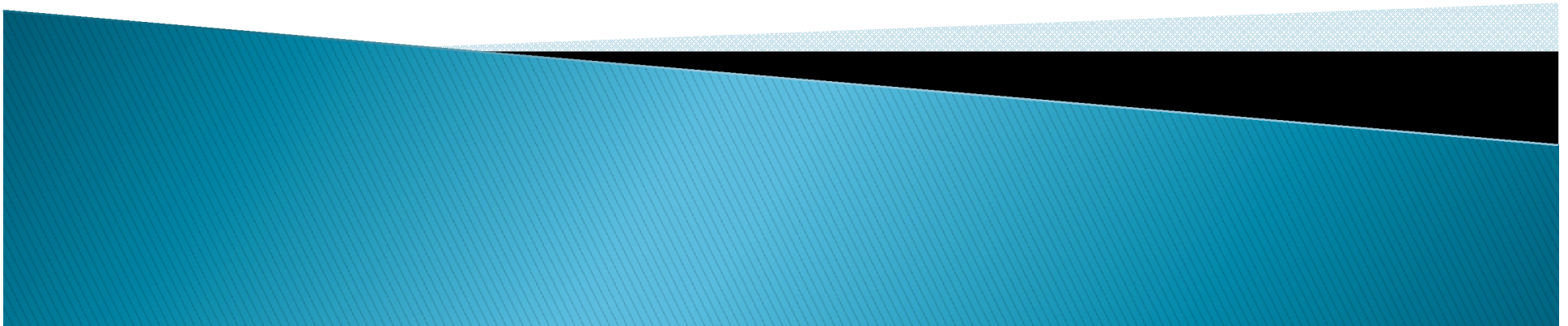
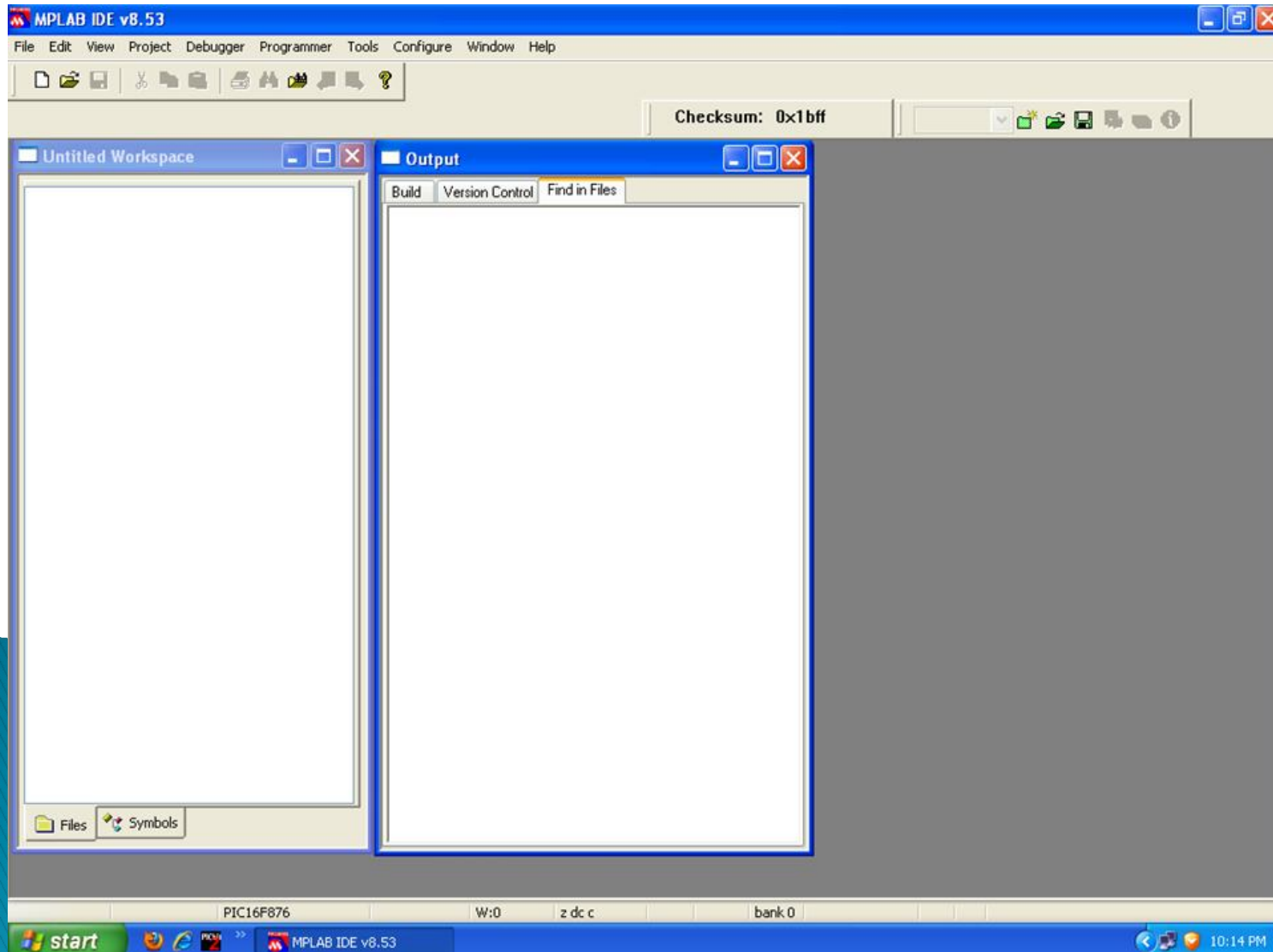


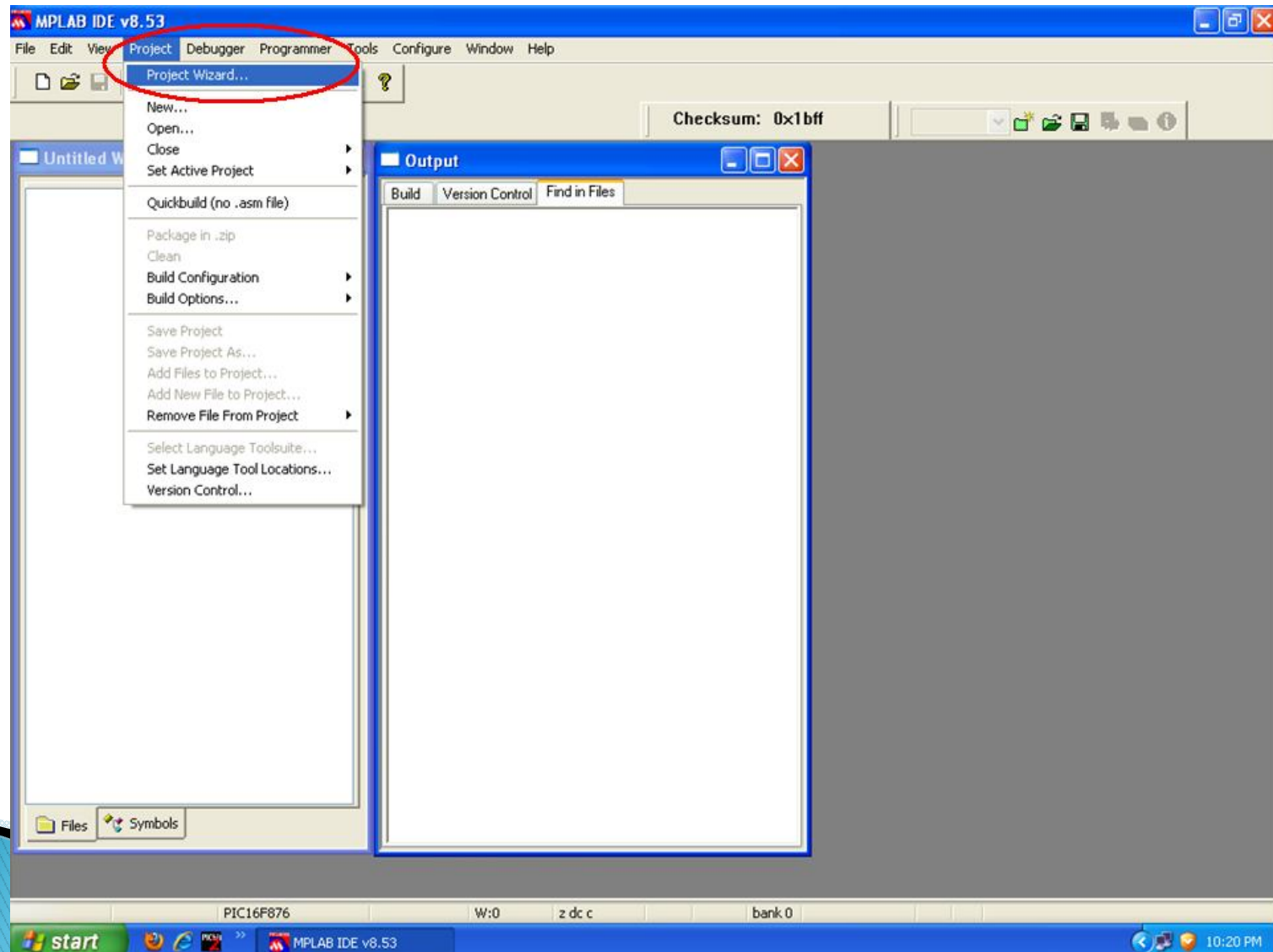
MPLAB PROGRAMMING



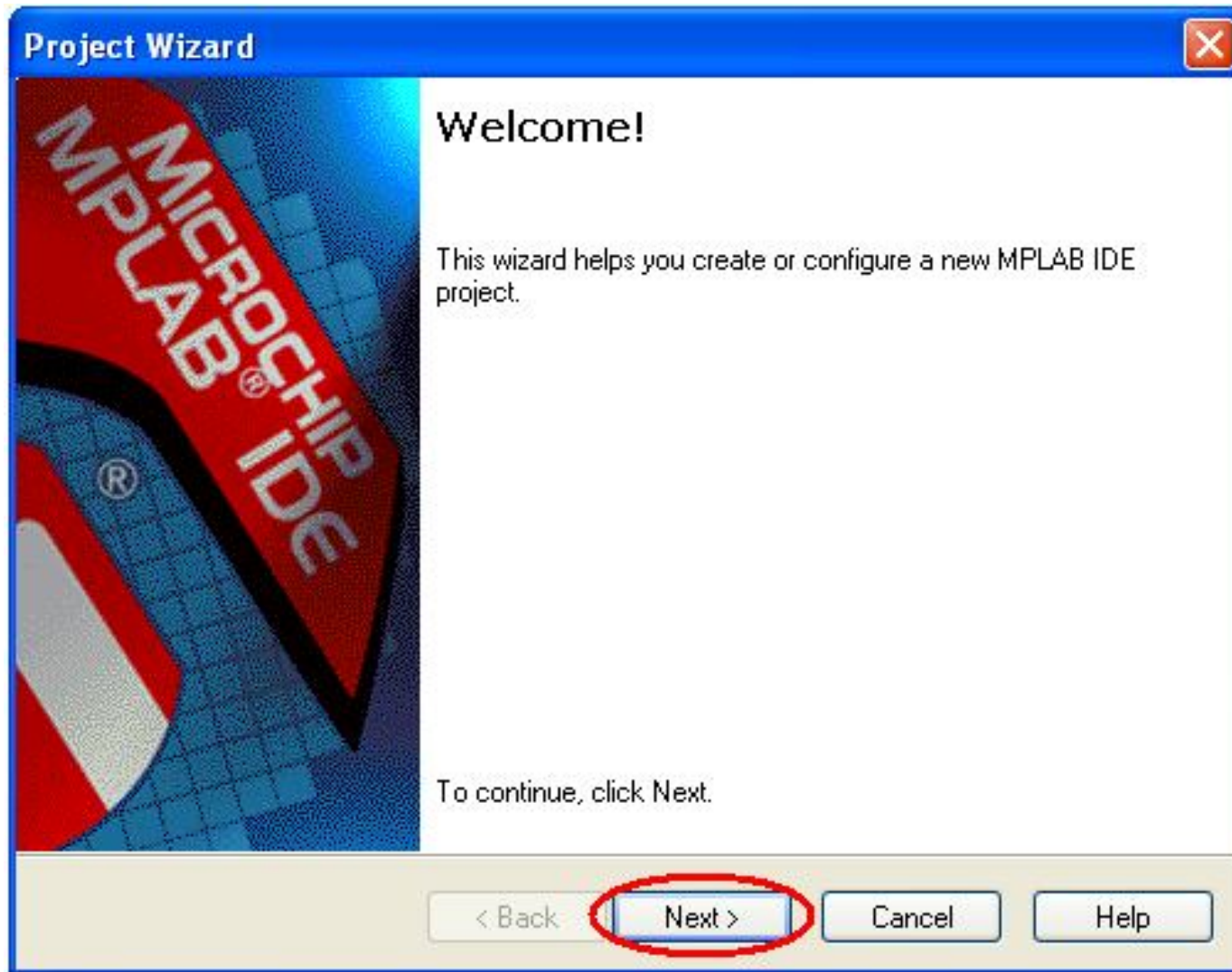
Double click on the icon installed on the desktop after installation or select ***Start>Programs>Microchip>MPLAB IDE >MPLAB IDE***. A screen will display the MPLAB IDE logo followed by the MPLAB IDE desktop as in diagram below.



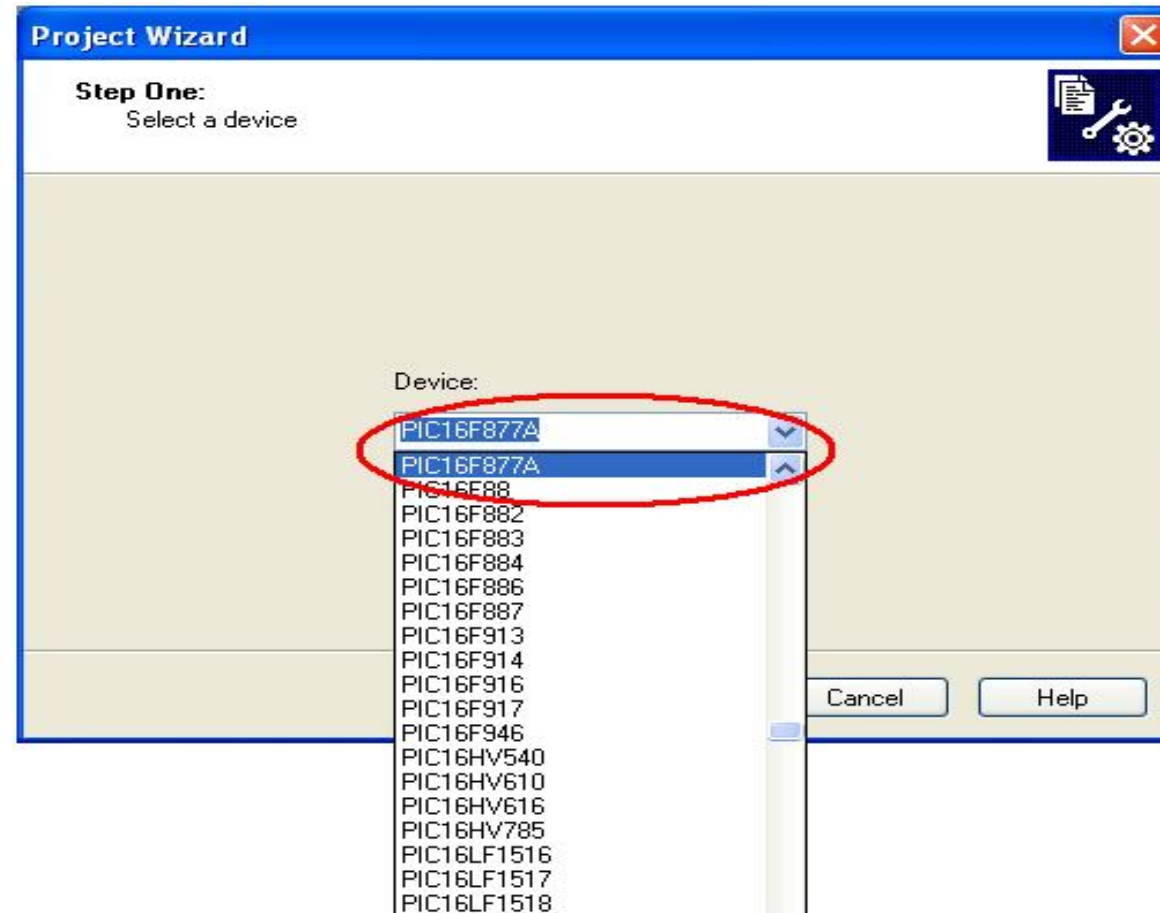
The next step is to create a project using the Project Wizard. A project is the way the files are organized to be compiled and assembled. We Choose *Project>Project Wizard*.



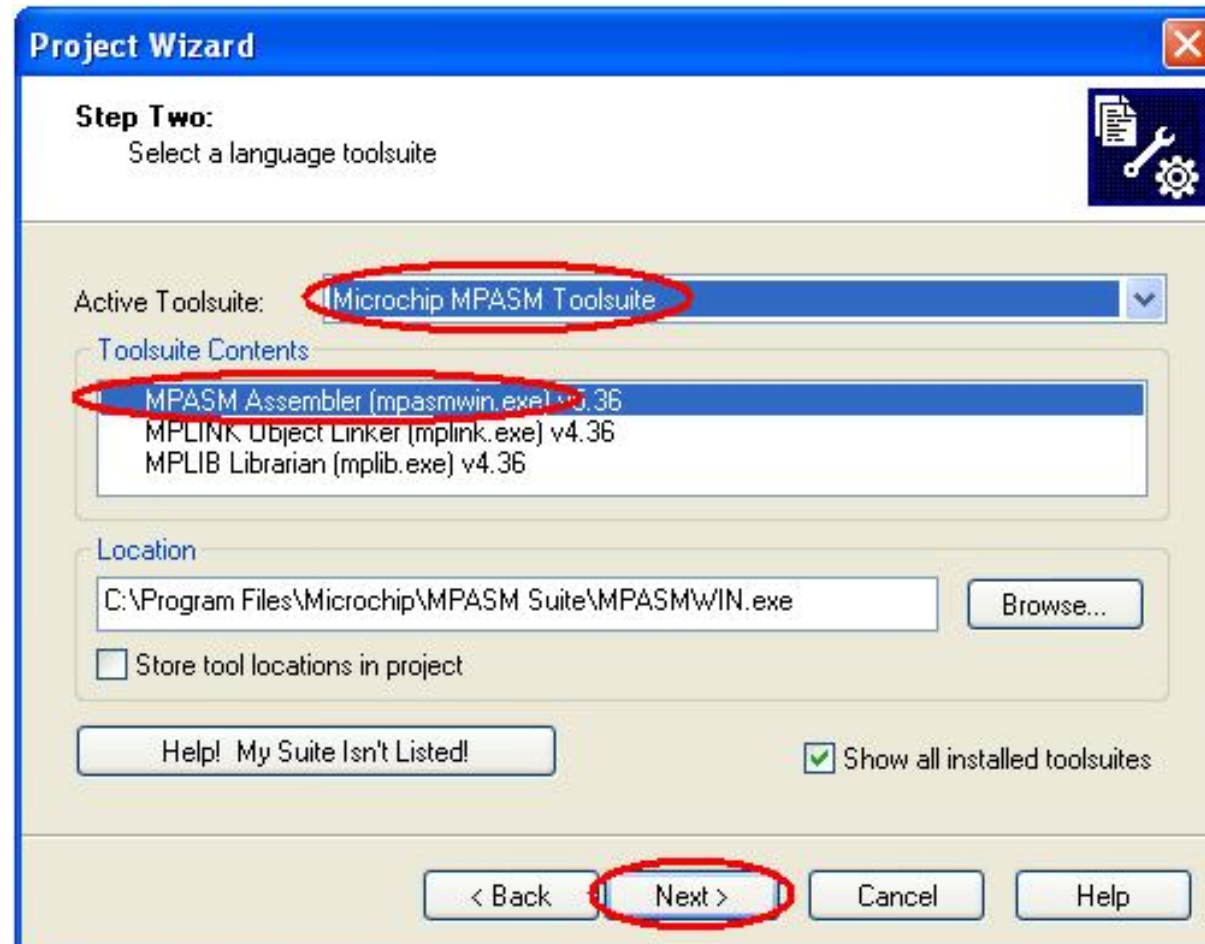
From the Welcome dialog, click on *Next to proceed.*



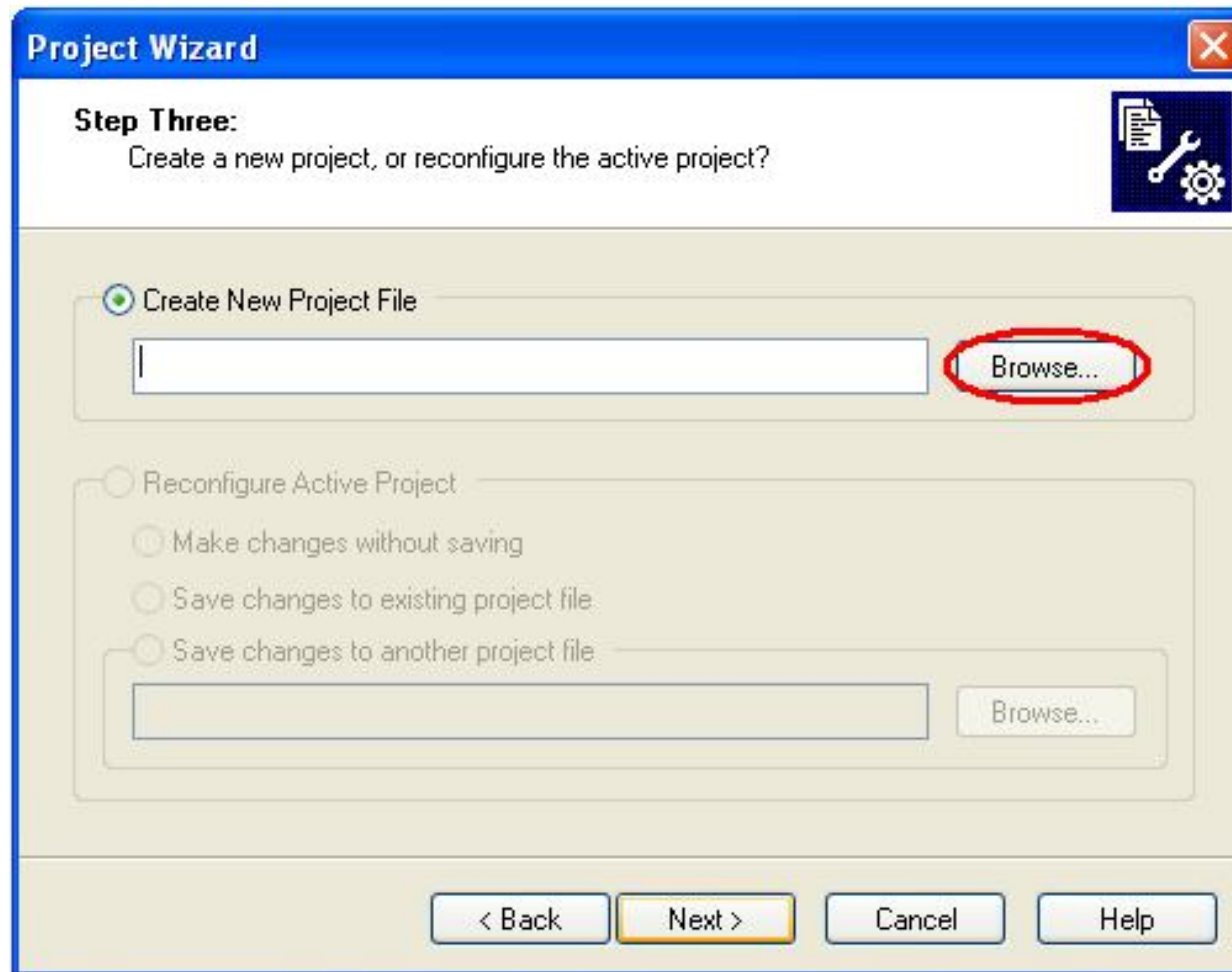
The next dialog (Step One) allows you to select the device. In this example, PIC16F877A was selected from the drop down menu. **Click Next>**.



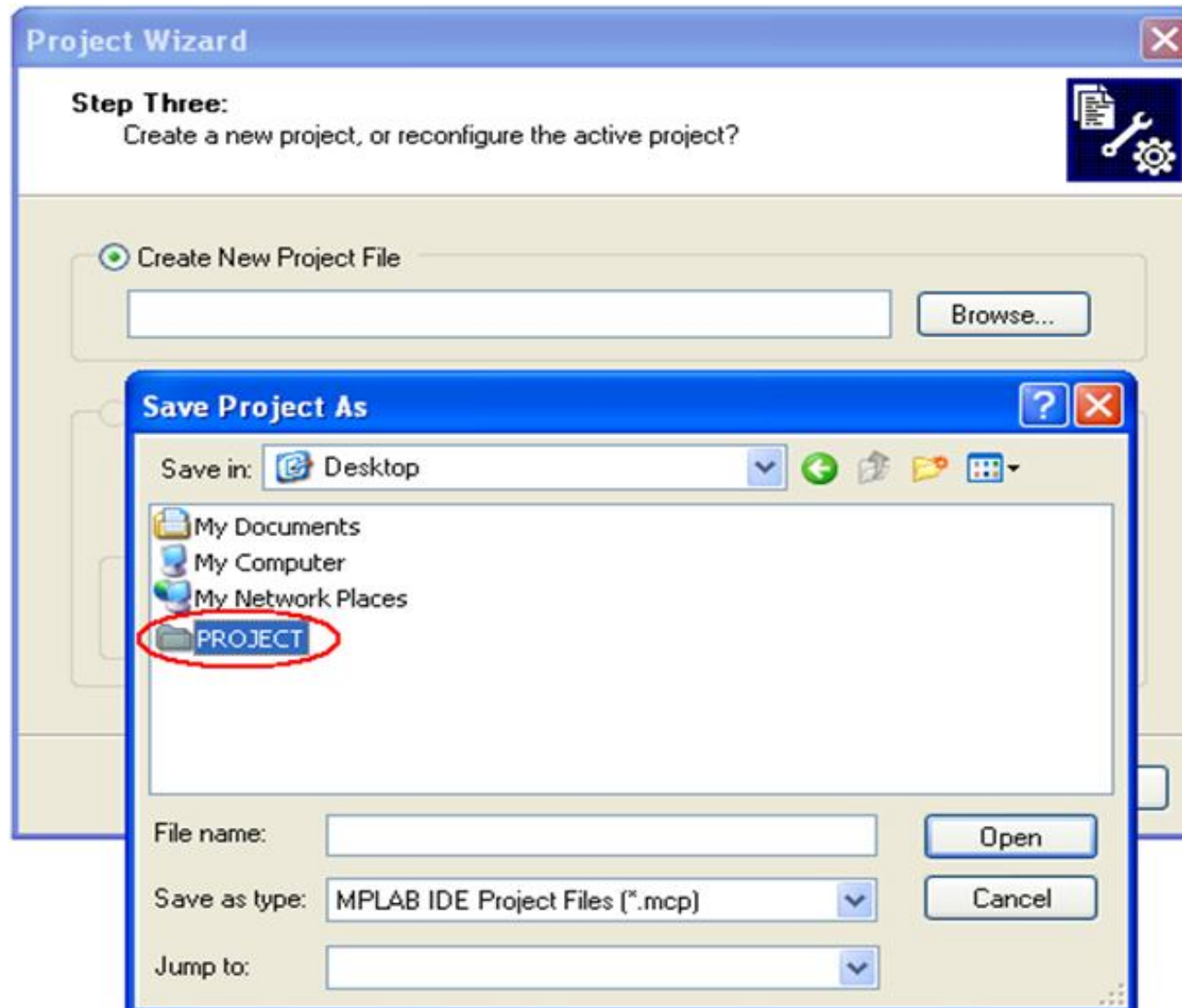
The next step of the Project Wizard is sets up the language tools that are used with this project. Select “**Microchip MPASM Tool suite**” in the Active Tool suite list box. Then select “**MPASM Assembler**” in the Tool suite Contents box. When you are finished, **click Next**.



Step three of the project wizard allows user to create new project file.



Create a folder, a folder named **Project** is first created at Desktop.



Then open the folder, *project*. Project named *project1* can be created by typing the project name in the column for “**File name**”, and click “*Save*”.

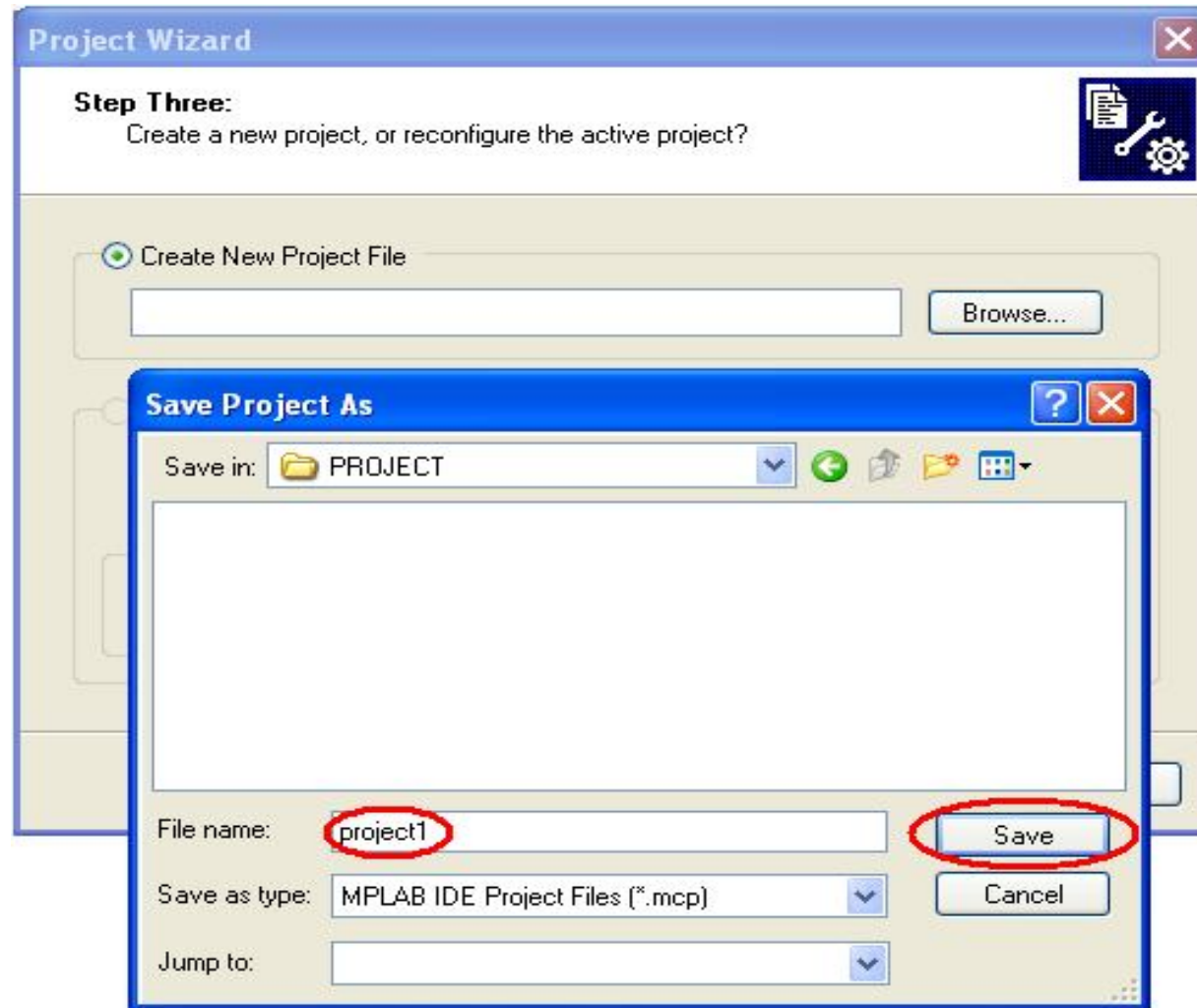
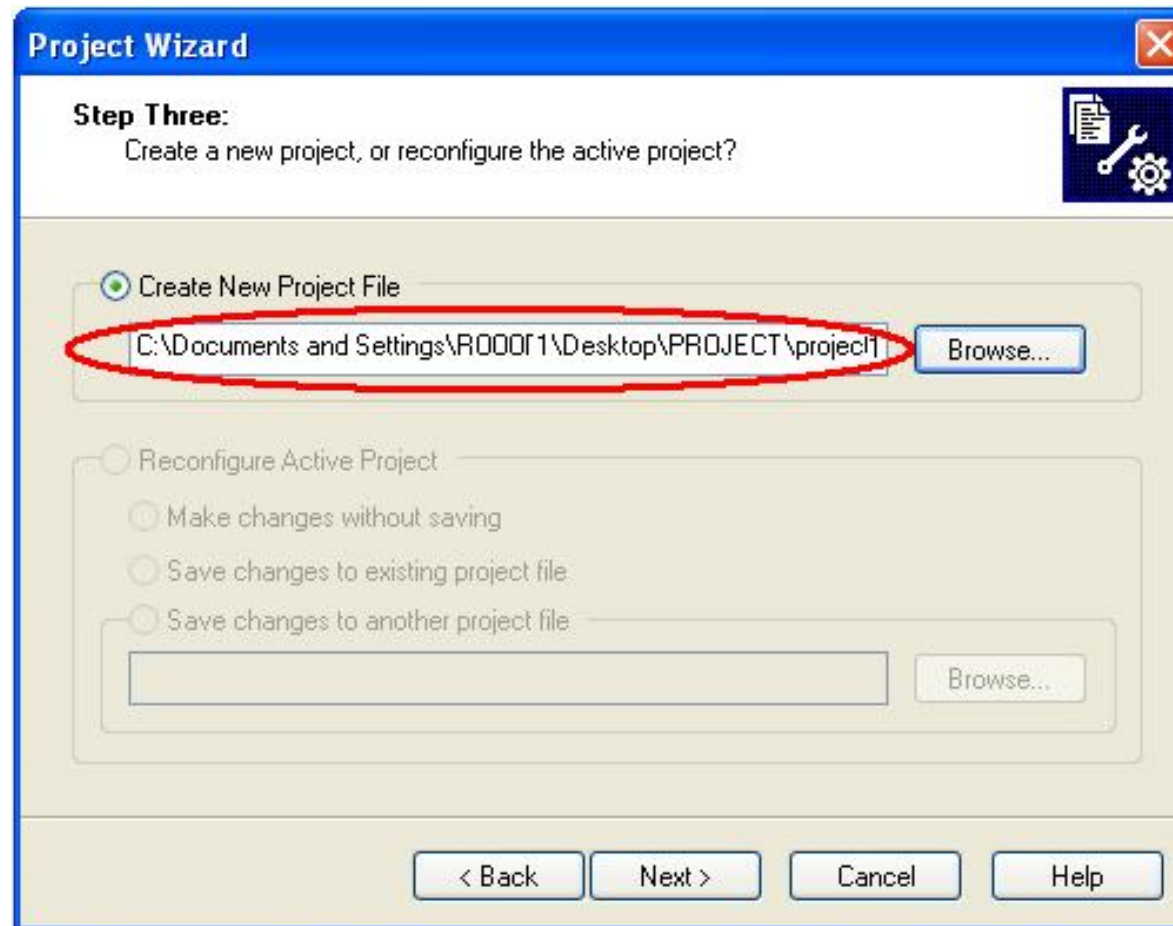
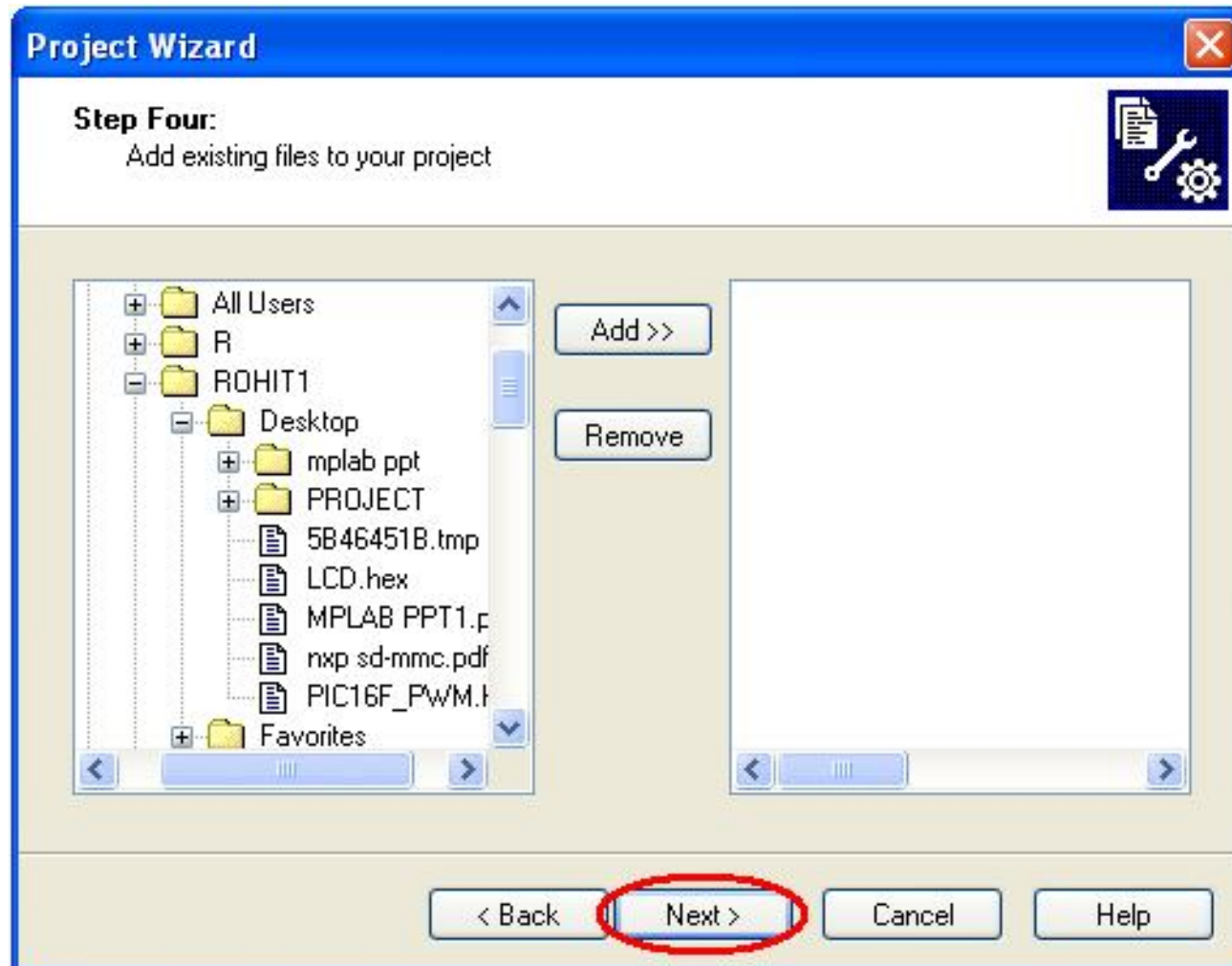


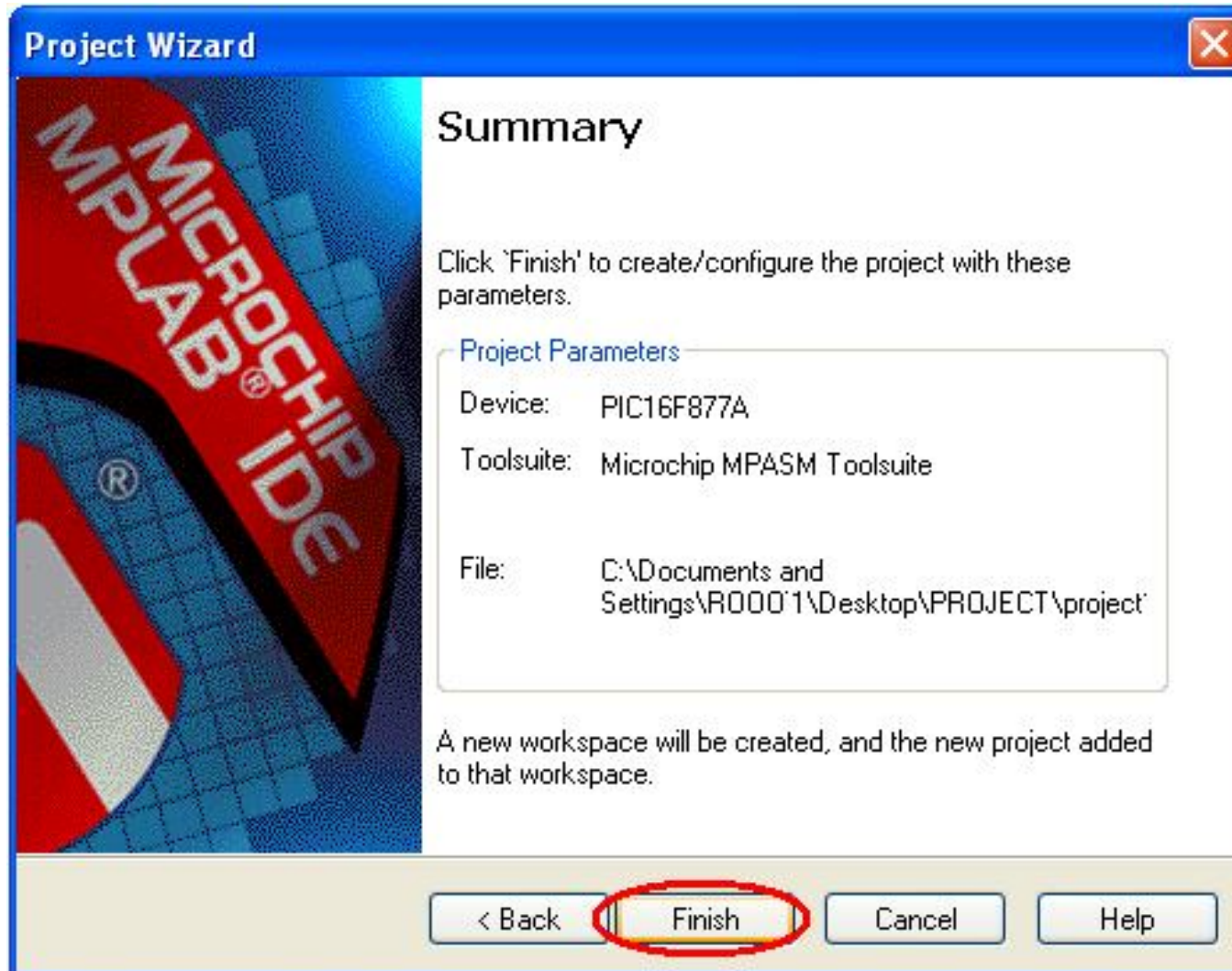
Diagram below shown the Project “*project1*” had been created and the directory. Click “*Next>*”.



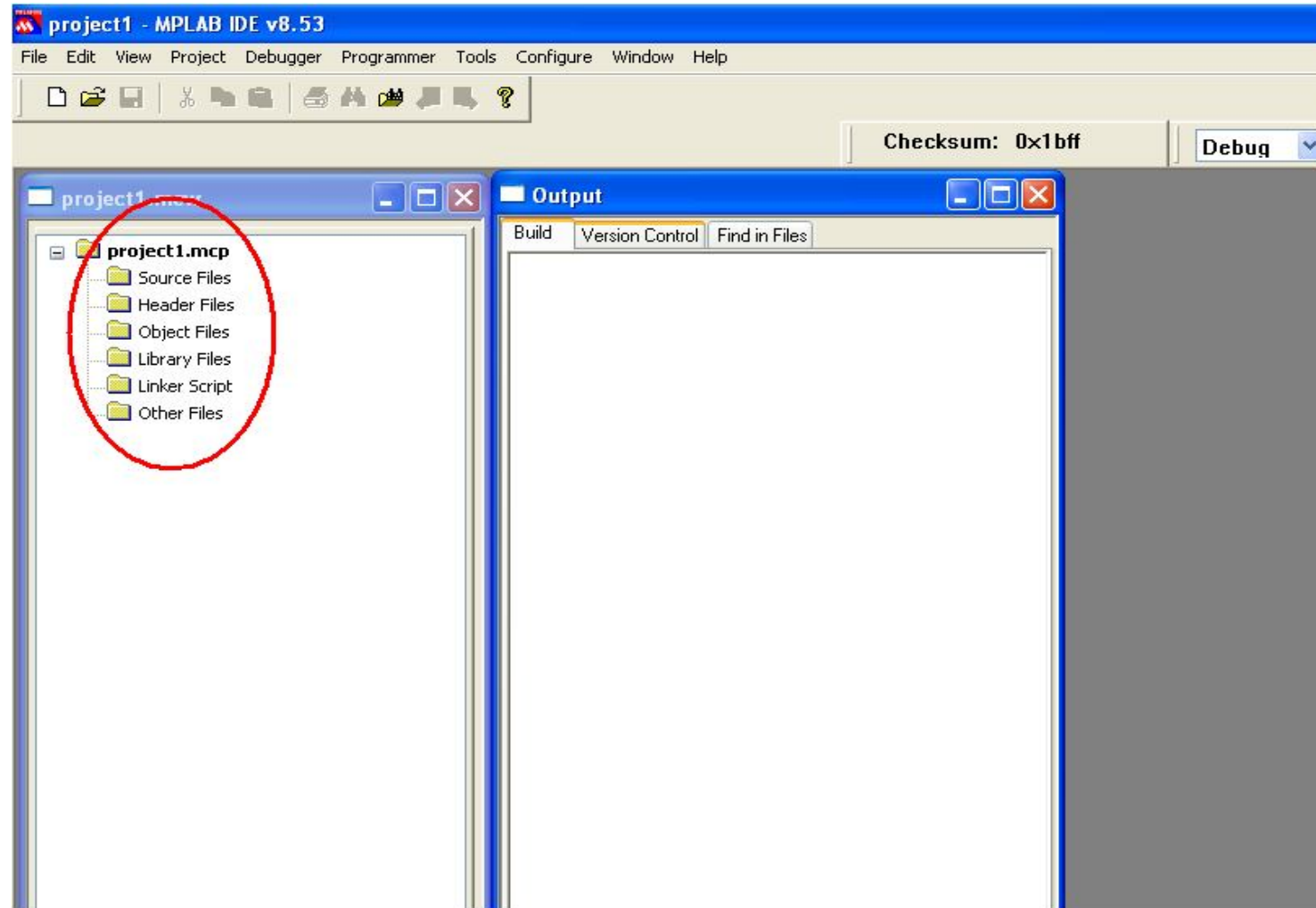
Step four of the project wizard allow user to add existing file to the project, however, for this example, no files will be added. Please click ***“Next>”*** to proceed.



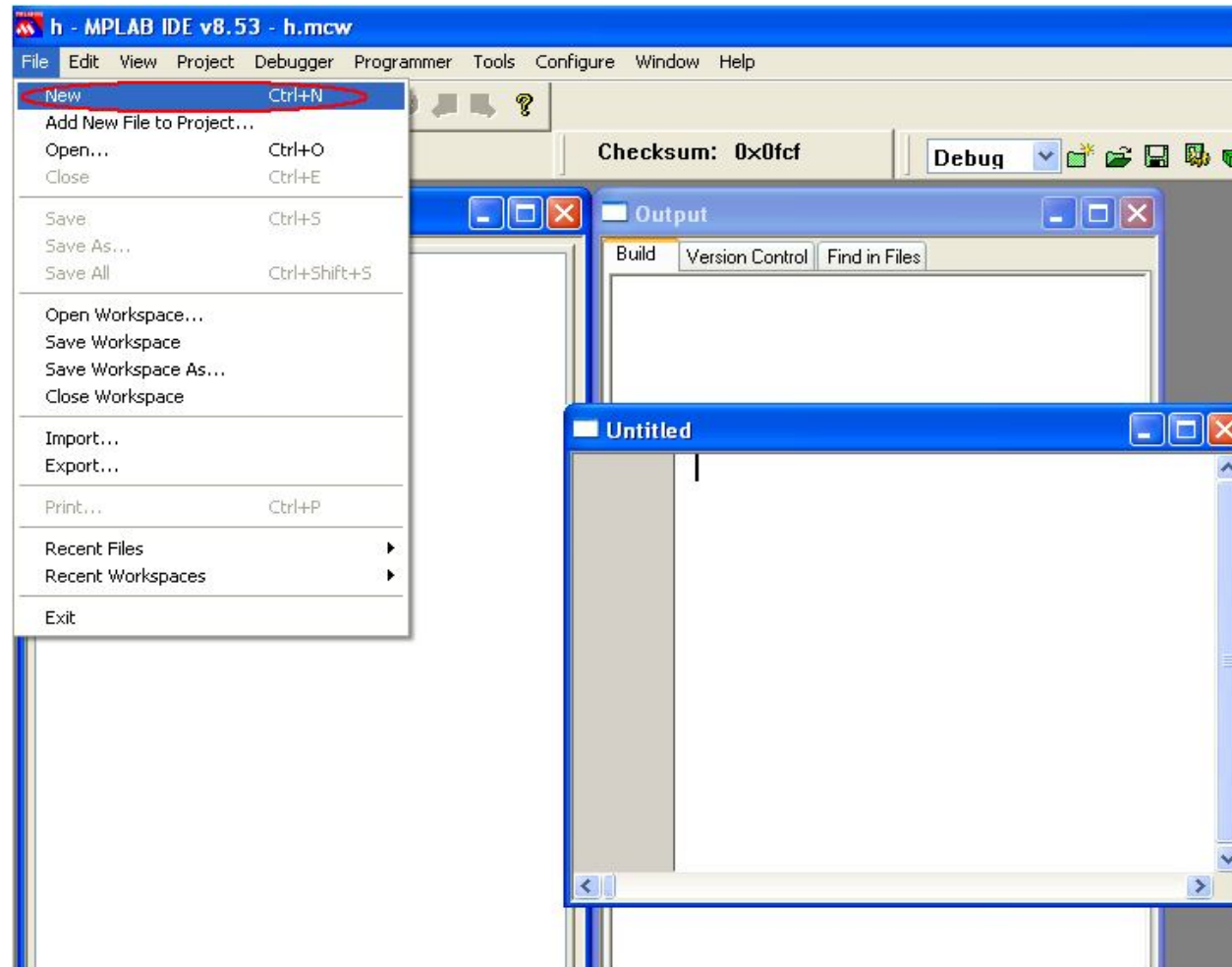
A summary will be shown at the end of project wizard, all the project parameters are shown. Please click ***“Finish”*** to exit from project wizard.



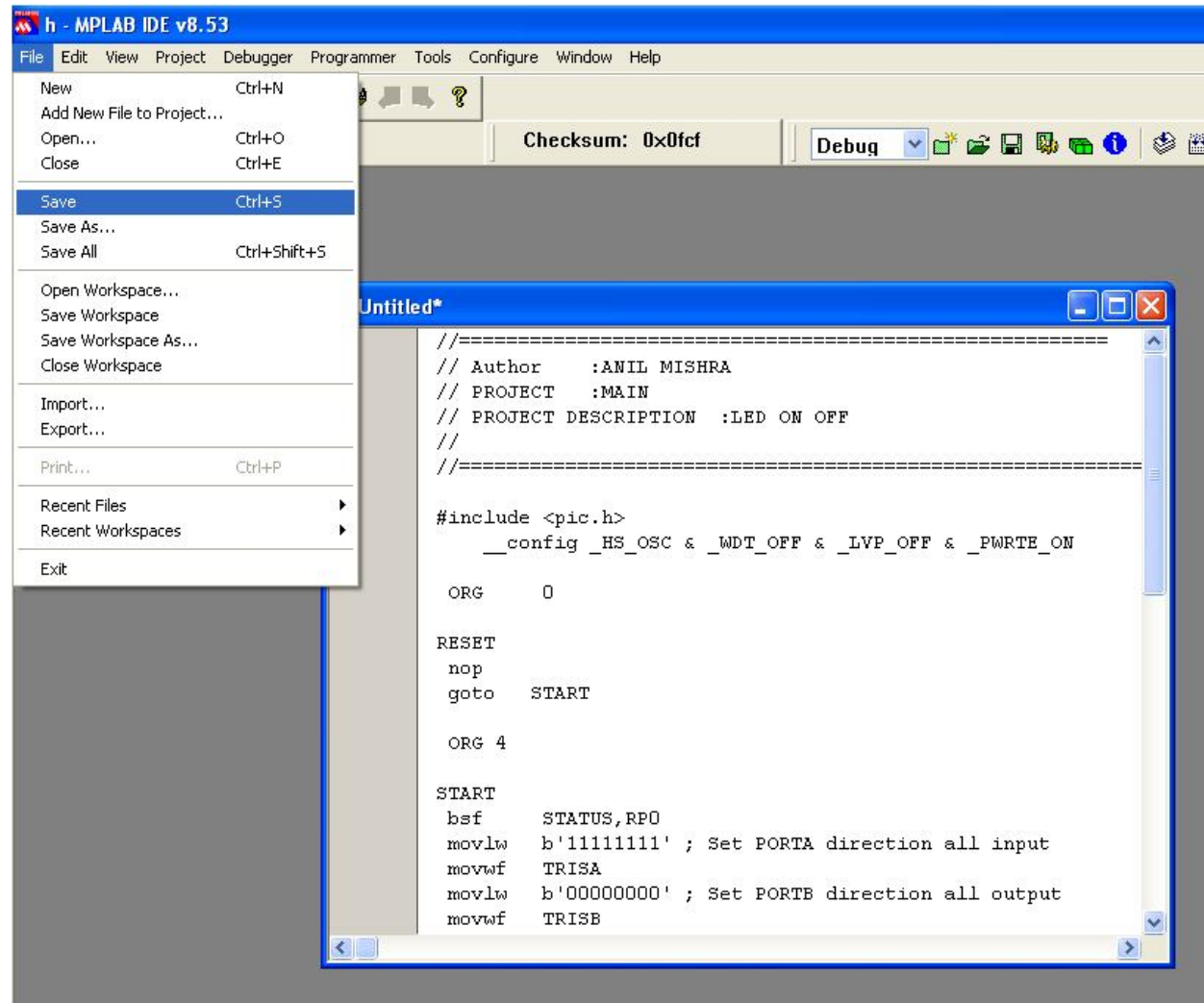
After pressing the *Finish* button, review the *Project Window* on the *MPLAB IDE* desktop. It should look like the diagram below. If the *Project Window* is not open, please select “*View>Project*”.



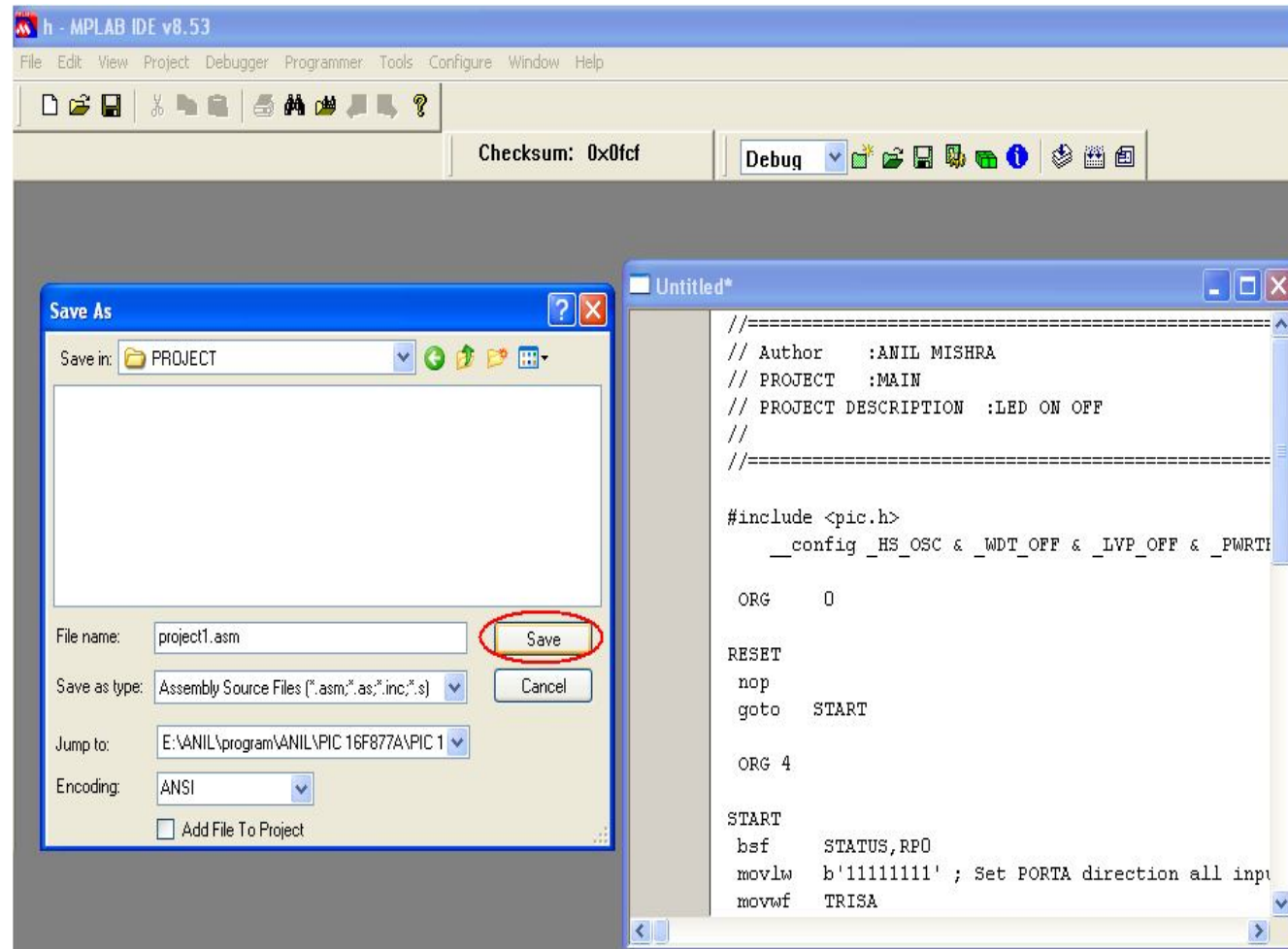
Click “**File>New**” option. A new window is opened for creating new program. See below.



After creating program, click **“File > Save”** option



A diagram is shown below, a **Save As** window is opened. Choose a correct folder where file was created, and then click **“Save”**.



In a programming window click right button of mouse, click **“Add To Project”** .

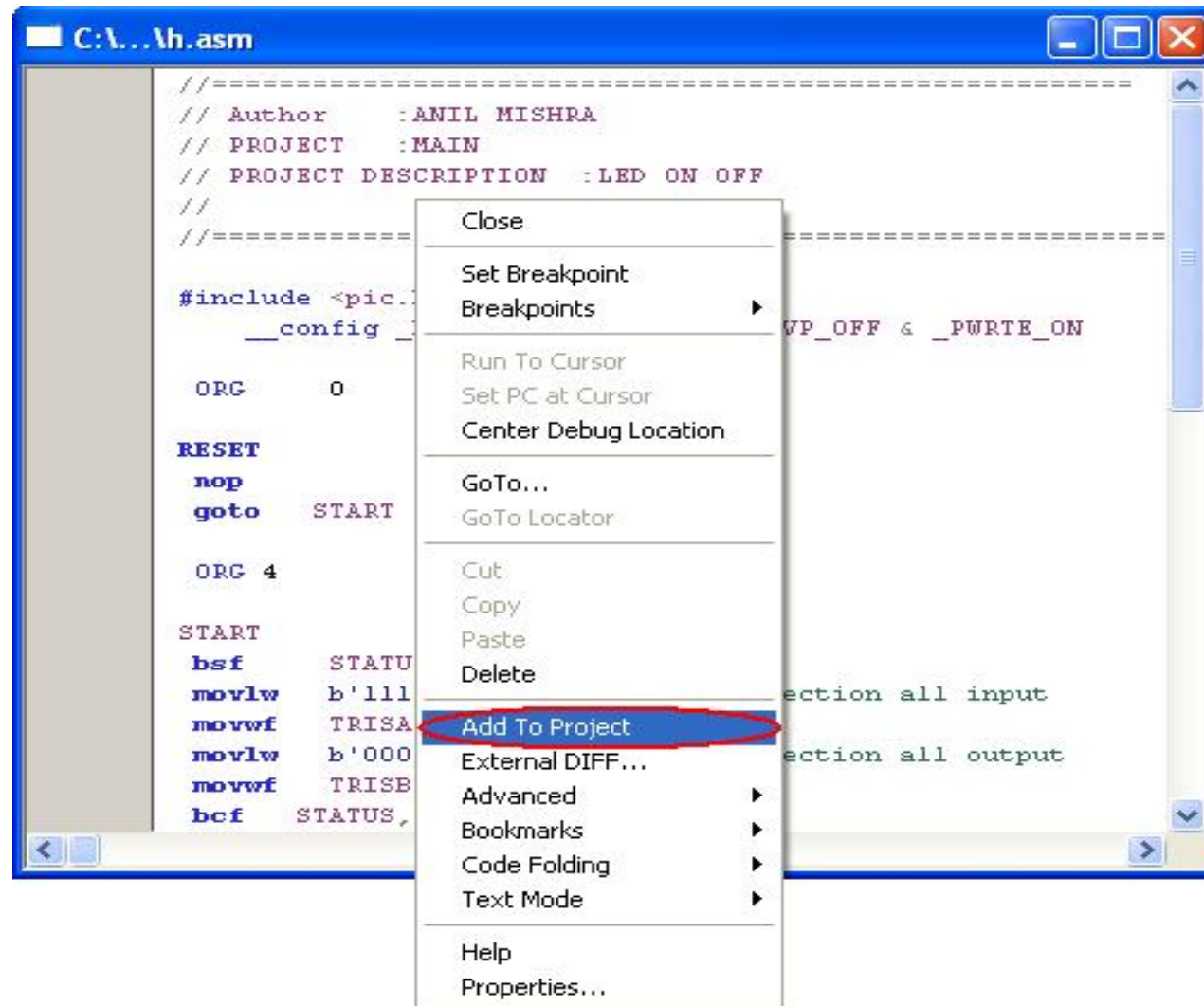
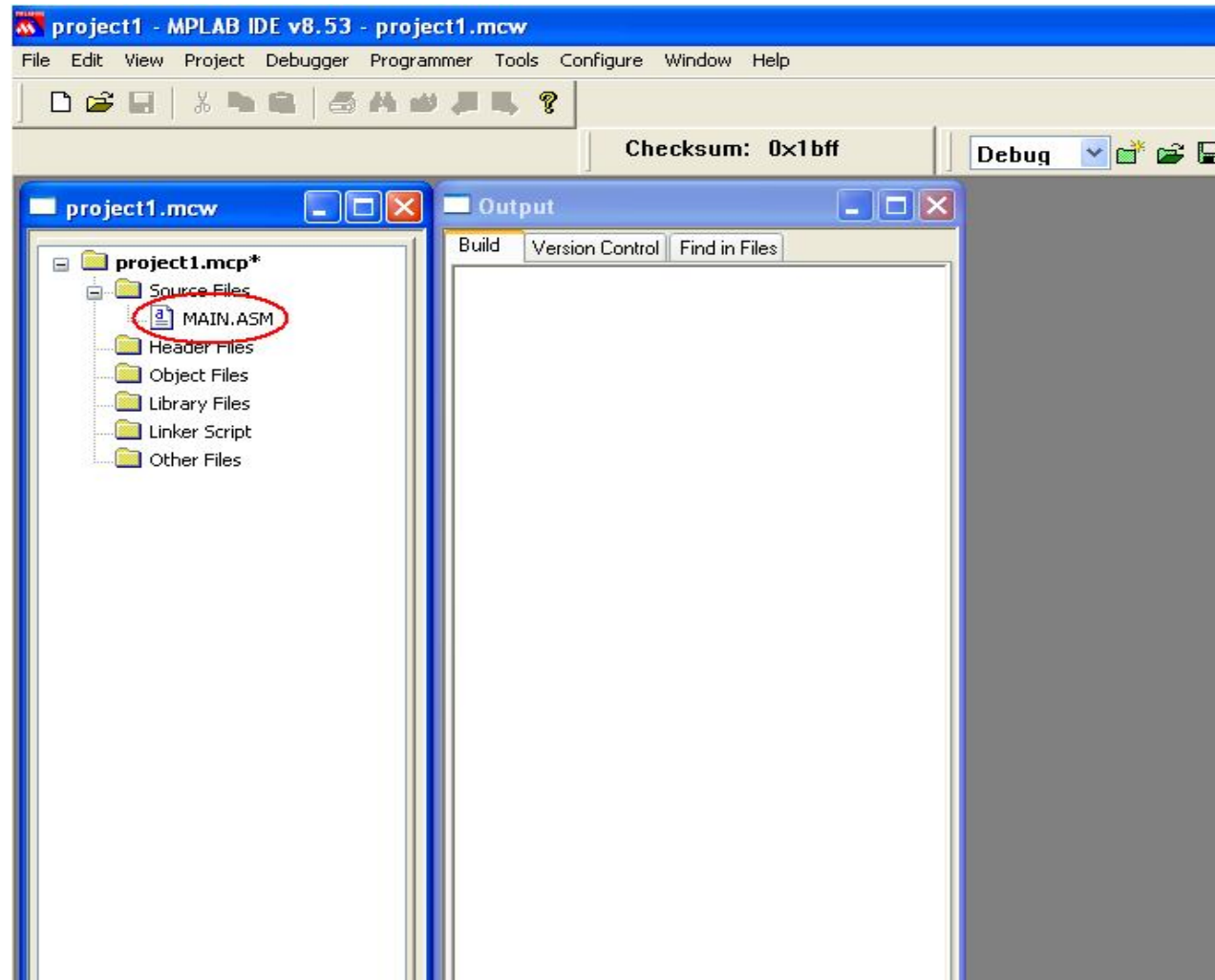
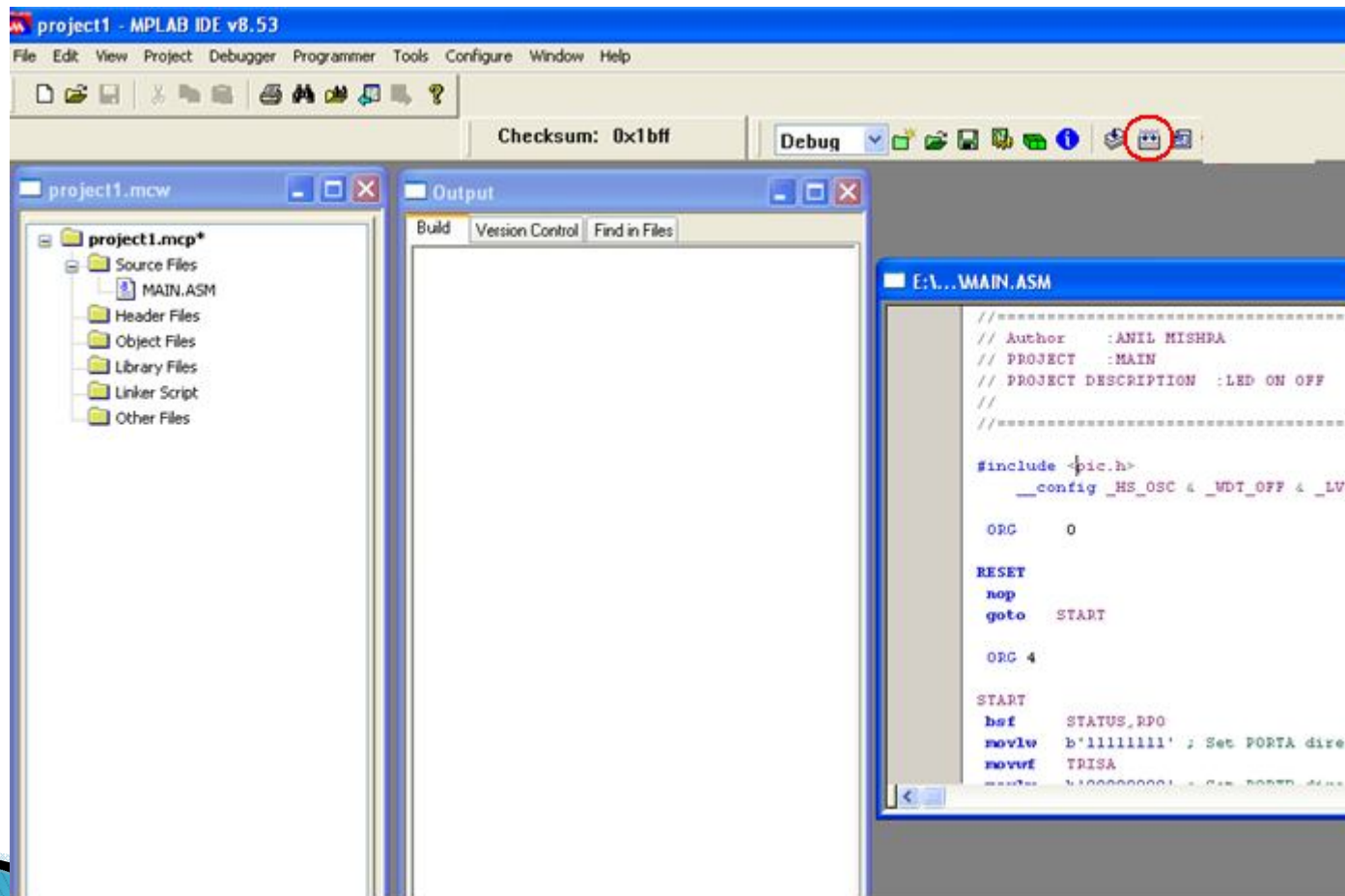


Diagram below shown **MAIN.ASM** added to the project.



After added the source file, user can open **MAIN.ASM** file in this workspace and try to compile it. Diagram below shown opened **MAIN.ASM** file. To compile, user can go **“Project>Build”** or the **build icon (in red circle) on menu bar** as shown in diagram below.



After build success, a message ***Build successful!*** will appear in output window like shown in diagram below.

