

## Duplicate Nets

Modified by Jason Howie on Jan 6, 2015

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

Default report mode:  Error

### Summary

This violation occurs when two nets with the same name have been detected within the design.

### 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:

Duplicate Net Names <*Object*> <*NetName*>,

where

*Object* is either Wire or Bus Slice or Element[n] (for a bus element)

*NetName* is the name of the affected net.

### Recommendation for Resolution

- The design is hierarchical, with separate sheet symbols used to reference distinct child sheets, and sheet entries connecting to ports on those child sheets. The violation will occur if the same net label has been used on both child sheets. This is because net labels defined on each sub-sheet, even with the same name, remain local to those sub-sheets. The resolution in this case is to ensure unique net labeling is used between sheets.
- The design is flat and ports have been used within the design. The violation will occur if the same net label has been used between sheets. This is because net labels defined on each sheet, even with the same name, remain local to those sheets. The resolution in this case is to ensure unique net labeling is used between sheets.
- The net continuity between flattened schematic sheets is broken by the inadvertent use of ports or offsheet connectors with different names. Use the **Details** region of the *Messages* panel to quickly cross probe to the duplicate net naming. Trace the net back to the incoming/outgoing port on each sheet and ensure the names for the ports are made the same.
- You may have the same net used in two different branches of a hierarchical design - i.e. different sheet symbols are used to reference different child sheets, but the same name is used for the top-level sheet entries and descendent ports, and the two symbols are connected by a physical wire or bus. The net continuity between these branches can be broken by the

inadvertent use of sheet entries with different names or the omission of a physical bus/wire connecting the sheet entries. Ensure that the physical wire connecting the two sheet symbols is in place and wired correctly and that the sheet entries are named the same.

---

**Source URL:**

[http://documentation.circuitmaker.com/display/CMAK/WorkspaceManager\\_Err-DuplicateNets\(\(Duplicate+Nets\)\)\\_CM](http://documentation.circuitmaker.com/display/CMAK/WorkspaceManager_Err-DuplicateNets((Duplicate+Nets))_CM)