Neel's Guide to PCB Design

8/15

If you have any questions, please email najjarapu16@lawrenceville.org (before May 2016), otherwise text 609651911948 or Facebook me or something.

Make sure you have my resources folder. It has many files useful files that you'll want to use.

Please use the book *Complete PCB Design Using OrCAD Capture and PCB Editor* by Kraig Mitzner as a supplement to this manual – I reference many of its pages.

Please excuse the grammar.
There are two sample boards that are fully loaded in terms of layers, symbol placement, spacing constraints, netlist, etc. You just need to route them, add the central cavity, add via arrays, add filters if necessary and generate artwork, but most of the initial setup is complete

If you just want to change the center cavity dimensions of an existing board, go to the very end.

Getting Access

1. Enable UNIX access for your netID: http://kb.princeton.edu/5216
2. Register to use the 'Nobel' UNIX cluster: http://www.princeton.edu/researchcomputing/computational-hardware/nobel/
3. SSH into the Nobel Server
   a. If you are using Linux
      i. Type *ssh –X yourNETID@nobel.princeton.edu* and login with your password
      ii. Read and do page 2 of the environment setup document in my folder under Reading Material
   b. If you are using windows, log in via Putty and Xming (not covered here)

Creating a Project

If all you have is a .brd file that you want to modify but none of these directories are setup, then it's a good idea to make a project, even if you aren't starting from scratch.

Alternatively, you can copy and paste my project folder into your directory and go from there – it's quite messy but it's already setup. Project name is

1. Open Project Manager
2. Type projmgr into the command line
3. Select "Allegro PCB Designer (Schematic)" and the "High Speed" product option and press OK. A graphic "Create or Open Project" window should open.
4. If you're opening and project, just select Open Project on the Create or Open Project Menu, select your file, and skip this part.
5. Create a folder in your H: Drive for your project files
   a. Ex. test_project in homeserver2/user2a/ajjarapu/
6. Click on the "Create Design Project" circle in the window.
7. Enter your project (all lowercase) and location, then click next
   a. Ex. Project Name: testproject   Location: /n/homeserver2/user2a/ajjarapu/test_project
8. Add project libraries – the only necessary one is "testproject_lib", then click next
9. Select "testproject_lib" and type in a design name
   a. Ex. testdesign
10. Click next and finish. In your "test_project" folder, you should see a "temp" folder, a "worklib" folder, a "cds.lib" file and a "testproject.cpm" file. In temp, you'll just see some "cfg… .log" files. In "worklib", you'll find your "testdesign" folder and in that shoulde be some "cfg…" folders and a "physical" folder. "physical" will be where all the board files are stored.

Files and Stuff

Copy everything from my "Symbol Library" folder and "Board Files" and paste the contents, not the folder, into the Physical folder. These are the files for components of the pcb.

Editing the Netlist

The netlist is a text file that describes the components of your board and how they are connected to each other. A "net" is common set of connections between components. Generally, the "microwave" net is used to connect the SSMA pins - they're not connected in the finished in the finished project but the connections make it much easier for board design. The SSMA's ground pins and the screw holes are connected via the ground plane.

I have a netlist my folder that already contains 12 SSMA symbols, 12 filter symbols and 16 screw holes. If you don't need more components than that, just read this section for its information (if you need less stuff, use my netlist and just don't place all the components on the actual board)

1. Open the basic netlist file. Here is a breakdown of what you will see
    a. "HOLE090P" - this is the package (aka symbol), or set of padstacks, that define the screw holes. This package only has one padstack, where the screw goes.
    b. "SSMA_BB" - this is the package that defines the SSMA connector. This package will have 3 labeled padstacks, the first is the signal pin, and the other two are ground pins. There should also be another, smaller, set of ground pins and a set of vias in the package, but they are not labeled
    c. "FILTER" – the package for the cavity that the filters fit to (a 6mm x 6mm cavity)
    d. "MTG…" - this is the reference designator (refdes) of the screw holes. Every package has a unique refdes (i.e. MTG1 MTG2 MTG3 etc.). The decimal value references a labled padstack (
        1. Ex. MTG1 is the first screw hole package
        2. MTG1.1 references the hole itself
    e. "J…" - likewise, the SSMA connectors have a refdes. J1 J2 etc. reference the entire SSMA packages. A ".1" references the center signal pin in the package and ".2" and ".3" reference the ground pins
        1. Ex. J.1 is the 1st SSMA_BB component
        2. J1.1 is the signal pin of the component
        3. J1.2 and J1.3 are the gnd pins

    f. "R…" The refdes for each added filter
    g. MICROWAVE - the net that connects all the signal pins of the SSMA connectors
    h. GND - the net that connects all grounded components
    i. THERMAL_CON_TYPE and DYN_CLEARANCE_TYPE - other properties that determine the thermal connectivity of the padstacks.

2. To add components, just add the test referencing ever package and pin. If there are J*N* packages, then add a J*N+1* and J*N+1*.1 , J*N+1*.2 , J*N+1*.3 to every list that contains them. Make sure to maintain the spacing
   a. Ex. If there are 6 J packages, add a J7 to SSMA_BB ! SSMA_BB ; list. Add a J7.2 and J7.3 to the $NETSGND list and so on.
3. To add a completely new symbol to the netlist
   a. Retrieve its name from the device file (it should look like [symbolname].txt). If you don't have this file, but you have the .dra file for the symbol, open in it and go to File > Create Device… While you're at it, if you don't have a .psm file for the board go to File > Create Symbol.
   b. Look at the format of the other symbols on the netlist
      i. [symbolname] ! [symbolname] ; N1 N2 N3…   Here you are creating an N number symbols that are allowed to be placed on the netlist. N1 N2 etc. are the refdes's of the symbol so make sure they match whatever the symbol has (if the .dra file shows N*, then you label the symbols N1 N2 etc.)
      ii. Add the pins of the symbol to whatever net they belong to. Ex. If N has 2 pins, the first of which goes to Microwave and the other to GND, then in the microwave list add N1.1 N2.1 N3.1 … and to GND add N1.2 N2.2 …
4. To remove components, just delete the refdes of number of packages that you don't want. Don't do this unless you really need to… you can choose not to use parts in the board design later
5. After all the text is added, wherever a list goes beyond 77 characters per line, put a line break at the last word before the 77th character and add a comma after that last word (see a previous netlist for an example)
6. Once you are done, save your netlist - I usually put in physical but it shouldn't matter - and exit.

Creating a Board

If you don't know all the parameters listed below, put in the recommended numbers. They can always be changed later.

If you have a board you're just editing, you can just read this.

1. Select "Layout" on the Board Design menu. You should come to a large window titled "Allegro PCB Designer (was Performance L): unnamed.brd Project: …/testdesgin/physical"
   a. A .brd file is a board file. It acts as a database for every part of the board. A single board file corresponds to a single board design
2. Go to file-> new. It opens a "New Drawing" window.
3. Enter the Drawing name and select Board Wizard and click OK
4. Go though the board wizard and select no and Next through Template, Tech and Parameter files (we will import some but we can't here), board symbol

5. In General Parameters select Units - Mils and Size - D. Set the origin whenever you'd like (I like center). Click Next
6. Select desired grid spacing (1.00 is fine) and number of layers - generally 4. Select "generate default artwork films" and continue
7. Enter your layer names and Types
   a. Set layer names to: Top, 02_GND, 03_SIG, and Bottom. All should be set to routing except 02_GND, which is a "power plane".
8. Define all minimums - I generally do 5 mils
9. To select the "default via padstack", press the " . " button to browse and select via10 and continue
   a. If you do not have via10, you did not put the "libraries" folder contents in the "physical" folder
10. Define your board outline and continue
11. Set your dimensions and keepin offset
    a. Anywhere from 0 to 20 should be okay for the keepin. The keepin defines the area in which packages and routes are allowed to be placed. If the distance is too large, then only a small part of the board will be useable
12. Click Finish. A rectangle or circle board outline should appear on your screen.

Board Setup
If you have a board, just make to do steps 1 and 2. The netlist and techfiles are already in the board

1. Go to Setup > Design Parameters. Under "Enhanced display modes" select display plated holes, thermal pads, and via labels. Leave the previously selected ones on as well. Click apply and ok to exit.
2. Go to Setup> Layer Cross Section. Make sure that all layer dimensions are correct according to your specifications
   a. TOP 03_SIG and BOTTOM should be Type: conductor and have no negative artwork
   b. GND should be "plane" with negative artwork and shielded
3. Go to file > import > Logic and to the Other page
   a. In the Import Netlist bar, select the text file you want o import
   b. If you just created/ edited the netlist, first do a syntax check – click the syntax check only box and then "Import Other". This will tell you if there are any errors – a common error is not starting a new line after 77 characters. If your list is running past 77 characters, start a new line at the last word before the 77th character – remember to add a comma after the last word of the line if you have a line break (see a current Netlist to see what I mean)
   c. f there are no errors, uncheck syntax only and select all the boxes below it – then import "Other"
   d. This will update or create a netlist from which you can now place symbols that have predefine connections to each other.

Navigating the Software
Reading these sections will give you help you figure out Cadence Allegro's basic tools

1. Please read the following sections in the book
2. pp 22-23 The PCB EDITOR WINDOW
3. Pp 23 Controlling the View
4. Pp 38 - 50 all - you'll probably use most of these tools at some point, so at lease skim through
5. Some important skills are
   a. using the small p on the bottom bar to precisely place object
      i. You can use this whenever you are asked to place an object, corner, shape, etc.
      ii. XY Coordinates are entered as : (X,Y)
      iii. Distance + Angle is entered as: D A   (just a space between the two numbers)
   b. Using the color menu
   c. Using the options pane
   d. Right clicks lets you undo an ongoing action with "oops", cancel with "cancel", and finish with "done"

Pouring Copper Planes
Every layer of this pcb will have a copper plane on it, which you need to add manually. Skip this if you have a board

Pouring Copper Planes Using Shape Add Tool (pp 284)
1. Select a shape (rectangle or circle) and go to the options menu. Select Etch, and start at top. Fill: Dynamic copper. Assign net: GND for all. You can place the shape in the options menu, manually or precisely.
   a. If the shape is smooth and transparent, then you're good. If it looks like mesh, then you didn't put in dynamic copper. Either redo the shape or go to shape > Change Shape Type and select the shape.

2. Planning. Add shapes in the plan layer to lay out some important aspect of the board. Visibility of key areas makes it easier to place objects

3. Add planning rectangles along the outer edges of the cavities
   a. Add > Rectangle. Select "Plan" for the top option. Select all for the second (since they are only guide lines, the exact layer doesn't matter)
   b. The outer dimensions I make are the size of the chip. The inner dimensions are half a mm less on every size (- 20 mils) . You should have two concentric rectangles or squares after you place both of them
4. Add a "plan" shape to show where the copper shielding will be placed so that you don't add an SSMA connector there by accident. You can use a circle or square depending on which shield you are using. The exact dimensions of the shields, + the orientation of the screw holes can be found in the CAD files of the resources folder.

Placing Objects

Manually Placing From Netlist

Before, we loaded up a netlist that contained all the components that we are using and which ones were connected to each other. Now we can manually place every component on the board – the program will automatically assign refdes/pin numbers and will display the virtual connections between the pins

Go to Place > Manually. In advanced settings, select database and library

1. Go to placement list select "components by refdes". Underneath should be the refdes's defined in the netlist.
2. Select the box next to the symbol/package you want to place. Do not press OK. The symbol will follow your mouse until you place it manually or precisely.
   a. If you are placing the SSMA symbols, go to the options plane and select mirror - the actual pcb boards have the connections coming out of the "Bottom" layer, so the padstacks needs to be facing down. You will know if its mirrored if the refdes is mirrored
   b. Add any necessary rotation in the option pane while placing
   c. The components are placed by their origin and will be centered wherever you decide to place it.
3. After a component is placed, it will have a check mark next to it and it's icon will turn green. You can now check another component to place it, or press OK on the menu to exit.

Importing Placement Files

Placement files save the placement and orientation of all the netlist symbols from a previous board. It is very useful if you want to start a new board and make slight modifications to the placement but want the majority of it the same. It will not transfer vias/via arrays that are not part of the netlist.

1. If you have a placement file from a previous board, you can go to File > Import > Placement and select the file.
   a. If you don't have the file but can to you use another board's placement, open that board and go to File > Export > Placement and safe the file. Then import the file in you board
2. Make sure that your netlist has as many or more components as the number that you are importing – if your netlist doesn't contain enough, the file won't load properly.
3. The placement file may not put the components exactly in the center of your board. In this case
   a. Select a single padstack and right click - select Show Element. This should tell you exactly which coordinates the padstack is located at. Remember this coordinate.
   b. Go back to the original file where the placement was imported from and record the same padstack's coordinates that same way

   c. Go back to your new board and mass select all the components of new board by either shift clicking every component or dragging a selection window over it. If you have a board outline and other components already loaded, be careful not to select them, or turn them off in the color dialogue before your mass selection. If this is too difficult, try the alternative method in #4

   d. After selecting all the placed parts, zoom into the original padstack whose coordinates you recorded until your mouse is exactly (w/in 1 mil) in the center of the padstack. You will know because the coordinates on the bottom right will match the coordinates you saw earlier.

   e. Right click and select move - this this move all the components you selected earlier but the defining "center" of the movement will be at the padstack you selected

   f. Enter the original coordinates of the padstack - this moves all the components relative to the padstack so they all end up where they were in the old board.

4. Alternatively, when you load the placement files, everything you loaded will be highlighted/selected. Before you load the placement file, save the board design. Then import the placement file and see the location of the padstack you want and record its original placement on the original board (see a, b, c above). Now go to File > Recent and select your board. DO NOT SAVE. This will open the pre-placement file board. Import the placement file again so all the components are highlighted. Go to the padstack whose locations you know and zoom in on it until your mouse is exactly at the center. Then right click and select Move – then precisely place it in its original location. All other parts will be moved relative to this repositioning

Rotating Parts (check Rotate in the book's index, there are many examples to see)
Even after a component is placed, it can be rotated around itself or another point. If you have multiple parts rotating at around the same point in a circle (not necessarily equally spaced) it is easier to place one component and copy and paste at given angles – which is explained after this section

1. To rotate a symbol, right click on any padstack or shape in it, go to symbol and press move. In the options pane, select the degree and rotation type

   a. absolute moves it exactly the number of degrees specified ex. 90 would be a 90 degree rotation

   b. incremental lets you choose between every multiple of the chose degrees => 90 degrees allows you to rotate 90, 180, or 270

2. Right click on the object again (it should be attached to the mouse) and select rotate. Then move the mouse to point it in the right direction. Click again to finish rotation and right click and select done.

Polar Copy and Place

1. If the placement follows a circle, place the first component. After that, right click the component, navigate to symbol and select copy. Then select the copy type (polar lets you place them round a given radius)
2. Choose the angle relative to the first placed object
3. Choose the origin (to place on a board, you'll usually precisely pick 0,0)
4. Move the mouse around and the copied symbol will follow at increments of the selected angle.
5. Click to place – place all of the components you need to before assigning a refdes (below)
6. Alternatively, you can use polar coordinates and rotation to manually place every component at a given radius

Assigning RefDes
If you place a symbol from the manual placement method, it should have a refdes assigned to it. If a symbol (usually from copying) does not have a refdes number and has a * (ex MTG*), use this method

1. Go to logic > assign refdes. Click on the period select a refdes. If you increment 1, then as you click around the circle of unlabeled symbols you'll given them sequential refdes's

Moving a Component
1. Click on some part of the symbol (refdes, outline, padstack). Right click and go to Symbol > move.
2. The symbol will now follow your mouse and can be placed manually
3. To do this more precisely, right click and select Symbol, > Show Element and see the symbol's origin. Zoom into the component until the mouse is at this coordinate and right click and select Move. From here, you can use the precise placement tool to move the component exactly where you want. If this is too annoying, just unplace the component and place it again.

Unplace a Component
1. Click on a part of the symbol
2. Right click and go to symbol > unplace
3. It will disappear and you manually place it again or assign it's refdes to another symbol

Determining Impedance
Keeping a constant impedance of 50 ohms throughout the board is important to prevent reflections.
1. In Simon's Coplanar Waveguide Circuits, Components, and Systems page 13 Figure 2.1 a) there is a diagram of a "sandwiched" coplanar waveguide.
2. Open my MATLAB impedance calculator. Knowing your dielectric heights and dielectric constants, experiment with the calculator to find a centerpin width and slot width that fit your needs.

3. Simon's diagram has h3 and h4 which account for airspace – I'd set these values to h1 and h2 respectively since there is no air between the copper plated and dielectrics
4. With the current dielectric setup (see previous board designs) I use a centerpin width of 19mils and slot width of 32.5 mils.
5. All centerpin and sloth width values can be set up in the constrain manager below.

<div align="center">

Constraint Manager
This sets up rules that the program obeys when creating and placing components such as minimum sizes, spacing, etc.

</div>

Creating Constraints
1. To give your component constant spacing go to Setup>Constraints>Constraint Manager.
   a. The constraint manager likes to go to the back of the windows, so right click and put it as always on top while you use it.
   b. If you plan to use the same CPW dimensions as another board, go to file > import > constraints. If that doesn't work (sometimes it crashes then just manually copy the changes values)
2. To set a distance between route and etch (useful for CPW) go to Spacing > Net > All Layers > Line. In the Microwave net, under Line to Shape, set your desired width.
   a. Width is the space from the outer edge of trace to inner edge of etch. Using MATLAB, enter in the dielectric heights and the rest of your variables (it's the first model in Simons) and figure out trace width. I currently use 19 mils
3. To set a space between the signal pin of the SSMA connector and the GND plane etch, go to Spacing > Net > All Layers > Shape and set the shape to thru pin microwave distance - I normally select 32.5 mils total which gives me 6.75 mil spacing
4. You can set any other constraints from the CM as well.

Importing Constraints
1. In the Constraint Manager go to file > import
2. Select the constraint file you want to import. The resources folder has once that was used on all previous boards (19 mil trace width, 6.75 mil trace to shape space, 30 something thru pin to shape spacing)

<div align="center">

Routing
Routes are the copper paths that connect the pins. In order for the board to work well, it's best to connect all the pins via routes (see previous board designs) and creating the cavity only during manufacturing. If you are routing a board with filters, see the Route Lengths section first

</div>

Straight Routes
1. Before routing, select Display > Show Rats > All. This show the connections for every net between components.
2. To create a route select route > connect.
3. Select the options plane and create/adjust your dimensions, angle, etc.
4. Click on the Padstack you want to connect and click on where you want it to end

      a. Click on other points as the route approaches the second padstack so the route follows a certain path
5. Right click done
6. If you need to vary the slot width, select the shape add button (6 sided polygon next to the rectangle) and in the options plane select Route Keepout on the top and the desired layer (03_SIG) on the bottom

Arc Routes
Arcs have gentler turns and (hopefully) are preferable to smaller turns
      1. After route>connect, in the options pane set the line type from line to arc.

Route Lengths
If you are using filters, the distance between the edge of the inner cavity of the filter and edge of the inner cavity of the center chip must be equal for all boards (to prevent standing waves of variable frequencies)
Before everything is routed properly, you'll need to do some "test routing" to determine lengths. Alternatively you can use lots of trig.

1. Ideally, all the filters are directly angled towards the center – you can do this by rotating when copying or by doing a fully ration. Know the angle from the center is very helpful.
2. Using trig, find the point where the inner cavity intersect with the route path to the filter. Read the Straight Route section above.
3. Rather than selecting a padstack to start the route, type in the intersection point and use the filter center as the second point.
4. Do this for all the filters.
5. Right click and show element on the route to see the route length.
6. Since the cavity is a square, if you have a symmetrical board, there will be a slight discrepancy depending on the filter's orientation and closeness to the squares corners.
7. Decide on a good distance.
8. Move the filters closer so that all thee distances match.
9. Then properly route the boards.

<div align="center">

Via Arrays
Once the routes have been laid out and most of the board is complete, we can fill the rest of the area in with vias
You cannot undo via arrays, only remove them afterwards. Sometimes it's easier to save the board before any arrays are added and if a mistake is made, open the previously saved version.
Useful Link: https://youtu.be/Ea3z9gbzxF0?list=PLCgZ9VoSvNf02B7EsSjkmrZWW3N266z7s

</div>

Placing Arrays
1. To create vias along the route select Place > Via Arrays > Boundary.
      a. In the options pane select your via - via10 - and the ground net. Under Global Ring Parameters select "on both sides of " and 1 ring.

b. Set object ring parameters for non-circular. The via-object distance should be greater than 15 mils and via-via distance depends on experimentation but I prefer 50 horizontal and vertical.

c. After selecting the right options, click on the route to see a preview of the via structure.

d. If the route looks bad, right click, select "oops" and adjust settings. If it looks good, click anywhere on the board to confirm the placement.

2. To create vias in the rest of the area select Place > Via Arrays > Matrix.

a. Select the operation mode you want. Board Mode covered the entire board, including packages in a via matrix, area mode lets you define a rectangle and shape mode lets you populate a pre-drawn shape. If the board has a large cavity and only a few places that need to be via-free (the packages, routes etc.) select board or shape (with the etch being the shape). Otherwise, select shape mode so you can put via matrices in individual areas. Note: the board matrices look a lot more uniform.

Remove Arrays

1. To remove/clean up the via arrays, select place > via arrays > unplace. In the options menu select "Enable via selection mode". Now drag a box over the vias you want to remove or click on them individually. When they are highlighted, click on any part of the screen to confirm their deletion.

a. This tool will only delete vias placed using via arrays, so don't worry about accidentally deleting padstacks from your symbols

b. Make sure that none of the vias are on the package, route, cavity or any important component of the board and that they all remain within the board outline.

c. If you are designing a board with multiple cavity sizes, first remove the vias in the smallest cavity area and save it as a new file. Then remove the larger ones so you don't have to add in an array again.

<div align="center">

Padstacks

</div>

Replacing Padstacks (pp 214)

1. Click on Tools > Padstack > Replace Padstack. The options bar will give you a box asking for the current via and the new via to replace it

2. At the top of the option pane, select whether you want to replace all vias of a given type or just the selected ones.

3. Click in the Current Via box and then click on the via you wish to replace.

4. Click the New Via box and either press the period to select a via from the database – if you don't see anything, go to the second page and select both database and library

5. Once the current and new vias are chosen, press okay to replace

Changing Via Designs

If you have a via you've been using and want to fundamentally change its design

Useful Link: https://youtu.be/5AV0dsDnlPI?list=PLCgZ9VoSvNf02B7EsSjkmrZWW3N266z7s

1. Go to Tools > Padstack > Modify Design Padstack (if it's on the board) or Library padstack if it's not.
2. Click on a padstack in your board / select from the library menu and from here you can edit the contents of the padstack.
3. Read the next section to learn what it means to edit the padstack.

Creating a Padstack (pp 206)
1. You can access the padstack designer from either Tools > Padstack > Padstack Designer or in the board creator menu of the project manager (somewhere in the tools section). Refer to the book to learn how to create padstacks.
2. Some things to note
   a. Make sure you're hole type is correct – if you are designing a board cavity you'll need a rectangular slot versus if you need a via, it will be a circle
   b. The drill diameter is the size of the hole. It will be constant throughout the via so you can't create stadium cavities (you can if you create two blind/buried vias on top of each other but its easier just to request the stadium cavity in the fab notes. If you need to do a stadium, use the padstack to create the smallest hole that goes through all the boards and request that they expose the signal layer with a hole with X by X dimensions.)
   c. Plating – this plates the entire inside of the hole with copper and so connects all the copper layers. You will want this for vias and GND connections, but not for anything that connect to the important traces on layer 3.
   d. The second page will have properties per layer. Pads are the copper layers circling the hole (see the gold on pg 206) and thermal reliefs are "vents" that disperse heat during soldering (see 219). Make sure the pads are large enough to solder to… if you are creating a cavity, you won't want pads, so set them smaller than the drill hole and no pads will be made. For your thermal relief flash symbol, use Th-64.

Creating a Symbol/Package (pp 208)
A symbol/package/footprint/component – all these names are interchangeable for your purposes. These are a collection of padstack that have a special purpose, such as being a filter cavity or SSMA connection.

1. To create a symbol, in the board editor go to File > New > Package. Don't use the wizard.
2. You will come to the Symbol editor
3. Go to the color dialogue and turn on Global Visibility
4. Make sure you've already created the necessary padstacks to create the symbol
5. Go to Layout > Pins. In the options pane, select the padstack that you want to add, the copy pacing (if you are planning to add many of the same padstack in a pattern) and the pin number. Every pin needs a pin number since different pins on the same symbol can connect to different nets (e.g. the SSMA symbol pin 1 connects to the Microwave Net

while pins 2 and 3 connect to ground). Vias on a symbol don't need pin numbers since nothing is connecting to them.

6. Place the pins in whatever shape you need – if you only are using one padstack, precisely place it on the origin
7. Go to Add > Rectangle and in the option menu select the Package Geometry class and subclass Assembly Top. Use this to draw an outline of your symbol. If you have a rectangular padstack, put the Assembly Top Square exactly on the outline. This shape defines the extent of the symbol
8. Add another rectangle of the same class and subclass Place_Bound_Top. Put it exactly over your Assembly Top rectangle. This shape defines the extent of the symbol to the constraint manager.
9. Add anything else to your symbol – more vias, a silkscreen text, etc.
10. Once the symbol is complete, go to Layout > Labels > Refdes. Then click somewhere on the symbol to add its Reference Designator. It's like a pin number but for symbols. It will end up looking like [refdes]* where the * will have an assigned number when the symbol is on the board.
11. Save the symbol. (See Creating a Netlist at the top)
12. Go to File > Create Symbol. This will create a .psm file that the board uses to reference the symbol along with the symbols .dra file. Save this file in your physical folder
13. Go to File > Create Device… this will create a Device File (.txt) that the netlist will use to create its connections. It will probably be a discrete symbol.
14. Use this file to add the symbol to the netlist.

Looking over the Board
The board is pretty much done at this point, so now just check it over and make sure that everything looks right. After this we add dimensions, notes and generate artwork.

GND planes connected to pins.

> On the color dialog, turn global visibility off and then select the etch for the top layer. Right click on it and select show element. If at the top says < SHAPE<auto-generated> >, class ETCH, subclass TOP (or whatever layer you're on), part of net name: GND, you're good. If not, remake the shape. Do this for all 4 layers
> Turn off global visibility and turn on all the parts of 02_GND. If you look at the large ground pins, you should see the 4 unconnected arcs around each pin (like the bottom picture of pp 286). These are thermal reliefs that prevent copper deformation when soldering
>> If you don't see them, check the design parameters to make sure thermal pads are visible in the enhanced display mode.

Dimensioning
Linear Dimensions
1. Add dimensions (esp. if you have a center cavity / something that isn't defined as a symbol/padstack)
2. Select Manufacture > Dimension Environment

3. Right slick and select "linear dimension"
4. Go to the color selector and set global visibility Off (so that we don't accidentally auto select the wrong component)
5. Use the precise pick coordinated and select the two points on the edge of the shape you are creating the dimension for.
6. Once both points are picked, the dimension text should appear with lines extending to the measured component
7. Once the text appears, if there are not enough sig figs, go to the options pane and override the dimensions in the text input box. You can add extra details here as well (ex. .392 Cutout 1-3).
8. Select a place on the screen to place the text.
9. Do this for any undefined but important areas on board and for the board outline
10. Right click and select next to place another dimension or click done

Drill Origin / Leader Line
1. Add Drill Origin.
2. To specify drill origin, select manufacture > dimension environment right click and select leader line.
3. Place the first point at 0,0 and second point wherever you want and a third point to create a leader line shape.
4. Right click and select done.
5. Go to Add > Text and click at the end of the leader line.
6. Enter 0,0 DRL

<div align="center">

Manufacturing Notes

</div>

Notes give the manufacturers clearer directions on how to make the board. I'll have some sample notes put up. They'll be split since adding a huge block of text makes the program crash. Select Class "board geometry" and subclass "assembly notes"

1. Go to Add > Text. In the options pane make the text size 6+ so its visible
2. Click on the location where you want to place text - a blinking gray bar will appear. Notes generally go on the right of the board
3. Right click and select "Read from File".
4. Navigate and select the text file you want to import
5. Wait for a while
6. When the mouse turns into a crosshair (instead of the loading bar), right click and select done and the process will finish. There is no other indication of it finishing but it can take 10+ minutes to fully load
7. Repeat for the next files

<div align="center">

Cross Section Legend

A graphic that shows the weights for every layer of the pcb

</div>

1. Go to tools > cross section and check that the height of every cross section is correct Then press ok
2. Go to manufacture > cross section - a pop up window will appear.

    a. Uncheck Material name (since we use FR408 not FR4 so don't accidentally confuse them)

    b. Make text size at least 6

    c. Make x scale 2 (for size)

    d. Click ok

3. A shape will follow your mouse - click to place. Generally its placed somewhere under the board

Add Photoplot outline

We need to create a board outline in the photoplot class so that the manufacturers can see it

1. Create a shape (same type as the board outline i.e circle rectangle etc.)
2. Before placing the shape go to the options place and select Class > Manufacture and subclass >Photoplot_Outline
3. Use the precise placement tool to put the shape exactly over your board outline

Generating Artwork

1. Select Manufacture > Artwork
2. Go to the general parameters page
    a. Make sure it looks like figure 10-8 on pp 407 of the book
3. Go to film control. If there are 7 folders listed (TOP, BOTTOM, 02_GND, 03_SIG, SOLDERMASK_TOP, SOLDERMASK_BOTTOM, PHOTOPLOT_OUTLINE or names that look like that) skip this step (#3). If a folder listed below is already under "Available films" skip that step
    a. 02_GND + 03_SIG : Select create missing files . Press OK
    b. Soldermask Top : Go to the color selector and turn global color OFF. Then go through every pane and select soldermask top and press ok. Go to manufacture> artwork and right click on an existing file and select add. Name the file "SOLDERMASK_TOP".
    c. Soldermask Bottom : Go to the color selector and turn global color OFF. Then go through every pane and select soldermask bottom and press ok. Go to manufacture> artwork and right click on an existing file and select add. Name the file "SOLDERMASK_Bottom".
    d. Photoplot Outline : Go to the color selector and turn global color OFF. Then go manufacturing and select photoplot_outline and select ok. Go to manufacture> artwork and right click on an existing file and select add. Name the file "photoplot_outline".
    e. To add a silkscreen refer t0 411 -413 (I don't use it at all...)
4. Click every file and on the right at undefined line width, enter 5
5. Aperture
    a. Go to aperture. The "Edit Aperture Wheels" pane will pop up. Select Edit
    b. In the new "Edit Aperture Stations" window select "Auto->" and without rotation.
    c. Then click ok until you're back to the artwork window

6. Click OK on the artwork window and go back to it through manufacture >artwork (otherwise your recent changes won't be applied)
7. Select "Create Artwork" - the files should be created with warnings. Ignore the warnings.
8. The files will appear in whatever folder the brd file is in. I recommend creating an artwork folder for that board (ex. Testboard_art) within that folder to hold the art and drl files. They will appear as 7 individual files names for every film (TOP, BOTTOM, SOLDERMASK_TOP, etc.)

Drill
Drill Info for the manufacturer
Drill Legend
    a. Manufacture > NC > Drill Legend.
    b. Click ok once the legend pane comes up (everything by default should be correct)
    c. Place the legend somewhere above the board
        i. The top two rows will look garbled, this is because there is a second part to the drill legend that the program overlays onto the larger one - you can see in previous files that there are two legends at the top
        ii. To fix this - hold shift and click on all the text from the top four rows as well as the lines separating. It's tricky but after clicking on all the elements the white outlines text should look like another legend but shorter (see previous files).
        iii. Right click and select copy and place this new legend above the larger one. Once placed, right click and select down
        iv. While the original elements are still highlighted, right click on them and select delete so the garbled text becomes clearer

Drill Files (.drl)
1. Go to Manufacture > NC > Drill Param and make sure it looks like 10-18 on pg 415 except make format 2:5
2. Go to Manufacture > NC > Drill and the page should open
    a. In root file name, make the file save in the artwork file you created earlier
    b. Select "Auto Tool Select, Separate files, repeat codes, and optimized drill head"
    c. Drill by layer pair
    d. Then click Drill
    e. The files should save in your artwork folder
    f. Move the nc-tools file that is left in your physical folder to the artwork folder as well
    g. All your padstacks will now have drill symbols on top of them too

NC Route
If you have large slots in your design, like the filter cavities, you might need a route file that defines the cutting patter of the cavity.
1. Go to Manufacture > NC > NC Route

2. Select separate files for plated and non-plated routing
3. Click route
4. The file will be generated in the physical file - move it to your artwork folder
5. If the only slots you have are the cavity holes, you'll only have one file since all the slots are non-plated

Plotting

Plotting gives you a pdf of the fabrication notes - pretty much a pdf screen shot of your screen. This shows your dimensioning, cross section, drill legend and notes.

1. Put my .cdsplotinit file in your h drive (not in any of the project folders!). It is normally used to create printouts of the board but I configured it to make pdfs of varying quality depending on your "printer" selection
2. Go to file > plot setup and select: Fit to page, Default Line Weight = 1, Auto center, Black and White, Screen Contents, Vectorized Text 1 width. Then press OK
3. Go to file > plot.
4. In destination, select printer name. Make sure it's set to PDF_6000dpi (you can also choose other printers but they're not as good)
5. Select OK
6. At the bottom command window it should say something along the lines of "plot file created"
7. Go to your H drive and there will be a pdf labeled plot6. Move this to your artwork folder and label it (ex testboard_Fab_Art.pdf)

Artwork

1. Your artwork should have:
    a. TOP.art
    b. 02_GND.art
    c. 03_SIG.art
    d. BOTTOM.art
    e. SOLDERMASK_TOP.art
    f. SOLDERMASK_BOTTOM.art
    g. PHOTOPLOT_OUTLINE.art
    h. [drill file 1].drl
    i. [drill file 2].drl
    j. [FabArt].pdf
    k. Nc_tools_auto.txt

These are what is sent to the manufacturer. To check if it all generated correct, either import them into PentaLogic ViewMate or to Cadence Allegro and make sure it looks right

Changing Center Cavity Dimensions

Rather than making you learn all that previous stuff, you just need a little.

a. Open the board file you want to use
b. Open the option pane on the right and click on the pin at the top to pin it to the window

Deleting Stuff

a. To delete, select an object and press on the red X under display. Delete the green board legends above the board as well as the two squares that denote the cavity in the board center
b. To delete the previous cavity dimensions go to Manufacture > Dimension Environment
c. Right click and select Delete Dimensions. Click on the two dimensions for the center cavity
d. If there's any text leftover, delete that normally (without the Dimension tool)

Add New Information

If you have a chip of size "A – squared" than the inner/smaller hole of the cavity will be "x-1mm – squared" and the expanded cavity that exposes layer 3 will be size "A squared"

a. Go to Add > Rectangle.
b. Change the Class to Plan and Subclass to All.
c. Click on the little P on the bottom of the window and a "Pick" window will pop up.
d. The origin is in the center so every coordinate will be +/- Dimension/2. Make sure to convert to mils. To enter a coordinate, enter x, y in the Value box and press pick. If you have a square, start with the coordinate in Quadrant I and add –'s to both values and press pick again to complete the square. After the square is done, right click and click Next.
e. Create the rectangles for the larger cutout that exposes layer 3 and the smaller one that goes through all
     a. Example: I want to create cavity for a 10mm x 10mm chip. For the larger cavity, I pick (196.9, 196.9) and then pick (-196.9 , -196.9). Then I click next. The smaller square will be 1 mil smaller per side so we subtract .5mm (19.7 mil) from every coordinate we enter => we enter (177.2,177.2) for the first point and (-177.2, -177.2) for the second.

Vias

If your cavity is larger than it was before you have to delete some of the vias that you are now on top of. If your cavity is smaller, you might want to add some.

Read Via Arrays on the Manual.

Dimensioning

1. Go to Display > Color/Visibility and in the top right corner, turn Global Visibility "Off". Then, in the Color Dialog window, go to Stack-Up > Plan and select Through All (the subclass you selected for the cavity dimensions). Then press OK.
2. Go to Manufacture > Dimension Environment. Right click and select linear dimensions.
3. Enter the coordinates for the top and bottom of the squares you created. This will create a dimension graphic.
4. In the options pane, in "text" copy the dimension and add Cutout 1-3 or Cutout Thru depending on which part of the cavity it is.
5. Click on the right of the board to place the text.
   a. Example. For the 10mm cavity, while in the dimension environment, pick the coordinates (0, 196.9) and then (0, -196.9). In the options pane, in text I will write .3938 Cutout 1-3 and place the text. Then I'll repeat for the smaller cavity but label it .354 Cutout Thru.

Art

Read the Artwork Section Above this section. All the artwork will have to be regenerated if you changes the vias. If you shrank cavity size but didn't add vias, then you just need to create a new "Plot".

Congratulations… you're done!

Quick thanks *to James Raftery and Dr. Houck for giving me this project.*

Donations between $200-200,000 are greatly appreciated.