

# The Picaxe Hatchet Board

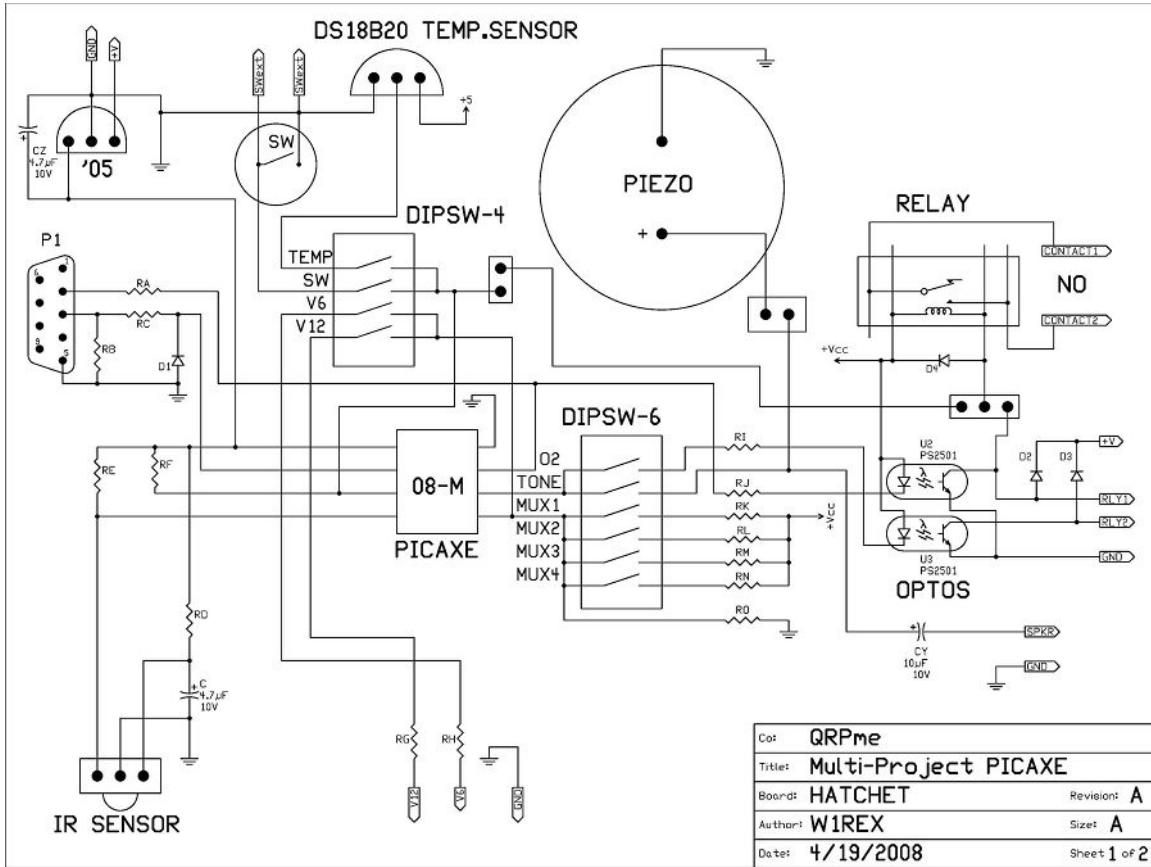
The Hatchet Board is a multi-project board comprising of a small easy to program Picaxe 08-M microcontroller and several input and output devices which enable it to be used in a variety of ways. I have written 6 different programs for the Hatchet board which are easily customized for personal preferences. Each program tailors the Hatchet board to perform as a useful controller around the shack or bench. These 6 basic applications are:

- Variable Delay T/R Switch
- Beacon Keyer
- I R Remote Controller
- Multi-Purpose Timer
- Multi-Setpoint Temperature Controller
- Remote Voltage Monitor

These 6 basic applications are provided as examples for what you can do with the Hatchet board. Use the programs as-is or modify away. If you come up with a neat application that you would like to share with other Hatchet users, please let me know and I will post it in the Hatchet folder.

As of right now, there is no detailed Step by Step tutorial on how to assemble the Hatchet kit. Assembly is quite straight forward; however, as the Hatchet board uses all through-hole parts, has a complete silk screen and solder mask, and has a minimum of parts. A basic knowledge of electronic parts and soldering techniques is required. Referring to the following schematic and parts list, one can see that there are 15 1/8 watt resistors, 4 diodes, 3 capacitors, 5 semiconductor devices and another 10 miscellaneous parts.

# Hatchet Schematic



# Hatchet Board Parts List

## Resistors:

A = 180 ohms	(Brown-Gray-Brown)
B = 10K ohms	(Brown-Black-Orange)
C = 22K ohms	(Red-Red-Orange)
D,I ,J = 330 ohms	(Orange-Orange-Brown)
E,F = 4.7K ohms	(Yellow-Violet-Red)
G = 100K ohms	(Brown-Black-Black-Orange)
H = 33K ohms	(Orange-Orange-Orange-Red)
K = 200K ohms	(Red-Black-Black-Orange)
L = 75K ohms	(Blue-Green-Orange-Red)
M = 33K ohms	(Orange-Orange-Orange-Red)
N = 12.5K ohms	(Brown-Red-Green-Red)
O = 50K ohms	(Green-Black-Black-Red)

## Capacitors:

X,Z = 4.7 uf
Y = 10 uf

## Diodes:

1 = 1N6263
2,3,4 = 1N4148

## Miscellaneous:

- RS232 Connector
- 2 - 8 Pin DIP Sockets for U1 and U2/U3 (U2 & U3 share a socket)
- 14 Pin DIP Socket for the Relay
- 4 & 6 position DIP Switches
- 2 pos & 3 pos .1" spacing Molex connector & 2 jumpers
- Pushbutton Switch
- Piezo Speaker
- DIP SPST Relay
- 2 - PS2501 Optoisolators
- Picaxe 08-M Microcontroller
- DS18B20 One-wire Temperature Sensor
- EL-IRM-8601S IR Receiver
- LM78L05AZP Voltage Regulator

## CONSTRUCTION:

Start with the smallest profile parts, the resistors. Note that there are 2 general style resistors. Several resistors have 3 color stripes on a solid color body. Other resistors have 'extra' color stripes and a different color body. These 'extra stripe' resistors are 1% tolerance resistors while the former are 5% tolerance. The 1 % resistors are used in the MUX switch and voltage divider circuits. The 5% resistors are general purpose resistors and are used in non-critical resistance circuits. Resistors are installed at locations A through O.

Next install the 4 diodes at locations 1 through 4. The 'different' colored BLUE diode is installed at location 1 and the other 3 diodes are installed at locations 2 through 4. Note the bar/stripe polarity on the diode silk screen markings and align the diode to match up the stripes. Ideally, 1N4004 diodes should be used across the coil terminals on the relays; but, they don't fit in the holes provided on the circuit board. You can mount the 1N4004 diodes EXTERNALLY across the coil terminals of the relays. Or you can use a little creativity.... Using clipped resistor leads, form two small eyelets and slip over the ends of a 1N4004 diode. Insert the leads from the eyelets into the diode holes and solder. Now solder the eyelets to the diodes and clip off all the surplus leads. OR the easiest but slightly less desirable solution is to use the 1N4148 diodes in place of the 1N4004 diodes. They work but don't provide as much protection as the 1N4004 diodes.

Now install the 3 DIP IC sockets at U1, U2/U3 and relay locations. NOTE: Insert the 4 position DIP switch next to the 8 pin U1 socket at the same time as the socket. Pay particular attention to the orientation of the part. IC sockets and chips have a small rectangular notch on the pin 1 end of the item. The silk screen also has a small notch noted on the pin 1 end of the item. Align part with the silk screen markings. Make sure both items are installed in their proper locations BEFORE soldering either item. These 2 items are spaced close together and completely soldering the socket and then inserting the DIP switches might prove difficult. So insert BOTH items first and bend over a couple of pins on each item to keep them in place. Now solder these two items in one step.

The 2 pin connector and jumper are mounted next to the 4 pos. DIP switch unit. Shorted, the relay is controlled by PI Caxe Pin4. The 3 pin connector and jumper are mounted next to the onboard relay. With the shorting plug to the right, the onboard relay is controlled from Pin4. When the shorting plug is to the left, the onboard relay is controlled by PI Caxe Pin0.

Install the pushbutton switch. You can also attach a pair of wires (not-supplied) to the EX pads and attach them to a remote user supplied switch.

The DS18B20 Temperature Sensor is usually mounted at the end of a 3 wire cable so that it can be mounted remotely in/at the item whose temperature is being monitored. Again, pay attention to the silk screen markings and orient the sensor accordingly. NOTE: the common ground lead between the pushbutton switch wire and the temperature sensor wire should have a small trace leading over to the ground plane...BUT it doesn't! The Fix: scrape a small piece of the solder mask close to the common lead between the switch and sensor. When installing one of the wires to the sensor or switch, strip it a little longer than necessary, insert it into its proper hole and extend it over to the ground plane and solder it in both places.

Install the IR Receiver chip paying attention to the orientation marked on the silk screen.

Next install the LM7805 Voltage Regulator, again, paying attention to the orientation marked on the silk screen. Attach a pair of wires (not supplied) to the + and - pads on the board to run out to your power source.

Now install the 6 position DIP Switch unit in its proper location.

The Piezo Speaker element is next. Make sure the + lead marked on the Piezo is inserted into the pad marked with the + on the silk screen.

Now install the RS232 Connector. You might want to add a little hot glue or something similar to the ends of the RS232 connector to add some rigidity to the connector/board mounting to prevent stressing the solder connections.

You should elect to power up the board and check out you handiwork now, BEFORE you install the easily damaged Picaxe microcontroller. Apply power to the power leads and check that there is +5 volts at U1 pin 1 and ground at U1 pin 8. Power down the board and proceed to the next step.

Next insert the Picaxe, Optos and Relay into the sockets and you are ready to try out the board for real. The Beacon program is probably the easiest application to use to test the board. Simply download and run. No modifications are needed as the supplied program has a default (mine) message to send and the Piezo sounder will provide a sidetone for checking that the program is working.

I won't try to detail the complete operation, theory and programming of the Picaxe micro. The Picaxe Hatchet board has the downloading circuit on board and connected to an RS232 connector instead of the 'standard' Picaxe 3.5mm stereo jack. This allows you to use a readily available DB9 PC serial cable to download programs into the Hatchet board. You can find a wealth of Picaxe info on programming, interfacing and downloading on the official Picaxe web site. The Programming Editor can also be downloaded for free at the official Picaxe web site.

OFFICIAL PICAXE WEBSITE:

[www.Picaxe.com](http://www.Picaxe.com)