

## Summary

Publish to PDF allows you to publish PDF documents from the OutputJob Editor.

Publish to PDF generates custom PDF documents based on all of the supported outputs in the OutputJob Editor including Schematic, PCB, PCB3D, OpenBus and BOM outputs. PDF bookmarks are created for each output and all of their corresponding components, nets, pins and ports where appropriate.

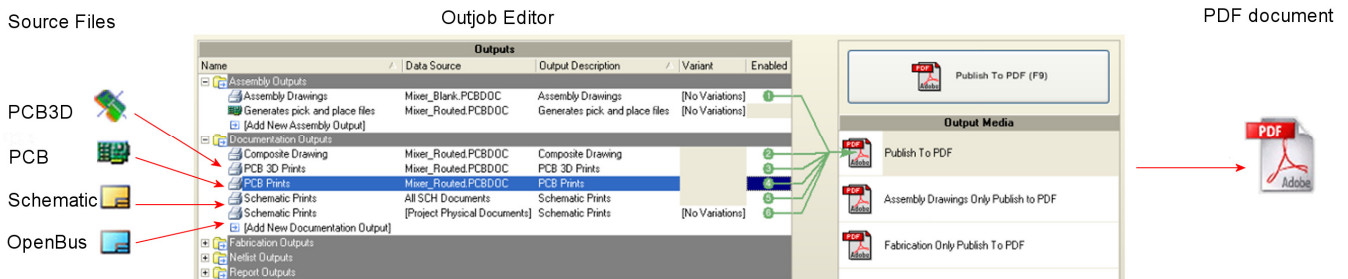


Figure 1. The Publish to PDF feature allows you to publish a PDF document from a number of different source files

## OutputJob Editor

The process of setting up outputs for your design and publishing them is performed in the OutputJob Editor. The OutputJob Editor allows you to define and manage Output Job Configuration files (\*.Out.job) for the current Design Project.

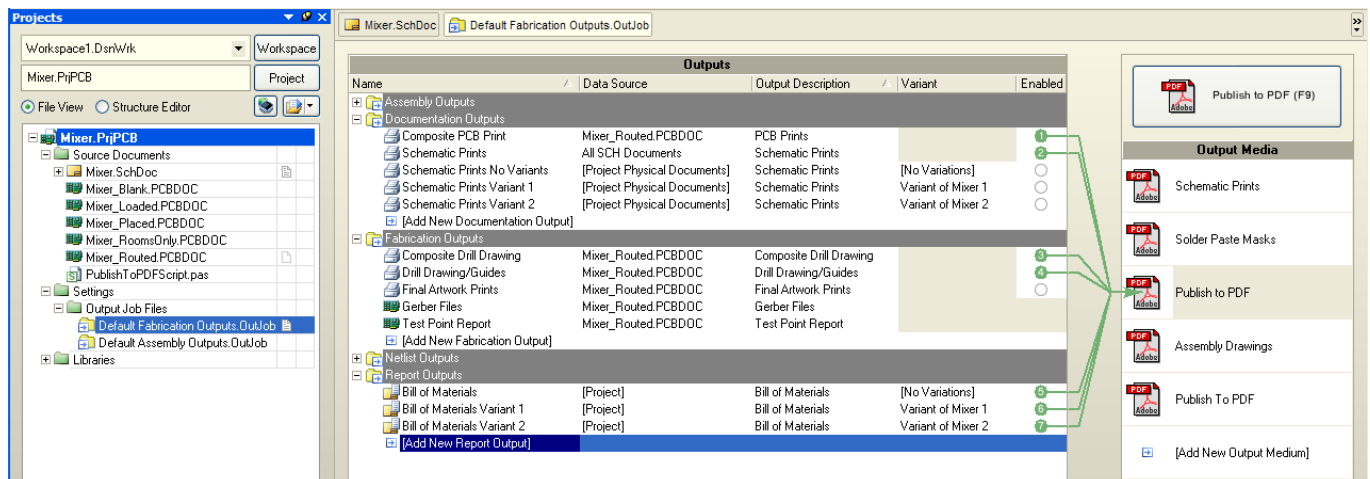


Figure 2. The OutputJob Editor with a set of Outputs linked to a Publish to PDF Media in the Output Media column.

An OutJob file consists of two parts:

- **Outputs** - Assembly Outputs, Documentation Outputs, Fabrication Outputs, Netlist Outputs and Report Outputs
- **Output Media** – choose to publish your Outputs from a variety of Output Media including PDF, direct printing and file generation.

You can create a number of Output Media with differing outputs to meet your publishing needs. You can also give your Output Media a custom name to differentiate between them. The green arrows in Figure 2 represent the linking of Outputs to an Output Medium.

## Adding New Outputs

When you create a new Output Job file, only the relevant outputs for your project are added to the file and configured using default setups. New outputs can be added to the file by either:

- Using the relevant **Edit » Add Outputs** menu items to choose the required output
- Clicking on the relevant **Add New** entry under each category and choosing the required output from the pop-up menu that appears.

## Adding Output Media

The **Output Media** column can contain a list of Publish to PDF, Print and File Generation publishing media.

Add a new Output Medium in the **Output Media** column by either:

- Clicking on the **[Add New Output Medium]** field and choosing one a publishing medium from the pop up menu that appears
- Dragging and dropping outputs from the **Outputs** section on the **Output Media** column
- Copying and pasting outputs from the **Outputs** section onto the **Output Media** column

To customize the name of your Output Medium, click once the Output Medium to focus it and click again or press **F2**, change the name to a meaningful name for your project.

## Assigning Outputs to the Output Medium

The availability of outputs is dependent on the type of publishing medium chosen from the **Output Media** column. Not all outputs can be published to PDF or generated as a file. This is indicated by the state of the **Enabled** field.

By assigning outputs to a given publishing medium, you are defining the contents of the publication. Green lines show which outputs have been selected for which output medium.

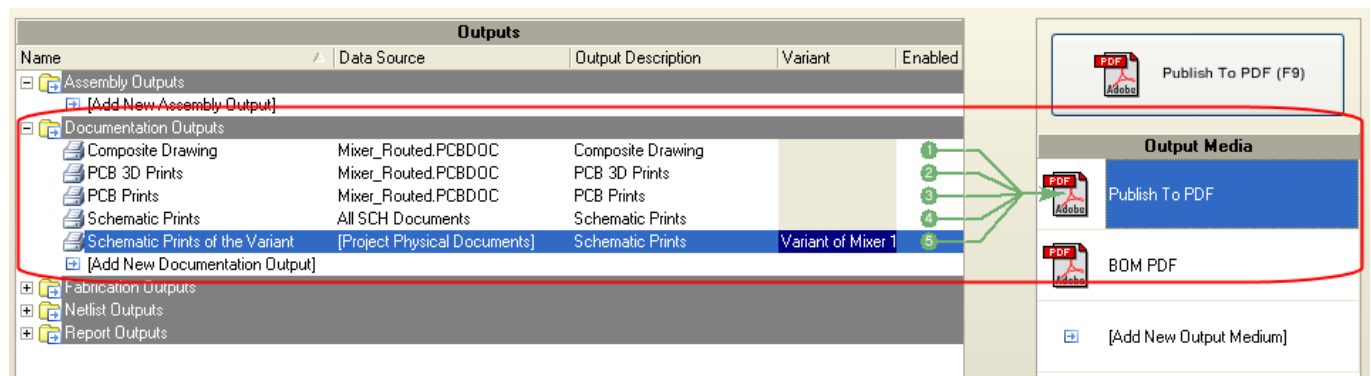



Figure 3. The Composite Drawing, PCB 3D Prints, PCB Prints and two Schematic Prints in the Documentation Outputs are assigned to a Publish to PDF Medium.

As you enable your outputs, they are numbered consecutively. If you remove one of the outputs from your Output Medium, the numbers are re-ordered accordingly. To change the order of your enabled outputs, double click on the number and select a number from the drop-down or alternatively, re-select each output in order.

## Publishing your Outputs

Publish the selected Output Medium by clicking on the button at the top of the **Output Media** column or by pressing the F9 short cut key.

 For more detailed information on adding or editing Outputs and Output Media in an OutJob File, refer to the [Output Job Editor Reference](#) document.

## Publish to PDF

---

The availability of Output Jobs is specific to the Output Medium selected. For a **Publish to PDF** Medium, the Outputs that can be linked to this PDF Medium have their **Enabled** field visible

The types of Outputs that can be assigned to a PDF Output Medium include:

### Assembly Outputs

- Assembly Drawings including the ability to customize based on Assembly Variants

### Documentation Outputs

- Schematic Prints - Project Physical Prints (including the ability to customize based on Assembly Variants)
- Schematic Documents – Logical Prints or selected prints
- PCB Prints
- PCB 3D Prints
- OpenBus Prints
- Logic Analyzer Prints
- SimView Prints
- Wave Prints
- WaveSim Prints
- Composite Drawing

### Fabrication Prints

- Drill Drawing/Guides
- Final Artwork Prints
- Power Plane Prints
- Solder/Paste Mask Prints
- Composite Drill Drawing

### Report Outputs

- Bill of Materials including the ability to customize based on Assembly Variants

Once the Outputs have been configured and assigned to a Publish to PDF medium, you can configure the *Publish to PDF settings* dialog prior publishing to a PDF.

## Configuring PDF Medium

The **PDF Setup** configures the PDF Output Medium prior to publishing.

Configure PDF settings by right clicking on the PDF Output Medium and choosing **PDF Setup** from the pop up menu that appears.

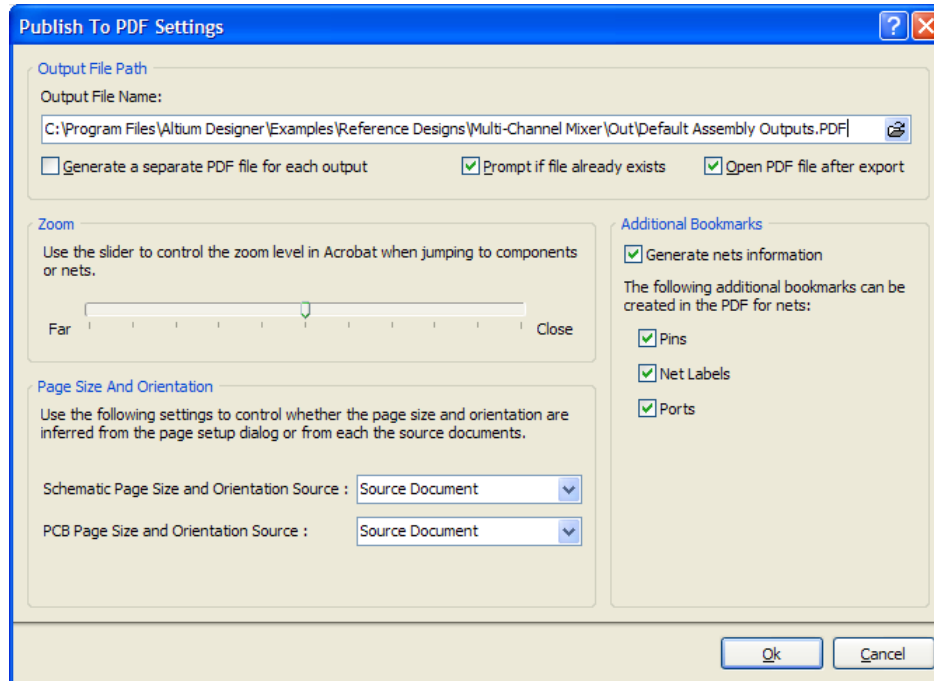


Figure 4. Publish to PDF Settings dialog

The *Publish to PDF Settings* dialog is categorized into four different groups:

### Output File Path

Use this field to specify the Output File Path and filename for your PDF. You can browse to locate your preferred Output File Path. By default, PDFs will be saved to the output path specified in the **Options** tab of your Project Options (**Project » Project Options**).

### Additional Output Settings

- **Open PDF file after export** – enable the checkbox to open your published PDF after it has been generated.
- **Prompt if file already exists** – enable the checkbox to prompt you to overwrite an existing PDF file.
- **Generate a separate PDF file for each output** – enable the checkbox to create one PDF for each of the selected outputs and also to use the **Name** of each output as the PDF Filename.

You can customize the filename and specify a folder for your PDFs, using the **Name** field in your OutJob file. Use the format `FOLDER\FILENAME`, giving you greater control over the filename, and location of your generated PDFs.

For example you can replace the `Schematic Prints` name with `Sch\Schematic Prints` and `PCB Prints` name with `PCB\PCB Prints` as shown in Figure 5. The PDFs for the Schematic Print and the PCB Prints will be generated in the Schematic and PCB folders respectively when the **Generate a separate PDF file for each output option** is enabled.

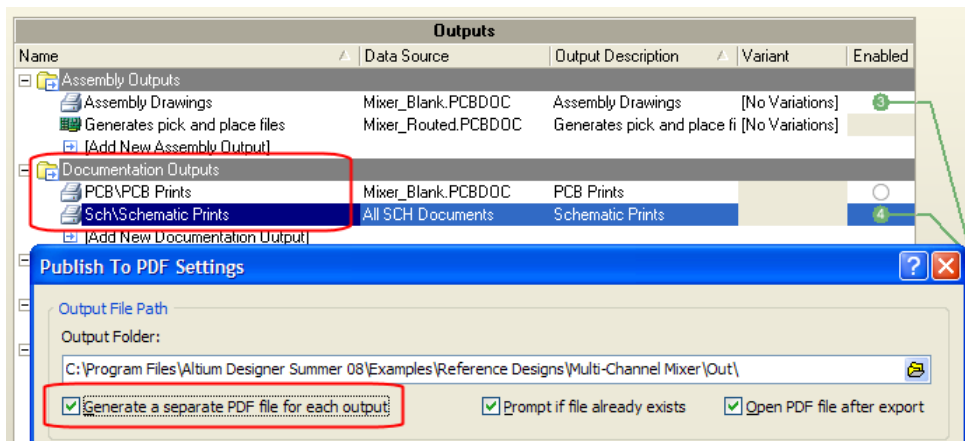


Figure 5. Generating PDFs in custom folders with the **Generate a separate PDF file for each output** option enabled in the Publish to PDF Settings dialog.

## Zoom

Use the zoom slider bar to specify the zoom level used in the PDF Viewer when browsing components, nets and ports.

## Additional Bookmarks

Use Additional Bookmarks to control whether net information is generated in the PDF or not. If the **Generate nets information** checkbox is enabled, additional bookmarks can be created for nets: **Pins**, **Net Labels** and **Ports**. Bookmarks for enabled objects (Pins, Net Labels and/or Ports) will be created in your PDF.

## Page Size and Orientation

The Page Size and Orientation settings controls whether the page size and orientation are derived from either the *Page Setup* Dialog or from the Source Document for both Schematic Pages and PCB Pages.

Page Size and Orientation can be configured for each Schematic Sheets by either:

- Using the **Design » Document Options** command from the Schematic Editor (choose **Source Document** for these settings to be implemented in your PDF)
- Using **File » Page Setup** command from the Schematic Editor (choose **Page Setup Dialog** for these settings to be implemented in your PDF)
- Right-clicking on the selected output in the OutputJob Editor and choosing **Page Setup** command from the pop-up menu that appears (choose **Page Setup Dialog** for these settings to be implemented in your PDF)

Page Size can be configured for PCB Designs by:

- Using the **Design » Board Options** command from the PCB Editor (choose **Source Document** for these settings to be implemented in your PDF)
- Creating a new PCB design from a template with a pre-defined page size (choose **Source Document** for these settings to be implemented in your PDF)

Page Size and Orientation can be configured for PCB Designs by either:

- Using **File » Page Setup** command from the PCB Editor (choose **Page Setup** for these settings to be implemented in your PDF)
- Right-clicking on the selected output in the OutputJob Editor and choosing **Page Setup** command from the pop-up menu that appears (choose **Page Setup** for these settings to be implemented in your PDF)

## Publishing to PDF

To publish your selected PDF Output Medium with the configured options, you can either:

- Click the button which has the selected Output Medium name with the PDF icon in the **Output Media** column.
- Right click and select the **Publish to PDF** option from the popup on the **Output Media** column.
- Press keyboard short cut **F9**

- Select **Publish to PDF** command (**Tools » Publish to PDF**)

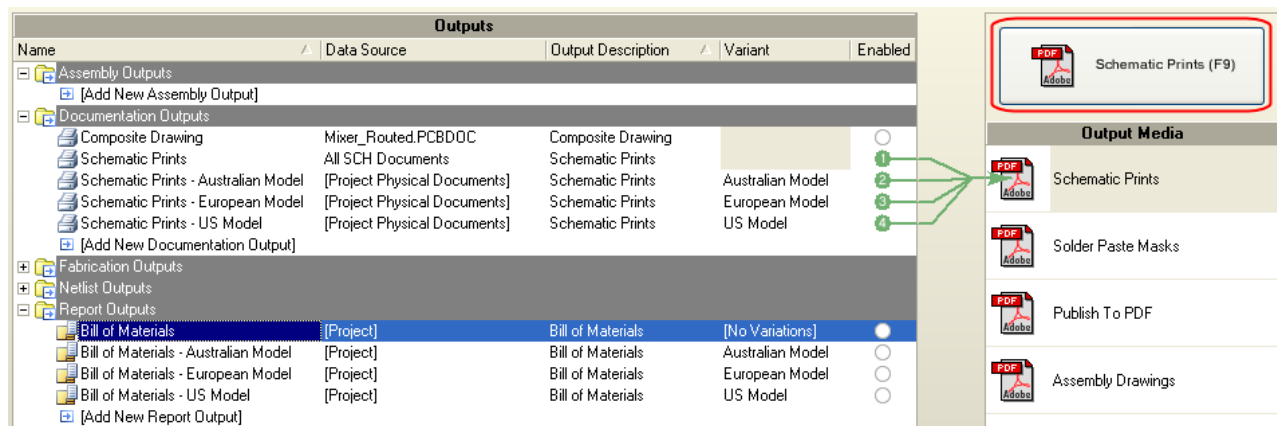


Figure 6. Output Generators assigned to the PDF to Publish medium which is ready to be published by clicking on the Output Medium (F9) button.

## Browsing the Generated PDF

To view your generated PDF, enable the **Open PDF file after export** checkbox from the *Publish to PDF Settings* dialog or navigate to your output file path and open your PDF.

The generated PDF document is a grouping of all of your selected outputs. Bookmarks are provided to enable you to navigate through your document and these are based on your **Additional Bookmarks** settings. If the source schematics are hierarchical, the hierarchy is reflected in the PDF Bookmarks.

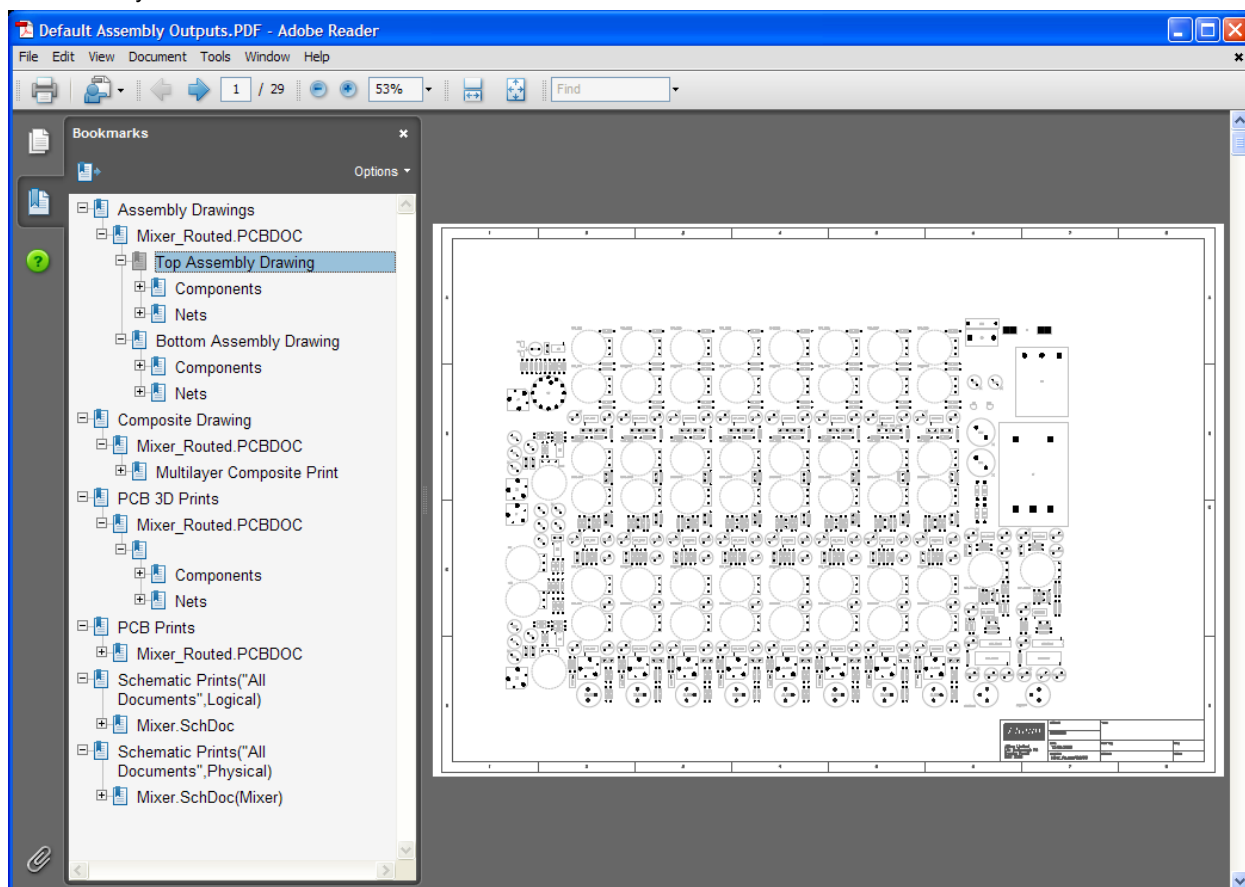



Figure 7. The result, a PDF published from a Publishing Medium with associated Outputs and its source documents in the OutJob Editor.

Clicking on a component, net, pin, port or net label bookmark will zoom to and highlight the area where the object resides. The level of zoom applied is determined by the zoom set in the *Publish to PDF Settings* dialog. Highlighting is applied when browsing by component, net, pin, port or net label bookmark for ease of reference.

## Publish to PDF using a Script

To automate the publication of Output Jobs, you can write a script to bypass the *Publish to PDF Settings* dialog and generate a PDF document quickly and easily. The `PublishToPDFScript` script as part of the Altium Designer installation is provided here to demonstrate how a PDF publishing job can be executed from an open OutJob file.

 The script project containing the `PublishToPDFScript` script is provided from the `\Examples\Scripts\Delphiscript Scripts\Processes\Process_Scripts.PrjScr` folder of the Altium Designer installation.

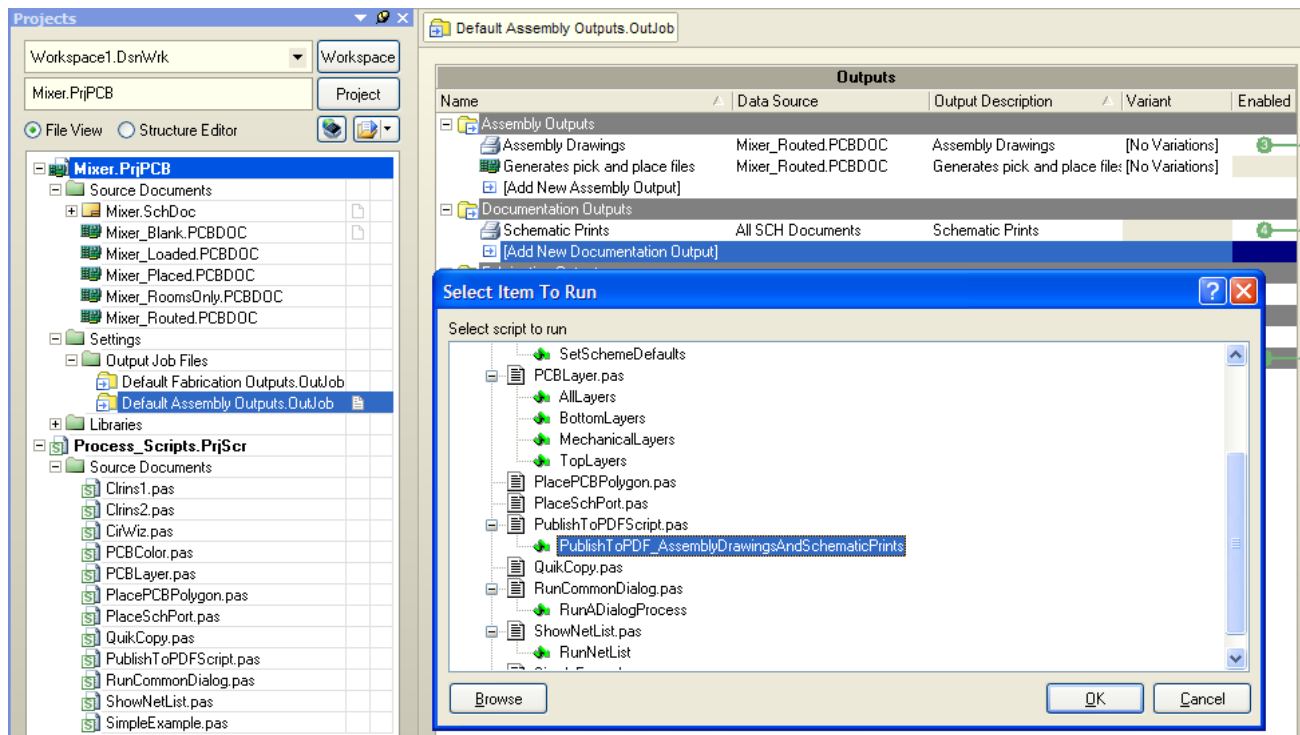
### How to Publish to PDF using the `PublishToPDFScript` script

To Publish to PDF using the supplied `PublishToPDFScript` script, you must load or create an OutJob file with a Publish to PDF medium.

Irrespective of which Output medium is set in the OutJob file, when you execute the `PublishToPDF_AssemblyDrawingsAndSchematicPrints` procedure from this script, a PDF file is generated for the existing Assembly Drawings and Schematic Prints Outputs.

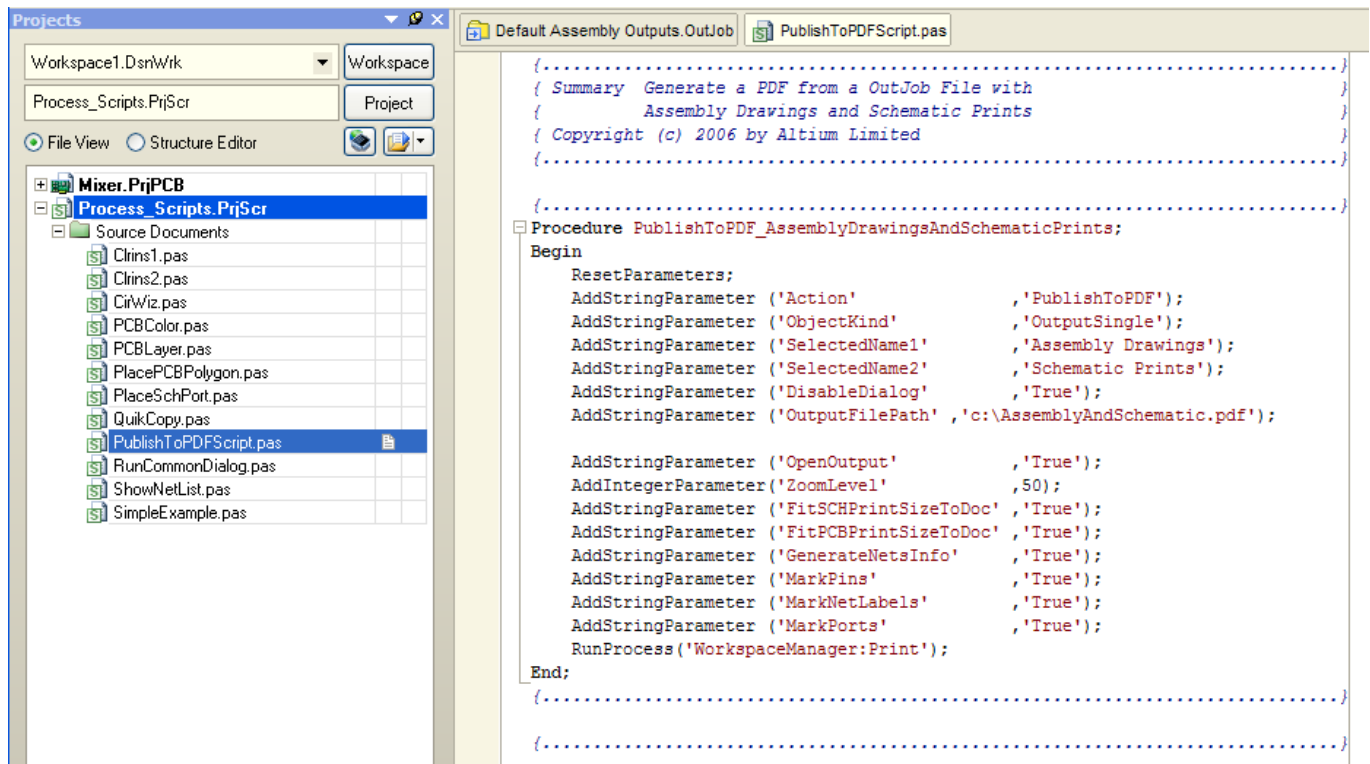
To publish a PDF from the OutJob File with existing Assembly Drawings and Schematic Prints;

- Load your design project which contains an OutJob file defining some Assembly Drawings and Schematic Prints. For example the `Default Assembly Outputs.OutJob` for the Mixer project from the `\Examples\Reference Designs\Multi-Channel Mixer` folder can be opened in Altium Designer
- Load the `Process_Scripts.PrjScr` that contains the `PublishToPDFScript` script from the `\Examples\Scripts\Delphiscript Scripts\Processes\Process_Scripts.PrjScr` folder
- Select the **DXP » Run Script** command which brings up the *Select Item To Run* dialog. The `Process_Scripts` project will be displayed in this dialog
- Select the `PublishToPDF_AssemblyDrawingsAndSchematicPrints` procedure under the `PublishToPDFScript.pas` script entry and click **OK**.



The PDF file with the Assembly Drawing and the Schematic Prints Outputs is published automatically by the script on the `Default Assembly Outputs.OutJob` file. This `AssemblyAndSchematic.PDF` file is generated in the `C:\` folder. You can have a look at the script by opening the `PublishToPDFScript.pas` file in Altium Designer.







## How to Customize the Script

The possible parameters and values for the `Workspacemanager:Print` process that are required to publish to a PDF from a script are outlined in the table below.

Parameter Name	Value	Default	Description	Needed
Action	String	PublishToPDF	If <code>Action</code> is set to <code>PublishToPDF</code> then you can use the other parameters in this table to customize the generation of your PDF document.	Yes
ObjectKind	OutputSelected / OutputBatch	OutputSelected	If <code>ObjectKind</code> set to <code>OutputSelected</code> then you need to specify the <code>SelectedNameX</code> parameters which represent the Outputs (the highlighted blue cells of Outputs on the OutJob file). If <code>ObjectKind</code> is set to <code>OutputBatch</code> then the enabled Outputs (with green arrows on the OutJob file) can be generated.	No
SelectedNameX	String		Where X is a number. <code>SelectedNameX</code> corresponds to the existing Output in the Name field of the OutJob file irrespective of the selected Output Medium in the Output Media column. The <code>ObjectKind</code> parameter needs to be set to <code>OutputSelected</code> as well. For example if you wish to generate the Assembly Drawings and Schematic Prints Outputs then you specify <code>AddStringParameter('SelectedName1', 'Assembly Drawings')</code> and <code>AddStringParameter('SelectedName2', 'Schematic Prints')</code> in a script.	No
DisableDialog	True / False	False	If the parameter is set to <code>True</code> and if any of the other parameters aren't specified, default values from the <i>Publish to PDF Settings</i> dialog are used for the generation of a PDF file. If <code>DisableDialog</code> parameter is <code>False</code> , then the <i>Publish To PDF Settings</i> dialog is invoked and once you close this dialog, the PDF is published with these values.	No
OpenOutput	True / False	*	If the parameter is set to <code>True</code> , the generated PDF is opened in a PDF reader. This parameter overrides the <b>Open PDF file after export</b> option in <i>Publish To PDF Settings</i> dialog.	No
ZoomLevel	Integer	*	The value can be set between 1 (far) and 100 (near). If the value is less than 1, it will default back to 1, if the value is greater than 100, it will default back to 100. The <code>ZoomLevel</code> parameter overrides the Zoom slider control on the <i>Publish To PDF Settings</i> dialog.	No
FitSchPrintSizeTo Doc	True / False	*	This parameter overrides the <b>Schematic Page Size and Orientation Source</b> control on the <i>Publish To PDF Settings</i> dialog. The <code>True</code> value corresponds to the <b>Source Document</b> setting and <code>False</code> corresponds to the <b>Page Setup</b> setting.	No
FitPCBPrintSizeTo Doc	True / False	*	This parameter overrides to the <b>PCB Page Size and Orientation Source</b> control on the <i>Publish To PDF</i>	No

Parameter Name	Value	Default	Description	Needed
			<i>Settings</i> dialog. The <code>True</code> value corresponds to the <b>Source Document</b> setting and <code>False</code> corresponds to the <b>Page Setup</b> setting.	
GenerateNetsInfo	True / False	*	This parameter overrides the <b>Generate nets information</b> control on the <i>Publish To PDF Settings</i> dialog. If the <b>GenerateNetsInfo</b> is set to <code>True</code> then additional <b>Pins</b> , <b>Net Labels</b> and <b>Ports</b> bookmarks can be created.	No
MarkPins	True / False	*	This parameter overrides the <b>Pins</b> bookmark control on the <i>Publish To PDF Settings</i> dialog. Defaults to <code>True</code> . The <b>GenerateNetsInfo</b> option has to be set <code>True</code> for this setting to work.	No
MarkNetLabels	True / False	*	This parameter overrides the <b>Net Labels</b> bookmark control on the <i>Publish To PDF Settings</i> dialog. The <b>GenerateNetsInfo</b> option has to be set <code>True</code> for this setting to work.	No
MarkPorts	True / False	*	This parameter overrides to the <b>Ports</b> bookmark control on the <i>Publish To PDF Settings</i> dialog. The <b>GenerateNetsInfo</b> option has to be set <code>True</code> for this setting to work.	No
OutputMedium	String		This parameter targets the existing PDF Output Medium for the current OutJob file in Altium Designer. If the <code>OutputMedium</code> parameter is not specified, the first PDF Output Medium in the <b>Output Media</b> list is used (the green arrows on the OutJob file will be updated to reflect this). If there are no PDF Output Media then nothing happens.	No
PublishMethod	0 / 1	*	This parameter determines whether the Outputs will be published as a Single File ( <code>PublishMethod = 0</code> ) or as Multiple Files ( <code>PublishMethod = 1</code> ). This parameter overrides the Generate a separate PDF file for each output control on the <i>Publish To PDF Settings</i> dialog. You will also need to specify the <code>OutputFilepath</code> parameter.	No
OutputFilepath	String	The path of the OutJob file with a PDF extension	If the <code>PublishMethod = 0</code> then the <code>OutputFilepath</code> should be a full path and file name.  If the <code>PublishMethod = 1</code> then the <code>OutputFilePath</code> should be a folder name.  If the <code>OutputFilepath</code> is not specified, then the path of the OutJob file but with a PDF extension is used for instance.	No

## Notes:

1. The asterisks, \*, in the **Default** column of the table correspond to the default settings of the *Publish to PDF settings* dialog.
2. The default values of the *Publish to PDF Settings* dialog are extracted from each Output Medium in the OutJob File. Every time you configure the Output Medium and save the OutJob file, the new settings for this Output Medium are saved in this OutJob file.

## Sample Script – Publish to a PDF using the specific PDF Output Medium

A sample script has been provided below to publish all enabled Outputs to a PDF. You will need an OutJob file that has a PDF Output Medium with the DocoPDF name before running the script on it.

To define the contents of the published PDF file, the default settings are used from the DocoPDF Output Medium. You can either use the *Publish to PDF Settings* dialog to change the settings or use the available parameters from the table above in the script to override the settings for this *Publish to PDF Settings* dialog.

**Procedure** PublishToPDF\_DocoPDF\_Medium;

**Begin**

```
ResetParameters;
AddStringParameter ('Action'           , 'PublishToPDF');

// Select an existing PDF Output Medium. If not found, first PDF Medium will be used.
AddStringParameter ('OutputMedium'     , 'DocoPDF');

// Publish all the selected Output Jobs
AddStringParameter ('ObjectKind'       , 'OutputBatch');

// Don't display the Publish to PDF Settings dialog
AddStringParameter ('DisableDialog'    , 'True');

//Publish selected Output Jobs in a single PDF file
AddIntegerParameter('PublishMethod'    , 0);

// Override the Output File path setting in the Publish to PDF Settings dialog
AddStringParameter ('OutputFilePath'   , 'C:\Doco.pdf');

RunProcess('WorkspaceManager:Print');
```

**End;**

This script contains the WorkspaceManager:Print process with various AddStringParameter and AddIntegerParameter calls to generate a DocoPDF.PDF file in the C:\ folder.

You can add this script in the PublishToPDFScript.pas file from the Process\_Scripts.PrjScr project and run the PublishToPDF\_DocoPDF\_Medium procedure from the *Select Item To Run* dialog in the OutJob Editor.



For an introduction to Scripting, refer to the [Getting Started with Scripting](#) document.



For more detailed information on building script projects, refer to the [Building Script Projects](#) document.



You can refer to the [Customizing the Altium Designer Resources](#) tutorial for details on how to create new shortcuts or update resources in Altium Designer.



To learn more about the benefits of scripts, study the script examples in the /Examples/Scripts folder of your installation.

## Revision History

Date	Version No.	Revision
19-Oct-2007	1.0	Altium Designer 6.8 Release
25-Jan-2008	1.1	Added information for new feature, Batch Publish to PDF
21-Apr-2008	1.2	Updated Page Size to A4
27-May-2008	1.3	Added information for new features, Output Media column and multiple PDF generation.
24-Aug-2011	-	Updated template.

Software, hardware, documentation and related materials:

Copyright © 2011 Altium Limited.

All rights reserved. You are permitted to print this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document is made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment.

Altium, Altium Designer, Board Insight, DXP, Innovation Station, LiveDesign, NanoBoard, NanoTalk, OpenBus, P-CAD, SimCode, Situs, TASKING, and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.