

## A



10

10

10

10

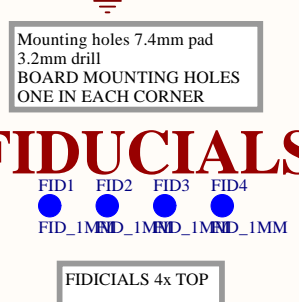
# COUNTING HOLES



**LAYOUT NOTE:**

- 1) Route all the **POWER** tracks with minimum track width 0.4mm.
- 2) Route all the other tracks by 0.4mm and change them by the design to 0.2mm. To change all of them at once, use this filter "(not InNet("+\*") and not InNet("GND")) and IsTrack and (OnLayer("L1") or OnLayer("L2"))" and then set 0.2mm width in PCB Inspector panel.

## USER BUTTON



**DESIGN NOTE:**  
This board can be powered from micro USB connector (J7) or a single +3.3V power rail (through J1 pin 4). If +3.3V is used, fit R26 and R28. In this case, JP4 & R27 must NOT be fitted, otherwise the board may be damaged.

