NEW PRODUCT REVIEWS

Editor: Richard Eckhouse, MOCO, Inc., P.O. Box A, 91 Surfside Rd., Scituate, MA 02055; Compmail+, r.eckhouse

Starting with this issue, we're introducing a new section to Computer *readers called New Product Reviews. The value of this section lies in the breadth of its coverage. It must appeal to the varying interests of our readers in all aspects of computer systems—software, hardware, and technology. This is a difficult task because it must be directed to both the sophisticated and knowledgeable practitioner, and the generalist wishing to know more.*

Our goal is to provide both the background to and representative samples of specific topic areas. With the help of guest editors drawn from the readership, we will try to cover as broad a range of subjects as possible. Thus, readers are encouraged to submit ideas for columns to me at the address given above so that they might serve as editors for future columns.

A partial list of topics of interest would include new solid-state developments; new releases of operating system, database, transaction, and networking software; vision systems and robotics; CAD/CAE systems; new developments in CAI; and real-time systems, personal computers, microcomputers, mainframes, multiprocessors, supercomputers, and fault-tolerant systems. For this first column, I've selected CAD/CAE as it relates to schematic capture and the production of printed circuit boards.

—Dick Eckhouse

Full functionality, low-cost CAD/CAE systems

Background

One of the dramatic changes that has occurred in the last few years has been in the area of computer-aided engineering and design (CAE/CAD). What I'm specifically talking about is the change in price and performance in schematic capture and printed circuit board (PCB) layout packages. Once the province of large mainframes, these systems have made the transition from mainframes to minicomputers, and now can be found on personal computers. Corresponding to the size change of the host machine has been an almost unbelievable price decline from systems costing several hundreds of thousands of dollars to $15,000 systems running on minicomputers, to systems selling for under $1000 that run on microcomputer engines.

According to market research I've seen, there is an estimated US population of about 75,000 PCB designers. A third of this group is using equipment approaching obsolescence. Another third is still using manual layout techniques. When you add to these figures the enormous number of engineers and technicians who manually sketch out their latest designs with pencil and paper, you come up with a marketer's dream—millions of people ready and waiting for something to fulfill their needs. And this is an audience who can easily afford and justify the low-cost desktop systems currently available in the marketplace.

When you dig further into it, there are really three distinct markets for those requiring the tools to automate the process of going from idea to finished board:

Schematic capture is becoming automated because of the lower cost and greater functionality now available.

(1) Large corporations whose extensive CAE/CAD systems are backlogged and who are seeking low-cost desktop alternatives for capturing schematic designs and prototyping boards;

(2) Medium to small companies who have traditionally not had the resources to buy expensive computers so have hired draftsmen to draw up the schematics and lay out printed circuit boards manually and are now seeking to become automated; and

(3) Entrepreneurs and consultants who can easily set up full-function circuit design bureaus with very little investment.

Schematic capture is becoming automated because of the lower cost and greater functionality now available in carrying out the design from concept to documentation. In the case of PCB fabrication, it is the demand for more circuitry per board that mandates automation because of (1) large component/connection capacity requirements, (2) fine-line design for surface-mount devices, and (3) multilayer capability. Fortunately, the latest offerings, while effectively shortening and simplifying the whole development cycle, are also dramatically offering capabilities not even thought possible heretofore.

The question to be asked at this point is just what are the desirable features that one would like to find in such systems? Not all of them will be available in all packages, but the surprising thing is that many of the same features are common to both the schematic capture and the printed circuit board layout programs. Thus, in the list below I've given an abbreviated statement of them all, without regard to the package in which they appear. The list is organized as to features I consider important from the user's point of view.

- Runs on anything from a PC to a Mac with all clones included. Supports a wide range of graphics standards, printers, plotters, and an optional mouse or tablet.
- Can produce schematics and PCBs as well as other technical drawings.
- Smooth, instantaneous screen panning (if raster) with small step size, or fast redraw with large workspace if vector based. Multilevel zoom plus windowing into sections of the drawing.
- Large libraries of standard objects (TTL, CMOS, microprocessor, discrete components, common device footprint, etc.) that can easily be moved, copied, deleted, mirrored, and rotated with a simple means to generate user-specific parts with the same editing commands used to develop schematics and printed circuit boards.
- Block moves with rubberbanding, deleting, and copying of parts or selected regions. Flexibility in moving labels and part designators around.
- Full range of standard sheet sizes (A to G) that can be personalized. Camera-ready output to printer or plotter plus fast check plots (1X and 2X).
- Different line types, arcs, circles, multiple text sizes (on any PCB layer) in addition to wires, buses, junctions, and components.
- Selectable snap grids with meaningful resolutions that can be turned on/off, or made visible/invisible. Autostep and editing repetition.
- Easily place and remove pads, traces, wires, and lines of varying sizes, shapes, styles, and widths.
- Support for routing of traces (rats' nest, shortest path, vertical/horizontal, L and Z paths, or completely automatic with vias).
- Multilayer (at least solder and component plus silkscreen) support for reasonably large boards (at least 12 × 12). Ability to flip and swap signal layers. Ground and power planes supported from schematic capture to PCB.
- Multiple trace widths with design rule checking for minimum spacing. Area fill/flood.
- Good tutorial and comprehensive manual with index.
- Can develop hierarchical designs. Automatically annotates components.
- Automatically generate related design documents, such as bill of materials, net and wire lists, and reference lists. Perform design rule checking.
- Produce and accept net lists from various schematic capture and PCB layout programs so that design/implementation can be compared and verified. Provide back annotation. Can transfer net and pin lists to other systems such as a circuit simulator.
- Automatically generate solder masks, padmaster plots, drill tapes, and industry-standard Gerber photoplotting.

The question to be asked is, just what are the desirable features?

Note carefully that just because I've chosen to review low-priced systems doesn't mean that they are low-end systems. In all cases, what I've found is that today's systems offer fantastic functionality and ease of entry with quick correction and revision. The bottom line is that they offer the same, if not better, ease of use, features, versatility, and performance as found in higher-priced systems.

Three sample systems

Once the only game in town, smART-WORK from Wintek was the first to break $1000 for a PC package to aid in the design and production of artwork for printed-circuit boards. Now, Tango-PCB from Accel Technologies has started offering a versatile PCB performer for just $495. Its autorouting companion package, Tango-Route, is just out for the same price. Both are available from Accel Technologies, Inc., 7358 Trade Street, San Diego, CA 92121, (619) 695-2000. HiWIRE, from Wintek Corporation, 1801 South Street, Lafayette, IN 47904-2993, (800) 742-6809, is a new offering at $895 from a company well known to users of its PCB package. Each exhibits a majority of the features outlined above and is truly an excellent buy in its category.

Tango-PCB and Tango-Route

What really excited me about Tango-PCB was its rich functionality plus its ability to interface to both the Omation and OrCAD schematic capture programs (reviewed in *IEEE Software* May 1986 and July 1986, respectively). Here was a system offering one-mil on-screen grid resolution (selectable from 1 to 200 mils). No more problems with unusual edge connectors and .156 mass terminations. I could design boards as large as 32 × 19 inches with up to eight layers that included power and ground planes. And best of all, I could use my 256K Deskpro in either EGA or CGA mode (I prefer EGA and strongly suggest it for clearer displays). A hard disk isn't required, but is certainly easier to live with. You also need DOS 2.0 or later.

The first thing I had to become accustomed to was the vector display method. Whenever I zoomed or recentered the screen, the display was redrawn item-by-item, rather than top-to-bottom or side-to-side. This was in contrast to the typical raster systems for schematic capture, but very similar to the mechanical CAD systems I'd used previously. The manual points out the reason for the vector approach—memory space. To get the resolution and board sizes that Tango-PCB supports in 256K on a raster system would require a gigabyte! A nice touch, and typical of the user's manual, to explain such things. Other sections are just as helpful when it comes to producing high-quality output, understanding net lists and photoplots, or just experimenting with adding new elements to the library.

Let me point out that even though Tango-PCB is a vector-based system, it pans and zooms a layout quickly on an XT and would be very fast on an AT or 386, partly because all calculations are done in integer. Not only is it faster, but a numeric coprocessor is not required.

Just because I've chosen to review low-priced systems doesn't mean that they are low-end systems.

As I've said, this is a very rich system, and there are a great many commands for placing and removing pads and traces; moving and rotating components; breaking, moving, swapping, and changing traces between layers; and block moving, copying, and deleting.

The design process involves a number of selection menus called up in the pro-

cess of creating your board. The first is the editor main menu from which you specify the active directory, the file to be edited, and possibly the net list. Each command is selected by pressing the highlighted letter in the command name. Additional commands invoke the editor and allow you to execute DOS commands, save or clear the workspace, choose the library to use or add components to, quit, and so on.

Once you have invoked the graphics editor, additional menus allow you to select the pad sizes and the display options. A multipage on-line help screen is available, and all of the commands can be found on either the quick-reference card or the function key overlay supplied with the system.

Within the editor you can choose from four track widths and eight pad sizes, and pull out predefined components (dips, sips, SMT patterns) from the library. All parts can be placed and moved with the rubberbanding of all connections (i.e., the ability to move and rotate components while maintaining track connections). Frankly, the

Today's systems offer fantastic functionality and ease of entry with quick correction and revision.

variety of options is so great that it will likely take the user a while to take advantage of all of them.

As I mentioned earlier, compatibility with many schematic capture program net-list files makes it easy to go from circuit diagram to routed and verified layout. This is important because you really do want to compare your layout net list with the original schematic net list. Also, if you take advantage of Tango-PCB's ability to "rats'-nest" route from the schematic net list, you will probably need the verify utility to tell you how well you've done in straightening out that mess.

While Tango-PCB can try to optimize the rats' nest for path length while biasing for horizontal or vertical connections, Tango-Route can make this job considerably easier. Routing on a 25-mil grid with a trace width of 12 mils, the system easily routed my test circuit in no time at all while minimizing the number of vias.

I tried two different approaches to perform the routing. In the first, I let the system assume a power and ground plane. In the second, I forced it to route as a two-sided board. In each case, the router was fast and 100 percent successful. If I had wanted to, I could have prerouted or defined "keep-out" areas, as well as specified the number of vias

Compatibility with many schematic capture program net-list files makes it easy to go from circuit diagram to routed and verified layout.

per connection. But what really makes this software "fun" was the dynamic graphical display of the routing process.

An important feature of Tango-PCB is that the user is able to turn on and off layers displayed, and change their colors. After routing, I called up the editor and was then able to change the layout to eliminate some unnecessary paths and vias.

Going back to Tango-PCB, another nice touch is "track stacking." This feature allows the user to repeat complex sequences of tracks, a particularly useful technique when routing memory chips. Rotation of components plus flipping in the X- and Y-directions was convenient if I needed to produce reverse-image boards, but I would have liked a real flip command for each layer.

Continuing with their design goal of ease of use and flexibility, the Accel people really outdid themselves with the print and plot utilities. There is every possible combination of plots possible, including changes in scale, offset, and padmaster sizing to name a few. You can stop the output at any time if you don't like the results. You can make full color plots for checking. Whatever protocol your printer or plotter uses, you can probably use it with this system. It is really impressive.

Finally, the well-written manual explains and teaches about PCB design. For those new to the subject, this manual is essential and goes to great depths to elucidate the subject. There is a section on how to get quality output from pen plotters, a detailed explanation of photoplotter output, a list of over 120 service bureaus providing fast turnaround, separate keypad and command summary overlays, plus a sample board showing various line and pad styles. The only limitation is that there is no index.

The software is not copy protected, but does use a locking device. The device is a connector block inserted in series with the parallel port. Because you may have two (or more if other software requires it), a high quality, free two-foot cable is included so that the devices don't stick out the back of your PC.

If you're looking into a layout and router system, Tango from Accel Technologies is an excellent value. Each package comes with a 30-day guarantee. If you buy both at the same time, you can purchase a combined update service at the price of one ($50). If you'd prefer, either one can be tested in a fully functional demo for $10.

HiWIRE

Wintek has been showing HiWIRE for a little while now, and its style will be very familiar to users of the smARTWORK PCB layout package. Since a thorough review of smARTWORK can be found in *IEEE Software*, May 1986, I won't cover it in detail again here. However, let me point out that smARTWORK has gone through a number of revisions that have added many new and versatile features. Among the ones I like best are (1) a revised manual; (2)

smARTWORK has gone through a number of revisions that have added many new and versatile features.

more trace widths and pad sizes; (3) Swap and Flip commands; (4) automatic mouse installation; (5) the ability to define sites, pins, and terminals; and, best of all, (6) an autorouter. All of these changes have not resulted in an increase in the price of the package ($895), and registered users can upgrade at any time for $20.

HiWIRE is one of the only CAD/CAE programs that I've seen that makes really effective use of the CGA graphic mode. Indeed, I found myself preferring CGA over EGA mode. A large number of users will find this a real plus. With HiWIRE, a mouse is essential; you cannot place and draw objects without one. I hope they change this in future versions because there are times when it is easier to use the cursor keys.

I used HiWIRE with smARTWORK to produce a simple test circuit. In no time at all I had read the manual, drawn the schematic complete with pictorial diagram, and laid out the PCB. It took a little longer to understand the Check program for verifying the schematic to be the same as the PCB layout. The end result was a very professional product complete with bill of materials. One thing I did not have time to check out, but is supported, is the excellent hierarchical design capabilities of the system.

HiWIRE is one of the only CAD/CAE programs that makes really effective use of the CGA graphic mode.

Turning to the details of the package, HiWIRE is a very easy to use system. I have always been impressed by the people at Wintek because their systems do not overwhelm the new user, and the excellent manual, complete with tutorial chapters, gets the user up to speed in very short order. Also, the way the menus are set up, and the mouse used, makes for a very efficient human interface. And let me note that this is also a versatile CAD system that can be used for a lot more than just schematics, particularly with its very large drawing area (approximately 30 feet square).

HiWIRE is a menu-based system, but the menu only appears when you press the left mouse button. In this way valuable space is not lost as it would be on a typical CGA system where the menu is constantly displayed. You drag the mouse over the list of menu items, releasing it when you wish to select either a conductive object (wires, buses, joins, and components) or nonconductive object (lines, boxes, arcs, circles, and text). Additional menu commands allow you to copy, move, delete, and change objects. Such objects change color to confirm their selection. Two second-level menus, called Ctl and Etc, turn on and off display coordinates, grid snaps, zoom in and out, spin and flip objects, undelete, and so on.

The connection rules for conductive objects are easy and very intuitive.

There are over 700 parts in five libraries, plus a library of predefined borders. While this may not seem like a large number of parts, it is, because the spin/flip commands make it unnecessary to store all configurations of any given part. Also, the simplicity of changing a component, say from the pinning of the A section to the C section on a quad OR gate, makes it unnecessary to keep more than just the A configuration. Users who don't have lots of disk space will appreciate this design feature. Also, by using the schematic editor recursively, the same commands are used to edit components as to lay out the schematic. Thus, the user isn't forced to learn a new set of object editing commands.

Text can be created in nine sizes, with independent control of height, width, and one of four rotation angles. Text can be visible or invisible (for instance, when you don't want to show pin numbering on a connector). Labels are placed like other components, and can be manipulated with the same commands. A nice feature is that by calling up the editor on a component you can change the placement of its reference designator or its identifier, in case they get in the way of other items being drawn. Changing the reference designator is just as simple, but an automatic annotation utility would be a nice feature to have.

There are a couple of other features that would make the system a little nicer to use. One is a continuous update of the coordinates being displayed, rather than having to wait until you stop moving the mouse. Another is the ability to select the colors being displayed. Also, there seems to be no way to stop output to the printer if things go awry. Redrawing of the screen as you pan is a bit slow. And finally, snap works a bit differently than other CAD programs in that the cursor moves freely and only the component is snapped to the grid.

Output to the printer is very fast if you don't use the feature that optimizes the quality of the plot. It is just like Wintek to offer such a feature because of their perfectionist attention to detail. Included in the chapter on printing and plotting is a section on troubleshooting and suggestions on how to improve the plot output. The print and plot commands are Unix-like with their optional parameters. The numerous parameters control the plotter type, paper and drawing size, output file name, and baud rates.

One nonstandard feature is the net-list output vis a vis other similar schematic capture programs. With HiWIRE you have a simple list of sites and pin numbers without all the parentheses. You could easily convert HiWIRE's format to that used by other programs, but it is not necessary if you are using smARTWORK with it. smARTWORK offers several net-list and check programs to verify the PCB and schematic representations. Finally, HiWIRE offers an EDIF format utility as well, but I didn't have any software to check it out for compatibility.

The connection rules for conductive objects are the easiest I've seen, and very intuitive. They don't limit you as to the number of wires that can come together at a junction, and don't require you to place a dot at that point. In this fashion they are much less limiting than the rules required in other similar packages. This is a real plus and eliminates many seemingly useless design rule errors (such as too many wires connected at a point).

HiWIRE is by far one of the easiest-to-use schematic capture programs.

Unlike smARTWORK, HiWIRE uses a hardware rather than a software lock. The lock device goes in line with your parallel printer port. This makes it easier to use than smARTWORK in that you don't have to have a key disk.

Wintek has been the leader in offering systems with a 30-day money-back guarantee, and HiWIRE also comes with this generous offer. The software requires 320K, two disk drives, parallel printer port, mouse, and DOS version 2.0 or later.

I recommend this system even though it is priced a bit above other systems I have evaluated. HiWIRE is by far one of the easiest-to-use schematic capture programs, and it is very flexible and versatile. It integrates nicely with smARTWORK and means that you can purchase a one-vendor solution to the CAD/CAE problem. Also, user support from Wintek has been excellent, as demonstrated by their continuous revision history. This last point is very important, since it shows how responsive Wintek is to its users' needs.

Richard H. Eckhouse, Jr.
New Product Reviews Editor