

Connection Tabs:

The idea is to create **6x solder filled ‘areas’** (2 X 3 zones), **on the bottom surface of the PCB** connection tabs, to provide for connection to the NAD main PCB, **and to leave their solder surfaces ‘open’ / accessible, e.g., with no solder mask.**

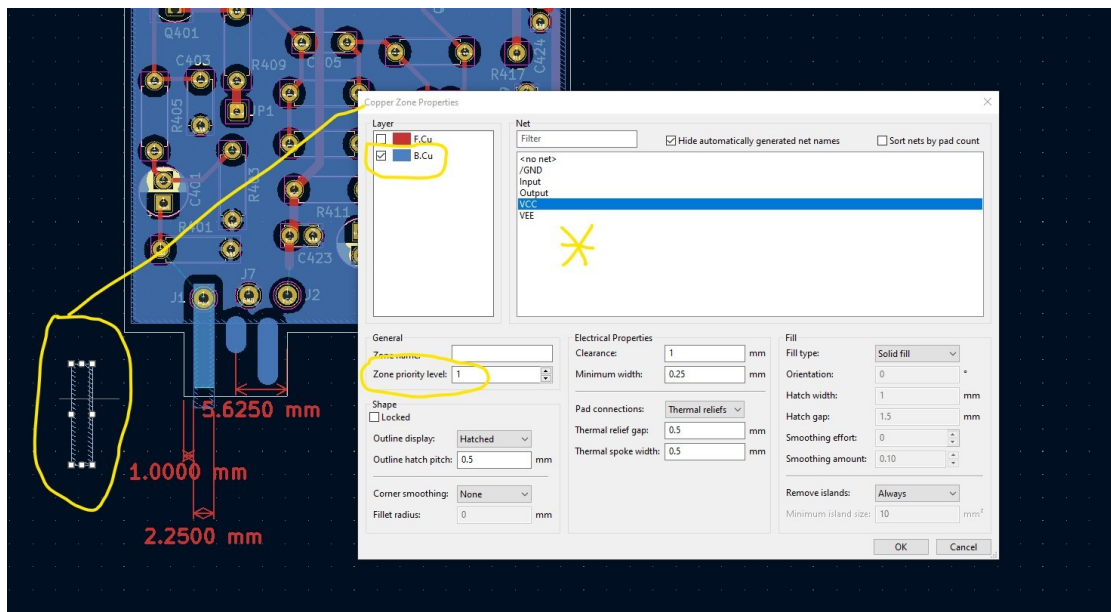
Initially, I’d suggest to just make the 6X ‘areas’ somewhere off to the side of the PCB – to avoid interfering with the PCB itself in any way... **hopefully, the bottom Ground Zone will already have been set to priority ‘0’, if not, now’s a good time.**

Prior to placing the connection ‘areas’ onto the PCB, ‘divide’ (calculate) each of the 11.25mm wide PCB connection tabs into 7 width segments, Left to Right, 1mm (empty), 2.25mm connection area, 1.25mm (empty), 2.25mm connection area, 1.25mm (empty, 2.25mm connection area, and 1mm (empty), just to get the spacing to match the NAD amp PCB. In this case, I used the exact x,y coordinates of the PCB tabs to locate the relevant positions. Zoom into the edge cuts to find the mid points of the lines etc., as needed, I used the grid set to 0.25mm as much as possible, but adjusted whenever necessary.... revert to a 0.25mm grid as much as possible, it’ll be easier ! It will also help if you put the ‘grid origin’ onto say the point circled below (use the ‘grid origin’ tool on the left, by the appearance manager).

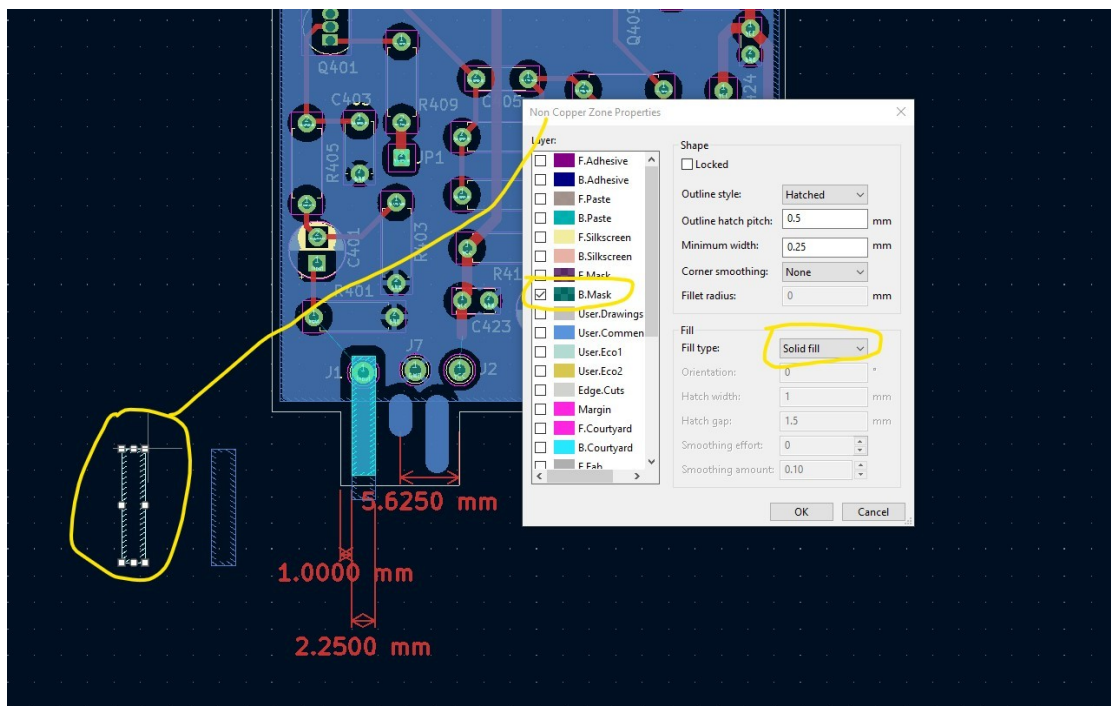
Note – if you get any of the x, y coordinates even slightly ‘off’, you may ‘break’ the edge cuts, and trigger new ‘errors’ – the edge cuts are good at the moment. If it helps, you could temporarily draw short (unconnected) 2.25mm wide PCB tracks at the appropriate positions, and then just drop the zones onto them.



Each of the 6 ‘areas’ consists of a ‘filled’ rectangular zone on the B.Cu layer (to match the NAD main pcb connections), 2.25mm wide, and approx. 10mm high.



A second 'filled' rectangular zone, of the same dimensions, is positioned directly on top of the filled solder zone, but on the B.Mask layer – NOTE, the 'mask' layers operate in the opposite sense to the copper layers; thus while a filled copper zone is filled with copper, a filled mask zone has NO solder mask. This therefore results in the underlying copper zone being open / exposed, and available for soldering.



The 2 superimposed filled zones may then be 'grouped' as a single object, and simply replicated 5X using copy & paste – each of the 6 grouped areas will become one of the connections.

To position each connection 'area' onto the PCB, it first has to be prepared – so ungroup one of the 6 'areas' (just so both of the zones are available to edit), select the filled zone on the B.Cu layer, and go to it's 'properties' page – set the zone to the appropriate 'Net', e.g., if it's for the

Input connection, set it to the Input Net, then set the zone priority to either '1' or '0' ('0' if it's to be used for either of the GND connections, otherwise '1'), then OK. The 'area' can then be grouped again, and simply dragged as one object to it's correct position on the PCB - it may be simpler to 'drag' the existing connection terminal on the PCB (J1, 2, 3, etc.) out of the way, while positioning the 'area', and to then drag it back afterwards, centred on top of the new 'area'. No other changes should be necessary to the PCB as such – obviously delete & re-draw the connection wire / track to the terminal (on the F.CU layer !).

Press 'B' to fill the zones, and then take a look with the 3D viewer... you should see the connection tab, with the exposed solder area in it's correct position.

If not, just post a [thumbnail](#) image for clarification, ideally before proceeding further with the other 5 'areas'.