

The Design Process, Part 3

Building the Library: Drawing things in the library - Line Components

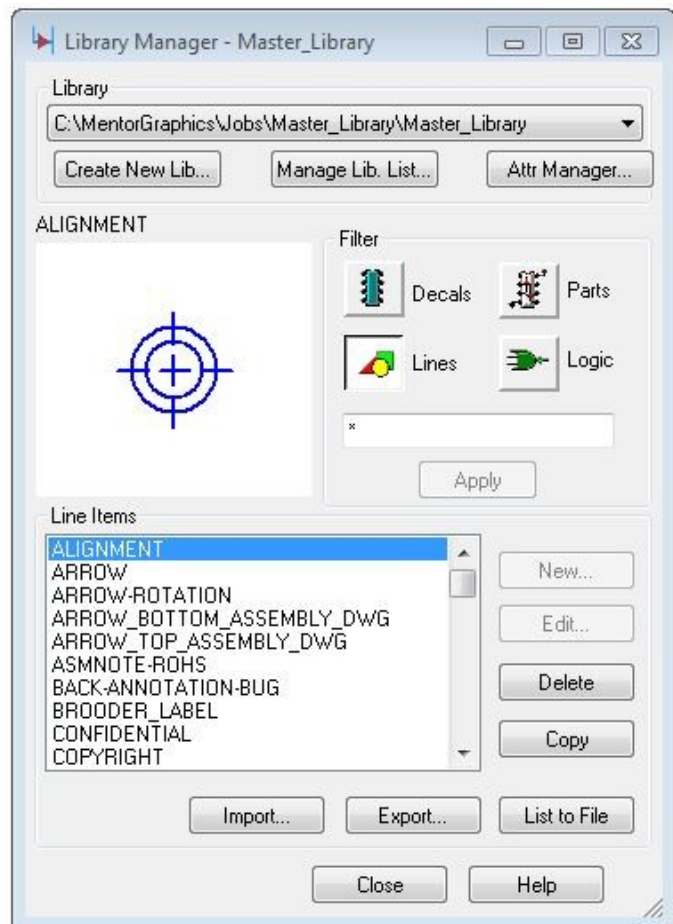
Sharon Hurtley-Durand
1 December 2021

Building the part library is the next step. I find it helpful to place each part on the schematic page as I build them. Then copy and move them as needed throughout the pages of the schematic.

In the library, you will find 4 categories: Decals, Parts, Lines, and Logic. In each category different types of things are drawn.

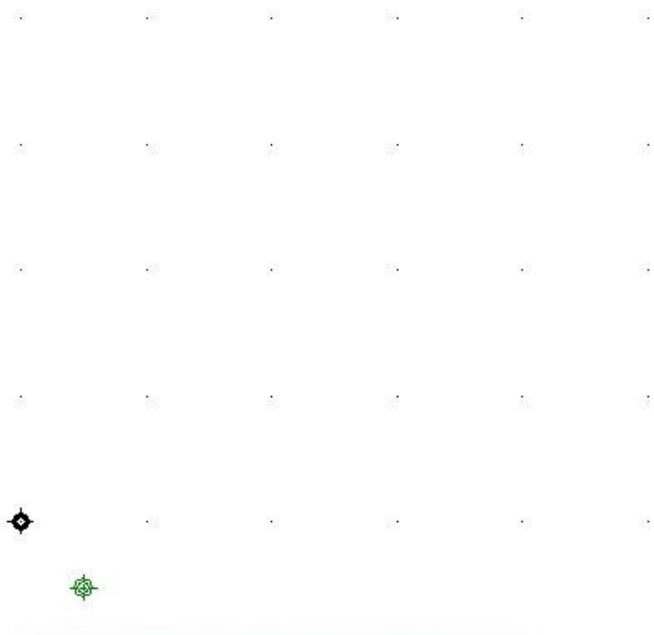
Here we show the category for Lines. In this area you can draw such things as the titleblock with text that appears on your schematic page, a logo, or as in the sample shown, an alignment target. Alignment targets are used on the Layout half of the software to keep different film layers aligned during the fabrication process.

The advantage of making a library component out of a group of lines and/or text is that you can just load them from the library whenever you need them instead of having to draw them from scratch each time they are used. None of these components are considered electrical, as in having any connection to any electrical component. They are merely pictures and text that are particular to their location in the schematic or layout. They may be inserted in either half of the software, but will not be linked to the other half, so will not automatically appear in the other location.



Here, you may also want to build line components for your default Layout page. Following is a screen capture of what I use. These components contain much the same information as the ones on the Schematic pages. Though in here, I have included default fabrication notes which inform the fabrication house of different necessary parameters. A lot of it is simple text which can be modified and to which numbers and additional text can be later added. It is a benefit to these line components that the text can be modified within the design. The lines can also be modified, but it is a more complex process.

FAB NOTES	
DIMENSIONS ARE IN INCHES (MM)	
DIMENSION TOLERANCES: DIMS = PLUS/MINUS 0.01" (0.254MM) DIMS = PLUS/MINUS 0.005" (0.127MM) ANGLES, PLUS/MINUS 1 DEGREE	
FABRICATION SPECIFICATIONS AND TOLERANCES, BOARD: [LEAD FREE /ROHS COMPLIANT MATERIAL: FR-4 THICKNESS = 0.062" (1.58MM) COPPER THICKNESS = 1 OZ (0.00157") COLOR: NATURAL FINISHED HARP OR 1V1ST: 0.090/12.000 MAX TEST: NONE VENDOR: DEFAULT LOGO, DATE CODE, UL	
HOLES: HOLE SIZES ARE AFTER PLATING FINISHED DIAMETERS: PLUS/MINUS 0.005" (0.127MM) COPPER PLATE: 0.001" (0.0254MM) THICK MIN LOCATION: 0.005" (0.127MM) ANNULAR RING OF TRUE POSITION MAX UNDRILLED LAMOS TOP 0, BOTTOM 0 SMT PAD DIMENSION MIN 0, SMT PAD-PAD CLEARANCE MIN 0, SMT PAD-PAD MINIMUM PITCH: 0, FINISH FOR CONDUCTOR PATTERN: ROHS COMPLIANT END PRODUCT VARIANCE: 10% MAX	
CONTACT FINGERS: FINGER USED TOP: NONE FINGER USED BOTTOM: NONE FINGER DIMENSION: NA FINGER PITCH: NA FINISH: GOLD PLATE: NA OVER LOW STRESS NICKEL: 0.0005" THICK MIN END PRODUCT VARIANCE: 10% MAX	
LINES: EXTERNAL TRACE WIDTH MIN: 0.008" (0.203MM) EXTERNAL TRACE CLEARANCE MIN: 0.008" (0.203MM) INTERNAL TRACE WIDTH MIN: NA INTERNAL TRACE CLEARANCE MIN: NA END PRODUCT VARIANCE: 10% MAX	
SOLDERMASK, TOP AND BOTTOM: SOLDERMASK OVER BARE COPPER MATERIAL: LM COLOR: GREEN	
SILKSCREEN, TOP: MATERIAL: PERMANENT HIGH-CONDUCTIVE EPOXY INK COLOR: WHITE	
LAYER ORDER FROM TOP OF BOARD: 1 - TOP SILKSCREEN 2 - TOP SOLDERMASK 3 - TOP 4 - BOTTOM 5 - BOTTOM SOLDERMASK	
DRILL PAIDS 1 - TOP-BOTTOM (MIN: 0.040" (0.127MM) THRU HOLE)	



Company Name	Layer:
Board Name, Rev.	BotTop
Date	BotTop Silkscreen
Printed Circuit Board Artwork	BotTop Fabrication Drawing
LAYOUT BY: Sherman Hartley-Burand	
 www.digintl.com	

CONFIDENTIAL – PROPERTY OF

PRELIMINARY DRAWING – DO NOT USE TO FAB BOARD

This particular set of Fab Notes deal with the most commonly used parameters of fabrication. Depending on your design, other elements may need to be specified. But, working from a basic list, you can easily add and delete elements as needed.

You may also want to add Assembly Notes to the Assembly Drawing Layers which might indicate where there are such things as keep out areas for manufacturers logos, or requirements for attaching heat sinks to certain components, or anything else that may not already be specified in the design.

Because PC Boards have multiple layers, so also does the drawing. The different colors used in the above reflect what layer that information is on. In this case what is in green is on all layers. The notes in purple are on the Drill Drawing layer (Fabrication Drawing layer). The different colored text in the titleblock reflect which layer that text is on. All of these colors are arbitrary and up to the user's preference. The black "target" above indicates the Origin of the drawing and is where everything else is internally dimensioned from.