CAM350 V7.0 Tutorial - Reverse Engineering a PCB

Written by Advanced CAM Technology,
and modified by Michael Raynes, Aspect Communications

Getting "legacy" designs, where you only have Gerber files, back into your CAD system - intelligently

This is the process of starting with Gerber files then adding information to the database such that an intelligent database can be brought back to a CAD system. Before starting make sure that you know what each component's reference designator is. If the silkscreen layer is incomplete, then get a fabrication drawing, BOM or some other method to determine each reference designator. If all reference designator are not known, you can make them up as you go.

Data Determination

First, figure out what kind of Gerber data you have. 274D data does not need any manipulation and can be used immediately (skip this section and go to **Reverse Engineering Procedure**). 274X data requires that the different electronic layers that are to be merged by the fabrication shop to make a final etch layer must be combined in CAM350 to create a single "copper" layer. Typically this can be a power plane with an imbedded trace, a plane with several other files to add in traces, etc. See the following steps:

1. This is not a necessary step, but I have found that thermals will be generated better in PADs, and pad stacks will be created cleaner, if the *thermals are now removed* from planes in the Gerber data. Unfortunately there is no easy way and this must be done manually.
2. Copy pads to a new layer (use this step only if the new layers are to be a surface layer?).
3. Select **Table/Composite** to access composite layer sets. Click on **Add**. For Composite_1 start choosing the layers you want to be combined into the final layer. Power and Ground planes will work better if the Bkg (background) button is changed to **Dark** and the plane layer being added in is **Clear**. The imbedded traces should be **Dark**. This results in a positive plane.
4. View the selected composite layers by clicking on **Redraw**. Only the selected composite layer will be shown. The data should appear as negative data.
5. Create composite into one "copper" layer using **Utilities/Convert Composite** command. Select the composite you want to create. A new layer will be automatically created.
6. Copy pads from new layer into the composite layer just created.
7. To return to viewing the board click on **View/Composite**. Proceed with the Reverse Engineering Procedure.

Reverse Engineering Procedure:

To completely ***reverse engineer*** a set of Gerber files into a PADS ASCII database requires the generation of parts or components. It is extremely helpful to have a Gerber layer that has a drill layer with pads the same size as the drill used (verify drill sizes against the pcb fabrication drawing). This can be used to generate the drill list that is then used by CAM350 to treat a drill location as a hole during netlist extract. The parts created become the footprint library for a design.

1) **Import the Gerber layers and set the draw and flash colors differently on all layers. Delete from any layers extra data (i.e. title blocks from design shop)**. If you want to keep original text in your PADs database you might want to create a new layer and move text items to that layer. This is done because text items can be wrongly converted as SMT pads on your top or bottom layer. I have found that thermals will be generated better in PADs, and pad stacks will be created cleaner, if the *thermals are now removed* from planes in the Gerber data. Unfortunately there is no easy way and this must be done manually.
   Optimize the data by running Edit, Line Change, Join Segments.
   Optimize the data by running Utilities, Data Optimization, Remove Redundant Pads (to remove covered data and overlapping pads that might look like a dual drill in one location). Run this only on the surface layers (turn off all layers except the top and bottom) one layer at a time!
   Optimize the data by running Utilities, Data Optimization, Remove Covered Data. Run this only on the surface layers (turn off all layers except the top and bottom) one layer at a time! Running this on plane layers could cause problems with flooded areas. Check on the *Delete pads covered by graphics* box. This will remove flashes that might be in a trace and don't need to be there. To see these click on the T box at the bottom right corner (this toggles which color of the flash/draw box is on top).
   Do a drill file extract using the drill layer (Tools, NC Editor, choose Utilities, Gerber to drill). Setting the <u>draw</u> color different from the <u>flash</u> color makes identification of drawn pads (SMT pads are sometimes drawn) much easier. Any large drawn area (ground plane under a TO220 regulator, connection from backplane connector or large area no drawn with a single decode) **must be identified as a raster drawn** area (Utilities, Polygon Conversion, Draw->Raster Poly). Use Fill (F key) to verify raster area.
   You can now save the file as something_step1 to identify it as the first step. All information becomes part of the .PCB file you save, so opening the database later will take over where you left off. No information is lost.

2) **Tag the layers properly (Tables/Layers) and put them in order (Edit/Layers/Reorder).** No system can know what the function of each layer is. "Tagging" each layer tells the software how to process the data (which is the outside, internal or plane). Only electrical layers are looked at, so masks and graphic layers are superfluous (graphic layers). This step is critical especially for internal planes. Most internal planes are tagged as "Negative Plane" *because the actual copper would be the opposite of what is displayed*. Use the N hot key to visualize a plane. After tagging, use Edit/Layers/Remove to remove any layers that are not required (i.e. delete paste, mask, hole plot and graphic layers because PADs is limited as to how much graphics can be put back into PADs).
   You can now save the file as something_step2 to identify it as the second step.

3) **Do draw-to-flash (Utilities/Draw to Flash) conversion as required.**
Activate Ignore Single Stroke Lines box. The step is required because netlist extract *defines the end of a net as where a draw ends at a flash*. In addition, <u>component pins must be flashed</u>. If you still have them convert thermals also. Review the results. You are looking for traces that did not convert to draws. Short traces will convert to oblongs by mistake. This will not export out properly into the ASCII file. One way to fix this is to replace the flashed traces with draws at this point. Another way is to choose Interactive Draw to Flash, it's a little slower but it won't convert the art you don't want to convert.
You can now save the file as something_step3 to identify it as the third step.

4) **Run Utilities/Netlist extract.**
This process will deduce net connectivity based upon the previous steps. Layers must be accurately aligned. If not, run Edit/Layers/Align first. Nets will be numbered. A small cross will be displayed on some pads to represent a through-hole padstack. (These padstack markers can be turned off under Settings/View Options.) No padstack marker means the pad was processed as SMT.
You may want to run Net Check (**Analysis, Net Check**). Net Check takes a while so be prepared to wait. Net Check will find floating traces or locations where Draw to Flash should have been run. This is not a required step, but it may give you important information. If errors are found, Net Check will create a new "netcheck" layer tagged as a temporary layer. Turn on that layer by itself to see the errors (if there were any).
You can now save the file as something_step4 to identify it as the fourth step.

5) **Run Utilities/Build Part.**
This step creates the footprint, or decal, that PADs will reference. This is the tricky part. Once you have started this process you may not go back and do any previous steps without doing a data reset (Edit, Change, Explode, All). The system will leave the original silkscreen layer untouched, but, as reference designators and part outlines become part of the components, they will be placed on new layers called refdestop and refdesbot. The build part process is as follows:
   a) **Turn on <u>only</u> the Top layer and the original Silkscreen layer**.
   Zoom into a component. During this process do not select single point nets (testpoints or mounting holes) because PADs can't handle a single point net, only nets with 2 or more connects.
   b) **Run Utilities/Build Part**.
   "REF" is attached to the cursor. Now is the time to check the button bar to make sure the reference designator text height and orientation is correct. *"T" turns the REF text 90 degrees and "M" flips it to the other side*.
   c) **Click the REF in the center the current silkscreen reference designator** (i.e. in the middle of the letters "C1").
   All of the old reference designator should change to white. This will now be the location of the new PADs reference designator after conversion. Location and orientation can be changed on an instance by instance basis later.
   d) **You will then be prompted to window around the current silkscreen outline of the part.**

The cursor is a "crossing window" so only those portions of the silkscreen you want represented in the new database outline need be selected. Anything selected will turn to white. If no silkscreen outline information is present, you may just enter two points to skip this step.

e) **You are prompted to enter the pin sequence**.
The button bar shows the starting pin # and the "inline pins" check box is defaulted ON. The pin numbering will automatically start with pin #1, and you only have to select the end pins that are in a row. The system will automatically number the intermediate pins.
Note: The next pin number will always be displayed on the button bar and, if desired, you can change the pin # in this field to another number, or even an alphanumeric (A1 will increment to A2, etc.) at the beginning, middle or end of adding pin numbers.

d) **End the pin entry sequence with the right button.**
After pressing the right button you will get a dialog box that allows you to go back and change anything that you did (outline or reference designator location for example). Assuming everything is correct just press CREATE. If a mistake in the pin numbering sequence is seen click on CANCEL. Sometimes a pad will come up with two pin numbers due to errant data within the pad area that is seen as a pad by CAM350. You might have to go in and manually edit the errant data out of the pad area.

e) **The next dialog box will allow you to enter the footprint (or decal) name for this item**.
This dialog box also tells you what side the system is building the part from. Make sure you're building from the correct side of the PCB! Enter the “Footprint Name”. This is the name/decal that the system will use to store the part in the internal part library. For example, a 16 pin DIP might be called *Dip16*. Do not enter any spaces in this name, and do not enter something like 7400. This is the footprint/decal name, NOT the device name! CAM350 will allow you to create different device names that only differ by character capitalization (RES1 as versus res1). Don’t do this as PADs doesn’t distinguish between the two and will pull the first one it finds! You only need to build one footprint for each unique part.

Note 1: If you miss a part you can always come back and add it later. It is recommended that, while you are learning the overall BuildPart/AddPart sequence, you do only one part at a time until the process feels comfortable. Also, you only need to build one part, regardless of the orientations used.

Note 2: Once a library has been created, you can always go to the Part Editor and export the library (.plb extension) for use in other reverse engineering jobs.

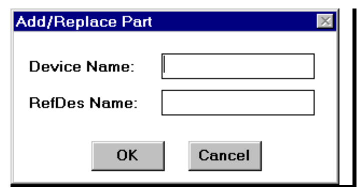
f) **Go to Tools/Part Editor and see how the footprint looks.** It can be edited in the Part Editor if the silkscreen is not correct. Verify the text height and thickness is what you want. Also verify that the pin sizes appear to be the same for all pins or are round when they are supposed to be square. If a pin size appears smaller than the rest of the pins it means that some errant data was picked up as the pad instead of the true pad. If a round pad appears as a square then that means that the pad stack was not interpreted correctly and CAM350 thinks it is a SMT pad. Select File, Exit Part Editor to return to the board.

Note: Only the pin locations are stored as part of the footprint. This is done so that when Add/Part is run later, the same footprint can be used even if the pad shapes are different from location to location. The Add/Part process will use whatever pad shapes are present as long as their locations match with the pin locations. Repeat this process for all parts on the design. You can now save the file as something_step5 to identify it as the fifth step.

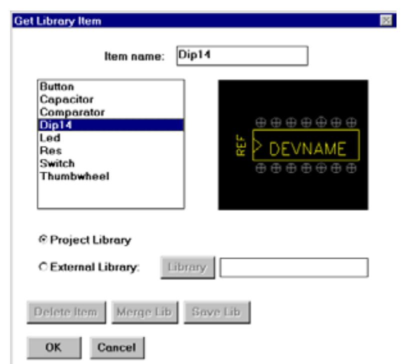
6) **After parts are available in the Part Library, run Add/Part**.

a) Add/Part is the process of actually instantiating each component. Add/Part replaces the data on the silkscreen layer with a part. A white box is drawn as a visual indicator that there is a part there rather than just Gerber graphics. **If you are working with double sided boards a solid white line is used for top-side parts and a dotted white line is used for bottom-side parts.**

b) Select a footprint from the local library and press OK. Alternatively you go to an external library and load a part from there. Any part loaded from an external library gets copied into the current project library.



c) You will be asked to enter a Device Name and Reference Designator. Only the reference designator is really required. The Device Name is the same as the part number (i.e. 74F647). Device Name will be added to the Refdestop and will run down the middle of the part if you do want to do it (some companies want the industry standard device type to appear on the silk for ease of identifying the replacement part for technicians).



d) **Pin #1 of the selected part becomes attached to the cursor.** The object snap is automatically turned ON so that the part snaps to the selected pad. It's a good idea to turn off the grid snap (S key) during this function. If you want to rotate the part <u>**before**</u> you place it, just press the **T (Turn) key**. If you want to place the part on the back side of the PCB, press the **M (Mirror) key** (all graphics and Reference Designators will mirror). **Once the part is placed, you press the right button to commit and the pins become part of the appropriate nets** (if the placement is incorrect then press the Cancel button on the part placement toolbar to stop the process). *You are now prompted to select the Reference designator and move it on top of the silkscreen Reference designator.* Here again, you can use the T and M keys to rotate or mirror as required. **After placing the Reference designator, you hit the right button to commit.** The part is instantiated, the Reference designator is automatically incremented, and you are ready to place the same part in the next location. You can change parts while in the Add/Part command by selecting the part pulldown on the tool bar. The next Reference designator number is always displayed on the tool bar. If you are not doing the placement in Reference designator order, you can change the Reference designator on the tool bar at any time. The system will not allow assigning the same Reference designator again. In addition, the next time you come back to this part, the lowest unused Reference designator will automatically be selected. Repeat this process for all parts on the design. You can now save the file as something_step6 to identify it as the sixth step.

Note 1: The instantiated component's Reference designator and outline information is placed on a NEW layer which is automatically tagged as refdestop or refdesbot. This is done so that the original silkscreen layer remains untouched and can serve as a check so that no parts are missed.

Note 2: We suggest selecting Settings/View Options and at the bottom of the dialog box set the part outlines to "draw after graphical data". This will make the part outlines always visible.

Note 3: We also suggest that you view part outlines for each side separately. Even though the part outlines on the top are shown as solid, and on the bottom as dotted, it can get crowded on a dense board.

7) **Name nets to real names.** If you want you can rename nets to proper names at this time (Edit, Change, Netname). It is handy to do at least the power and ground so they can be easily identified in PADs. If you have to explode nets at this point you will lose the net names you have entered! You can now save the file as something_step7 to identify it as the seventh step.

8) **When you are done, unless you must have the original silkscreen layer, remove the old silkscreen layer.** It is not needed on the PADs side because you have created a new refdestop and refdesbot layer. When you have removed the silkscreen layer (save the file as something_step8 to identify it as the eighth step) it is time to export a PADs ascii database (File, Export, CAD Data and select PowerPCB whatever version you want). If you select PowerPCB 2.0 the exported database will have all signal layers present, but the power and ground planes will only be exported as a “ratsnest” due to raster line limitations in PowerPCB 2.0. This does not occur in PowerPCB 2.1 or greater.

Special Notes and Considerations

Building parts from the backside

If you build a part from the backside, the system should detect this automatically, and show you that the part is on the backside in step 7E above. Regardless of the side the data was built from, the system will always store the part in the library as if you built it from the topside. This makes the overall process more reliable because parts are always stored the same way, independent of how they were built and how they are instantiated.

Adding parts to the backside.

Check View/Backside (under the View menu). This does not mirror the data or in any way affect the database. This effectively just "flips the board over," making it easier to read the Reference designator and place the parts.

Building double-sided parts

An example of a double-sided part would be an edge connector that has SMT connections on both sides of the PCB. CAM350 can handle these, as long as your target CAD system can. We suggest turning ON only the topside when the topside pins are being selected then, while still in the Build/Part command, turning ON the backside then selecting those pins. Also, if there are multiple rows of pins, you can use inline pins for each row, but remember to uncheck the Inline Mode when you select the first pin of the next row.

Checking your Padstacks before you get started

CAM350 displays a black cross on every through-hole padstack. Before you get started, make sure that SMT pads do not have a padstack marker and that through-holes do.

Exploding Nets

If the netlist extraction was incorrect for any reason, such as you forgot to do draw-to-flash conversion, you can use Edit/Change/Explode Netlist to remove the net connectivity. Otherwise, you can just run the netlist extract again and let it automatically explode the nets.

Exploding Parts

If you make a mistake when adding parts, you must explode the incorrect part before trying to add another part.

Netlist Extract note

Version 5.0 will use drill files to assure proper layer to layer connectivity, which is particularly important for boards that contain blind and buried vias. If no drill file is present, the system will assume that round pads or square pads that exist on both sides on the PCB, and are collinear, are through-holes. If this case exists, BUT YOU INTEND ON THE PADS BEING SMT, make sure you change the dcode type to SMT in the aperture table for those dcodes before running Netlist Extract.

Add Part note

Add/Part will verify that the pin locations match the Gerber data before it will allow the part to be added. It uses a 2 mil tolerance to handle pads that are not perfectly aligned. Nevertheless, the overall pin count must match exactly.

PowerPCB Design Hints

1. Do a file import on the file. Nets might show a “ratsnest” even though the traces are present. Because of alignment issues PADs might not associate a trace end with a drill hole. PADs wants a flash at the end of a trace and wants it to match up with a drill hit. Set the colors for the layers and turn off reference designators, types and outlines (Setup, Display Colors). Label layers (Setup, Layer Definition). Be aware that the file is quite large because of reference designator P parts (ref des and types). The P parts can be deleted to lower the file size.
2. View Nets and turn off the power plane and ground plane nets.
3. Verify the drill sizes for component’s leads are not the same size as the pad. If they are then PADs will not allow you to route to the pad. Edit components if necessary to change the drill size for pads.
4. Look for traces that didn’t convert properly. These will appear as diagonal traces going across the board where a trace shouldn’t be. You might want to go back into CAM350 and determine why the trace is not being properly (one cause is if a net is created with a trace and copper on opposite sides of the board, if so then change one of the traces to raster data in CAM350 and re-export). Look for “ratsnests” where PADs could not determine if a piece of copper was connecting 2 points. Most of the time double-clicking on the trace in PADs where the fly-line goes to will reroute the trace to correct the error. Sometimes vias do not come through the conversion process. Look for traces on two layers meeting and not having vias (another trick is to go back into CAM350 and try using Edit, Move Vtx/Seg to snap that line to the center of the pad, and then rerun netlist extract). Be aware that you might have to check net rules and verify if a trace has been allowed to route on layers. Many times traces were only allowed to route on top and ground!
5. Turn thermals on (right click, select pins, then edit/modify and turn off and on thermals).
6. A plane layer will appear to be present, but it is only a 2D line. I have always had to delete this 2D line and recreate the plane outlines to be able to pour planes.
7. Associate power plane nets with the power plane and associate the ground plane nets with the ground plane (Setup, Layer Definition, Associate Nets).
8. If you had imbedded traces in power or ground planes they will need to be rerouted in PADs. The outline will be there, but not the trace. Reroute the trace and delete the shape that used to be the imbedded trace and its keepouts.
9. Take the entire design and save off all the decals to a unique library. Rename the decals to meaningful names and then replace the decals in the design with the newly named decals.
10. A final check would be to run a netlist comparison using the gerber netlist and the PADs database netlist, or, to export gerber files from PowerPCB and overlay them against the original gerber files in CAM350.
11. Once you are happy with the design create a schematic in PowerLogic.
12. Use OLE to compare schematic and database. Part Differences that show up can also be 2 leaded parts with the opposite pins connecting on the same nets (swap pins in schematic to fix).