

Project Setup: A Tutorial on Proteus and CCS C

December 7, 2018

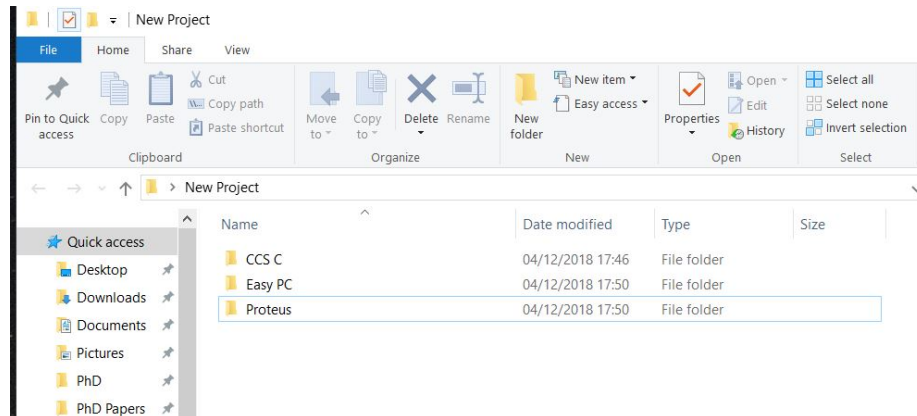
1 Introduction

The Proteus Design Suite by Labcenter is a propriatry software tool suite used primarily for electronic design. Combined with the CCS C compiler allows for impressive design and simulation capabilities utziliing microcontrollers. This document provides a step-by-step tutorial for the initial set-up of a new project to get you started.

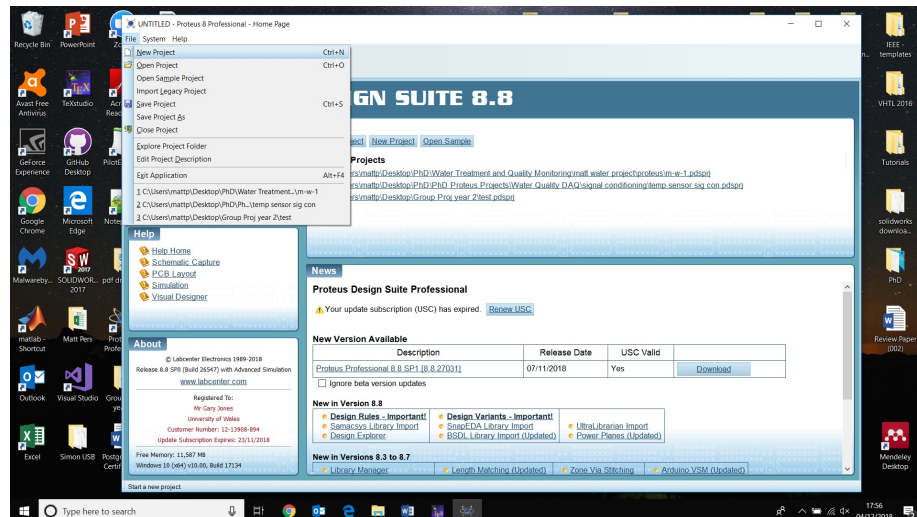
1) To begin, start by creating a new folder on your desktop and label it 'new project'.



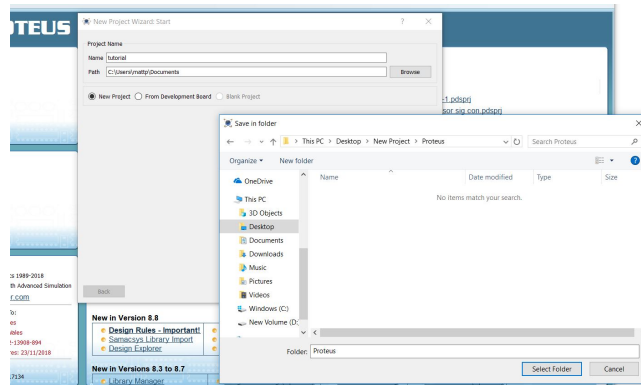
2) Within your 'new project' folder, create an additional 3 folders and label them CCS C, Easy PC, and Proteus respectively.



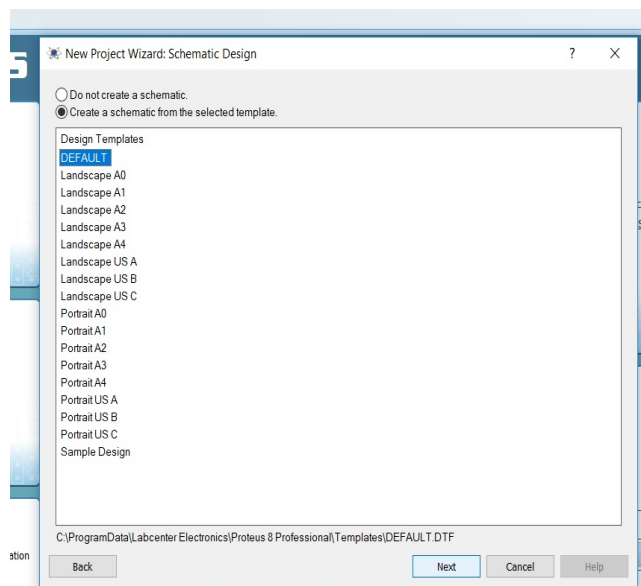
3) Now that you've setup your folder, you can begin working on Proteus. Open Proteus, click File, and New Project.



4) Provide an appropriate name for your project, and ensure you save it into the Proteus folder created in step 1.

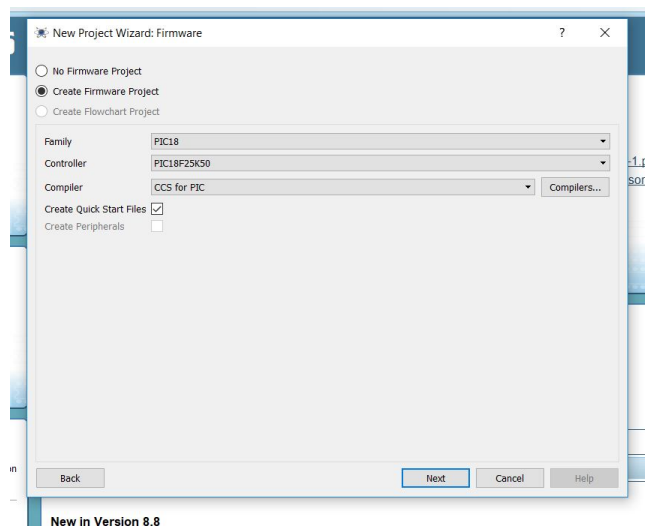


5) Ensure you use a default template if you're unsure (dimensions of your schematic can always be adjusted later).

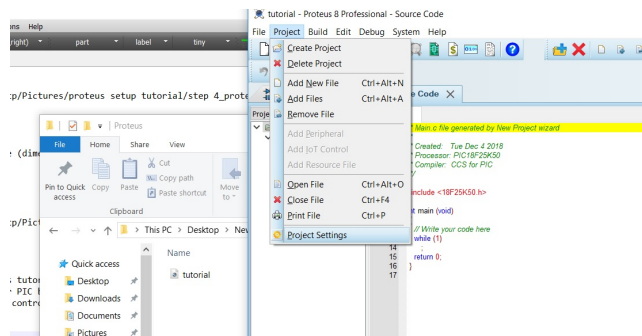


6) Select the 'create with firmware' option. For this tutorial I'm using the popular 25k50 micro from the PIC18 family. First, select the 'family' that your PIC belongs from the initial drop down menu. Secondly, find your device in the drop down menu for 'controller'. Finally, ensure you select 'CCS C for PIC' in the compiler drop down menu.

Click finish to finalise your project.

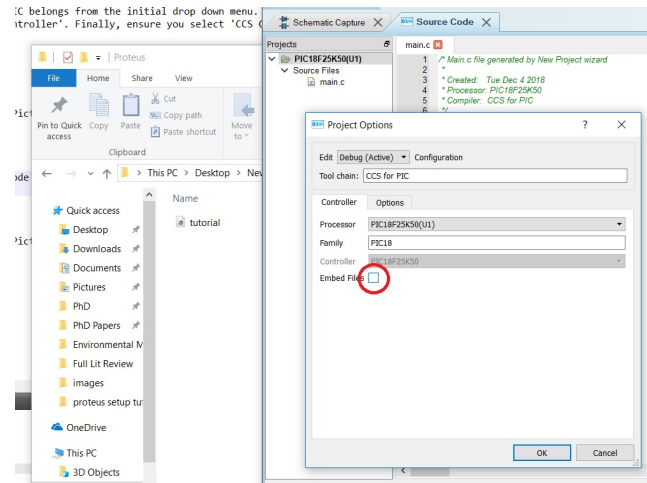


7) Now that you are within Proteus, under the 'Source Code' tab, click Project on the taskbar and select Project Settings from the drop down menu.

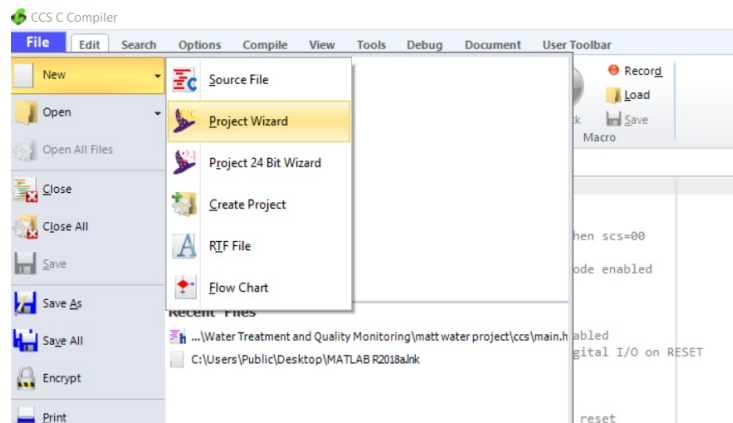


8) Untick the option for Embedded Files. You will notice that a new folder has appeared in your New Project > Proteus folder named after the microcontroller of your selection, in this case PIC18F25k50.

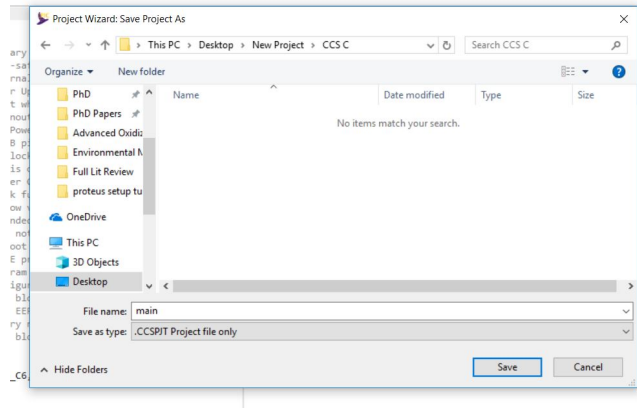
Note: Before you continue with your setup, it is highly recommended you complete your schematic design within Proteus. Doing this before you continue onto CCS C will inform you of what pins are being used and for what operation.



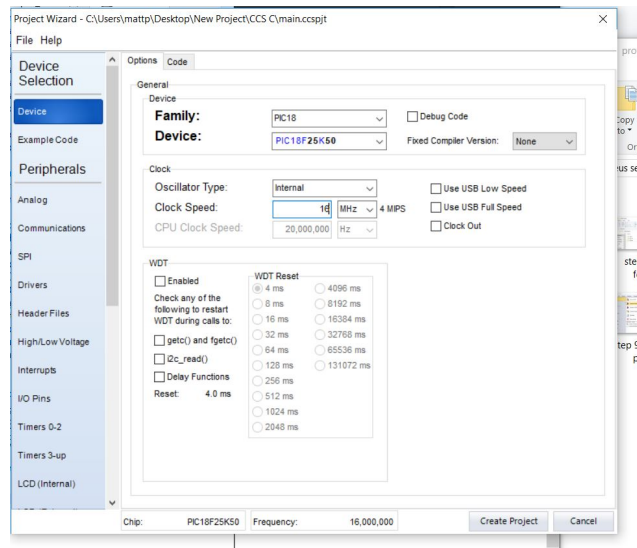
9) Minimise Proteus, and open the CCS C compiler. Click File from the taskbar, and select New > Project Wizard from the drop down menu.



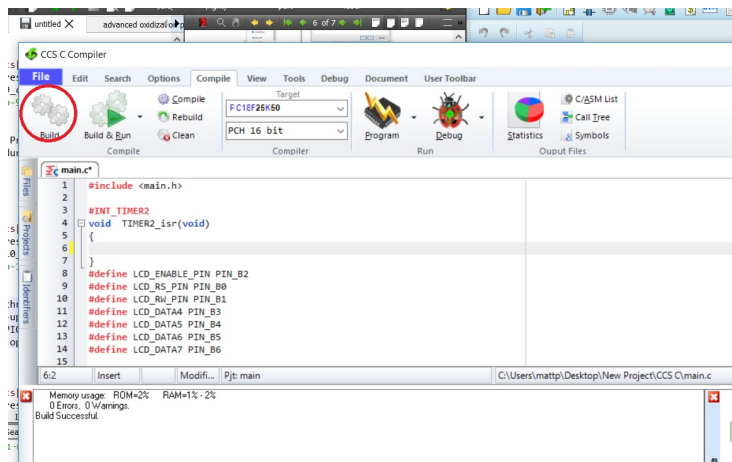
10) Save your new Project into the CCS C folder we created during step 1, do not rename this file.



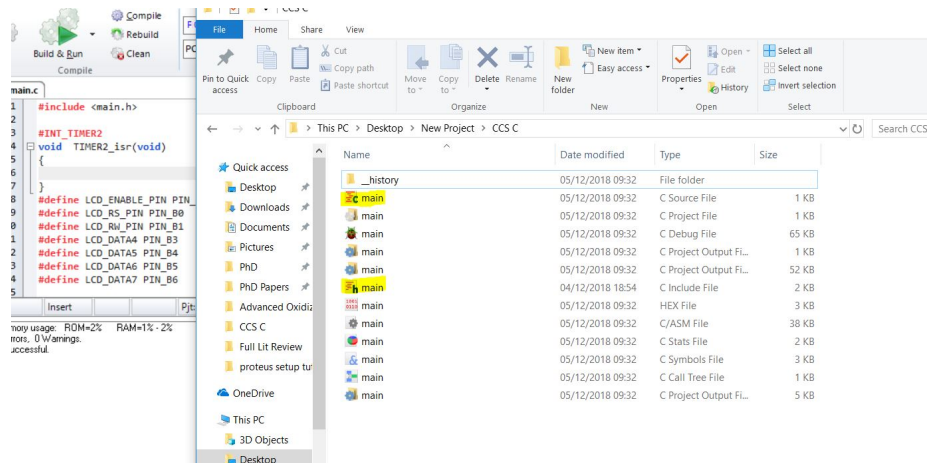
11) Carefully go through each of the options available, and set-up the features you wish to utilise with your PIC. Once finished, click the create project option.



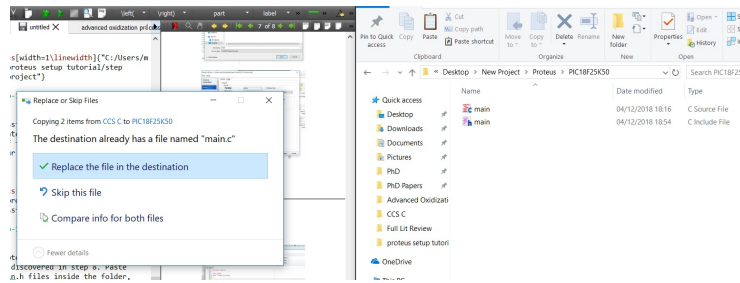
12) Once you've finished the project wizard, CCS C will open your main.c file. Make a change to the file (enter spacebar on an unused line) and build the project.



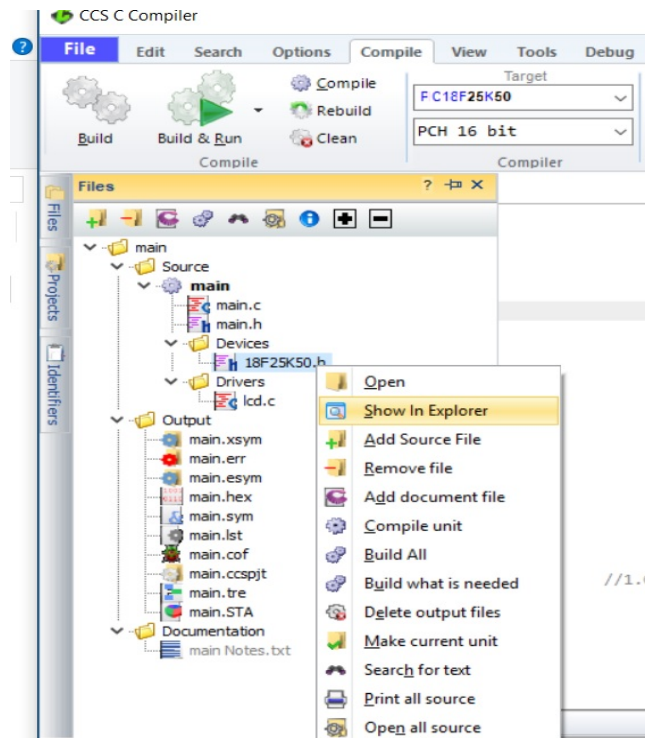
13) After a successful build, go to your CCS C folder that we created during step 1. You should now see a number of files within that folder. Select and copy your main.c and main.h files.



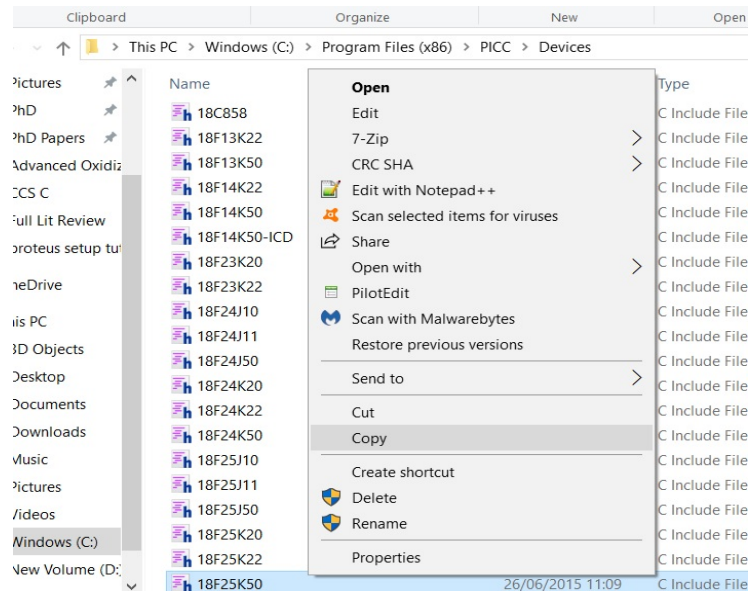
14) Go to your Proteus folder, then select the PIC folder that we discovered in step 8. Paste your main.c and main.h files inside the folder, overwriting your previous main.c file.



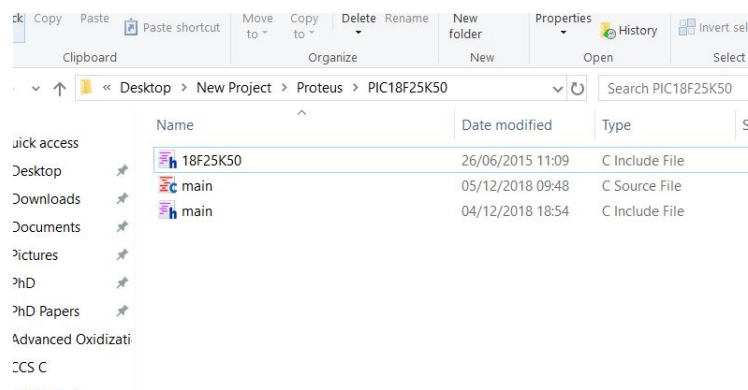
15) Return to your CCS C project. On the left hand side, click Files on the taskbar. Under devices, right click on the .h file of your PIC and select show in explorer.



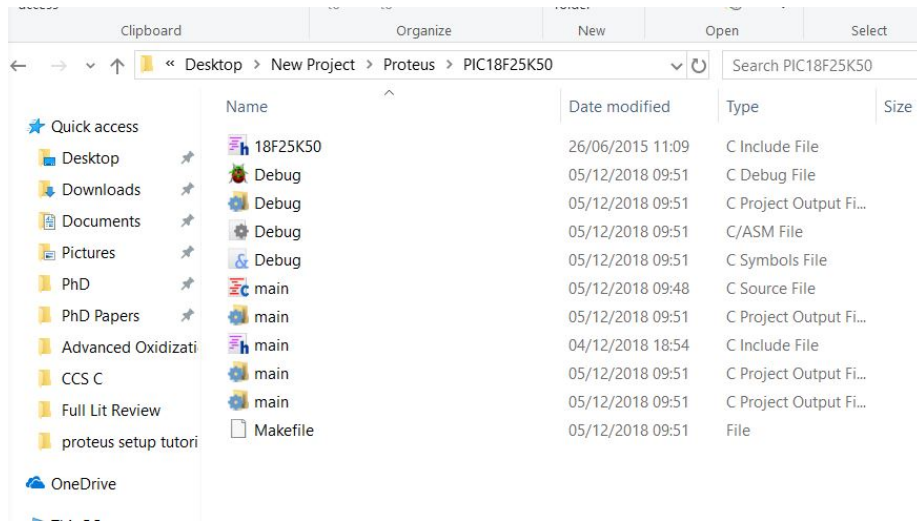
16) This will take you to your PIC header file within CCS C's devices folder. Select and copy the header file.



17) Paste your header file into the Proteus > PIC folder as we did with main.c and .h in step 14. **Note;** you will need to do this for any devices you include in your CCS C project wizard ”(i.e. if you’re using an external LCD, you need to include the devices header file in this folder)”.



18) Return to your Proteus project. Go to your source code tab, make a change to the code (a space bar on an empty line), select build from the taskbar and click rebuild project. This will create the remaining files required within your PIC folder.



You are now ready to begin working on your project, enjoy!