

Download AD 17 [trial here](#).

What's new?

- ActiveRoute™

ActiveRoute is a ground-breaking interactive routing technology. It routes using the design rules for widths and clearances by net, net class, rooms, and layers. It does not add any vias because it uses a River Routing approach. Therefore, it is imperative to fanout the design first before routing on inner layers. It is fast, really fast. It has a focus on quality, both subjective and objective quality.

It is very important that users follow the instructions in the technical documentation. If they don't, the routing will fail.

- Draftsman improvements

- Support for Ordinate (single reference) dimensions
- Implemented a Custom Table
- Dual Unit capability for Dimensions and for global Document Options
- New way to represent components in the Assembly View (Body Projection Outline)
- Ability to enable separate colors for SMD and through-hole pads on an Assembly View

- Back drilling

Back drilling is a technique used to remove the unused portion, or stub, of a copper barrel from a thru-hole in a printed circuit board. When a high-speed signal travels between PCB layers through a copper barrel, it can be distorted. If the signal layer usage results in a stub being present, and the stub is long, then that distortion can become significant. See the technical documentation for more information.

- Smart Copper Editing

Smart Copper makes the exercise of creating and editing copper less intensive and more user-friendly. Now it is possible to combine, split, and edit existing polygons by extending it. New modes of editing apply to all objects based on polygons - polygons, regions, and the boundaries of the circuit board. See the technical documentation for more information.

- Other improvements include:

- Selection & Display improvements
- New Project Releaser
- New Post-route Glossing Tool
- Design Rule Improvements
- Dynamic Cross Probing and Cross-Selecting
- Layer Stack Management Enhancements
- Parameters in PCB Footprints
- Improved Cross Selection
- New Pick and Place Generator
 - Additional Enhancements and Improvements
- Improved Outjobs to support PDF/A for use as an Output Container
- Improved Undo-Redo functions
- Added support to automatically register all exporters and Save As formats as Export Outputs
- Clicking the DRC report will bring you to PC

- Includes Popular BugCrunch Fixes

- #9676 Added a lasso selection tool for use in PCB/Schematic documents (BC:962).
- #12051 Added a new Pick and Place generator to select pick and place options before generating output (BC:999;

BC:2067).

- #12092 Improved Smart copper editing to include options for merging and subtracting polygons, as well as the addition of improved editing (BC:1295).
- #9649 Improved PCB STEP with new option to export as a single part or as one model (BC:5582).
- #9476 Added support to automatically register all exporters and Save As formats as Export Outputs (BC:1324).
- #11919 Improved SCH component parameters, so they now synchronize with PCB footprints (BC:477).
- #12213 Fixed a bug causing Flip component command to not work correctly for test point properties (BC:2971).
- #9183 Improved BOMs exported to PDF, which now contains page breaks for increased readability and printing (BC:58).
- #13769 Improved the method to modify/edit/finish polygon borders (BC:1031).
- #10341 Improved Un-routed net rule with a new option to 'Check for incomplete connections' (BC:1119).
- #10338 Improved Primitives Clearances with the addition of an Add Hole constraint (BC:5744).
- #7937 Added Zuken CR-5000 ASCII format PCB, schematic, and library files support for Import Wizard. (BC:1467).