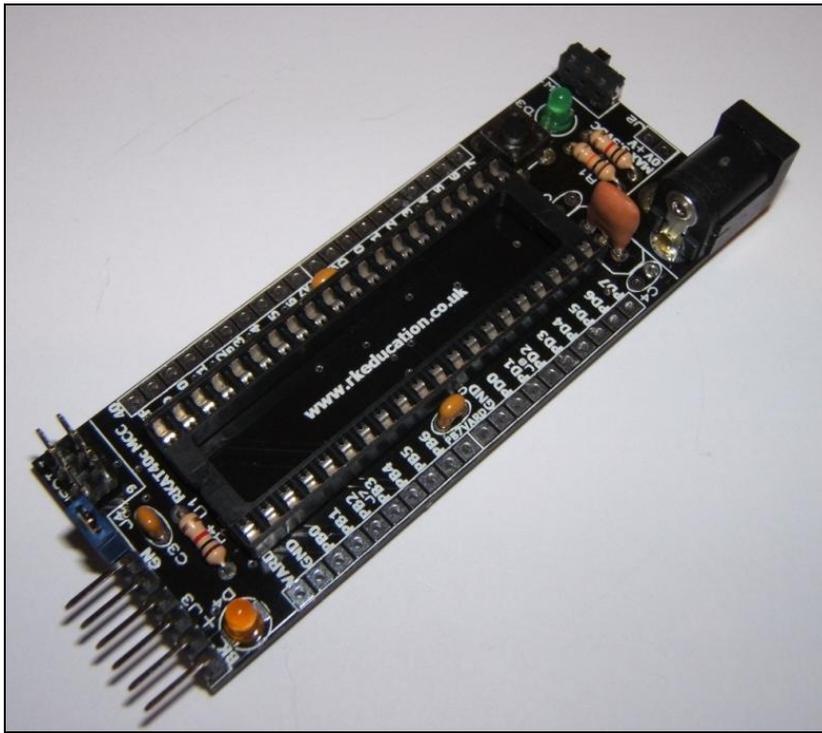
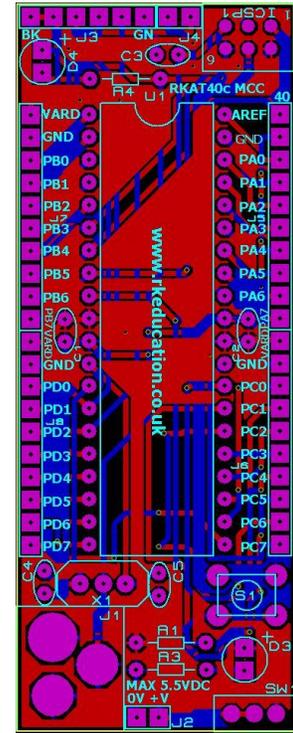


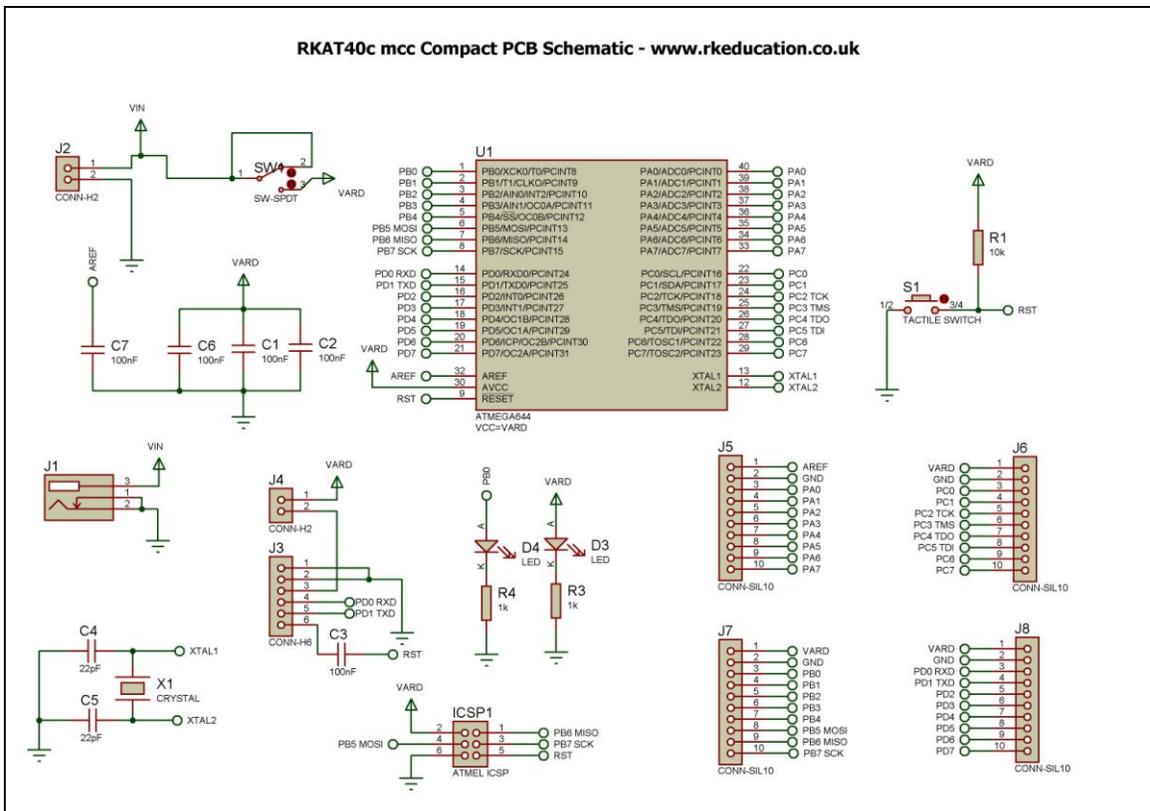
# RKAT40c MCC Component List and Instructions



Constructed PCB



PCB Layout



Schematic Diagram

## Description

The RKAT40c MCC compact project PCB has been designed to use ATMEL microcontrollers such as the ATmega1284p, ATmega644p and Sanguino

- A low cost method of producing Atmel projects
- Uses a high quality, double sided PCB
- Software is downloaded from a PC into the microcontroller from an Atmel AVR programmer or a FTDI USB cable
- The clock reference can be a ceramic resonator or crystal oscillator
- All input and output pins have a PTH
- Easily interfaced to peripheral devices
- Power supplied via either a DC socket or 2 way header – 5.5VDC max
- Power can also be supplied from the FTDI cable, there is a 2 way header plug to select USB power
- Power switch and LED power indicator

## Component List

C1, C2, C3, C6, C7 – 100nF multilayer ceramic capacitor

D3, D4 – 3/5mm LED

ICSP1 - 2x3 way header plug

J1 - DC socket

J2 - header for power supply

J3 - 6 way right angled header plug

J4 - 2 way header plug with jumper tab

J5 ~ J8 - 10 way SIL sockets (optional)

R1 – 10k ¼ watt resistor (brown black orange)

R3, R4 – 1k ¼ watt resistor (brown, black, red)

S1 - tactile switch

U1 – 40 way DIP socket with ATMEL microcontroller

Ultra miniature slide switch for power switch

X1 –ceramic resonator or crystal oscillator, when using an oscillator C4 & C5 will need to be used

## Instructions

The PCB has been designed to use Atmel microcontrollers and MCUs based on Atmel such as Sanguino, for instructions on using your chosen microcontroller please see the appropriate website and/or forum.

## Connecting Power

Power is connected to the PCB via the DC socket J1 or the 2 way header next to J1 marked J2. The 0V input, usually black is marked clearly as is +VE which is usually red, a regulated power supply should be used. The recommended voltage range for an Atmega1284P-PU MCU is between 1.8 and 5.5VDC, please consult the technical information for your chosen MCU.

A power switch has been included and is to the right of J2. When the PCB is powered the LED D3 should light up.



RKAT40c mcc Compact PCB Schematic - www.rkeducation.co.uk

