AAdd Balance User Guide

Purpose

Adds copper balance pads to the copper layers on the circuit board.

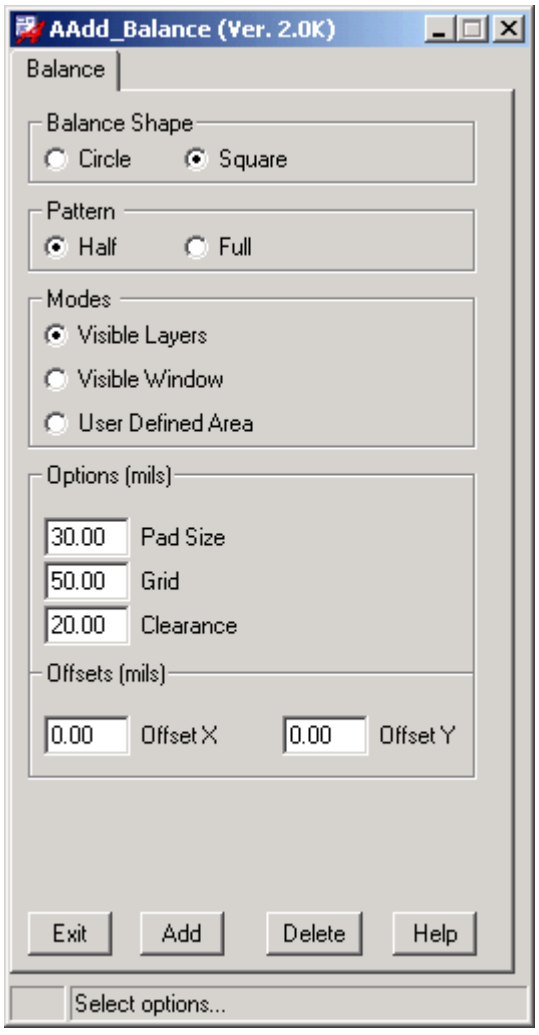
Description

Adds non- functional, circular or square shaped pads to any copper layer of the board, in areas where copper features (pads, track and planes) are not preset.
The pad size, pad grid, and clearance to other copper features can be user defined.

Benefits

Distributes copper evenly across the board (side to side) also helps balance the total amount of copper between layers, vertically in the stack-up.

Improves PCB manufacturing yield (layers etch evenly)
Reduces board warpage during reflow or wave or soldering, as the board will heat and cool evenly, if the copper is distributed evenly.



Menu Functions:

Balance Shape → Choose between circular or square pad shapes.

Pattern → Add pads in **Full** (pads at all grid positions) or **Half** (Checker Board) patterns.

Features→ Add or delete copper balance pads.

Modes of Operation

Visible Layers → Add or delete pads to all visible copper layers on the board.

Visible Window → Add or delete pads to all visible layers inside the display window.

User Defined Area → Add or delete pads to all visible layers in a user defined (two point pick) window.

Options

Pad Size → User can change the diameter of circular pads or the X,Y size of square pads. All imperial and metric drawing units are supported, the default size is 30 thou.

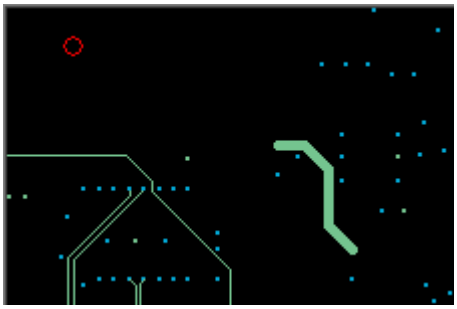
Grid → Spacing grid used to place the pads. Grid size should include the minimum clearance times two. e.g. If the pad size is 30 and minimum copper spacing is 5, the minimum grid size should not be less than 40 or (30 + (2 x 5)).

Clearance → Sets the spacing between the copper balance pads and all other copper features. Default value is 25 thou, if set lower, minimum spacing for the PCB is respected.

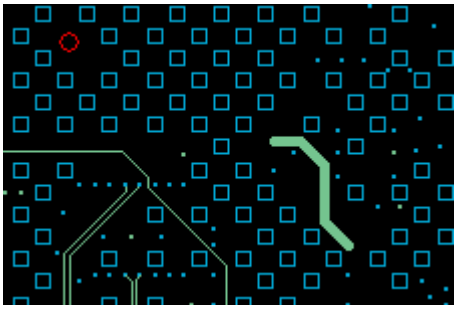
Offsets → Optional pad offsets allows the user to change the starting point of the balance pads.

Screen Shots

Before



After



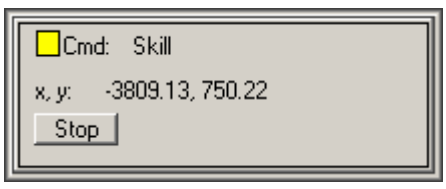
Note 1: The Add_Balance program uses the "Route Keepin" area to place the copper balance pads. If a "Route Keepin" area is not defined on the board, the maximum extends of the "Board Outline" area will be used to define the working area.

Note 2: To ensure that no un-intentional exposed copper pads are created (pads without solder resist covering them), the Add_Balance program will avoid placing pads inside solder resist openings on the "Top" and "Bottom" layers. Types of openings avoided are: solder resist openings around fiducials, SR openings around through hole connectors, etc.

Running the Program
---

1) Select the pad shape, square or circular.
2) Choose the mode of operation → Visible Layer, Visible Window, or User Defined Area.
3) Set the pad size, grid and clearance, and optional x and y offsets.
4) Click the "ADD" button.

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 Add Balance program at any time.

For more information, please contact CopperCAD at **(905) 488-8958** or **www.CopperCAD.com**