

AMod_BrdTidy User Guide

© Copyright 2002-2005, CopperCAD Design inc.

Purpose

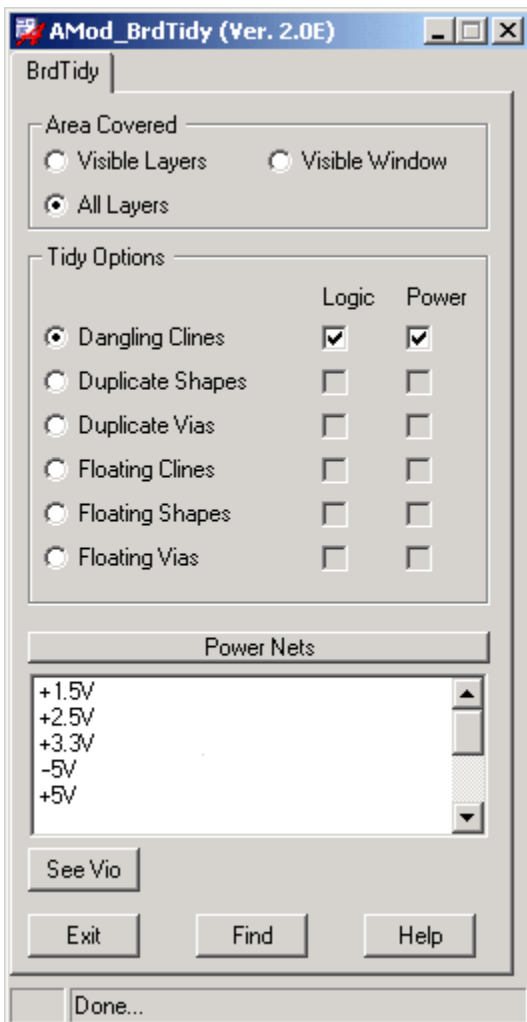
Provides Board Designers with a process to quickly find and delete redundant copper objects (vias, clines and shapes).

Description

As a designer nears the completion of a design, a large number of dangling, floating and duplicate copper objects, (created accidentally during the layout process) need to be located and tidied in order to achieve 100 percent clean Summery Drawing Report. AMod BrdTidy shows the users each violation and gives them the option of automatically or manually fixing the copper. Several hundred objects can be cleaned up in just a few minutes.

Benefits

Reduces the time required during cleanup. Typically three to four hours of cleanup time can be reduced to a few minutes.



AMod BrdTidy Menu

Visible Layers → Checks currently displayed layers only.

Visible Window → Checks all visible layers in the currently displayed window.

All Layers → Checks all of the copper layers in the design file.

Tidy Options:

Dangling Clines → Clines with a net name, with only one end connected to a via or pin

Duplicate Shapes → Identical shapes with the same net name, in the same location, and are not part of a component or symbol.

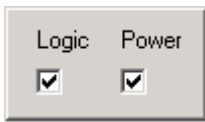
Duplicate Vias → Vias in the same location +/- 1 thou, that have the same padstack and net name. Floating vias are excluded.

Floating Clines → Clines without a net name, and are not part of a component or symbol.

Floating Shapes → Shapes without a net name and are not part of a component or symbol.

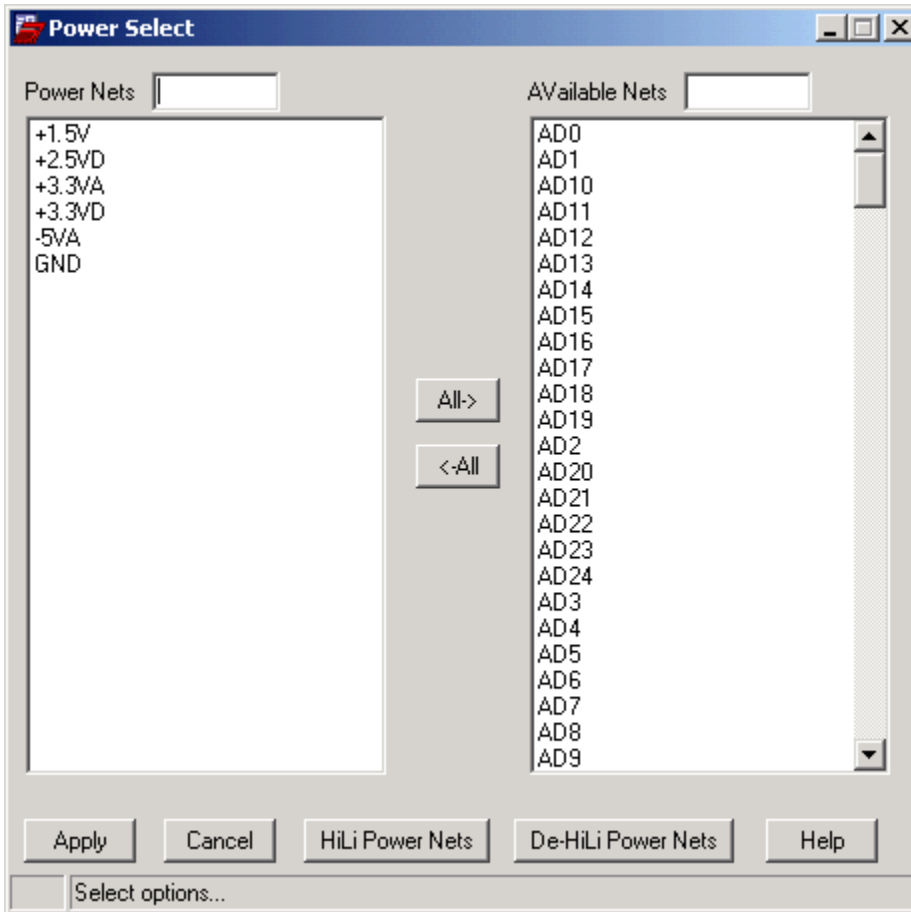
Floating Vias → Vias without a net name, and are not part of a component or symbol.

Logic and Power Nets



The optional net type setting allows the user to tidy the board in stages, by selecting both logic and power (all) nets, logic only nets or power only nets. Logic nets often can often be tidied using the “Tidy All” button, however power nets may require more individual attention to ensure the intended connectivity remains intact

Power Net Selection Menu



Power Nets → Power Nets are populated automatically at start-up by sorting nets that are identified as power nets and by collecting nets attached to all shapes on all layers.

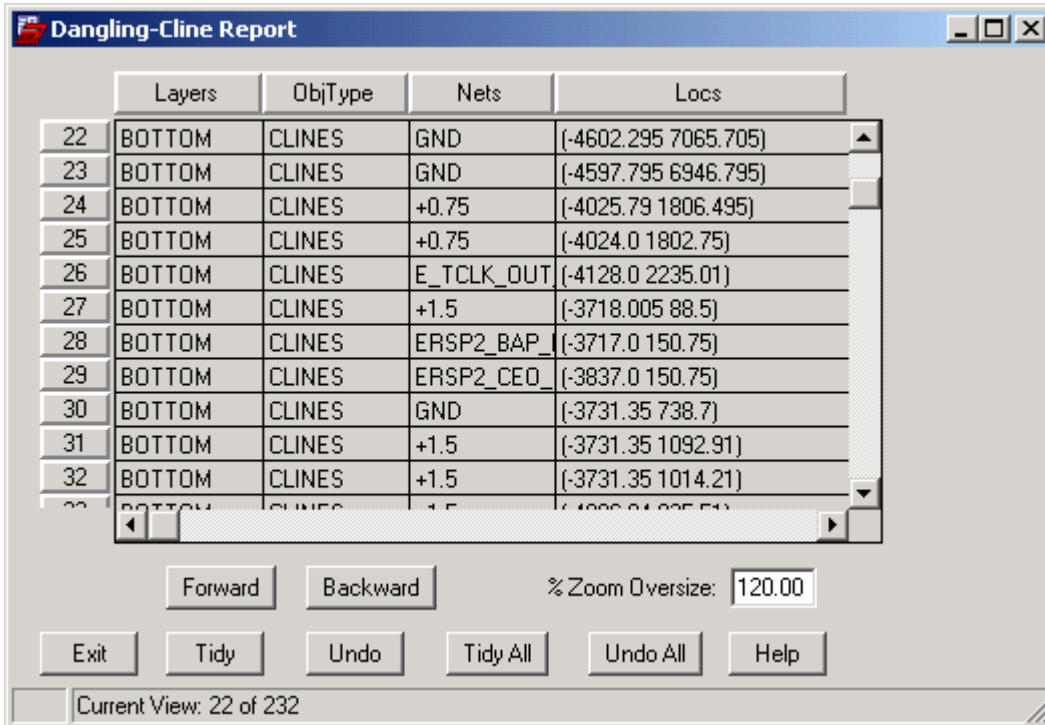
If the power nets are not identified inside the design, they can be populated manually using the menu.

HiLi and De-HiLi → Selected power nets can be highlighted and de-highlighted using these two buttons.

Running the Program

- 1) By default (initially), all layers are selected.
- 2) Select the type of tidy verification check and click the **“Find”** button.
- 3) When complete, click the **“See Vio”** button to visit each violation and tidy each violation or use the **“Tidy All”** button to automatically tidy all the violations.

By clicking the **“See Vio”** button, the following violation report will be displayed.



The screenshot shows a window titled "Dangling-Cline Report" with a table of violations. The table has columns for Layers, ObjType, Nets, and Locs. Below the table are buttons for Forward, Backward, % Zoom Oversize (set to 120.00), Exit, Tidy, Undo, Tidy All, Undo All, and Help. The status bar at the bottom indicates "Current View: 22 of 232".

	Layers	ObjType	Nets	Locs
22	BOTTOM	CLINES	GND	(-4602.295 7065.705)
23	BOTTOM	CLINES	GND	(-4597.795 6946.795)
24	BOTTOM	CLINES	+0.75	(-4025.79 1806.495)
25	BOTTOM	CLINES	+0.75	(-4024.0 1802.75)
26	BOTTOM	CLINES	E_TCLK_OUT	(-4128.0 2235.01)
27	BOTTOM	CLINES	+1.5	(-3718.005 88.5)
28	BOTTOM	CLINES	ERSP2_BAP_	(-3717.0 150.75)
29	BOTTOM	CLINES	ERSP2_CEO_	(-3837.0 150.75)
30	BOTTOM	CLINES	GND	(-3731.35 738.7)
31	BOTTOM	CLINES	+1.5	(-3731.35 1092.91)
32	BOTTOM	CLINES	+1.5	(-3731.35 1014.21)

Navigate to individual failures simply by clicking on any line.

The **“Forward”** and **“Back”** allow easy sequential navigation of the violations.

The **“% Zoom Oversize”** allows you to customize the size of window view based on individual objects extents.

Right Mouse Click on any column heading will resort the data.

The Tidy Button

As PCB designers tend to be cautious when using automatic functions. So in addition to the fully automatic **“Tidy All”** Button, we give the user the option to visit and view each violation, and tidy them one at a time using the individual **“Tidy”** button. If the result is unsatisfactory, the **“Undo”** button restores the original objects.

Once you are comfortable and familiar with how tidy and undo operate, you will find yourself clicking the tidy button, which will flash before and after the tidy operation for each. For repeated violations with a single click for each it is possible to step through them at about 2 per second using this method, using undo when one occurs that the tidy operation is unsatisfactory

Note 1: When using the individual tidy button, re-sizing the Allegro window will cause last **“Floating Shape”** to be tidied, to be restored to its original state. To confirm all tudies are complete run the same **“Find”** function at least two times.

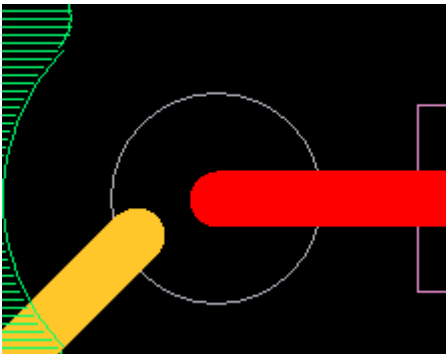
Note 2: Currently HDI technologies are unsupported. Some via in pad fanouts may be detected as dangling.

Tidy Types

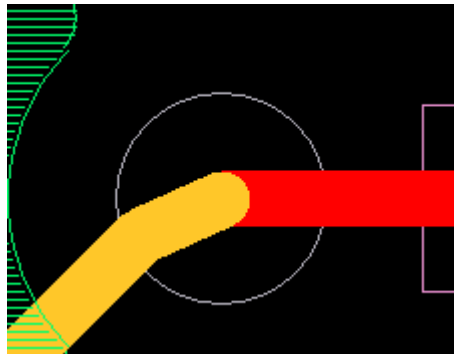
- Dangling Clines** → The entire dangling portion is deleted all the way back to the via, pin or T-junction. If the dangling cline end is inside a via or pin with the same net name, a connection will be made to the centre of the pad.
- Duplicate Shapes** → Identical shapes with the same net name, in the same location, and are not part of a component or symbol will be deleted. The original shape will remain after the tidy.
- Duplicate Vias** → All duplicate vias in the same location +/- 1 thou, that have the same padstack and net name will be deleted. The via with the highest connectivity (most number of clines connected to it's pad centre) will remain after the tidy.
- Floating Clines** → The floating clines will be deleted, including duplicate floating clines (no net name) in the same location. Dummy nets are ignored
- Floating Shapes** → The floating shape will be deleted, including duplicate floating shapes (no net name) in the same location. Dummy nets are ignored
- Floating Vias** → The floating vias will be deleted, including duplicate floating vias (no net name) in the same location. Dummy nets are ignored

Screen Shots

Dangling line inside a pad



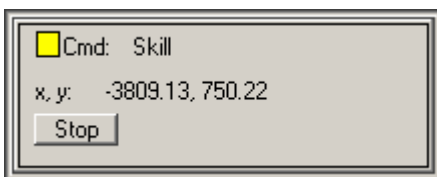
Dangling line automatically fixed



Duplicate via with less than 1 thou offset



The Stop Button



While the program is running, the stop button will be displayed in the bottom right corner of the Allegro tool window. The stop button can be used to terminate the AVerify program at any time.

For more information, please contact CopperCAD at [\(905\) 488-8958](tel:9054888958) or www.CopperCAD.com

(c) Copyright 2000-2005 All Rights Reserved. CopperCAD Design Inc.