


Adding Hidden Net to Sheet

Modified by Jason Howie on Oct 20, 2014

Parent category: [Violations Associated with Nets](#)

Default report mode:  Warning

Summary

This violation is related to multi-part components and occurs when you have specified one or more pins to be hidden and connected to an existing net within the design - typically a power pin connected to VCC or GND for example.

Notification

If compiler errors and warnings are enabled for display on the schematic (enabled on the **Schematic - Compiler** page of the *Preferences* dialog) an offending object will display a colored squiggle beneath it. A notification is also displayed in the *Messages* panel in the following format:

Adding hidden net

Recommendation for Resolution

The problem arises when the following properties for the offending pin(s) are evident (in the associated *Pin Properties* dialog):

- The **Hide** option is enabled
- The **Connect To** field contains the specific power net name
- The **Part Number** is that of the placed part (i.e. other than).

To resolve this issue, you could enable the display of the pin(s) in the workspace (simply disable the **Hide** option). This option may prove to be less than desirable, especially if you have many hidden pins connected to power nets. Revealing these pins in the workspace can cause clutter as each pin would need to be wired to the appropriate power port object - preventing the design schematic(s) from being easily read.

A better solution is to clear the **Connect To** field and set the **Part Number** field to . Leave the **Hide** option for the pin enabled. Repeat for each pin that has been connected to a power net in this way. Ideally, the power net connections should be assigned through use of part in the source library component.

Notes

Only one instance of this violation type will be listed in the *Messages* panel. When investigating the

error using the **Details** region of the panel, a single entry will be listed reflecting the net which is being added. There may be multiple nets added - such as GND and VCC - but only one will be listed, determined by alphabetical order. If you clear the violation for a particular net, the next net (in order) will appear under this violation type.

Source URL:

[http://documentation.circuitmaker.com/display/CMAK/WorkspaceManager_Err-AddingHiddenNetToSheet\(\(Adding+Hidden+Net+to+Sheet\)\)_CM](http://documentation.circuitmaker.com/display/CMAK/WorkspaceManager_Err-AddingHiddenNetToSheet((Adding+Hidden+Net+to+Sheet))_CM)