Output Documentation

Now that you've completed the design and layout of the PCB, you're ready to produce the output documentation needed to get the board reviewed, fabricated and assembled.

Available Output Types

Because a variety of technologies and methods exist in PCB manufacture, CircuitStudio has the ability to produce numerous output types for different purposes:

Assembly Outputs

- Assembly Drawings - component positions and orientations for each side of the board.
- Pick and Place Files - used by robotic component placement machinery to place components onto the board.

Documentation Outputs

- Composite Drawings - the finished board assembly, including components and tracks.
- PCB 3D Prints - views of the board from a three-dimensional view perspective.
- Schematic Prints - schematic drawings used in the design.

Fabrication Outputs

- Final Artwork Prints - combines various fabrication outputs together as a single printable output.
- Gerber Files - creates manufacturing information in Gerber format.
- NC Drill Files - creates manufacturing information for use by numerically controlled drilling machines.
- ODB++ - creates manufacturing information in ODB++ database format.
- Power-Plane Prints - creates internal and split plane drawings.
- Solder/Paste Mask Prints - creates solder mask and paste mask drawings.
- IPC-2581 Standard - creates an XML-based single file format that incorporates a rich range of board fabrication data – from layer stackup details though to full pad/routing /component information and the Bill Of Materials (BOM).

Report Outputs

- Bill of Materials - creates a list of parts and quantities (BOM), in various formats, required to manufacture the board.
- Report Single Pin Nets- creates a report listing any nets that only have one

connection.

- Electrical Rules Check - formatted report of the results of running an Electrical Rules Check.

Individual Outputs or Managed Output Generation

CircuitStudio has 2 separate mechanisms for configuring and generating output:

1. **Individually** - the settings for each output type are stored in the Project file, you selectively generate that output when required, via the buttons on the **Outputs** tab. These outputs are written to the folder specified in the **Output Path** setting in the **Options** tab of the *Options for PCB Project* dialog.
2. **Managed Release** - all output settings are stored in a special file in the project folder, you then generate all enabled outputs in a single action via the *Generate Output Files* dialog. Using this approach gives you confidence that all the correct outputs were generated from the same version of the source schematic and PCB files. The dialog is accessed either from the **Project | Project Actions | Generate Outputs** button, or the **Home | Project | Project » Generate Outputs** menu entry. These outputs are written to a folder named \Default Configuration. Once you have configured and enabled each required Outputer, click the **Generate** button to generate the outputs in the \Default Configuration folder.

For this project, we will use a Managed Release

Preparing for a managed release:

1. The *Generate Output Files* dialog can be opened at any time, with a schematic open, a PCB open, or no document open - click the Generate Outputs button on the **Project** tab to open the dialog. Note that when you open the dialog the special file that holds the output settings is automatically created in the project folder, so the project file will now be marked as modified.
2. Note the list of Outputers included, for the tutorial you will be using the **Gerber Files** outputer, and the **Bill of Materials** outputer.
3. Click Configure... for the **Gerber Files** to open the *Gerber Setup* dialog. This will be configured in the next set of steps.

Configuring Gerber generation:

1. In the *Generate Output Files* dialog, click Configure... for the **Gerber Files** output, the *Gerber Setup* dialog will open, as shown in the image above.
2. Since the board has been design in Metric, set the **Units** to **Millimeters**. in the **General** tab of the dialog.
3. The smallest unit used on the board is 0.25mm for the routing and clearance, but because most of the components have their reference point at their

geometric center (and were placed on a 1mm grid), some of their pads will actually be on a 0.01 grid. The default **Format** of **4:4** on the **General** tab will more than ensure that the resolution of the output data is adequate to cover these grid locations. **Note:** the NC drill file *must always* be configured to use the same **Units** and **Format**.

4. Switch to the **Layers** tab, then click the **Plot Layers** button and select **Used On**. Note that mechanical layers may be enabled, these are not normally Gerbered on their own. Instead they are often included if they hold detail that is required on other layers, for example an alignment location marker that is required on every Gerber file. In this case the **Mechanical Layer** options on the right side of the dialog are used to include that detail with another layer. Disable any mechanical layers that were enabled in the **Layers to Plot** section of the dialog.

5. Click on the Advanced tab of the dialog. Confirm that the **Position on Film** option is set to **Reference to relative origin**. **Note:** the NC drill file *must always* be configured to use the same **Position on Film** option.

6. Click **OK** to accept the other default settings and close the *Gerber Setup* dialog.

7. Now the Gerber settings are configured, the next step is to configure other outputs. For this tutorial you will also configure the BoM in the next set of steps.

Configuring the Bill of Materials

CircuitStudio includes a highly configurable BoM generation feature which can generate output in a variety of formats, including: text, CSV, PDF, HTML and Excel. Excel-format BoM's can also have a template applied using one of the pre-defined templates, or one of your own.

1. BoM output is configured in the *Bill of Materials For Project* dialog. Since we intend to use the managed release approach, open the *Bill of Materials For Project* dialog from the *Generate Output Files* dialog.

2. Down the left of the dialog there is a list of every component attribute, for all components in the design. Enable the checkbox for each attribute you would like to include in the BoM, clear the checkbox for an attribute you wish to remove.

3. The default settings for the BoM is to cluster by like components. Clustering is achieved by adding component attributes to the **Grouped Columns** region of the dialog. Click and drag these attributes out of the **Grouped Columns** and drop them back in the **All Columns** region if you prefer every component to be on its own row in the BoM. For this project, we will have one line item per component.

4. The main grid region of the dialog is the content that is written into the BoM. In this region you can: click and drag to reorder the columns; click on a column heading to sort by that column; ctrl+click to sub-sort by that column; define value-based filters for a column using the small dropdown in each column header; right-click to Force the columns to fit the current dialog width.

5. The BoM generator sources its information from the schematic, enable the

Include Parameters from PCB option to access PCB information, such as location and side of board (note that this feature can also be used to configure and generate a configurable pick and place file, if required).

6. Set the BoM Template field to BOM Default Template.XLT - When using a Template the order and location of the information presented is determined by the Template instead of by the configuration in this dialog, so when using a template it is not necessary to change column order. You do, however, need to make sure all values referenced in your template are available in the report you generate or these fields in the template will receive no information.

Generate the Outputs

Now that our Gerbers and BoM have been configured, make sure these two outputs are enabled on the Generate Output Files dialog, and press the Generate button. The status column will let you know the current status of the generation, and will show as Passed when it completes.

You can now locate your output files in your project directory.

This concludes this series of training videos for CircuitStudio, I hope you have found them useful.