



Overview

OrCAD® PCB Productivity Toolbox provides a suite of utilities focused on improving your PCB designer’s productivity by enabling or streamlining many design tasks that are common or emerging today. The toolbox utilities are integrated directly into the OrCAD PCB Editors GUI and command structure, providing little to no impact to the designer’s use habit of the tool, while delivering measurable benefit.

Features

While varying in their capabilities, the utilities in OrCAD PCB Productivity Toolbox focus on improving existing use models and reducing designer input/interaction.

Highlights

- Comprehensive collection of design capability and designer productivity features
- Seamlessly integrated within OrCAD PCB Editor
- Available for OrCAD PCB Designer Professional and Standard users
- Improve PCB Designer efficiency and productivity, while reducing design time

Polar Grid

The Polar Grid utility provides the ability to quickly and easily manage component placement and interconnect routing for circular boards, as shown in Figure 1. The polar grid utility is comprehensive and can be considered mandatory for designs that require this type and style layout.

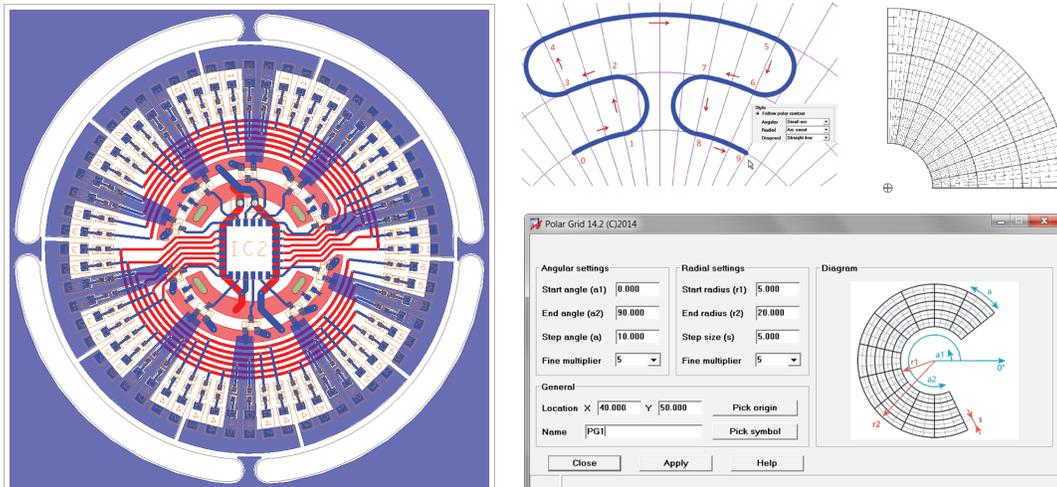


Figure 1: The Polar Grid utility provides a set of functions that are very useful for boards where circular placement and routing is required

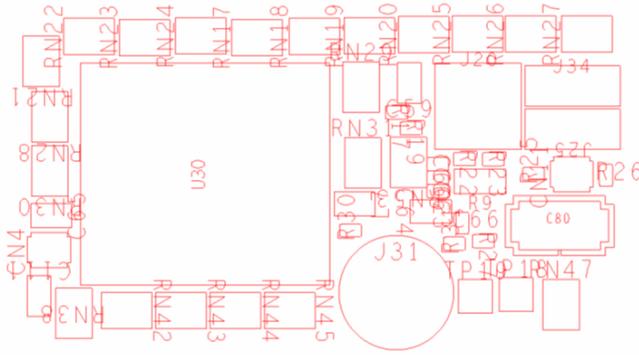


Figure 2: Typical label situation during PCB layout

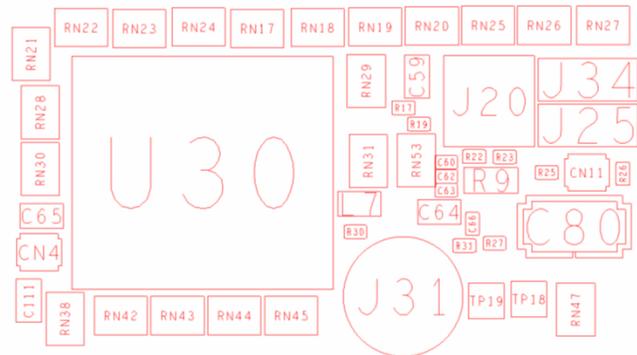


Figure 3: Result after running the Label Tune utility

Label Tuning

The Label Tuning utility provides auto-silkscreen component-name location/rotation optimization. It takes the unreadable label situation shown in Figure 2, and based on your settings, can auto-create the manufacturable version shown in Figure 3.

Toolbox Features

Other highly useful and productive utilities include:

- Assign Net to Existing Via — Changes the signal or net name on an existing via without deleting and re-adding the via
- Cross Copy — Copies elements to a different layer while keeping their original features
- Barcode — Creates a barcode image of the design part number or other user defined information
- Drafting Utility — Creates shapes that are not based on a grid but on intersections of other objects
- Batch Plot — Creates a defined document batch as a single process for printing as a PDF
- Padstack Finder — Locates and highlight padstacks in design by color, separated by component pins and vias
- Post Processing — Creates post-processed manufacturing data in a set order that can be saved and easily reproduced to match manufacturers' requirements

- Color by Netclass — Color spacing net classes for review and documentation,, useful for high-speed or high voltage/current designs
- PCB Library Plot — Documents all the PCB footprints in a library and prints based on a selected template
- Quick Symbol Edit — Locally edits/modifies design-specific symbol
- Shape Utilities — Provides a powerful set of Boolean operations (OR, AND, ANDNOT, XOR) for use on shapes, including resultant shape scaling

For the latest product or release information, visit us at www.orcad.com or contact your local Cadence Channel Partner.

Sales, Technical Support, and Training

The OrCAD product line is owned by Cadence Design Systems, Inc., and is supported by a worldwide network of Cadence Channel Partners (VARs). For sales, technical support, or training, contact your local channel partner. For a complete list of authorized channel partners, visit www.orcad.com/CCP-Listing.