
MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

WHAT IT DOES

The MIDI2CLOCK (M2C) takes a MIDI input, and outputs CLOCK pulses and gates, START, STOP and CONTINUE pulses. These can be either +5V or +10V by installing pairs of resistors, or wire links. An onboard two-stage divider network with GATE and TRIGGER outputs allows for combinations of different pulse divisions, pulse widths and time signatures to be set up, using a rotary switch and 8 SPST toggle switches. The INITIAL DIVIDER stage also has a pulse output.

It decodes MIDI CLOCKS, **not** MIDI TIMECODE (MTC). It's intended for use with analogue synthesizers and modules that require either GND to +5V or GND to +10V gates and triggers.

It's designed to run from $\pm 15V$ power supplies, but $\pm 12V$ should work OK with no part changes.

There are LED indicators for the various outputs, and a MIDI RECEIVE LED that lights whenever MIDI is received at its MIDI input. There are also several pin-headers on the PCB which can be used for future expansion. Other MIDI messages could be decoded using these and some extra circuitry, and CV to MIDI transmission is also possible, again with additional circuitry. No programmable ICs are used, and all components are in current production as of August 2011.

BEFORE BUILDING THE MIDI2CLOCK, REMEMBER TO TAKE ANTI-STATIC PRECAUTIONS WHEN HANDLING CMOS DEVICES, AND PLEASE READ THE DISCLAIMER AT THE END OF THIS DOCUMENT.

BUILDING THE M2C

I recommend testing all the diodes, resistors, capacitors and transistors before installing them. The build may take a bit longer, but you can be sure that each component is the correct value and actually works. This will save a lot of time in the event you ever have to debug your build. I also recommend printing out the BOM and marking each component on it as they are installed - this helps avoid missing a part out (which is surprisingly easy to do!)

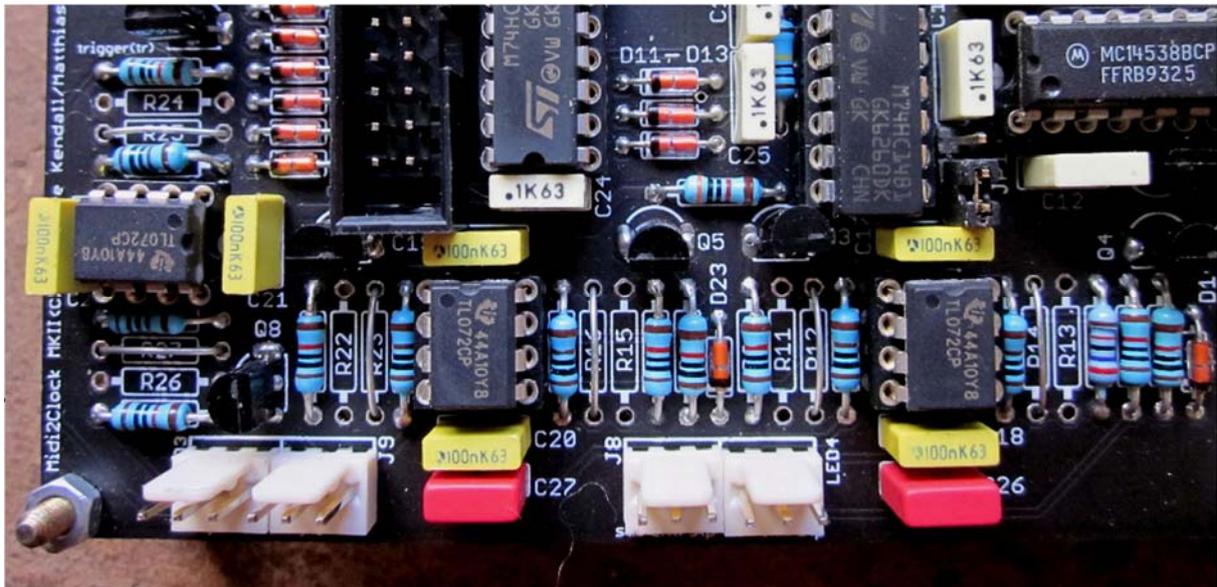
Remember to use anti-static precautions when handling CMOS ICs. Most modern CMOS ICs are pretty robust, but on a cold dry day, with your feet on a nylon carpet, you can definitely risk killing a chip with a static discharge. You **really** wouldn't want to accidentally do that to U3...

MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

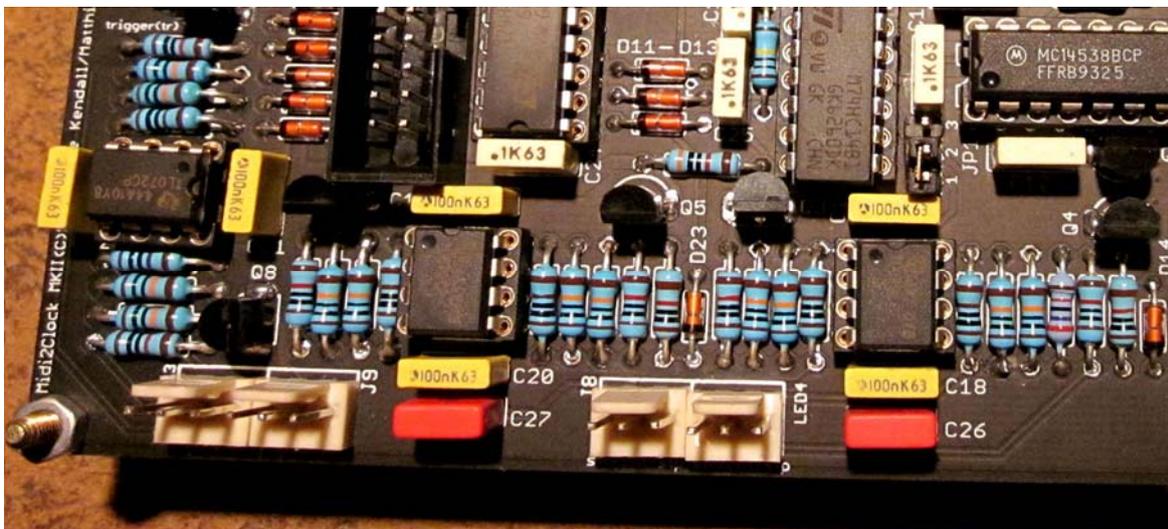
RESISTORS AND DIODES

If installing the resistors and diodes first, as many people do, first decide whether you want the GATE, and other pulse outputs (START, STOP, CONTINUE, TRIGGER and INITIAL DIVISION) to be +5V or +10V. For +5V pulses, install wire links for R12, R14, R16, R23, R25, and R27. Do not install anything in the positions for R11, R13, R15, R22, R24 and R26.

The picture below shows the positions for the links and resistors for +5V outputs.



For 10V gates and pulses, install 100K resistors for R11, R12, R13, R14, R15, R16, R22, R23, R24, R25, R26 and R27. The picture below shows the resistors installed for +10V outputs.



R17, R18, R19, R28, R29 and R30 are all 1K protection resistors for the opamps used in the GATE and other outputs. R5 and R8 MUST BE metal film resistors, as they're used as part of two timing circuits for the 4538 dual monostable. It's probably best to use 0.25W - 0.6W metal film resistors throughout, as the cost is scarcely higher than that of carbon resistors.

MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

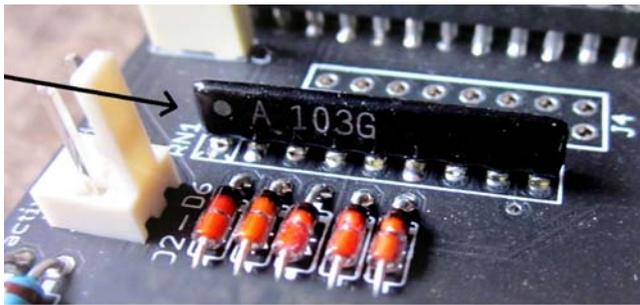
RN1 is a 9-pin 10K commoned resistor network (RESNET), with 8 separate resistors, where pin 1 is the common, and is connected to GND (or 0V) in the M2C circuit. The “top” ends of the 8 resistors are joined to the common. The other 8 ends of the resistors are connected to the pads for pins 2 - 9 of RN1 on the PCB.

If you have difficulty sourcing RN1, you can make your own home-brew resistor network by mounting resistors vertically, and joining their top leads together as shown in the picture below.

RN1 10K RESNET, standard metal film resistor, 1N4148 diode and (right) homebrew 10K RESNET.



The arrow below shows pin 1 of RN1. The photo also shows the correct orientation of D2 - D6



Ferrite beads F1 and F2 could be replaced with low ohm value resistors, say 10 - 22 ohm.

Diodes D1 to D23 can either be 1N4148 or 1N914. Check when installing them that the black band on the body of the diode lines up with the white band on the PCB markings. Check the photos in this document, if unsure of the the correct orientation of the diodes.

MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

ICs AND IC SOCKETS

I recommend using high quality IC sockets, for example turned-pin sockets throughout the PCB (except for X1 - see below for details). If you don't use sockets for every IC, then definitely use sockets for U2, U3 and U4. U2, the 6N138 opto-isolator, can be a fragile little chip, and U3 is a large and relatively expensive IC that would be very tedious to de-solder if the need ever arose.

Take great care when straightening the pins for U3 and when installing it in its socket. Because of the large number of pins, it's easy for one or more to get bent and squashed under the chip, and the circuit may then not work at all..... Take it slowly and carefully on this one.....

All of the IC's except U2, U3 and U4 are 74HC types, with a maximum recommended positive power supply voltage of +5V. In some cases it should be possible to use regular CMOS types. If doing this, be aware that Motorola ICs add a "1" to the front of the chip type, so for example a Philips HEF4017 has the same functionality as a Motorola MC14017.

U1 is a 74HC4040. An early prototype used a couple of regular 4040s (CD4040 and MC14040).

U4 is a regular 4538, as I found it harder to source 74HC4538 types from my regular suppliers. If using a regular CD4538, R5 should be 27K, whereas if U4 is a 74HC4538 R5 should be 39K. The PCB has only been tested with a CD4538 and MC14538 so far, which both work perfectly well.

U7 and U8 are 74HC4017s, although regular CD4017s have been used successfully. The HC4017 is recommended if you have a choice, due to its higher operating speed at +5V. There will probably not be any noticeable difference, and if sockets are used both types can be tried.

X1 can be in a few different packages. It is a crystal oscillator (XTAL OSC), and it's a different part to a regular 2-pin crystal which needs a small network of extra components to function correctly. The 3 types of XTAL OSC that will fit on the PCB are a +5V DIL 8-pin SPXO, an 8-pin +5V metal can, and a 14-pin +5V metal can. The SPXO is the neatest fit if using an IC socket. If using the metal can packages, note that the pins have to be cut down, as they are too long and otherwise the chip will not fit flush with the top of the socket. They also don't fit quite as snugly in the socket as the 8-pin SPXO type, so it's probably best to directly solder a metal can type.

The XTAL OSC used in all the prototypes was a 4mHz type. There is a jumper (J1) that allows the optional use of 8mHz or 16mHz types, although this has not yet been tested.

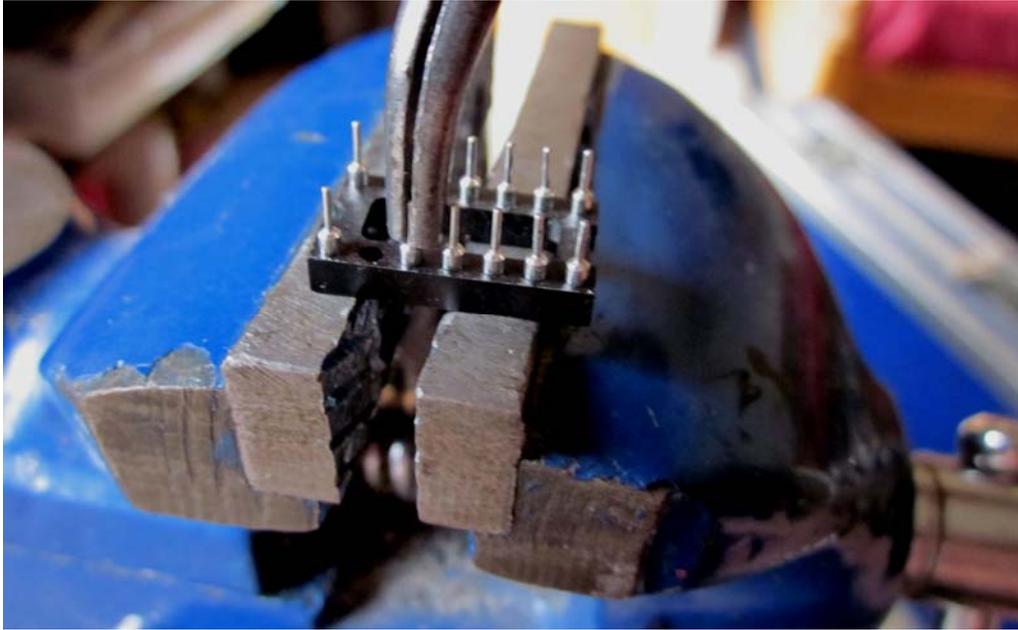
Left to right; +5V 4mHz SPXO, +5V 16mHz 8-pin DIL (untested), and +5V 4mHz 14-pin DIL.



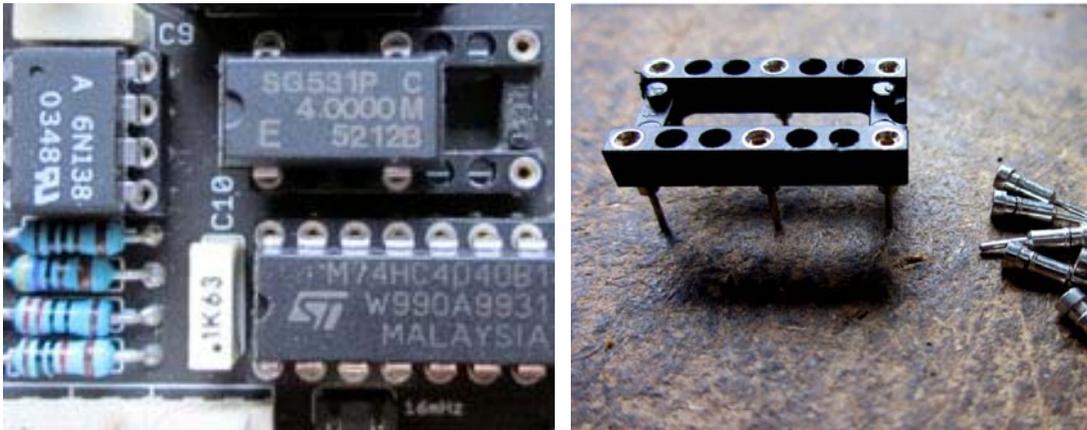
The footprint for the socket for X1 (if using one) is a little unusual, in that both the 8-pin and 14-pin types only use 4 pins, regardless of the size. The PCB is set up to take both sizes. The photos below show how to make a suitable "combo" socket with a 14 turned-pin socket, a bench vice and some fine needle-nosed pliers. Place the socket upside down over the vice, and push the pins down and out of the socket. The unused pins make useful testpoint pins, so are worth keeping.

MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

I made one of these to test various sizes of XTAL, but as noted, a socket is not recommended for the 8 and 14-pin metal can types, as they may work loose. They're best soldered in directly.



Correct orientation for an 8-pin XTAL OSC (SPXO shown) and a modified 8/14-pin socket for X1



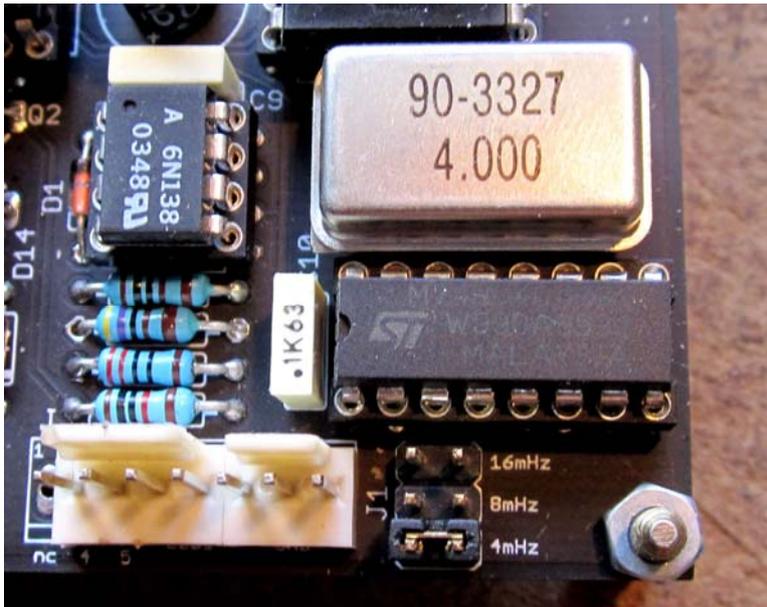
MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

JUMPERS

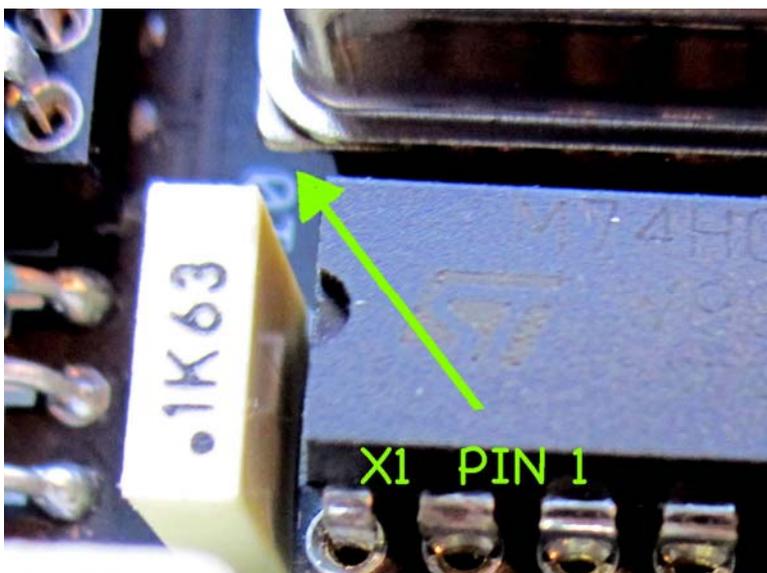
There are quite a few pin-headers and jumpers on the PCB, but not all are needed for the circuit to work correctly. J3, J4 and J5 are pin-headers that allow expansion of the PCB, and are not needed for correct operation of the circuit. A list of what is on what pin is at the end of the document.

J1 must be fitted. It allows you to choose what speed XTAL OSC you are using for X1. The type is a 0.100 (2.54mm) 3 x 2 pin header, of the type and spacing commonly found on computer motherboards and cards. If you like, a wire link can be used instead.

The photo shows the J1 jumper setting for a 4mHz XTAL OSC, and a 14-pin metal can XTAL OSC.

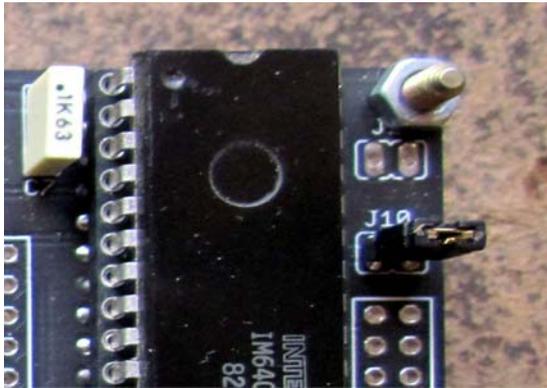


Identification of X1 pin 1 on metal can XTAL OSCs.



MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

J10 must also be fitted. It was added as it may possibly come in handy if the circuit is being heavily modded, but again, either a jumper or a wire link could just as easily be used here.



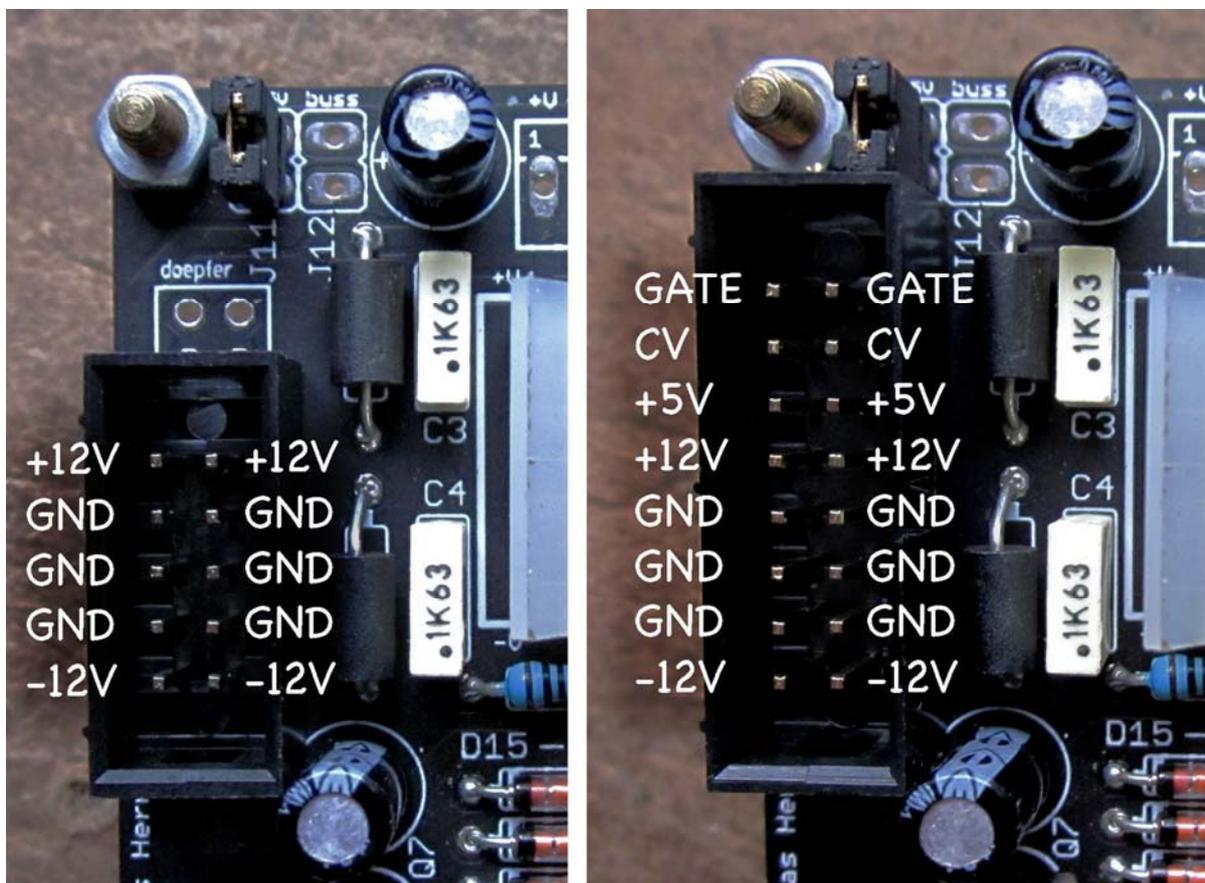
POWER HEADERS AND JUMPERS.

Three types of power input connectors can be installed on the PCB. Either a 0.156" (3.96mm) pitch 4-pin header (MOTM, CGS and OAKLEY compatible), a SYNTHESIZERS.COM 0.100" (2.54mm) pitch 6-pin header, or a 10-pin or 16-pin DOEPFER 0.100 (2.54mm) pitch IDC box header. The DOEPFER and SYNTHESIZERS.COM connectors can have +5V from the M2C's onboard +5V regulator connected to them, by linking J11 or J12. Only link either if +5V is *NOT* being supplied to these connectors from any other power connector in the system. If you don't need it, don't install it!

If installing the 10-way DOEPFER header, there are 4 unused holes visible above the connector.

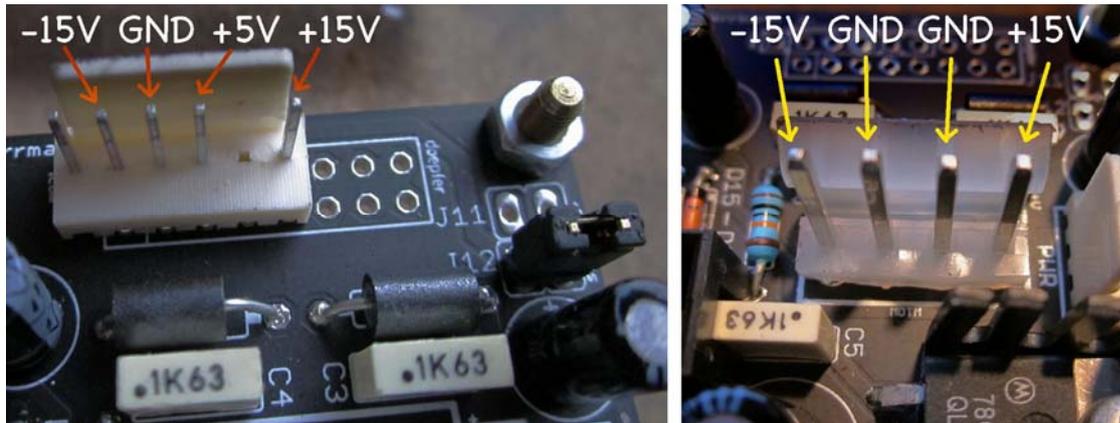
DON'T APPLY POWER TO MORE THAN ONE TYPE OF POWER INPUT SOCKET SIMULTANEOUSLY!

The photo below shows 10-pin and 16-pin DOEPFER connectors, with J11 linked in both cases.



MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

The photo below shows the correct orientation for a SYNTHESIZERS.COM power header with Pin 2 removed as specified. J12 is linked to give +5V. Below right, the MOTM/CGS/OAKLEY header.

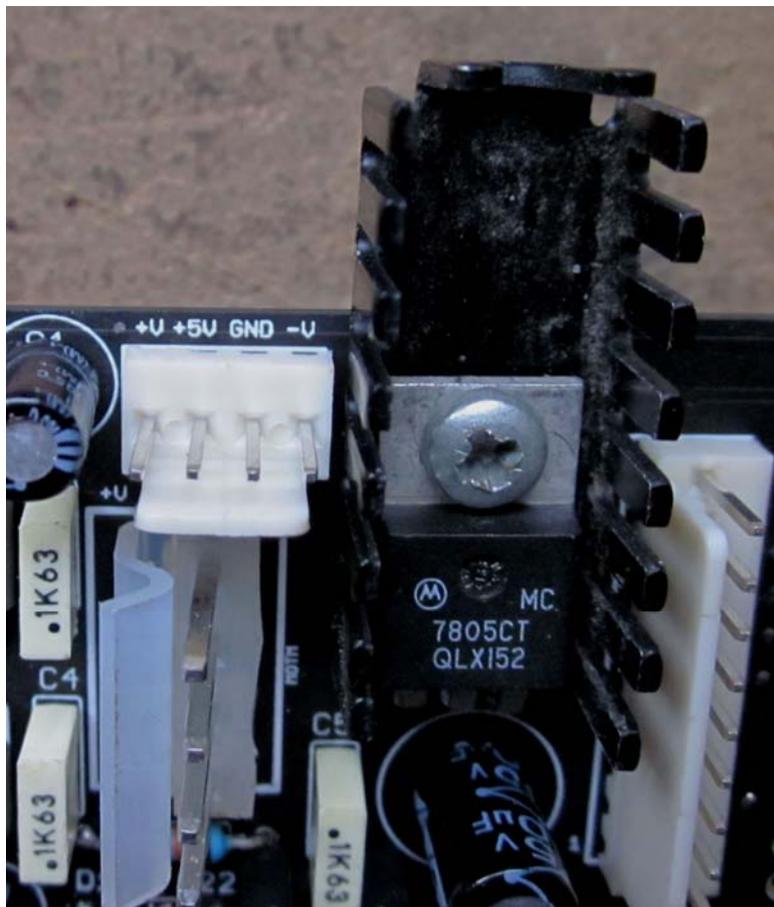


The power-out header (marked PWR on the PCB) is a 4-pin 0.100" (2.54mm pitch) type.

NOTE THAT IT IS NOT THE SAME PINOUT AS THE 0.156" MOTM etc. HEADER!

The +V, -V and *ONE* of the GND pins are the same, but it has +5V instead of the other GND connection. This header is designed to be used to power other possible future expansion boards, and is not a standard configuration. You do not need to fit it for the standard circuit to work correctly. If installing an optional heatsink, the metal body of the voltage regulator does not need to be insulated from the heatsink, as it is at 0V or GND. Use M3 fixings to secure it.

The photo below shows the PWR out header, and a heatsink installed on IC1 (the LM7805).



MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

CAPACITORS

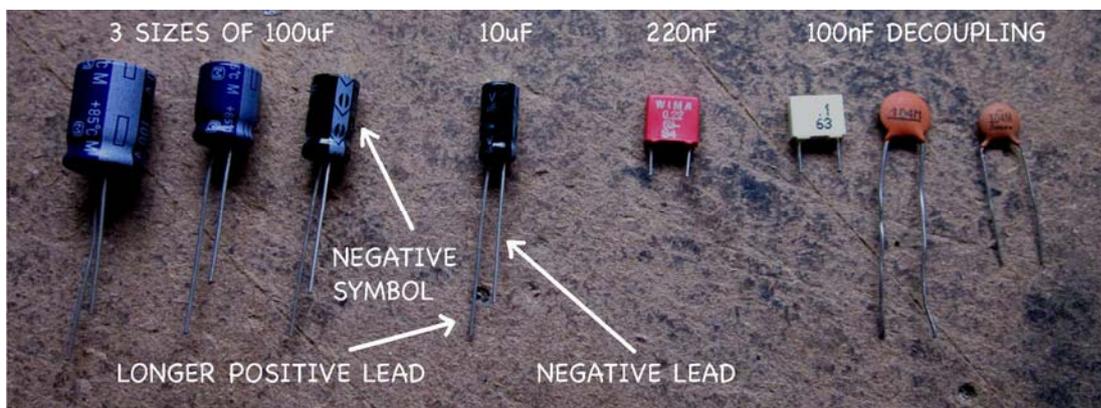
All capacitors should be 35V or higher rated.

The DECOUPLING capacitors (see BOM) on the PCB are 5mm spacing, and can be either ceramic, or "box" type capacitors, which were used in the PCB photos. The type is not critical - use what you have that fits. Note that if using "box" type capacitors, the decoupling capacitors for U1, U4, U5, U6, U7 and U8 are quite close to the IC. This is not a problem, and will not affect performance, but oddly, the type of IC used in the build for U5 (HC14) was slightly longer than the IC socket, and indeed the other ICs, so it crushed up slightly against C13. It would be worth installing the decoupling caps for these ICs *after* the ICs and/or sockets have been installed.

The electrolytic capacitors are all radial types with 2.54mm spacing except for C6 and C8, which are slightly wider spacing. Slightly different sizes should work, if the pins are slightly bent to fit.

When installing the electrolytic capacitors, check that the positive lead goes into the pad hole on the PCB marked +. The minus side of an electrolytic capacitor is usually marked with a lighter-coloured band with (-) signs on it. The other types of capacitor (10nF, 100nF and 220nF) are non-polarized, and can be fitted either way round on the PCB.

The photo below shows suitable types of capacitor for the M2C.



MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

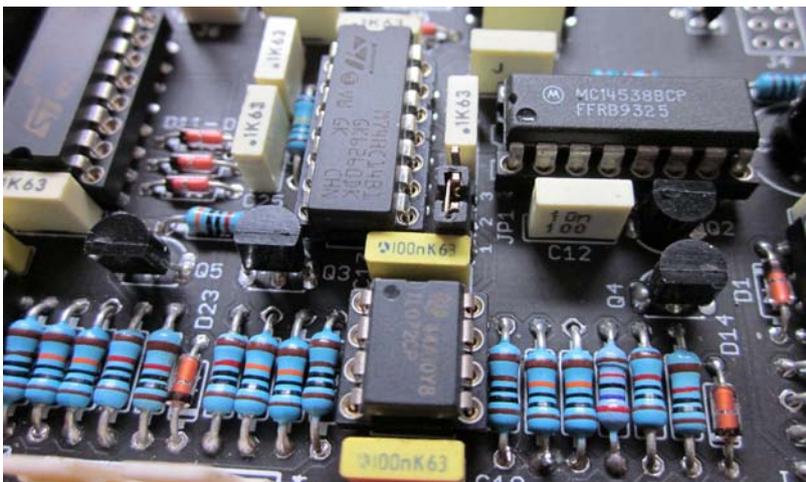
TRANSISTORS

2N3904 NPN transistors are specified, but any general-purpose NPN transistor with the same pin-out could be used. BC547, BC560 etc. would work if they were soldered **IN REVERSE** to the outline on the PCB. It is a good idea to check the transistors with the tester on a digital multi-meter before installing, as occasionally batches of transistors with different pin-outs turn up.....

Correct orientation of Q1 and D8 - D10, Q8 (left) and Q6 (right) and Q7 and D15 - D22



Correct orientation of of Q2 - Q5 and U4, U5, U8 and U9, and D1, D11 - D14 and D23, and JP1 (if using long pulse for STOP)



LEDS

The LEDs can either be regular types, in which case 1K - 2K7 resistors are used for the RLEDs, or the newer superbright and ultrabright LEDs, which would need the RLED value to be between 10K and 30K or so. These newer types are a better bet, as they consume far less current, as well as having a wider range of colours. 10K resistors were used in one of the test builds.

If omitting the LEDs from any output for any reason then omit the transistor and it's associated RLED resistor. If leaving the LED off the START and STOP outputs, also omit D14/D23, C26/C27 and R37 and R38. Whatever you include or leave out, it would be well worth installing Q2, R6 and LED1, as this LED indicator circuit gives a useful indication of MIDI received.

JP1 must be installed for the STOP pulse to work. The normal mode of setting is to connect pins 1 and 2 together. This gives a longer pulse which is needed to fix a bug with some sequencers, where occasionally a rogue clock pulse is sent out **after** the STOP pulse. This can lead to erratic behaviour in the divider sections of the M2C. Pin 3 of JP1 carries the standard short pulse, called "STOP RAW" in case it's needed for whatever reason. It's probably best simply to link pins 1 and 2 of JP1. A permanent wire link could be used instead of a jumper.

MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

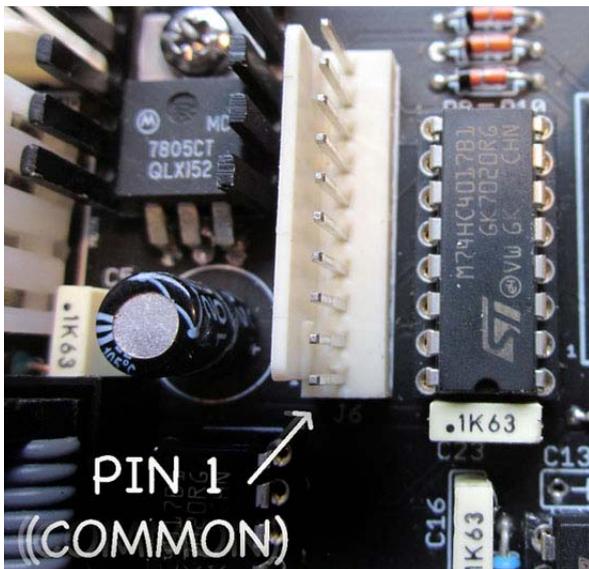
VOLTAGE REGULATOR.

There is room on the PCB for a small heatsink (size - about 31mm x 12mm x 12mm). This is only necessary if you plan to expand the board, using the various pin headers for that purpose.

SWITCH CONNECTIONS AND OUTPUTS.

The M2C has been designed to use simple controls, like those on a modular synth. J6 is a pin-header that uses a 0.100 (2.54mm) 10-way connector to link to a 1-pole 12-way rotary switch, of which 8 (or 9) positions are used. This switch divides MIDI's 24 pulses per quarter note (PPQN) into a more useable number. The available divisions are divide by 2, 3, 4, 5, 6, 7, 8, and 9. External selection of initial division would require a 9th rotary switch position and extra circuitry, but isn't necessary for basic use, as the 8 positions give plenty of interesting and useful patterns.

J6 (LEFT). PIN 1 is connected to the switch's common, PIN 2 to the FIRST switch position etc.



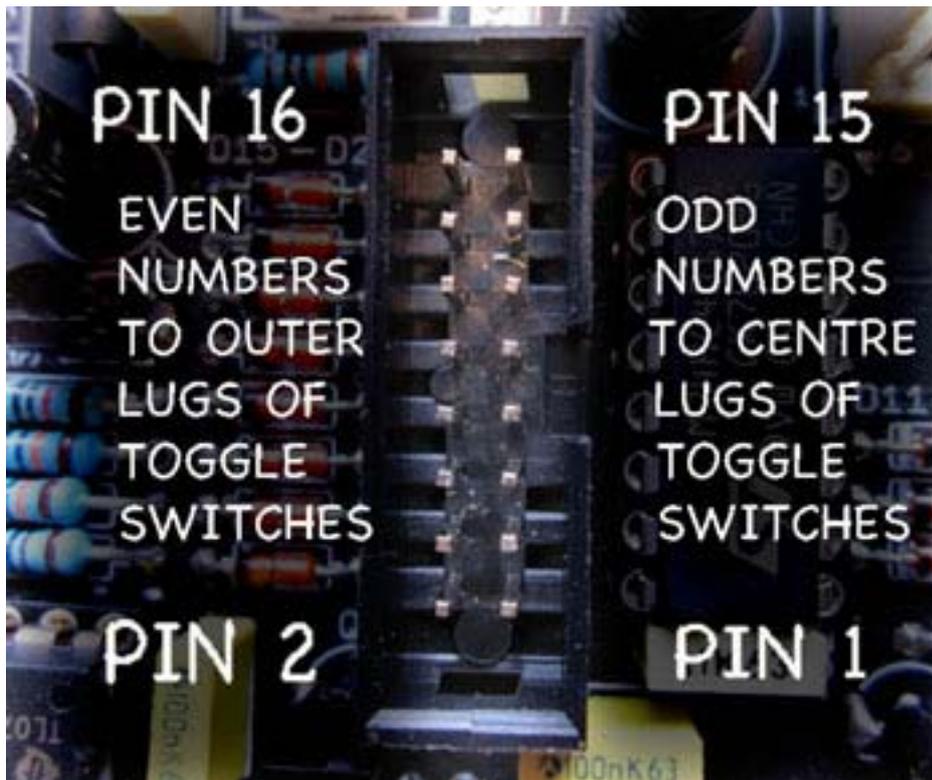
The common is connected to pin 1 of J6. Pin 10 of J6 is not used or connected to the rotary. J6 Pin 2 is connected to the first position of the rotary - divide-by-2, J6 pin 3 is connected to the second position of the rotary, divide-by-3 and so on up to pin 9, divide-by-9 at the last position.

The photo below shows the completed connector and rotary switch, with the wires heatshrunk onto the terminals of the rotary to give them some additional mechanical strength.



MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

J7 is a bit more fiddly..... The connector is a 16-way IDC 0.100 (2.54mm) spacing box header.



I strongly recommend fitting this header, rather than direct-wiring, as it allows for easy modding later on - including such things as GATE-controlled access to the toggle switches' action, which could be achieved with a separate expansion board.

The easiest way to connect the toggles to the header is by soldering them to one end of a flat 16-way ribbon cable, in which the wires have been separated out at one end for a couple of inches. Do this with a stanley knife or an Xacto type craft knife.

On the other end attach a 16-way cable mounting socket. This comes in 3 parts - the socket which has 2 rows of sharp metal jaws, a bracket that closes down over the jaws, and a second strain relief bracket. The cable can be assembled in a regular bench vice.

The ribbon cable has a red stripe along one edge to help identify direction. In the photo on next page, it identifies PIN 1. Unlike the numbering for a DIL socket the pins are labelled in pairs - 1 and 2 on the same row, 3 and 4 the same row, and so on. Check the photo above for details.

If using Dual-throw toggle switches, (whether dual pole or single pole) only solder pins 2,4,6,8,10, 12, 14 and 16 to ONE of the outer lugs of the toggle switch **not both !**

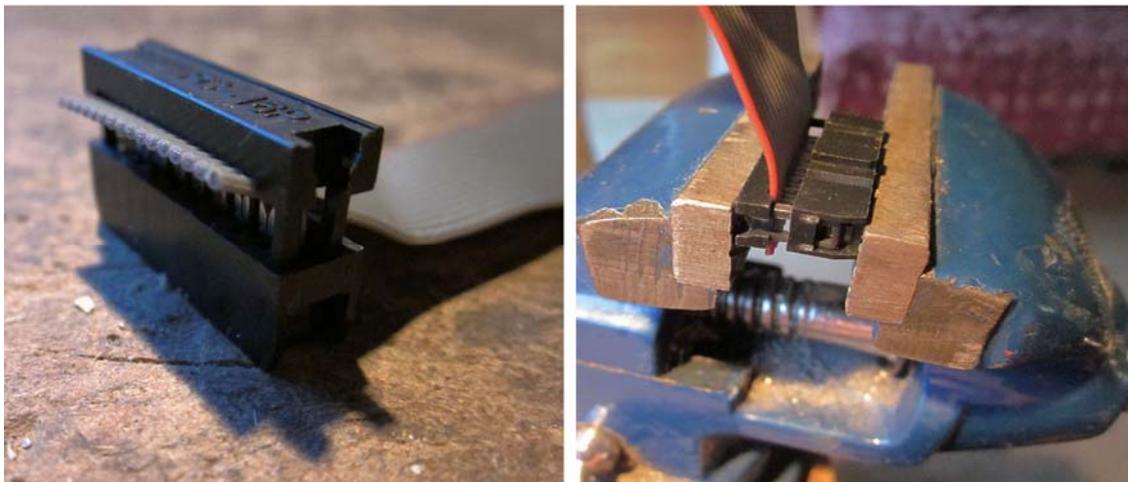
With single-throw switches you only have one outer lug, so no problem.....

MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

The photos show pin numbers below left and cable-mounting socket components below right.



First, take the cable and place it between the jaws of the main body of the socket and the first bracket, with a little spare at one end. Hold it together, then carefully place in the jaws of the bench vice, making sure the cable is perpendicular to the long face of the socket.

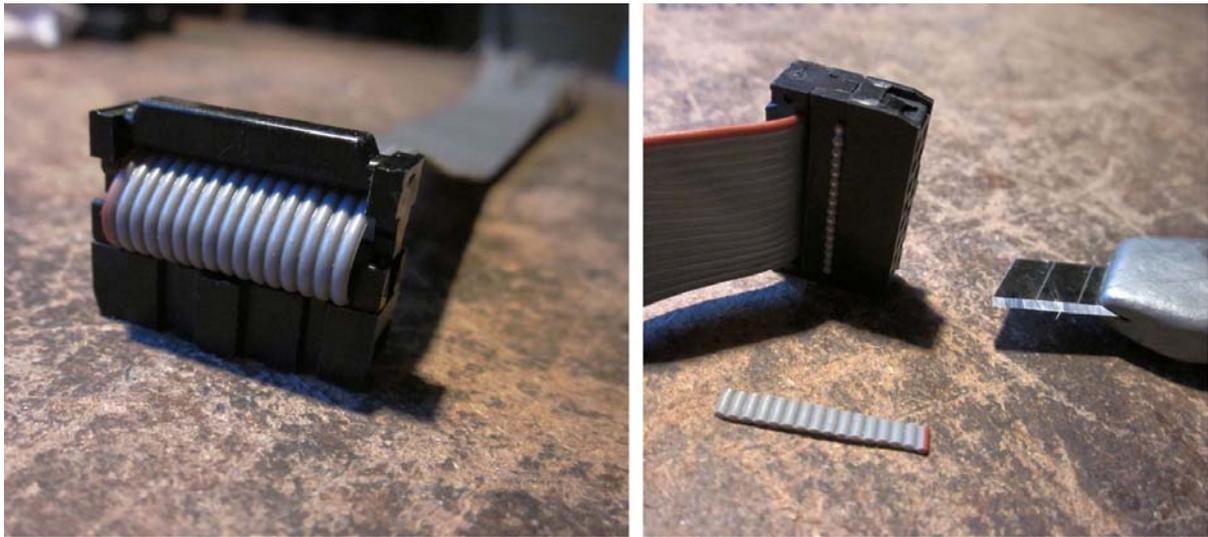


Carefully close the vice until the jaws of the socket are no longer visible. Then take the cable, and bend it back tightly over the the top of the first bracket. Then take the second bracket and loosely fit it into the top of the bracket. Again, gently tighten it together in the vice. This second bracket gives strain relief for the cable.

Finally carefully cut off the small amount of protruding spare cable from the socket. See next page for photos.

MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

Photos showing the final stages of the ribbon cable construction.



Finally, solder the free ends of the ribbon cable to the lugs of the toggle switches. Pins 1,3,5,7,9,11,13 and 15 connect to the middle (common) lugs of the toggle switches, and pins 2,4,6,8,10,12,14 and 16 connect to the outer lugs. Use SPST, SPDT, DPST or DPDT CHANGEOVER toggles. (CENTRE-OFF types will work too, but the middle **and** one of the outer positions will both be OFF). The photo shows DPDT switches. DEFINITELY use heatshrink sleeving on these lugs, as the cable strands in the ribbon cable are very thin, and easily broken. The heatshrink will give extra mechanical strength to the joint.

The cables connecting to pins 1 and 2 are paired, and are connected to the same toggle switch - TOGGLE 1 in the second stage divider. Pin 1 goes to the middle lug of the toggle, and pin 2 to the outer lug (or either of the outer lugs with dual-throw type sockets). If using dual-pole toggles, leave the other outer and centre pins unconnected. The same scheme applies to pins 3 and 4, and so on up to pins 15 and 16. Check page 12 for details.

The finished cable.



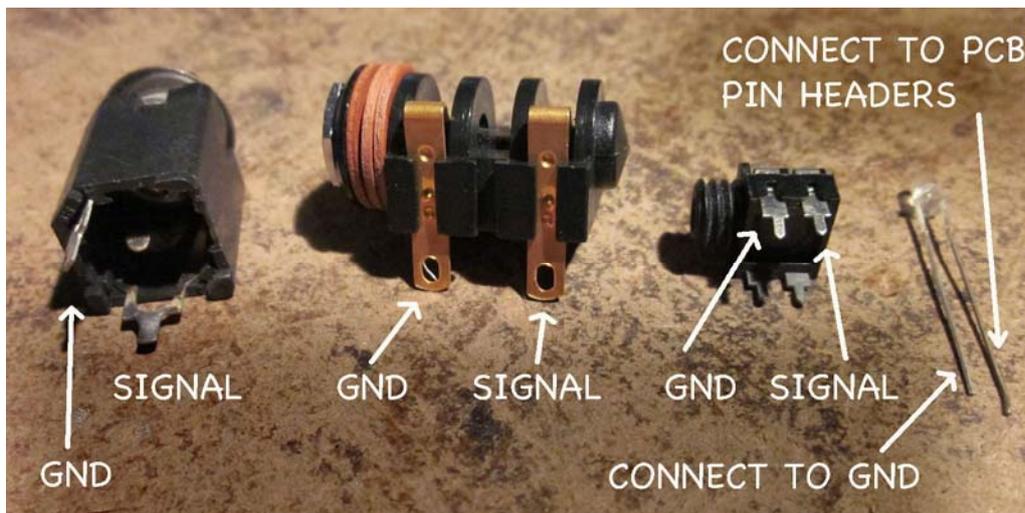
MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

OUTPUT SOCKETS

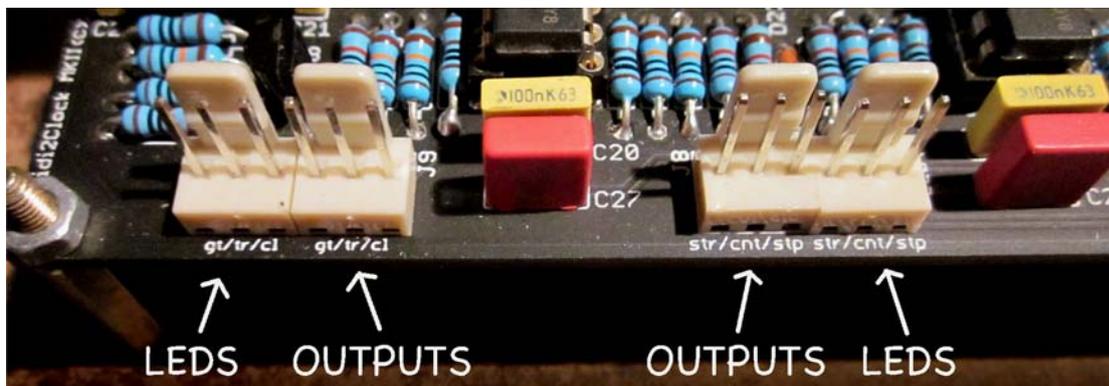
The final stage is to fit the gate and pulse output sockets and LEDs, and the MIDI INPUT socket.

There are 6 pulse or GATE outputs. If using pin-headers, make up the correct cable where the pins connect to the relevant signal lugs of 3.5mm or 1/4" jack sockets or banana sockets. GND connections are available from W1, W2 or W3. If using jack sockets, connect the GND pins of the jack sockets together. All these connections can be hard-wired if desired.

Gate and pulse output sockets and LED orientation.



The photo below shows the 2 groups of outputs, GATE, TRIGGER and INITIAL DIVISION (called gt, tr and ck on the PCB) and START STOP and CONTINUE (str, cnt and stp) I used 3-way end-stackable pin headers. A 6-way and an 8-way header, or direct wiring is also OK.

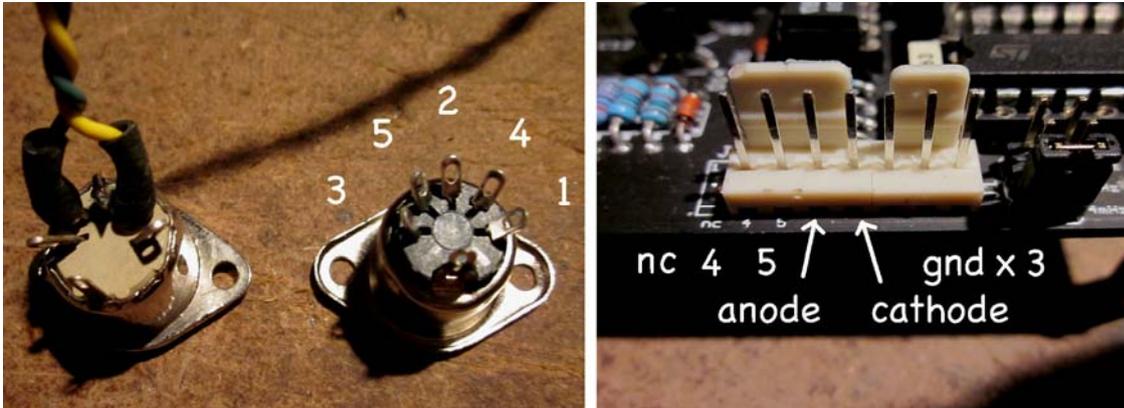


Connect the 6 LED outputs on the PCB to the ANODE or positive lead of the LEDs. This lead is usually the longer one. Connect the shorter leads to GND, perhaps from W1, W2 or W3.

MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

MIDI SOCKET

The pinout of a MIDI socket (5-pin DIN 180 degree) is a bit unusual, in that the pin numbers do not run sequentially. Only pins 4 and 5 need to be connected. The LED connection on the PCB has connections for both anode and cathode, so no separate GND connection is needed.



MIDI 2 CLOCK MKII V2 DESCRIPTION AND BUILD NOTES

USING THE M2C

The M2C is a relatively simple basic circuit, and easy to use. It handles MIDI CLOCK messages, and will start and stop in sync with a MIDI sequencer if some simple settings are made in the sequencer. The circuit may seem complicated, but actually it's very straightforward to use, once you get your head around it. Connect up the GATE output to a module that requires clock pulses. Set the initial divider to one of its divisions - divide by 6 and divide by 3 are probably the most useful, as they give maximums of 1/16 and 1/32 clock pulses respectively, in 4/4 time. Then simply experiment with switching in the toggle switches to get different divisions and patterns.

Er, ... that's all there is to it..... :-)

TESTING IT

The easiest way to check the M2C is doing what it should, is to connect up the MIDI input to a source of MIDI clocks - a computer or hardware sequencer for example. Check that the sequencer is set to transmit MIDI clocks, and also that it is set to transmit START, STOP and CONTINUE commands. These are important for keeping the M2C in sync, even though it does not recognise Song Position pointers (SPPs). Finally, make sure that the sequencer is set to start playing **always** from the beginning of a bar. If you stop in the middle of a bar, and then resume playing, the M2C will still output pulses in time with the sequencer, but the pattern you set with the dividers may not be what you expect!

Start the sequencer, and the MIDI RECEIVE indicator LED should light up rapidly. Set the initial divider to Divide by 6. This will give you a pattern of 16 beats per bar from the initial divider output socket. This output **always** provides pulses from the Initial divider stage when the MIDI sequencer is running. Plug the INITIAL DIVIDER output into a module that needs a gates to run it, or simply into a VCO or VCF set to self-resonance. With the latter 2 options you should hear the pitch jumping from high to low 16 times per bar.

Now set all toggles in the secondary divider stage to "off" and connect the GATE output to your ADSR or VCO or whatever. Switch in the INITIAL divider toggle, and the results will be the same as before. Now switch it off, and switch on toggles 1, 3, 5 and 7. You should now hear only 8 beats to the bar. switching in only toggle 1 gives you 2 beats to the bar. These toggles can be switched in live while the sequence is running without affecting the timing or sync of the unit.

Switching the INITIAL divider to either divide by 4 or 8 will give you a 3/4 pattern. Divide by 5 and 7 are simply weird. Switching the initial divider while the sequence is running will usually make the sequence glitch - it won't damage the M2C, and it will still produce clocks at the correct tempo, but they will not be in sync with beats and bars. Stopping and re-starting the sequencer will get things back on track again.

To summarise,

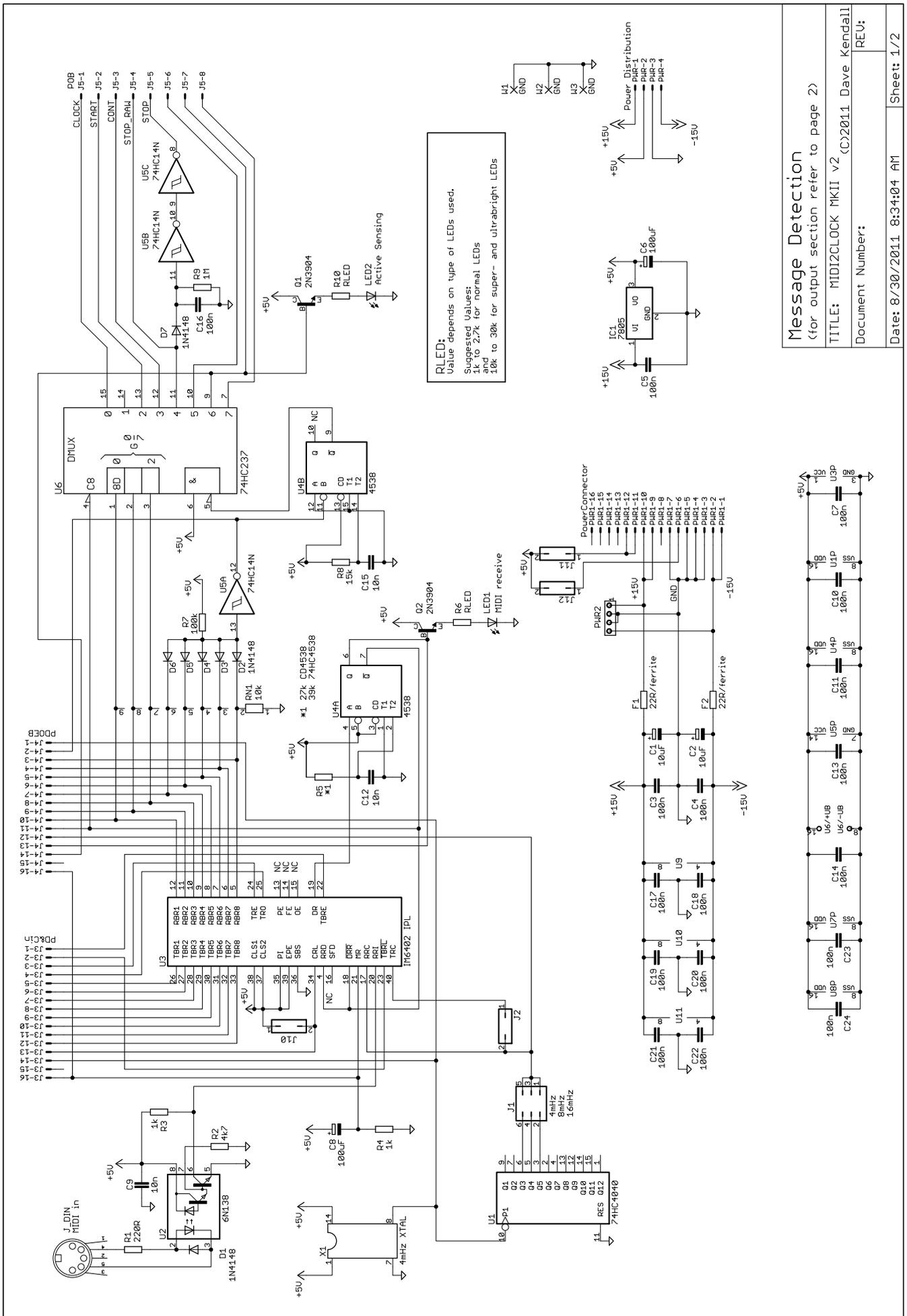
- Set your sequencer to always resume playing from the start of a bar.
- Set your sequencer to always output START and STOP commands (most do by default)
- Set the Initial divider up, then don't adjust it while the sequence is running.
- The toggles can be freely switched on or off while the sequencer is running. Experiment!

The TRIGGER output will output a short pulse with the start of the the first toggle that's switched ON, but won't produce 2 pulses in 2 consecutive stages if two toggles next to each other are ON. It will only produce another pulse after going through a stage that's been switched OFF.

The START output produces a pulse when the sequencer is first started. If it is stopped and then re-started, it will produce another pulse. If the sequence is PAUSED, then when it's re-started, the M2C will output a short pulse from the CONTINUE output. Some sequencers have different behaviour regarding START and CONTINUE.

Whenever the sequencer is stopped, the M2C will produce a pulse from the STOP output. The normal setting (pins 2 and 3 bridged) is for quite a long pulse to be produced. Set JP1 to bridge pins 1 and 2 to change this to the shorter pulse if you **really** need to.....

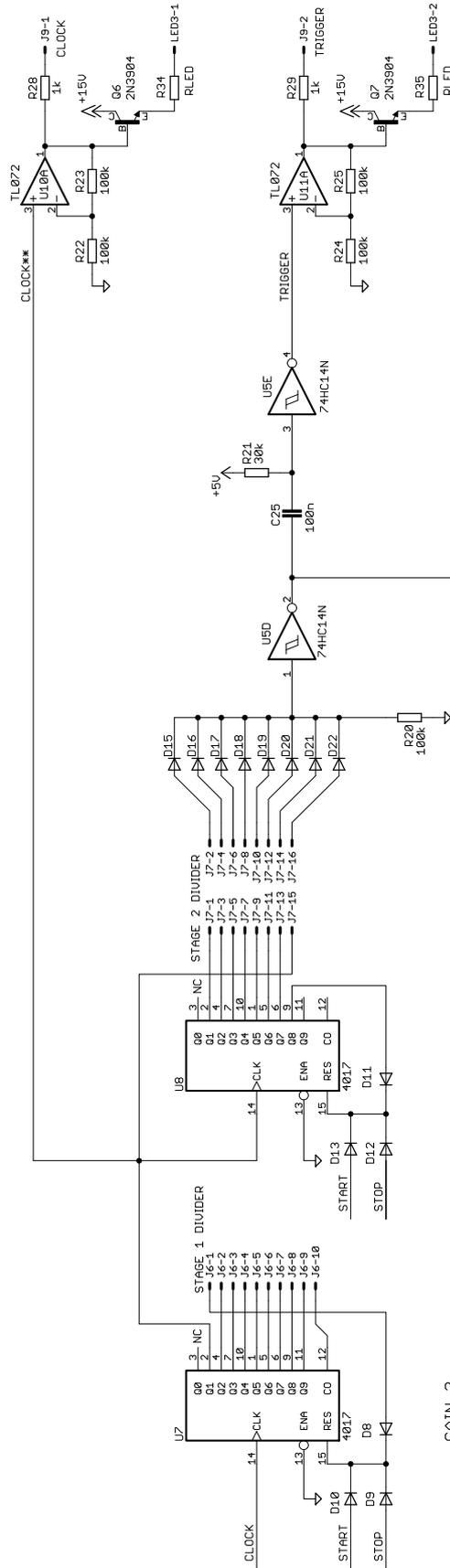
Have fun.....!



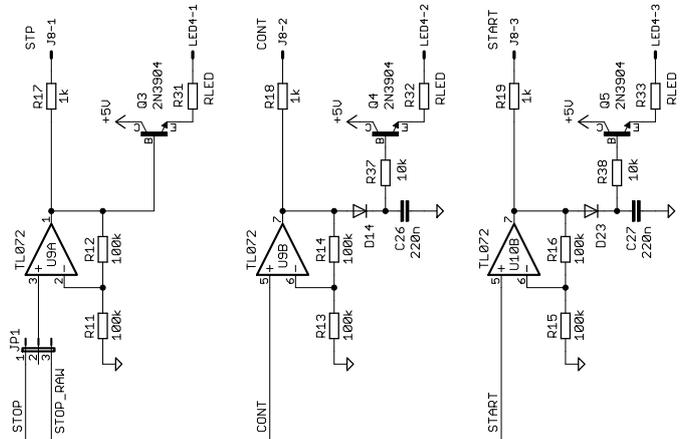
Message Detection
(for output section refer to page 2)

TITLE: MIDI2CLOCK MKII v2 (C)2011 Dave Kendall
Document Number:
Date: 8/30/2011 8:34:04 AM
Sheet: 1/2

GAIN 2



GAIN 2



ILED:
Value depends on type of LEDs used.
Suggested Values:
1k to 2.7k for normal LEDs
10k to 30k for super- and ultrabright LEDs

+5V vs +10V gates/triggers
For +10V gates/triggers
Install 100k resistors for
R21, R22, R23, R24, R25, R26, R27.
For +5V gates/triggers
Fit wire links for R23, R25, R27, and
do not install and/or fit in the positions for
R11, R13, R15, R22, R24, R26.

Output Section

(for message detection refer to page 1)

TITLE: MIDI2CLOCK MKII v2 (C)2011 Dave Kendall
Document Number:
REV:
Date: 8/30/2011 8:34:04 AM
Sheet: 2/2

APPENDIX

BILL OF MATERIALS

- R1 = 220 ohm
- R2 = 4K7
- ~~R2~~, R3, R4, R17, R18, R19, R28, ~~R29~~, R30 = 1K
- R5 = 27K or if using a 74HC4538 for U4, R5 = 39K
- R8 = 15K
- R9 = 1M
- R7, R11, R12, R13, R14, R15, R16, R20, R22, R23, R24, R25, R26, R27 = 100K
(NOTE = R11, R13, R15, R22, R24, R26 NOT NEEDED FOR +5V GATE OUTPUTS)
- R21 = 30K
- R6, R10, R31, R32, R33, R34, R35, R36 are marked RLED: If you are using regular LEDs, use a value between 1K and 2K7, for super and ultrabrights, use a value between 10K and around 30K. 10K resistors were used with superbright LEDs in the beta test builds. The lower the resistance, the brighter the LED.
- R37, R38 = 10K
- RN1 = 9-pin 10K commoned resistor network 2%

Metal film resistors between 0.25W and 0.6W are recommended. R5 and R8 **must** be metal film

- D1 to D23 = 1N4148 or 1N914 small signal diode
- F1, F2 = axial ferrite bead. 22ohm resistors could be used instead.

ALL CAPACITORS MUST BE RATED AT 35V OR MORE..

- C1, C2 = 10uF/35V 2.54mm pitch radial electrolytic capacitor.
- DECOUPLING CAPACITORS:
C3, C4, C5, C7, C10, C11, C13, C14, C16, C17, C18, C19, C20, C21, C22, C23, C24, C25 = 100nF/35V ceramic or polyester box type 5mm pitch capacitor
- C6, C8 = 100uF/35V radial electrolytic capacitor 3.5mm pitch (other sizes should fit too)
- C9, C12, C15 = 10nF/35V ceramic or polyester box type 5mm pitch capacitor
- C26, C27 = 220 nf/35V ceramic or polyester box type 5mm pitch capacitor

- IC1 = LM7805 +5V positive voltage regulator 1.5A TO220 package (Optional heatsink for above = 31 x 12 x 12mm)

- Q1 - Q2 = 2N3904 NPN transistor TO92 package. Any general-purpose small signal transistor would work, as it is just used to drive the LED. BC547, or 560 would work if the transistor **IS REVERSED** with respect to the PCB artwork. It is a good idea to check pinouts with the tester in a DMM before installing, as sometimes they can vary from different sources.....
- LED 1 - 6 = either regular LEDs, or super-bright/ultrabright. - see resistor values above

- U1 = 74HC4040 binary ripple counter
- U2 = 6N138 optoisolator. 6N137 and 6N139 could probably be used, but have not been tested. R1 and R2 may have to be adjusted if not using 6N138
- U3 = IM6402IPL UART (Harris) or HD-3 6402R-9Z (Intersil)
- U4 = CD4538, HEF4538 or MC14538 dual precision monostable.
- U5 = 74HC14 Hex schmitt trigger inverter.
- U6 = 74HC237 1 of 8 non-inverting decoder/multiplexer
- U7, U8 = 74HC4017 counter/divider. Regular CD4017 etc could be used and has been successfully, but the 74HC variant is recommended if you have a choice.
- U9, U10, U11 = TL072 dual opamp.

CRYSTAL OSCILLATORS MUST BE +5V RATED - NOT +3.3V !

- X1 = 8-pin 5V 4mHz Crystal oscillator module (SPXO package) recommended, or 8-pin or 14-pin 4mHz Crystal oscillator module (metal can package)

If using metal can XTAL OSCS be aware that the fit in a socket is not that tight. You're probably better off soldering these directly. 8mHz and 16mHz Crystal oscillator modules can be used, but these have not been tested as yet.

APPENDIX

- J2, J10, J11, J12, JP1 = 2-pin 2.54mm pitch header and jumper (like commonly found on computer motherboards and cards)

Power connector = one of;

- 0.156" (3.96mm) 4-pin connector (MOTM, OAKLEY, CGS etc)
 - 6-pin single-row 0.100" (2.54mm) connector with pin 2 REMOVED (for SYNTHESIZERS.COM)
 - 10-way 2.54mm IDC box header (DOEPFER)
 - 16-way 2.54mm IDC box header (DOEPFER)
- This option gives access to the DOEPFER buss

-
- J6 = 10-pin 0.100 (2.54mm) stackable single row header
 - J7 = 16-way 2.54mm IDC box header

These 2 connectors are recommended, as they make adding additional mods to the circuit easy.

- GT/TR/CK = 3-pin 0.100 (2.54mm) stackable single row header
- LEDs for above = 3-pin 0.100 (2.54mm) stackable single row header
- STR/CNT/STP = 3-pin 0.100 (2.54mm) stackable single row header
- LEDs for above 3-pin 0.100 (2.54mm) stackable single row header

Alternatively, a single 6-pin header could be used. If using 3-pin headers, make sure they are the stackable type, so they sit side by side on the board,

- MIDI in = 3-pin 0.100 (2.54mm) stackable single row header
- MIDI RECEIVE LED = 2-pin 0.100 (2.54mm) stackable single row header
- W1, W2, W3, = 3-pin 0.100 (2.54mm) stackable single row header

Similarly, an 8-pin header could be used here for MIDI in, the LED and W1, W2, W3 and a 2-way for the MIDI RECEIVE LED

- ACTIVE SENSING LED = 2-pin 0.100 (2.54mm) single row header

OPTIONAL SOCKETS FOR EXPANSION

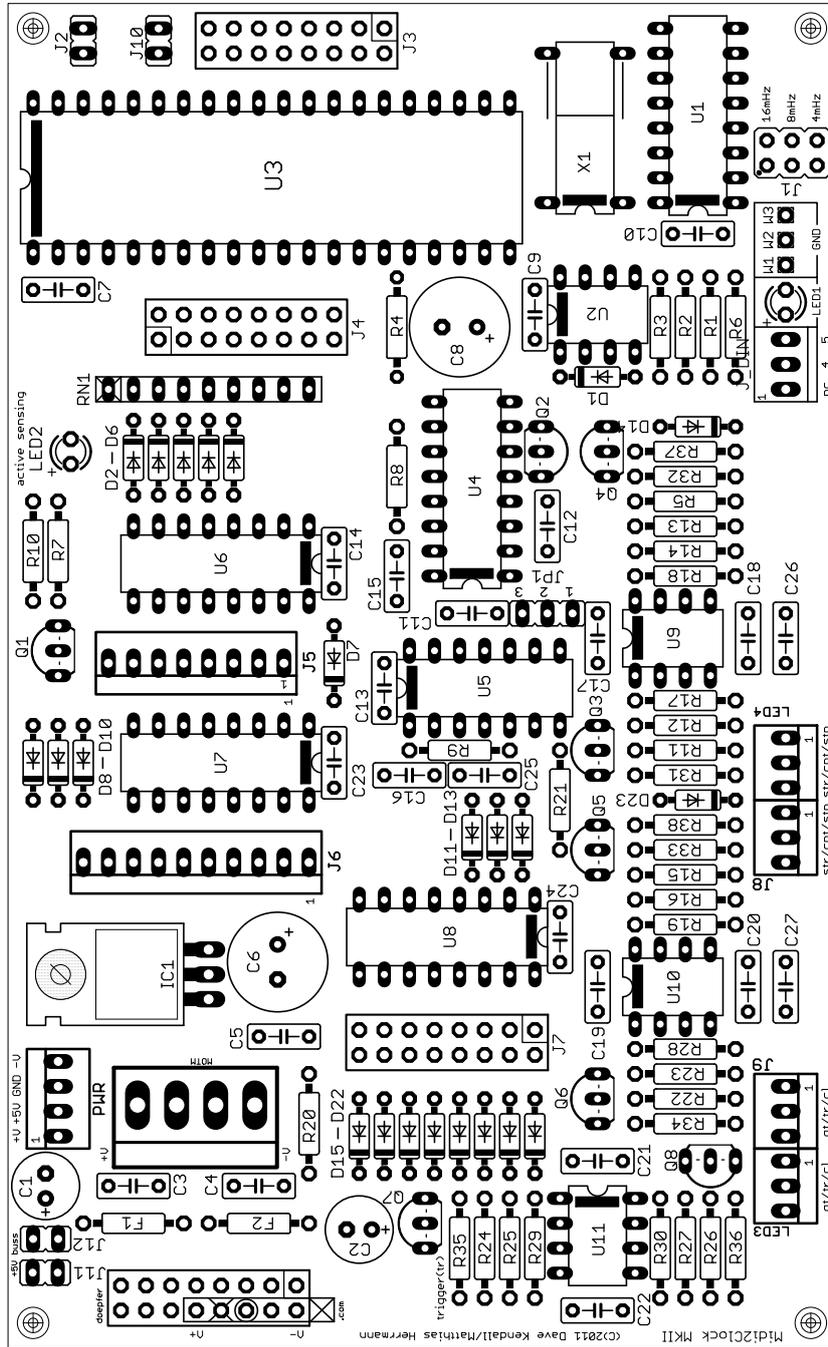
- J5 = 8-pin 0.100 (2.54mm) stackable single row header
- J3, J4 = 16-way 2.54mm IDC box header
- PWR OUT = 4-pin 0.100 (2.54mm) stackable single row header

If fitting the above you will need the corresponding cables, or make your own with the same type of cable mounting sockets and the relevant crimp pins.

Of course, direct wiring could be used instead.

-
- MIDI input socket = 1 x 5-pin DIN 180-degree chassis socket
 - 8-pin turned-pin socket (U2)
 - 16-pin turned-pin socket (U4)
 - 40-pin turned socket (U3)
 - The above are pretty much essential. If using sockets throughout, add
 - 3 x 8-pin turned-pin socket
 - 3 x 14-pin turned-pin socket
 - 3 x 16-pin turned-pin socket
 - 6 x output sockets eg. 3.5mm jack socket, 4mm banana socket = see text for details.
 - 12-way 1-pole rotary switch (only 8 or 9 positions are used)
 - 8 x SPST, SPDT, DPST or DPDT toggle switches.

HEATSHRINK SLEEVING is recommended for the connections to the toggles if using ribbon cable as described in the text, as the cable is very thin and easily broken. An outer diameter of between 1mm and 2mm will work for fine hook-up wire.



APPENDIX

PCB INFO

The PCB is a solder-masked, plated-through-hole, double-sided ROHS-compliant board. M3 fixings can be used to mount the PCB.

Any modifications to the board are at the users own risk, and any damage to or failure of the board will be the user's sole responsibility if attempting any modifications. The following information is given for those who are familiar with this type of circuit, and are experienced in performing modifications.

LIST OF PINOUTS FOR HEADERS

IMPORTANT NOTE

All of these connections are not buffered, and are mostly raw CMOS +5V logic signals. They cannot be connected directly to other modules, but are designed to interface directly with other CMOS devices. To use these in a modular synthesizer, appropriate output buffers and other circuitry must be used to prevent damage to the circuit. Do not use these unless you know what you are doing!

J1 - XTAL SPEED SELECT

J1 -1 16mHz,
J1 -2 8mHz,
J1 -3 4mHz,
J1 - 4, J1 -5, J1 -6 linked. Install the correct jumper for the speed of XTAL OSC you are using.

J2 TRANSMIT CLOCK LINK

J2 Links 500kHz CLOCK to U3 pin 40
(not used in basic design)

J3 UART MIDI TRANSMIT

(not used in basic design)
J3 - 1 to J3 -13 - see MIDI2CLOCK schematic and UART schematics for details of pin connections.
J3 - 14 RAW XTAL OSCILLATOR CLOCK OUTPUT
J3 - 15 NOT USED
J3 - 16 POWER-ON RESET PULSE

J4 - UART MIDI RECEIVED_

(not used in basic design)
J4 - 1 RAW XTAL OSCILLATOR CLOCK OUTPUT
J4 - 2 SYSTEM REAL-TIME RECEIVED PULSE
J4 - 3 MIDI DATA MSB
J4 - 4 MIDI DATA
J4 - 5 MIDI DATA
J4 - 6 MIDI DATA
J4 - 7 MIDI DATA
J4 - 8 MIDI DATA
J4 - 9 MIDI DATA
J4 - 10 MIDI DATA LSB
J4 - 11 MIDI RECEIVED PULSE negative going
J4 - 12 500kHz CLOCK
J4 - 13 MIDI RECEIVED PULSE positive going
J4 - 14 ACTIVE SENSING PULSES
J4 - 15 NOT USED
J4 - 16 POWER-ON RESET PULSE

J5 DECODED MIDI PULSES

(not used in basic design)
J5 - 1 24PPQN CLOCK PULSES
J5 - 2 START PULSE
J5 - 3 CONTINUE PULSE
J5 - 4 STOP PULSE ("STOP RAW")
J5 - 5 STOP PULSE (LONGER PULSE)
J5 - 6 PIN 10 OF U6 - RESERVED/NOT USED IN THE MIDI SPECIFICATION
J5 - 7 ACTIVE SENSING PULSES
J5 - 8 RESET PULSE

Note 1: These signals ARE NOT the same as DIN-SYNC. Extra circuitry will be needed for that!

Note 2: JP1 selects between J5 - 4 and J5 - 5 to be output via buffer U9A.

J6 U7 INITIAL DIVISION

J6 - 1 RESET (COMMON)
J6 - 2 DIVIDE BY 2
J6 - 3 DIVIDE BY 3
J6 - 4 DIVIDE BY 4
J6 - 5 DIVIDE BY 5
J6 - 6 DIVIDE BY 6
J6 - 7 DIVIDE BY 7
J6 - 8 DIVIDE BY 8
J6 - 9 DIVIDE BY 9
J6 - 10 CARRY OUT

J7 - STAGE 2 DIVIDER.

J7 - 1 Q1 OUTPUT
J7 - 2 Q1 SELECT
J7 - 3 Q2 OUTPUT
J7 - 4 Q2 SELECT
J7 - 5 Q3 OUTPUT
J7 - 6 Q3 SELECT
J7 - 7 Q4 OUTPUT
J7 - 8 Q4 SELECT
J7 - 9 Q5 OUTPUT
J7 - 10 Q5 SELECT
J7 - 11 Q6 OUTPUT
J7 - 12 Q6 SELECT
J7 - 13 Q7 OUTPUT
J7 - 14 Q7 SELECT
J7 - 15 Q7 OUTPUT
J7 - 16 Q8 SELECT

J11 DOEPFER +5V ENABLE

Linking J11 allows +5V from the onboard +5V voltage regulator to provide power to pins 3 and 14 of the power header, if using a 16-way connector.

J12 SYNTHESIZERS.COM +5V ENABLE

Linking J12 allows +5V to be routed to pin 3 of a SYNTHESIZERS.COM 6-pin connector.

CAUTION - ONLY SUPPLY POWER TO ONE TYPE OF CONNECTOR! MAKE SURE THAT IF J11 AND/OR J12 ARE USED, +5V IS NOT ALSO BEING SUPPLIED BY A DOEPFER OR SYNTHESIZERS.COM POWER CABLE AT THE SAME TIME !

APPENDIX

I cannot promise to always answer queries and problem-solve build issues - I have a life and only do this as a hobby, this is not a business. Support can be found on forums such as electro-music.com and muffwiggler.com. There are many helpful and knowledgeable people there who can also help.

My thanks to Neil Johnson for advice and coffee, and encouraging me in the project's early stages, and many thanks to Matthias Herrmann for encouragement and ideas in the final design stages. Without you this project would never have happened as a proper PCB. Cheers Matthias! :-)

I hope you all enjoy the M2C !
Dave Kendall, 24th August 2011.

The Legal bit:

DISCLAIMER.

BUILDING THE MID2CLOCK REQUIRES SKILL AND PRIOR EXPERIENCE IN BUILDING SIMILAR ELECTRONIC CIRCUITS. KNOWLEDGE OF SAFETY PROCEDURES, AND SAFE WORKSHOP PRACTICE IS REQUIRED.

NO LIABILITY CAN BE HELD BY THE DESIGNER, ANYONE ASSOCIATED WITH THIS DESIGN OR THE AUTHORS OF THIS DOCUMENT FOR A NON-WORKING OR DAMAGED CIRCUIT, OR ANY DAMAGE ARISING FROM USE OF THE CIRCUIT, TO PROPERTY OR TO THE HEALTH OF ANY PERSON CONSTRUCTING OR USING THE MID2CLOCK. IF YOU ARE IN ANY DOUBT AS TO YOUR ABILITY TO BUILD THIS CIRCUIT SUCCESSFULLY, THEN DO NOT DO SO, OR EMPLOY A SKILLED ELECTRONIC TECHNICIAN TO BUILD IT FOR YOU.

ALTHOUGH MAINS VOLTAGE IS NOT USED IN THE CIRCUIT, A MAINS TRANSFORMER WILL BE NEEDED TO POWER THE CIRCUIT. THE RELEVANT DETAILS ARE IN THE MAIN BODY OF THE DOCUMENT, AND IT IS ASSUMED THAT THE BUILDER WILL UNDERSTAND HOW TO POWER THE MID2CLOCK SAFELY AND SUCCESSFULLY.

BUILD IT AT YOUR OWN RISK!

THE TRADE NAMES DOEPFER, MOTM, CGS AND SYNTHESIZERS.COM ELECTRO-MUSIC.COM AND MUFFWIGGLER.COM ARE THE PROPERTY OF THEIR RESPECTIVE OWNERS, AND ARE USED IN THIS DOCUMENT ONLY AS A POINT OF REFERENCE. NO AFFILIATION WITH SAID PARTIES IS CLAIMED OR IMPLIED.

THE CIRCUIT DESIGN IS COPYRIGHT DAVE KENDALL 2003 - 2011, THE PCB LAYOUT, SCHEMATIC AND ALL PCB ARTWORK COPYRIGHT MATTHIAS HERMANN 2010 - 2011.

THIS DOCUMENT IS COPYRIGHT DAVE KENDALL AND MATTHIAS HERMANN 2011.

IT IS ENCOURAGED FOR THIS CIRCUIT TO BE BUILT AND ENJOYED FOR PERSONAL USE.

COMMERCIAL USE, INCLUDING BUILDING MULTIPLE CIRCUITS FOR RE-SALE AND DISTRIBUTION OF THE SCHEMATIC, PCB ARTWORK OR THIS DOCUMENT IS STRICTLY FORBIDDEN WITHOUT PRIOR WRITTEN CONSENT OF THE COPYRIGHT HOLDERS WHO ARE NAMELY DAVE KENDALL AND MATTHIAS HERMANN.

HITCHIN, ENGLAND, AND WIESBADEN, DEUTSCHLAND 24TH AUGUST 2011

APPENDIX

Documentation Change Log

v1 – 2 SEP 2011	Original Version	
v2 – 14 SEP 2011	Page 2 of 24, para 1	“Install wire links for R11...” now reads “Install wire links for R12...”
	Page 25 ADD	Documentation Change Log added
V3 – 20 SEP 2011	Page 20	R2 = 4.7k NOT 1k
		R29 = 1k