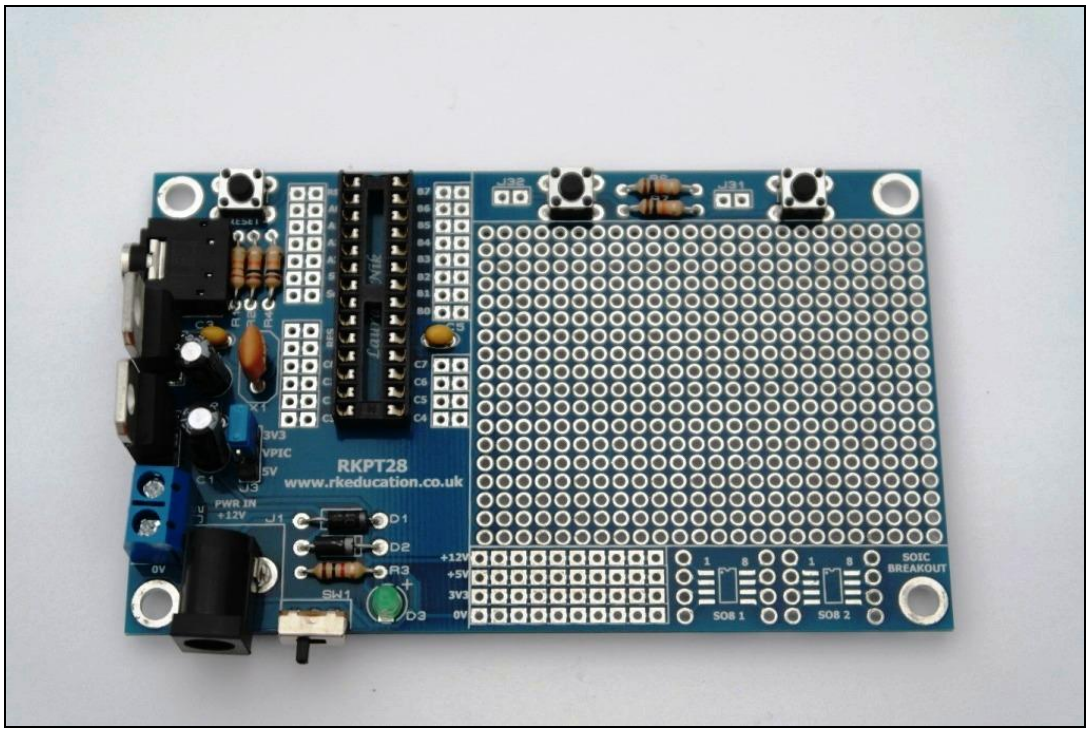
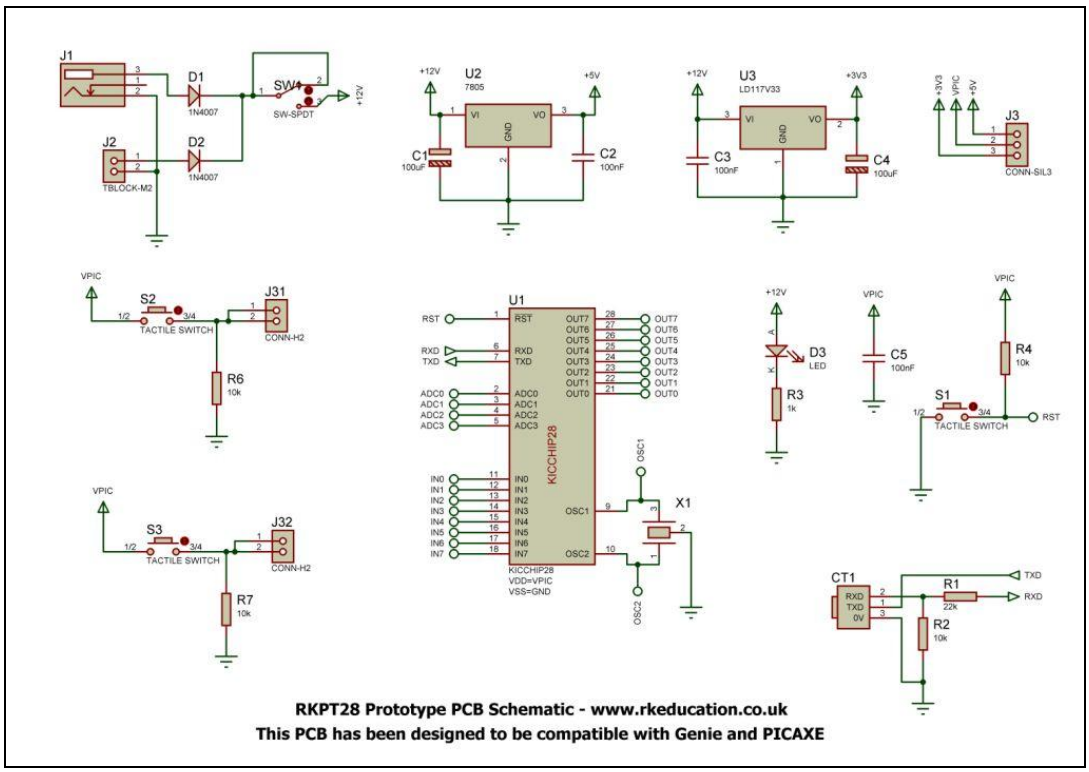
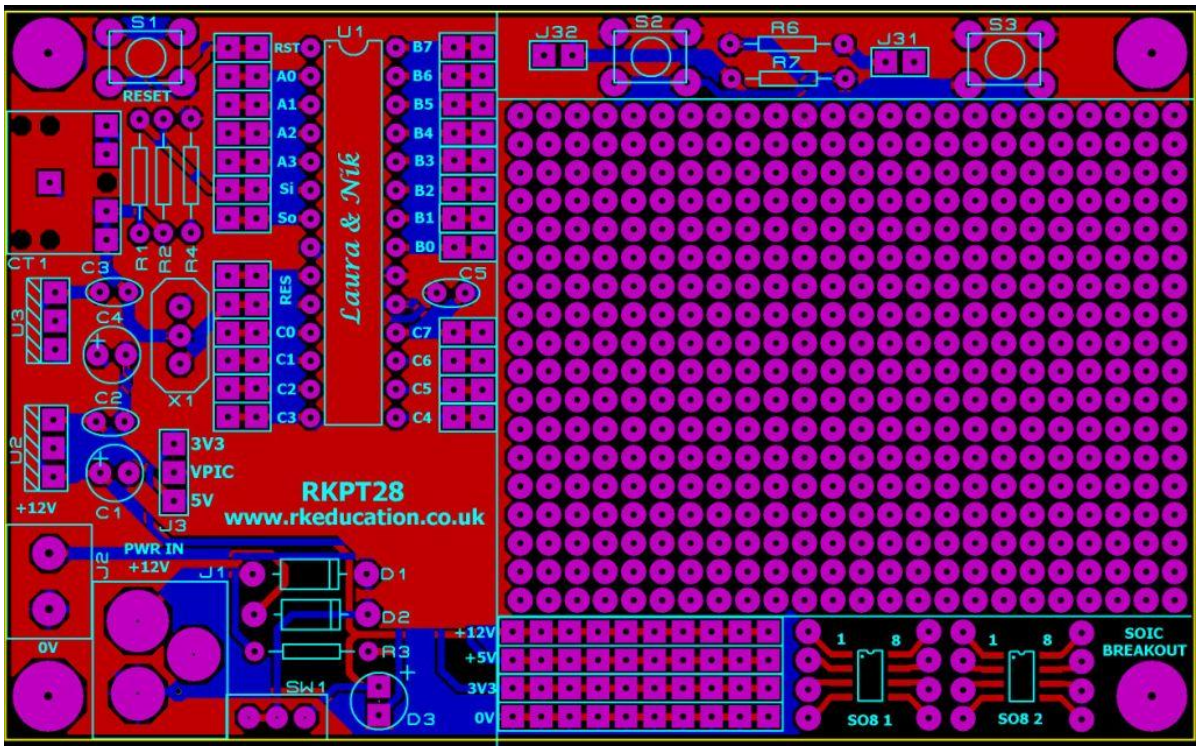
RKPT28 Component List and Instructions



Constructed PCB



Schematic



PCB layout

Description

The RKPT28 prototype project PCB has been designed to use PIC microcontrollers such as Genie and PICAXE

- Software is downloaded from a PC into the microcontroller via a 3.5mm stereo socket
- The clock reference is from a ceramic resonator, 4MHz supplied
- All input and output pins have a PTH
- A large prototyping area
- 2 tactile switches available for prototyping
- 2 SOIC footprints for prototyping
- Power rails on the prototyping area
- Powered via a terminal block or DC power socket
- +12VDC input and +5VDC and +3V3DC regulated outputs
- Power to the MCY selected using a jumper link, either 3V3 or 5V
- LED used to indicate power
- Power switch and LED power indicator
- High quality, double sided PCB

Component List

J1 – 2.1mm DC socket
J2 – 2 way 5mm pitch terminal block
J3 - 3 way 2.54mm PCB header plug with jumper
C1, C4 – 100uF electrolytic capacitor
C2, C3, C5 – 100nF multilayer ceramic capacitor
D1, D2 – 1N4007
D3 – 3mm LED (power indicator)
R1 – 22k ¼ watt resistor (red red orange)
R3 – 1k ¼ watt resistor (brown black red)
R2, R4, R6, R7 – 10k ¼ watt resistor (brown black orange)
CT1 – PCB mount 3.5mm stereo connector (software download)
S1, S2, S3 – 6mm tactile switch (S1 reset, S2, S3 prototyping)
SW1 - Ultra miniature slide switch for power switch
U1 – 28 way DIP socket with microcontroller e.g. Genie/PICAXE
U2 – 7805 5V voltage regulator TO220 package
U3 - LD1117V33 3V3 voltage regulator TO220 package
X1 – 4MHz ceramic resonator

When constructing always start with the components that have the lowest profile and work high, for example start with the resistors and end on the 7805 voltage regulator.

Instructions

The PCB has been designed to use microcontrollers based on the PIC e.g. Genie/PICAXE, for instructions on using your chosen PIC please see the appropriate website.

Connecting Power

The power is connected to the terminal block marked PWR IN, the 0V input, usually black is put in the lower terminal and the +VE, usually red, is put in the upper terminal, power can also be supplied via the 2.1mm DC socket, a regulated 12VDC 1Amp power supply should be used. The circuit incorporates 7805 and LD1117V33 voltage regulators and 3V3, 5VDC and input V is available on the prototyping area as 3 power rails, a heat sink may need to be added to the regulators if a high current is required and heat is an issue.

A power switch has been included and is labelled SW1.

Downloading software

Once the software has been written using the Genie/PICAXE Programming Editor (or equivalent) it can be downloaded into the Genie/PICAXE (or equivalent). This is downloaded using a download cable that connects either to your PC's serial port or USB port. Insert the download plug into the download socket and activate the program function in your Programming Editor. If all goes well it will tell you the program download was successful.

Using the prototype area

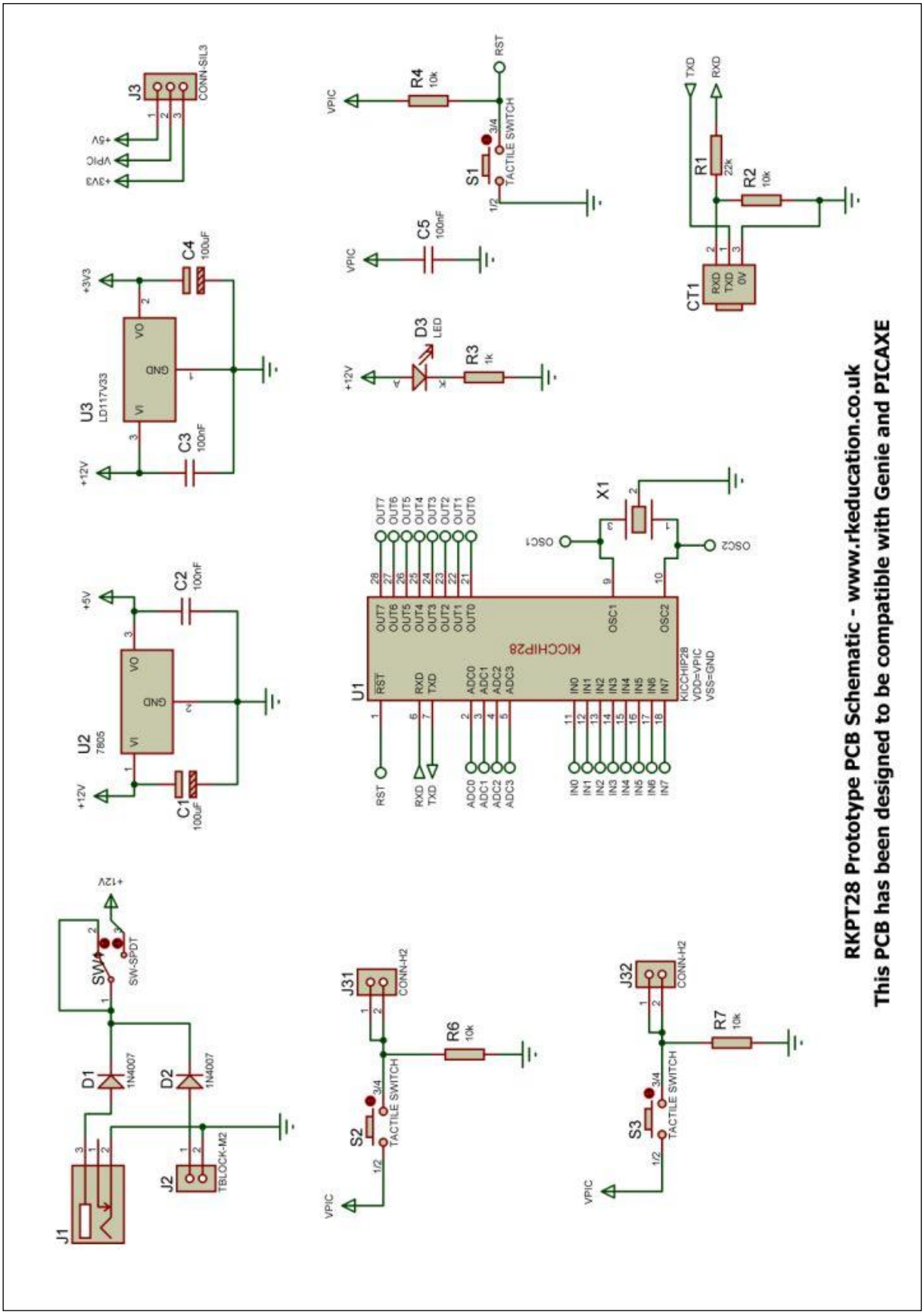
Using the prototype area is simple and how it is used is dependent on what is being done. Access to all of the pins of U1 is gained by through holes near the pins of U1, simply connect using jumper wires. There are power rails at the bottom of the prototyping area, these are labelled 0V, 3V3, +5V, +12V. Please note the +12V is the input voltage minus the voltage drop accross the 1N4007 siode, typically 0.7V.

Please visit our website

www.rkeducation.co.uk

If you have any comments or queries please email us at

technical@rkeducation.co.uk


RKPT28 Prototype PCB Schematic - www.rkeducation.co.uk
This PCB has been designed to be compatible with Genie and PICAXE
KICCHIP28
U1
U2
7805
U3
LD117V33
J1
J2
TBLOCK-M2
J3
CONN-SIL3
J31
CONN-H2
J32
CONN-H2
CT1
RXD
TXD
0V
S1
S2
S3
TACTILE SWITCH
SW1
SW-SPDT
D1
D2
D3
LED
1N4007
X1
R1
22k
R2
R3
R4
R6
R7
10k
1k
C1
C2
C3
C4
C5
100uF
100nF
+12V
+5V
+3V3
VPIC
RST
TXD
RXD
OSC1
OSC2
ADC0
ADC1
ADC2
ADC3
VDD=VPIC
VSS=GND