

ACCEL EDA merges the best features of Master Designer, TangoPRO, and significant new features. This document describes the general capabilities, new features, and enhancements in ACCEL EDA from a TangoPRO V3.0 perspective.

New Features:

- 1. Platform Support.** ACCEL EDA is a native Windows 3.1 application that runs under Windows 95 and Win-NT on Intel hardware platforms.
- 2. Documentation.** Documentation is provided in hard copy and on CD-ROM.
- 3. Net Classes.** Net classes may be named, saved, and modified. DRC checks clearances using hierarchical inheritance rules for class-to-class, net, net class, and design level precedence. Net classes and clearance values are passed to the CCT router. This feature closely coupled with the ability to attach attributes such as clearance and width to the PCB expands design control and communication to new levels.
- 4. Layer Sets.** Sets of layers can be defined to facilitate setting up printing, Gerber, NC Drill, DXF OUT, and active display layers. Sets can be recalled using hot keys. Coupled with Pad and Layer Ordering, this feature enhances and speeds editing and output processes.
- 5. Move Component By RefDes.** A component, identified by its Ref Des, is moved to the current cursor location for placement.
- 6. Split Plane Support.** Power and ground planes may be split for multiple net assignments. Splits are defined by polygonal areas and may be given unique colors. Pins are automatically tied and DRC supports the connections. Draws, prints, and photoplots in the negative.
- 7. Interactive Routing, InterRoute.** This interactive shape-based, assisted routing tool avoids obstacles, provides copper hugging, and honors all routing, net, and class clearances. Guide the route's path or allow the tool to choose the path. Vias are autoplaced when there is a layer change. Removes redundant tracks ("backtracks").
- 8. Dimensioning.** Basic dimensioning capabilities including baseline, point-to-point, center, leader, radial, diameter, and angular dimensions. Enhances the ability to completely document the design process and pass this information to the manufacturing facilities.
- 9. Auto Save.** Design files are automatically saved. The user specifies the number of versions to maintain and the time interval between saves.
- 10. (SCH) True Type Fonts.** TrueType fonts are available for all text on Schematic sheets.

Enhanced Features:

- 1. DBX.** ACCEL's powerful application programming interface, DBX (for DataBase eXchange), is extended to include *PCB-IN* and *SCH-OUT*. DBX provides what is believed to be the most open access to an EDA database in the market place. By adding supplied DBX function calls to a C, C++, or Visual Basic program it is possible to extract data from, or add data to, an active design. Using DBX an end user can customize their ACCEL design system in limitless ways.
- 2. User Interface.** ACCEL EDA has a "modern" Windows look-and-feel including chiseled buttons and menu tabs. Changes are geared towards productivity and include:
 - reducing keystrokes required for common actions

- direct editing from the status line
 - keyboard co-ordinate input for lines, wires, and component placement
 - user-assigned shortcut keys
 - context-sensitive right-mouse menu and dialog boxes
 - double-click for object properties dialog
 - object “sub-selection” (e.g., to select a component’s pads, net’s name, copper pour’s islands)
 - user-specified file viewer
3. **Object Properties.** A context-sensitive *Properties* command replaces Query/Modify for all key data. Multiple object-oriented access methods (right mouse, Edit menu, double clicking an object) offer comprehensive design data, editable fields, and direct access to pertinent properties. Accessing and changing information is done in a Windows standard format and modality.
 4. **Cut/Copy.** Graphics are cut to the clipboard in bitmap form (Windows Metafile format) for pasting in other Windows programs such as Word, AutoCAD, FrameMaker, etc. Facilitates design documentation.
 5. **Attributes.** Improved creation, editing, and *display support* for component, net, and design level attributes. Changes in handling PCB Global vs. Local components and Schematic Part vs. Component attributes. The expanded ability to pass all attributes from the front end vehicle to the physical layout package redefines “control”.
 6. **Copper Pour.** Copper pours now include *island detection and removal* based on island size or interior placement. An island can also be sub-selected and interactively deleted or excluded through the use of a Polygon Void. A connection line is automatically placed when an island is not connected to the net. This “connectivity-awareness” carries through to DRC. Traces added after the pour has been created are automatically plowed. The back-off distance from traces, pads, and vias optionally follows design rule clearances or is a fixed value. Three levels of smoothness are supported in defining curved edges in the pour.
 7. **Pad/Layer Ordering, Blind/Buried Vias.** *Layer order is supported and presented visually.* Pad and via definitions include visual representation of hole range using a virtual three dimensional slice through the physical design. DRC, NC drill, and Gerber functions appropriately use the layer order and the hole range. These features, tied with the ability to define and recall layer sets, aids customization and provides speed enhancements for editing and output activities.
 8. **(Library Manager) Library Management.** Simultaneously view all component data in multiple dialogs to facilitate component creation and editing. Includes graphic views of the pattern and symbol, a spreadsheet view of pin information, and a dialog box of component data. Pins and pads are highlighted, and spreadsheet rows are selected to match selections within any of the browse windows. Much of the data is filled in automatically. Spreadsheet editing was simplified. Errors can be checked before the component is saved.
 9. **(SCH) Additional Bus Connector Styles.** Three bus connector styles are available. Styles may be set as a default and modified for individual connectors.
 10. **(PCB). DXF Z-Axis Component Support.** A predefined component height attribute can be added to the design and output as an extruded component outline in DXF format (DXF-Out).