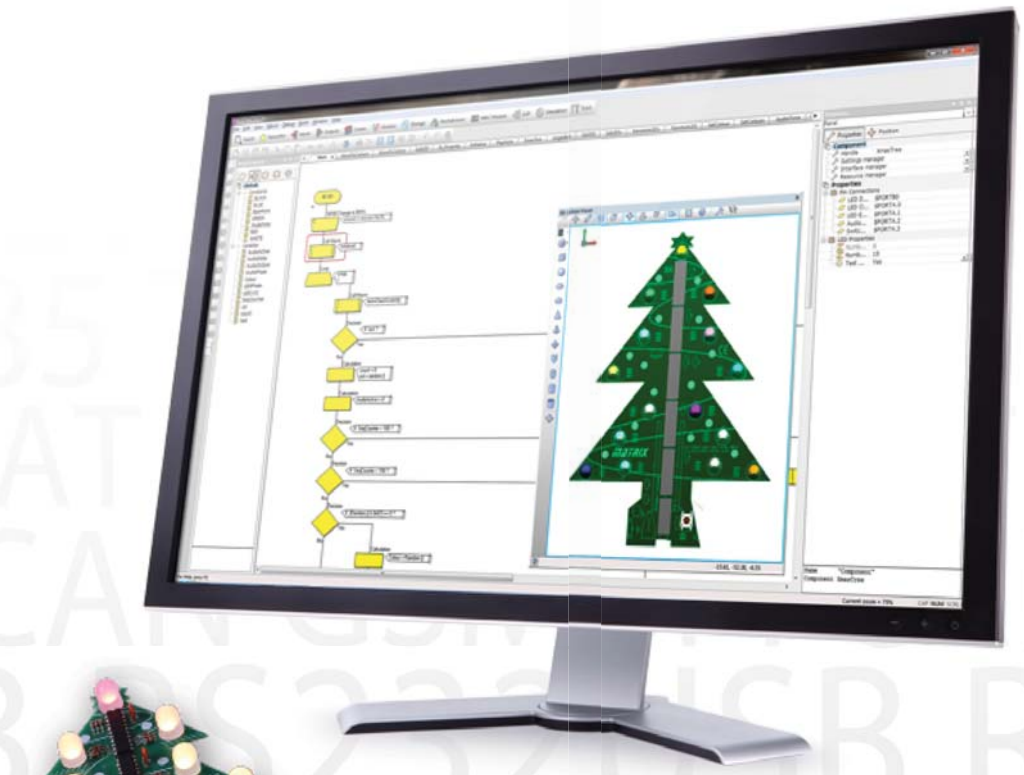
MATRIX

eBLOCKS®

Christmas Tree Project



Introduction

The kit contains all the components you need to create your own Christmas tree board which consists of:

- 15 x individually controlled RGB LEDs
- Audio output with on-board speaker and volume control
- Switch to trigger the audio output or LED animation

The PIC16F88 microcontroller supplied with the kit comes pre-programmed with working firmware which can be used to test the functionality of the board once assembled and provide a working demonstration program.

Tools you will need to complete the board:

- Soldering Iron
- Solder
- Wire cutters
- Multi-meter (optional)

To re-program the microcontroller please refer to the section entitled re-programming the kit.

Kit Contents

Before starting with the building of the kit you should first make sure that all the parts have been supplied.
We have taken the upmost care to ensure that the kits are correct but please contact us if you think there has been an error in the kitting of parts for your tree.

QTY	ITEM
2	Resistor 4K7
31	Resistor 470R
15	Resistor 560R
1	Electrolytic Capacitor 10uF
6	Ceramic Capacitor 100nF
1	Pre-Programmed Microcontroller Brain PIC16F88
1	18-Way DIL Socket
1	5V LDO Regulator
5	Octal Buffer IC
1	Small signal NPN Transistor
15	RGB LED Common Cathode
1	Reverse voltage protection diode
1	Battery Clip With Cable
1	Mini Speaker With Cable
1	Potentiometer 4.7K
1	Shaft for POT
1	Small Square Push Switch
1	MIN TACTILE SWITCH

About this project

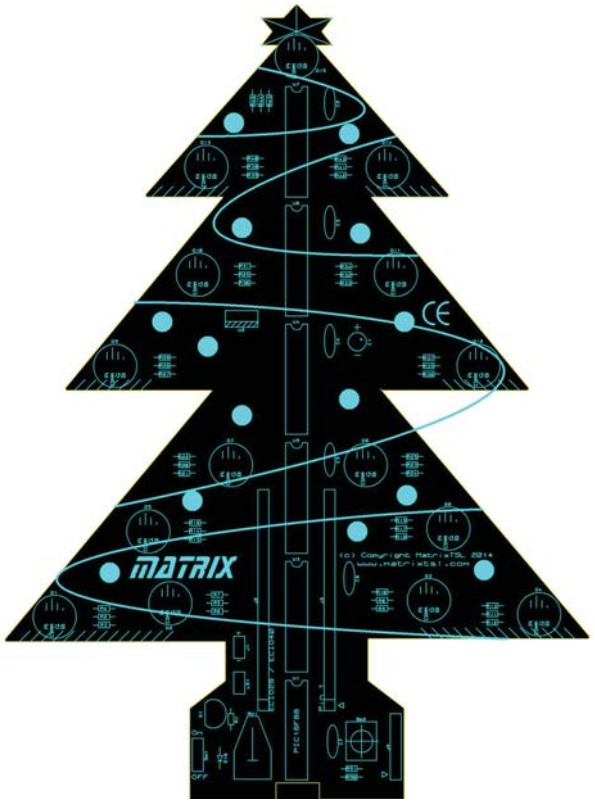
The kit contains a microcontroller brain which is used to drive the various electronic items on the board. The 16F88 microcontroller is capable of processing 2 million 8-bit instructions per second and is plenty powerful enough to drive the electronics.

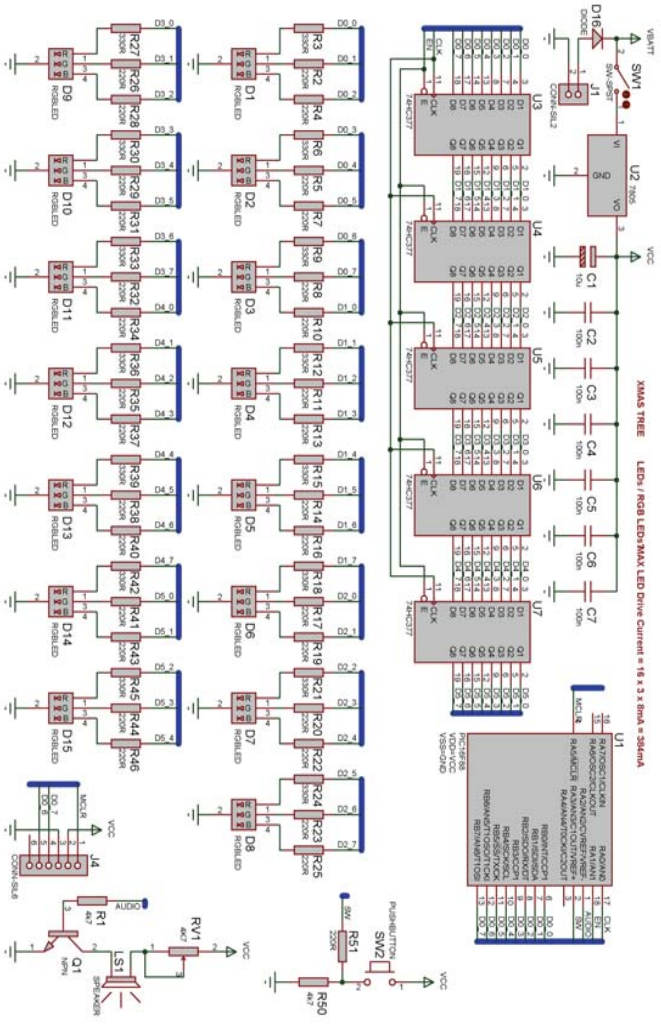
There are 15 RGB LEDs which are controlled by using digital buffers in a daisy chain to provide enough outputs to allow all 15 x 3 individual LEDs to be controlled. Each LED is made up of three internal LEDs which control the Red, Green and Blue colours. By switching the individual LEDs on and off at high speed you can vary the brightness of each LED therefore controlling the overall output colour of each RGB LED.

The LEDs are automatically taken care of as part of a timer interrupt so all the user has to do in their program is change the values in the colour variable for each LED.

The audio output is driven by using a single digital output toggling at a certain frequency to output a tone to the speaker. By using a sequence of monophony tones it is possible to create an audio melody similar to the ring tones found on early mobile phone devices. A NPN transistor is used as a very simple amplifier to take the signal from the microcontroller and boost the current to a suitable level to drive the speaker. This boosted current is then controlled by using the potentiometer which has the effect of altering the output volume from the speaker.

Board layout and Schematic





Building the kit

1. Begin by fitting the 15 x 560R resistors, both legs must be bent to a 90 degree angle before being inserted into the PCB in locations R3, R6, R9, R12, R15, R18, R21, R24, R27, R30, R33, R36, R39, R42 and R45. Solder both ends of the resistor and then cut off any excess wire. The resistor can be fitted either way into the circuit board.

2. Fit the 30 x 470R resistors into positions R2, R4, R5, R7, R8, R10, R11, R13, R14, R16, R17, R19, R20, R22, R23, R25, R26, R28, R29, R31, R32, R34, R35, R37, R38, R40, R41, R43, R44, R46, R51 and solder into position before cutting off the excess wire.

3. Fit the 2 x 4K7 resistors into positions R1 and R50 and solder into position before cutting off the excess wire.

4. Fit the 100nF capacitors into locations C2, C3, C4, C5, C6, C7. These capacitors can also be inserted into the PCB in either orientation. Solder into position before cutting off the excess wire.

5. Fit the 5 x Buffer ICs into locations U3, U4, U5, U6, U7 taking great care to get the orientation of the chip correct to the circuit board. Pin 1 is noted with a square pad compared to all the other pads which are round. After double checking all the orientations are correct solder the chips into place by soldering two of the four corner pins of each chip. Triple check your orientations and make sure the chips are flush to the circuit board before soldering the rest of the pins.

6. Fit the 18-pin socket into location U1 again double checking the orientation and alignment with the PCB.

7. Next the two switches SW1 and SW2 can be inserted and soldered into position.

8. The potentiometer RV1 can be inserted and soldered into position.

9. The LEDs can now be inserted, check that each LED is inserted with the correct orientation otherwise that LED will not work correctly. Compare the legs of each LED with the print on the PCB to check that the longest leg goes into the hole for the cathode marked COMM. Take care whilst soldering the LEDs to try and avoid any potential shorts between pins. Solder into position before cutting off the excess wire.

10. Next the NPN transistor can be fitted into location Q1 after double checking the orientation matches that shown on the circuit board. Again solder into position before cutting off the excess wire.

11. The Diode can be fitted into location D16 taking care to get the line on the diode in the same orientation as the line on the circuit board. Solder into position before cutting off the excess wire.

12. The 10uF Capacitor can be fitted into location C1 taking care to get the negative pin on the capacitor in the same orientation as the negative hole on the circuit board. Solder into position before cutting off the excess wire.

13. Bend the legs of the 7805 voltage regulator so they form a right angle with the regulator. Then fit the regulator into position U2 and solder into position before cutting off the excess wire.

14. Solder the wires from the speaker into position LS1 before hot gluing the speaker to the front or back of the PCB depending on if you want it to be visible or not.

15. Solder the wires from the PP3 battery clip into position J1 taking care to place the red wire in the hole marked "+" and the black wire into the hole marked "-".

16. Finally push the pre-programmed 16F88 microcontroller into it's socket at position U1 on the PCB ensuring it is in the correct orientation with pin 1 going to the square pin on the PCB.

Testing the kit

First double check that there are no shorts between solder joints on the board specifically around pins that are close together like the LEDs. Powering up the board while there are shorts may risk damaging the hardware so use a multi-meter to double check if you're not sure. Shorts can be removed by heating up the solder joint and using a solder sucker or some multi-strand copper wire to pull off any excess solder.

Once you are fairly certain there are no shorts and that the build instructions have been completed ensure that the on/off switch is in the off position and connect up a 9V PP3 battery to the battery clip. Briefly move the switch to the on position and all 15 LEDs should light up white.

If one or more of the LEDs do not show the correct colour then move the switch back to the off position and double check your solder joints for the LEDs that were showing problems.
After a few seconds of being powered the LEDs will start to display random colours.

Once you have all the LEDs working as they should you can switch on the board and press the push button switch to test the audio interface. Once the switch has been pressed a jingle bells tune will play. If you cannot hear any audio then try adjusting the potentiometer clockwise until the audio is audible. Pressing the push button again will re-start the audio. If no audio can be heard then move the on/off switch back to the off position and double check your solder joints around the audio components: RV1, Q1, R1 and LS1.

Reprogramming the kit

Should you wish to modify the firmware and create your own custom animations and audio then you will need to re-program the microcontroller. The board cannot directly re-program the microcontroller brain so there are a number of different ways to do this:

- Use an EB006 or other PIC programmer to re-program the supplied 16F88 device. The chip will need to be removed from the XMAS Tree PCB and inserted into the programmer each time the firmware is changed.

- Remove the 16F88 from its socket completely and use an ECIO device which contains USB re-programming circuitry. All ECIO devices should be compatible with the Xmas Tree PCB though you will likely need to purchase sockets to allow the ECIO to be removed from the PCB in future should you wish to use it in another project.

- Use a PICkit2 / PICkit3 to re-program via the ICSP socket on the board. This allows you to re-program the firmware without having to remove the chip from its socket.

The demo tree firmware was created using the PIC chip pack for Flowcode 6. If you don't already own a copy of Flowcode 6 then don't worry the firmware will also work with the 30-day trial which can be obtained by downloading the free version and then signing up for a Matrix user account. Once Flowcode is installed on your system use the username and password from your Matrix account to activate your 30-day trial.

The demo firmware source is available from the HPTREE product page on the MatrixTSL.com website. It is recommended to start with this file when creating new firmware as it contains lots of tips, tricks, comments and routines designed to help simplify the software and help with creating your own custom animations and audio jingles.

Matrix Technology Solutions Ltd.
The Factory
33 Gibbet Street
Halifax, HX1 5BA, UK

t: +44 (0)1422 252380
e: sales@matrixtsl.com

www.matrixtsl.com

HPTREE-30-1