

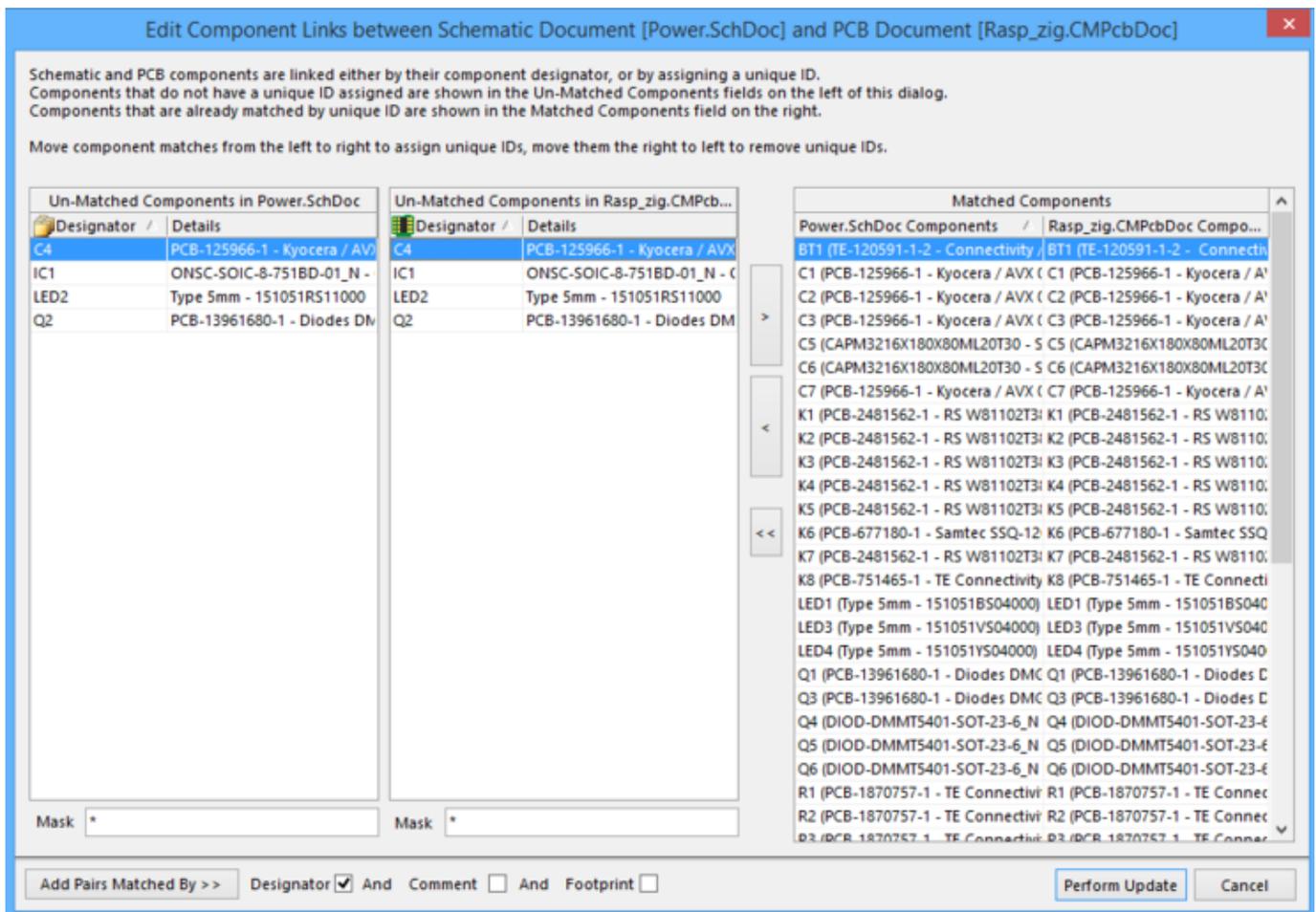
Edit Component Links

Modified by Susan Riege on Jul 30, 2015

Other Related Resources

[Design Compilation & Synchronization](#)

Parent page: [WorkspaceManager Dialogs](#)



The *Edit Component Links* dialog

SUMMARY

This dialog allows the designer to check and control the status of the links between components in the schematic and PCB domains. When a component is placed on a schematic sheet, it is automatically given a unique ID. As a precursor to comparison, CircuitMaker scans the source schematic and target PCB documents for linked components. These are components that have been previously synchronized with one another and share a unique ID. This dialog provides controls to manually match and link components between the two domains.

It is a good idea to have all components matched using unique IDs, so that annotation of designators in either the schematic or PCB document can be carried out, with the safe knowledge that the documents can still be resynchronized at any stage.

When component information is transferred for the first time between schematic source documents and a blank PCB design document using the Synchronizer, all components will automatically be linked by unique ID - the ID information from each schematic component being assigned to the corresponding component footprint.

ACCESS

The dialog is accessed from the PCB Editor, by clicking **Tools | PCB Links**  from the main menus.

Manual linking of components is only carried out from within the PCB document because only the PCB component footprints need to be updated with the unique ID information - it is already present on the schematic side.

OPTIONS/CONTROLS

- **Un-Matched Components** - the left-hand side of the dialog presents components that are currently unmatched. The two lists are used for unmatched components on the source schematic documents (left) and those unmatched components on the target PCB document (right). Each component is listed in terms of its designator, footprint name, and comment.

✓ As you type within the **Mask** field below a list, the list is filtered to only show strings that match the mask string. You can use the ? (any single character) and * (any characters) wildcards in the mask string. For example, "*" will display all components and "D?" will display all components that start with the letter D.

- **Matched Components** - the right-hand side of the dialog presents all components that are currently matched. The region's two columns list the schematic components on the left and the PCB components on the right, with each entry consisting of designator, footprint name, and comment.

Keep in mind that any components that are manually matched using the dialog will be moved from the **Un-Matched Components** to **Matched Components** regions immediately, but the actual links are not created until the **Perform Update** button is clicked.

-  **Match Selected** - click this button to manually match the currently selected unmatched schematic component, with the currently selected unmatched PCB component. The entries will disappear from their respective **Un-Matched Components** lists, and a single entry for the two will be added to the **Matched Components** region.
-  **Un-Match Selected** - click this button to un-match the currently selected component entries

in the **Matched Components** region. The entries will be removed from the region and the components will be added to their respective **Un-Matched Components** region lists.

-  **Un-Match All** - click this button to un-match all components currently listed in the **Matched Components** region. The entries will be removed from the region and the components will be added back to their respective **Un-Matched Components** region lists.
- **Add Pairs Matched By** - click this button to automatically match unmatched components in accordance with the selected options to the right. Matching can be attempted by any combination of **Designator**, **Comment**, and **Footprint**. Successful matching will result in applicable entries disappearing from their respective **Un-Matched Components** lists, and a single entry for each linked couple being added to the **Matched Components** region.
- **Perform Update** - click this button to effect changes made to linking. If you have un-matched any components, a confirmation dialog will appear alerting you to the fact that some existing component associations will be broken by proceeding. To continue, click **Yes**. An information dialog will display, showing how many links were modified and/or how many links were removed.

Adding a component link (matching unmatched components) adds the unique ID of the schematic component, to the linked PCB component. Conversely, removing a component link will remove the unique ID from the corresponding PCB component only. The schematic component retains the unique ID unless a new one is generated (using a reset unique ID-related command).

Source URL:

[http://documentation.circuitmaker.com/display/CMAK/WorkspaceManager_Dlg-ConfirmCompMatchesForm\(\(Edit+Component+Links\)\)_CM](http://documentation.circuitmaker.com/display/CMAK/WorkspaceManager_Dlg-ConfirmCompMatchesForm((Edit+Component+Links))_CM)