Important Notes about new PCB designs

The aim of this page is to give the community the possibility to publish improved layouts of MBHP modules. The original layouts by TK are mostly quick solutions with focus on “first-time-right”, and therefore not very well optimized with regard to area size or number of bridges.

Even small modifications on the original layouts lead to a long process: new etch masks have to be created, the new PCB needs to be tested before it's produced in larger quantities and last but not least the documentation has to be changed: sometimes new photos are required, existing schematics and connection diagrams need to be updated in a way which avoids confusion between different layout revisions, and the support effort in the forum increases, since different layouts have to be taken into account when helping on troubleshooting.

However, all these “issues” should not prevent you from sharing your improvements with the community. They could be part of the next official layout revision. Note that the update cycle is ca. 1-2 years, so it's enough time to think about the best solutions!

When publishing your layouts here, please consider the following:

- change the version number in a meaningful way (e.g. v1 → v1_1)
- don't remove the names of the copyright holders. But you can add your name if you want
- for best utilization, the board dimensions should be choosen in a way which matches with the “eurosize format” 160×100 mm (e.g. 80×100, 80×50, ...)
- try to let ca. 5 mm free at the borders, otherwise chances are high that tracks will be damaged during splitting the boards to smaller pieces
- consider mounting holes at the edges
- single sided boards should be prevered before double sided boards
- It is against the law to use any pirated/cracked software, and this is absolutely not to be done here. If you have a registered copy of Eagle, or a freeware/Eagle Light version, go right ahead. Cracked software is not only illegal, but will also corrupt the .brd files (they cannot be opened with the freeware version anymore).
    - Consider Kicad if you need a larger layout. It is open source and free to download from http://kicad.sourceforge.net/
- **IMPORTANT**: never change the part numbers, they must match with the schematic to ensure consistency

---

Improved Layouts

Edit this page to add a new one

What	Who	Where	Notes
Bridgeless MBHP_AOUT Module	NorthernLightX	http://home.quicknet.nl/qn/prive/alex.span/midibox/mbhp_aout_v1_bridgeless.brd	
Bridgeless MBHP_BURNER Module	NorthernLightX	http://home.quicknet.nl/qn/prive/alex.span/midibox/mbhp_burner_v1_bridgeless.brd	

What	Who	Where	Notes
Some Layouts (including core)	rdoursenaud	http://rdoursenaud.free.fr/midibox/	
Improved MBHP_OPL3 Module	selfservice	http://www.cronokart.com/midibox/MBHP_OPL3FM_V1_SCH.pdf	Take care about the CORE→OPL3 connections, some pins are swapped
“”	“”	http://www.cronokart.com/midibox/MBHP_OPL3FM_V1_BOT_V2.pdf	This is the bottom layer
“”	“”	http://www.cronokart.com/midibox/MBHP_OPL3FM_V1_AST_V2.pdf	This is the assembly top

Ideas

Suggestion from Rowan: I've got an idea for a small modification to the layout of the DIN and DOUT boards which could cut down wiring in some situations.

What I sugget is to change the spacing BETWEEN J3 -J6 of the DIN and DOUT (I guess this would be useful on the AIN too) to a multiple of 0.1“.

The reason I suggest this is it would allow the DIN/DOUT/AIN to be used as “Plug-in” Modules/daughter boards for strip/matrix board that have DIL Sockets mounted on them. I could mount all my buttons, pots, encoders etc on Strip board, then mount some DIL sockets spaced the same distance apart as J3 - J6 on the other side of the board and plug the module directly on to the board which has the controls on it.

From:
https://www.midibox.org/dokuwiki/ - **MIDIbox**

Permanent link:
https://www.midibox.org/dokuwiki/doku.php?id=layout_improvements&rev=1214872634

Last update: **2008/07/01 00:38**