mnmlCore

we want to create a small STM32F4 based Board (for example 10x4cm in size), with limited Pins/sockets...
FantomXR had made such a thing before, so we start with his working design, and invest some time in it
see also this article: Another Midibox Core
I have started a new Tread in the Forum, so we can discuss things there: mnmlcore, but i hope that most of the things will be done in the wiki, so it is more clear

Introduction

...

a Name

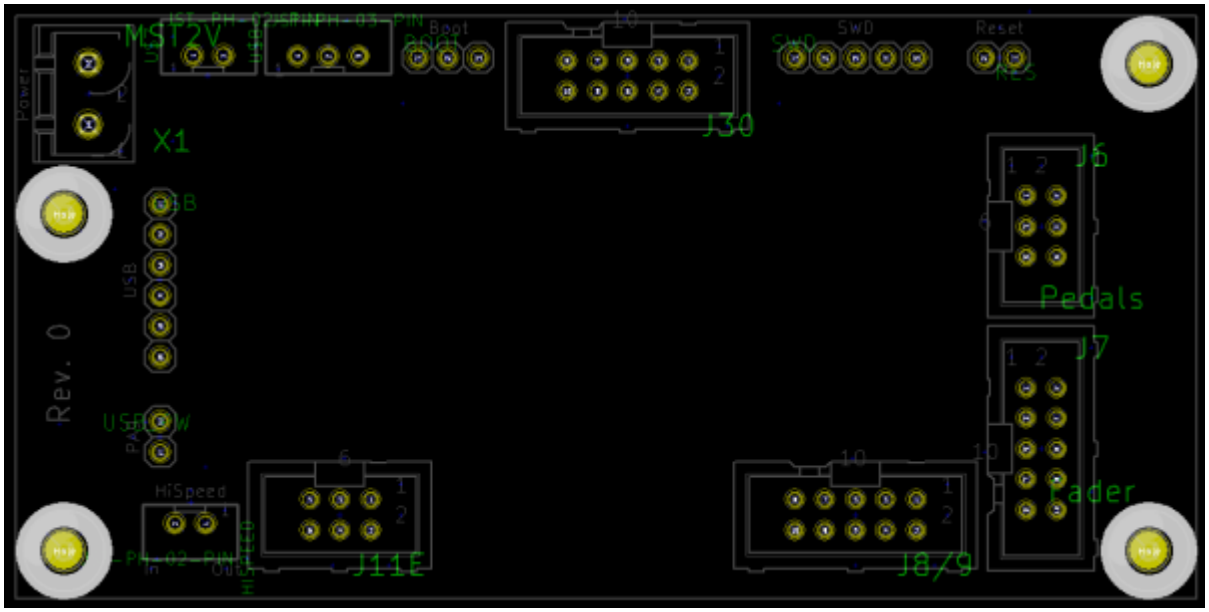
mnmlCore, miniCore32, 10x4CORE, RFcore, fundimentalCore, basisCore, baseCore, XRcore, midicore

PCB

the original [[FantomXR]] Board:

Features

- **Size V0:** 98,5mm x 48,5mm
- **Micro-SD-Slot**(flat @back on the board)
- **J8/9 SRIO** (for normal shiftregisters)
- **Midi A+B** (+HiSpeed?)(need external Optocoppler)
- **J30 Displays**(need external HC595)
- **Bootloader-Switch**(for program-alphas...debug)
- **onboard-Voltage-Regulator** (5V input regulated to 3,3V)
- **USB over Pinheader** (no onboard-socket)
- **USB-Power-Jumper**
- **J6+J7 - Pedals+Fader**:“for an AINSER64-like-analog-scanning”, “I made some pcbs with faders which I connect directly to that connectors”
- **SWD-Header** (to program BOOTLOADER into board by using for a example a Discovery-Board)
- **Reset-Header**



our modified Board:

improvements

FantomXR made some improvement in his other projects, so we also should change this things:
“For example I changed:
the oscillator to SMD
all none-VDD-traces to 0.2032mm
all vias to round-style and 0.32mm diameter.
all 0603 SMD parts to 0805. removed the ID-pin from USB.
added an inductor behind the USB connector for EMC-improvements
”

PCB Fabrication/Pick and Place:

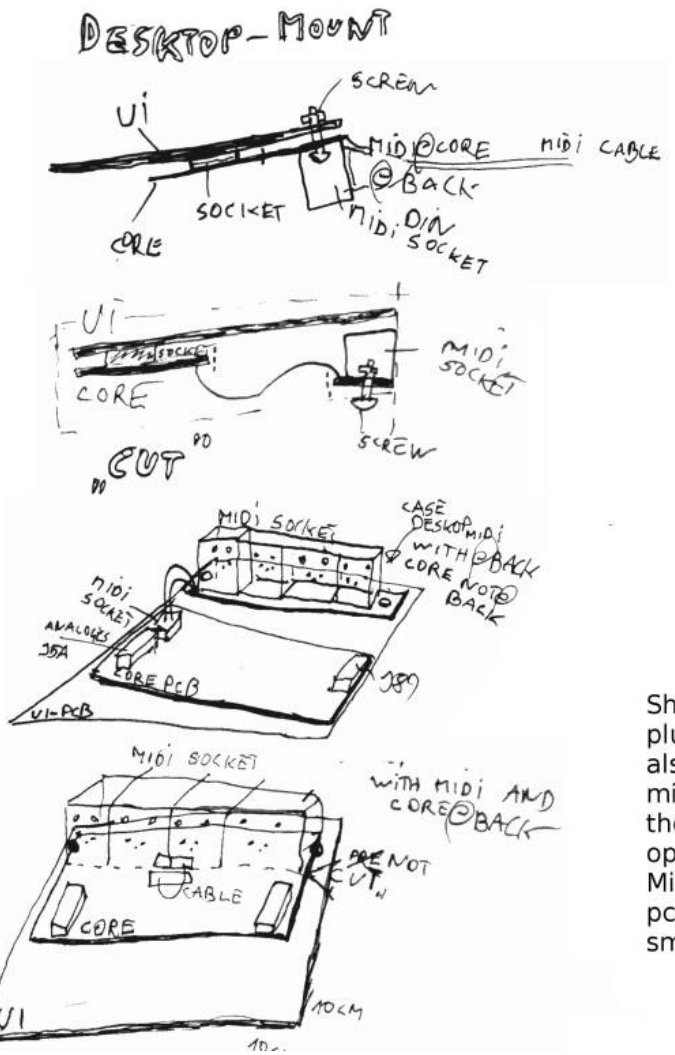
FantomXR have a pick and place machine, price he not knows up to now

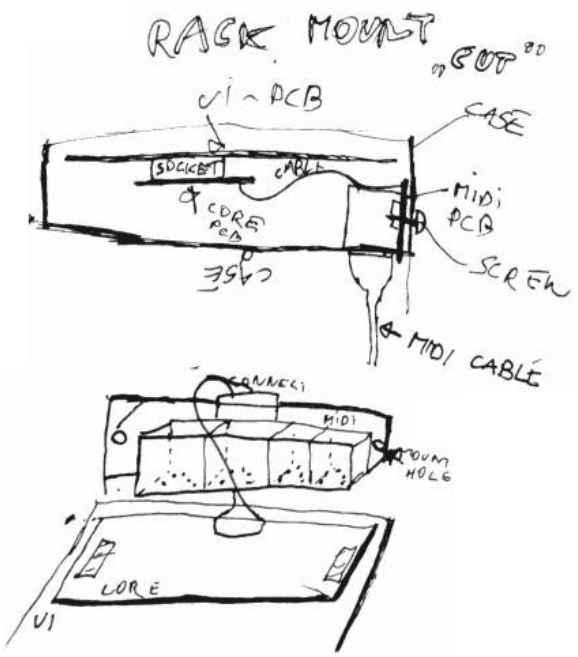
Hardware Requirements

- a jtag programmer (or a Discovery Board)
- Soldering Iron, Wires, PCB….
- USB Power Supply

Design Concepts

phatlines idea with onboard-split-cut MidiIO:





Showing a 10x4cm sized Core32 plugged into a 10x10cm UI-PCB also a midiport-solution is shown, where a midiPCB is fabricated on the core PCB but with the possibility to break it down, the optocopplers diodes and Resistors for the Midiports are on the Core PCB, so the Midi-IO pcb can be small as possibil, so we achive a small Project (small case, small ui...)

Resources

Forum to discuss

Community users working on it

- **Phatline** = Documentation, PCB-layout-or at least give it a first try...

Just let a Private message on the forum to user already involved

From:
https://wiki.midibox.org/ - **MIDIbox**

Permanent link:
https://wiki.midibox.org/doku.php?id=mnmlcore&rev=1533178573

Last update: **2018/08/02 02:56**