Using Altium Documentation
Modified by Jason Howie on Aug 7, 2017

Parent category: Violations Associated with Components

Default report mode: Error

Summary
This violation occurs in multi-part components, where a hidden pin common to more than one sub-part is connected to different nets.

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:

Mismatched hidden pin connections in Pin <Identifier> and Pin <Identifier>

where:
- *Identifier* is used to identify the pin in question. The identifier appears in the format PhysicalComponentName-PinDesignator (e.g. U2-7).

Recommendation for Resolution
Reassign the offending pin(s) to the correct nets. Edit the pin(s) using the Component Pin Editor dialog - accessed from the Component Properties dialog by clicking the Edit Pins button. From the Component Pin Editor dialog, access the Logical tab of the Pin Properties dialog for the offending pin (for each sub-part in turn) and check/modify the net entry in the Connect To field.

Source URL:
https://www.altium.com/documentation/display/ADES/WorkspaceManager_Err-MismatchedHiddenPinConnections((Mismatched+Hidden+Pin+Connections))_AD