



CAM350 Release 6.0 Tutorial - Reverse Engineering a PCB

Distributed by DownStream Technologies, and modified by Michael Raynes, Aspect Communications

Intelligently creating "legacy" designs for your CAD system from old Gerber files

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.

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 be combined to create a single "copper" layer. 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. Use **Table/Composite** to select composite layer sets.
4. View the selected composite layers by clicking on **View/Composite**. Only the selected composite layer will be shown.
5. Create composite into one "copper" layer using **Utilities/Convert Composite** command. This layer will be a raster filled layer.
6. Copy pads from new layer into the composite layer just created.
7. Run data optimization to remove covered data (in this case the polygon pads will be replaced with flashed pads).

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. This can be used to generate the drill list which 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.**

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.

Optimize the data by running Edit, Line Change, Join Segments.

Do a drill file extract using the drill layer (Tools, NC Editor, choose Utilities, Gerber to drill). Setting the draw color different from the flash 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.

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). **Also 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 you top or bottom layer. You can then use the layerord.scr macro to set the correct layer order.

3) **Do draw-to-flash (Utilities/Draw to Flash) conversion as required.**

The step is required because netlist extract *defines the end of a net as where a draw ends at a flash.* In addition, component pins must be flashed. Convert thermals also. Change the color on each layer so that the draws are different colors than the flashes. 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.

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. **Save the PCB.** The PCB database stores all the information, and since you have gone through several steps it's a good idea to save it at this point.

5) **You may want to run Net Check also.**

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).

6) **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 only 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 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.).

f) **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.

- g) **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! You only need to build one footprint for each unique part. Once you have done this for all parts, then go to Add/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.

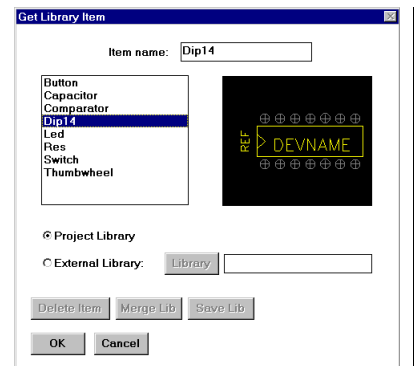
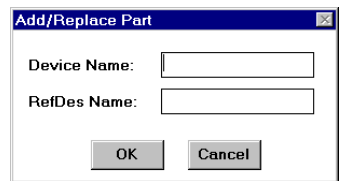
Note 2: Once a library has been created, you can always go to the Part Editor and export the library (.plb extension) for use in other reverse engineering jobs.

- h) **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

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.

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

- Select a footprint and press OK
- You will be asked to enter a Device Name and Reference Designator. The Device Name is the same as the part number (i.e. 74F647). Only the reference designator is really required.
- Note also, that this is where 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.



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. A solid white line is used for top-side parts and a dotted white line is used for bottom-side parts.

- 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 **before** 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.** 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 part in the next location. You can change parts while in the Add/Part command by selecting the part pulldown on the button bar. The next Reference designator number is always displayed on the button bar. If you are not doing the placement in Reference designator order, you can change the Reference designator on the button 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.

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.

8) **Check your work and save it.**

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.

9) **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 it is time to export a PADS ascii database (File, Export, CAD Data and select PowerPCB 2.X). 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. Other 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. Reroute the trace in PowerPCB (this will more than likely clear the problem). 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.

PowerPCB Design Hints

1. Label layers (Setup, Layer Definition).
2. Associate power plane nets with the power plane and associate the ground plane nets with the ground plane (Setup, Layer Definition, Associate Nets).
3. View Nets and turn off the power plane and ground plane nets.
4. Turn thermals on (right click, select pins, then edit/modify and turn off and on thermals).
5. Take the entire design and save off 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. 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 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.
6. 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 (delete connections and reverse part in schematic to fix).

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.