# EAGLE BOARD LAYOUT

## User Notes & Tips

PRODUCED BY AIS CUBE

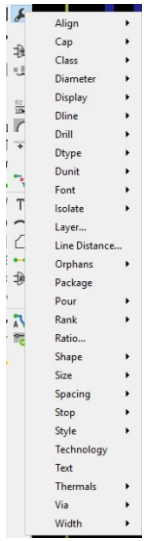
*COMMONLY USED ACTIONS
**SELDOM USED ACTIONS

EAGLE
www.cadsoft.de

### Command Buttons

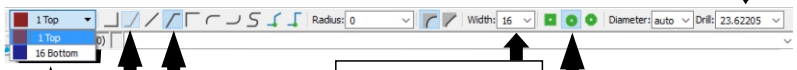
- *Opens properties of selected item. Great for obtaining position of parts for alignment* — **INFO**
- **LAYER SETTINGS** — *Enable/ Disable visible layers*
- **MOVE** — *Right click while moving to rotate*
- **MIRROR** — *Positions item on top/ bottom layer*
- **GROUP** — *Left click (hold and drag) to select an area. Left click and release to form multiple points for a polygon. (right click closes polygon)*
- **PASTE**
- **DELETE**
- PINSWAP
- LOCK
- **NAME** — *Name Nets & Components*
- **SMASH** — *Use smash to move and edit the **text** of parts eg. Adjusting the font size/thickness and location*
- SPLIT
- MEANDER
- **ROUTE** — *Route air-wire connections*
- **DRAW LINE**
- **CIRCLE**
- **RECTANGLE** — *Draw symbols using these to indicate polarity/ part dimensions on tDocu/bDocu layers or silkscreen layers (eg. tPlace/bPlace)*
- VIA
- **HOLE** — *Place a hole in the board (eg. Mounting holes)*
- DIMENSION
- **RATSNEST** — *Used to create a solid GND plane after using polygon*
- ERC
- ERROR LIST

- *Highlights selected item on schematic & board* — **SHOW**
- MARK
- **COPY**
- ROTATE (ON ORIGIN)
- **CHANGE** — *Usually used with "group" to change properties of selected components in one shot (eg. "change" > select on change desired > "group" > select group of items > right click > change: group*




- **ADD ELEMENT** — *Add **footprint element** from library eg. Silkscreen Logo*
- REPLACE
- **VALUE** — *Specify values of components (eg. Resistor/Capacitor/Inductor, LED colour etc)*
- MITER
- OPTIMIZE
- **RIP UP** — *Rip up routed trace. Selecting air-wire removes the entire trace*
- **TEXT** — *Add Text (eg. Labels on silkscreen layer)*
- ARC
- **POLYGON** — *Use this to create signal planes by naming it*
- DIMENSION
- ATTRIBUTE
- AUTOROUTE
- **DRC (DESIGN RULE CHECK)** — *Generates warnings/errors detected based on design rules with reference to Fabrication house capabilities*

## ROUTING PARAMETERS




- *Size of drill hole for vias (unit according to what is set in grid) Min: 14mil*
- *Select layer on which to route trace*
- *Alternate between these 2 styles to route traces, they automatically create 45 degree bends*
- *Width of trace (unit according to what is set in grid) Min: 8 mil *Typical: 16mil Power lines: 2x of signal lines (eg. 16x2 = 32mil)*
- *Shape of Via (square/circle/octagon) select **circle***

## ABOUT TRACE WIDTHS

*Width of trace*
*(unit according to what is set in grid)*
*Min: 8 mil*
**Typical: 16mil*
**Power lines: At least 2x of signal lines (eg. 16x2 = 32mil)*
**These numbers are good for most hobby projects.*
*Always measure your total current consumption and plan accordingly.*

IPC Recommended Track Width For 1 oz cooper PCB and 10 °C Temperature Rise

| Current/A | Track Width(mil) | Track Width(mm) |
|---|---|---|
| 1 | 10 | 0.25 |
| 2 | 30 | 0.76 |
| 3 | 50 | 1.27 |
| 4 | 80 | 2.03 |
| 5 | 110 | 2.79 |
| 6 | 150 | 3.81 |
| 7 | 180 | 4.57 |
| 8 | 220 | 5.59 |
| 9 | 260 | 6.60 |
| 10 | 300 | 7.62 |