

A kit for a QTU

Enclosed is a kit of parts (or a partial kit if requested) for you to assemble a QTU board.

The exact list of components used varies with the scale you are using, and with where the board fits in your system, and the components you actually use may well vary again according to your own likely usage.

Examine your package and check that you can identify all the parts and tick the \checkmark column to confirm you have received the correct parts.

This is just page 1 of the QTU assembly instructions (file *QTU Assembly.pdf*). Download the whole document from the address at the bottom of the page.

In the same folder is a *QTU parts chart.pdf* print this and arrange your parts on it (see photo: *QTU parts chart.jpg*).

Also download *QTU silkscreen.pdf* - open this in acrobat so you can zoom in for a closer look at the component placement. There are also pictures and schematics as mentioned in the main text.

For partial kits the list of parts to get from Rapid are in *QTU Component list.txt*

Qty	Value	Markings	\checkmark
Resistors			
8	1R, 1%	Brown, Black, Black, Silver, Brown	
4	150R, 1%	Brown, Green, Black, Black, Brown	
5	220R	Red, Red, Black, Black, Brown	
5	820R, 1%	Grey, Red, Black, Black, Brown	
6	1K, 1%	Brown, Black, Black, Brown, Brown	
18	4K7, 1%	Yellow, Purple, Black, Brown, Brown	
17	33K	Orange, Orange, Black, red, Brown	
5	180K	Brown, Grey, Yellow, Gold	
5	8 * 100K	•104 - dot is pin 1	
4	5K6	Green, Blue, Red, Gold	
Capacitors			
4	22pF	22	
2	0.47uF	474	
12	0.1uF	104	
4	10uF, 63V	10uF 63V	
1	470uF	470uF 25V	
1	2200uF	2200uF 35V	
Diodes			
15	1N4148	41 48	
2	1N5401	1N5401 DC	
9	1N4001	1N4001	
1	BZX79C30V	BZX 33C 30	
1	BZX79C8V2	BZX 55C 8V2	
Transistors			
5	BC327	C32740	
14	BC337	C33725	
4	TIP120	TIP120	
1	7812	L7812CV	
1	7805	L7805CV	

Integrated Circuits			\checkmark
1	ATMega32	ATMEGA32 16PU	
3	CD4051	MFC4051B	
1	CD4555		
1	TLC5620		
4	CD4094		
4	ULN2803A		
4	CD4021		
2	LM324		
1	CD4069		
Miscellaneous			
1		3 pin power strip & connector	
1		10 way pin strip to cut into 3	
3		shorting links	
1		10 pin R/A plug & socket	
1		8 way straight pin header (needs filing)	
1		8 way housing	
9		10 way straight pin header	
9		10 way housing	
2		12 way straight pin header	
2		12 way housing	
100		Crimp terminals	
8	BT47	Relays	
3	LEDs	Red, Yellow and Green	
1	Resonator	16.00MX	
1		PCB and heatsink	
6		TO220 mica and plastic washers	
9		6mm M3 bolts	
6		4mm M3 spacers	
G scale extras			
4	0R22	Red, Red, Silver, Gold	
2	P600G	P 600 G	
1		8 way pin strip	
4		2 pin interlocking connector block	
RS485 extras			
2		RJ-45 connectors	
2	ST485	ST 485	
1		6 way DIP switch	
RS232 extras			
4	10uF, 25V	10uF 25V	
1	DB-9	RS232 socket	
1	ST232	ST 232	
2		RJ-45 connectors	
2	ST485	ST 485	
1		6 way DIP switch	

Optional parts

Some components have alternate values for different scales, and some are only fitted to some boards and it is important to determine these before you start, to prevent mistakes later. Look through these options and decide which you have components for, and which you want to implement. Then look through the assembly instructions for *OPTIONAL* and cross out or modify to suit your requirements.

Scale dependant components

Component	Function	N scale	00 scale	G scale
D1, D2	Main rectifier	1N5401	1N5401	P600G
R101, R201, R301, R401	Current measurement calibra	1R	1R	R22
R102, R202, R302, R402	Secondary component		1R	
D100, R200, D300, D400	Reverse feed protection	1N4001	1N4001	1N5401
RP6	Wiring leakage bias			4K7
Transformer	Power feed	15-0-15V	15-0-15V	18-0-18V

The track output connector can take many guises. For smaller scales the kit includes an 8 way molex connector. For G scale an 8-way pluggable connector block (like the AC-in connector) is supplied. Also PCB mount connector blocks and even press-to-release connector blocks are compatible with the PCB (make sure the connector uses a 5mm pitch).

Slow motion turnout drive

The slow motion turnout motor drive components are optional.

For Conrad/Hoffman motors a full 500mA output drive is required, so the output current limit resistors are a low value - 8K2.

Tortoise motors take much less current so a much larger value can be substituted and this protects the transistors in the even of a short.

The diodes are not required if a bipolar (forward-reverse) output is not required, and the space can be used for series resistors to drive LEDs.

For LEDs I suggest they are fitted with resistors in the range 1K to 2K2 and fed from the V+ output. This minimises heat dissipation in the regulators and so reduces heatsink heating.

Component	Function	Tortoise	Conrad	LED	Other
Q2, Q3, Q4, Q5	Bipolar output drive	Fit	Fit	omit	omit
R35, R36, R37, R38	Output current limit	33K	8K2	omit	omit
D11, D12, D13, D14	Reverse isolation	Fit	Fit	1K5 resistor	wire link
Q7	Voltage regulator	Fit	wire link C-E	omit	omit

Relay voltage

The relays can be either 12V or 24V (dependant upon where and when ordered). Link J1 should be fitted with a wire link to suit the relays supplied.

Computer interface

The computer interface is also customisable. The choices are RS485, RS232 and RPC stack. Boards fitted into RPC stacks below the stack header board should not have the RS232 or RS485 components fitted as they interfere with the connectors of the adjacent board. Stack header boards must have the RS485 components, and may occasionally have RS232 fitted as well (only one such board is required per layout).

Component	Function	RS232	RS485	Stack
U9: ST232	RS232 buffers	Fit	optional	omit
C27-C30	RS232 voltage generators	Fit	optional	omit
SK7: DB-9	RS232 connector	Fit	optional	omit
U7,8: ST485	RS485 buffers	Fit	Fit	omit
SK1: RJ45	Control panel connector	omit	omit	omit
SK2, 3: RJ45	RS485 connector	optional	Fit	omit
PL1: R/A plug	RPC uplink connector	optional	optional	Fit
R27	Disable serial buffer	omit	Fit	Fit

What else will you need?

You will need the following tools and supplies:

- Small side-cutters
- Small snipe-nosed pliers and/or tweezers
- Magnifying glass or illuminated magnifier.
- Size 1 cross-head screwdriver.
- Straight edge such as 1" wide steel rule
- Multimeter
- Soldering iron with small bit (Maplin N78AR, see tool tips below)
- 18SWG and 22SWG (or similar) solder.
- Heatsink compound (Rapid 36-0400)

The side-cutters are wire-cutters with one side of the blade straight. These give cleaner cuts than wire cutters designed for thicker wires, which just crimp the wire until it breaks.

Other, optional, but useful tools include the following, also see assembly tips below:

- Bendy jig (Merg)
- De-soldering pump (Rapid electronics code 85-0600)
- Crimping tool (Rapid 85-0262)
- Soldering sponge (Rapid 85-0882)
- Support foam (Rapid 87-1526)
- PCB stand (Rapid 34-3622)

Tool tips

A 'bendy jig' (available from MERG) will make bending component leads much easier - or you can make your own. If you make a mistake a 'de-soldering pump' is recommended.

For the crimp connectors you will also need a crimp tool. It is possible to crimp with pliers, but it is very slow, you will get a less reliable joint, risk damaging the wire, and there is a chance that the crimp pins will be difficult to insert into the shrouds.

A reasonably fine soldering iron tip is essential, smaller than the 'pads' on the 'solder side'.

The ideal iron has a less than 2 mm 'chisel' tip and is temperature controlled to 300C.

A 1 mm pointed tip also works well for most thin-lead components.

Larger tips can be used but you will need to take more care to avoid solder bridges and splashes.

Use 22 SWG resin-cored solder for the small components, keeping the more readily available 18 SWG for the power components. (A suitable soldering iron is the Maplin soldering station code N78AR).

Wipe the iron tip on a damp sponge before each use and wet it with a little solder to improve thermal contact

with the work. Then place the tip so that it touches both the PCB trace and the wire to be soldered, and feed solder in until it wets both the wire and the PCB trace and surrounds the wire. Never use the iron to carry the solder to the joint - the resin that cleans the joint surfaces will boil off before the solder reaches the board.

The ideal joint has been described as 'Mount Fuji' shaped. Any similarity to a 'ripe tomato' suggests that the solder did not 'wet', and the joint should be re-made. Avoid breathing the resin smoke. I suggest using a small desk fan to create a slight air movement, but not so much that the soldering iron cools too much!

With practice, each joint should take one second or less (longer for components with thick leads, such as the rectifier diodes). If the joint is not formed after the iron has been in contact for about 3 seconds, allow the component to cool and try again later.

Time must be allowed for cooling before soldering a second (or third...) joint on the same component. This is especially important for semiconductors. To save time, fit several components at once, solder one lead of each component, then return to solder the other leads. After completing each batch, use the magnifying glass to check for good joints and remove any solder splashes that might short-circuit between tracks.

Finally use side cutters to cut off any surplus wire above the solder.

Some components are mounted quite close and it is important to crop the leads to ensure that adjacent leads cannot short if pressure is applied to the board. Yes, I could have made the board bigger and spaced everything out more, but the PCB accounts for 16% of the total cost and price is proportional to area.

You will see a black residue forming on the tip after a few joints. Wipe it off on the sponge before it drips onto the board and leaves a sticky mark. Not only is it messy but makes inspection difficult - you cannot tell if there is a solder bridge under there!

If you do make a mistake you will need to remove a component.

This is where a 'de-soldering pump' (also known as a 'solder sucker') comes in handy.

First operate the plunger so that pressing the 'trigger' button will cause it to fly back and 'suck'.

Place the nozzle in contact with the joint where you fed in the solder, push the iron tip on to the opposite side and, as soon as the solder melts, release the 'trigger'.

Hey presto! The solder disappears. You may have to do this more than once to remove all the solder.

When you have removed solder from both (or all of) its joints, the component can be removed by hand - sometimes it will need to be jiggled slightly as the lead is touching part of the hole and a tiny spot of solder is still holding it. Make sure that the hole is clear of solder before you insert a fresh component or you risk damaging the PCB.

Note that desoldering is more than twice as hard with a double sided board than you might be used to with a single sided board. This is because the copper track runs through the holes and so it is difficult to remove all the solder. This is why commercial PCB rework stations use vacuum devices like Rapid code 85-1190 which most of us cannot afford. It is much safer not to make a mistake in the first place!

Assembly tips

These instructions are written assuming little knowledge or experience. However, even experts must examine the options section carefully.

The PCB has a 'component side' with white outlines (known as the silk screen) showing where each component fits (also shown in appendix 1) and a 'solder side'. Both sides have tracks to connect the components together. A single sided board (with tracks on one side only) is impractical for complex designs like the QTU. Therefore a double sided board with copper tracks on both sides is used. Both sides also have a 'solder resist' which is the green colour wash over the whole board. The only places not covered by solder resist are where you should solder. A solder resist is used to reduce the likelihood of making solder bridges, but as the component density is high, great care still needs to be taken.

On the PCB, component names (R1, C1 etc) are printed outside their outlines where possible so that they will remain visible after assembly.

Components fit over their outlines on the component side. A suitable procedure for most components (but not the ICs) is:

1. Remove the component from its packaging, or from the paper strips.
2. Bend the leads to suit the PCB hole spacing.
3. Insert into the board from the component side, taking care to use the correct orientation.
Some outlines are marked to show which way round the component should go.
Other components can be inserted either way round.
4. Bend the leads outwards, thus holding the component in place.
5. Trim the leads leaving 2-3mm
6. Solder to the board on the solder side.

I find it efficient to process about a dozen components at once - bend them all, insert and secure them all and so on.

Even though components like resistors can be inserted either way around, it is a good idea to adopt a strategy of inserting them so that the colour rings can be read easily so you can double check that the correct component is fitted. Try to orientate resistors so that the first colour ring is uppermost, leftmost, or whatever you are most comfortable with.

It is vital that every component should go into its allocated place and the right way round where appropriate. It takes less time to double-check every component before soldering than to correct mistakes. If you do find a mistake, put it right before continuing. Take special care with the integrated circuits. If you solder one in the wrong way around, do NOT try to remove it unless you have specialist equipment. Instead cut all the leads, discard the chip and then remove the pins one at a time. Yes it costs a new chip but all the chips are much cheaper than a new board.

Assembling the PCB:

Components are fitted generally in order of thickness so that each new component will be pressed close to the PCB when you turn it over for soldering. It can be useful to have a piece of foam to lay the board on (component side down) during soldering so that components are held in place better than simply lying on the desk. An even better solution is a PCB stand which holds the board giving access to both sides,

1. Fit mounting spacers

Turn the PCB upside down (silkscreen down) and identify the SIX mounting holes. You will need to support the board off the desk, but horizontal and stable and leaving space for the soldering iron (see below).

Take care to avoid the heatsink mounting, grounding pad and RS232 mounting holes.

Tin these 6 holes with solder.

Holding the soldering iron point upwards, using pliers put a spacer over the soldering iron tip. When it is hot enough, put 2 or 3 small drops of solder on the uppermost end of the spacer.

Push the iron through a mounting hole, thus holding the hot spacer against the PCB. Ensure the solder between spacer and board is liquid, Use pliers to hold the spacer against the board and extract the iron. Quickly ensure the spacer is centered on the hold.

Turn the PCB the right way up when the solder has cooled.

Mounting the spacers first means that the bottom of the board is not covered with sharp points as you assemble the board, thus protecting both the assembly and whatever you place it onto. If you have a suitable scratch resistant and heat resistant surface then you might like to delay fitting the spacers until the connectors are fitted, as these support the board for fitting the spacers. Another chicken and egg situation!

2. Assemble heatsink (30 mins)

Position the heatsink on the PCB with the vanes overhanging the end of the board and centred in the silkscreen box.

Consider the end near the power-IN (three way connector, CN1) to be the left hand end of the heatsink, mark with pencil near the slot for the six transistors, at these points from the left-hand end:

5mm
17mm
30mm
50mm
69mm
88mm

This slot will take the M3 bolts supplied, but is not pre-threaded. When you insert a bolt ensure it stays perpendicular to the heatsink as you screw it in. If it goes crooked unscrew and try again.

If you find this stage difficult then a good trick is to drill a 3mm hole in a block (approx 30mm x 30mm x 5mm) of your preferred material (wood, plastic, metal). Drill the hole about 7mm from an edge of your block and draw a line from the centre of the hole to the edge of the block. Then using a longer M3 bolt, pre-thread the heatsink by holding the block such that the line aligns with a mark on the heatsink.

Take the 12V regulator U23 (L7812CV) and push a plastic insulating washer through the mounting hole from the front of the transistor. Smear the back with a thin layer of heatsink compound if you have any (not included in kit) and place a mica washer on it. Smear more heatsink compound on the mica washer. Only use a very thin layer otherwise heat conduction will be retarded, not enhanced!

Bolt the assembly to the heatsink using one of the 6mm M3 bolts, at the 5mm mark. The bolt should be tightened almost all the way, but allow the transistor to pivot slightly so it can be aligned when the assembly is fitted to the PCB. Do NOT attach the heatsink to the PCB yet!

Ensure the mica washer sticks out all around the transistor and that the plastic washer engages in the hole of the mica, and does not tear into the mica.

Repeat with the 5V regulator (L7805CV) at the 17mm mark.

Repeat with the output transistors (TIP120) at the other four marks.

Offer up the heatsink assembly to the board and get all the leads through their holes.

With the heatsink centred in its silkscreen box, tighten the six bolts, while ensuring that the transistors do not twist around.

Remove the heatsink assembly. Wipe off any excess heatsink compound.

Using a continuity tester or multimeter, check that none of the 18 leads connect to the heatsink (put one probe on any one of the six screws, not the black metal). If you have a connection the mica washer is probably incorrectly positioned or is damaged.

3. Fit small passive components mounted flat.

Using the techniques mentioned in 'assembly tips', begin assembling the board. Mark the page with a tick when you have completed each line of the list. The first column indicates what spacing to bend the leads at.

The location column shows where the components are: x/y measured in millimetres. With the board held with the heatsink to the right, which means most of the silkscreen designations are the right way up 0/0 is the bottom left corner - near one mounting hole of the RS232 connector. Where a group of components are listed only one dimension will be iterated, such as 10/1,2,3,4 being short for 10/1, 10/2, 10/3, 10/4.

When mounted, the board is normally other way up - heatsink on the left, RS232, RS485 and stack plug

uppermost. Extra RPC boards plugging in below one another.

Type & Value	Markings	Designations	Location
Wire link		J1	153/87: relay 12V or 24V
		Q7, D11, D12, D13, D14	128/90, 90,85,80,75/27 *OPT*
Resistors			
0.3 33K, 1%	Or, Or, Blk, Red, Brn	R106, R206, R306, R406	182/28,31,58,61
0.3 150R, 1%	Brn, Grn, Blk, Blk, Brn	R103, R203, R303, R403	182/25,33,55,63
0.3 820R, 1%	Grey, Red, Blk, Blk, Brn	R5	176/84
0.3 4K7, 1%	Yel, Purp, Blk, Brn, Brn	R4	179/84
	1R, 1%	Brn, Bk, Bk, Silver, Brn	R101, R102 etc
			195/27,24,45,64,83 *OPTIONAL*
Diode			
0.4 30V	BZX 33C 30	ZD1	181/84
0.3 1N4148	41 48	D6, D8, D7	183,186,189/84, other way to ZD1
0.3		D4, D5, D9, D10	144,154/91, 132/99,97
0.4 8V2	BZX 55C 8V2	ZD2	130/94, band towards heatsink
Resistor			
0.3 33K	Or, Or, Blk, Red, Brn	R22	184/84
0.3		R8	131/93
0.3		R6, R7	98/23, 101/23
0.4		R28	93/41
0.3		R18, R20	62/59, 56/54
0.3		R27	6/21 *OPTIONAL*
0.4 4K7	Or, Purple, Blk, Brn, Brn	R25, R26	98/37, 103/105
0.3 220R	Red, Red, Blk, Blk, Brn	R23, R24	143/100, 154/100
0.4		R1, R2, R3	39,41,30/57
0.3 1K	Brn, Blk, Blk, Brn, Brn	R9	103/23
0.3		R21	69/57
0.3 180K	Brn, Grey, Yellow, Gold	R19	57/55
Capacitor			
	0.47uF	474	C13, C18
	0.1uF	104	C4, C20
			C3, C5-C12, C14
	22pF	22	C100, C200, C300, C400
			169/23,34,54,65
Diode			
0.4 1N4001	1N4001	D101, D201, D301, D401	195/13,32,51,70
0.4		D3	192/92
0.4 varies	See optional components	D100, D200, D300, D400	195/15/34/53,72

If you have a V1.4 board the cathode ends (the end with the ring) of D400 and D401 need to be connected. Somehow this link got lost during the PCB update!

Photo [QTU Flat Passives.jpg](#) shows the board at this stage.

4. Fit most ICs

ICs are static sensitive. Leave in tubes until ready to insert. Handle as little as possible. Avoid plastic worktops. The pins are usually splayed apart more than the hole spacing. Place a chip on the board in the orientation indicated by the silk screen. Double check chip orientation - they are very hard to take out once soldered in!

Manoeuvre the chip so that one row of pins drops through their holes. Now use a finger on the back of the

board to lift the chip so that those pins have only dropped through to the back of the board and no further. Using a thin straight edge, such as the end of a steel rule, press the whole second row of pins sideways towards the chip so that all the pins will go into their holes. Now push the chip fully home. The pins will spring back and hold the chip in place.

Check that all legs have gone through the board! It is possible to bend the legs under the components and this can detrimentally affect the performance of your board!

I insert all the ICs first, then check the pins have gone through the board, then solder the corner pins and check that the devices are flat against the board. Then I solder one complete row of pins on a chip, then wipe the iron on the sponge, then do a row of a different chip.

Type	Designations	Location
LM324N	U25, U26	168/29, 168/59
HFC4051BE	U2	168/44
	U3	108/46
	U4	43/75
ULN2803A	U12, U14, U16	12/50, 33/37, 63/37
	U17	110/73
HFC4094	U11, U13, U15, U18	Next to the four ULN2803A chips, but opposite orientation
ST485	U7, U8	22/15, 22/26 *OPTIONAL*
ST232CN	U9	11/20 *OPTIONAL*
CD4021	U19, U20, U21, U22	77/92, 52/92, 28/92, 17/75
MC14555BCP	U5	99/85
CD4069	U27	127/47

At this point three ICs are still missing from the board - the most expensive ones:

U1 - the processor

U10 - the D/A converter

U6 - the eeprom (which is for future expansion and not included in the kit).

The first two will be added once the power supply has been tested.

Photo *QTU ICs.jpg* shows the board at this stage.

5. Fit upright passive components, switch, LEDs

Type & Value	Markings	Designations	Location
Network			
100K	• A 104G - dot is pin 1	RP1, RP2, RP3, RP4, RP5	75,50,20/99, 9/85, 35/75
Resistor			
8K2 or 33K	* OPTIONAL *	R35, R36, R37, R38	93/27, 88/27, 83/27, 78/27
820R	Grey, Red, Blk, Blk, Brn	R110, R210, R310, R410	160/22, 160/34, 160/53, 160/65 adjacent to U25, U26
1K, 1%	Brn, Blk, Blk, Brn, Brn	R105, R205, R305, R405	160/18, 160/38, 160/50, 160/69
4K7	Yel, Purp, Blk, Brn, Brn	R109, R209, R309, R409	162/20, 162/36, 162/51, 162/67
		R107, R207, R307, R407	170/18, 170/38, 170/50, 170/68
		R108, R208, R308, R408	189/22, 189/40, 189/65, 160/75
		R111, R211, R311, R411	200/24, 200/43, 200/62, 200/81
180K	Brn, Grey, Yellow, Gold	R104, R204, R304, R404	180/20, 180/35, 180/51, 180/66
Switch			
6-way		SW1	30/70 *OPTIONAL*
LEDs			
RED	Longer lead is +ve	LED1	55/59
YELLOW		LED2	45/59
GREEN		LED3	35/59
Diode			
1N4148	41 48	D11, D12, D13, D14	90,85,80,75/27 *OPTIONAL*
		D102, D202, D302, D402	191/19, 191/33, 191/57, 191/76
1N5401	1N5401 or P 600 G	D1, D2	190/104, 190/96

Check that components are upright and not touching.

6. Fit all transistors (15 mins)

Fit all the transistors at once, then trim the legs, then solder one lead of each transistor before moving on to the second lead.

Type	Markings	Designations	Location
BC337	C33725	Q100, Q200, Q300, Q400	175/22, 175/38, 175/53, 175/68
		Q102, Q202, Q302, Q402	185/22, 185/38, 185/53, 185/68
		Q2, Q3, Q4, Q5	92,86,81,76/32 *OPTIONAL*
		Q6	183/75
		Q7	128/88 *OPTIONAL*
BC327	C32740	Q101, Q201, Q301, Q401	196/20, 196/39, 196/59, 196/77
		Q1	65/53

7. Fit connectors, relays and remaining capacitors.

These connectors can be fiddly - they do not stay in their holes. I tend to persuade a few at a time to sit roughly in position while I solder a single pin of each. Then one at a time re-melt the solder while pushing them into position. When all are positioned correctly I solder them in. Remember to wipe the iron regularly.

Take care with SK6. If you are using the molex style (0.1") header - the board has NINE holes! This is to allow a resistor network to be fitted for garden railways. Do not use pin 1 (with a box around it on the silkscreen) - position the connector to the heatsink end of the nine holes. Also if you have a V1.4 board the holes are a little small for the molex connectors (they resized to suit the resistor pack!). Filing the corners of the connector pins should make it fit. Alternatively allow it to stand proud, or use a 0.2" style connector.

Type	Designations	Location
3 pin links	LN1, LN2, LN3	12/62, 20/37, 48/37
10 way straight	PL5, PL6, PL7, PL8	By resistor networks. Tab to inside of board
	PL9, PL10, PL11	5/50, 40/23, 80/20. Tab towards board edge
	PL3	115/50. Tab away from processor
	PL13	150/105, Tab to inside of board
12 way straight	SK4, SK5	120/11, 140/82. Tabs to relays
8 way	SK6	165/12. Tab towards relays *OPTIONAL* straight header or pinstrip
3 way pinstrip	CN1	175/102
BT47	RL1-RL8	Between 12 way connectors
Capacitor	C22, C23, C24, C26	10uF 63V 143/97, 148/97, 155/97, 163/97
	C2	470uF. 111/93
	C1	2200uF. 168/86

Photo [QTU Connectors.jpg](#) shows the board at this stage.

8. Inspect carefully

Using the magnifier and good lighting (or better still, an illuminated magnifier) examine carefully every joint on the solder side of the board. Check that there are no solder splashes or solder bridges, and that all joints are wetted properly, and that the joints are actually soldered!

Re-examine the component side and check that every component is in the correct location, and in the correct orientation.

Mistakes found at this stage are easier to fix than once power has been applied and some magic smoke emitted.

At this point the board should be missing a few components:

Type	Designation	Function	Why missing
IC	U1	Processor	Test power before risking chip
	U10	D/A converter	Test power before risking chip
	U6	EEPROM	Future expansion
	U7, 8	RS485 buffer	Optional
	U9	RS232 buffer	Optional.
Resistor	R10-R17	Protection	Future expansion
	R102,201 etc	Current scaling	optional component
	R27	Buffer disable	Fitted if U9 omitted
Connector	SK1	RS485	Only used for control panel link
	SK2, SK3	RS485	Optional, not yet fitted
	SK7	RS232	Optional, not yet fitted
	PL1, PL2	RPC stack	Optional, not yet fitted
	PL4	Expansion	Future expansion
	SK6a	Track output	Second row for press to release connector
Link	J2	Programming	Development only
	J3	5V	Test point
	J4	Ethernet	Future expansion
Capacitors	C27-C30	RS232	Optional
	C15, C16	Crystal loading	Not required with ceramic resonator
Transistor	Q2, Q3, Q4, Q5	Tortoise drive	Optional, with associated components
Heatsink			Omitted until board inspected

9. Fit heatsink assembly

Reposition the heatsink onto the board, and fix with the remaining 6mm M3 bolts. Solder in the six transistors.

10. Power up and check supplies

Connect the AC input. To play safe you could insert a resistor in series with each of the two feed wires (the outer wires on the three way connector), perhaps around 100 ohm.

Connect a meter to ground, use either the middle pin of the power connector, or insert an M3 bolt and washer in

the ground point (near the track output connector). The ground point is the hole with the track connecting it to the track around the edge of the component side. The other hole (without a ground connection) is a mounting hole.

Turn on power and check the following:

Meter scale	Probe point	Should read
AC Volts	PL13 pin 1	15V (18V for G scale)
	PL13 pin 2	15/18V
DC Volts	PL13 pin 3	21V (25V for G scale)
	PL13 pin 4	+22V
	PL13 pin 5	-22V
	J3 heatsink end	5V
	PL13 pin 9	7V or 12V (tortoise motor supply)
	PL13 pin 10	12V
	D6	21V and 28V
DC Amps	Across J3	< 40mA

The last measurement - the current consumption on the 5V line is rather variable. Ideally it should be less than 1mA, but it is possible for some CMOS chips to oscillate as the processor is still not fitted in which case 40mA is plausible.

If you get less than 1mA then everything is fine.

If you get over 40mA assume that a chip is in upside down or you have a short and inspect the board again.

If you get readings in between do the repeat inspection anyway, but if all looks good proceed to the next step.

11. Fit last components

Some connectors have been left to last as they interfere with any PCB holding mechanism if you have any.

Type	Designation	Function	Location
IC			
	U1	Processor	83/60
	U10	D/A converter	170/73
Resolator			
	X1	Timing reference	95/65
Wire link			
	J3	Connect 5V	193/89
Connector			
	SK2, SK3	RS485 connector	62/10, 45/10 *OPTIONAL* (NOT SK1)
	SK7	RS232 connector	17/7 *OPTIONAL*
	PL1	RPC stack uplink	122/2 *OPTIONAL*
	PL2	RPC stack downlink	122/102

Photo *QTU assembled.jpg* shows the board at this stage.

12. Power up and check LEDs

The processor is supplied pre-programmed with the bootloader. This means that when powered up the red LED should flash constantly once per second. This flashing indicates that the bootloader is running. Other LED combinations include:

RED	YELLOW	GREEN	Meaning
off	off	off	No power (probably)
wink	wink	off	Normal power up indication
off	off	flashing	Local script running, no serial comms One flash per 256 executions of script
off	off	on	The green led is flashing very fast
off	flashing	off	Serial comms working, probably downloading EEPROM
off	flashing	flashing	Normal working using computer messaging
flash	flashing	flashing	Errors occurring on serial comms Check RED is not regular 1Hz
1Hz	off	off	No code, bootloader waiting
1Hz	flashing	off	Initial comms to bootloader
1Hz	flashing	winking	Each green wink is writing a page of code

13. Connect to PC and download desired script.

The QTU holds THREE different pieces of software:

1. The bootloader. This is what allows Tcc to download firmware updates. The bootloader is pre-programmed into the chip and cannot be changed.
2. The firmware. This is what makes the board function as a QTU and is updated automatically by Tcc.
3. Any QTU script you may have written and compiled on Tcc. You do not need a script to drive a QTU from Tcc.

Using whichever comms link the board is configured for (RS232, RS485 or RPC stack) connect the board to the computer and start Tcc using qtu.tcl (<http://groups.yahoo.com/group/mergtcc/files/QTU%20kits/qtu.tcl>)

If you haven't already configured the network:

Open the network configuration window (Menu->Configure->Network).

Add a serial interface. (New interface->Serial interface)

Add a Merg Stack (New interface->MERG stack)

Open the stack (click on the icon to the left of the new stack item)

Check that the stacks' interface is the one you have just created.

Set stack protocol to QTU and preferred rate to 20.

Set the address switches on the board to a value not used on other boards (unless its a stack slave).

Switch 1 (nearest the green LED) is bit 0. Switch 5 is bit 4. Switch 6 is not used currently. Bits 7,6,5 take the value '010'. Set the stack address to match the address you have just set on the hardware. Addresses range from 64 (01000000, all switches 'on') to 95 (01011111, all switches 'off')

Add a QTU module to the new stack (Stack->New module->QTU)

Open the serial port (click on the icon to the left of the new serial interface)

Select the port for the serial interface.

Set the baud rate of the serial interface to 57600

If you are testing a second (or subsequent) board on the same stack then check in the network configuration window that your stack has the correct number of QTUs defined.

If another is required then use Stack->New Module->QTU.

If you are testing a third (or subsequent) board, then qtu.tcl will require extending suitably.

Start the stack running. (Stack->Start)

Check that the orange LED start flashing on the QTU. If it doesn't then look for an error message on the serial interface in the network config window. If that is clear then check the log window. The log window should show a few messages sent to and received from the QTU (labelled 'Send' and 'Rcvd'). If the log shows messages being sent but the orange LED doesn't flash then check the serial lead, and that its in the correct serial port on the PC.

[To be reworked as a fault finding list]

If all is well the QTU (in the new stack in the network configuration window) should show 'xx% code updated' indicating that all is well and new firmware is being downloaded.

14. Check green LED flashes (indicates script is running)

Once the firmware has been downloaded the green light will flash very quickly - it will probably look as if its on constantly. Thats because there is no script downloaded and so the end of the script is reached very quickly!

15. Check basic I/O

All the inputs should show up as ones in the sensor window. Connect something (such as a DIP switch connected to a connector block) to the four input connectors one at a time to check they are working.

Test outputs 24..27 using the controls window and check that the relays click. If they don't then check the relay power strap J1 near the power output connector.

[This section not yet finished. Much work to do...]

Symptom	Check	Possible cause
One channel dead	Rx01 burnt out or open circuit	Heatsink washer allowing transistor to short to heatsink.
General misbehaviour	AC supply	If only one of the two AC feed inputs are connected then strange things can happen.
No LEDs on or flashing	AC supply connected	Not switched on and plugged in
	5V link or resonator missing	Assembly incomplete
Red LED flashing	Serial lead to PC, Tcc running	No firmware downloaded.
Yellow LED not flashing when Tcc is started	Tcc Log - should show data being sent	Serial interface wrongly configured (try a different baud rate)
	Address switches wrong - change to all switches 'on' and address 64	
	Serial cable connections	