

The following is a more detailed description of several ACCEL EDA key features.

◆ **Complete Windows Implementation and Unique Selection Tool**

ACCEL EDA runs under the Windows 3.x, Windows 95 and NT platforms, and also offers network support. Within the Windows interface, our implementation of the *Select* tool is unique; it can be used to execute five major functions: you can *Query*, *Move*, *Copy*, *Delete*, and *Modify* practically anything on the screen. It is completely context-sensitive. Double-click an item to directly view and edit its properties. Shortcut keys speed creation and editing; for instance, just press the Ctrl key, then click and drag an item to create a copy!

The *Select* tool can be used to sub-select items as well. For example, sub-select to quickly change a component's pad, move a RefDes or wire name, and delete copper islands. ACCEL EDA supports *Cut and Copy* to other Windows applications. The power is not only in the amazing ease of use ACCEL EDA brings to the customer, but the incredible increase in productivity as well.

◆ **DBX - ACCEL's Database Exchange (API)**

Customize! Finally, a true application programming interface (API) for the PCB design world. DBX allows ACCEL, our VARs, and our customers to customize the design environment in nearly every conceivable way. DBX gives the user full control over the database, while the design is active. With these provided utilities, a C, C++, or Visual Basic program allows the import and export of data to and from the ACCEL EDA PCB database. In the Schematic package users can export information. Many valuable "sample" DBX applications are provided, ranging from basic data extraction and modification (Attribute Report, Duplicate Reference Designators, Circular Array Placement) to very complex programs allowing the user to analyze data integrity and performance (Signal Integrity Analysis, Test Point Sites, Routability, Pattern Wizard). Source code is provided for many of these for user modification. DBX is the tool for third party developers to integrate other applications directly into our environment. This is an extremely powerful toolset.

◆ **Integrated Libraries**

Correct by Design. Our fully integrated library allows for true forward and backward annotation (*ECO Processing* - process independent) between the Schematic and PCB. One schematic symbol is mapped to each PCB pattern and all the packaging information is included in this definition. *Cross probing* between the schematic and the PCB is possible because of this integration. Thus a part on the PCB can be selected and highlighted and in the schematic all the gates that are included as part of the component are instantly highlighted. The ACCEL Library Manager provides users with an easy to use toolkit for the creation and modification of schematic symbols and PCB part patterns as well as the ability to manage the packaging information. Integrated libraries are unique to ACCEL EDA and guarantee *correctness by design* throughout the design process.

◆ Rules-Based Design

Design rules drive the design. Users can establish design rules in the Schematic and pass them forward to the PCB and autorouters in the form of net attributes. Modified design rules in the PCB can also be back-annotated to the schematic. Attributes can be user defined, or system-supplied such as routing line widths, clearance values, component height, etc. Component attributes can also be applied to assist with simulation, part placement, thermal analysis, clearance checking, and purchasing. Additionally, all CCT design rules are supported and can be assigned at any stage in the design process. Nets can also be grouped into *classes* and these classes can have complex rules assigned to them as well. This helps speed data entry since many nets can belong to a class and just one rule can be set for that entire class. These rules can even be assigned to govern copper pour backoffs. Net attributes such as net-net and class-class clearances are fully supported by our on-line DRC capability. ACCEL EDA allows you complete control so that designs come out right the first time.

◆ Mixed Signal Support

Unique features for today's complex mixed signal designs. The *split plane* feature of ACCEL P-CAD PCB greatly simplifies the assignment of multiple nets on power and ground planes, commonly required in mixed mode design. Split plane areas are defined as negative polygons and may be distinguished by unique colors. Pins carrying the split plane area's signal are automatically tied to the plane, saving considerable time and eliminating errors. In fact, the connections are recognized for complete *design rule checking* (DRC). Additional shielding can be added to signal layers with the *copper pour* tools. The shielded area can be defined as solid or patterned with ties to a signal. The copper pour can have automatic island removal based on island size or whether they are internal to the copper pour boundary. *Blind and buried vias* are fully supported and via and pad holes can be graphically shown for true DRC hole checking. *Layers* can be added, modified, and deleted at the users request. Layers can be grouped into *sets* and turned on and off with keyboard hot keys. ACCEL EDA provides powerful tools to complete the most difficult mixed mode designs.

◆ High-Completion Autorouters

Autorouters to complete the job - fast. Our interactive router, dubbed *InterRoute*[™], avoids obstacles, provides copper hugging, and honors all routing, net, and class clearances. Users can guide the route's path or allow the tool to choose a path. As expected, vias are automatically placed when the user changes the layer. Redundant tracks are removed when backtracking over an existing trace, eliminating time-consuming cleanup.

ACCEL PRO Route is a high-completion autorouter available as an option for ACCEL P-CAD PCB and ACCEL Tango PCB. This router is seamlessly integrated into the layout packages. It features automated operation and set-up, rapid iteration to 100% completion, and manufacturing cleanup passes. Special handling for wide traces and surface mount devices, plus 45-degree diagonal and T-routing connections effectively accommodate a wide range of design types. ACCEL PRO Route supports blind and buried vias, outputs a comprehensive statistical report, and generates a backup file. ACCEL PRO Route 2/4 is the same router limited to designs of 2 layers, or 4 layers and 4000 pins.

◆ High-Completion Autorouters (contd.)

ACCEL EDA is also tightly integrated with Cooper and Chyan's high-completion SPECCTRA autorouters. These optional shape-based products are accessed directly from the PCB menu. A "Do Wizard" assists in complete and correct router set-up. Like all attributes, those supported by the SPECCTRA routers can be defined by the engineer when the schematic is created. They are transferred to the layout and automatically picked up for use during routing. Advanced options are available for supporting high speed and hybrid designs, and improving a design's manufacturability. CCT's SP2 and SP4P reduced-capacity and interactive routing tools are also effective ACCEL EDA companions.