Generates Gerber Files and Drill Tables for Selected Layouts
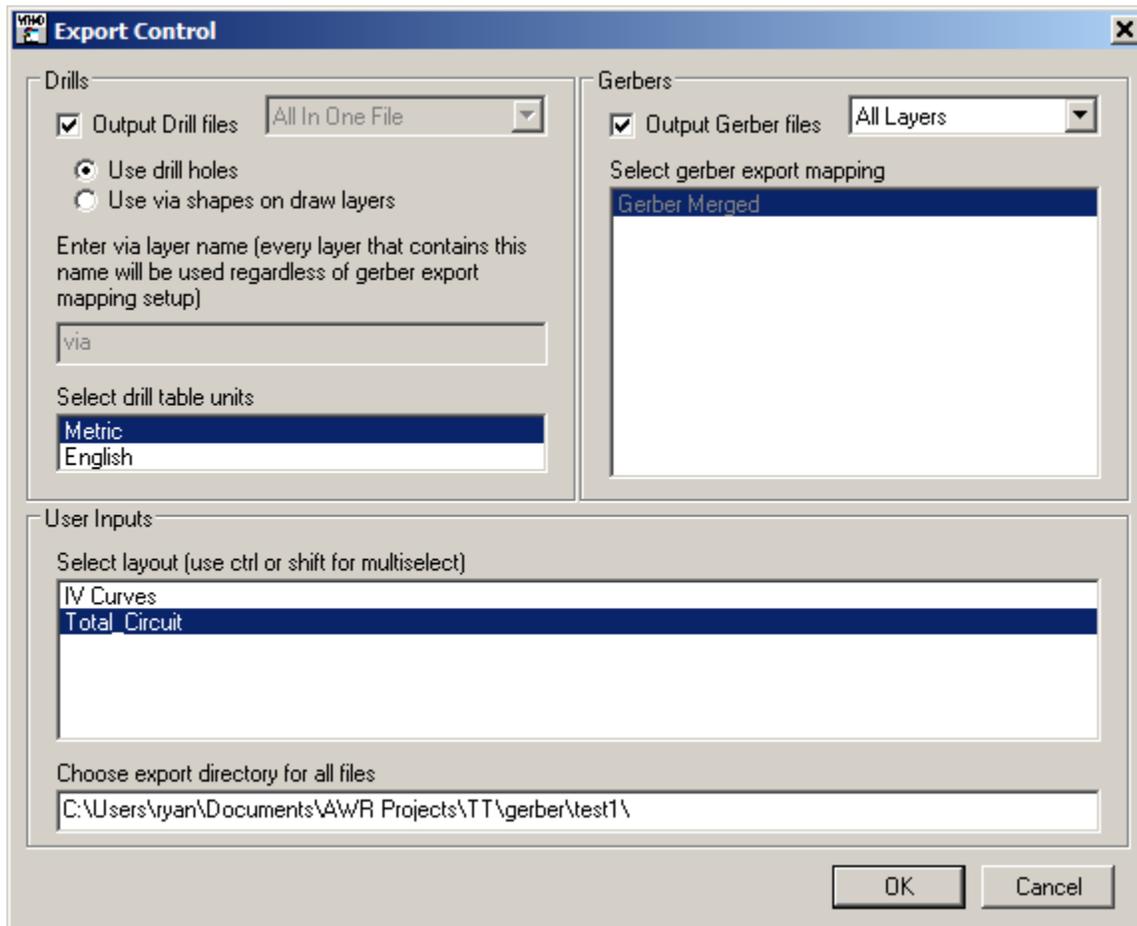
To run this script

Select **Scripts > Layout > Export_PCB_Drill_Gerber** from the Menus.

Or, in versions that support the script, you can run the utility directly from this page using this button.

Description

This utility provides a single dialog to export the manufacturing files for every layer of every PCB style design in the project, rather than exporting them one at a time for each layer of each layout. It has settings for how the drill and Gerber files should be exported for the current project. When the script is run, the following dialog will appear.

The dialog has the following groups of controls:

- Drills: Controls how drill files are exported. If you used drill hole objects in your layout, you should use the **Use drill holes** selection. Drill holes are limiting as they have no concept of layers. Otherwise, if your vias are drawn as regular shapes (circles) in your layout, you should use the **Use via shapes on draw layers** option. If using this mode, you can then decide if you want all the drills in one file or one file per unique layer found. This will depend on your manufacturing process. Additionally, you provide part of a layer name that indicates which layers have vias. Typically the word "via" is in the layer name. Finally you can select if you want the drill tables in Metric or English units.

- Gerbers: Gerber files do not have a concept of layers, so this script handles exporting one Gerber file per layer. You must have a Gerber export mapping table defined in your LPF. Please see the AWRDE documentation for setting up these tables. Your first selection is which Gerber export mapping table to use. Then in the pull down menu, you can select All Layers or User Selected. All Layers will export one Gerber file for every layer in the export map regardless if the Write Layer option is selected. The User Selected option will only export data on the layers have the Write Layer option selected. **Note** : The AWRDE has a concept of positive, negative and normal layers that are configured with + and - symbols on the end of these layers. This script detects this situation and exports those layers as a group to the base layer name (the layer name without the + or - symbol).

- User Inputs: Select which layout you want exported. Hold down the **Ctrl** key to select multiple layouts. Note the exported files have the file name of the schematic and then the layer name at the end. Finally, you can type in (or copy and paste from Windows Explorer) the path where the files should be written. The field will be originally filled in with the path for the current project.