

# PCB Clustering for OrCAD

## Enhanced clustering, placement and design reuse technology

### Accelerate the Component Placement Process

As designs become more complex and project timelines more compressed, it's important to identify opportunities to enhance design processes. PCB Clustering™ for OrCAD® provides AutoClustering™ technology, intelligent design (IP) reuse, and replication technology that can significantly reduce board placement time.

Component placement has typically been a manual and tedious process, accounting for a significant portion of the design cycle. Engineering teams have struggled through these long placement cycles due to the impact correct placement can have on routability, signal quality, and overall design closure.

PCB Clustering for OrCAD enables a more efficient and effective PCB layout and placement process through intelligent clustering, replication, and reuse technologies. With PCB Clustering designers can deliver higher quality results than a manual approach in a fraction of the time.

### Seamless Integration with Cadence Technologies

PCB Clustering for OrCAD resides directly in the OrCAD PCB Editor canvas. Once installed, a new menu item enabling access to the PCB Clustering technology is available through the OrCAD user interface. As you use PCB Clustering, all changes can be automatically applied to the OrCAD PCB Editor database. This avoids any translation or synchronization issues that can occur when using separate programs.

### Auto-Clustering

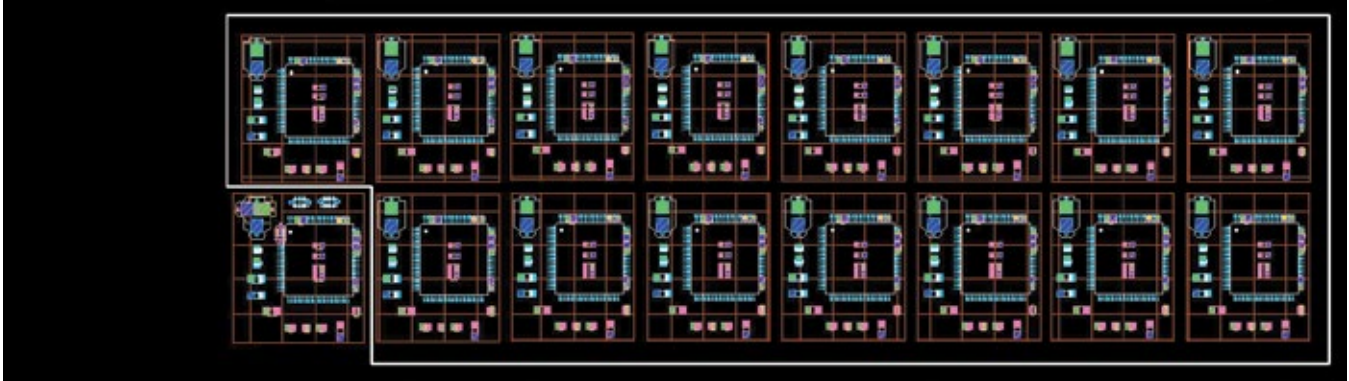
Related components are typically grouped together in the schematic. PCB Clustering can leverage this information to auto-generate tiled placement clusters. These clusters can be based on hierarchy, pages, reference designator prefixes/suffixes and/or ROOM properties defined in your schematic. This allows designers to get a quick logical grouping in the physical realm, enabling faster placement.

### Highlights

- Create functional groups of components based on design information
- Cross-Probe between PDF schematic and OrCAD PCB Editor
- Create multiple replicas of a source cluster's net topology
- Propagate cluster membership, placement and shape changes to specified clusters
- Save a design checkpoint at any time and compare it against other checkpoints
- Create a cluster from a specified group of components
- Create templates containing a cluster's membership, net topology, and placement information
- Propagate cluster reference designator text locations to target clusters
- Reuse design templates in new or legacy designs
- Perform early 'what-if' spatial analysis using generic circuit templates

### Automatically Replicate Cluster Configuration

Once you have defined your clusters, PCB Clustering can automatically replicate that configuration using its matching algorithm by finding other like circuits in your design. This innovative bottom-up approach allows for partial match configurations and works regardless of component orientation. This greatly speeds up the completion of boards with replicated circuitry or channels. The user need only define a circuit once and PCB Clustering will apply that definition to all relevant matches.



### Define Templates and Design Reuse

Clusters can be saved for reuse as a template that contains net topology, placement information, and etch. Templates can be referenced and used in current and subsequent designs. Creating a library of templates allows the design team to raise their level of abstraction by focusing on placement by template (group of components) as opposed to one part at a time. Templates are saved in a generic abstraction, without regard to reference designators or netnames.

#### Key Benefits

- Shortens design cycle timeline
- Raises design abstraction from individual devices to functional blocks
- Provides a flexible design reuse environment
- Helps avoid costly errors by ensuring replicated circuitry is in synch
- Works seamlessly with existing design methodologies
- Improves communication between EE and Layout Designers

### Create Hierarchical Clusters

Clusters can be hierarchical, allowing cluster within blocks of functionality as a cluster while maintaining individual pieces that can be modified, updated, and replicated as needed.

### Cross-Probe between the Layout and Schematic

Access to the logical design makes the PCB design much easier. Often the PCB designer will only be given a PDF view of the schematic which does not enable intelligent communication between the logical and physical design. PCB Clustering for OrCAD provides bi-directional cross-probing functionality between a PDF schematic and the OrCAD PCB Editor canvas. Users can select components in the PDF and have the corresponding footprint highlighted in the layout and vice versa. You can also create clusters based on components selected in the PDF.

### Modify Clusters and Propagate Changes

Placement and components often change as a design progresses. PCB Clustering allows you to modify your clusters and propagate those changes to all replicated clusters in your design. This enables a 'change once and apply where used' methodology which ensures your design stays in synch with the desired intent and avoids costly errors down the road.

### Sales, Technical Support, and Training

For more information, please contact EMA Design Automation, a Cadence channel partner.

877.362.3321  
 info@ema-eda.com  
[www.ema-eda.com](http://www.ema-eda.com)