DWG to PDF ActiveX Control User Guide

DWG2PDF-X, an ActiveX control letting you convert dwg to pdf, dxf to pdf, dwf to pdf directly, no AutoCAD required.

Steps to use this control

Step 1 Please download the 32bit or 64bit version program according to your own requirement

DWG2PDF-X(32 bit) DWG2PDF-X(64 bit)

Step 2 Please unzip the program to the destination folder

Step 3 Register the control

Open the program folder and double click the file reg.bat to register the DWGTOPDFX.dll or DWGTOPDFX64.dll.

If you run Win7 and the reg.bat file cannot work, please click start button>>All programs>>Accessories, then find the "Command Prompt" and right click it to choose "Run as Administrator" option to open the command prompt, then try to use the command regsvr32 to register the DWGTOPDFX.dll or DWGTOPDFX64.dll.

Step 4 Do a test

Prior try out the active-x program with your own application, please try to give it a very first test with the example code enclosed with the free trial package, please make sure it will work as expected and then try out with other environments.

Help For Developers

- **Width**

  specifing the output PDF file width , unit is millimeter

  Property Let Width As Double

  No parameters

- **Height**

  specifing the output PDF file Height , unit is millimeter.

  Property Let Height As Double

  No parameters

- **ColorMode**

  Set the background color of the PDF file.

  Property Let ColorMode As Long

  No parameters

  Set Colormode Value as AutoCAD colorindex.

- **InputFile**

  specifing the input drawing file that want to be conveted.

  Property Let InputFile As String

  No parameters

- **DPI**

  Set PDF DPI (dots per inch) value, the default value is 100.

  Property Let DPI As Long

  No parameters

  VB script example:

```
Dim obj
Set obj= CreateObject("DWGTOPDFX.ConvertPDF")

obj.Width = 250 'uint is mm
obj.Height = 150 'uint is mm
obj.ColorMode = 1 '1--white&black 0--256color mode

Obj.DPI = 300

obj.AddFontPath "D:\Program Files\AutoCAD 2005\Fonts"
obj.InputFile = "E:\tempaa\44650-39b.dwg"
obj.PsPath = "c:\windows\system32\ps"
obj.Convert "E:\tempaa\44650-39b1.pdf"
```

- **PenWidth**

  specifing the pen width of the color, unit is millimetre.

  Property Get/Let PenWidth(nPenWidth As Long) As Double

  VB script example:

```
Dim obj
Set obj= CreateObject("DWGTOPDFX.ConvertPDF")

obj.Width = 250 'uint is mm
obj.Height = 150 'uint is mm
obj.ColorMode = 1 '1--white&black 0--256color mode

obj.AddFontPath "D:\Program Files\AutoCAD 2005\Fonts"
obj.InputFile = "E:\tempaa\44650-39b.dwg"

obj.PenWidth(1) = .1 'set line width, unit in mm
obj.PenWidth(2) = .1 'set line width, unit in mm
obj.PenWidth(3) = .1 'set line width, unit in mm
obj.PenWidth(4) = .1 'set line width, unit in mm
obj.PenWidth(5) = .1 'set line width, unit in mm

obj.PsPath = "c:\windows\system32\ps"
obj.Convert "E:\tempaa\44650-39b1.pdf"
```

- **PsPath**

  specified the ps path.

  Property Let PsPath As String

  No parameters

  Must setting PsPath before using convert method.

- **AddFontPath**

  Add a directory to the folder list in which DWG2PDFX searches for shx font file.

  Sub AddFontPath(szFontPath As String)

  VB script example:

```
Dim obj
Set obj= CreateObject("DWGTOPDFX.ConvertPDF")

obj.Width = 250 'uint is mm
obj.Height = 150 'uint is mm
obj.ColorMode = 1 '1--white&black 0--256color mode

obj.AddFontPath "D:\Program Files\AutoCAD 2005\Fonts"
obj.InputFile = "E:\tempaa\44650-39b.dwg"

obj.PenWidth(1) = .1 'set line width, unit in mm
obj.PenWidth(2) = .1 'set line width, unit in mm
obj.PenWidth(3) = .1 'set line width, unit in mm
obj.PenWidth(4) = .1 'set line width, unit in mm
obj.PenWidth(5) = .1 'set line width, unit in mm

obj.PsPath = "c:\windows\system32\ps"
obj.Convert "E:\tempaa\44650-39b1.pdf"
```

- **ViewCount**

  Get the drawing's layout count.

  Property Get ViewCount As Long

  No parameters

  Use Viewcount after the method InputFile.

  VB script example:

```
Dim objImage
Set objImage = CreateObject("DWGTOPDFX.ConvertPDF")

objImage.Width = 1024
objImage.Height = 768
'objImage.Background = 0
objImage.InputFile = "d:\aa.dwg" "'8th floor furniture.dwg"
```

```
objImage.Convert "e:\testa.pdf"

For I=1 to objImage.ViewCount
strViewName = objImage.ViewName(I)

strOutput = strViewName&".pdf" 'output file name, include path
objImage.ConvertView strOutput, strViewName
Next
If Err.Number < 0 Then
MsgBox Err.Description
End If
```

- **ViewName**

  Get layout name by layout index.

  Property Get ViewName(nIndex As Long) As String

  Use Viewcount after the method InputFile.

- **Convert**

  Convert the drawing merge all layout into a single PDF file.

  Sub Convert(szoutFile As String)

- **ConvertView**

  Convert specified layout in the drawing to PDF file.

  Sub ConvertView(szOutputFileName As String, view)

  *Use convertView after the method **InputFile**.*