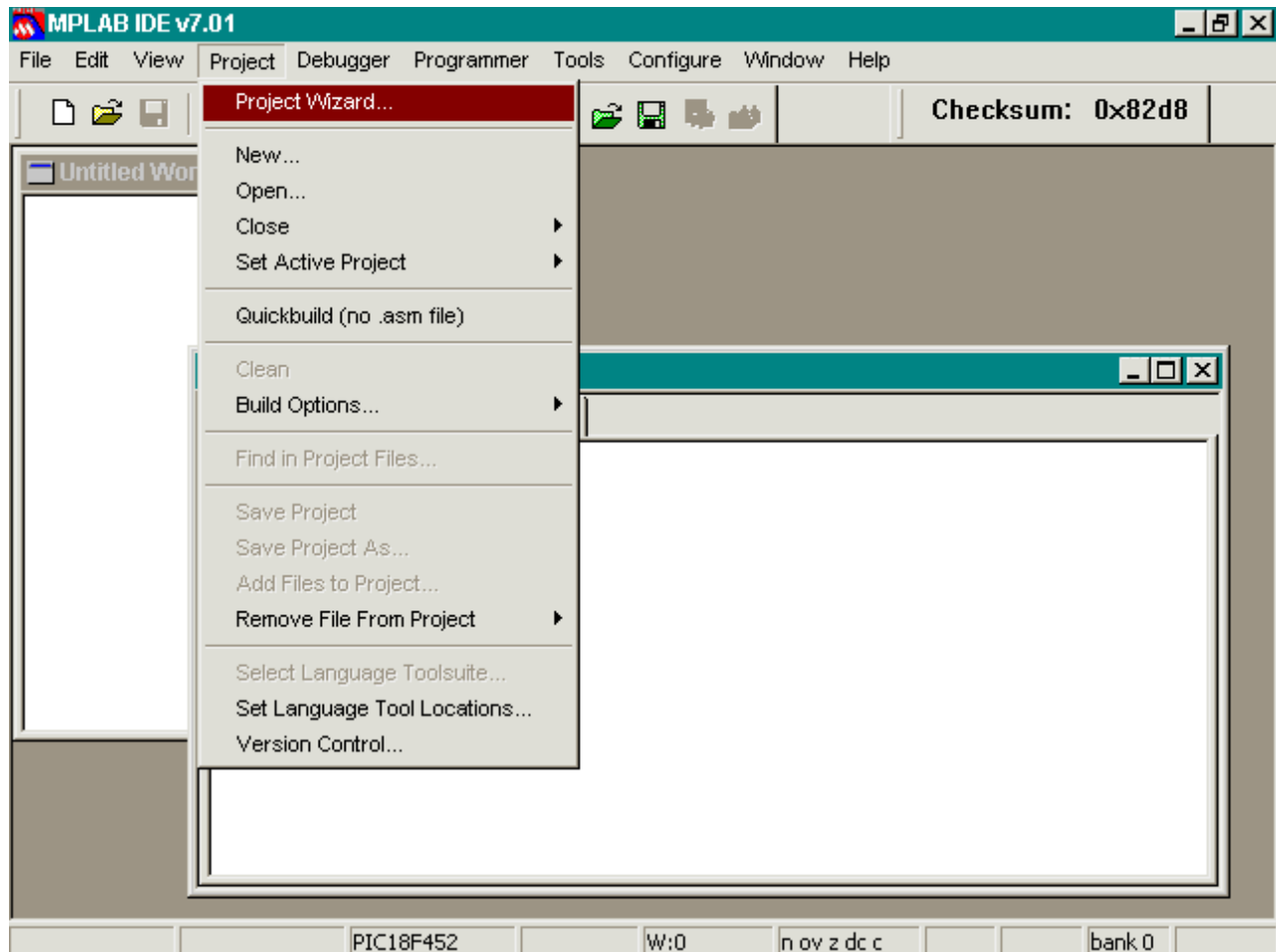


# MPLAB<sup>®</sup> Integration Help

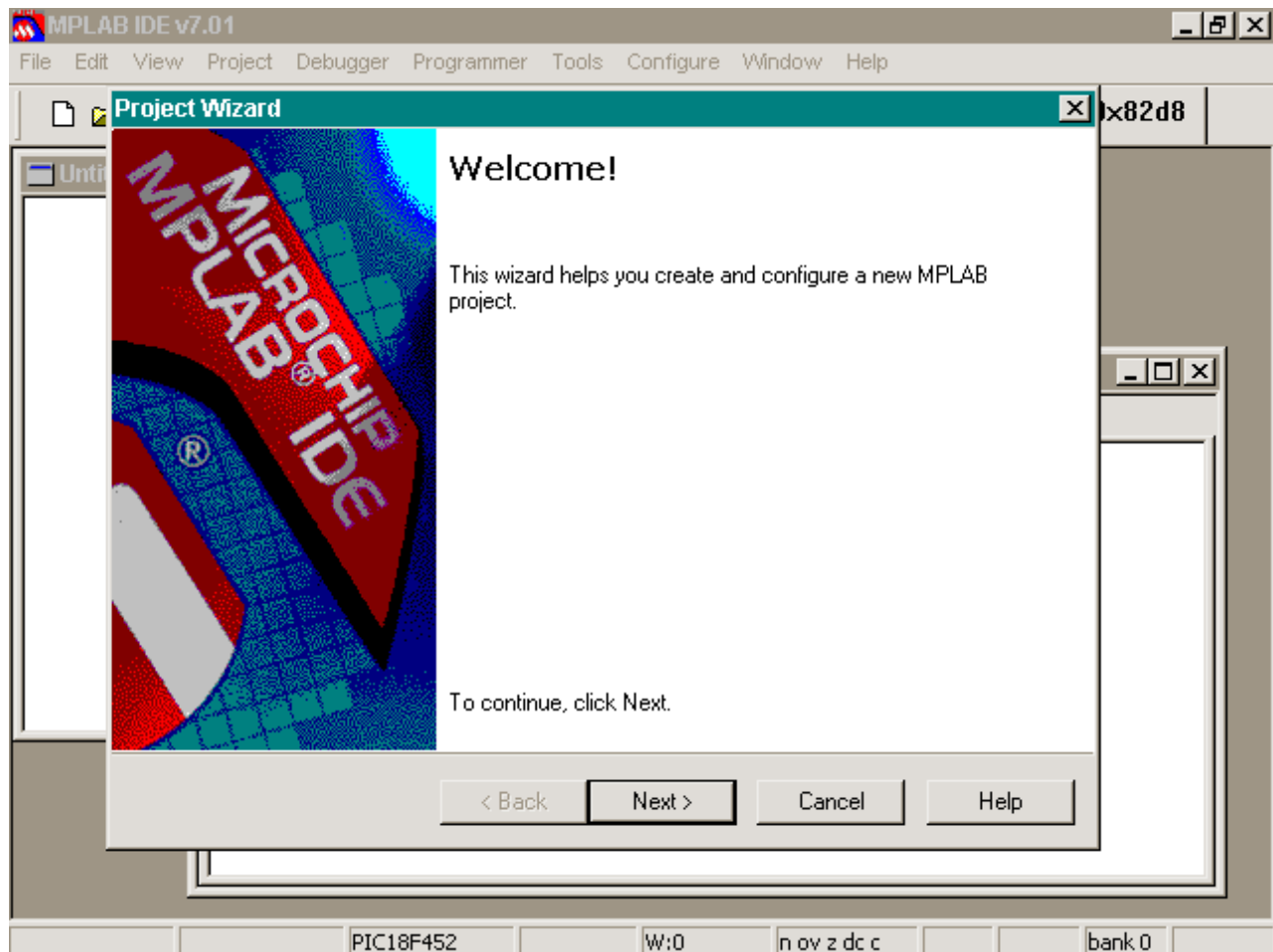
## How do I get CCS to work inside MPLAB<sup>®</sup> IDE?

This FAQ will show how to create a simple project for the `ex_sqw.c` sample provided with the compiler. After you have success creating this project then you should not have problems creating projects with your own code.

1. Make sure CCS is installed correctly on your PC. Try compiling an example program provided with the compiler to verify it works.
2. Verify that MPLAB 6.xx or 7.xx is installed.
3. If you haven't done so already, download and install the [CCS MPLAB 6.xx/7.xx/8.xx plugin](#)
4. Start MPLAB<sup>®</sup>. Start MPLAB's Project Wizard by selecting Project->Project Wizard:

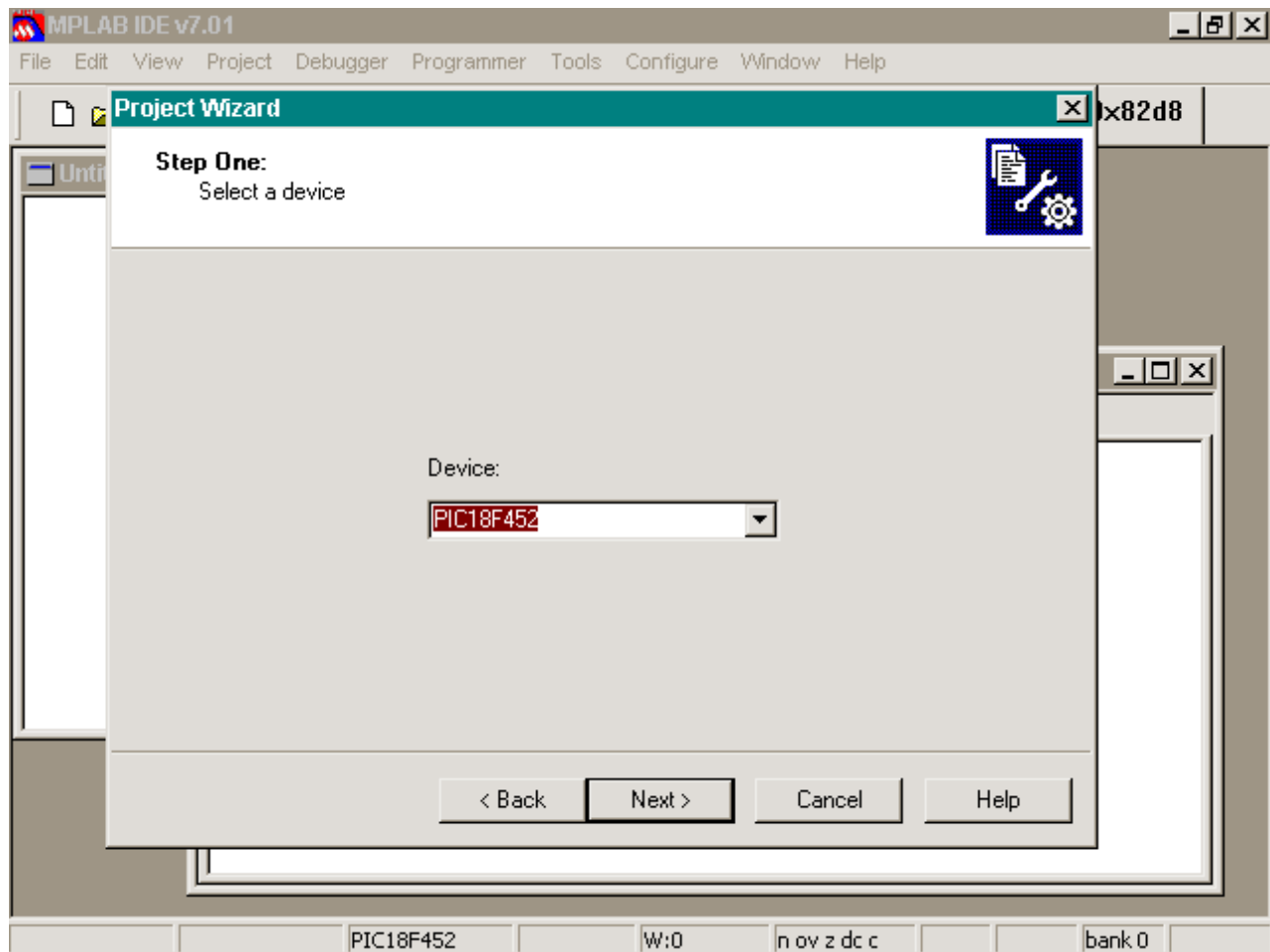


You will then be greeted with the Wizard splash screen:



Press NEXT to continue

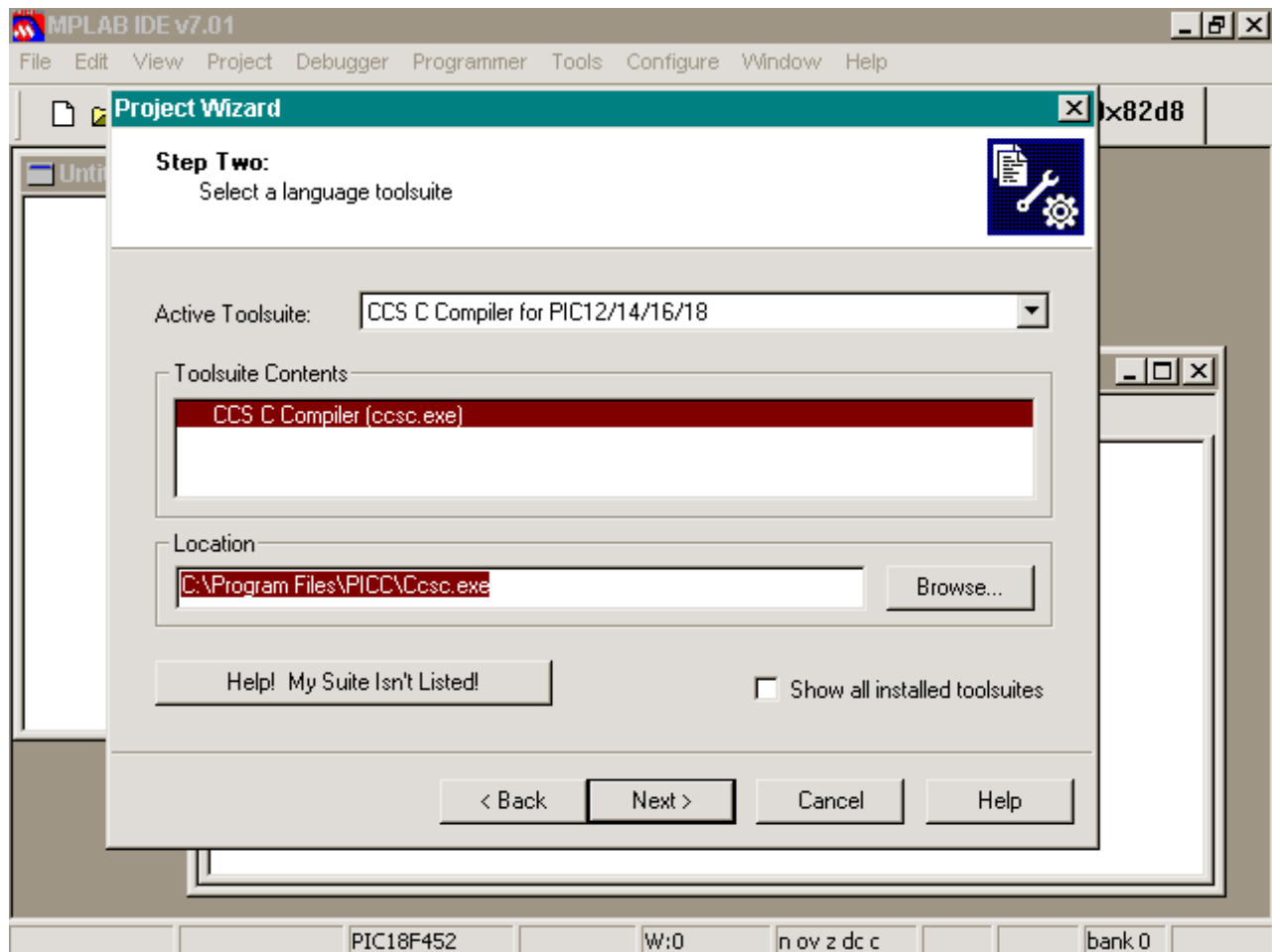
5. Select the device you wish to compile for and press NEXT to continue:



- The next screen of the wizard will ask you which build suite you wish to use for your project. Select the tool suite labeled **CCS C Compiler for PIC12/14/16/18**. If you do not see this option then the [CCS MPLAB 6.xx/7.xx plugin](#) was not installed correctly.

**DO NOT SELECT THE TOOL SUITE LABELED *CCS C Compiler*. THIS IS A LEGACY OF MPLAB5 AND DOES NOT WORK CORRECTLY IN MPLAB6 AND MPLAB7**

Your tool-suite select screen should look like this:

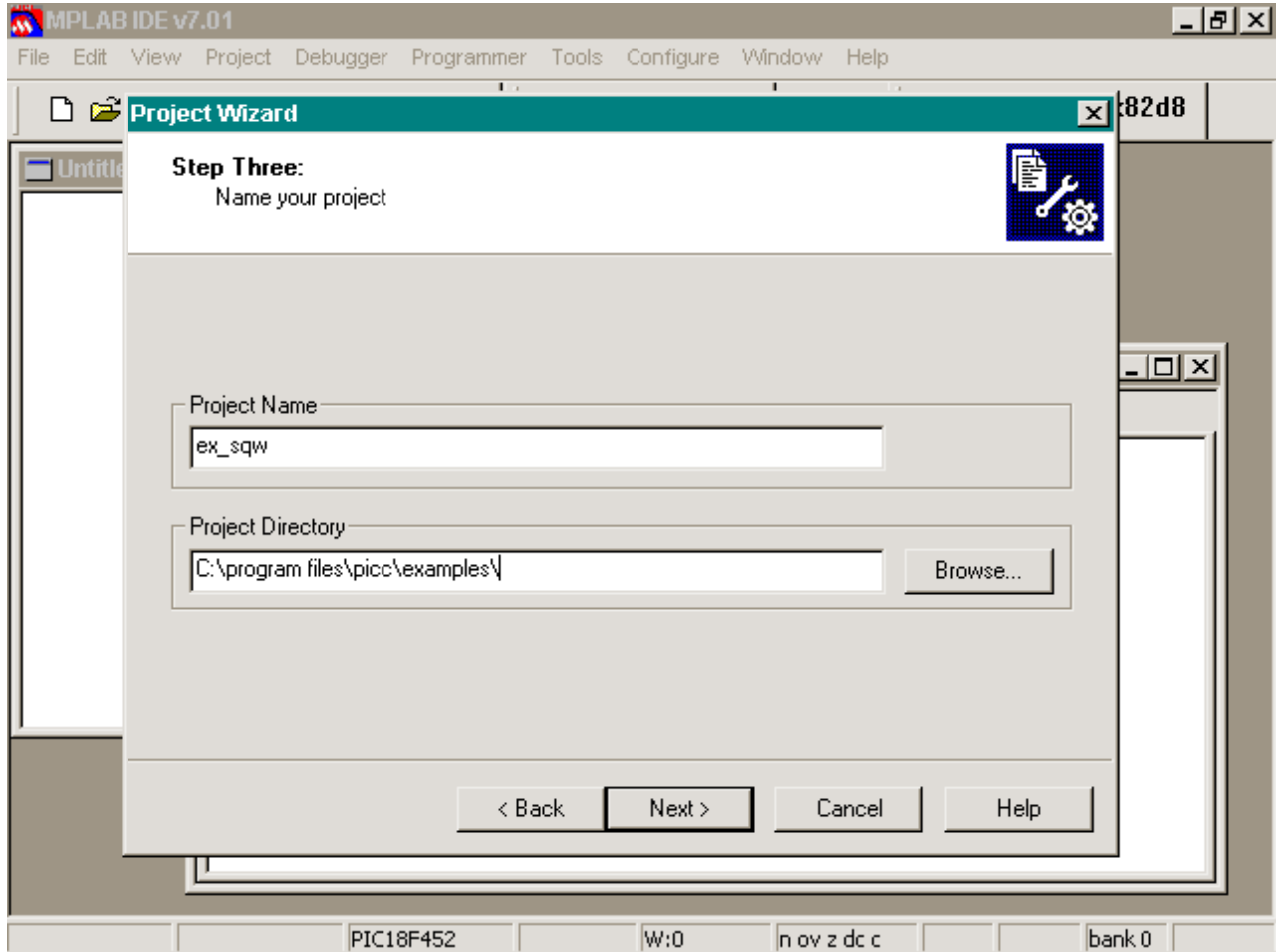


Verify the location field is where you installed the compiler. Press NEXT to continue.

- 7. The next step of the New Project Wizard will ask you what to call the project and where to save the associated project files. Since we are creating a project of `ex_sqw.c` then we shall call the Project Name `ex_sqw` and save the project files to the examples folder `C:\program files\picc\examples`.

This is assuming you installed the compiler to the default location. If you installed it into a different location then please put that location there

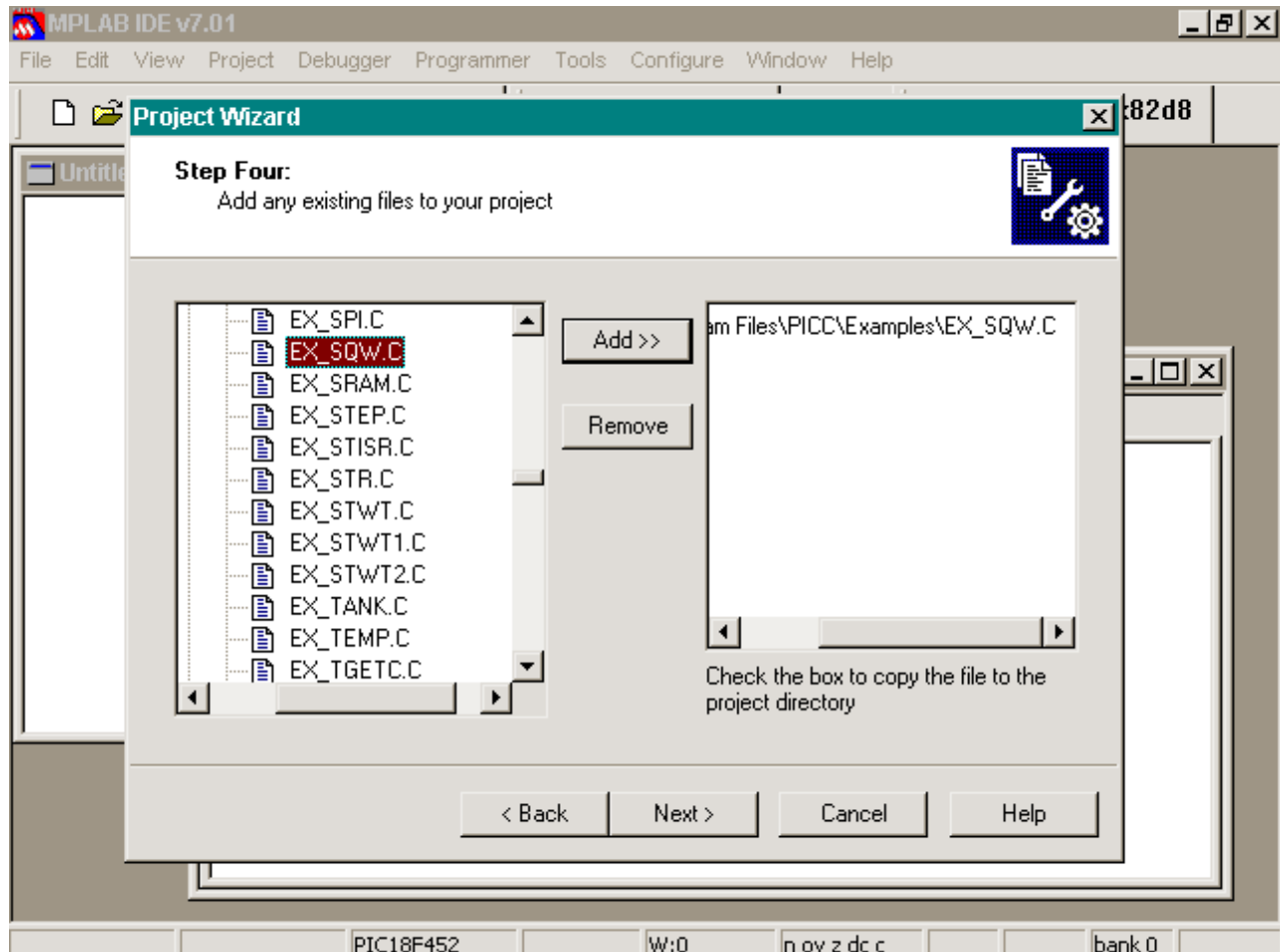
When done your screen should look like this:



Press NEXT to continue

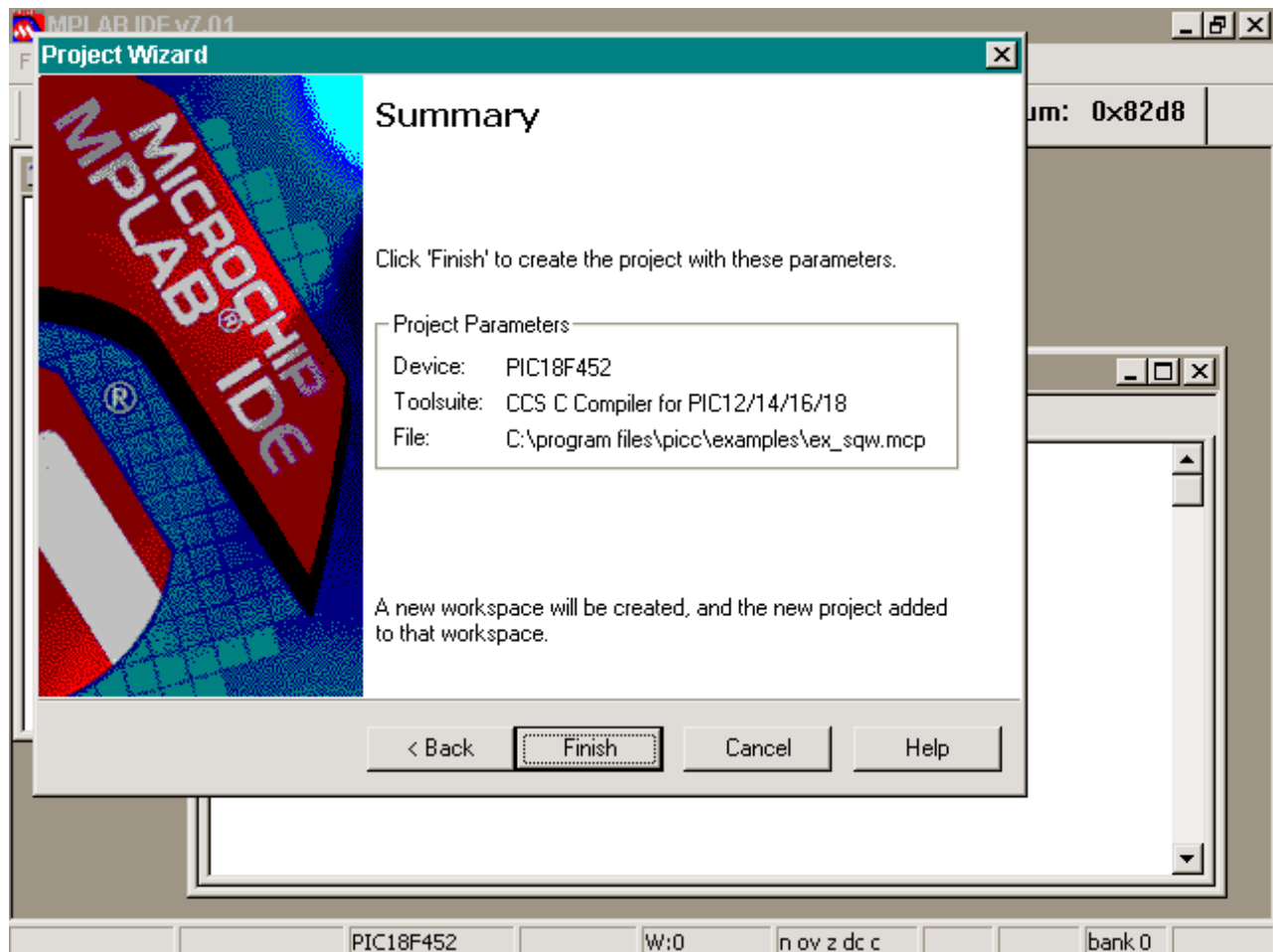
- The next step of the New Project Wizard asks you what files you wish to include with the project. This will be the .C file containing your main(), or the .C file that you call from CCSC.EXE or PCW to compile. In our example this is ex\_sqw.c - so open C:\program files\picc\examples, highlight ex\_sqw.c and press the *Add >>* button.

When done your screen should look like this:



Press NEXT to continue

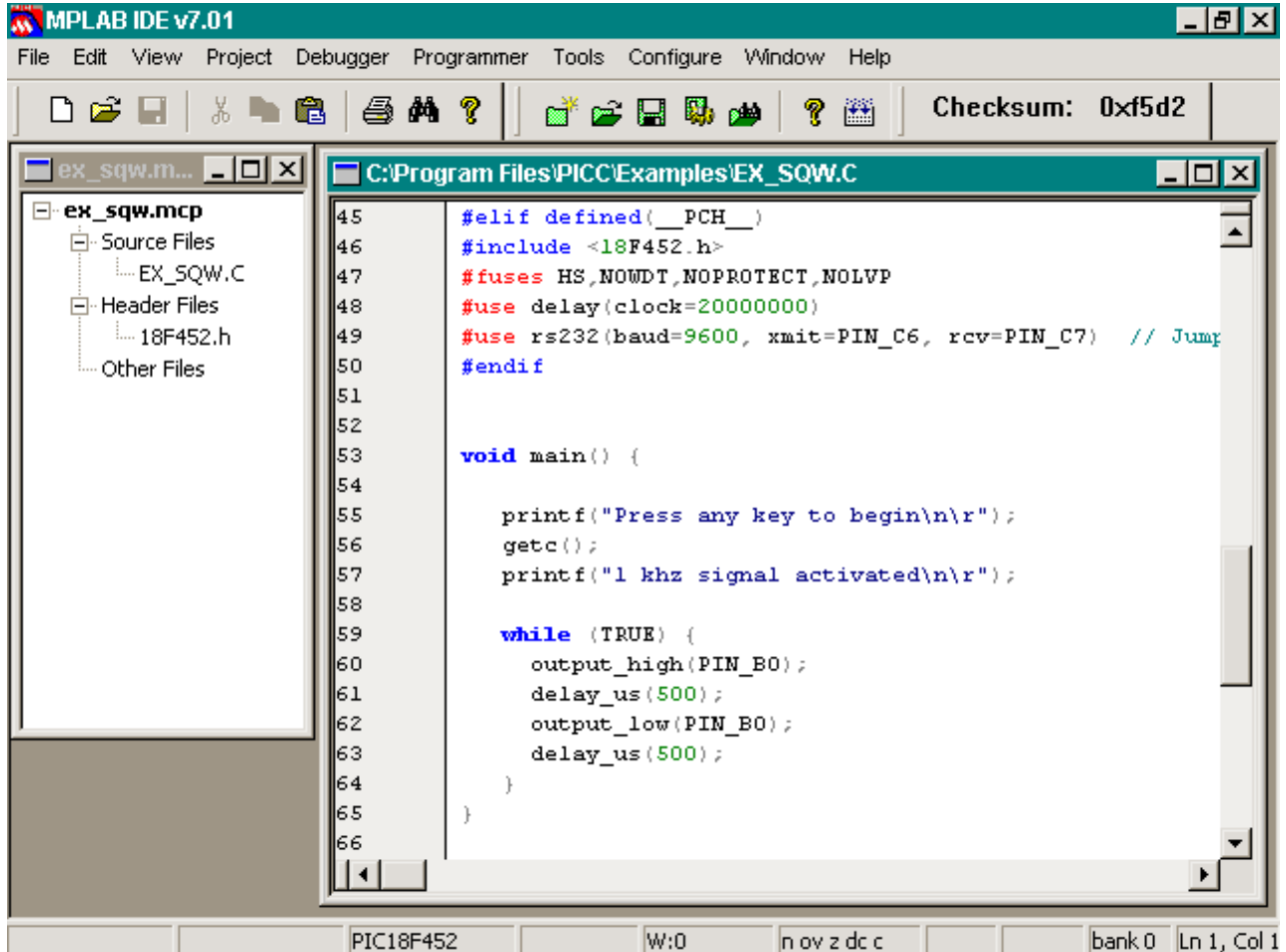
9. MPLAB Project Wizard will now give you a summary:



Press FINISH to continue

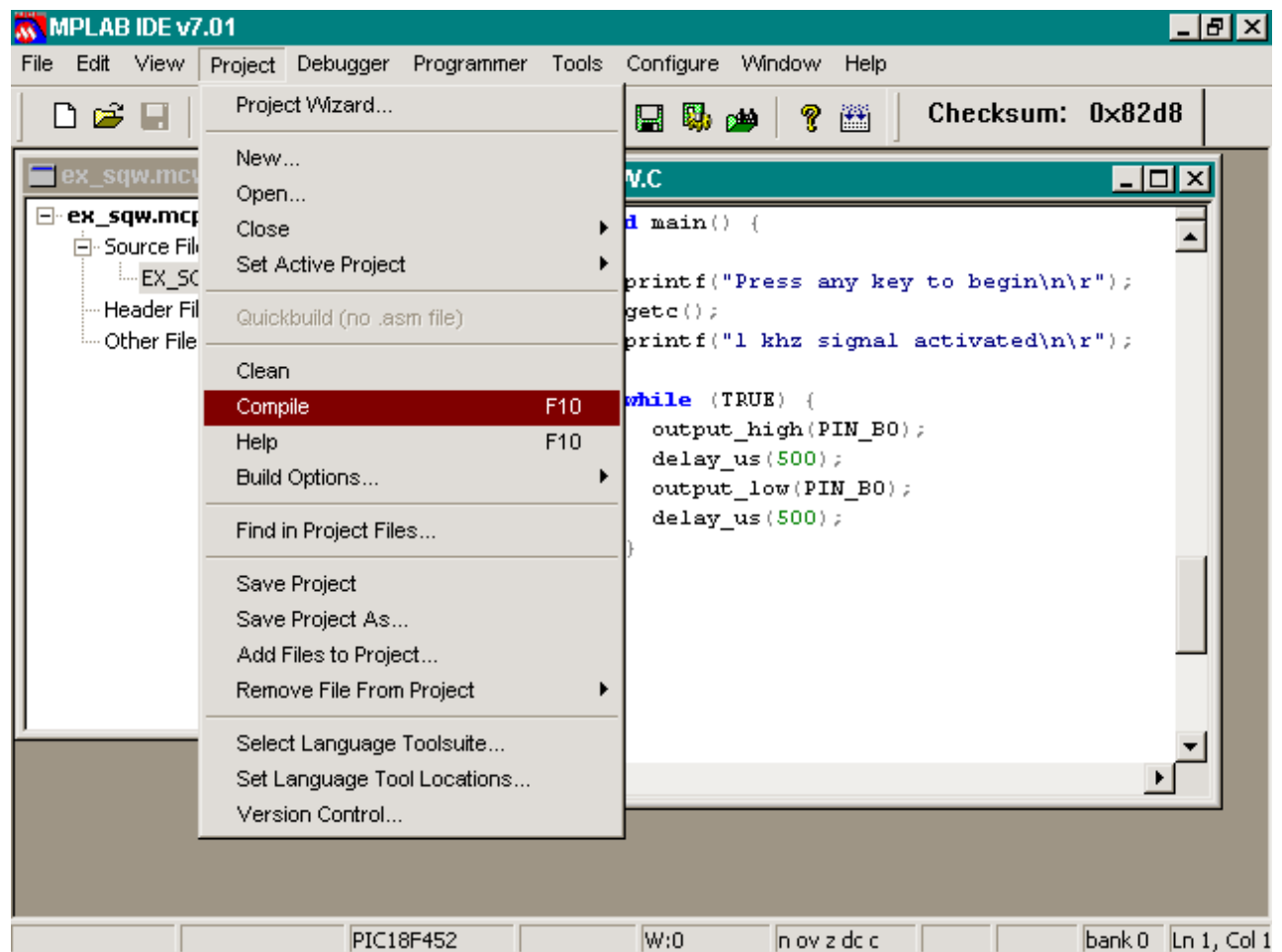
10. The Project Wizard will now close and MPLAB® will be back to normal mode. In the Project Window on the left side you will see EX\_SQW.C listed under *Source Files*. Double-clicking on this file will open EX\_SQW.C with the MPLAB Editor.

Your screen should look like this:

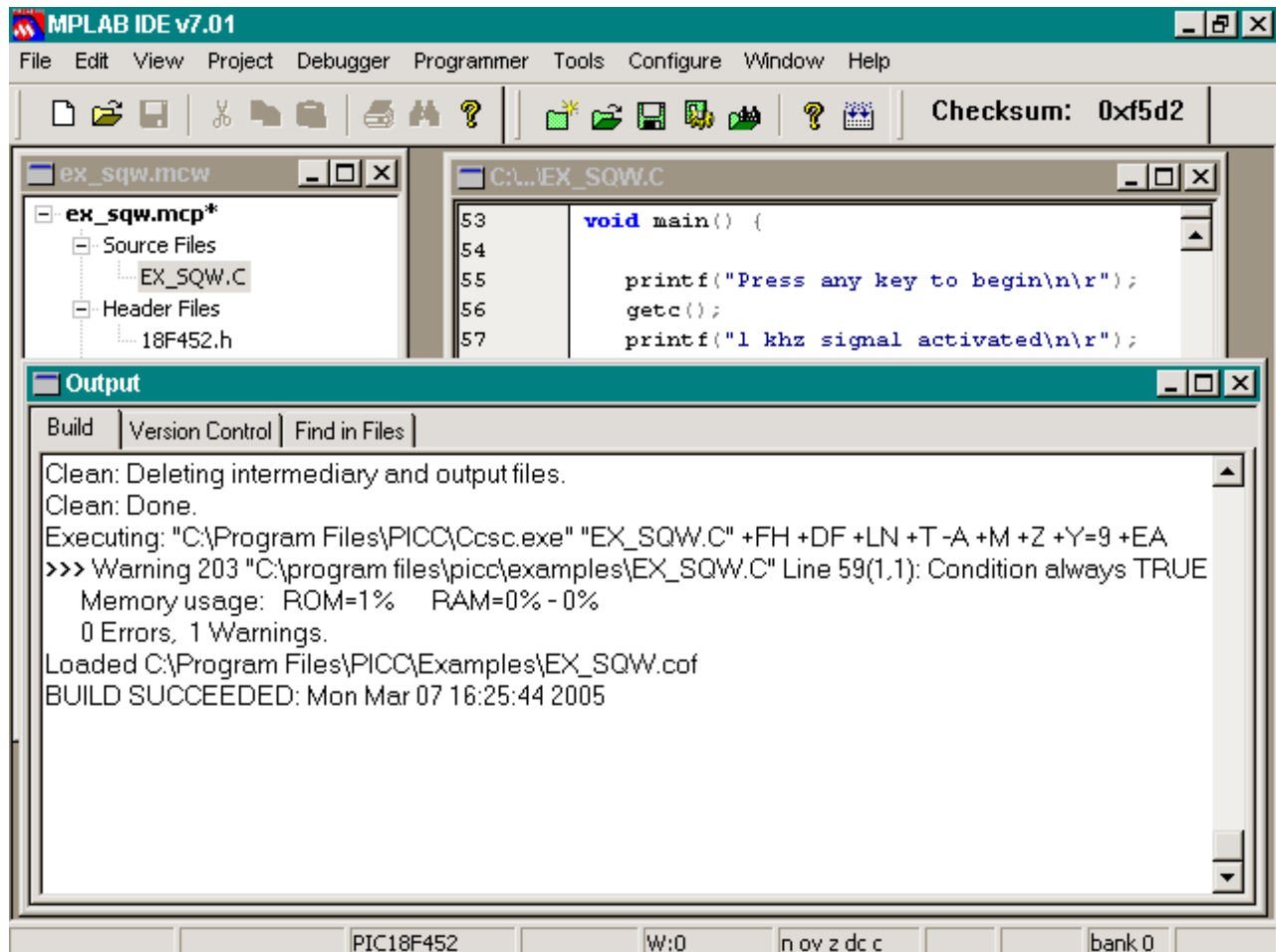




11. Now that the project is completed, you can compile it by selecting Project->Compile:



OR you could press F10 to compile your project. The output window will display results of the compile, and if successful it will say BUILD SUCCESSFULL:



If there were any errors with your compile it would display the errors in the output window.

## Some More Tips

- If creating your own project, in Step 7 specify your own project name (usually something descriptive to help you find your project) and specify the directory where you want to save your own files. (If the directory doesn't exist MPLAB<sup>®</sup> will create it.) In Step 8 you can either use the *Add >>* button to include your files or you can skip that screen of the wizard and add your file later. To add your file later, right click on the *Source Files* of the *Project Window* and select *Add Files*.
- If you add multiple .c files to the project, they will be compiled to .o files and then linked in the final stage. An already compiled .o file can be added to the project, and will be linked during the Make/Build process. If there is only one source in the project, it will be compiled and linked in one phase (no .o file will be created).
- Many project build options (such as output directory, include directories, output files generated, etc) can be changed by selecting “Project -> Build Options“ from the MPLAB toolbar.
- If the compile fails with an error that says something like “Target chip not supported” or “Compiler not found” make sure that a.) you have the proper PIC selected (use “Configure -> Select Device” from the MPLAB toolbar), b.) the CCS C Toolsuite has been selected for this project (use “Project -> Set Language Toolsuite” from the MPLAB toolbar) and c.) the path for CCSC.EXE is configured correctly for your installation of the CCS C Compiler (use “Project -> Set Language Tool Locations” on the MPLAB toolbar)
- If you are still having problems with MPLAB<sup>®</sup>, try opening a DOS command prompt and invoking the compiler from the command-line to verify that the compiler is working properly. See the CCS manual for help on using the command line compiler.

## Helpful Hints

### How do you view float variables inside MPLAB<sup>®</sup> IDE?

When you add a watch for a float variable MPLAB commonly only watches it as an 8bit register. In the Watch Window, highlight the float register you want to watch and right click and choose properties. A dialog box will open, you will want to change the properties to these settings:

**Size:** 32 bits

**Format:** MCHP Float

**Byte Order:** Low:High

### How can I link an MPASM object into my project?

You can link in a Microchip MPASM object like this:

```
#import (file=yourmpasmfile.o, COFF)
```

You need to do this before you attempt to use any of the MPASM symbols in your C program. A good place is after you include the device header file in the main file.

Note: This may work to link a COFF file made by C18, but this functionality was not tested.

### How can I use my HEX files with MPLAB tools to program my target?

MPLAB is the IDE Microchip provides and provides programming support for all Microchip tools (PICSTART Plus, Promate, ICD2, etc). This provides a simple guide on how to import the HEX file into MPLAB so it can be programmed into your target. For more MPLAB documentation please refer to Microchip's documentation.

First, be sure that your HEX file will run stand-alone (doesn't need a debugger). [Click here for pointers on creating a stand-alone HEX file.](#)

Follow these steps:

1. Start MPLAB. All steps in this FAQ that refer to a toolbar means the toolbar of MPLAB.

2. Make sure no other project/workspace is open by selecting *File -> Close Workspace* from the toolbar.
3. Select your target device by selecting *Configure -> Select Device*. This will bring-up a dialog where you select your target PICmicro (PIC16F877A, PIC18F4520, etc). It also will tell you if your programming tool is supported for this device.
4. Open the Configuration Bits dialog by selecting *Configure -> Configuration Bits* from the toolbar. This will open a new dialog window, at the very top of this window you will see a checkbox labeled 'Configuration bits set in code', make sure this checkbox is SET. Failure to do this will cause invalid configuration bits to be written to the device.
5. Select your programming tool by selecting the *Programmer* item from the toolbar. This will give you a list of selectable programmers.
6. Import your HEX file by selecting *File -> Import* from the toolbar. This will load the HEX file into memory. Importing the hex BEFORE you selected programmer or selected target device will cause the HEX file to be cleared from memory, so it is important to do it in the order shown here.
7. Program your device. This is done via the *Programmer* item from the toolbar. This process is different for each programming tool. But for the ICD2, you first need to connect to the device with *Programmer -> Connect Device*, and then program the device with *Programmer -> Program*
8. Repeat the previous step for each target microcontroller that you wish to program with this HEX file.