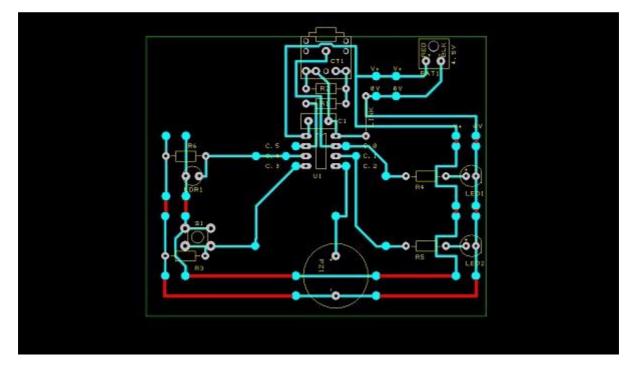# Using PICAXE Create within DesignSpark PCB

Introduction

To use PICAXE Create within DesignSpark PCB you must:

1) Download and install the DesignSpark PCB software
2) Download and install the free PICAXE Create library, supplied in a zip format file from www.picaxe.com/DesignSpark

The PICAXE Create zip file contains two separate system features required to use the PICAXE Create system with DesignSpark PCB:

1) The first feature is the actual component libraries. These hold all the component, schematic symbols and PCB footprint symbols that can be used to draw circuits. These libraries contain all the commonly used educational components such as resistors, LEDs, LDRs, PICAXE microcontrollers etc. These files are DesignSpark PCB 'library' format files.
2) The second feature is the Create Modules themselves – the sections of PCBs that can be merged together to form PICAXE Create projects. These modules are in DesignSpark PCB file format.

Note that each module is a completely separate PCB file. Therefore to merge them together it is necessary to open each module (using File>Open) and then cut/paste the multiple sections of PCB into one master PCB file. Within DesignSpark PCB this is very simple to do, simply drag out a selection box around the section of PCB and use Edit>Copy menu to copy it.

Library Installation Instructions

1. Start DesignSpark PCB and select the File > Libraries menu
2. Select the last tab, Folders, and note the default Folder path which is displayed in the top box. It will typically be something like

   C:\Users\Public\Documents\DesignSpark PCB\Library

3. Exit DesignSpark PCB
4. Create a new Folder with the same path, but ending \PICAXE Create instead of \Library e.g.

   C:\Users\Public\Documents\DesignSpark PCB\PICAXE Create

5. Extract all the files from the downloaded zip file into this new '...\PICAXE Create' folder.
6. Restart DesignSpark PCB and select the File > Libraries menu.
7. Select the last tab, Folders, and click the 'Add' button. Click the 'browse' button and then select this new '... \PICAXE Create' folder. Click Open, then OK.
8. You should now have two folders listed in the top box. Select the '...\PICAXE Create' folder and click the 'Up' button to ensure it is the top item in the list. This is important as it then gives this folder 'priority' over the original folder.
9. *Optional* – Select the original '...\Library' folder and uncheck the 'Folder Enabled' box. This will then prevent DesignSpark PCB showing the original libraries contained in this folder, or in other words the DesignSpark PCB library system will now become much simpler as *only* the PICAXE Create components will be displayed. This is recommended for educational use.
10. Click Apply at the bottom of the screen.

The library folder system should now be setup as shown in this screenshot. Now exit and restart DesignSpark PCB to initialise this new library structure.

Note for network administrators, the folder settings are saved in the Registry keys:

HKEY_CURRENT_USER\Software\DesignSpark\DesignSpark PCB\Libraries\Directories\Dirs

HKEY_CURRENT_USER\Software\DesignSpark\DesignSpark PCB\DisabledFolders

Design Process

The design process is very straight forward, simply first select the 'building blocks' of your circuit from the Circuit Creator modules, for example:
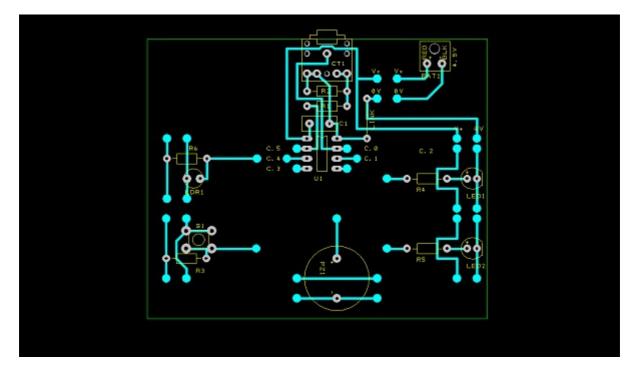
- PICAXE motherboard (in this example we are using the 08M2)
- Power supply type (4.5V battery box)
- Output Components (two LEDs and a piezo sounder)
- Input Components (switch and LDR)

Step by Step Guide

Step 1

Each of these modules is provided as a separate file, so the first task is to simply open the PCB module files and cut and paste the different module sections onto the same PCB design. The easiest way to do this is to open the motherboard module and save it with a new name to become your new project file. Then open the input/output modules one by one and copy and paste them on to the master project file.

Note whole modules can be rotated 90 degrees if required (as shown by the piezo module below, which has been rotated to fit on the bottom of the PCB). Use the 'R' keyboard key to rotate the selected area.
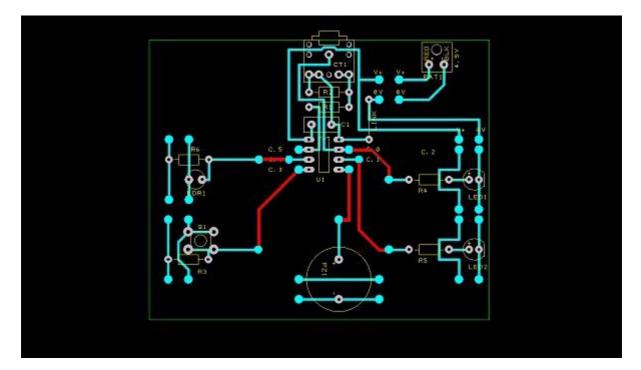

Normally you would then reduce the size of the PCB by moving the components closer together where possible, however for clarity in these pictures we have left all components in their original positions.
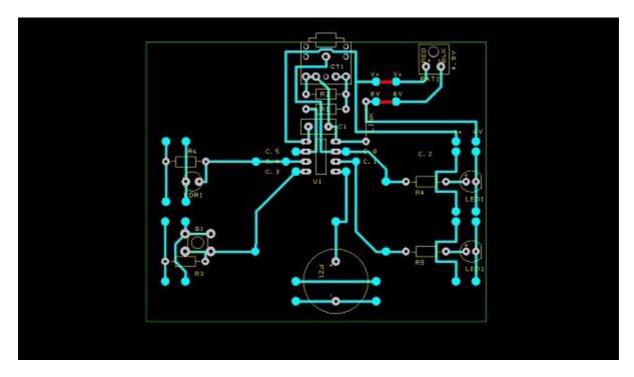
Step 2

Now we need to use the 'add track' tool to connect each of the 5 i/o modules to the correct PICAXE-08M2 pin. Simply click on the dot at the microcontroller pin to start a new track and then draw in the track as required. When you click on the connection dot of the module the track will automatically finish and connect the modules together.

The tracks are shown in red here to illustrate where the added tracks are but they will be given the normal track colour when you actually add them.
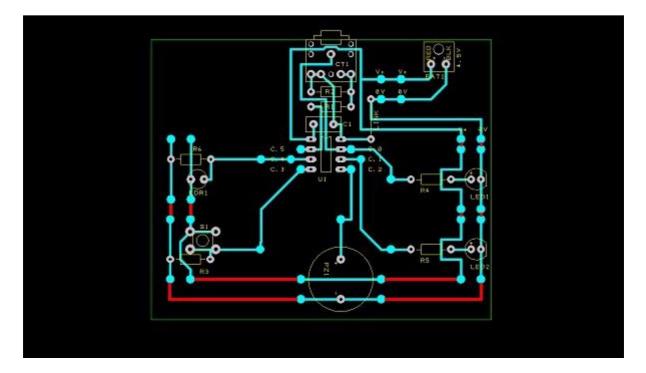
Step 3

The next task is to draw two tracks to connect the selected power supply module to the motherboard (see top right of the picture).

Step 4

The last task is to then connect the two parallel power rails (V+ and 0V) all the way around the board. Make sure all the modules are connected together (note the four short tracks added on the right hand side to link the supply to LED1 and then from LED1 to LED2). Your design is then complete!

Outputting the design

Your design can now be:

- displayed as a '3D image' to act as a reference during assembly
- exported as PDF or bitmap image for the student's portfolio
- printed for photo-etching masks
- used to export a Bill of Materials
- converted to Gerber files for commercial production. You could even use isolation routing software such as CopperCAM to manufacture the PCB on a milling machine.