

OurPCB Gerber Generation - Altium

Summary

Each Gerber file corresponds to one layer in the physical board – the component overlay, top signal layer, bottom signal layer, the solder masking layers and so on. The *Gerber Setup* dialog provides you with tools to completely configure your Gerber file output options.

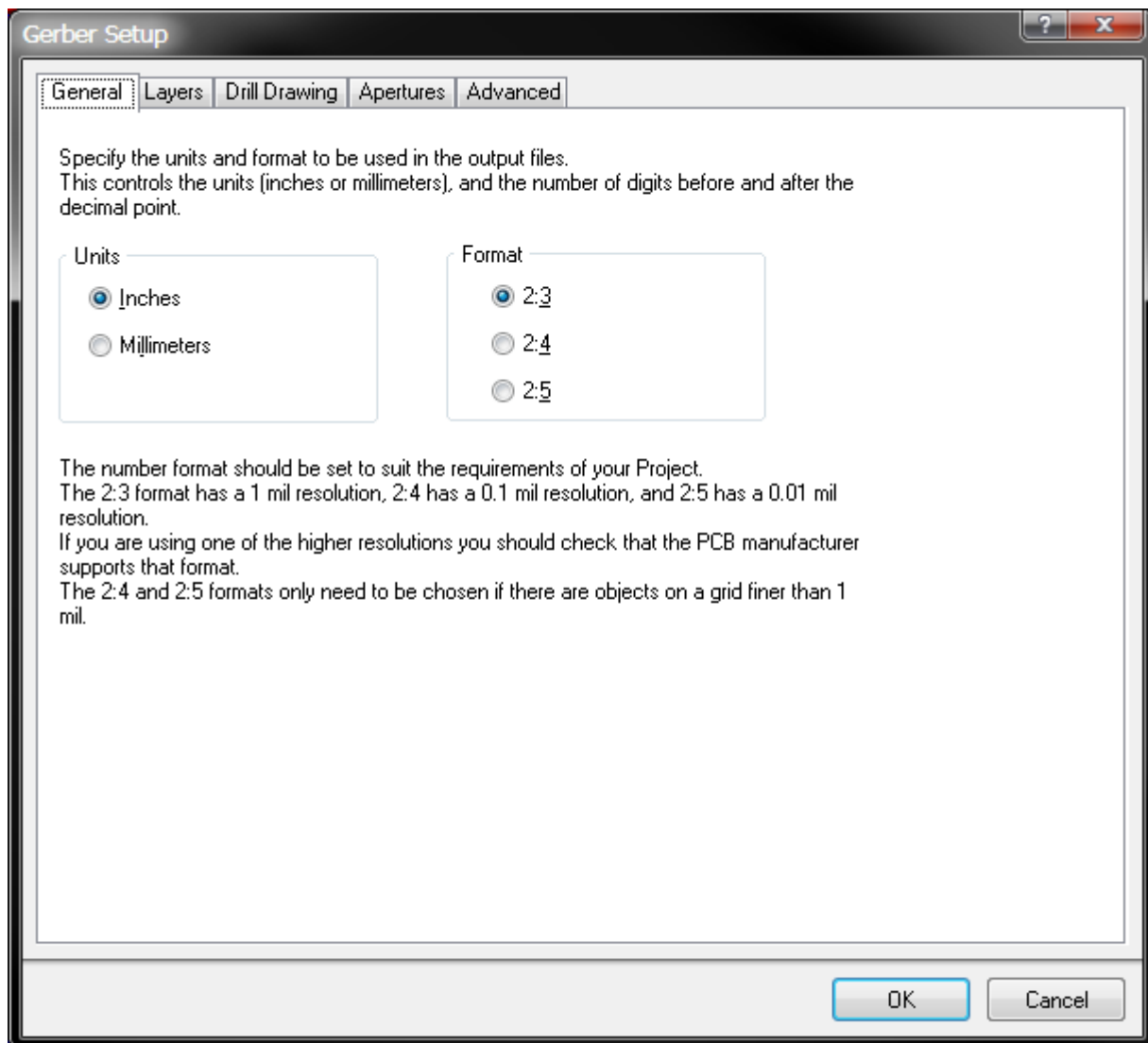
Access

Gerber output can be generated in one of two ways:

- Using a Gerber output generator in an OutputJob Configuration file (*.OutJob). Output is generated when the configured output generator is run.
- Directly from within the active PCB document using the **File » Fabrication Outputs » Gerber Files** menu command. Output will be generated immediately upon clicking **OK** in the *Gerber Setup* dialog.

Note : The settings defined in the *Gerber Setup* dialog when generating output directly from the PCB are distinct and separate to those defined for the same output type in an Output Job Configuration file. In the case of the former, the settings are stored in the project file, whereas for the latter they are stored in the Output Job Configuration file.

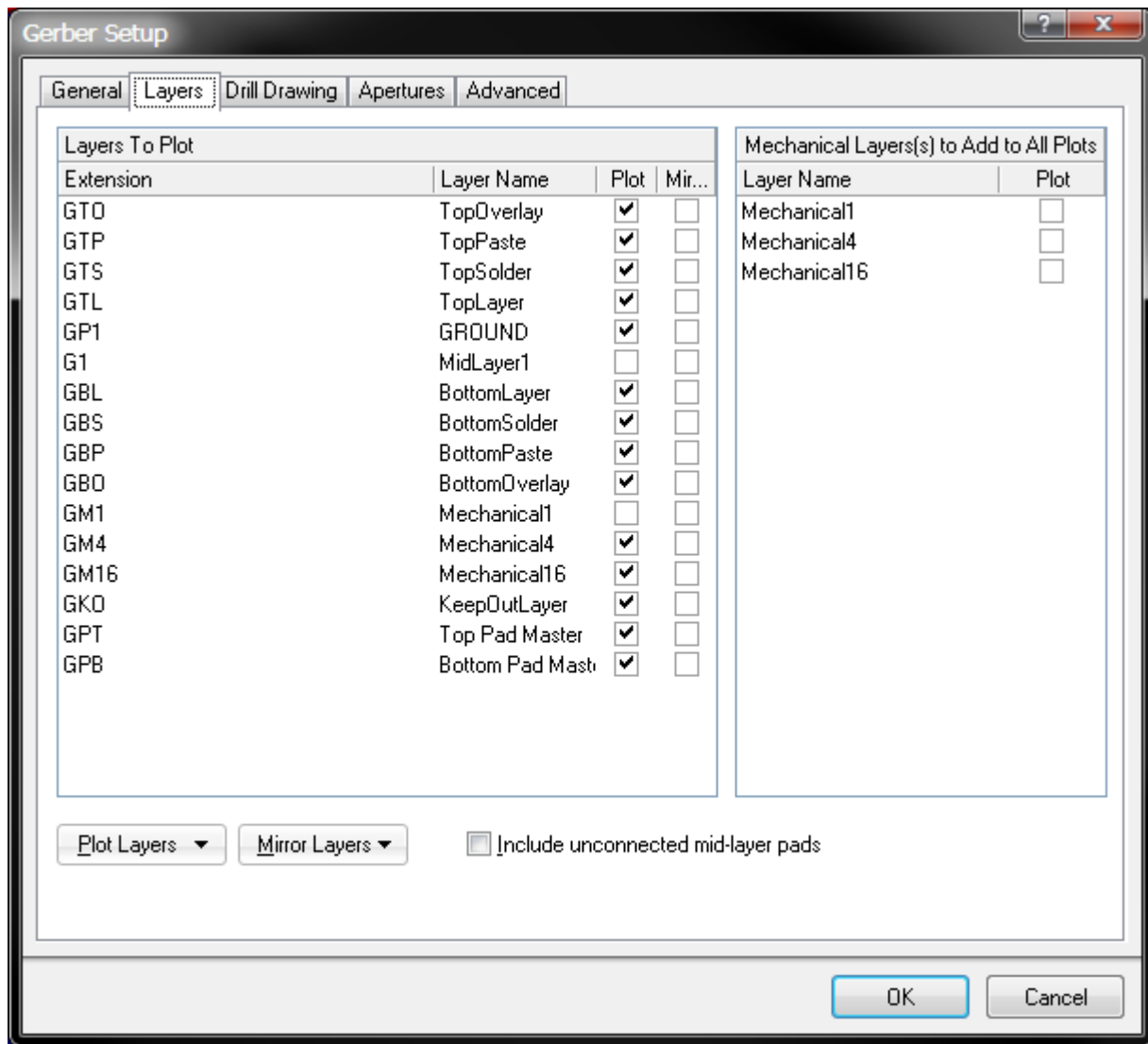
General Tab



The Gerber Setup - General Dialog.

- **Units**- The units can be either inches or millimeters.
 - Inches - Check this option to use inches as unit
 - Millimeters - Check this option to use millimeters as unit
- **Format**- The format specifies the precision of the coordinate data, which must be selected to suit the placement precision of the objects in the PCB workspace.
 - 2:3 - The **2:3** format has a resolution of 1 mil (1/1000 inch).
 - 2:4 - The **2:4** format has a resolution of 0.1 mil
 - 2:5 - The **2:5** format has a resolution of 0.01 mil

Layers Tab



The Gerber Setup - Layers Dialog.

Layer To Plot

Use this tab to enable each layer that you want to generate a Gerber file for.

- **Extension** - List the file extension for each Gerber
- **Layer Name** - List the layer name.
- **Plot** - Check the box at the right of each layer to enable the gerber creation for this layer
- **Mirror** - Check the box at the right of each layer allows a mirrored Gerber file is to be created.

Mechanical Layers to Add to All Gerber plots

Use this control to nominate which mechanical layer(s) are to be added to all Gerber plots.

- **Layer Name** - Display all the mechanical layer name
- **Plot** - Check this option to add selected mechanical layer to all Gerber plots.

Plot Layers

- **All On** - Select this option to check all boxes in Plot column, which means Gerber plots will be created for all listed layers.
- **All Off** - Select this option to clear all boxes in Plot column, which means NO Gerber plots will be created.

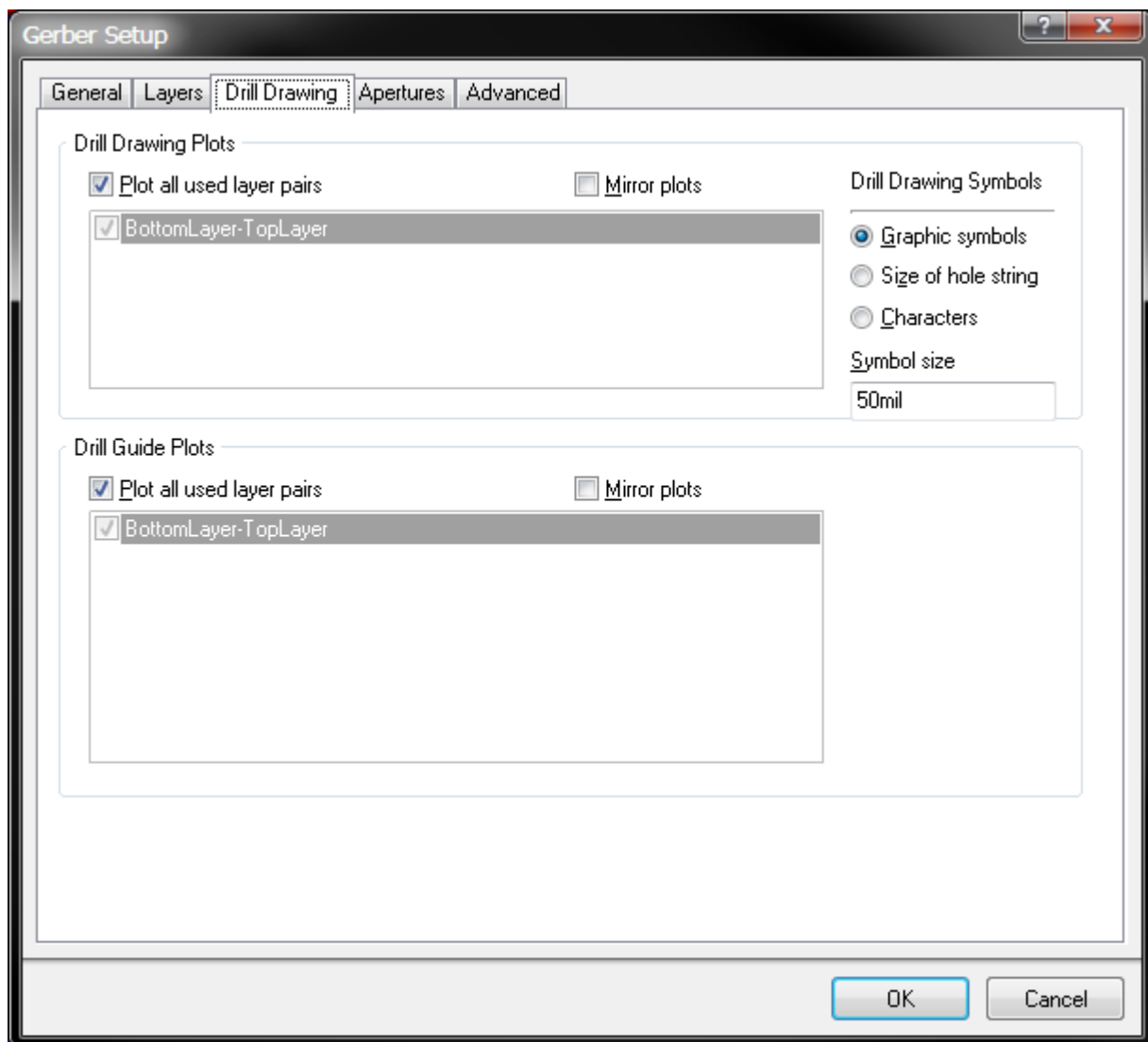
- **Used On** - Select this option to check all the used layers in Plot column.

Mirror Layers

- **All On** - Select this option to check all boxes in Mirrored column, which means mirrored Gerber plots will be created for selected layers.
- **All Off** - Select this option to clear all boxes in Mirrored column, which means NO mirrored Gerber plots will be created.
- **Used On** - Select this option to check all the used layers in Mirrored column.

Include unconnected mid-layer pads - Check this option to allow unconnected pads in mid-layer on Gerber plots.

Drill Drawing Tab



The Gerber Setup - Drill Drawing Dialog.

Use this tab to specify which layer-pairs a drill drawing is required for (mirrored output files can also be generated).

Drill Drawing Plots

- **Plot All used layer pairs** - Check this option to plot all defined layer pairs in drill drawing plots.
- **Mirror plot** - Check this option to mirror layer pairs in drill drawing plots
- **Layer Pairs Region** - This area shows all defined layer pairs in the design. Check the box in front of the layer pair to draw that layer pairs in drill drawing plots.

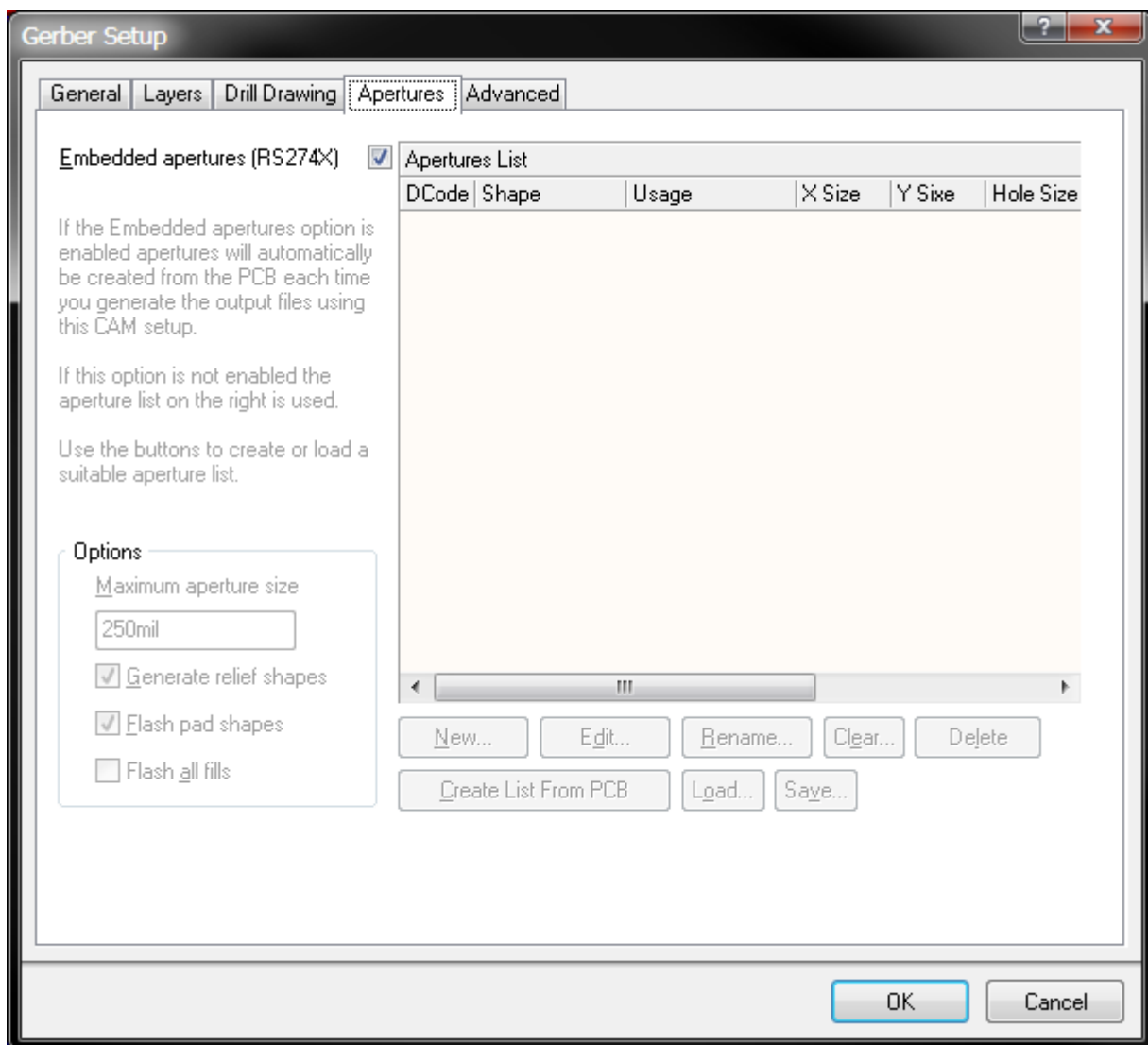
Drill Guide Plots

- **Plot All used layer pairs** - Check this option to plot drill guide in all defined layer pairs.
- **Mirror plot** - Check this option to mirror drill guide plots
- **Layer Pairs Region** - This area shows all defined layer pairs in the design. Check the box in front of the layer pair to plot drill guide for that layer pairs.

.Legend Symbols - Define the appearance of the legend symbol. Available symbols include Graphic Symbols, Size of hole string, Characters.

Symbol size - Define the size of the legend symbol.

Aperture Tab



The Gerber Setup - Aperture Dialog.

Use this tab to enable/setup the required aperture information for the design.

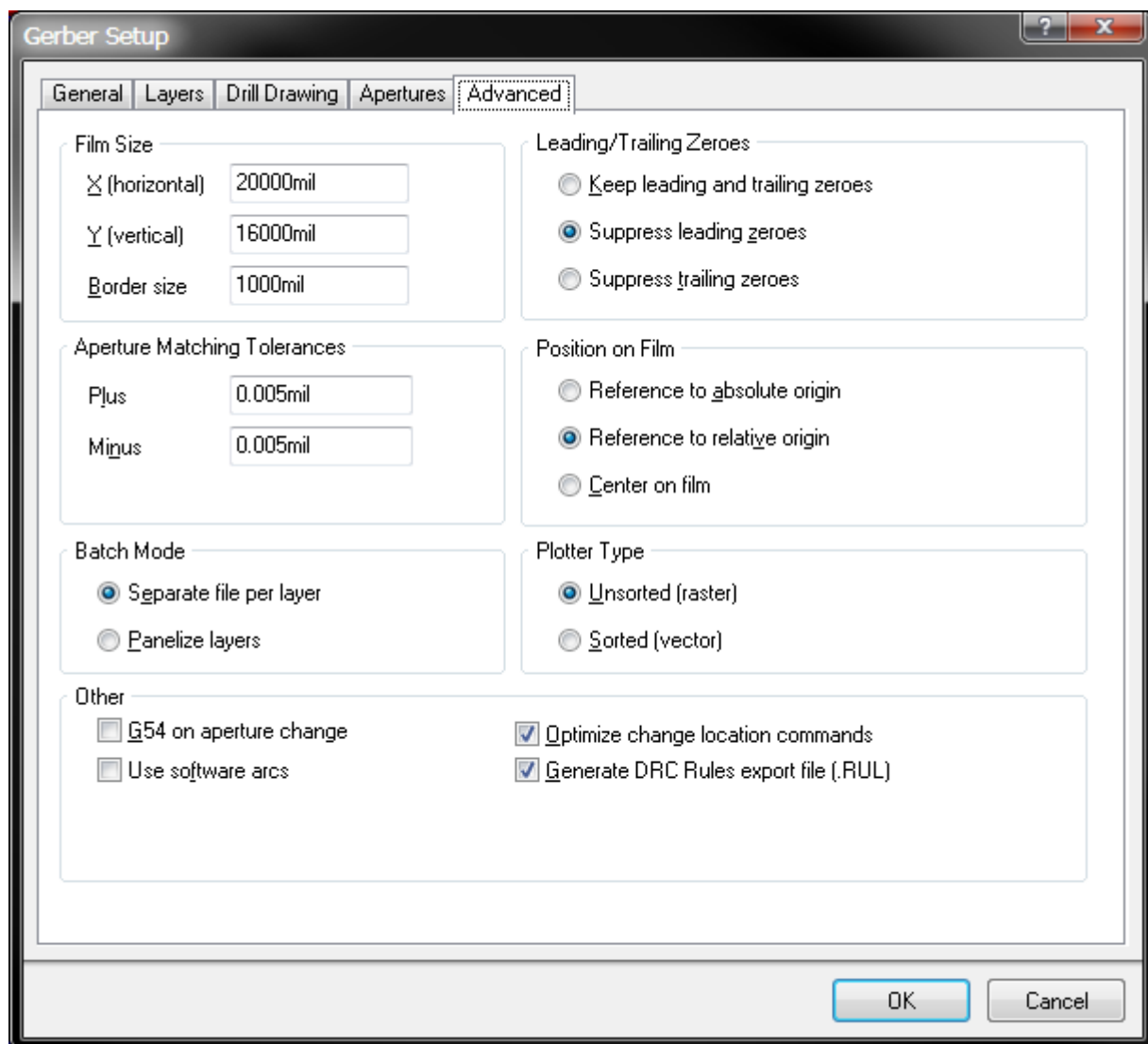
Embedded apertures (RS274X) - When you enable the **Embedded apertures (RS274X)** option, the aperture list is created automatically, each time you generate the Gerber files. The apertures are then embedded in the Gerber files, according to the RS274X standard. This feature means that you do not need to worry about whether the current aperture list includes all the required apertures. If you do not enable this option, the main apertures region of the tab becomes enabled, from where you can load, create, edit and save an aperture table exactly as you require.

Tips on Apertures

It is highly recommended that you use the **Embedded apertures (RS274X)** option. Our modern photoplotters are raster plotters which can accept any size aperture.

If you are using automatic aperture generation, the required apertures for Drill Drawing symbols and other text is automatically provided with the stroke width being proportional to the text height. For example, the default 50 mil Drill Drawing symbol size yields a 7mil aperture for stroking the symbols. 60 mil text height yields 8 mils stroke width. You should take care in defining text sizes for Silkscreen (or Overlay) layers, as very fine strokes may not be successfully screened onto the board.

Advanced Tab



The Gerber Setup - Advanced Dialog.

Use this tab to specify options such as the virtual film size to be used during Gerber generation, aperture matching tolerances and the use of zero suppression.

Film Size

- **X(horizontal)** - Virtual film length
- **Y(vertical)** - Virtual film width
- **Border size** - Border size of the virtual film

Leading/Trailing Zeroes

For example, consider the generation of NC Drill files with format **2:5** . Using this format can yield the following data values: 00.00001 10.00000

- **Keep leading and trailing zeroes** - If this option is enabled, these values will keep as: 00.00001
10.00000
- **Suppress leading zeroes** - If this option is enabled, these values will appear in the file as: 1
10.00000
- **Suppress trailing zeroes** - If this option is enabled, these values will appear in the file as:
00.00001 1

Aperture Matching Tolerances

- **Plus** - Define plus tolerances
- **Minus** - Define minus tolerances

Position on Film

- **Reference to absolute origin** - Refer to absolute origin
- **Reference to relative origin** - Refer to relative origin
- **Center on film** - Put the gerber center on film

Batch Mode

- **Separat file per layer** - Each layer will generate a separate gerber file
- **Panelize layers** - Only one gerber file will be generated in the format of panelization.

Plotter Type

- **Unsorted(raster)** - Use raster machine
- **Sorted(vector)** - Use vector machine

Others

- **G54 on aperture change** - Check this option to be compatible with obsolete photo plotter.
- **Use software arcs** - Check this option to use software arcs.
- **Optimize change location commands** - Check this option to optimize the change location commands.
- **Generate DRC Rules export file(.RUL)** - This file contains all design rules defined for the source PCB document from which the Gerber data is being generated. This file is generated provided that the Generate DRC Rules export file (.RUL) option is enabled on the **Advanced** tab of the *Gerber Setup* dialog.

Generated Files

The following file extensions are used to identify each Gerber file. The filename for each Gerber file is the PCB filename when the Gerbers are generated via the **File » Fabrication Outputs » Gerber Files** menu command. For Gerbers generated via an OutputJob, the default is to use the PCB filename, however this can be overridden if required.

Gerber Extension	Description
G1, G2, etc	Mid-layer 1, 2, etc
GBL	Bottom Layer
GBO	Bottom Overlay
GBP	Bottom Paste Mask
GBS	Bottom Solder Mask
GD1, GD2, etc	Drill Drawing (assignment based on order of drill pairs appearing in the <i>Drill-Pair Manager</i> dialog)
GG1, GG2, etc	Drill Guide (assignment based on order of drill pairs appearing in the <i>Drill-Pair Manager</i> dialog)
GKO	Keep Out Layer
GM1, GM2, etc	Mechanical Layer 1, 2, etc
GP1, GP2, etc	Internal Plane Layer 1, 2, etc
GPB	Pad Master Bottom
GPT	Pad Master Top
GTL	Top Layer
GTO	Top Overlay
GTP	Top Paste Mask
GTS	Top Solder Mask
P01, P02, etc	Gerber Panels
APR	Aperture File (generated when embedded apertures (RS274X) are used)
APT	Aperture File (generated when embedded apertures (RS274X) are not used)

Once generated, the output will be added to the project and appear in the **Projects** panel under the Generated folder, in an appropriately-named sub-folder. If you have used a separate folder for each output type, then corresponding (separate) Generated folders will be added to the **Projects** panel (e.g. Generated (Gerber Output)).

Location of Generated Files

The output path for generated files depends on how the output was generated:

- From an OutputJob file - the generated files are stored in a folder within the project folder, the naming and folder structure is defined in the **Output Container** that the Gerber File output is targeting.
- Directly from the PCB -the output path is specified in the **Options** tab of the *Options for Project* dialog. By default, the output path is set to a sub-folder under the folder that contains the Project file and has the name: Project Outputs for ProjectName. The output path can be changed as required. If the option to use a separate folder for each output type has been enabled in the **Options** tab, then the Gerber files will be written to a further sub-folder, named: Gerber Output.

Automatically Opening the Generated Output

When generating Gerber output, you can specify that the output be opened automatically in a new CAM document. The way in which this is accomplished depends on how you are generating the output:

- From an OutputJob file - enable the **Gerber Output** auto-load option in the *Output Job Options* dialog (**Tools » Output Job Options** from the OutputJob Editor).
 - Directly from the PCB - ensure that the **Open outputs after compile** option is enabled on the **Options** tab of the *Options For Project* dialog (**Project » Project Options**).
-