CircuitSpace

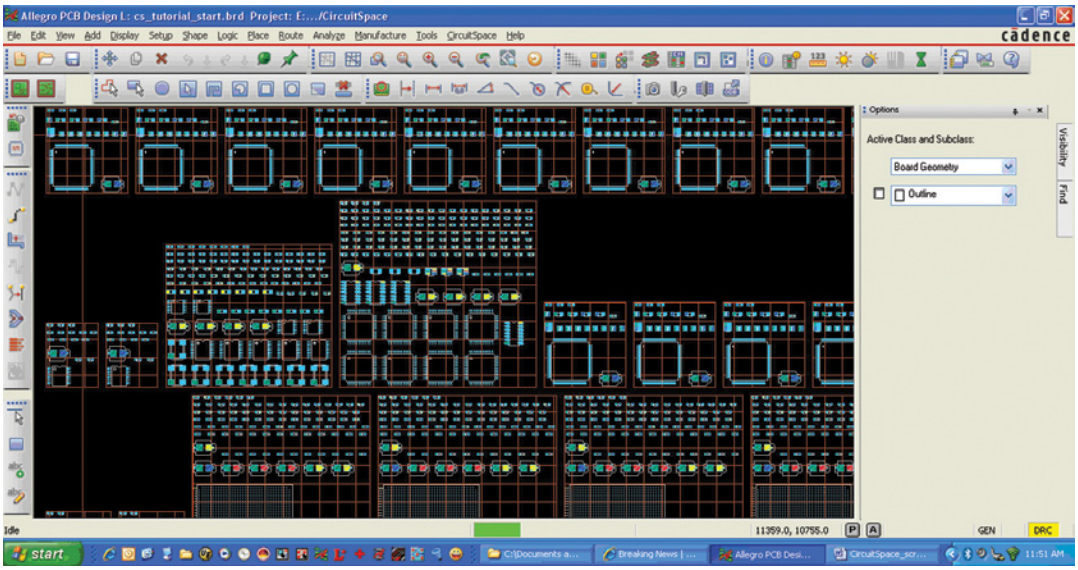
Enhanced clustering, placement and design reuse technology

Accelerate

CircuitSpace Key Features

- AutoClustering - create functional groups of components based on design information
- Cross-Probing - bi-directional communication between Cadence Allegro and OrCAD PCB Editor and (PDF) schematic
- 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

As designs become more complex and project timelines more compressed, it's important to identify opportunities to enhance design processes. CircuitSpace® AutoClustering™ technology, intelligent design (IP) reuse, and replication technology can reduce board placement time from weeks to minutes.



CircuitSpace automatically gathers components into interconnected groups called clusters. A rough placement of the cluster's components is then performed, simplifying a designer's task of manual placement.

Accelerate the Component Placement Process
Component placement has typically been a manual and tedious process, accounting for weeks to months 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.

CircuitSpace provides a solution to enable a more efficient and effective PCB layout and placement process through intelligent clustering, replication, and reuse technologies. With CircuitSpace designers can deliver higher quality results than a manual approach in a fraction of the time.

Seamless Integration with Cadence Technologies
CircuitSpace resides directly in the Cadence® Allegro® and OrCAD® PCB Editor canvas. Once installed a new menu item enabling access to the CircuitSpace technology is available through the Cadence user interface. As you use CircuitSpace all changes can be automatically applied to the Allegro 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. CircuitSpace can leverage this information to auto-generate tiled placement clusters. These clusters can be based on hierarchy, pages, and/or ROOM properties defined in your schematic. This allows designers to get a quick logical grouping in the physical realm, enabling faster placement.

CircuitSpace

Easily replicate user defined clusters

CircuitSpace Key Benefits

- Shortens design cycle timeline by weeks
- 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

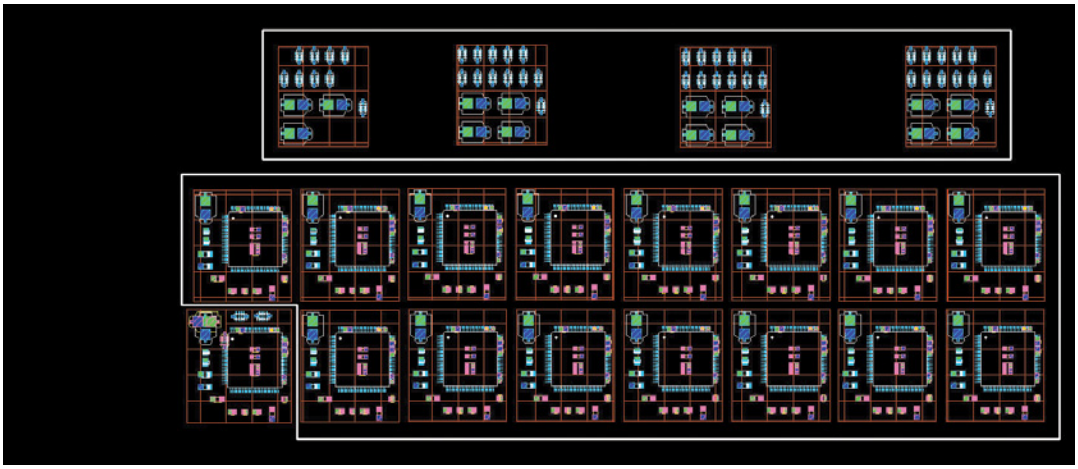
For More Information

For sales and pricing information contact EMA Design Automation, a Cadence Channel Partner.

EMA Design Automation, Inc.
225 Tech Park Drive
Rochester, New York 14623

Phone: 585.334.6001
Fax: 585.334.6693
eMail: info@ema-eda.com
Web: www.ema-eda.com

CircuitSpace® seamlessly integrates with Cadence® Allegro® or OrCAD® PCB Editor and allows users to achieve board layouts in a fraction of the time it would take to complete by hand.



CircuitSpace instantiates source cluster's modified typology, placement and cluster shape changes to targets.

Automatically Replicate Cluster Configuration
Once you have defined your clusters CircuitSpace can automatically replicate that configuration using its matching algorithm by finding other like circuits in your design. CircuitSpace's 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 CircuitSpace 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, reference designators and/or netnames will not need to be maintained from one design to the next.

Create Hierarchical Clusters
Clusters can be hierarchical, allowing cluster within cluster configurations. This enables the designer to define whole 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 times 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. CircuitSpace provides bi-directional cross-probing functionality between a PDF schematic and the Allegro or 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. CircuitSpace allows you to modify your clusters and propagate those changes to all like replicated circuits 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.

cadence™ CHANNEL PARTNER

EMA Design Automation™ ema-eda.com