

Allegro Quick-Place Utility

Pre-Placement Strategy

-Venkata Ramanan

Objective

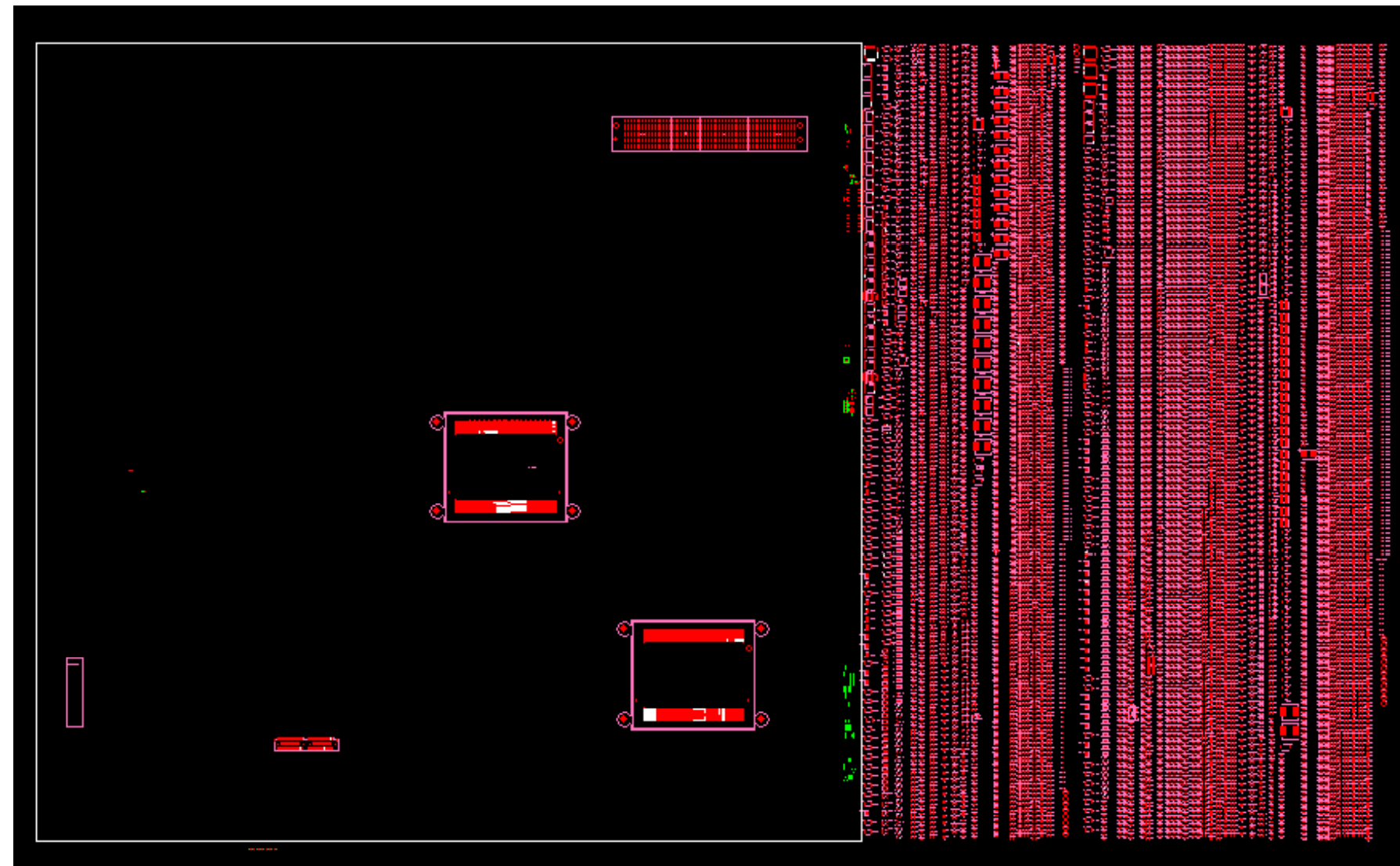
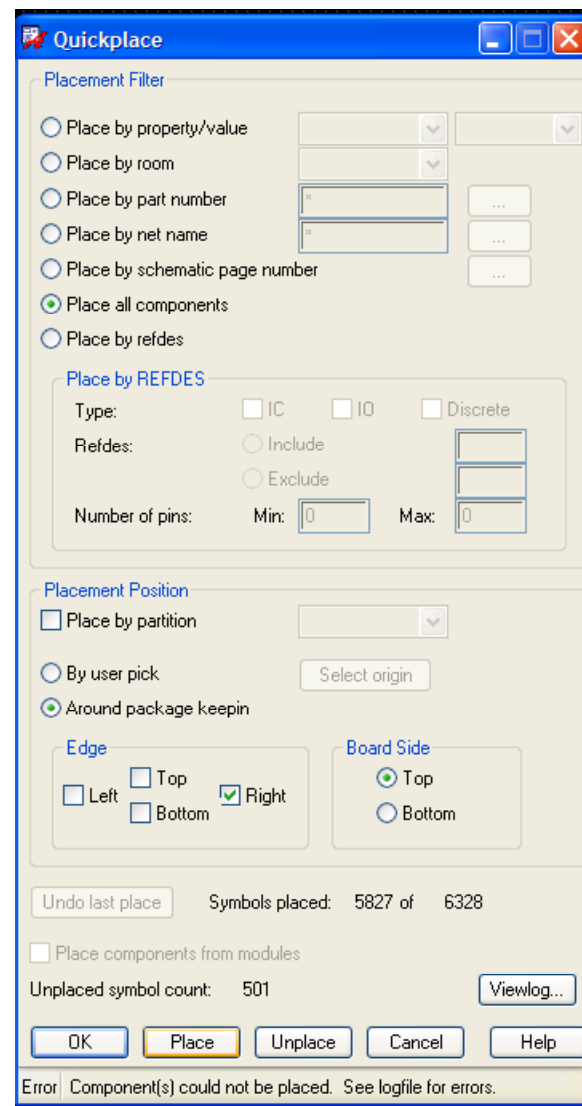
- Scope of this Presentation
- Traditional Placement Method
- Disadvantages of Existing Method
- Introducing QuickPlace Tool
- Functionality
- Limitations and FAQs
- Summary

Scope Of This Presentation

- This presentation lists the current method of pre-placement on PCB designs
- Explains the pitfalls and drawbacks
- Introduces the new QuickPlace Tool developed at Infinera
- Explains the Functionality and the Rational
- This tool will be used by PCB Designers and Board level Engineers

Traditional Placement Method

- Cadence Allegro Provides a Quick place tool which will place all the components to any side of the PCB outline



Disadvantages of Existing Method

- A typical large scale design at Infinera has multiple Hierarchy and numerous sub-designs
- It is quite common to have more than 150 pages of schematics
- Any given component on a particular page has an underlying requirement to be placed near its circuitry
- This requirement drives the Engineer to group them by pages and by connectivity
- It is highly impractical to assign ROOM property to all circuitry in all the pages

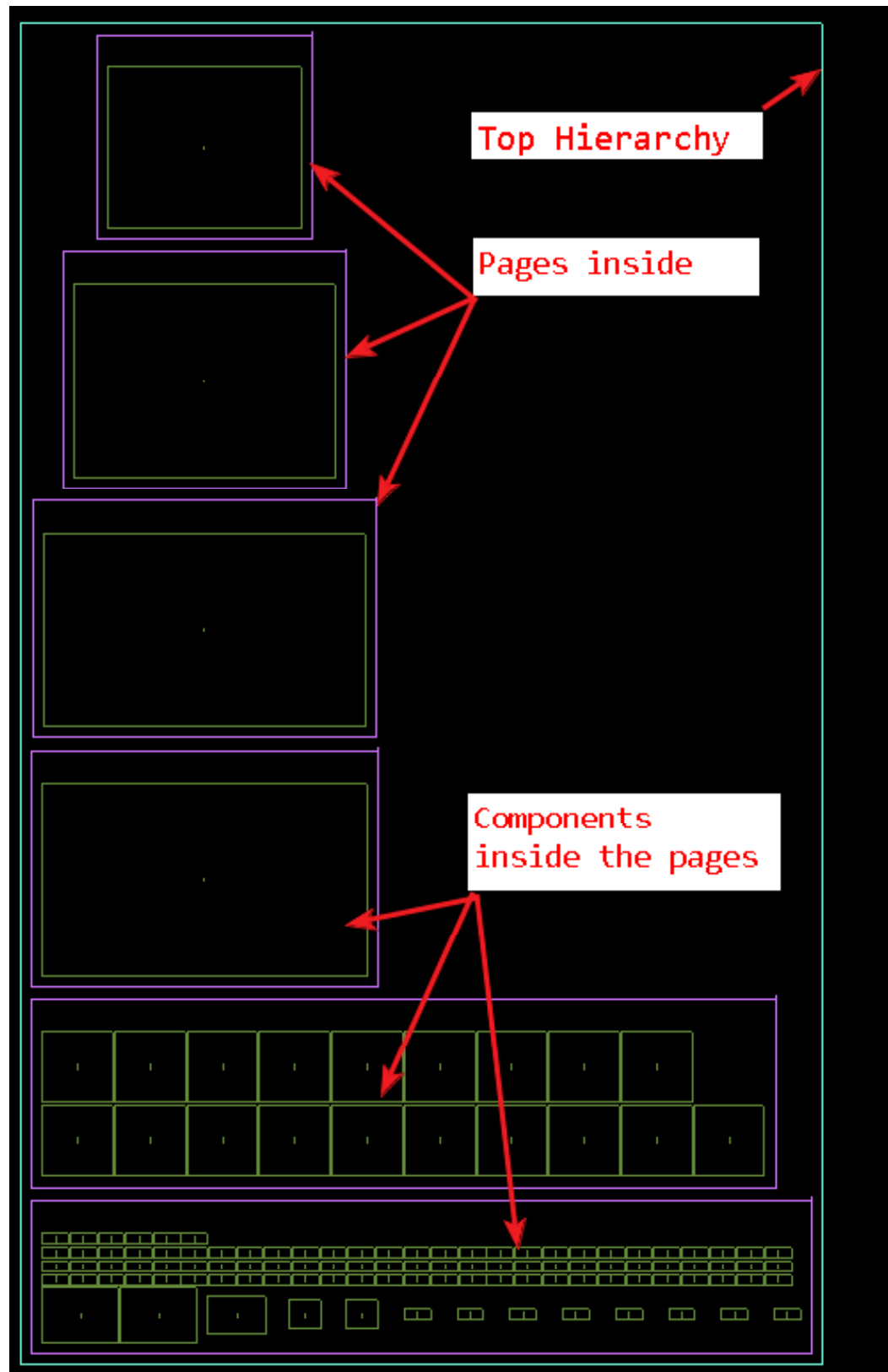
Disadvantages of Existing Method - cont

- A typical large scale PCB layout starts by placing all the components on the side of the board outline
- Then either with the help of a hard copy of schematic or by using Concept the components are grouped by page by page
- For a component count of 5000 plus this could take days before this initial grouping is done
- After placing them there is no easy way to track or review a group of components from a the same circuitry from schematic to Layout
- It is difficult to identify the parent page of a given set of components

Introducing QuickPlace Tool

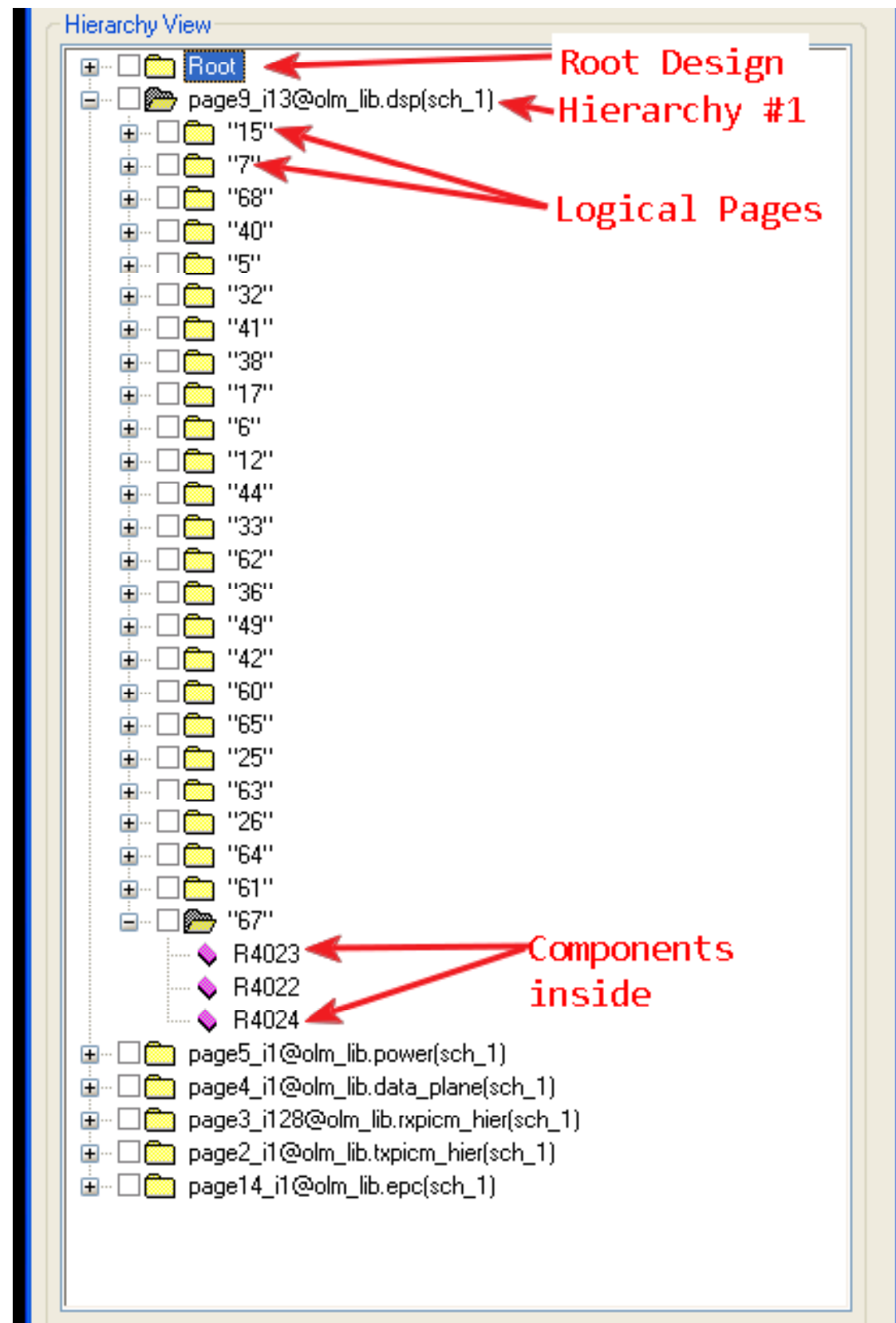
- By Using the Skill language a QuickPlace tool has been developed in Infinera
- This will expedite the Initial Placement grouping and will cut-down days of work to minutes
- This tool also will provide an interface similar to the Concept HDL's Hierarchical View
- This will help in highlighting a particular Page/Hierarchy or a single component on the Layout with respect to the Parent Schematic

Functionality



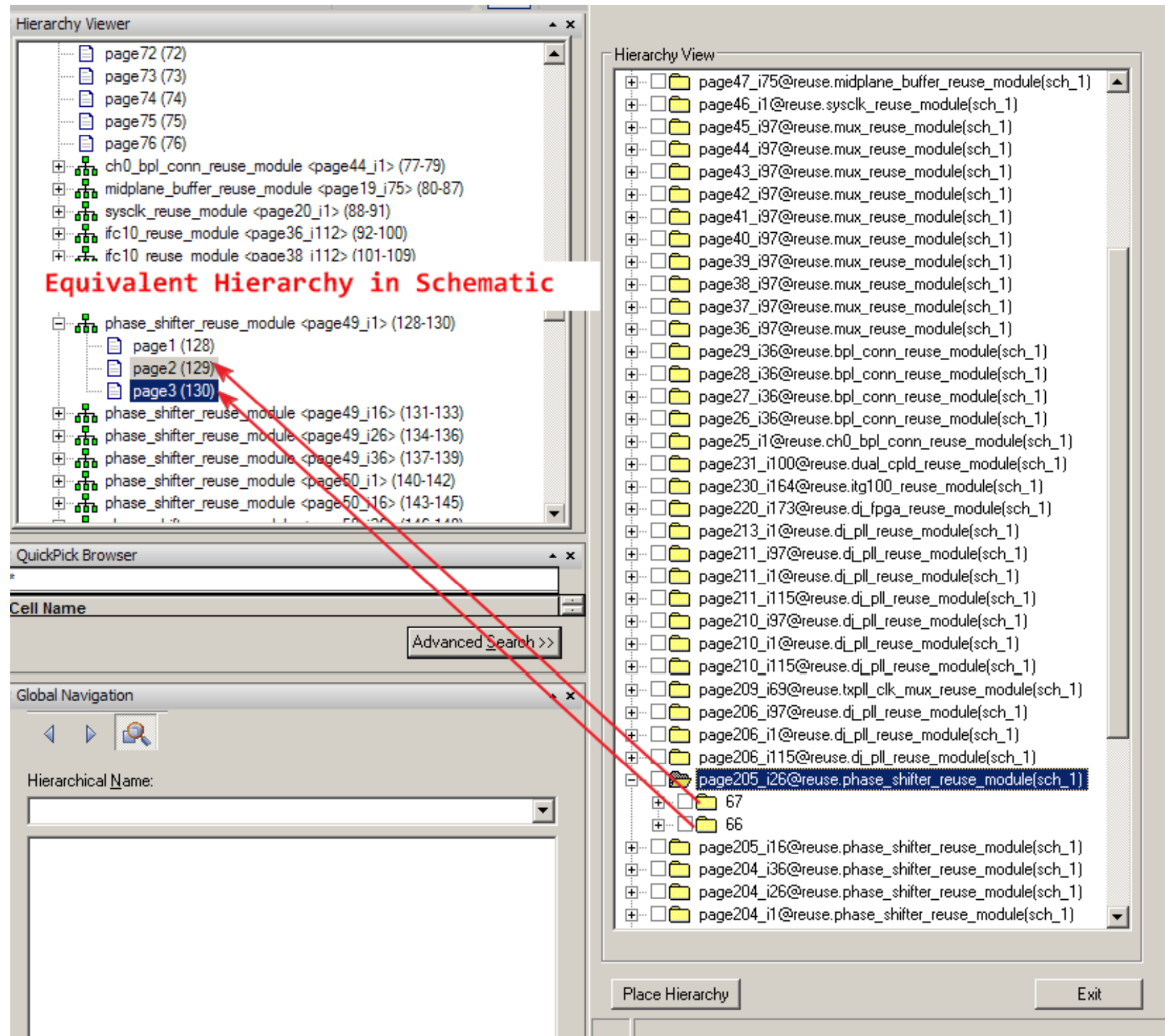
- The QuickPlace tool will place all the components with respect to its top hierarchy, then its page
- The components inside the pages are sorted by their size
- This tool does not care about the logical page number assigned by Concept HDL

Functionality



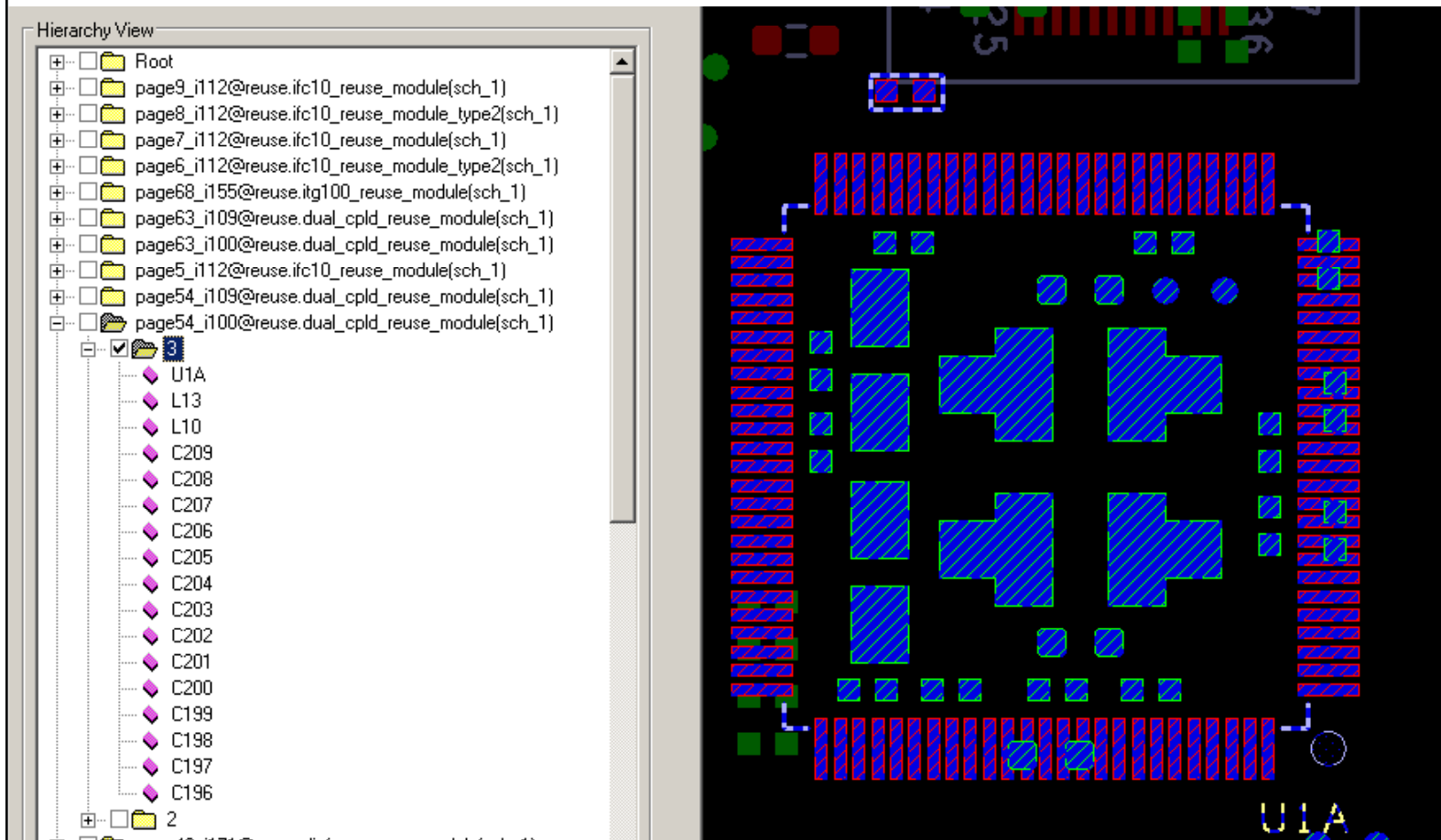
- The Hierarchy browser will list all the pages/hierarchies and their underlying pages/components
- By selecting them the respective components/Pages will be highlighted

Functionality



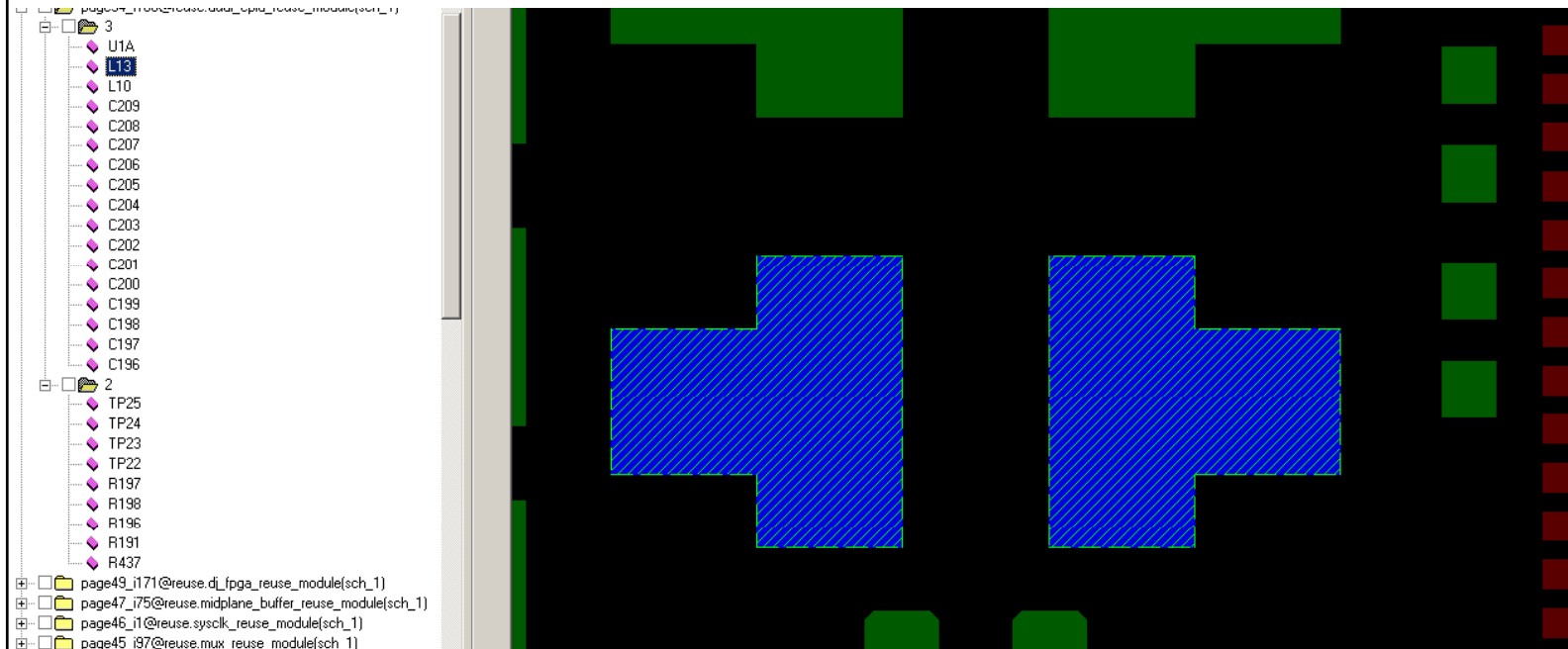
- This hierarchy view in the layout is similar to that of in the schematic.
- It helps in efficient cross probing

Functionality



- Selecting a logical page under a Hierarchy will highlight all the components in that page.

Functionality



- Selecting a single component will make the allegro to zoom in to that particular component

Limitations and FAQ

- The Pages listed under the hierarchy are logical page numbers not the physical
- Hierarchical page names are not the same as that viewed in the HDL, But it is easy to identify.
- The Pages are sorted by the amount of components in them not the same sequence as that of the schematic

- What is the Rational?
 - In a large scale PCB layout after the initial netin , the layout engineer spends a considerable amount of time just grouping the components together w.r.to their page. This can lead to days of productive time spent on just grouping. This tool eliminates that initial time spent
- Why ?
 - By using the inbuilt quick-place there is no way to track a particular component or a group of components to its parent schematic page
 - These are very much manual process and not efficient
- How Quick is this Tool?
 - It can place/group a 9000+ components w.r.to hierarchy and underlying pages under 80 seconds

- Who will use this?
 - A Layout engineer for initial quick place
 - Layout Engineer/Board Engineer to quickly review components that should be placed w.r.to their parent schematic locations
- How this tool is developed?
 - This tool is developed at Infinera and coded in Cadence Skill language
 - Skill is similar to Lisp and C.
 - This tool will work only under Cadence Allegro
- What it is Not
 - This is not a solution for everything, still the layout engineer and the board engineer should review the placement and relative locations

Summary

- This is another set of tool that would help engineers to manage the layout in an efficient manner
- Your comments and suggestions are Welcome
 - Regards
 - Venkata Ramanan