

Data Flows

eNewsletter from DownStream Technologies
Early Winter 2010

IN THIS ISSUE:

Automating the Documentation Process

Written by Mike Buetow
pages 1-3

Next Generation Post-Processing

On-Demand Overview and Demo
page 4

Maximizing Engineering Value

On-Demand Presentation
page 5

Panelization of Multiple Boards

CAM350 Tech Tip
page 6-8

DownStream Record Sales

Quarter Results
pages 8

Virtual PCB

Join us online in March
pages 8

Automating the DOCUMENTATION PROCESS

Dedicated tools help reduce the once-manual job by 20% of the typical design cycle time. **by Mike Buetow**

The question posted to the DesignerCouncil email forum was simple and straightforward: “Why on earth can’t these companies make a product out of the box that does engineering documentation well? I find it hard to believe there can’t be a boilerplate level delivered that could handle 80 to 90% of most companies’ documentation needs out of the box. ... Even the BoM reports delivered out of the box from Oracle are absolutely useless. You need an integration company or an internal team of Oracle experts to spend a fair amount of effort just to get a usable BoM report.”

What’s fascinating, if not somewhat predictable, is how many firms take software that is intended (and good) at one function and repurpose it for something else. Take, for example, two of the best-known product life management tools: Oracle’s PLM and PTC’s Windchill. Good programs, both, but they were never intended to handle circuit board documentation. It’s one reason why it takes, on average, two man-days of the typical 10-day design cycle to complete the documentation (**Figure 1**).

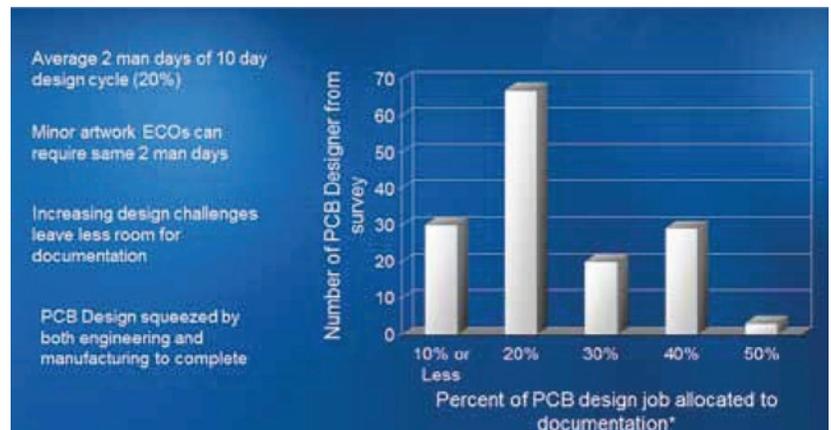


Figure 1 Studies show documentation takes 20% of the typical design cycle.

Best-in-class processes reduce the amount of time spent creating the documentation; are automated so as to create a “living” document, or one that’s not fixed in time, unlike a paper printout of a pdf where, in the event of an engineering change order, the documentation needs to be regenerated; and facilitates product flow to the extent that the end product quality is enhanced. As Downstream Technologies cofounder Joe Clark says, “Documentation is much bigger than just design, fab and assembly.”

Designers are pros at stretching the limits of CAD tools and other software, but at some point the deficiencies become too glaring to tolerate.

It’s why I’m surprised more haven’t turned to Blueprint, which purports to solve the headaches by providing access to all the electrical intelligence contained in the PCB CAD database to create detailed documentation in reportedly a far shorter timeframe than using traditional ECAD or MCAD tools.

The brainchild of DownStream, Blueprint was developed by former board designers who understand from firsthand experience the pains of their excolleagues. It uses a drawing- and sheet-based approach to create actual PCB documents (drill drawing, assembly drawing, parts list, and so on), and stores them in a digital release file. Blueprint uses the CAD data to automatically create unlimited views, details, and charts of the PCB, and can revise the documentation to reflect new PCB changes, reportedly in seconds. It’s not just a tool change; it’s a change in methodology.

Like MCAD, Blueprint can document drafting and dimensional specifications. Where Blueprint separates itself from the traditional MCAD tools, however, is in the number of required translations. Mechanical tools aren’t designed for the specific nature of a printed circuit board. To wit: MCAD cannot distinguish between a layer on a circuit board and a floor of a building. Meanwhile, Blueprint is a full 2-D drafting tool designed specifically to create PCB documentation and capable of creating anything that the CAD, fabrication and assembly departments might need. It eliminates constraints imposed by the PCB design tool for documentation, but retains the intelligence lost when using an MCAD tool for documentation.

The best example of this is Blueprint’s support of unlimited PCB design views on any drawing sheet. Each instance of the design can be set with different display settings, scaling and formatting. However, each is also linked to the source CAD to automate ECOs, requiring one file translation from the source PCB CAD data to Blueprint (Figure 2).

As our emailer suggested, most users try to generate documentation from their ECAD tool. But ECAD was never intended for that purpose: Dimensioning is the domain of mechanical design. In practice, it’s another story. Asserts Clark, “Users do [documentation] in ECAD because they need to get

it onto the piece of paper. There are lots of things done in the electrical design process that have nothing to do with electrical design; it’s all aimed at the documentation.” For instance, the use of assembly variants or mechanical components, which are never simulated but are pushed into a design capture or PCB layout tool to ensure they are represented in the final bill of materials.

Blueprint was developed to work with existing EDA tools. The MS Office environment requires a single load of the entire PCB CAD database to start the documentation process, and uses all the CAD intelligence to help create the documentation. ECOs are added by “refreshing” the source CAD data. Blueprint refreshes the documentation package, adding the new data in every instance of a board view, detail and chart.

“Blueprint brings ECAD design into the documentation tool,” says Clark. There, in the gallery (also known as the library, the term “gallery” is used so as not to be confused with the data for the PCB packages and land patterns), users create a drawing sheet, dragging and dropping the elements onto it. Any element – the number of zones, the font, colors, format, and so on – that is dragged and dropped on a sheet is “intelligent”; links remain active from the underlying design database, so if the links change, everything within the drawing is updated.

Inside the file are contained via stackup and drill layer pairs; drill symbols, patterns and charts; notes and callouts; mechanical component support; parts lists and assembly drawings; tooling, and so on. Schematic, Gerber, DFX, PDFs, plus any graphic or audio files are all stored. Or, in the words of Rick Almeida, DownStream’s cofounder, president and a former designer, “Any data used to create documentation and drive manufacturing.”

Automation of documentation is created by first creating the graphical “pictorial” of how the user wants to see a specific documentation element, for example, a “Via Stack-Up.” The graphical instruction defines how the via stackup should look.

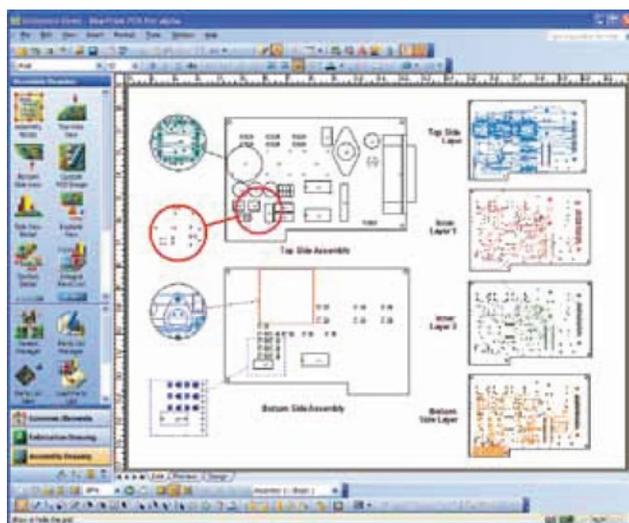


Figure 2 Each design view is linked to the source CAD, simplifying ECOs and reducing translations.

The via stackup could be amended, and the template modified to company specific versions and saved for reuse. The template graphics are linked to data fields that pull the correct information from the PCB CAD data to intelligently create a via stackup that reflects the design intent and remains linked to the CAD data, so when the CAD data are updated (e.g., new layers or via ranges added or deleted) the detail is also automatically updated. “Essentially we use the same approach that is found in MS Excel chart build, where empirical data are married to user specific graphics,” says Almeida. This same process goes for drill charts, parts lists, finger chamfers, coordinate lists, exploded views, etc.

BluePrint supports automation of the assembly drawing as well. This includes parts lists, which, even if not shown in the documentation set, are in the underlying design database and can be referenced as needed. The search function can find specific parts and component outlines. What it is not is a bill of materials management tool. “The parts list is a key part of the documentation,” explains Clark. “Documentation affects the parts lists, because as designers do the layout, they add mechanical parts, such as face plates, screws, washers. They need to be able to add to the parts lists and export them to the right department so they can add to the BoM and bring in all the attributes of the parts.” With BluePrint, users can import an external parts list from PLM or other enterprise systems, and sync the parts list with the assembly document, then export the parts list back to the enterprise system for ongoing management

BluePrint follows the orientation of the board to place component reference designators. (The latter can be hyper-linked, too.) Assembly variants can be created at both the parts list

level and also with a graphical assembly pictorial, with special legends to show what’s been installed, uninstalled or substituted. Documents such as relevant standards can be inserted as hyperlinks into the database. It also can generate rework sheets, showing cuts and jumpers, and include installation procedures.

One complaint about reference designators in Oracle and SAP is that they do not display all instances of a particular part number with the part number and description in a single screen or menu. That’s not an issue for BluePrint. The documentation set in the GUI looks just as it will print on paper (Figure 3). The documentation package also can be exported as a pdf and posted to the Web, or as html on a company Intranet.

“It treats the end product like a PCB, not a generic widget,” Clark says. “We don’t know how good our documentation is until we print it. Then, if something is wrong, we need to go back upstream and fix the ECAD database, then print again. What we do is allow the user to work with the documentation as it will be delivered upon release. This is a full 2-D drafting tool; it’s not intended to replace the mechanical group, but you need to be able to move and place things when doing the documentation. The handcuffs need to be removed.”

As we reported several months ago, Qualcomm has become one of the largest companies yet to standardize on BluePrint. Come this time next year, I would expect many more such announcements – and far fewer plaintive missives like the one at the beginning of this story.

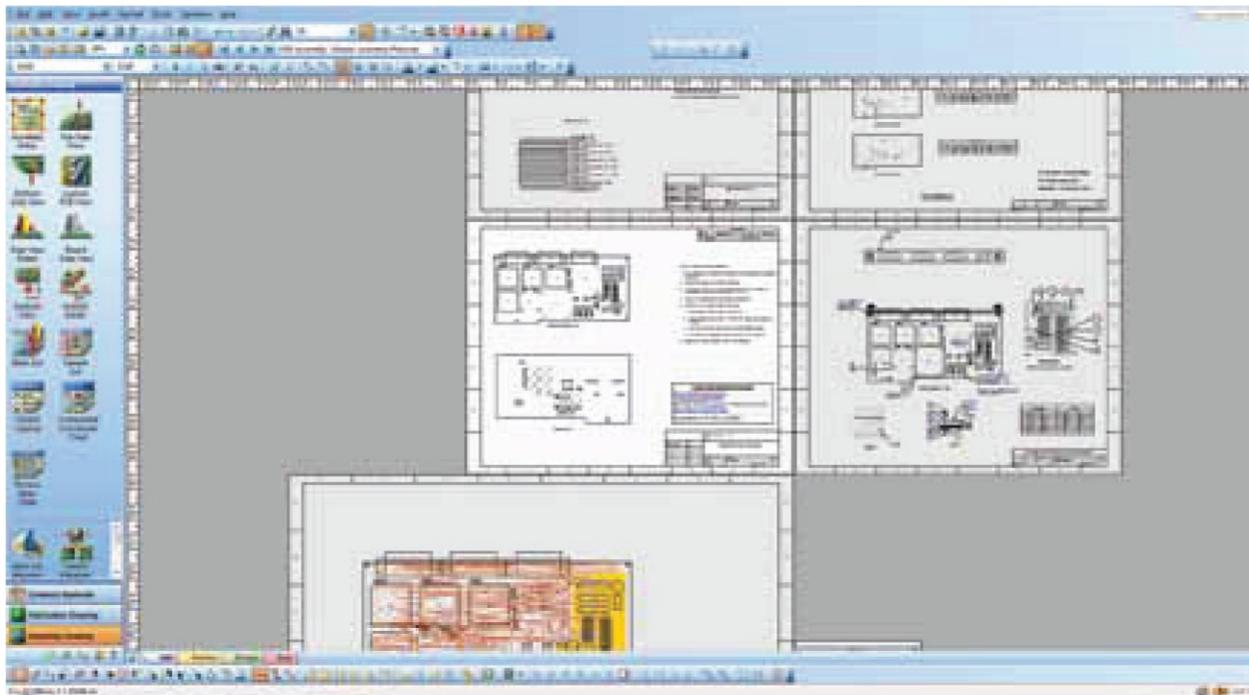


Figure 3 Unlike other GUIs, the documentation set looks the same on paper as on the screen.

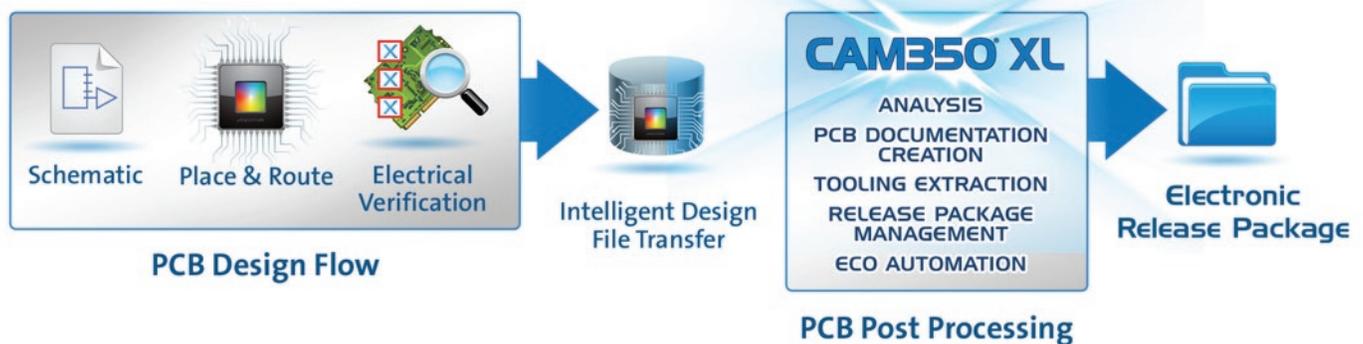
Discover the Next Generation of PCB Post-Processing and Management Solution

Available On-Demand: Pre-Registered Guest Login / Non-Registered Guests

Remember when Placement and Routing tools revolutionized the way you PRE-Processed your PCB designs? At this free webinar you will learn how CAM350 XL can do the same for POST-Processing your designs.

CAM350 XL is an integrated suite of tools that addresses the needs of the PCB designer who is charted with releasing a PCB design to manufacturing. It imports an intelligent design file and focuses on the key areas of PCB post-processing. The result is a single electronic Release Package, that is portable and viewable by others, that ensures successful manufacture anywhere, anytime.

CAM350 XL sets the new standard for post-processing PCB designs. With CAM350 XL you can verify design and Gerber files, optimize the files for manufacture, extract tooling to generate Gerber, Drill and Mill data in industry standard formats, create comprehensive PCB documentation and easily distribute a comprehensive electronic Release Package with all the deliverables for successful PCB fabrication.



Demonstration Includes How CAM350XL™:

- Verifies design and Gerber files for Manufacturing
- Extracts Tooling information to generate Gerber, Drill and Mill data
- Automates Engineering Change Orders
- Creates comprehensive PCB documentation
- Creates an electronic Release Package for distribution

Who Should View this Presentation and Product Demonstration

- PCB Designers
- Engineering and CAD Managers
- Project Engineers and Managers

Available On-Demand: Pre-Registered Guest Login / Non-Registered Guests

Maximizing Engineering Value By Automating PCB Documentation

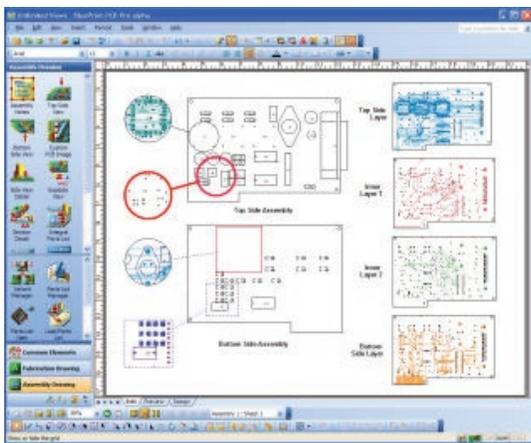
Explore the latest trend in PCB Documentation. Now by using BluePrint-PCB you can automate the tedious task of creating PCB documents. Not only will BluePrint automate the process, the results are more detailed, comprehensive, easy to use documentation. With BluePrint we offer three unique ways to distribute and share the end documentation as well - making PCB Documents and ECOs a simple, streamlined endeavor.

[View the On-Demand presentation to learn more!](#)

Overview

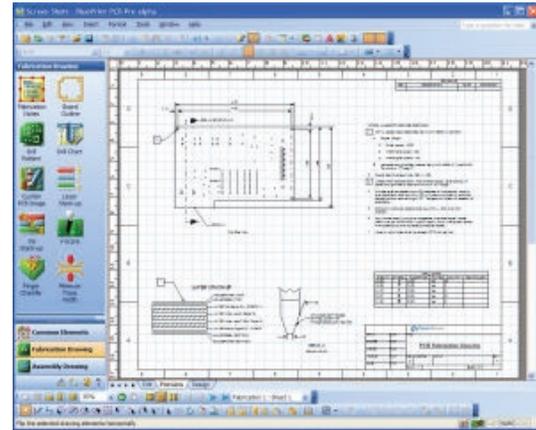
During this informative webinar you will see how BluePrint-PCB can dramatically change the way you create PCB documentation by using intelligent design data to automate the PCB documentation process. Topics to be covered include:

- Creating and using templates to quickly populate a documentation set
- Creating comprehensive PCB Fabrication Drawings
- Implementing engineering change orders (ECOs).



Fabrication Drawing as seen by BluePrint-PCB

BluePrint-PCB “knows” it is creating a PCB document and imports the entire PCB CAD database once to automatically create and link unlimited PCB views, details and tables. By importing the CAD data you preserve the design intelligence and overcome the constraints imposed by both PCB and mechanical CAD alternatives. The result is an electronic document, which better articulates the instructions for successful PCB manufacturing.



Unlimited PCB Views using BluePrint-PCB

BluePrint-PCB will:

- Use PCB intelligence to create common drawing elements
- Create live and interactive documentation with hyperlinks and multi-media
- Store all information associated with the PCB assembly in one electronic “release package”
- Support instantaneous engineering changes (ECOs)
- Allow viewing of documents without the application - via the web, PDF, or a free viewer

With BluePrint, you can create an electronic PCB release package that is more complete than what is being created today. BluePrint also supports the integration of digital multi-media content such as JPEGs, audio files, and even video to better communicate the often complex instructions needed to fabricate and assemble a PCB.

In addition, BluePrint’s hyper-linking capability allows the creation of links between different document elements, external files, URLs and email addresses. With BluePrint, all the data is logically, simply displayed - PCB documentation is live, interactive, and much more informative.

Video On-Demand

Who Should View this Presentation

- Anyone who has responsibility for, or whom is involved in, documenting PCB designs.
- Project Engineers responsible for the finished product
- Engineering and CAD managers who are evaluating ways to improve their process to meet time-to-volume and quality production goals
- Anyone who wants to learn about new developments in design automation tools aimed at the manufacturing and fabrication phases of new product development.

Panelization of Multiple Boards in CAM350

The CAM350 Panel editor can step and repeat the design loaded to create a manufacturing panel. But this solution only works on one board at a time. To create a Panel with multiple designs, the Merge command must be used.

The Merge command is under FILE | MERGE in the CAM350 menu. It allows the user to load in a complete CAM350 database as a Merged Database. Merged Databases function as a single item in the editor. This allows the user to Merge in multiple CAM350 databases and arrange them as desired using the Edit menu commands.

CAM350 Database Preparation

As a Merged database is treated as a single unit, it is best that everything concerned with that database be complete. Proper preparation will make the merge process easy and help avoid common mistakes.

After loading your database, make sure all the layer types are properly labeled. With Gerber data you will typically have graphic layers. Using TABLES | LAYERS change the layer type as appropriate. Properly labeled layer types will make layer identification easier during the layer mapping step of merge.

Note that Part and Net database intelligence will not be merged. It is recommended these be exploded (if present) prior to saving. Part and Net info can prevent exploding custom apertures, discussed below. If your data came from Gerber or DXF then there is no Part or Net data present. But for ODB++ and other intelligent formats, it is a concern.

- Turn on all layers
- EDIT | CHANGE | EXPLODE | PART

Select all

- OK
- EDIT | CHANGE | EXPLODE | NETS

Select all

- OK

Merge will use all data in the database, so it is important to remove data you do not want on your panel. This includes data outside the panel edge such as text or logos as well as graphic layers that are not part of the final board.

- Use Edit | Delete, remove the unnecessary data from a layer.
- Use EDIT | LAYERS | REMOVE LAYERS to remove extra layers.

Some data needed for the panel may not be part of your initial database. This is common with NC Mill data for example and to a lesser extent NC Drill data. Make sure to run Gerber To Mill and Gerber To Drill as needed to produce the layers you want in your merge database.

Custom apertures are a problem for the Merge process. CAM350 assumes Custom apertures of the same name have the same definition. But this may not be the case. It is recommended that all Custom Apertures be exploded for your Merge database.

- Turn on All Layers
- EDIT | CHANGE | EXPLODE | CUSTOM

Select all

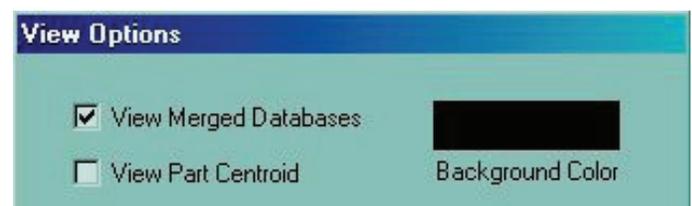
- Answer “No” to the Vector fill question
- OK

It is recommended that the layers be arranged by number using some kind of consistent system. This is especially true if the layer names are not very descriptive or if you have a large number of the same type of layer. Using the NAV Pane to drag and drop you can re-arrange the order of the layers. Right click to select “renumber all layers” once you are finished. You can also use EDIT | LAYERS | REORDER to do this.

Once database is fully prepped, use FILE | SAVE or SAVE AS to create a CAM350 database.

Merge Process

By default merged databases are displayed as an outline. This is useful while arranging the databases, but most users prefer to see all the data in the merge database at all times. You can use SETTINGS | VIEW OPTIONS to check the “View Merged



Databases” box in order to see all merge data in the CAM editor.

To make this a default

- FILE | NEW
- SETTINGS | VIEW OPTIONS to check the “View Merged Databases” box
- FILE | SETUP | SAVE DEFAULTS



To merge a database use FILE | MERGE then select your CAM350 database. After selecting it, the layer mapping screen will appear.

This screen controls where the incoming layers will end up. Data on the left (in grey) is info about the incoming layers. On the right is the destination layer. [New Layer] will create a new layer and send the merge data to that layer. If layers were arranged in the same order in merge database, then you can select the “By Num” button on the bottom left. By default layers will map to the same type.

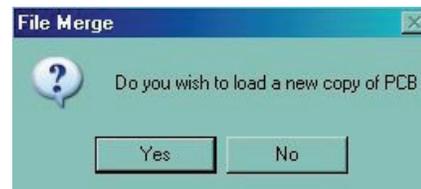
If there are multiple layers of the same type, CAM350 sets mapping to [New Layer]. The user must correct this prior to completing merge. Using the pull down the user can select which layers the merge layers should get mapped to. In the above example the incoming database has 2 Neg Planes that CAM350 has set to [New Layer].

If the type in question does not exist then it goes to a new layer. NC data is always set to a new layer. When pointing it to an existing layer, make sure that layer is a NC layer.

Once mapping is set, select OK. Click in the location in the CAM window where you want the database to be initially placed. Do not worry about accuracy in this case as you can move items around in the edit step. Just place items in the general location you will want them.

Merge Messages

When loading a merged database that has already been merged:



This is just a warning. By merging in a new copy you have to go through layer mapping again. A more efficient way of getting an extra copy of a board is to use Edit | Copy on the merged database.

When an electrical layer going to a new layer:



This is fine if you are merging into a blank database. But if you are trying to merge 2 databases into a panel then you need to go back and fix the layer mapping.

When caps (custom apertures) of the same name are detected:



A warning like this will not appear if all custom apertures (caps) in the merge database were exploded. If it does appear then the incoming caps of the same name as existing ones will be assumed to have the same definition. If they are different then the caps in the merged database will be changed. But if you are sure the caps are the same then this error can be ignored.

Edit Commands On Merge Data

All standard edit functions in CAM350 will work on Merge data as a single unit. So if you use Edit | Move and click on any part of a merge database, the whole database is selected. Using Merge, arrange items as you like. Using the Move command with the coordinate box, you can move selected data to a precise location. To learn more about Advanced Edit Techniques like the coordinate box, see the [Spring 2008 issue of Dataflows article “Data Selection Modes for Editing”](#)

Editing elements of merged data is possible, but not recommend. Such edits should be done prior to saving the merge database. But by using Edit | Change | Explode | Merged Data and selecting it, you can change merged data into Standard CAM350 data. This cannot be undone, so if you want to move the database after explode you'll have to use windowing selection.

As mentioned before you can use Edit | Copy to create extra copies of a merged database. The Mirror command also works, but it does not change layer mapping, so it will not work to flip the board front to back side. If that is a desired, merge in a new copy of the board and map the layers as desired.

Panelization Of Merged Data

Once your merged databases are arranged as desired, the resulting data can be stepped further using the Panel Editor. But keep in mind all the data loaded in the CAM editor will be viewed as a single "board" by the Panel editor. So if you have 4 databases arranged in a group, that group would be stepped as a single database by the panel editor. This can make Venting difficult.

The best work around for this is to use Add | polygon to create any venting needed. Note that venting patterns can be created using add polygon. For more info see the [Spring 2008 issue of Dataflows article "Add Polygon – Overlooked features"](#)

Final Note

The FILE | MERGE process should not be needed for single database panels. Using features such as Edit | Spin Image and Edit | Flip you should be able to fully control how your single board is displayed for any particular circuit image.

DownStream Announces Record Sales Quarter

Continued customer satisfaction and adoption of DownStream products and services has led to the announcement of record sales for the quarter ending September 2010. Increased sales were seen across the board as demand for the company's core products CAM350XL® and BluePrint-PCB® grew. In addition to overall increased demand, BluePrint-PCB® sales were impacted by a number of larger orders placed for the product as a variety of companies made it their standard documentation tool.

Sales in Q3 2010 were the third largest ever recorded in DownStream's history and they represented a 10% growth over the same period in 2009. In addition, the company has announced an overall sales growth of 8.5% year to date.

"At DownStream, we are particularly pleased with the sales numbers because we have always been a company driven by customer requirements," said Joe Clark, one of the Founders of DownStream. "As always, we stay committed to meeting the technology and service required by our customers and look forward to the partnerships we have forged with them," he said.



March 8-9, 2010

Virtual PCB (www.virtual-pcb.com) - the industry's only live, fully interactive, Web-based conference and trade show for PCB engineers, designers, fabricators, assemblers and managers worldwide returns for its fourth year live March 8 - 9, 2011 and available on demand through April 9, 2011. Virtual PCB brings together buyers and sellers from around the world.