
Subject: Multicomp PCBs

Posted by [lenzjo](#) on Tue, 01 Dec 2015 15:12:27 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi,

I've just got a Cyclone II board and would like to build a 6809 system with it. I noticed mention of some pcbs for it in the wiki. Are these pcb's still available?

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Fri, 04 Dec 2015 00:45:04 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi Lenzjo,

the Cyclone II PCB "motherboard" was available from <http://www.smarthome.jigsy.com/fpga> but the designers (Dr Acula and James Moxham) have now moved on to a PCB designed for a different, Cyclone IV-based daughtercard. You can try contacting them to see whether they still have any of their older design available.

If not, it's fairly straightforward to construct this system using a prototyping board. I used point-to-point wiring with (soldered) wire-wrap wire. It's about 45 wires. Use Grant's diagram as a guide:

<http://searle.hostei.com/grant/Multicomp/index.html>

The end result is not beautiful but it's All My Own Work:

https://github.com/nealcrook/multicomp6809/blob/master/photos/re_20150429_multicomp.jpg

For the RTL itself, since you want a 6809 system, I suggest you use my updated set of files which you can find here:

<https://github.com/nealcrook/multicomp6809/tree/master/multicomp>

With this hardware setup, you can run the following software:

Brad Rodriguez's Camel Forth (I use this as the ROM "boot monitor" and everything else is loaded and started from images on SD card)

Microsoft 8K BASIC

Lennart Benschop's 6809 debug monitor, BUGGY

Dave Dunfield's CUBIX

FLEX 09

Nitros9 (level 1, currently)

images for all of this and source code for (nearly) all of this is available here:

<https://github.com/nealcrook/multicomp6809>

and documentation here:

<https://github.com/nealcrook/multicomp6809/wiki>

The Nitros9 sources are not on-line but the executable disk image is online and the sources (as a patch from the open-source repository) is available from me on request.

If you have any problems getting up and running or in following any of the documentation on my wiki, let me know; I'm always keen to facilitate bringing another 6809 system into existence.

Neal

Subject: Re: Multicomp PCBs
Posted by [lenzjo](#) on Sat, 05 Dec 2015 10:48:27 GMT
[View Forum Message](#) <> [Reply to Message](#)

Hi Neal,

THX for the info. I had seen Grant's site, that's what prompted me to go further with the project. I am building a real 6809 system and thought that the FPGA might be a quick way of test-driving Flex and OS9 before committing on the real system. Went to the Smarthome site no contact info but will dig and find one. Don't really want to do point-to-point with the memory..

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Sun, 06 Dec 2015 23:29:35 GMT
[View Forum Message](#) <> [Reply to Message](#)

I haven't got any cyclone II boards left (sorry) but I am putting together some kits for cyclone IV boards. Should be available soon. We found we needed a larger FPGA when doing things like multiple serial ports and a memory management unit.
If you built a 6809 on the cyclone IV you would need to change all the pin assignments as the cyclone IV has more pins/different numbering.

Subject: Re: Multicomp PCBs
Posted by [lenzjo](#) on Sun, 06 Dec 2015 23:40:03 GMT
[View Forum Message](#) <> [Reply to Message](#)

thx dr acula for answering this thread I might be interested in the cyclone IV pcb/kit though I won't be able to get hold of a cyclone IV till the new year. I've only found 'em in China, none so far here in the UK. If you're not making any more cyclone II pcbs would you be willing to release the gerbers so that I could get one made?

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Sun, 06 Dec 2015 23:55:00 GMT
[View Forum Message](#) <> [Reply to Message](#)

I'll see if I still have the gerbers - had a computer crash a few months back and lost a pile of data. I might have one soldered up somewhere though - will rummage around in the shed and see what I can find.

Grant's design will fit on the free (10x10cm) size of Eagle - I don't know if you feel keen trying to design a board using Eagle?

Subject: Re: Multicomp PCBs
Posted by [lenzjo](#) on Mon, 07 Dec 2015 00:08:04 GMT
[View Forum Message](#) <> [Reply to Message](#)

I use Kicad, so size is not a problem.. It's more the time that I'm lacking right now otherwise I wouldn't be bothering you with this.

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Wed, 09 Dec 2015 10:02:41 GMT
[View Forum Message](#) <> [Reply to Message](#)

My original boards added a touchscreen, which I always had hopes of getting working but the code didn't quite fit on a cyclone II so all of that part of the board is not needed. Hence the move to the cyclone IV. Just now I costed out the real cost of a cyclone IV kit. With all the parts, including three serial ports, dual ram, wifi access and cyclone IV fpga, it comes to \$120AU, maybe \$100US. I'm putting together some kits right now and should have them available soon. One big plus for me is Max's software where he puts partitions on an SD card so one card can do CP/M2, CP/M3 and MP/M plus software to easily copy between FAT and CP/M partitions.

Having said all that, the cyclone IV project has different pin layouts due to the FPGA board, so the 68xx part of Grant's original project has been left behind. Given this discussion, I thought it well worthwhile to go back to Grant's original design and do another PCB. The cyclone II fpga is cheaper, there are less parts on the board and an complete kit would be cheaper too.

I've got a PCB designed just tonight. It is 10x10cm, and is pretty much exactly the same as Grant's webpage <http://searle.hostei.com/grant/Multicomp/index.html> except for i) the serial ports also include max3232 chips for proper RS232, and ii) one pulldown on the NC pin 1 of the ram chip which means the socket can take a 128K or a 512K ram chip.

The thing is I am 99% sure that this pcb is correct but not 100%. It is based on version 9 of boards I already made and those 9 versions ironed out a lot of bugs, including things like flipping the fpga board upside down and adding two pins for power. It uploads fine to Seeed PCB and the processed gerber looks correct. The quote is \$21.90 for 10 boards (minus shipping).

So the next step. I'm happy to send this pcb package to anyone and you can get your own boards made, but no guarantee it is perfect. If you want to do this, log into Seeed and upload the attached

zip file.

Or I can get boards made and we wait a few weeks and I can post them out, which of course will be cheaper but will take longer.

There probably is a market for Grant's original design as it is, because it is cheaper than the cyclone IV option. Though you need an obsolete version of Quartus to program it and the fpga is pretty much full, so clever things like MP/M and more serial ports run out of room.

In an ideal world, I think there is a place for both the cyclone II and IV, and as prices come down, maybe someone might port over the 68xx part of Grant's project into the cyclone IV (not me though, I'm more a Z80 sort of bod!).

Cheers, James Moxham

File Attachments

- 1) [Image1.jpg](#), downloaded 627 times
 - 2) [Image2.jpg](#), downloaded 619 times
 - 3) [Cyclone2v110.pdf](#), downloaded 781 times
 - 4) [Cyclone2v110.zip](#), downloaded 417 times
-

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Wed, 09 Dec 2015 12:38:24 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi James,

nice work in recreating a smaller version of your original. Are you working to a fixed board size? I'd (respectfully and politely) lobby for the following changes:

- 1/ add a second RAM device like your original board (it looks like this might fit if you rotated the MAX devices and switched to a mini SD socket). Routing should be straightforward because you can arbitrarily scramble address and data lines to the RAMs
- 2/ add the option to power the keyboard from 5V rather than 3V3 (power jumper plus 2 optional diode clamps as suggested by Ian May)
- 3/ 2-pin jumper connecting pin <cannot remember offhand> to Gnd. In my 6809 codebase, as deployed by Kip Koon, I used that to swap the address decode of UART1 with the virtual uart (VGA/PS2). This allows the system to selectively boot to either device with no sw change
- 4/ 4-pin header to support connection of RTC chip + button cell (ebay module)

If you're open to any of these suggestions I'd be happy to provide more details and/or review schematics/layouts. I'd be interested in one or two of such boards and would be willing to fund a purchase of 10 (in the UK).

I see that you have 5V on this board, for the MAX devices. Is there a connection on the FPGA daughtercard header to route power up (I don't recall seeing one) or do you need to use separate power connection to the FPGA daughtercard?

My current 6809 codebase, compared with Grant's original, includes a GPIO module, interrupt controller, memory mapper, timer and hardware single-step support and more synchronous timing. The FPGA has not yet declared itself "full" but it is pretty close.

thanks,

Neal.

Subject: Re: Multicomp PCBs

Posted by [dr_acula](#) on Wed, 09 Dec 2015 12:55:11 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi Neal,

Thanks for the suggestions.

0) The board size is 10x10cm because Seeed charge in 5x5cm increments and going to 10x15cm is quite a bit more so I guess that is the thinking. Plus for some very strange reason, my local post office charges more that double the price of postage to the USA for a 10x15cm board of a 10x10cm one. It goes over a weight threshold. On the cyclone IV we put things in the middle of the fpga chip and there is spare room too on this cyclone II so we can fit things in the middle there too. So there is spare space for (fairly low-rise) things.

1) Second ram kind of led us (me and Max) down the path of how do we use the second ram, and that led to a MMU using both MP/M and a ram drive at the same time and that led to more vhdl which pushed us to the cyclone IV. There probably is a cunning way to put in a MMU in the cyclone II but might mean only one serial port. There is another idea - Grant's board only used 64k of a 128k chip, but with three jumpers to three spare fpga pins it would be possible to use the same 32 pin ram socket to use a 128k chip or a 512k chip, and so go from 64k to 512k with just one chip. Is this enough to do clever things with a 6809?

2) Happy to take suggestions about the keyboard power. Ian did lots of research on this. Grant's design powers from 3V3 but can easily change to 5V. Or put in two jumpers for either/or.

3) Can you post a schematic for that - should be ok to do.

4) Sounds a good idea, have you got a schematic?

In general, Grant's design has lots of spare pins and so extras should be possible if they can fit in 10x10cm. I'd very much prefer that everything stay backwards compatible with Grant's original design, but then add in things that add extra value but which can be left off a board if not needed.

5) Power is 5V, this goes to the fpga which has a local 3V3 regulator. I don't think anything on the board is powered by 5V except the fpga. Oh and one pin on the VGA which tells the display it is enabled (and which Grant leaves not connected, not sure about that...) Max3232s are powered by 3V (max232s are 5V).

6) Your 6809 sounds brilliant and is exactly the sort of addon that this design needs. In an ideal world, I'd like a board that is Grant's original design, adds in your extras and is all backwards compatible.

Great fun brainstorming this - I look forward to adding in your suggestions

Cheers, James

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Wed, 09 Dec 2015 20:40:21 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi James,

thanks for your explanations. I agree that your rationale for keeping the board size down is sound, and I agree that a board that works with Grant's original design is highly desirable.

Here is a (sorry, hand-drawn) schematic showing RTC dimensions/hookup, kbd connections and the uart address jumper. Also (if there's room.. a connector for 8 GPIO which are software accessible in my design. I showed the dimensions of the RTC module. It would be OK to fit this on the bottom of the board (mirror the connector). Search in ebay.co.uk "arduino rtc ds1302" for a picture.

My code-base does include a memory mapper that has 2 contexts and can map up to 1MByte of SRAM in 8K chunks. It also has the ability to disable the ROM, making the whole address space appear as RAM (apart from the 16-byte memory mapped I/O space (yes, yes, memory-mapped. No sniggering from you z80 types!! . The memory mapper has been designed to be functionally compatible with that on the Tandy Coco, with the hope that it will allow a level-2 port of NITROS9 (I already have a level-1 port running). As I mentioned, it does all fit along with the VGA color display, 2xUART, GPIO etc. (I guess a 6809 is somewhat smaller than a z80).

That is why I would lobby for (i) connecting all the address lines to the existing RAM (ii) adding a second RAM if it is at all possible. However, I guess that worst-case, a second RAM could be added by piggy-backing and a flying lead for the extra chip select.

I understand that wiring the extra address lines for the RAM would break compatibility with Grant's design, in the sense that his project would need a couple of 1-line additions in order to work with a 128Kx8 device, but I think it's too sad to pass up the opportunity to access 1MByte of RAM. It was my unhappiness at tying off the high-order address line of the AS6C1008 that led me to develop my mk1 memory mapper, then learning about NITROS9 led me to redesign it to accommodate a bigger address space and the capabilities required for NITROS9.

regards,

Neal.

File Attachments

1) [multicomp_pcb_mods.pdf](#), downloaded 568 times

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Wed, 09 Dec 2015 23:37:28 GMT

[View Forum Message](#) <> [Reply to Message](#)

James,

now that I'm staring at your layout and holding the FPGA daughtercard, a couple more questions occur to me..

1/ do you need the 5V connector on your board? why not take power in on the connector on the FPGA daughtercard and use the 2-pin connector to route power down to your board (only used for the kbd so some may not need it at all. Perhaps it could be a flying lead and not require the FPGA daughtercard to completely sit within the footprint of your PCB

2/ if you rotated the FPGA daughtercard by 180 the programming connectors would align with the edge of the PCB. You could put a "knockout" area on your PCB and provide free access to those connectors - and access to the reset button, and somewhat sight of the LEDs.

apologies if you've already considered and rejected these ideas.

Neal.

Subject: Re: Multicomp PCBs

Posted by [dr_acula](#) on Wed, 09 Dec 2015 23:51:26 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi Neal,

Thanks for the great schematic.

I've added the changes - please take a look and let me know. Which pins do you want to use for A16 to A18?

Cross post with your one above, yes, once get the schematic right can look at layout. And yes, if the fpga overhangs the side of the board, better access to the programming socket, and it may well free up space for that second ram chip. However, then no access to the two 5V supply pins. Would need flying leads. Or maybe some other solution...

File Attachments

1) [Cyclone2v110.pdf](#), downloaded 541 times

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Thu, 10 Dec 2015 08:25:06 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi James,

schematic looks good. Here is the pin assignment (from your previous cyclonell board). BTW, I don't think the jumper is needed on the RAM pin1 because that pin is NC on the smaller device.

```
set_location_assignment PIN_4 -to n_sRamWE
set_location_assignment PIN_126 -to n_sRamCS
set_location_assignment PIN_79 -to n_sRamCS2
set_location_assignment PIN_134 -to n_sRamOE
set_location_assignment PIN_32 -to sRamAddress[18]
set_location_assignment PIN_31 -to sRamAddress[17]
set_location_assignment PIN_25 -to sRamAddress[16]
set_location_assignment PIN_8 -to sRamAddress[15]
set_location_assignment PIN_30 -to sRamAddress[14]
set_location_assignment PIN_24 -to sRamAddress[13]
set_location_assignment PIN_28 -to sRamAddress[12]
set_location_assignment PIN_136 -to sRamAddress[11]
set_location_assignment PIN_132 -to sRamAddress[10]
set_location_assignment PIN_139 -to sRamAddress[9]
set_location_assignment PIN_142 -to sRamAddress[8]
set_location_assignment PIN_143 -to sRamAddress[7]
set_location_assignment PIN_141 -to sRamAddress[6]
set_location_assignment PIN_137 -to sRamAddress[5]
set_location_assignment PIN_135 -to sRamAddress[4]
set_location_assignment PIN_133 -to sRamAddress[3]
set_location_assignment PIN_129 -to sRamAddress[2]
set_location_assignment PIN_125 -to sRamAddress[1]
set_location_assignment PIN_121 -to sRamAddress[0]
set_location_assignment PIN_122 -to sRamData[7]
set_location_assignment PIN_120 -to sRamData[6]
set_location_assignment PIN_118 -to sRamData[5]
set_location_assignment PIN_114 -to sRamData[4]
set_location_assignment PIN_112 -to sRamData[3]
set_location_assignment PIN_113 -to sRamData[2]
set_location_assignment PIN_115 -to sRamData[1]
set_location_assignment PIN_119 -to sRamData[0]
```

I will do a double-check/review of the whole schematic against this assignment file but that won't happen until the weekend.

regards,

Neal.

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Thu, 10 Dec 2015 12:37:04 GMT
[View Forum Message](#) <> [Reply to Message](#)

Thx for the layout. Ok, maintaining the backwards compatibility but adding extras...

1) overhang the board upside down. One disadvantage - board overhangs. But advantages - frees up some space on the pcb for a second ram chip, frees up space to attach the jtag programming

header, and doesn't need to use tall headers (which are a bit hard to find anyway as dual row, so need to use lots of single row ones. Standard height ones are much cheaper than tall ones.

2) add one jumper cable like you say, to bring out the 5V onto the board. Leave it out if not using any 5V things (keyboard mainly).

3) This frees up some space for the second ram. It is entirely optional but also completely backwards compatible with Grant's original design.

4) Yes agree that header is not needed for the NC pin on the 128K ram chip. Is easy to add the pads though on the pcb and they don't cost anything. Realistically, many folks might be tempted to use a 512k rather than a 128k chip as they are almost the same price and then you have the flexibility to do more things. Just bridge the pads with solder.

5) Changes on the attached pdf are i) adding the second ram chip and ii) deleting the power jack on the board.

I need to remind myself that now need to design the board so the cyclone II power jack doesn't get blocked by anything on the pcb.

Is this all ok?

File Attachments

1) [Cyclone2v110.pdf](#), downloaded 540 times

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Thu, 10 Dec 2015 12:56:16 GMT

[View Forum Message](#) <> [Reply to Message](#)

James,

looks great. I did play around with some PCB price comparison sites and can see that, in general, moving from 10x10 does step up the price, so the current board size is an attractive goal (to the extent that I'd probably be happy with piggy-backing the second RAM if it didn't fit, rather than growing the board). You could maybe fit the VGA resistors inside the FPGA connector footprint, and you can certainly put the RTC module on side 2. It would be good to avoid the double-height connectors for the FPGA. Happy routing!

Neal.

Subject: Re: Multicomp PCBs

Posted by [mscane](#) on Fri, 11 Dec 2015 07:55:21 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi James,

One of the things I really like about the larger Cyclone II board is having all the connectors along

one edge.

The board sits nicely on the desk with all the cables going out the back.

However, as you said, it does add to the cost.

Since the pin layout is going to be different to the first two Cyclone II boards I wouldn't worry too much about being compatible with the previous designs (or Grant's for that matter) I generally create a new project in Quartus and assign the correct pins for that project. You can still leverage all the common components from other designs.

I also have a register (CPUid) which allows the software to know which board it is running on and adapt accordingly.

A couple of things I would like see on the next design:

1. Add CTS to each serial port (same as Cyclone IV) this is used to determine which console is used with auto boot.
2. Remove the USB connector it was the wrong type anyway.
3. To reduce board size maybe remove one DE9 and replace it with a 6 Pin header (same as Arduino serial header). That should bring it back to 10x10 with connectors on one edge I think. You can also connect a bluetooth board with the same pinout.
4. If adding a 6 pin header have a Jumper for 3/5 volts. Some USB serial cables supply 3 and other are 5. That way you can run the board from 1 USB cable.
5. If you are going to add an RTC why not use an I2C type? That way you could expand the board with other I2C chips.
6. Bring out the data bus, 3v, gnd and a couple of signals for I/O selects to a header for expansion

Has anyone considered re-working the Cyclone II module to put the pins around the 'right' way?

Cheers!

Max

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Fri, 11 Dec 2015 10:28:19 GMT
[View Forum Message](#) <> [Reply to Message](#)

Hi Max,

Great ideas there.

- 1) CTS to each port. Yes, can do that.
- 2) USB connector removed.
- 3) See below brainstormed a bit more
- 4) I might need to think about that. See the posts above, ended up removing the supply jack off our pcb and using the supply jack on the cyclone ii board instead. So then 3V and 5V comes off that board onto our board.
- 5) I2C is on the 'to do' list. But it has been on there forever... The RTC on this board is at least a working design with software written as per previous posts. Or maybe we do both? Adding I2C is really just a matter of declaring two pins on the cyclone II to be I2C and adding a couple of 10k pullups to 3V. So it is really just a header and two resistors.
- 6) some general I/O pins on this board already done, but might run out of pins if we start adding I2C and CTS. Will need to check...

I like this idea of having things all along one edge and maybe it can be done with a 10x10cm format. Hmm - , that is going to be vga, keyboard, one serial port and sd card? When I get home i'll test that out. I have ordered some little micro sd sockets on pcbs - they actually end up cheaper to buy now than a standard sized sd socket. That makes the sd part smaller. The 'single edge' design becomes more feasible.

Technically, if we are replicating Grant's project, he doesn't have either serial port converting to RS232 levels, and brings out both as TTL. So..., keeping to that philosophy, it should be quite possible to bring all the I/O to one edge. Maybe one D9 RS232 and one TTL serial port - that could be a good compromise.

I'll see if it fits. If it does, the ram chip (s) probably will end up some distance from Grant's original location, which then means it doesn't matter any more about the cyclone ii being flipped. The autorouter is easily completing on Eagle in under a minute or so.

@Max, what would be really cool would be to bring as much as possible from the cyclone IV project back to this cyclone II project. Especially your boot monitor, your multiple partitions on an sd card, and your diagnostics. They make setting up and testing boards a breeze and it is so nifty to be able to copy CP/M files between partitions rather than downloading them via serial. I think you free up some fpga space by moving some of the bootloader into the sd card. So maybe there is room in the cyclone ii for at least vga, keyboard, two serial and ?? the MMU

If it almost fits but not quite along one edge, can also move the 2 corner holes in maybe 2cm so the full 10cm is available.

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Fri, 11 Dec 2015 12:48:44 GMT
[View Forum Message](#) <> [Reply to Message](#)

Quick experiment with the PCB layout trying to get all the important things along just one edge.

If move the mounting holes back, then it is possible to get VGA, then Keyboard, then one D9, then a micro sd board just to fit in 10cm.

The micro sd module is this one

<http://www.ebay.com.au/itm/New-Mini-SD-Card-Module-Micro-SD-Card-Module-/201107614941?hash=item2ed2f2acdd:g:j64AAOSwcnpT mVvk8>

So - TV not along this edge. But ?? if you needed TV it is just two wires and may as well put a RCA socket on a front panel of a box and solder two wires over.

And the second serial port can be TTL.

The cyclone II extends off the side of the board. So can the RTC. And there could well be enough space on the board for a ESP8266 wifi module, which also should extend off the board so less interference to the antenna. Can have 3 jumpers so can connect to the second serial port. So this could add wifi.

On Grant's board he leaves 16 pins free - pins 40,41,42,43,44,45,47,48,51,52,53,55,57,58,59,60

In posts above we have allocated pins 40,41,42 to the RTC and pin 48 to VDUFFF0. And pins 51 to 60 as general purpose I/O pins to a header. So that leaves 4 pins free. Is it ok to allocate 2 of those to CTS on serial ports for full flow control, and another 2 to an I2C bus?

All still fully backwards compatible with Grant's design. Just bringing things out to headers and they can always be used for other things anyway.

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Fri, 11 Dec 2015 13:21:04 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi James, Max,

That is all sounding good. In particular, having 1 DB9 and 1 header sounds even better than 2 DB9, given that most PCs don't have RS232 so that many people will need a converter at one or other end of the cable. Please can the DB9 be "port A" (pins 103/104).

Would it be possible/worthwhile to add a 6-pin header for both serial ports, so that the Maxim/DB9 could be omitted? I haven't used any RS232 to date so I don't have a strong feeling on this, just trying to keep all the options open (after all, the autorouter doesn't sound as though it's even breaking a sweat on this, yet)

For the 6-pin header, the 3v/5v selection could be just as planned for the kbd socket. @Max: the current "plan" is to require a flying lead to grab 5V power from the FPGA card (requiring the 2-pin 5V connector on the FPGA card to mate with the 10x10 board constrains the layout too much).

Allocating the 4 spare pins to CTS and "I2C" sounds perfect.

For the RTC module, I have a mild preference for it not extending over the edge of the 10x10 -- eg by mounting it on the bottom of the board. My reasoning is that I think it's easier to get a good mechanical connection - using a 2-sided sticky foam pad. Admittedly the 10x10 then needs to be on stand-offs to keep the whole thing stable. (Of course, in the end it's your call as you are the one doing the work..)

BTW: I zoomed in and didn't like the look of the soldering on that micro sd module!

Neal.

Subject: Re: Multicomp PCBs
Posted by [Andrew B](#) on Fri, 11 Dec 2015 17:35:16 GMT
[View Forum Message](#) <> [Reply to Message](#)

When you guys get this all worked out, let's make a new board page on the wiki - since this seems like it will probably be the last iteration of the Cyclone II Multicomp PCB.

Subject: Re: Multicomp PCBs
Posted by [mscane](#) on Fri, 11 Dec 2015 19:29:00 GMT
[View Forum Message](#) <> [Reply to Message](#)

I would stick to the normal SD card connector. Those micro SD cards are too small and fiddly. Rather than put it on the back of the board put it on the front like the current Cyclone II board. That makes it easy to access and keeps the size of the board small.

So on the front you would have SD card, reset, leds etc

On the rear you would have VGA, Keyboard, DE-9. The TTL serial port doesn't have to be on the edge

That should work with 10x10.

I have a Cyclone II version of my new software pretty much working at the moment. I had to remove a few goodies (like the baud rate generator etc) but it is compatible with the Cyclone IV system. You can put the SD card in either board and boot CP/M without any problems.

Cheers!

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Sat, 12 Dec 2015 00:15:32 GMT
[View Forum Message](#) <> [Reply to Message](#)

Thanks for all those suggestions. This board is going to be brilliant!

Agree with the micro sd socket comments including the dodgy soldering - ok, stick with the standard size one.

Changes on this schematic are:

- 1) added 2 pins for reset (bring out pin 144 which is the board's reset)
- 2) I2C
- 3) remove one max3232
- 4) added 6 pin headers for TTL serial for both ports.
- 5) Added some 4 pin headers in the supply section for 5V, 3V and Gnd (always handy to have)
- 6) Added 3 leds - duplicates the ones on the cyclone ii board. Can be handy for diagnostics.

Plan is that on the back of the board, VGA then Keyboard, then D9 and on the front of the board, SD leds, reset, and TV.

Attached - preliminary PCB layout. It all fits and autoroutes. The RTC is a large module - would it be ok if it sits over the ram chips? Maybe on taller headers, or two headers stacked? Can then avoid overhang, stick it to the top of the ram chips and it doesn't take too much pcb real estate.

Addit: Fixing Leds - they are active low on the cyclone II.

File Attachments

- 1) [pcbDec12.jpg](#), downloaded 509 times
 - 2) [Cyclone2v110.pdf](#), downloaded 544 times
-

Subject: Re: Multicomp PCBs

Posted by [mscane](#) on Sat, 12 Dec 2015 06:20:34 GMT

[View Forum Message](#) <> [Reply to Message](#)

Looks great!

One suggestion: It might be a good idea to increase the clearance at the back for the VGA and serial connectors. There are few types around and some have more depth than others. Having that bit more clearance would allow a greater number of brands to be used.

Cheers!

Max

Subject: Re: Multicomp PCBs

Posted by [mscane](#) on Sat, 12 Dec 2015 11:13:26 GMT

[View Forum Message](#) <> [Reply to Message](#)

Another suggestion. If you bring out the current 8 data lines as a second row of pins under the GPIO pins (convert to 2x9 header) you would have a ready made I/O bus expansion capability.

Cheers!

Max

Subject: Re: Multicomp PCBs

Posted by [dr_acula](#) on Sun, 13 Dec 2015 00:11:16 GMT

[View Forum Message](#) <> [Reply to Message](#)

Good idea re the data bus. I hope it is ok, the autorouter is just starting to struggle so I brought out the data bus as 8 pins on a physical header just near the data bus on the cyclone II. Ended up with short traces. Ditto the reason the GPIO header is where it is. So they aren't near each other, but shouldn't matter much and like you say, could create a nifty expansion bus with those two.

I managed to find an extra 0.05" above the vga and serial connectors, is that enough?

Any other suggestions/changes from anyone?

File Attachments

- 1) [Cyclone2v110.pdf](#), downloaded 605 times
 - 2) [pcb13Dec.jpg](#), downloaded 538 times
-

Subject: Re: Multicomp PCBs

Posted by [yoda](#) on Sun, 13 Dec 2015 00:48:05 GMT

[View Forum Message](#) <> [Reply to Message](#)

Just a thought - how about swapping SP3232 with TTL serial headers. That way if a user want so only use TTL serial with USB adapters they don't populate DB9 and cables can run there. With current have to run the cables over the top of DB15 which may not be optimal.

Subject: Re: Multicomp PCBs

Posted by [dr_acula](#) on Sun, 13 Dec 2015 03:32:27 GMT

[View Forum Message](#) <> [Reply to Message](#)

Good idea yoda. And the GPIO and Data headers now end up next to each other. Is this ok?

File Attachments

- 1) [PCBDec14.jpg](#), downloaded 524 times
-

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Sun, 13 Dec 2015 12:05:23 GMT

[View Forum Message](#) <> [Reply to Message](#)

James,

the board is looking better and better. Here are my comments/suggestions

1/ I note the VGA connector has +5V. This is inherited from your earlier design, but Grant's design (and mine on veroboard) had 3.3V. Is 5V necessary here? It creates another dependency on 5V.

2/ the designator for the 3-pin kbd voltage jumper "3V3_k_5V" should be "5V_K_3V3" (or "5V_K_3V" for consistency) - at the moment the silk screen is marking the 3V3 side as 5V and vice-versa.

3/ The jumper VDUFFF0 should be named VDUFFD0. I apologise for this mistake in my hand-drawn schematic. Not worth fixing if it will result in a lot of work.

4/ suggest changing the designator for the RTC connector from "RTC" to "RTC_3V_0V_CLK_IO_CE"

5/ C21 C22 suggest swapping silk screen value/designator to match C23 C24 (getting really picky here, only because the connector might obscure the lower text)

6/ suggest a review of the 3V connections to see whether there's any opportunity to add more traces to "grid up" the power

7/ likewise 0V (I can't inspect these from the JPG because where tracks are coincident top/bottom only one track is visible)

Additional annotation for the schematic :

1/ in VGA add text "HSYNC" to PIN_71 and "VSYNC" to PIN_72

2/ in LED add text "SD ACTIVE" to PIN_3

3/ in RTC add text "SCLK" to PIN_41 and "I/O" to PIN_40 and "CE/RST" to PIN_42. Add general text "RTC module contains DS1302 and button cell"

4/ in GPIO add text "GPIO7" to PIN_60 ... "GPIO0" to PIN_51

5/ in DATA BUS add text "D0" to PIN_119 .. "D7" to PIN_122

6/ in VDUFFD0 add general text "Fit link to swap address decode of VDU and Serial A"

7/ in Upside down socket:

In Serial A add PIN_45

In Serial B add PIN_47

Delete Group A/Group B

Add CLK PIN_17

Change "Input only group" to "Unused input only group" and remove PIN_17

Add RTC PIN_40 PIN_41 PIN_42

Add GPIO PIN_51 PIN_52 PIN_53 PIN_55 PIN_57 PIN_58 PIN_59 PIN_60

Add I2C PIN_43 PIN_44

Add VDUFFD0 PIN_48

Delete comment "use stackable female headers"

Maybe add comment "<http://www.retrobrewcomputers.org>"

Delete comment "I/O spare pins.."

regards,

Neal.

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Sun, 13 Dec 2015 12:57:02 GMT
[View Forum Message](#) <> [Reply to Message](#)

Thanks ++ Neal.

I think all of those should be possible. I have work commitments for the next 48h but after then should have some time to make all these changes. Re points 6 and 7, do you mean make the 3V an 0V traces thicker?

Subject: Re: Multicomp PCBs
Posted by [nealcrook](#) on Sun, 13 Dec 2015 17:35:38 GMT
[View Forum Message](#) <> [Reply to Message](#)

>> Re points 6 and 7, do you mean make the 3V an 0V traces thicker?
Well, widening those traces is never a bad thing, but not quite what I was thinking about. Anywhere where (eg) the 0V trace connects to one (non-decoupling cap) component and then goes on to connect to another (ie, daisy-chains) see if there's room to add a trace that connects the far end of the chain back to one of the FPGA 0V connections.. or to any other 0V connection. The end result will be a set of loops -- so that the whole topology is a mesh rather than a star or daisy-chain. This is a simple board so I don't expect any problem but having been bitten by ground bounce one before (and not even realising what had happened until some years later) having the best 0V distribution possible has no down-side. Same (to a lesser extent) for 3V3.

Neal.

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Mon, 14 Dec 2015 13:56:25 GMT
[View Forum Message](#) <> [Reply to Message](#)

Hi Neal,

Wow, great list there. I have been through it one line at a time, *hopefully* have left nothing out!

Autorouter is on 12 mil for the data lines and 25mil for the ground and 3V lines - that is about the best it can do. Anything more than 25mil for power and it fails. Adding a ground plane also fails with polygons that fall apart. So the whole decoupling star thing is going to be hard unless hand rout it all. Which is the difference between 1 minute and many days. My experience though with much more complex boards with many more chips is the decoupling and ground bounce seems ok, so long as decoupling randomly placed around the board so average trace length is low to a 0.1. I think the SD needs 0.1 and 470uf near it - have learned that the hard way. The sp3232 ends up with short traces, and it is really just the ram chip where the decoupling is not perfect. However, multiple boards on the cyclone II and IV seem ok with the less than perfect autorouter.

The attached files are .bmp so can zoom in better than jpg. Lots of updates on the schematic. The gerber is downloaded back from Seed and they always seem to make the fonts slightly larger so

have tried to allow for that.

Have changed the title so leaving out my name and replacing it with www.retrobrewcomputers.org. This board truly is an international joint effort now

I had a PM about the eagle source code - this is attached below as well. It will likely change, but at least here is a snapshot for anyone who wants to fork this. This project is open source.

Please let me know if I have left anything out.

File Attachments

- 1) [cyclone110.bmp](#), downloaded 496 times
 - 2) [cycl110brd.bmp](#), downloaded 532 times
 - 3) [Cyclone2v110.pdf](#), downloaded 532 times
 - 4) [Cyclone2v110_eagle_source.zip](#), downloaded 457 times
-

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Mon, 14 Dec 2015 23:37:21 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi James,

nice job. Much Kudos to you for all the work and for sharing the results. I have just placed an order through the excitingly-named www.dirtypcbs.com -- Boards due in about 30 days. The Eagle .brd file uploaded OK and the rendered Gerber they generated looked OK on quick inspection. I will report back on progress. I plan to take some photos during assembly for the Wiki, and I expect to have some spare boards available at about £2.00 each in due course.

Neal.

Subject: Re: Multicomp PCBs

Posted by [mscane](#) on Tue, 15 Dec 2015 02:58:53 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi James,

You guys are too quick for me...

I think there is an error in the TTL wiring of the serial ports.

The intention was to make them Arduino serial port compatible so you could plug in a USB adapter, Bluetooth adapter, USB serial cable etc.

I'm looking at the ones at Sparkfun and the pinouts are to as follows:

1. Ground
2. CTS (Output)
3. Power (5V or 3.3V)
4. TX (output)
5. RX (input)
6. RTS (input)

I think your schematic should have the following connections:

Serial A

FPGA	-	DIR	-	SP2332	-	TTL
103 (TX)	->	11 (T1-In)	->	5 (RXI)		
101 (RX)	<-	12 (R1 Out)	<-	4 (TXO)		
104 (RTS)	->	10 (T2 In)	->	2 (CTS)		
45 (CTS)	<-	9 (R2 Out)	<-	6 (RTS)		

TTL Ground - 1

TTL Power - 3

Serial B

FPGA	DIR	TTL
100 (TX)	->	5 (RXI)
99 (RX)	<-	4 (TXO)
96 (RTS)	->	2 (CTS)
47 (CTS)	<-	6 (RTS)

TTL Ground - 1

TTL Power - 3

Also, since the USB adapters can supply 3.3 volts, it might be a good idea to put a jumper on the power pin (3) of the TTL headers so that you can isolate the power. I don't think the 5V->3.3V regulator on the FPGA board would be happy with an external 3.3 volt supply.

If the header is used to supply power (eg Bluetooth card etc) then you can put the jumper in to power the card.

Cheers!

Max

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Tue, 15 Dec 2015 03:47:17 GMT
[View Forum Message](#) <> [Reply to Message](#)

Hi Max,

I thought about that, but figured that there are probably all sorts of adapters out there and no real standard for layout. eg I've got some of these
http://www.ebay.com.au/itm/RS232-To-TTL-Converter-Module-Bui-lt-in-MAX232CPE-Transfer-Chip-With-4PCS-Cables-/280839635021?hash=item416358c04d:m:mqCJoT5QIM9ODooNWAHK_0A but they are not even on a single row header.

Happy to change the order of the pins if needed though.

Subject: Re: Multicomp PCBs
Posted by [mscane](#) on Tue, 15 Dec 2015 05:00:42 GMT
[View Forum Message](#) <> [Reply to Message](#)

Yes I was thinking that Arduino is pretty popular so there should be plenty of devices compatible with its serial port.

How about these:

USB to TTL adapter
<https://www.sparkfun.com/products/9873>

Bluetooth Adapter
<https://www.sparkfun.com/products/12580>
This one works quite well. I have had it working on the Cyclone IV and switched console to it

USB to serial cable
<https://www.sparkfun.com/products/9717>
This one has 3.3 volt signals but 5 volt power which might be an issue (hence the jumper suggestion)

Cheers!

Max

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Tue, 15 Dec 2015 05:09:54 GMT
[View Forum Message](#) <> [Reply to Message](#)

Should be ok to swap them around.

Subject: Re: Multicomp PCBs
Posted by [Andrew B](#) on Tue, 15 Dec 2015 07:29:04 GMT
[View Forum Message](#) <> [Reply to Message](#)

There is a page for this board at <http://www.retrobrewcomputers.org/doku.php?id=boards:sbc:multicomp:cycloneii-c:start>

Subject: Re: Multicomp PCBs
Posted by [nealcrook](#) on Tue, 15 Dec 2015 08:02:16 GMT
[View Forum Message](#) <> [Reply to Message](#)

James,
if you mod. the board now please will you add a "REV2" marking near the www.retrocomputers.org to distinguish them from the boards I have in the 'shop.
thanks,
Neal.

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Wed, 16 Dec 2015 09:51:43 GMT
[View Forum Message](#) <> [Reply to Message](#)

Ok, I'm kind of stuck with that modification.

Sparkfun sell two products - this TTL cable <https://www.sparkfun.com/products/9717> and they say "This cable has almost the same pinout and functionality as our FTDI Basic Breakout board" and link to this board <https://www.sparkfun.com/products/retired/8772>

But the operative word there is "almost" because one is bringing out the CTS/RTS pair and the other is bringing out CTS and DTR. Going back for years, I think picaxe and arduino and basic stamp have used DTR as their reset pin. Probably because it was never used for anything much on a RS232 link so the pin was kind of spare.

Go to ebay and search with FTDI board or cable or breakout and there are a whole range of modules that come up that are meant for arduino, and they are all using the DTR pin. A 'proper' cable would bring out eight pins but they all seem to bring out just six. Sort on price and the cheapest cable that brings out the RTS/CTS pair we want is this one <http://www.ebay.com.au/itm/FTDI-FT232RL-USB-to-Adapter-Module-USB-TO-TTL-RS232-Arduino-6Pin-Cable-/191732706593?hash=item2ca428ed21:g:bM4AAOSwo0JWPH4b>

But the order of the pins is

Different to Max's suggestion, different to the board we made and different to sparkfun's.

Also, some boards are 5V and some are 3V so it is not going to be a 6 way header but more likely individual jumper leads with power not connected? In which case it doesn't matter what order the pins are, so long as they are clearly labeled.

Hmm, now I'm thoroughly confused.

Subject: Re: Multicomp PCBs

Posted by [dr_acula](#) on Wed, 16 Dec 2015 12:06:09 GMT

[View Forum Message](#) <> [Reply to Message](#)

Talking to Max offline, suggestion is to leave it as it is so no revision 2.

So, Andrew B, I'm not having much luck moving files into the wiki. I think I have about 5 different user names and passwords for the wiki all with the one email address. So, plan B:

Attached here are the eagle files for the Cyclone II board, and also the Cyclone IV board. Any chance you could move these to the correct wiki?

This includes gerbers and should be enough to send straight to places like Seeed PCB.

For editing the eagle files, I am hoping it has included custom library parts, but if not, I can add these.

File Attachments

-
- 1) [CycloneIVJune2015.zip](#), downloaded 639 times
 - 2) [Cyclone2v110.zip](#), downloaded 505 times

Subject: Re: Multicomp PCBs

Posted by [lenzjo](#) on Wed, 16 Dec 2015 12:09:04 GMT

[View Forum Message](#) <> [Reply to Message](#)

You guys have been busy! Really didn't expect this.. I've been away for a week or so - had to deal with RL...

Regarding the FTDI, if you are trying to keep it "Grant Searle compatible" maybe it might be a good idea to keep the FTDI pinout he uses on his board so there's no confusion as people will probably go to his site if they haven't already for documentation. Otherwise you have to include documentation with the pcb that stresses the difference between your's and his pinout.

@nealcrook: Can you put me down for one of your pcbs when they get here please?

Subject: Re: Multicomp PCBs

Posted by [dr_acula](#) on Wed, 16 Dec 2015 12:29:27 GMT

[View Forum Message](#) <> [Reply to Message](#)

Re the ftdi, Max and I already sort of broke compatibility with Grant's board by adding bidirectional flow control with RTS and CTS. We have tried to make it backwards compatible as much as possible - I think this board is backwards compatible so it can run all the things on Grant's site. Though it adds more - two ram chips, a different bootloader that shifts more code into the SD card, which frees up more LE space in the FPGA, which means room for a MMU, multiple partitions on one SD card, and maybe other things like variable baud rates if we can get it to fit.

BTW, offline, Max has just managed to get the cyclone IV bootloader to run on the cyclone II. This means all the diagnostic stuff like detecting ram chips will also work on the cyclone II.

On another topic, and I sent Max a PM about this, the latest Quartus version is needed to run the cyclone IV, but I think Grant says you need an older version of Quartus to run the cyclone II. I wonder if there is a way to program the cyclone II on the latest version of Quartus? I don't think you can looking at <http://dl.altera.com/devices/> which is a bit of a pain having to have two versions of quartus on one PC.

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Wed, 16 Dec 2015 13:24:58 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hey! This is officially the busiest topic on the forum (the first to get to 2 pages)

I read (<http://stackoverflow.com/questions/11325105/cyclone-ii-fpga-active-serial-using-usb-blaster>) that you can program from the command line thus:

```
quartus_pgm c byteblasterII m jtag o bpv;design.pof
```

I guess that is the best chance of being able to program an old device from the latest tools.

@James: is the Cyclone2v110.zip you posted above equivalent to the earlier eagle zip or are there any (design) changes?

@lenzjo: OK, have you down for a PCB.

I guess Grant is busy doing other stuff; he hasn't updated his web page for a while. I swapped email with him in the summer and he kindly gave me permission to repost files for the 6809 version of multicomp.

Regarding serial port hookup, I anticipate that we will provide build-instructions on the Wiki that will remove any ambiguity about connections to the PCB, and provide links or repositories for the 6809 software and the z80 software. As James says, we are still compatible in the sense that an FPGA built from Grant's design can run on the new board with no change (not even pin assignment changes).

It would be nice if someone evolved the 6502 variant further. I read on www.6502.org that some folk there had built Grant's design, but I don't know whether anyone went further than using the software that Grant provided.

Neal.

Subject: Re: Multicomp PCBs
Posted by [dr_acula](#) on Wed, 16 Dec 2015 23:32:52 GMT
[View Forum Message](#) <> [Reply to Message](#)

Hi Neal,

No, no design changes. I was anticipating a change, so I ripped up the traces but then changed nothing and redid the traces, so it will be the same.

Re my question about the versions of Quartus, Max has just sent me a message about this - Quartus 13.1 will also program cyclone IV.

And yes, it would be great to do clever things with the 68xx emulation as well.

Subject: Re: Multicomp PCBs
Posted by [ian-may](#) on Fri, 18 Dec 2015 10:10:17 GMT
[View Forum Message](#) <> [Reply to Message](#)

Regarding eBay TTL to RS232 modules -
I purchased some of these (AU\$3.80)
<http://www.ebay.com.au/itm/MAX232-RS232-TTL-Converter-Adapter-Module-Board-New-/171289209569?hash=item27e1a1eae1> which actually have max3232 chips and bring out RTS and CTS on a dual row header. The listing states 5 Volt power required. Unfortunately there are only 5 available at the moment - so probably not much use to anyone else.

I also purchased some of these (which at AU\$1.19 each are astoundingly cheap)
<http://www.ebay.com.au/itm/MAX3232-RS232-Serial-Port-TTL-Converter-Module-Transfer-Chip-W-4-Jump-Cables-/201347929016?hash=item2ee14593b8> . On this module the remaining max3232 transmitter and receiver are unconnected so adding a few wires will allow connection to RTS and CTS but they will have to be connected via wires at the TTL end since there are no spare header pins. The listing claims 3.3-5.5 Volt supply.

Neither module should actually be used with a 5 Volt supply because all of the capacitors are 0.1uF which according to the max3232 data sheet (page 9) are OK for a 3.0-3.6 Volt supply only. 0.47uF capacitors are needed for 3 of the capacitors for a 5 Volt supply. The first module does not have a bypass capacitor on the supply line either which probably isn't a good idea.

Note that the module that James referenced in message #177 (http://www.ebay.com.au/itm/RS232-To-TTL-Converter-Module-Bui-It-in-MAX232CPE-Transfer-Chip-With-4PCS-Cables-/280839635021?hash=item416358c04d:m:mqCJoT5QIM9ODooNWAHK_0A) has a specified supply voltage of 3.3-5 Volts in the Ebay listing but if it really is a MAX232CPE it can only be run on 5 Volts which means 5 Volts on the outputs which is going to be BAD for the FPGA inputs.

In summary, a module with a MAX3232 will be required since it must be run from the 3.3V supply or the FPGA inputs will likely be damaged.

Hopefully this information will be helpful to someone

Cheers, Ian.

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Thu, 24 Dec 2015 18:58:43 GMT

[View Forum Message](#) <> [Reply to Message](#)

My PCB order has shipped (on a slow plane from China). The passives arrived in a jiffy bag today. Waiting, waiting. Soon it will be Christmas peace,
Neal.

Subject: Re: Multicomp PCBs

Posted by [Andrew B](#) on Sun, 27 Dec 2015 18:25:11 GMT

[View Forum Message](#) <> [Reply to Message](#)

I'll get the files uploaded to the wiki, want to make sure we are correct on the versions though - this is Cyclone II-C (3rd version of the Cyclone II) and (2nd version of Cyclone IV), correct?

Can we get pictures of these boards from people who have build them as well?

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Tue, 05 Jan 2016 21:13:26 GMT

[View Forum Message](#) <> [Reply to Message](#)

I'm excited to report that the boards arrived today. I ordered them on 14th Dec and they arrived on 05th Jan, using the cheapest shipping option to the UK. Nice job from Dirty PCBs.

I haven't started assembly yet (*and* I'm still waiting for the FPGA board to arrive.. The one I already have has a short between two of the pins that are assigned for the RAM connection). I put some of the connectors in place and things seem OK for fit and clearance.

I will do some nicer photos than this for the WIKI - both of the bare board and the assembly

process.

I will have about 7 PCBs spare and they will be available at £2.20 each including UK postage. Best to wait a few days and see it work first though!

Neal.

File Attachments

1) [pcb_photo2.jpg](#), downloaded 1443 times

Subject: Re: Multicomp PCBs

Posted by [Andrew B](#) on Wed, 06 Jan 2016 06:23:13 GMT

[View Forum Message](#) <> [Reply to Message](#)

I am excited to see the first boards with the new website address!

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Thu, 07 Jan 2016 00:14:40 GMT

[View Forum Message](#) <> [Reply to Message](#)

DirtyPCBs.com offer a service where they keep hold of your data so that you can reorder boards. Also, you can share your data so that anyone else can order boards without uploading any data. I have enabled sharing in case anyone else wants to order a batch for shipping to their geography (\$25 for 10 +/- 2, including shipping). The design is called "Cyclone2v110" (no, I didn't select the option where they bump up the price and give me a kick-back).

Board #1 is part-assembled and I have done a bunch of updates on the WIKI including photos. <http://www.retrobrewcomputers.org/doku.php?id=boards:sbc:multicomp:cycloneii-c:start>

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Thu, 07 Jan 2016 00:45:44 GMT

[View Forum Message](#) <> [Reply to Message](#)

it's alive. 6809 with all the trimmings, but running with bare minimal external hardware: internal RAM only. VDU is running 40 columns because there isn't room in the FPGA For both internal RAM and the 80-column VDU. Yes, that VGA monitor definitely needs dusting.

File Attachments

1) [minimal_boot.jpg](#), downloaded 1288 times

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Fri, 08 Jan 2016 23:38:23 GMT

[View Forum Message](#) <> [Reply to Message](#)

Everything is now tested, and there is a full build/test write-up on the WIKI. I even managed to fix the adjacent-pin short on my FPGA board so I was able to boot from the SDcard image into FLEX, CUBIX and NITROS9. The only problem I experienced was that I could not get the RS232 to work, connecting to my old laptop with a null modem cable. I got unreliable intermittent operation as though the baud rate was wrong. I need to investigate further.

Question: the schematic shows an SP3232. Is this the same as a MAX3232 (I hope so, that's what I fitted).

I need to upload an SDcard image and the FPGA project files - should be done over the weekend.

Neal.

Subject: Re: Multicomp PCBs

Posted by [dr_acula](#) on Sat, 09 Jan 2016 05:10:55 GMT

[View Forum Message](#) <> [Reply to Message](#)

Looking very good!

Re the SP3232, I have about 50 so called max3232 chips from ebay, and they are all fakes. They tend to work in one direction and give garbage in the other direction. They all have max3232 written on them. I got them from four different suppliers as a test. The common feature is they all cost under \$3, and as low as 30c each. A real max3232 can be \$7 or more. So I specified the SP3232 as these don't seem to exist as fakes anywhere I can find, and also they are about a third of the price of genuine max3232 chips.

Search fake max3232 on google eg

<http://blog.bogpeople.com/2015/01/fake-max3232-rs232-line-drivers.html>

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Sat, 09 Jan 2016 16:15:27 GMT

[View Forum Message](#) <> [Reply to Message](#)

>> I have about 50 so called max3232 chips from ebay, and they are all fakes
Thanks for the explanation. I think I had heard this from you before, because I bought mine from a reputable UK distributor rather than EBAY. I will boot up my bitscope and take a look at some signals.

In "other news", the WIKI is now fully populated with 6809 FPGA and SDcard images, and I posted the schematic and eagle .zip file.

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Mon, 11 Jan 2016 21:37:27 GMT

[View Forum Message](#) <> [Reply to Message](#)

OK, I worked out my problem with the serial port. I expected the connector to be male (as on a PC) and to use a null-modem interconnect cable (2-3, 3-2, 5-5). Eventually I realised (and then saw the connector labelled F on the schematic) that you expected the connector to be Female (as on a MoDem) presumably with an interconnect cable wired straight (2-2, 3-3, 5-5).

I need to make up a new cable to verify with my PC but I got the loopback working OK and it looked good on the scope.

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Tue, 12 Jan 2016 20:52:57 GMT

[View Forum Message](#) <> [Reply to Message](#)

After sitting down with a glass of wine I decided my easiest fix was a gender-bender-twist adaptor, and sure enough it all works beautifully now. Wiki updated accordingly. Also, I swapped email with Grant and sent him a PCB.

Subject: Re: Multicomp PCBs

Posted by [computerdoc](#) on Sat, 16 Jan 2016 12:42:31 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi Neal, James and Max,

I have been wondering why I haven't seen any emails for a little while and it suddenly dawned on me that this forum is the only way to communicate. Maybe forum messages could also be emailed as well. I like having all me email in one place.

Now on to the Cyclonell and Cyclone IV boards. I'd like to order at least one of each with the possibility of getting 2 of each. I want to be a tester again for these boards. By the way James, I am in the process of adding a DS1302 RTC chip to the v1.09 PCB, the one with all the connectors on one edge except for the TV. I'm going to use a little 0.33F supercap for VBAT instead of a lithium battery.

Neal, The DS1302 has a trickle charge register that can be set for 1 or 2 diodes and 2K, 4K or 8K resistors. Could you add code for this register with some way for the user to chose battery or supercap?

Of course on the new boards I'll get what you guys have decided on.

Is this doable?

Subject: Re: Multicomp PCBs

Posted by [nealcrook](#) on Sat, 16 Jan 2016 16:14:43 GMT

[View Forum Message](#) <> [Reply to Message](#)

>> I'd like to order at least one of each with the possibility of getting 2 of each
I can't speak for the Cyclone IV boards, but for the new Cyclone II boards, aka, V110, I suggest 2 options: (1) for \$25 including postage you can order a batch of 10 directly (see above) then sell off the extras in the USA if folk are interested. (2) I will sell you a couple, but with airmail postage that will add up to \$8.55

>> Could you add code for this register
You can test the RTC without having the charging working (it just won't be non-volatile). I'm just putting the finishing touches to some CamelForth mods that will (finally) allow code to be stored to SD. After that, I'll look at making this mod. Should be straightforward.

Subject: Re: Multicomp PCBs
Posted by [computerdoc](#) on Tue, 19 Jan 2016 16:49:35 GMT
[View Forum Message](#) <> [Reply to Message](#)

Hi Neal,
Thanks a bunch. I think I will order some boards as you suggested. They are the best Cyclone II boards yet. With 10 boards, I'll be able to retire most of my earlier boards.
James, Please let me know when the Cyclone IV Multicomp is ready. I really appreciate all the effort you have put into your Multicomp PCBs. Thanks a bunch.

Subject: Re: Multicomp PCBs
Posted by [gkaufman](#) on Mon, 25 Jan 2016 16:18:36 GMT
[View Forum Message](#) <> [Reply to Message](#)

A huge thanks to Neal, James, Max and others.

The board went together easily and works nicely.

I had 10 boards fabricated thru dirtypcb, delivery was pretty quick and the quality looks fine. Amazing for \$25.

- Gary

Subject: Re: Multicomp PCBs
Posted by [computerdoc](#) on Wed, 27 Jan 2016 06:07:33 GMT
[View Forum Message](#) <> [Reply to Message](#)

Hi James,
Thank you for mentioning this. I know this is not the topic for this thread, but I have to tell you that your message about the fake MAX232 chips you received from ebay is just what I needed know to revisit troubleshooting my Mark IV board. I'm going to order some more from Digikey and see if I have success. Thanks a bunch for your keen insight. I really appreciate this.

Subject: Re: Multicomp PCBs

Posted by [computerdoc](#) on Wed, 27 Jan 2016 11:27:45 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi James,

I'm putting together an order for parts for the Cyclone IV v1.3 board, but the part numbers you cited for the switching regulators and the ones I found are different. They seem to be backwards. The part numbers you used were non-stock items for me. I want to be sure I'm getting compatible parts. Here are the urls.

< <http://www.digikey.com/scripts/DkSearch/dksus.dll?Detail&itemSeq=188602895&uq=635894685996415881>>

< <http://www.digikey.com/scripts/DkSearch/dksus.dll?Detail&itemSeq=188603003&uq=635894685996415881>>

Thank you for your help.

Subject: Re: Multicomp PCBs

Posted by [dr_acula](#) on Thu, 28 Jan 2016 03:08:45 GMT

[View Forum Message](#) <> [Reply to Message](#)

Sure, no problem, but can you repost the links, they both go to a generic search page on digikey.

Subject: Re: Multicomp PCBs

Posted by [computerdoc](#) on Sat, 06 Feb 2016 08:54:01 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi Neal, James and Max,

Another Multicomp Cyclone IV v1.3 PCB is soldered up and passes the smoke test. No Smoke! Whoo-Hoo! Now to get the FPGA programmed and an SD Card image written to the SD Card then I will be able to another Cyclone IV Multicomp is online! Just a few more hours...

Subject: Re: Multicomp PCBs

Posted by [Andrew B](#) on Tue, 09 Feb 2016 07:49:26 GMT

[View Forum Message](#) <> [Reply to Message](#)

Just received one of the Cyclone II-C boards from Gary K. Nice to see the new web address on the board!

Subject: Re: Multicomp PCBs

Posted by [computerdoc](#) on Tue, 08 Mar 2016 06:53:05 GMT

[View Forum Message](#) <> [Reply to Message](#)

Hi Guys,

I have finally received my Cyclone II-C boards and I was wondering what modules to order off ebay. I've started to build it with the parts I have on hand and I'm about ready to order what's needed to finish the build. Please let me know which modules you all recommend. Thanks a bunch.

Subject: Re: Multicomp PCBs
Posted by [nealcrook](#) on Thu, 30 Mar 2017 22:51:30 GMT
[View Forum Message](#) <> [Reply to Message](#)

I logged in to my dirtyPCBs account today and saw that 17 people have ordered batches of these PCBs since I uploaded James's gerbers. Added to the original 10 that I ordered, that's 180 PCBs -- not to mention folk who have had boards fabricated elsewhere. That's almost mass-production!

Neal.

Subject: Re: Multicomp PCBs
Posted by [tor](#) on Fri, 31 Mar 2017 10:36:45 GMT
[View Forum Message](#) <> [Reply to Message](#)

Well, yes.. we can only order 10 or more.. I expect to use only two of the ten I got made. And afaiik there aren't anyone else nearby who would be interested in getting one from me.

Subject: Re: Multicomp PCBs
Posted by [nealcrook](#) on Thu, 13 Apr 2017 20:34:12 GMT
[View Forum Message](#) <> [Reply to Message](#)

From another poster here I learned that I can reference my upload of James's design files at dirtypcbs like this (saves having to browse through pages and pages of designs since the search facility seems less than useful:

<http://dirtypcbs.com/store/designer/browse/9663>

since most people ordering boards will not use all 10, I suggest that anyone with excess boards post on this topic, mentioning where in the world they are. I have sold boards (at cost+postage) to folk in the UK and the EU.

Neal.

Subject: Re: Multicomp PCBs
Posted by [kubik](#) on Fri, 21 Apr 2017 15:00:39 GMT
[View Forum Message](#) <> [Reply to Message](#)

I've got a few spares of the PCB for the Cyclone IV FPGA board. If you are interested, let me know. The boards are located in Munich, Germany.

Subject: Re: Multicomp PCBs

Posted by [znac049](#) on Tue, 09 May 2017 08:52:27 GMT

[View Forum Message](#) <> [Reply to Message](#)

I have some spare V110 pcbs for the Cyclone-II board in the UK. Message me if interested. I also have a drawer full of the SD card slots.

-Bob
