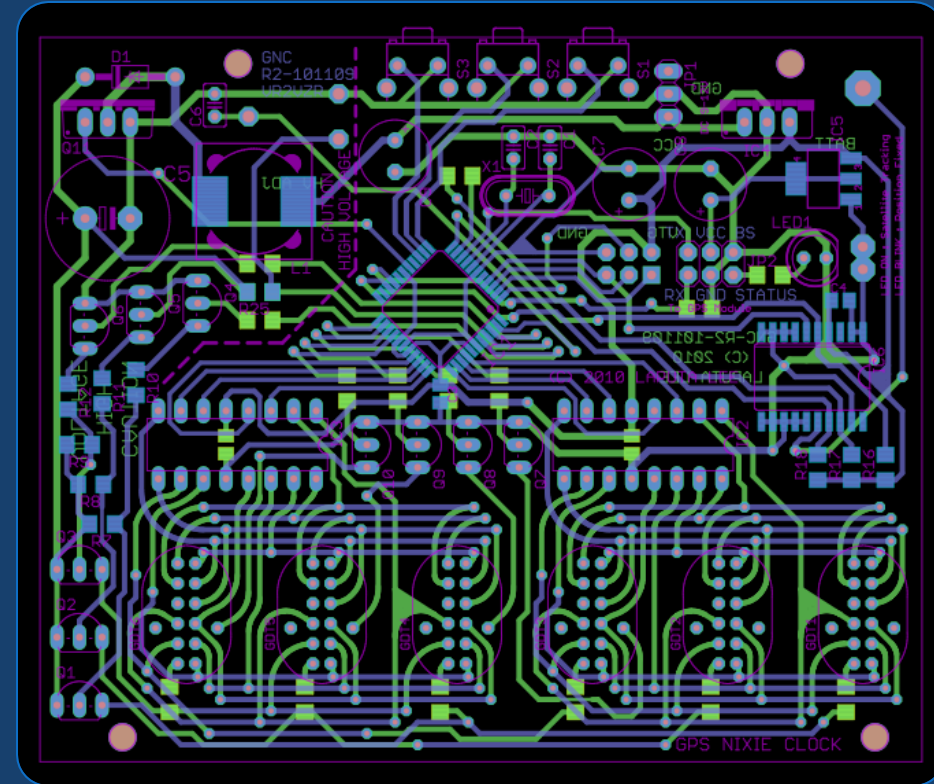


MCL Bootcamp

Gerber Files

What is a Gerber File?

- Standard electronics industry file format
- Used to communicate design information to manufacturers
- The electronics industry's equivalent of a PDF



Types of Gerber Files

- RS-274D – “old” standard
 - Spreads information for a single layer across at least two files
 - External aperture list or file
- RS-274X – most common
 - Allows all information for a layer to be contained in a single file
 - The aperture list is embedded
- ODB++
 - Intelligent, single data-structure to transferring PCB designs into fabrication, assembly, and test
- More file types may evolve as the industry progresses

How to View Gerber Files

- Many programs out there to view Gerber files
- MCL uses:
 - CAM350
 - Pentalogix View Mate
 - GC Prevue
 - Ucamco



Typical File Types Inside a Gerber

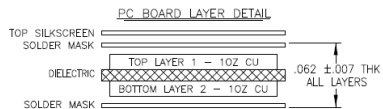
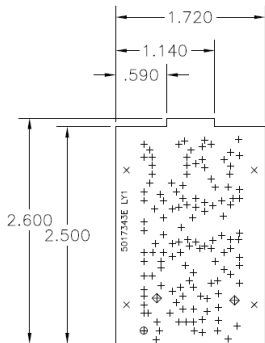
- Fabrication drawing/specs
- NC Drill file
- Drill map – there may be multiple drill files if blind/buried vias are present
- Individual copper layers – top/bottom copper, any internal layers
- Silkscreen files
- Solder mask files
- Outline or profile layer
- May have carbon print file, any other separate manufacturing processes, or paste

Fabrication File/Specs

- May be in PDF format, DXF, DWG, Gerber, Readme, or Microsoft Office
- Stack up
- May contain the drill map file

PROPERTY OF CANBERRA INDUSTRIES, INC.
COMPANY CONFIDENTIAL MATERIAL
UNAUTHORIZED DUPLICATION OR DISTRIBUTION OF ANY OR ALL
OF THE MATERIAL CONTAINED HEREIN IS STRICTLY PROHIBITED.

Rev	ECN	Description	By	Approv	Date
E	2343	SEE ECN	TAG		9-79
F	2493	REVISED	RAM		4-23-80
G	5529	Ø.152 WAS .165, ADD 60° CSK	RAM		2-14-90
H	5925	ADDED .086 60° CSK	WP		5-21-91
J	9310	REVISED ADDED SHT 2	HMC	MAL	12-15-94
K	9321	REVISED PER ECN	HMC	MAL	1-6-95
L	9334	REVISED WEBBING SHT 2	HMC	MAL	1-27-95
M	--	SEE ECO AR-C23818	KM	IN	7/1/08
N	--	SEE ECO 500000000416	KM	TBP	4/7/09
P	--	SEE ECO 500000005138	KM	FLB	



HOLE	SIZE	QTY
+	0.036 Plated	126
Φ	0.086 $\frac{5}{16}$ 60° NonPlated	1
X	0.136 NonPlated	4
⊕	0.152 $\frac{5}{16}$ 60° TO TOPSIDE NonPlated	2
TOTAL:		133

FABRICATION NOTES:

- Latest versions of all referenced material to be used unless otherwise specified.
- Fabricate in accordance with IPC-A-600 Class 2. Board to meet performance spec IPC-6012.
- Base laminate to meet requirements of IPC-4101/21.
- Plated through holes to have copper wall thickness as determined by IPC-6012 Class2. Plated through hole sizes are after plating.
- Silkscreen nomenclature to be white, non-conductive, non-soluble electronic grade ink and not to cover any portion of a component pad. Fabricator to clip any non-conforming silkscreen as required.
- Apply Green Liquid Photolimageable Solder Mask over bare copper per supplied artwork per IPC-SM-840 Type T Class 2. All exposed areas to be HASL. Surface mount pad surface finish must be flat to a maximum of 0.003" above board surface.
- The following fabricator supplied markings are required.
All other fabricator supplied markings are optional.
All markings to be in silkscreen,
Manufacturer's name or logo
Date code
UL logo (UL 94V-0, flammability rating)
- Fabricator shall not modify any elements of this design within board boundary except as specifically permitted. Fabrication masks may be adjusted to achieve finished line width shown in supplied artwork.
- Finished copper thicknesses are shown. Board thickness shown includes outside copper layers.
- Dimensions are in inches, unless otherwise specified.
Dimensional tolerances, unless otherwise specified:
All holes must be within .003 of radial true position
Hole to hole $\pm .003$
Hole to edge $\pm .005$
Edge to edge $\pm .005$
Hole diameter $\pm .003$
Layer to layer registration $\pm .005$

Canberra Industries, Inc.
800 Research Parkway
Meriden, Connecticut 06450
Tel. (203) 238-2351
Fax. (203) 235-1347

Title:

DRILL PRINT
SSB PREAMPLIFIER

Drawn	K.MESSERSCHMIDT	06/11/2008	Project	--	Last Sheet	01
Checked	J. PROVENZANO	6/23/08	CAGE Code	Size	Drawing Number	Sheet
Approved	IN	7/1/08	24114	C	17343	01
Scale	1/1					P
CAD Stamp	--					

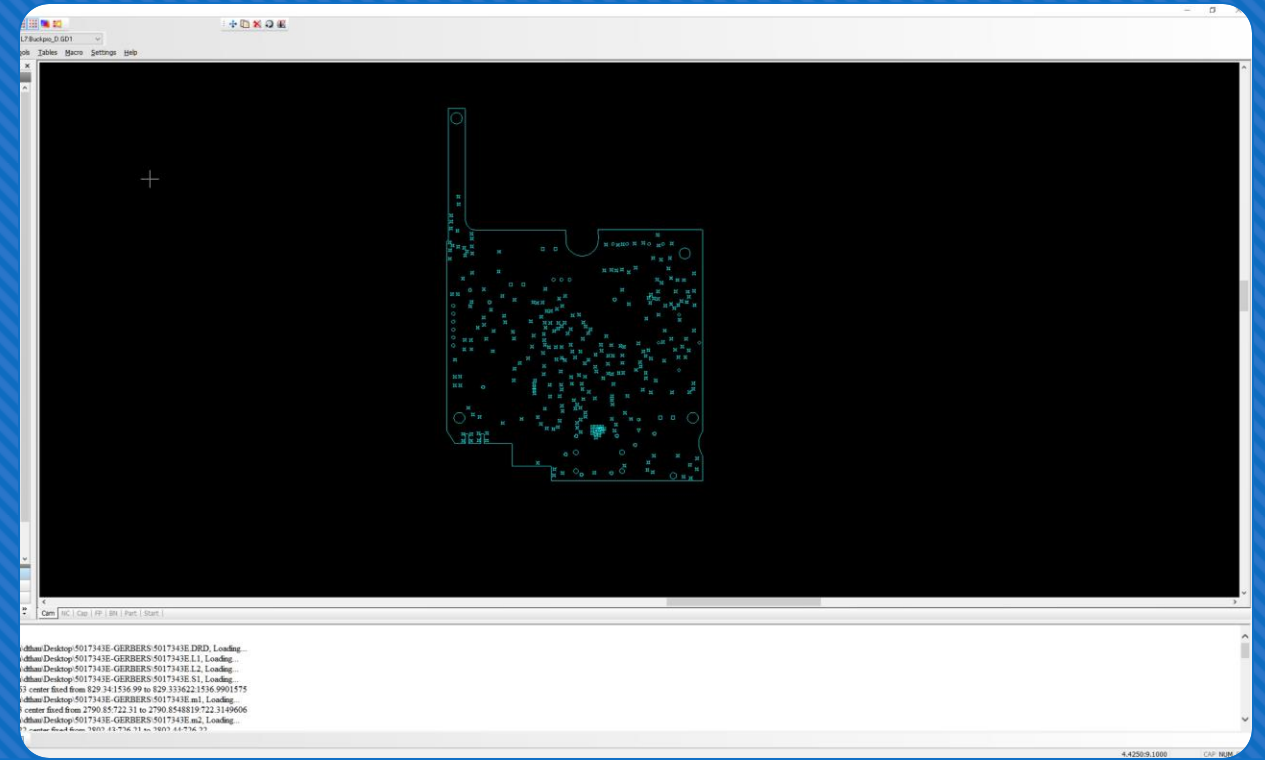
NC Drill File vs Drill Map

- NC drill file shows x and y coordinates of all the holes, but it shows them so that they are measurable
 - Typically in Excellon format
- Drill map lists all the drills in a chart
 - Uses symbols to differentiate between the drills

NC Drill File



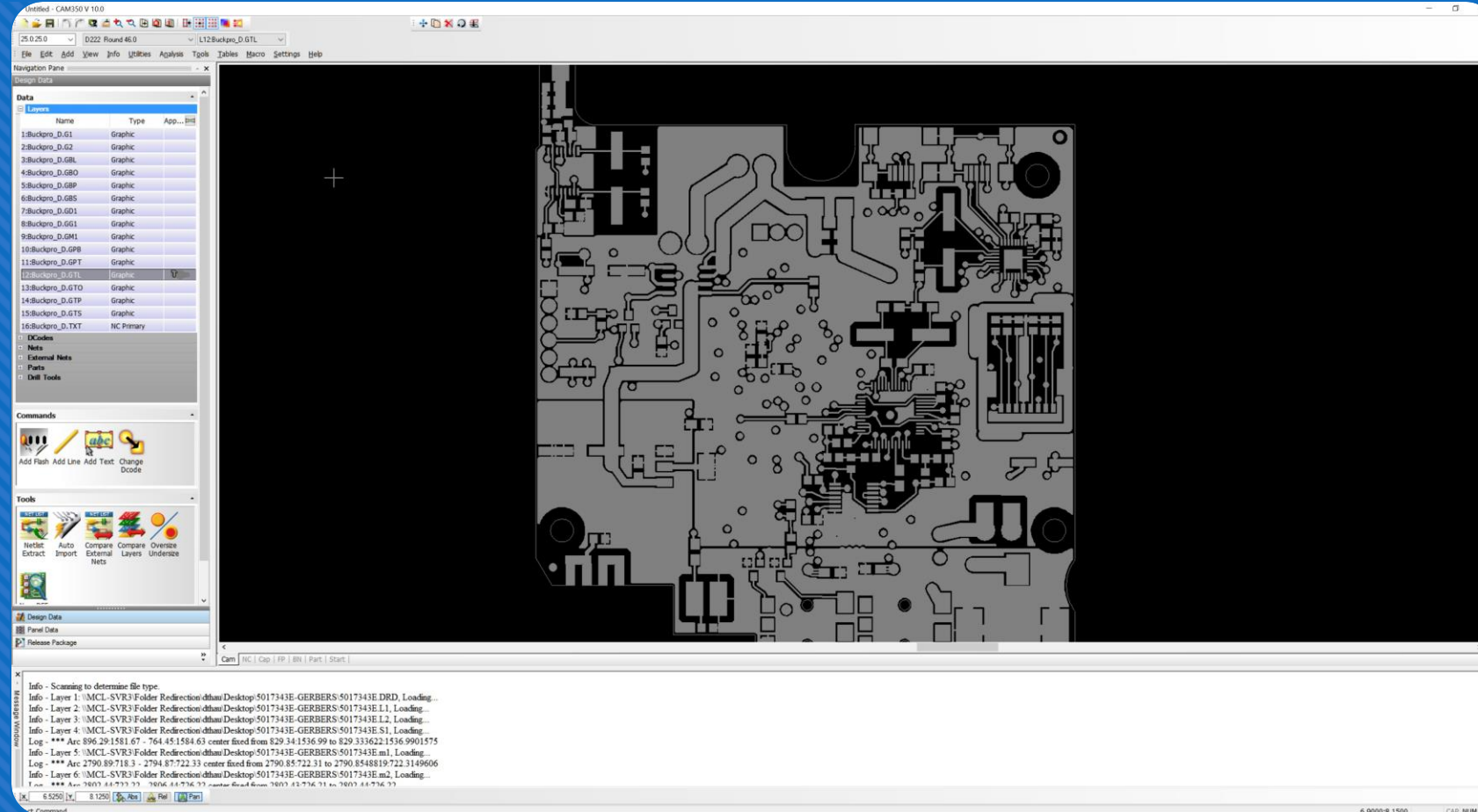
Drill Map



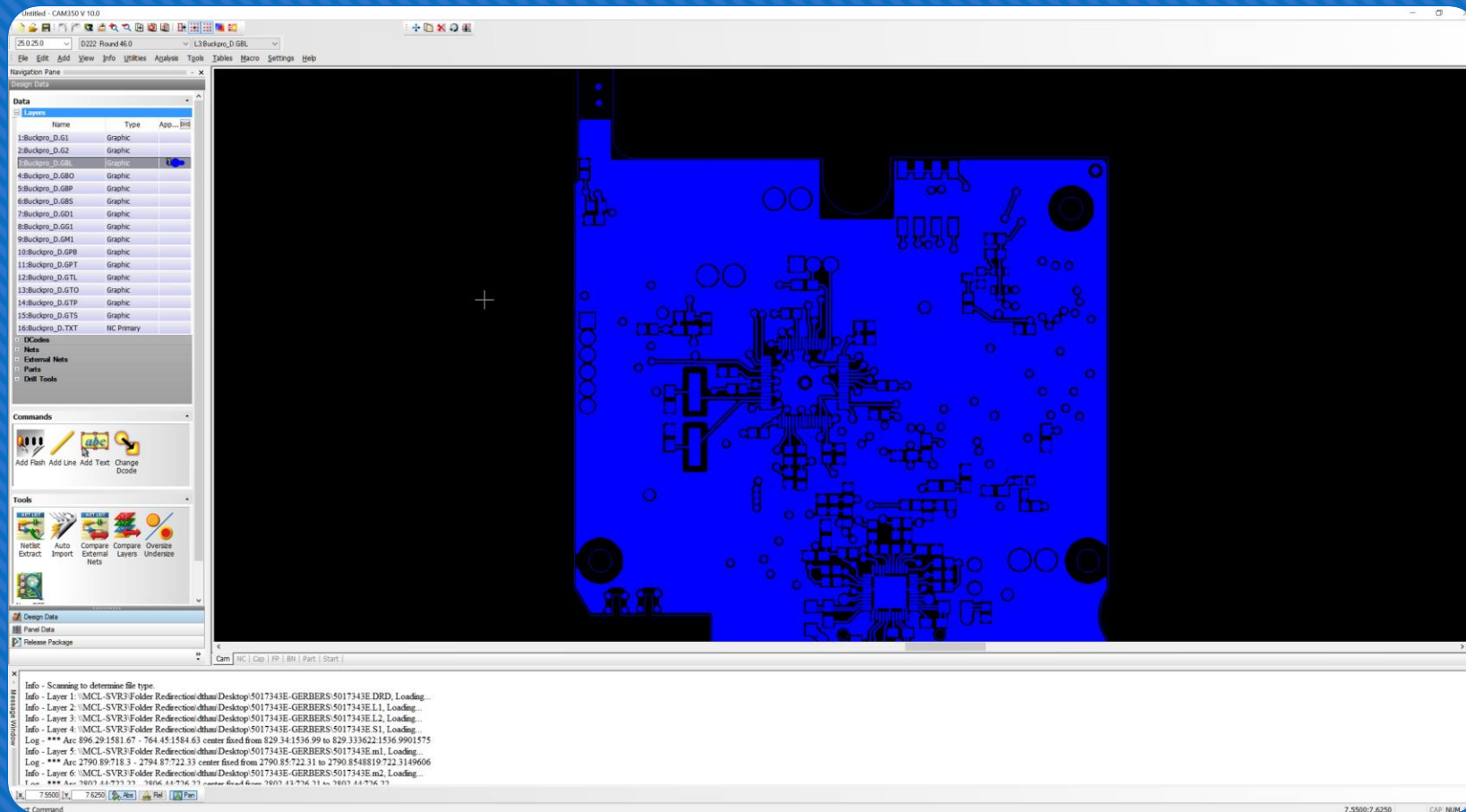
Side by Side NC Drill File vs Drill Map

Individual Copper Layers

- Typically shown as positive image
- All layers will appear similar but can differentiate the top and bottom due to the presence of component footprints as well as the extension of the Gerber File
 - Top layer - .top, .tl, .cs,
 - Bottom layer - .bot, .bl, .ss



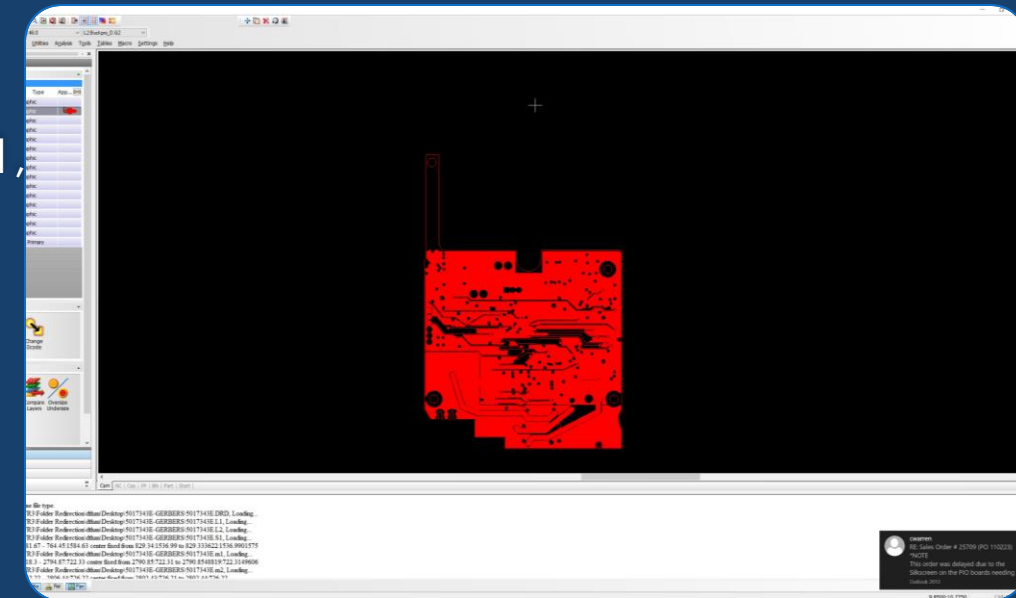
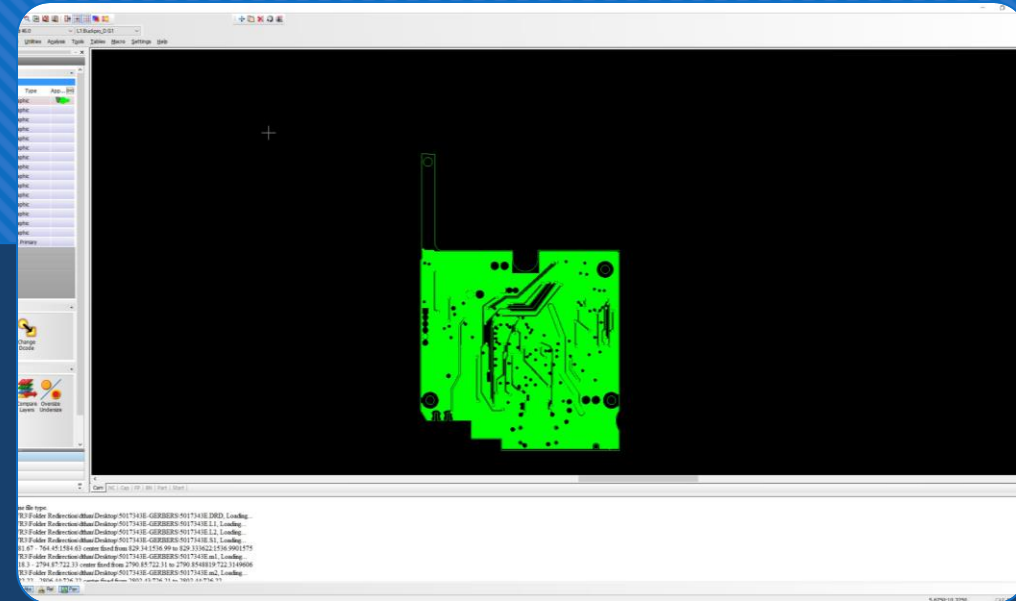
Top Copper Layer



Bottom Copper Layer

Inner Layers

- Thermals
- Won't see any component footprints
- Differentiated by extensions such as .l2, .gnd, .pwr, .in1, etc.



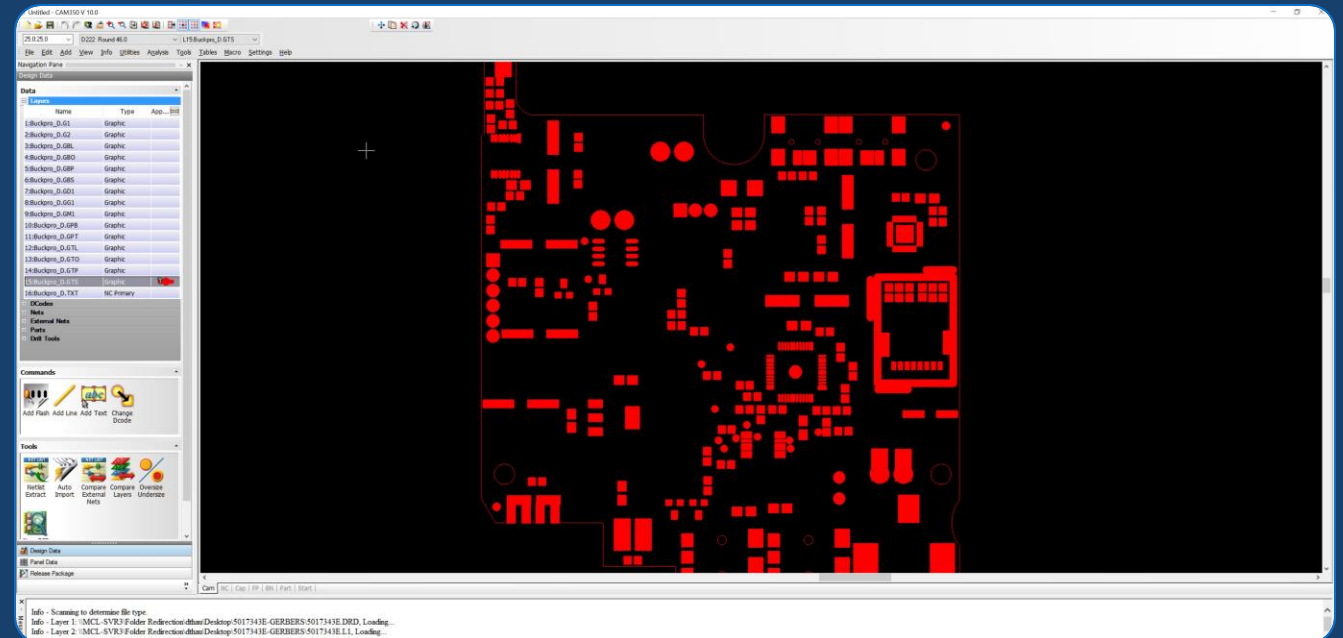
Silkscreen Files

- Viewed as the marking or the nomenclature that shows the component locations
- Any text on the board
- Typically shown as .to, .bo, .tsilk, .silk, .silk



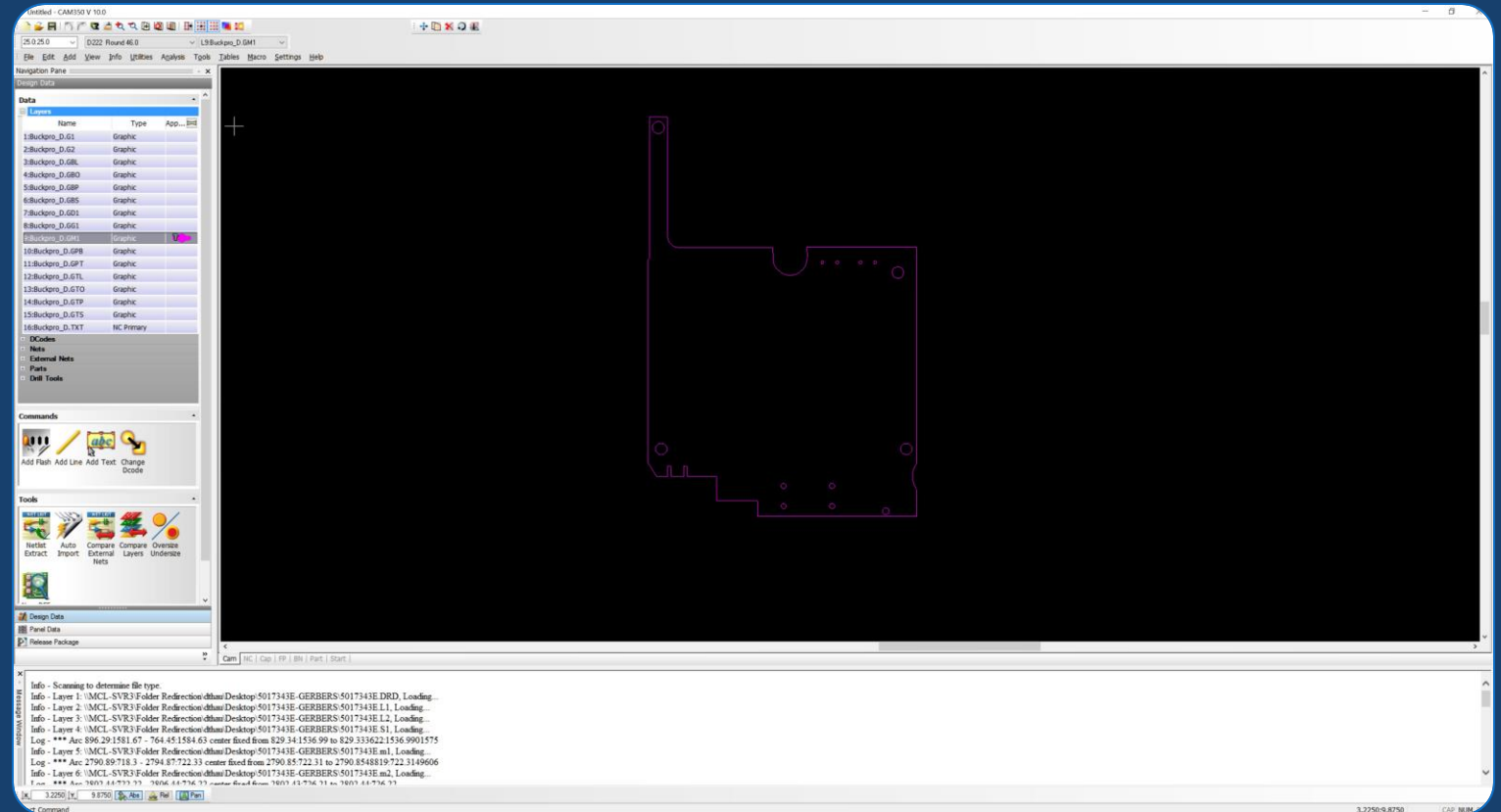
Solder Mask Files

- Actually a resist file – typically the black areas shown in the Gerber will translate to where solder mask will be placed
 - The areas with color denote the locations in which solder mask will not be added
 - During manufacturing any locations that have exposed copper and no solder mask will result in the finish of the board



Outline or Profile Layer

- The outline of the board – it may show internal cutouts/routes
- May show v-scores



How to Differentiate One Layer From Another

- How the file extension was named
- Information in the stack up – only way to know for sure
- Component footprints

