Altium Designer Guide

PURPOSE

The purpose of this document is to introduce the reader to Altium Designer, show how to create and document a new printed circuit board design, and output design files.
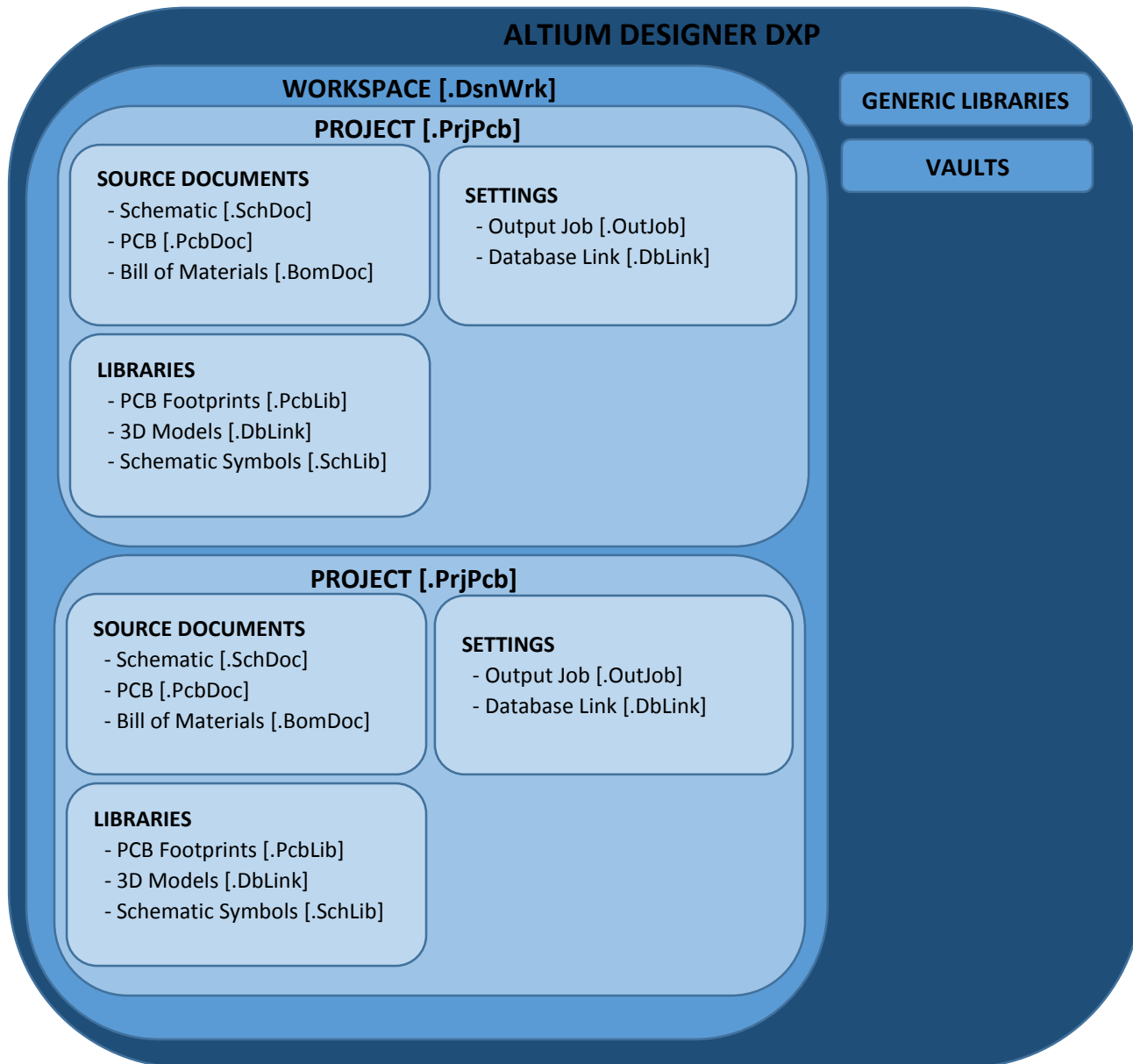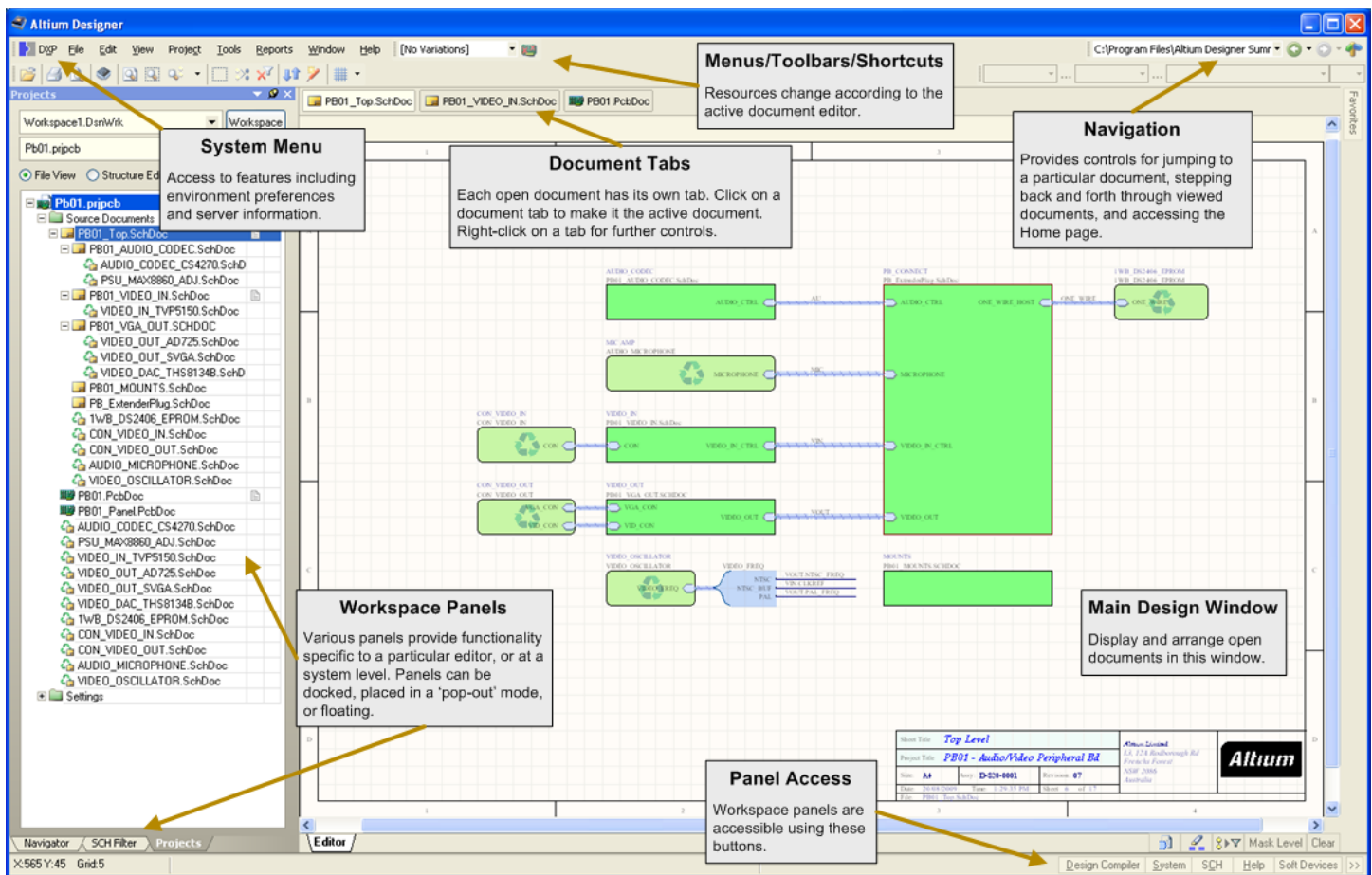
CONTENTS

ALTIUM DESIGNER ENVIRONMENT

Altium Designer is a unified electronics design tool used primarily for schematic capture and PCB layout. The tool is structured as illustrated below. At the top level is the Design Explorer (DXP). This houses the user interface along with all the tools and files used in schematic capture and PCB design. A Workspace is a container for multiple projects and preference settings. At the lower level, projects contain all the actual design documents; schematic sheets, PCB layout files, custom libraries, and output settings.

Resource: http://techdocs.altium.com/display/ADOH/The+Altium+Designer+Environment

ALTIUM DESIGNER DXP

WORKSPACE [.DsnWrk]

PROJECT [.PrjPcb]

SOURCE DOCUMENTS
- Schematic [.SchDoc]
- PCB [.PcbDoc]
- Bill of Materials [.BomDoc]

SETTINGS
- Output Job [.OutJob]
- Database Link [.DbLink]

LIBRARIES
- PCB Footprints [.PcbLib]
- 3D Models [.DbLink]
- Schematic Symbols [.SchLib]

PROJECT [.PrjPcb]

SOURCE DOCUMENTS
- Schematic [.SchDoc]
- PCB [.PcbDoc]
- Bill of Materials [.BomDoc]

SETTINGS
- Output Job [.OutJob]
- Database Link [.DbLink]

LIBRARIES
- PCB Footprints [.PcbLib]
- 3D Models [.DbLink]
- Schematic Symbols [.SchLib]

GENERIC LIBRARIES

VAULTS

CREATE A NEW PCB PROJECT

The following instructions step through the process of creating a new PCB Project and adding files.

1. Select **File > New > Project > PCB Project** from the top menu.
   A new project will appear in the *Projects Panel*, including a PcbDoc, SchDoc, and OutJob file.
2. Save and rename the new project files. Select **File > Save Project As**.
   Navigate to the location where you would like to store the project documents (Z drive recommended) and enter a name in the **File Name** field then click **Save** for each document.
   Notes: Altium Designer does not automatically manage project file structures. You have the liberty to choose where files are stored. Project files stored in the same folder as the project file or in a child/grandchild folder are linked to the project using relative referencing, whereas files stored in a different location are linked using absolute referencing.
3. As you draw your schematic, you may need to add additional schematic sheets. To do this right click on the project name in the *Projects Panel*. Select **Add New To Project > Schematic**.

SCHEMATIC CAPTURE

1. Change sheet size if the default size B [11x17] is not desired. Select **Design > Document Options.** Under the **Template** tab, choose the template sheet size you would like to use from the **Template file** drop down menu. *Note, only sheet sizes A and B are available as these are common and print nicely to standard printer paper, 8.5x11 and 11x17 respectively.*

2. Populate the Title Block. First populate the Global Project Parameters by clicking **Project > Project Options…** Under the Parameters tab enter in values for **DraftsmanName**, **EngineerName**, **GlobalDate**, **GlobalTitle**, and **RevNumber**. Click **OK**. These parameters will automatically map to the respective place holders in the schematic title block.

   Next, populate the Sheet Title parameters. Click **Design > Document Options…** Under the parameters tab, insert a value for the **Title** parameter.

   Lastly, populate the sheet numbering. Select **Tools > Number Schematic Sheets**. Click **Auto Sheet Number,** then click **Update Sheet Count**. Click **OK**. You should see the sheet number and sheet total values update.
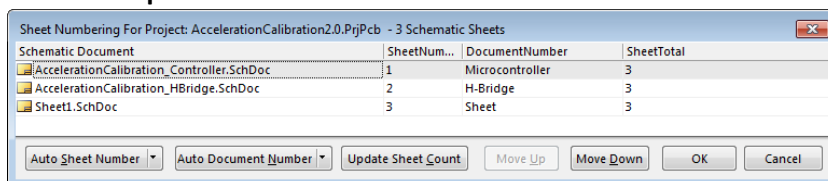
3. Add components to the schematic. Components can be found in two locations; *Vaults* and *Libraries*.

   Altium Content Vault

   I recommend using the Altium Content Vault as your first source for components. This vault contains high quality components that include validated symbols, footprints, 3D models, and manufacturer data. New content is added regularly by Altium. To access the vault, select the **Vaults** tab on the right toolbar and browse through the Vault Folders or use the search box at the top right to find components. Right click on the desired component and click place. Move the cursor to your schematic sheet and single click to place the component.
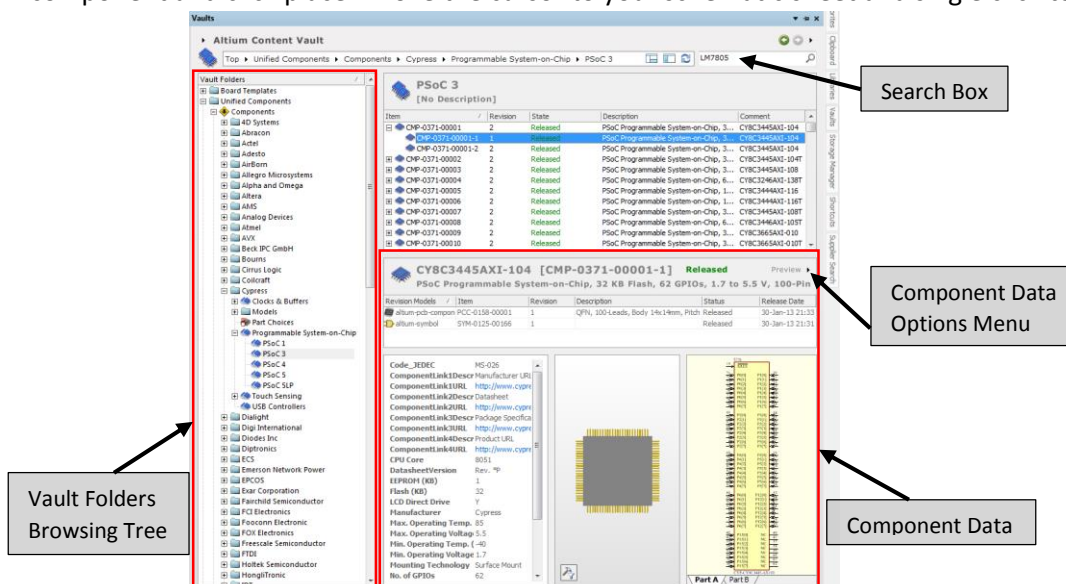

   Libraries

   The libraries included with Altium are a good alternative source if you are unable to find a component in the Vault. Select the **Libraries** tab on the right toolbar. Choose from the available libraries in the drop down menu. Browse to or search for the desired component. Right click on the component and select **Place**.

4. Add supplier data to components. In the Supplier Search tool on the right, enter the complete manufacturer part number and click **Search**. A list of suppliers that offer this component will appear. Find a supplier that has stock of the component and offers the best price in the quantities needed. Right click that row and select **Add Supplier Link and Parameters to Part**. Click on the schematic symbol to add parameters to component.
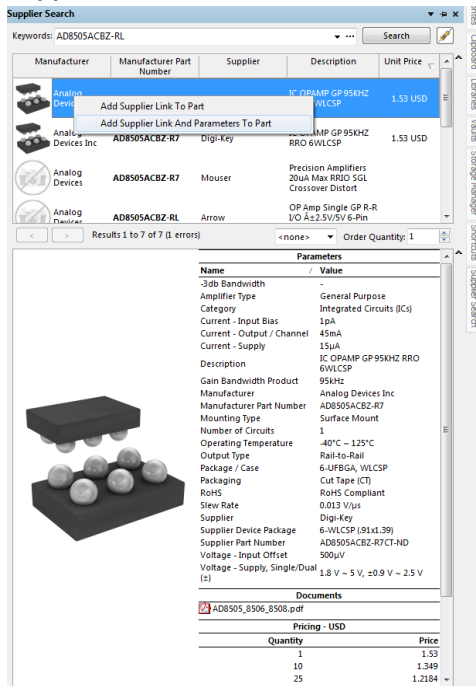
5. Configure parameter visibility. Double click on the component to open the properties window. For specialty components such as ICs, uncheck the **Supplier Part Number 1** and **Comment** parameters, then check **Manufacturer Part Number** parameter. For generic components like resistors and capacitors, uncheck **Supplier Part Number 1** and check **Value**. Here are two examples of how parameters should appear:
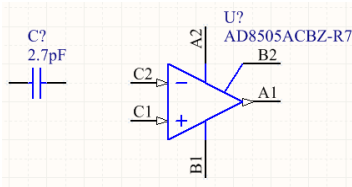
6. Wire components together. Click the **Place Wire** button in the main tool bar or type '**w**'. Move the crosshairs to a component lead, single click to start a wire and single click on the destination lead to terminate a wire.

7. Add meaningful Net Labels to vital nets. Click the **Net Label** button on the main tool bar or type '**p**', '**n**'. Hit **Tab** to open the properties window and enter a net name. Click **OK**. Move the crosshairs to the net and single click to attach the label.

8. For nets that span multiple sheets, add Port Flags in addition to Net Labels. Click the **Place Port** button or type '**p**', '**r**'. Hit **Tab** to open the properties window and enter the net name.

9. For Power and Ground nets use the **VCC Power Port** and **GND Power Port** tools. Select the desired power port and select **Tab** to open the properties window. Enter a meaningful power net name and click **OK**. Here is an example of proper net labeling with ports.
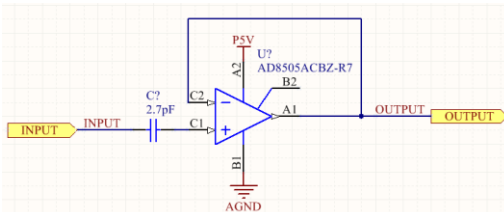
10. Annotate schematic. Select **Tools > Annotate Schematics…** Click **Update Changes List**, click **OK** to acknowledge the Change(s) made prompt, then click **Accept Changes (Create ECO)**. In the Engineering Change Order window click **Validate Changes**. If no errors exist, click **Execute Changes**. Click **Close** to exit. Your schematic symbols should now have numbered reference designators.

PCB LAYOUT

The following steps assume you are using the WWU EE template project that automatically includes a .PcbDoc file. If this is not the case, you must first add a .PcbDoc file to your project by selecting **Project > Add New to Project > PCB.**

1. Forward schematic information to PCB Layout. Select **Design > Updated PCB Document** *{document name}***.PcbDoc**.
2. In the Engineering Change Order window uncheck the **Add Rooms** checkbox, then click **Validate Changes.** If no errors exist, click **Execute Changes**. Click **Close** to exit.
3. Configure Design Rules. Select **Design > Rules…** Right click on **Design Rules** in the rules tree. Select **Import Rules…** Highlight all the rules by selecting the top rule, holding **Shift** and selecting the bottom rule. Click **OK**. Browse to R:\Altium\Templates\Design Rules and open **2LayerOSHPakrRules.RUL**. Click **Yes** when asked to clear existing rules prior to import. Click **OK**.
4. Arrange components by dragging them to the desired location. Hitting the space bar while dragging a component will rotate it 90 degrees counter-clockwise.
5. Route circuit connections. Select the **Interactively Route Connections** tool from the main tool bar or click '**w**'. Single click on a pad to start a route. Single click to create a corner and double click on the destination pad to end a route. Press **Escape** to release the tool.
   To modify the trace width, select **Tab** with a trace started. This will open the properties window for that trace. Modify the **Width** parameter.
6. Adjust board shape. Select **View > Board Planning Mode** or type '**1**'. Select **Design > Move Board Vertices** or **Design > Redefine Board Shape**. Adjust the board shape as desired. Type '2' to return to the standard PCB view.
7. Reset Origin to bottom left corner of board. Select **Edit > Origin > Set**. Click on the bottom left corner of the board shape.
8. Draw board outline. Select the **Place Polygon Plane** tool from the main tool bar. In the properties window select **None (Outlines Only)** for the Fill Mode. Under Properties enter "Board Outline" for **Name** and set **Layer** to Mechanical 1. Click **OK**. Draw the board outline along of the edge of the board shape.
9. Check for design rule violations. Select **Tools > Design Rule Check…** Uncheck **Create Report File** under DRC Report Options and click **Run Design Rule Check…** Any existing errors will appear in the Messages panel. Double click on the error and the PCB editor will zoom in to the location of the error. Correct all errors and run the Design Rule Check again. Repeat until all errors are gone.

   Note: A common error occurs when hidden silkscreen text is outside the bounds of the board shape. The error will be reported as "Silk To Board Region Clearance Violation". If this occurs, look for components near the edge of the board. Double click suspect components and uncheck the Hide box under Comment. Move the text so it is within the bounds of the board shape and re-hide the text.

OUTPUT DESIGN FILES

Altium Designer consolidates all design file output configuration into a single Output Job File (*.OutJob). This file can be configured to output documentation such as schematics, BOMs, 3D views and models, as well as fabrication files like Gerber, pick and place, and NC Drill files. The default WWU EE PCB Project Template automatically includes a pre-configured Output Job File. The following instructions step through how to output the various design files.

Fabrication Files

This output container includes Gerber, NC Drill, Pick and Place, and Assembly BOM files that are all commonly required by PCB manufacturers and assemblers.

1. Double click the *.OutJob file in the left file browser to open.
2. Highlight **Fabrication** under Output Containers.
3. For all enabled Outputs indicated by a green dot and line/arrow, ensure the Data Source is set to **[PCB Document]** for Assembly and Fabrication Outputs, and **[Project]** for Report Outputs.
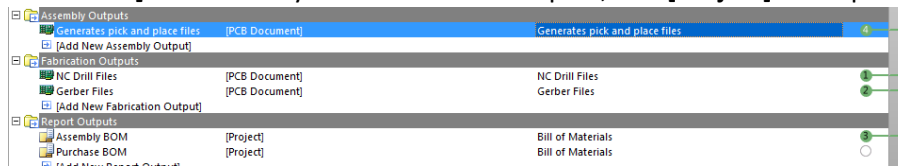
4. If you wish to assign different file names from the standard, right click on **Fabrication** output container and select **Properties...** Click **[Output Name]** in the Fabrication Settings window, select the **Use custom file output names** radial button, and enter a new name. Click **OK**.
5. Click **Generate Content** in the Output Containers window pane. Altium will automatically generate the fabrication files and place them inside the project directory in the following structure.

Folder	Sub-Folder	File	Description
Fabrication	NC Drill	{GlobalProjectName} – Rev. {RevNumber}.DRR	Drill report
		{GlobalProjectName} – Rev. {RevNumber}.LDP	ASCII format drill pair report
		{GlobalProjectName} – Rev. {RevNumber}.TXT	ASCII format drill file
	Gerber	{GlobalProjectName} – Rev. {RevNumber}.REP	Gerber generation report file.
		{GlobalProjectName} – Rev. {RevNumber}.EXTREP	Extension report file
		{GlobalProjectName} – Rev. {RevNumber}.apr	Aperture file
		{GlobalProjectName} – Rev. {RevNumber}.APR_LIB	
		{GlobalProjectName} – Rev. {RevNumber}.GBL	Bottom Copper Layer
		{GlobalProjectName} – Rev. {RevNumber}.GBO	Bottom Silkscreen
		{GlobalProjectName} – Rev. {RevNumber}.GBS	Bottom Soldermask
		{GlobalProjectName} – Rev. {RevNumber}.GM1	Board Outline
		{GlobalProjectName} – Rev. {RevNumber}.GTL	Top Copper Layer
		{GlobalProjectName} – Rev. {RevNumber}.GTO	Top Silkscreen
		{GlobalProjectName} – Rev. {RevNumber}.GTS	Top Soldermask
	BOM	{GlobalProjectName} – Rev. {RevNumber}.xlsx	Assembly Bill of Materials
	Pick Place	{GlobalProjectName} – Rev. {RevNumber}.txt	Pick & Place

Documentation

This output container includes schematics, assembly drawings, 3D views, dimensional drawings, and validation reports.

1. Open the *.OutJob file.
2. Highlight **Documentation** under Output Containers.
3. Ensure **[Project Physical Documents]** is selected as the data source for Schematic Prints and **[PCB Document]** is selected for the remaining enabled Outputs.
4. Click **Generate Content** in the Output Container window pane. Altium will automatically generate and open a PDF. The PDF is saved in a Documentation folder within the project directory.

Purchase BOM (Bill of Materials)

This output container includes the Purchase Bill of Materials.

1. Open the *.OutJob file.
2. Highlight BOM under Output Containers.
3. Ensure [Project] is selected as the Data Source for Purchase BOM.
4. Click Generate Content in the Output Container window pane. Altium will automatically generate an excel spreadsheet and save it to the BOM folder within the project directory.