How-To Design Electronics Modules and Use EAGLE for Schematic and Layout

Biorobotics and Locomotion Lab

by Emily McAdams with Jason Cortell, 2009

Motivation

A new electronics system was at the core of the overhaul of Ranger. The new electronics design was conceptualized at a high-level as a "nervous system." The idea was to create one "main brain" board that communicates with satellite boards that control and interact with different parts of the robot. The idea for the electronics redesign was to create a certain number modules each designed to perform a specific task. The different modules would be combined together as necessary with to form the satellite boards. The "main-brain" board as well as the "user-interface" board were designed separately because of the very specific nature of their task.

One of the main advantages of such a system was design efficiency. Instead of having to create each board from scratch, the boards could be created by combing different modules. For example, every board needs a power supply module and most need a microcontroller module. There are also some specific modules such as motor control that are only used on a specific board. This system also allows for quick creation of new boards if necessary. If the need arises for a new board the idea is that only a module needs to be designed and that module can be combined with others that have already been designed to form the new board. The purpose of this documentation is to help students working on design and layout of new modules and boards. This documentation contains information about the software, "how-tos," and particular specifications for all boards built in this lab.
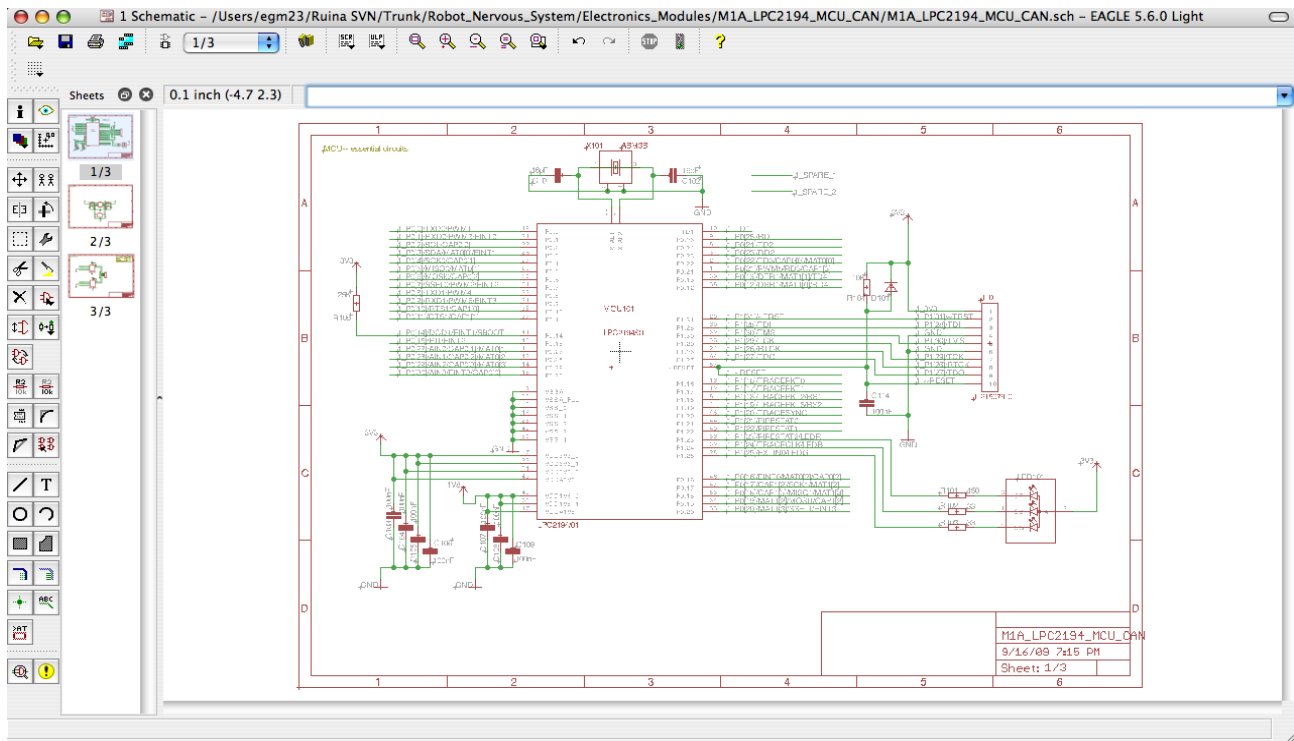
Creating a Schematic

1. *Obtain Necessary Software:* EAGLE is the software used in the lab for both schematic drawing and layout. EAGLE Light is available as a free download from http://www.cadsoftusa.com/download.html. While EAGLE light is a good starting point, most designs will require features only available in the full version. You can obtain the license key to upgrade your software from the lab manager.

2. *Begin a New Project:* Open EAGLE and select New --> Project. Begin by first drawing the schematic. Choose New → Schematic
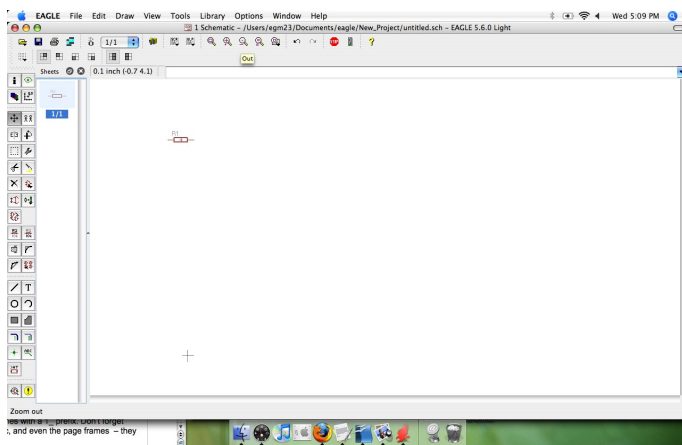
Picture 1: New Schematic Sheet in EAGLE

3. *Adding Components to the Schematic:* In this step there are two cases, either the component exists in the EAGLE library or you must create a library entry for that component. For the latter, refer to the next section on creating a library part, otherwise click on the "Add" icon in the left-hand side toolbar. The library should open. Browse or search the library to find your desired component. Highlight the component. If there are multiple packages for the device consult the part information and choose the appropriate package. It will be very helpful during layout if you have the right package selected. Click OK to exit the library screen and then place the component on the schematic by clicking anywhere on the sheet. At this point you can add as many instances of that component you need. You can leave "Add" mode by hitting ESC.

4. *Modify the Components*: Be sure to label each component with the correct name and value.

5. *Wire the Components:* Connect the components using nets. It is helpful to label these nets because it will help you fix any wiring errors.

6. *Run ERC*: Once you have finished the schematic, run ERC to try to catch any errors you have made in the design of your schematic. Do not move on to the layout phase if your schematic does not pass ERC.

7. *A Few More Things*: EAGLE schematic allows you to use multiple sheets. This is very helpful for a complicated module with lots of parts. The nets do not have to be physically attached as long as they are labeled the identically.

2: Example of A Finished Schematic

Creating an EAGLE Library Part

In some cases the part you need for your circuit cannot be found in the EAGLE library. In these cases, it is necessary to create a library entry in EAGLE for the part. The following are steps compiled by Jason Cortell to successfully add a part in EAGLE:

A. Use exactly one component per library file. This is not standard EAGLE practice, but it is necessary to maintain a version control of the boards. To get started, from the main EAGLE menu choose File → New → Library. From this library you will create symbols and packages and link them together to form a device.

B. Library Component Naming Convention: It is important to adhere to this convention because it makes the part searchable later and very clear what part it is when you are assembling the board.

> **1.** Use all capital letters. Separate sections of the file name with underscores, not spaces or hyphens.
>
> **2.** The first letters (or two) is the component type: C for capacitor, R for resistor, L for inductor, J for connector or jack, D for diode, T for transformer, Q for transistors, and U

for ICs

3. The next two sections of the name should give some clue to the function, if not obvious, and the manufacturer name, if not generic. E.g., J_USB_HIROSE

4. The next section of the name should give the exact manufacturer's part number of the component, if the part is not generic. E.g. SN74LVC1G07DRLR

5. The final section of the name should give the standard designation for the package type, if available, or the full number assigned to it by the manufacturer, if available.

C. To copy a preexisting library device or package, open the EAGLE control panel and find the desired device in the list view. If you don't see the folder you want click on the Options menu, then Directories..., and add the path to the library path search string. Open or create the destination library from the File menu in the Control Panel. You should see and empty window. Drag the desired device from the list view to the open destination library window. Save the destination library.

D. To copy a library symbol (or package), open the source library and the desired symbol or package, then use the dashed-square group selection tool to select the entire package or symbol (make sure all the layers are displayed). Then use the cut (scissors) tool to copy it to EAGLE's clipboard, open a new symbol or package window in the destination library, and use the paste tool to insert it. Note that even if you open multiple copies of EAGLE on the same computer they won't talk to one cannot and you cannot copy from one and paste to the other.

E. Making a new Library Symbol

1. From the library main menu, choose Symbol and enter the name of the new symbol. Following the naming convention specified above.

2. Put the origin of the symbol in the exact center of the part, unless you have a very good reason not to. This makes placing the symbols and moving much easier later.

3. Use the Symbols layer to draw your schematic representation of your component. A simple rectangle is an easy way to get started with an IC.

4. Add pins. I usually put the input pins on the left and the output pins on the right. Power supply pins can be at the top and bottom. Choose appropriate directions for the pin functions. E.g. input, output, power, NC, passive. Avoid using supply pins because they have the power to rename your signals and nets, possibly causing trouble (use power pins for Vcc and GND, for example).

5. Add pin function names where needed. If the functions are obvious from the symbols shape you don't need them. (Eg. Diode cathode and anode)

6. Turn on or off pin names and numbers. (The numbers will come from the package and the connections linking symbol to package when you put them together to make a device).

7. Add >NAME and >Value on layers Names and Values, using 0.07 proportional font at 8% ratio.

8. For the symbol, use only a 0.1 inch grid; otherwise you will not be able to connect wires to your part. No metric here (for the package it is fine).

F. Making a new library package:

1. From the library, choose package and edit. Name the package appropriately.

2. Put the origin in the exact center of the part, unless you have a very good reason not to. Machine part placement will not work otherwise.

3. Use 8 mil lines for silk screen layers (tNames, bNames, tPlace, bPlace). For surface mount devices, put the lanes on the top copper.

4. Use layer tNames for silkscreen text information. Use later tPlace to show placement location and orientation of parts. These should not overlap the pads, solder mask, etc.

5. Use layer tDoc for visual representation of the part for layout and documentation purposes. This should show the actual profile of the part and its leads, etc. Its okay for this to overlap the pads because it will not be on the silkscreen. This also helps to maintain clearance between parts during board layout.

6. Solder mask layer tStop should be 3 mils larger than the copper in every direction. This is normally a setting in the DRC, but if end up making oddly shaped pads with polygons then you'll need to know this.

7. Add >NAME and >VALUE on layers tNames and tValues, respectively using 0.04 vector font at 20% ratio.

8. Add a tKeepout boundary if the part is larger than the land pattern in any direction or if extra room is needed for soldering. Add tRestrict layer lines to keep our polygon/power layers from under a part.

G. Making an New Library Device

1. From the main library window, select device and name accordingly.

2. Add symbol from the list of available symbols in the library.

3. Add package or packages from the list of available packages. In some cases the exact same part is available in different packages and you may want to create a package for

each one. Instead of creating a new part for every different package, multiple packages can be attacked to one symbol.
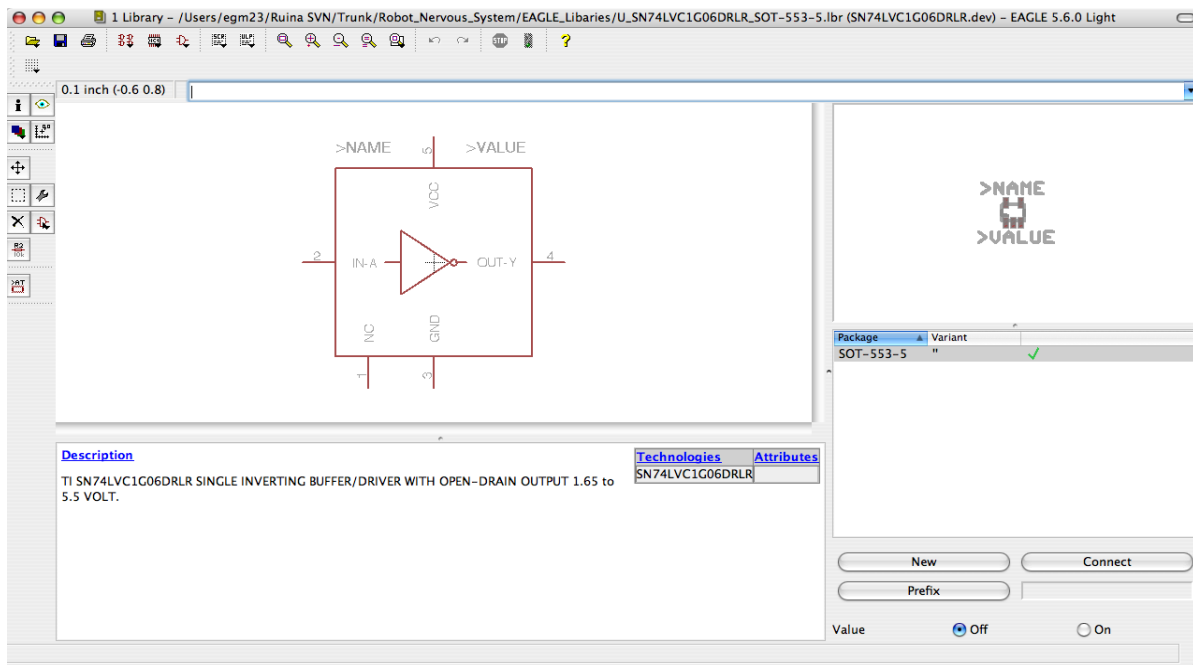
Picture 3: Example of a Library Part Entry

Creating a Layout

1. *Starting a Layout:* To switch from board to layout, go to File → Switch to Board and create a new board. Please carefully read and the layout specifications in the table below. In this lab, we only design 4 -layer boards because it keeps manufacturing costs down. The table at the end of this section, compiled by Jason Cortell, provides specific criteria for board sizing, appropriate layer use, and running the Design Rules Check.

2. *Layout Tips and Tricks:*

A. *Designating Board Area:* Carefully following the specifications in the table, set your board area. All components must be placed within this area.

B. *Planning Out Board:* Before laying down any traces, arrange components within the area. There will be yellow lines called "airwires" between components indicating necessary connections. Try to arrange components such that the traces do not have to be very complex between components.

C. *Using Supply planes:* Certain layers are primarily for use as either ground or power supply planes. Occasionally it will be necessary to create power polygons on other layers of the board. There is a polygon drawing tool.

D. *Viewing layers:* The display button allows you to choose which layers you want to view. Also, when you are putting down traces, the top left tool bar indicates which layer you are drawing traces on. There is a drop-down menu to change layers.

E. *Using Vias:* Vias allow you to connect traces on different layers as well as connect to the ground and power planes. General rule in the lab is that the fewer vias on the board the better because vias increase the cost of the board.

F. *Removing traces:* Use the "rip-up" tool to remove unwanted traces.

G. *Running DRC check:* You must run DRC with the setting specified in the above table. This is crucial to ensure the board will be functioning. See section on final error checks for all checks that must be run before the board is completed.
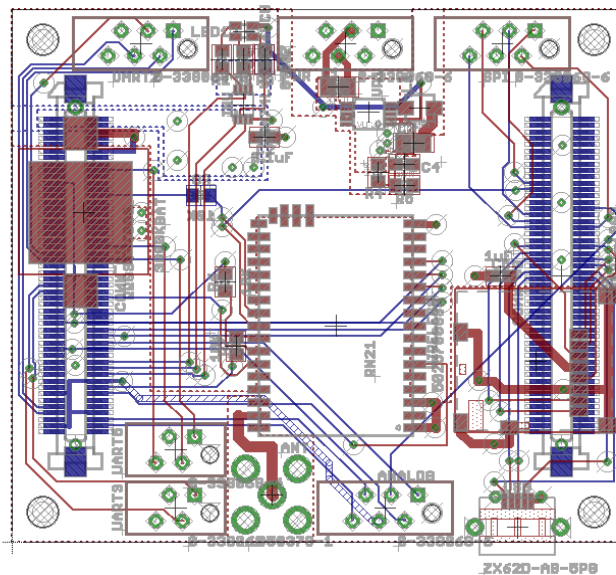
Picture 4: Example of a Finished Layout

Criteria for All Electronics Boards Designed for the Lab:		
Parameter	**Value**	**Comments**
Height (Y direction)	30 mm (1.182 inches)	Modules with Connectors
Height (Y direction)	18 mm (0.787 inches)	Modules without Connectors
Length (X direction)	Multiple of 1 mm (0.0394 in)	As short as is feasible
Mounting Holes		
Number of Holes	2	
Vertical Position	Centered	
Horizontal Positions	>=3mm (0.118 in) from ends,	It may be useful on some boards to

	in 1 mm (0.0394) increments	have a vertically oriented connector on the end of the board with the mounting screw located closer to the board center.
Hole Diameter	3 mm (0.118 in)	For 3mm or #4 screw
Board Layers – 4-Layer Boards		All modules need to be 4-layer boards
First (top) copper layer	EAGLE Layer 1	For internal module wiring and power polygons (planes) when possible. Use mostly vertical traces. Try to minimize use of horizontal traces
Second (internal) copper layer	EAGLE Layer 2	Primarily for module to module interface wiring. Leave plenty of space for future module wiring. All traces should be horizontal. NO Vertical traces allowed
Third (internal) copper layer	EAGLE Layer 15	Primarily for ground planes. Should not be used for other connections, but may be separated into analog and digital ground planes. A LAST RESORT for unexpected connections. Try to avoid damaging the ground plane continuity.
Fourth (bottom) copper layer		Primarily for power supply planes, with some traces and components as needed. Traces should be mostly vertical – Try to avoid horizontal traces.
Additional EAGLE Layers		
Pads	EAGLE layer 17	
Vias	EAGLE layer 18	
Unrouted	EAGLE layer 19	Should be empty when you are done with your layout!
Dimension	EAGLE layer 20	For module board outline only. Use width of 0 to avoid ambiguity. Include in silkscreen gerber files, though they probably won't appear on the board.
tPlace	EAGLE layer 21	Default location for top-side component placement lines. EAGLE library parts have 5 mil line widths. These are too narrow for board houses

		so they all need to be increased to 8 mil.
bPlace	EAGLE layer 22	Default location for bottom-side component placement graphics. See tPlace.
tOrigins	EAGLE layer 23	Component origins, top layer. Select components with these. Turn off the layer if you don't want to select them accidentally.
bOrigins	EAGLE layer 24	Component origins, bottom layer. Select components with these. Turn off the layer if you don't want to select them accidentally.
tNames	EAGLE layer 25	Component names, top layer. These will appear in the silkscreen graphics. All text must have an 8 mil line width. To achieve this, use 0.04 in height, vector font, and 20% width/height ratio
bNames	EAGLE layer 26	Component names, bottom layer. These will appear in the silkscreen graphics. All text must have an 8 mil line width. To achieve this, use 0.04 in height, vector font, and 20% width/heigh ratio.
tValues	EAGLE layer 27	Component values, top layer. Do not include in silkscreen gerber file.
bValues	EAGLE layer 28	Component values, bottom layer, Do not include in silkscreen gerber file.
tStop	EAGLE layer 29	For generation of top soler mask gerber files
bStop	EAGLE layer 30	For generation of bottom solder mask gerber files.
tCream	EAGLE layer 31	For generation of top solder paste stencil file
bCream	EAGLE layer 32	For generation of bottom solder paste stencil file.
tGlue	EAGLE layer 35	For generation of top glue stencil file (holds components down temporarily for soldering). We probably won't use this.

bGlue	EAGLE layer 36	For generation of bottom glue stencil file (holds components down temporarily for soldering). We probably will not use this
tKeepout	EAGLE layer 39	Shows component space required for ease of installation, top side
bKeepout	EAGLE layer 40	Show component space required for ease of installation, bottom side.
Drills	EAGLE layer 44	Shows drill locations for vias and pads (plated holes). Don't include in gerber silkscreen file, but do include in gerber/ Excellon drill file.
Holes	EAGLE layer 45	Shows drill locations for unplated holes. Don't include in gerber silkscreen file, but do include in gerber/ Excellon drill file.
tDocu	EAGLE layer 51	Shows component icons on top layer, to make it easier to understand the layout. Don't include in silkscreen.
bDocu	EAGLE layer 52	Shows component icons on bottom layer to make it easier to understand the layout. Don't include in silkscreen
Layout Design Rules for 4-Layer Boards		These rules are intended to help make the various modules compatible and to keep fabrication costs down.
Minimum trace width	7 mils (0.178 mm)	This and the space width below are a little larger than the minimum for 1-oz thickness copper, but we would like to be able to use the same module designs with motor controllers and other power circuits on 2-oz copper.
Minimum space between copper objects	7 mils (0.178mm)	See above.
Minimum hole size	15 mils (0.381mm)	Vias, pads, holes, etc.
Minimum annular ring - vias	5 mils (0.127 mm)	Radial distance. EAGLE calls this "restring." The copper shape around a via should be at least 10 mils larger in diameter than the hole itself.

Minimum annular ring - pads	7 mils (0.178 mm)	Radial distance. EAGLE calls this "restring." The copper pad around a component pin hole should be at least 14 mils larger in diameter than the hole itself.
Minimum copper to PCB edge distance	20 mils (0.508 mm)	Distance from board edge to closest traces.
Minimum inner layer clearance	10 mils (0.254 mm)	Distance from board edge to closest traces.
Solder mask clearance	3 mils (0.076 mm)	Distance from edge of pad to start of solder mask.
Silkscreen line width	8 mils (0.203 mm)	Minimum line width for text, part placement graphics, etc.
Font	EAGLE vector font	This font allows user-selected thickness/height ratio; the proportional one does not.
Character thickness/height ratio	20% EAGLE "ratio" parameter	Gives 8-mil line thickness with 40-mil character height
Minimum text height	40 mils (1.016 mm)	
EAGLE DRC (Design Rules Check) Setup for 4-Layer Boards	Do a DRC check by clicking on an icon showing a magnifying glass full of traces and pads.	Note that many of the DRC settings go beyond error checks and actually change your board layout.
Layers	(1+2*15+16)	Setup
Clearance	7mil (all)	
Distance	20mil	Copper/dimension
	8mil	Drill/hole
Sizes	7mil	Minimum width
	15mil	Minimum drill
Restring	7mil	Minimum pads
	5 mil	Minimum via
Masks	3mil	Solder Mask Swell, Min
	10%	% of mind. SMD Dimension
	4mil	Solder Mask Swell, Max
	0mil	Cream
Misc.	Angle Check	On

Copying a Module to a Board

The boards are meant to be combinations of difference modules in order to complete a certain function. This section provides step-by-step instructions for successfully copying a module to the board compiled by Jason Cortell:

1. Check that all your component and net names are unique between the two boards. Each module/unit should have a unique prefix number. For example, the LPC2194-CAn unit has component numbers starting at 100 and net names with a 1_prefix. Don't forget to check the ground and power symbols on the schematic, and even the page frames. They have names too.

2. Make backup copies of your files. This process can completely ruin your layout and schematic if something goes wrong.

3. Open the source board file.

4. Turn on all layers, then use the selection tool to select the whole board.

5. Use the cut (scissors) tool to copy the board layout to the paste buffer.

6. From the same instance of EAGLE, open the destination board file. DO NOT open the schematic file. Turn on all layers.

7. Use the paste (paintbrush) tool to copy in the layout.

8. Save and close the board file.

9. Open the source schematic. Starting with the first schematic sheet, use the selection tool and select the entire schematic, including the page frame.

10. Use the cut (scissors) tool to copy the schematic to the paste buffer.

11. Open the destination schematic. Ignore the warning that the schematic and board are not consistent. DO NOT do anything else – e.g. ERC, RATSNEST

12. Create a new schematic sheet and paste in the module schematic. Save the file

13. Repeat the previous four steps for each additional schematic sheet in the source file

14. Now run the ERC check. This hopefully will show you that the operation was successful. It also reestablishes the forward-back annotation so that you can start editing the combined files.

Finishing Your Module or Board

1. *Run the DRC Check:* Are all layers on? Stop mask errors are generally due to silkscreen printing in areas with no stop mask. Make sure there aren't any when you turn off the silkscreen layers (tPlace, tNames, bPlace, bNames, Dimension). Drill errors tend to indicate that there are two vias in the same place.

2. *Run the ERC Check:* Watch out for connections that look real but aren't. You may need add junctions here and there. Since we are building modules intended for connection to other modules, you can approve the various errors related to pins with only one junction.

3. *Check your library parts:* Even devices that were in already in an EAGLE library can have errors. Some possible problems:

> a. Do the pins physically line up with the copper areas (lands). The lands should not be too small in any area.

> b. Is the pin number of the pads/SMDs the same as the pin numbering shown on the data sheet. Use the info tool to see the numbering.

> c. Check the connections against the data sheet one more time.

4. *Update Parts:* Update your library parts from your module library

5. *Name all Parts:* Make sure that all components on the schematic and layout are named.

6. *Rerun all Checks:* This includes ratsnets, which will show any remaining airwire, DRC and ERC. This is in case anything went wrong in the previous checks.

7. *Double Check:* Export gerber files and upload them to www.freedfm.com for an additional check. If any repairs are needed, you'll have to run the ratsnest, ERC, DRC, again.