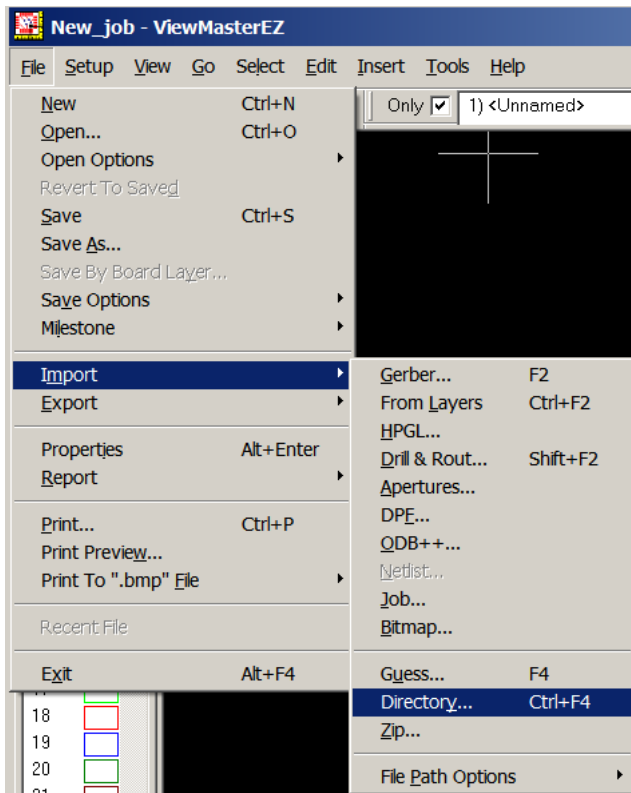# How To Panelize PCB Data Using ViewMasterEZ

You can import PCB CAM data in ViewMasterEZ and then edit it to produce panels that you can send to PCB manufacturers. ViewMasterEZ imports multiple file formats: Gerber, ODB++, NC drill data, Barco DPF, HPGL etc. In this app note we will focus on importing Gerber data, panelizing the data and then exporting the Gerber files for the panels.
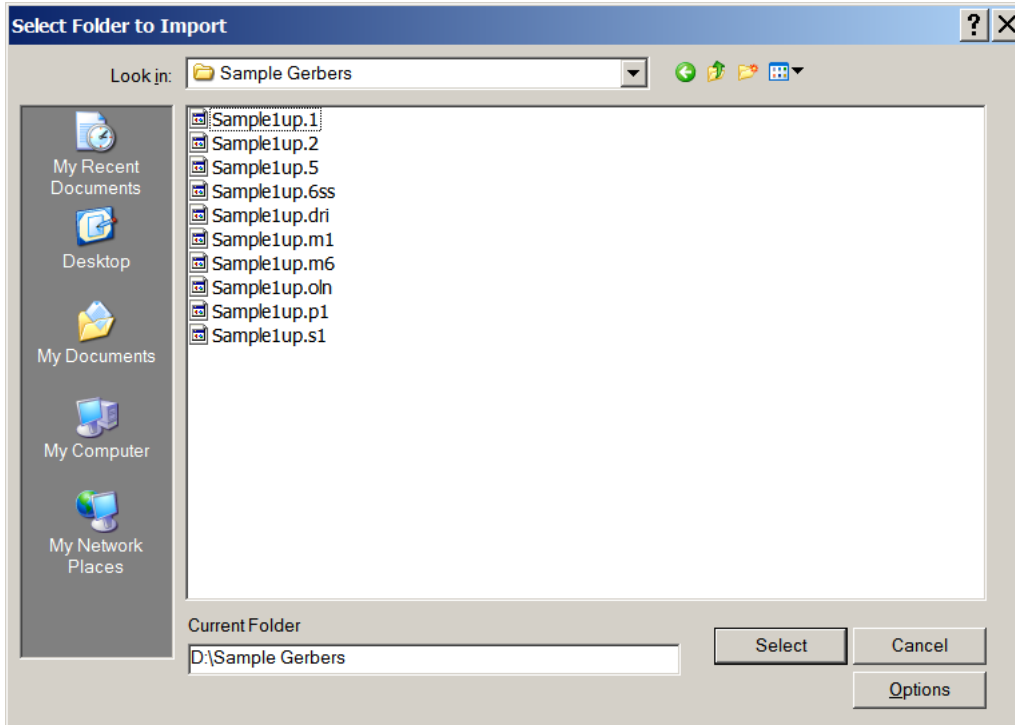
Importing The Files

There are many ways to import Gerber files into ViewMasterEZ:

- "`File/Import/Gerber`" or "`File/Import/Guess`": convenient when importing one file at a time. This method imports a file into the "current layer", so you have to make sure you have the current layer set before importing.
- Dragging and dropping a Gerber file onto the layers toolbar or into the main view. When dropping on the layers toolbar the data will be imported into the layer where the file is dropped. When dropping onto the main view, all previous data is erased and the data will be imported into layer 1.
- "`File/Import/Directory`": this is the preferred method when you have several files that make up a design (one file per layer).

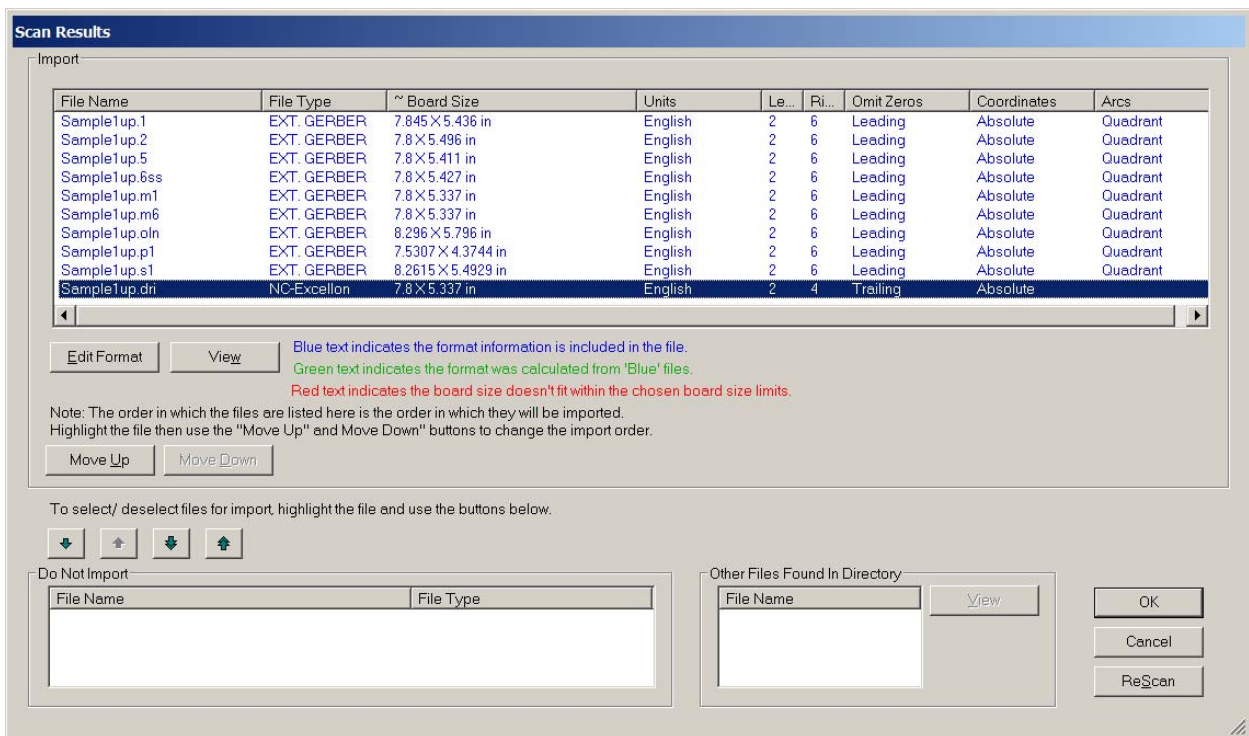In our example we will use "`File/Import/Directory`". Activate the command as shown below:

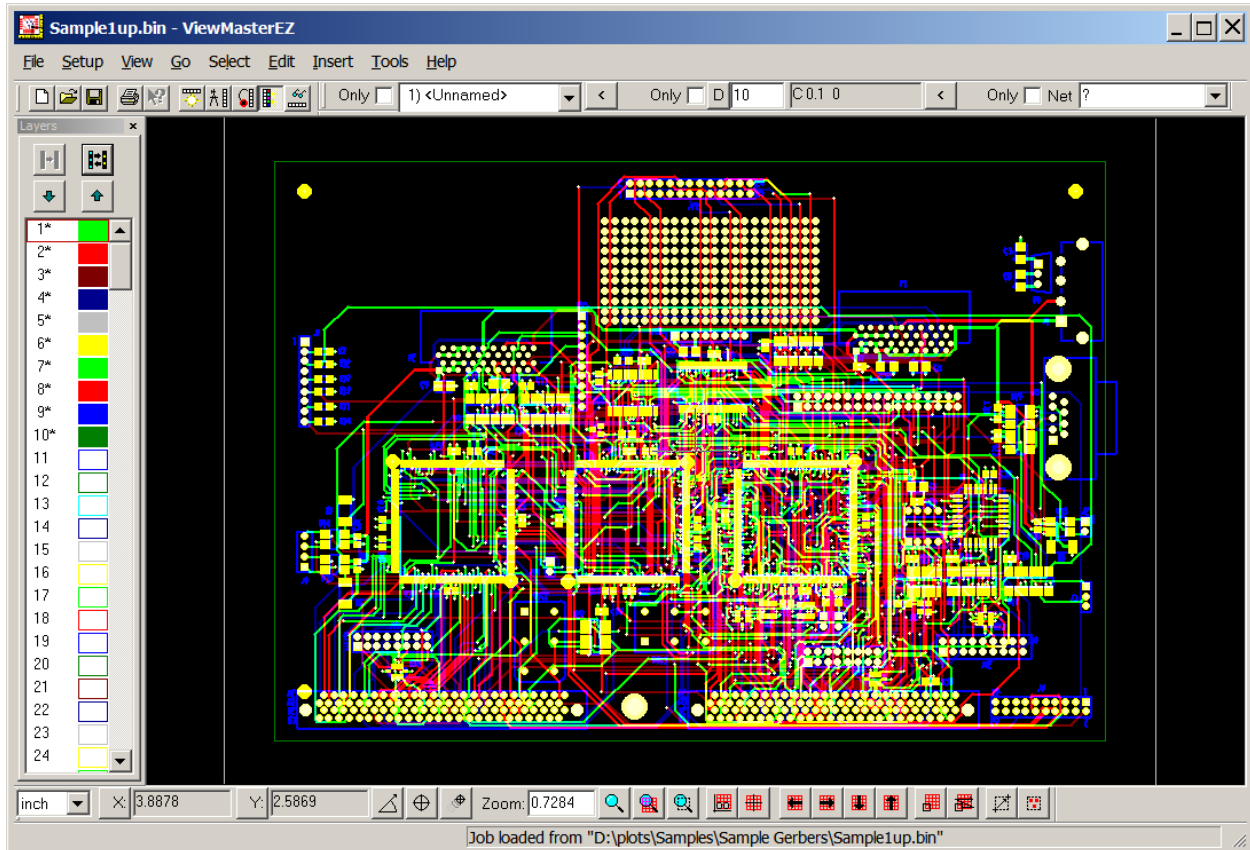Navigate to the desired folder and press "`Select`":

It is best if the folder contains only the data to be imported, because ViewMasterEZ will examine all the files in the folder and try to determine the file type and format of each. If there are many extra files, the work to interpret them is wasted time.

You will get a result dialog as below:

ViewMasterEZ has found a number of Gerber files and one Excellon NC-drill file. If desired, you can rearrange the order of the files using the "Move Up" and "Move Down" buttons. You can also remove files from the list to be imported by pressing the down arrow in the dialog. Note that the format of the files has been detected by the scan routines so now you are ready to import. Press "OK".
The data is imported as shown:

If you want to see what each layer contains, you can do "Setup/Layers" and you will see:

Layers Table

L#	B Layer	Name	Type	Traces	Pads	Poly	Color	Vis	Neg	Sc
1	1	Sample1up.1	CPU	2462	2742	0		✔	☐	
2	2	Sample1up.2	SIG	1262	1668	0		✔	☐	
3	5	Sample1up.5	SIG	1045	1668	0		✔	☐	
4	6	Sample1up.6ss	CPL	1265	1677	0		✔	☐	
5	7	Sample1up.dri	DRI	0	1668	0		✔	☐	
6	8	Sample1up.m1	MSU	0	2742	0		✔	☐	
7	9	Sample1up.m6	MSL	0	1677	0		✔	☐	
8	10	Sample1up.p1	SPU	0	1065	0		✔	☐	
9	11	Sample1up.s1	SKU	4807	0	0		✔	☐	
10	12	Sample1up.oln	BOL	4	0	0		✔	☐	
11	11		SIG	0	0	0		✔	☐	

We have adjusted the layer "Type"s manually, but they have no relevance for step-and-repeating.

As you see, this data is properly aligned. Often the data in input files is not aligned, so this would be the next step to do before panelizing. This is commonly needed to align drill files with the Gerber data. We will not show you in this tutorial how to do the alignment and will assume that the data is aligned.
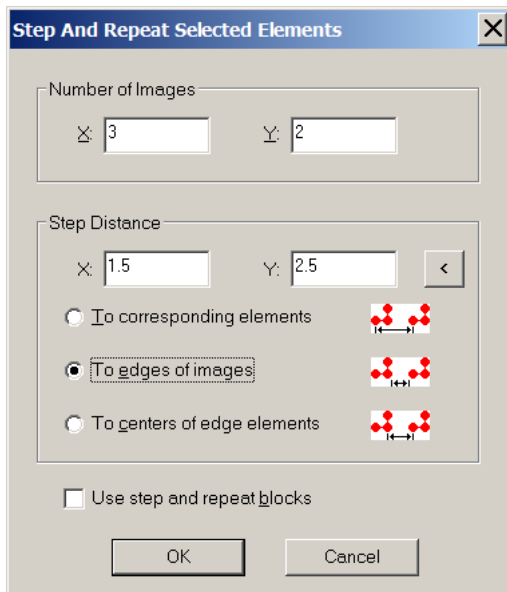
Panelizing The Loaded Data

The next step is the panelization. For this you will need to use the commands in the "`Edit`" menu. This tutorial shows the simplest and most common way of panelizing which replicates the data in a matrix. You can also use editing commands to generate more complex step and repeat patterns, where the steps are not the same or some images are rotated (etc).

The step and repeat commands ("`Edit/Edit Selection/Step And Repeat`") act on the "selection". You could in principle select only part of the data and step and repeat it, but in this case we want to step and repeat the whole data.

So, the first step will be to select the whole data ("`Select/Select All`"). Note that most of the View-MasterEZ commands have shortcut keys (which you can see when you pick the command from the menu). For "`Select All`" it is "`Ctrl+A`", so you could have just pressed `Ctrl+A` to select all the data. The selected data is shown in white, so at this point your view should be drawn in white.
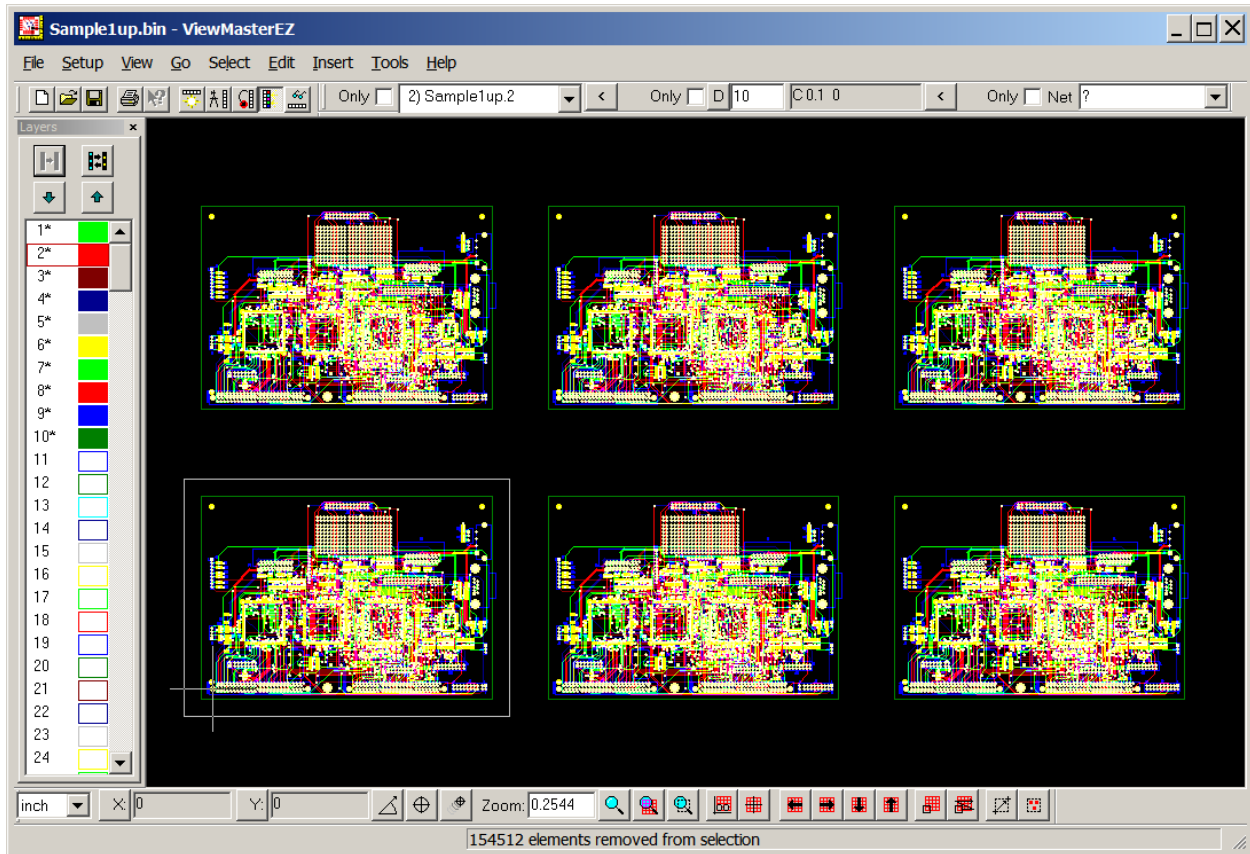
We will now step and repeat the selection by picking "`Edit/Edit Selection/Step And Repeat/New`". The following dialog appears:

We have specified 3 copies in X and 2 in Y which will create a matrix of 3 X 2, so 6 copies. The step between copies can be specified in the three ways shown. "To edge of images" will create the specified spacing between images, so 1.5 inches in X and 2.6 inches in Y.

We leave "`Use step and repeat blocks`" unchecked. If checked, the generated data will not be editable after the step and repeat and will be exported with step and repeat commands (so resulting in smaller files).

After pressing `OK`, deselecting ("`Select/Clear Selection`") and zooming out, we get the following view:

The data is step and repeated as desired. We are now ready to export the data to generate the files to be sent to the fabricator.

Exporting The Panelized Data

For this we will use the "File/Export/Gerber Extended" command. The export mode that we will pick uses the layer names as the file names of the exported data. Since we do not want to overwrite our original imported files, we will have to rename the layers. We do this in "Setup/Layers". You could type in any names you want in the "Name" column for each layer, but there is an automatic way to rename all the layers. For this, you type "Panel.*" (or "anything.*") in the name box for layer 1, then press Enter.
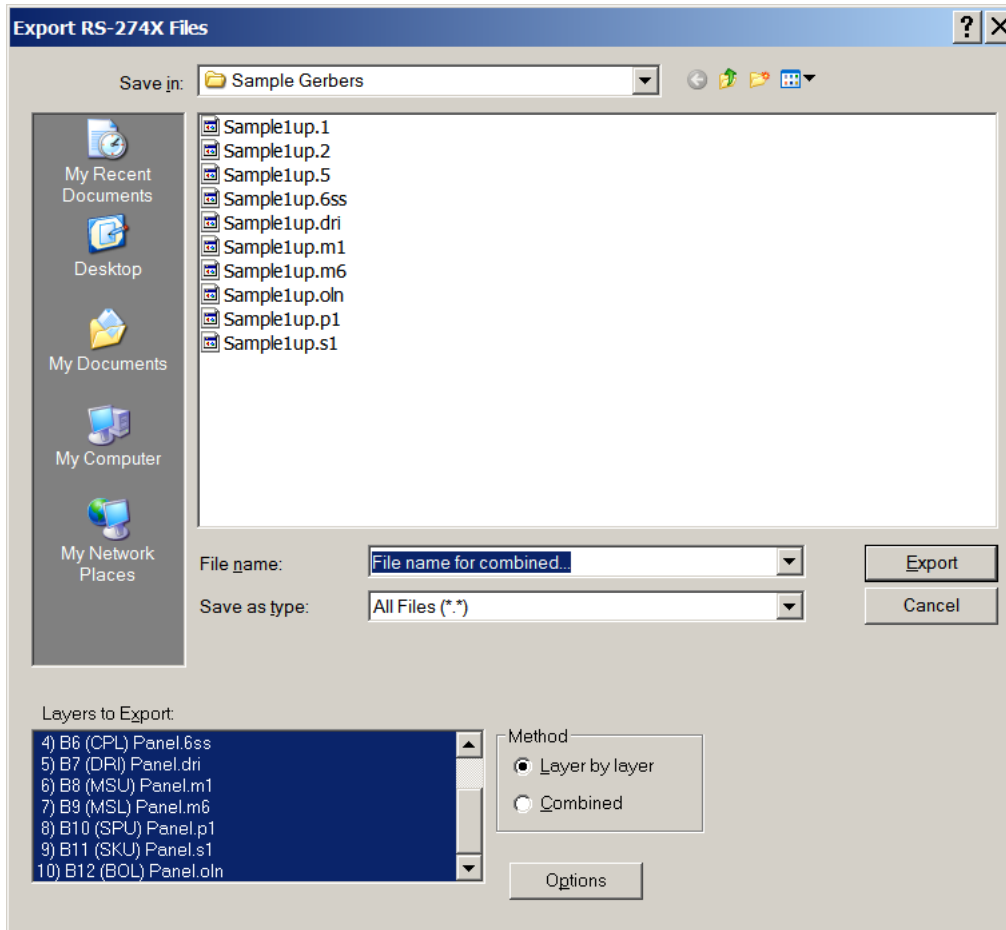
This will cause all the name prefixes to be replaced with "Panel", while keeping the suffixes (file extensions), as shown below:
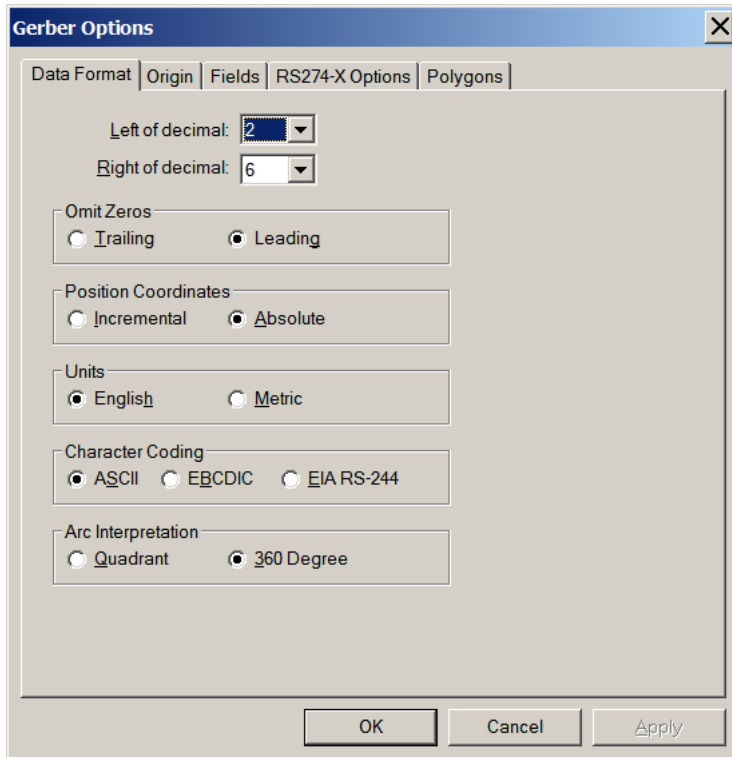

After closing the dialog, do "`File/Export/Gerber Extended`". The following dialog appears:

You can use this to navigate to any folder where you want the output file to go.

All layers are selected in the "Layers to Export" box and the "Method" is "Layer by layer". This will cause each layer to be exported to its own file with the file name given by the layer name. The "Combined" method causes all the layers to be exported merged into one file, whose name can be specified by typing in the "File name" box.

There is one more thing to set up: the format of the Gerber files. For this, press the "Options" button. You will get a dialog with many tabs with format options. To set these, you need to know about Gerber format. If you don't, you can leave the defaults or copy the settings from the import files (you can see those in the "File/Import/Directory" "Scan Results" dialog (see above).

The only important setting is the "`Right of decimal`", which specifies the resolution of the output data (the implied decimal point in English units is for inches). The larger you set this value, the less likely you are to get rounding problems. So, if the site you will be sending the data can accept it, best to set this value to the maximum (6 – which gives a resolution of a millionth of an inch).

After setting the options (you only need to do this once since your settings will be remembered), you can press "`Export`" in the export dialog. Your folder will now look like this:

The "Sample1up.*" files are the originals and the "Panel.*" files are the panelized Gerber data.

Notes

- One of the exported files is a drill file ("Panel.dri") and is exported in Excellon NC-drill format (not Gerber). This is because any layer which has the type set to "`DRI`" will be exported as NC-drill, even if "`File/Export/Gerber Extended`" is used. If you don't want this, you can always set the layer type to something else ("`SIG`", for instance). Drill files have their own format parameters and you will have to set them (once) to match your desired format. This can be found in "`File/Export/Drill`" – "`Options`"

- While working on a job it is often useful to save the state of the job so you can reload it. You can do this with "`File/Save`" or "`File/Save As`". This will create a binary file (with extension ".bin"), which contains all the data for your job and all the settings for shapes and layers. You can later re-open these files with "`File/Open`" or double-clicking on the file. Moreover, if your fabricator has PentaLogix CAM software, you can send them .bin files (no need to export Gerber). This way accuracy is guaranteed and you only need to send one file.

- You can download a manual for our CAM products from: ftp://ftp.pentalogix.com/Documents/CAMGuide.pdf. This manual is a bit out of date, but it contains descriptions of all the basic editing operations. Be sure to scan the release notes (.pdf file in the ViewMasterEZ installation folder) for new features and commands, as many have been added since the manual was created and they will make your life easier.