event on execute so the it can be executed:

```
Private Sub m_sampleButton_OnExecute(Context As NameValueCollection) Handles m_sampleButton.OnExecute
    MsgBox("You just pressed your first Addin Button", MsgBoxStyle.YesNo, "First Add In Button")
End Sub
```

```
Public Sub Deactivate() Implements ApplicationAddInServer.Deactivate
    Throw New NotImplementedException()
End Sub
```

```
Public Sub ExecuteCommand(CommandID As Integer) Implements
ApplicationAddInServer.ExecuteCommand
    Throw New NotImplementedException()
End Sub
```

```
Public ReadOnly Property Automation As Object Implements ApplicationAddInServer.Automation
    Get
        Throw New NotImplementedException()
    End Get
End Property
End Class
```



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## The complete code:

### The complete code should look like this:

```
Imports System.Runtime.InteropServices
```

```
Imports Inventor
```

```
<ProgIdAttribute("MyInventorAddIn.StandardAddInServer"),
```

```
GuidAttribute("ad0a3353-0c3b-42a6-87ba-e1a86f4529e7")>
```

```
Public Class StandardAddInServer
```

```
Implements Inventor.ApplicationAddInServer
```

```
Private m_inventorApp As Inventor.Application
```

```
Private WithEvents m_sampleButton As ButtonDefinition
```

```
Private guid As String = Nothing
```

```
Public Sub Activate(AddInSiteObject As ApplicationAddInSite, FirstTime As Boolean) Implements  
ApplicationAddInServer.Activate
```

```
MsgBox("My First Add In Activated")
```

```
m_inventorApp = AddInSiteObject.Application
```

```
MsgBox(m_inventorApp.UserName)
```

```
Try
```

```
Dim t As Type = GetType(MyInventorAddin.StandardAddInServer)
```

```
Dim customAttributes() As Object = t.GetCustomAttributes(GetType(GuidAttribute), False)
```

```
Dim guidAttribute As GuidAttribute = CType(customAttributes(0), GuidAttribute)
```

```
guid = "{" + guidAttribute.Value.ToString() + "}"
```

```
Catch
```

```
End Try
```

```
m_sampleButton = m_inventorApp.CommandManager.ControlDefinitions.AddButtonDefinition("My First  
Button", "MyFirstButton", Inventor.CommandTypesEnum.kShapeEditCmdType, guid, "My Description", "My Tool  
Tip Text",,, ButtonDisplayEnum.kDisplayTextInLearningMode)
```

```
If FirstTime Then
```



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*AddToUserInterface()*

*End If*

*End Sub*

*Private Sub AddToUserInterface()*

*Dim r As Ribbon = m\_inventorApp.UserInterfaceManager.Ribbon.Item("ZeroDoc") 'Part Assembly Drawing, Ifeature, Presentation*

*Dim rt As RibbonTab = r.RibbonTabs.Item("id\_TabTools")*

*Dim rp As RibbonPanel = rt.RibbonPanels.Add("My First Ribbon Panel", "MyFirstRibbonPanel", guid)*

*Dim cmd As CommandControl = rp.CommandControls.AddButton(m\_sampleButton)*

*End Sub*

*Private Sub m\_sampleButton\_OnExecute(Context As NameValueCollection) Handles m\_sampleButton.OnExecute*

*MsgBox("You just pressed your first Addin Button", MsgBoxStyle.YesNo, "First Add In Button")*

*End Sub*

*Public Sub Deactivate() Implements ApplicationAddInServer.Deactivate*

*Throw New NotImplementedException()*

*End Sub*

*Public Sub ExecuteCommand(CommandID As Integer) Implements ApplicationAddInServer.ExecuteCommand*

*Throw New NotImplementedException()*

*End Sub*

*Public ReadOnly Property Automation As Object Implements ApplicationAddInServer.Automation*

*Get*

*Throw New NotImplementedException()*

*End Get*

*End Property*

*End Class*



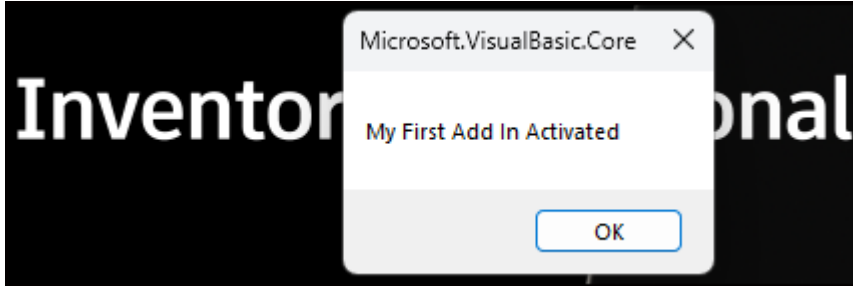


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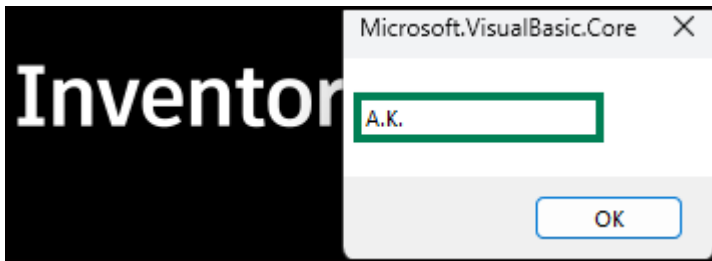
## Testing:

Rebuild again (see tutorial001) and start Autodesk Inventor'

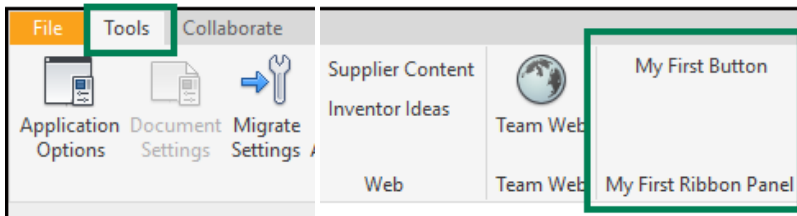
You should receive:



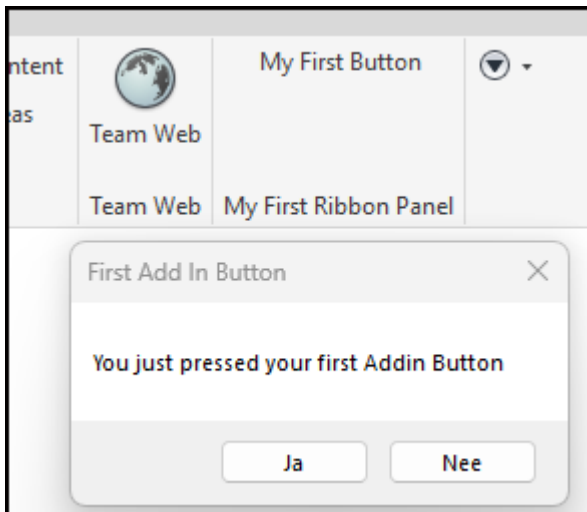
The username from the Inventor Software:



And when Inventor is loaded:



When pressing the button you should receive this message:





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Congrats you are now a step further and created your first add-in with a working button.