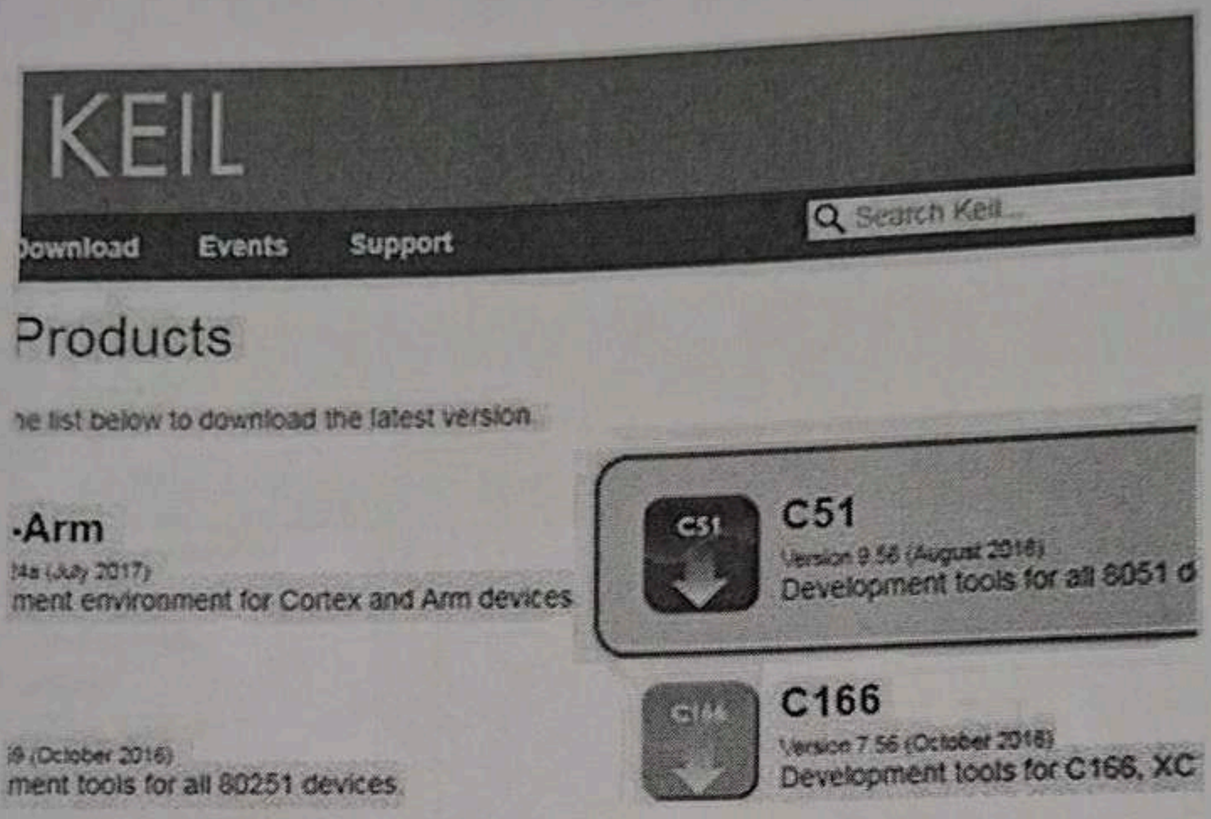
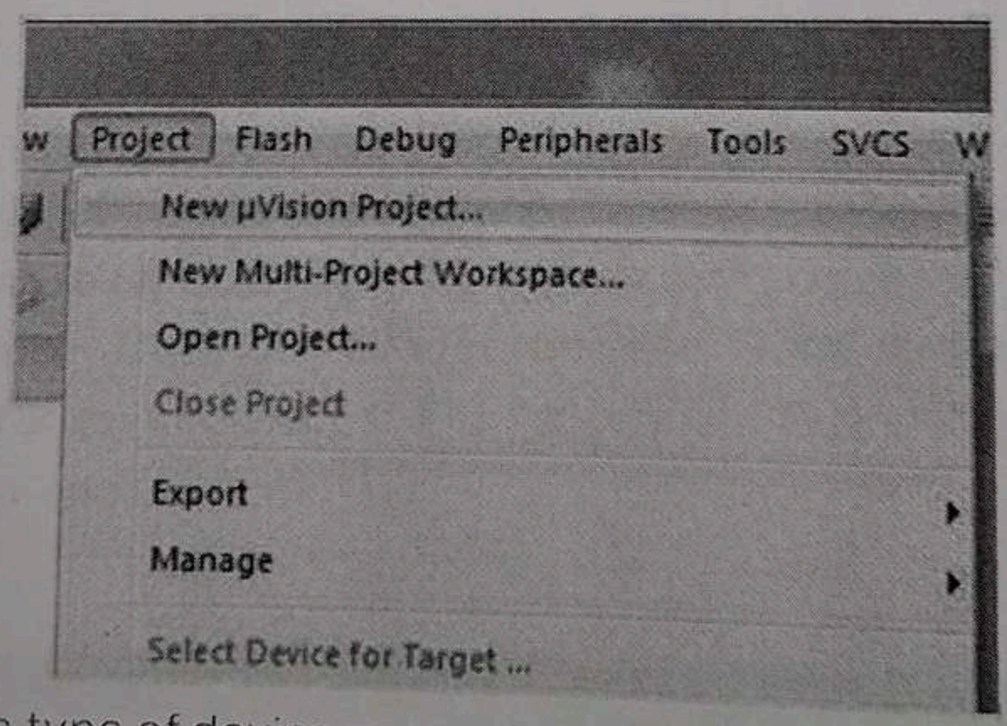


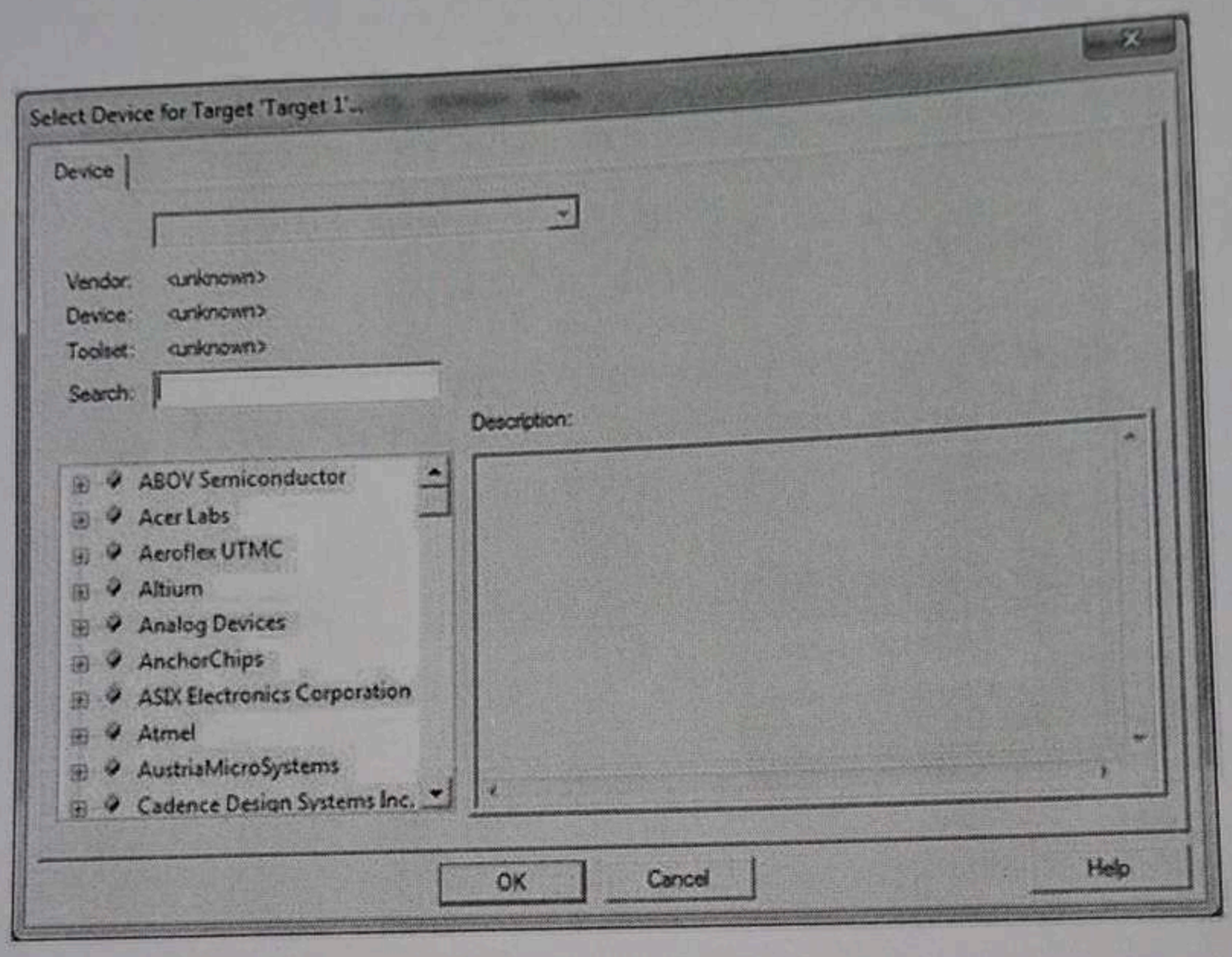
Step 2: To initiate the programming you must create a project using the keil Uvision IDE

The option to create a new project will be available under the project tab in the toolbar. Next, you have to store the project in a folder and give a suitable name to it.



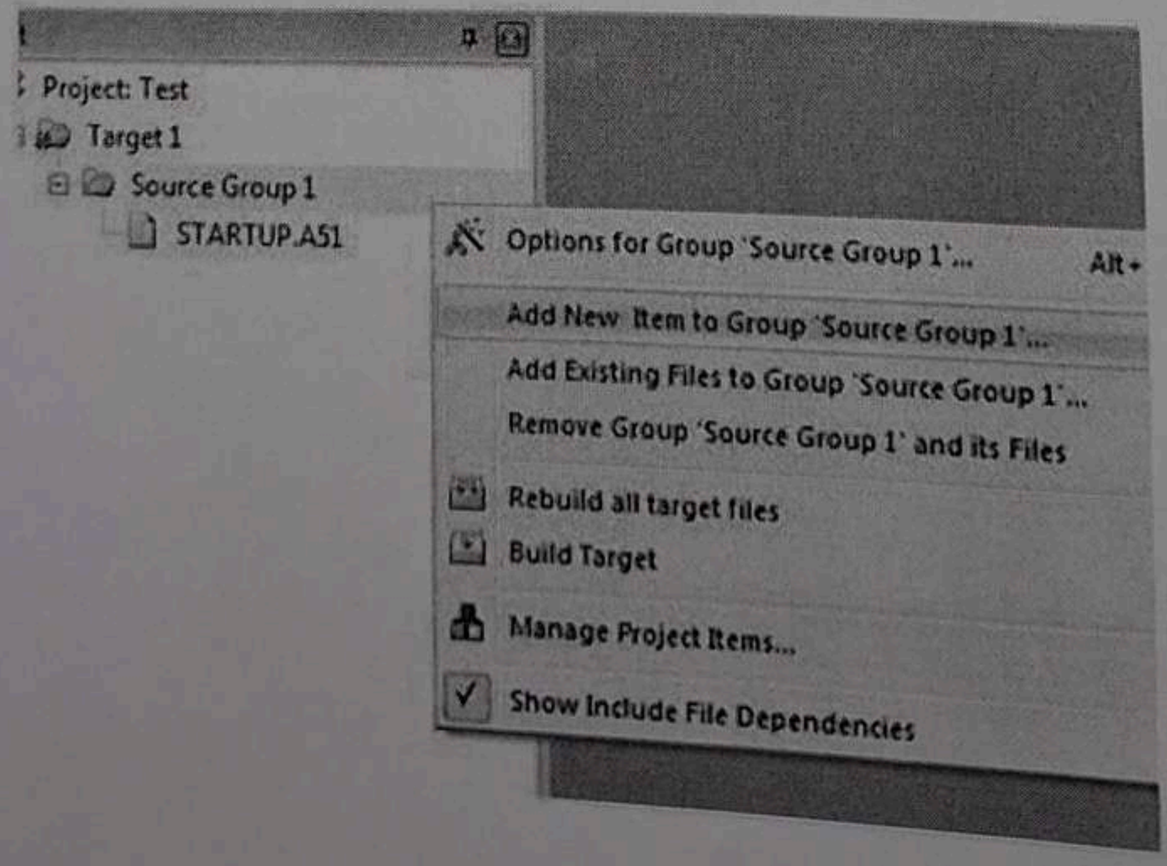
Step 3: Selecting the type of device you are working with

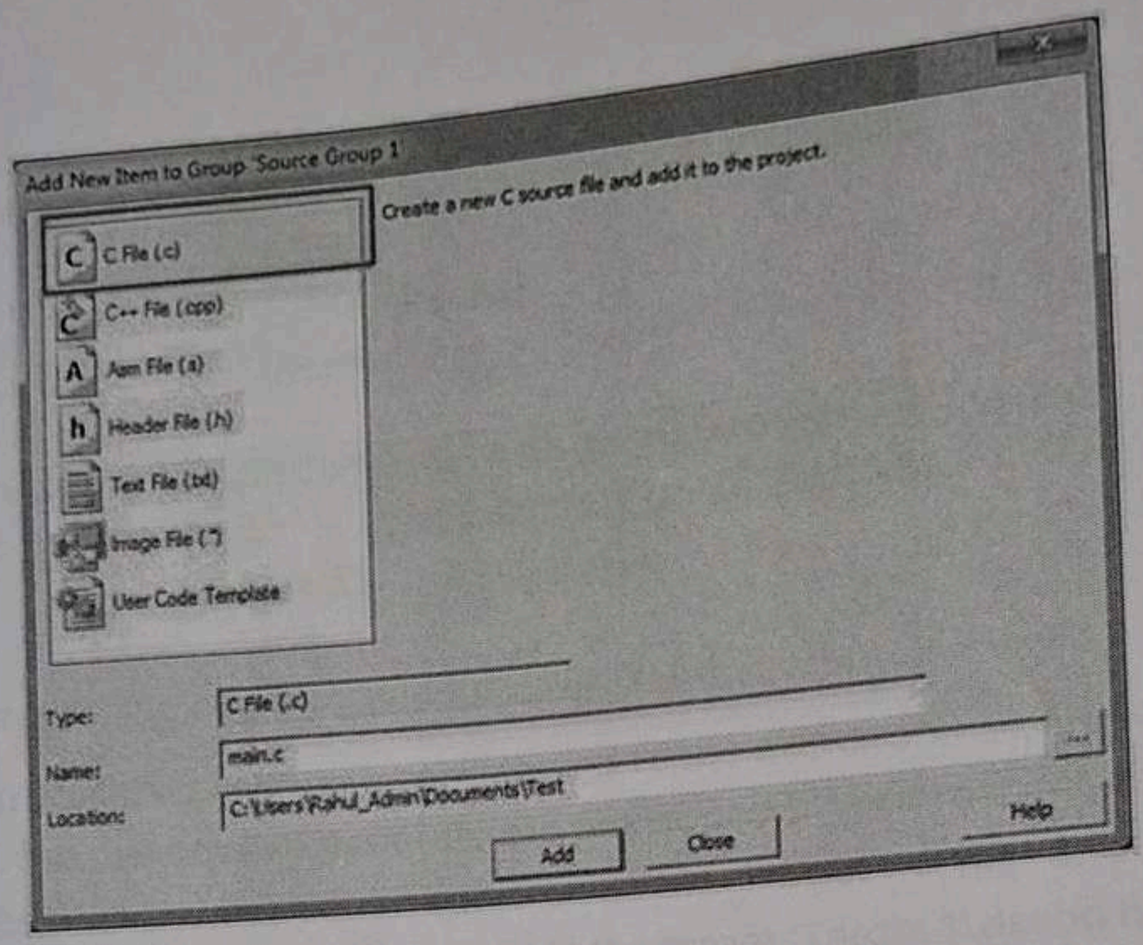
The device selection dialog provides you with the option to select the 8051 derivatives for which you want to develop the program. If you are not sure about your device you can refer to the description of the devices which is displayed on the left pane of the dialog. Accordingly, select your device and click OK to confirm.



Step 4: Adding C files to your project

You must add C file to your project before you begin coding. This can be done by right-clicking on your project from the project pane and selection "Add a new item to source group 1". In the next dialog box, you will be given with the choice on what type of file you want to add such as C, C++, header, text etc. Since here we are dealing with Embedded C programming, select C File (.c) option. Provide the necessary name, location and click on add.



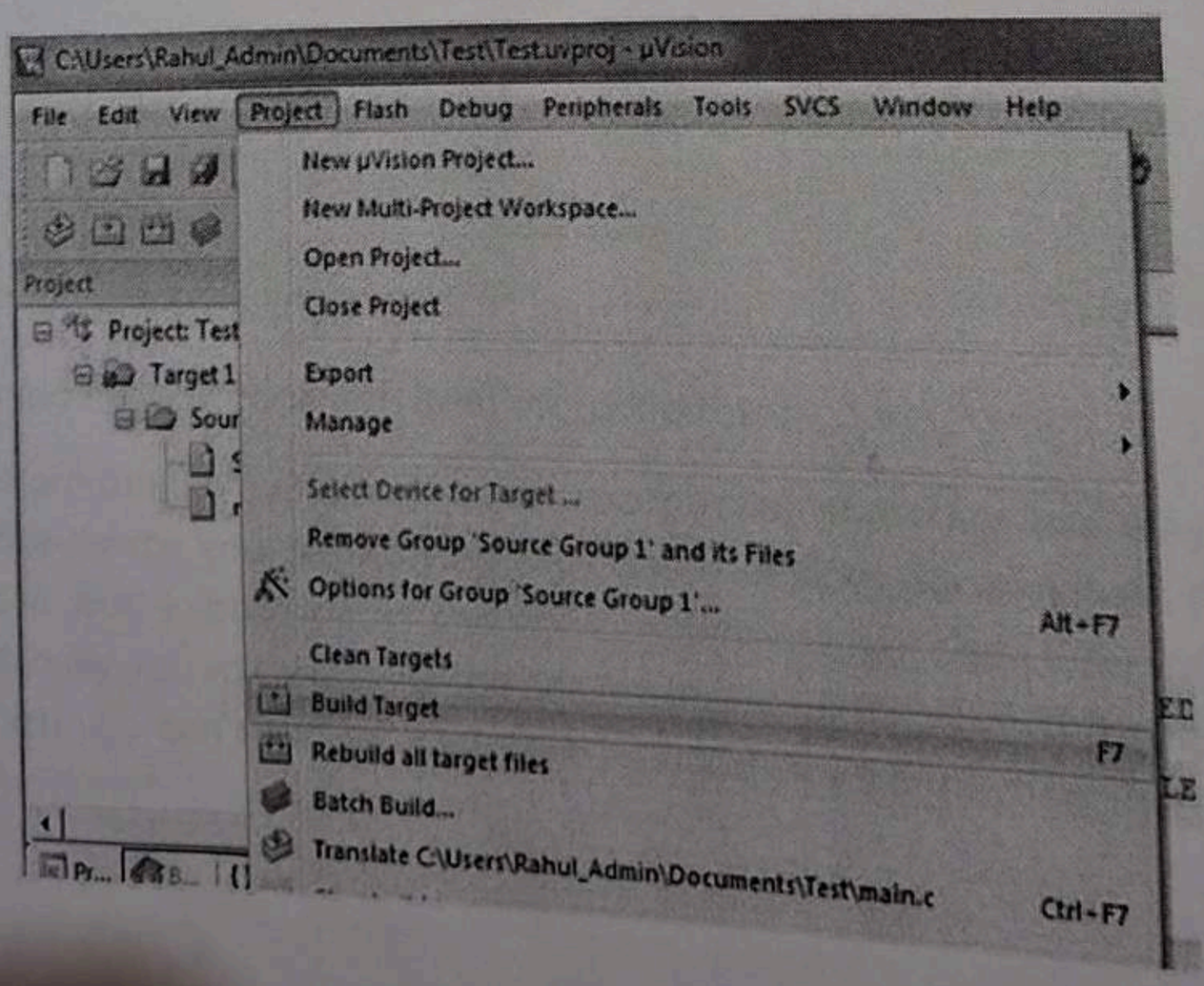


Step 5: Coding in C

The main part has finally arrived, so now you can go along with programming in C with your respective microcontroller.

Step 6 : Compiling and building the C project using Keil Uvision IDE

In order to build recently created C program go to Project tab and click on Build Target on the menu bar. An alternate way to do this is by pressing the F7 key. If the code that you have written is correct, the code will successfully compile without any errors. You can check your output in the Build Output pane.

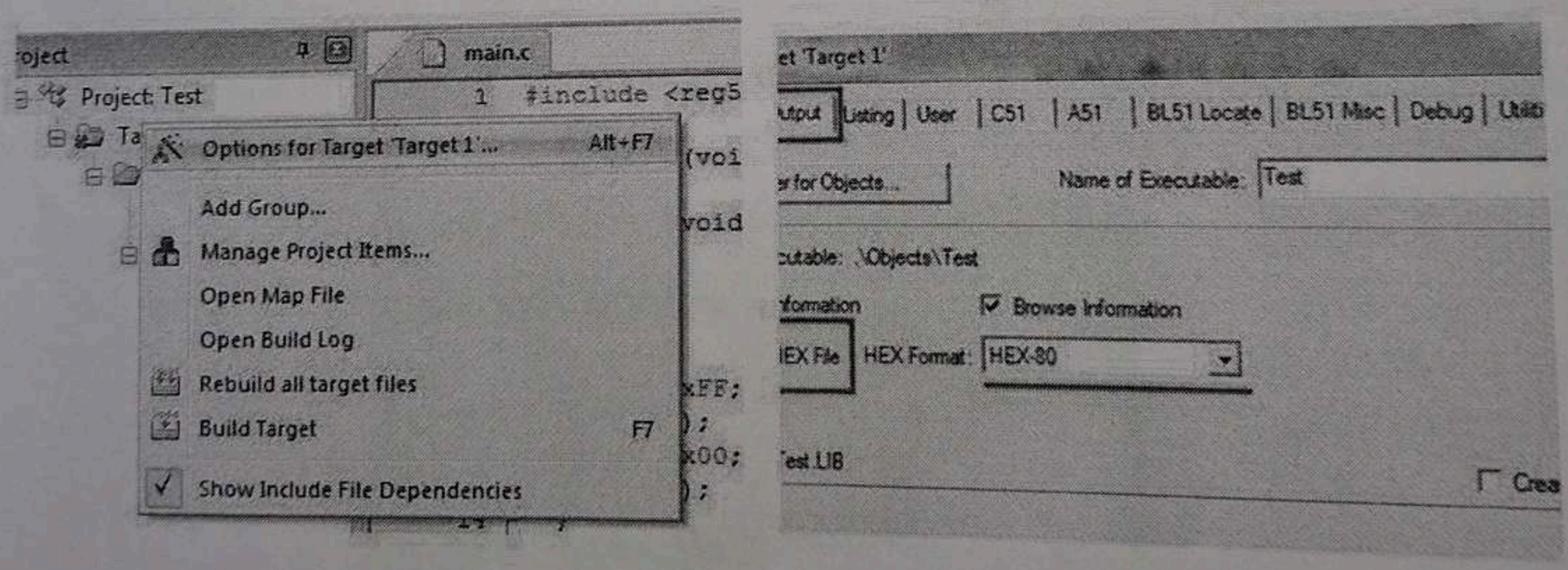


```
 'Target 1'
TARTUP.A51...
in.c...

: data=9.0 xdata=0 code=56
est" - 0 Error(s), 0 Warnir
lapsed:  00:00:01
```

Step 7: Generating the hex file using Keil Uvision IDE

The code you compiled cannot be directly fed to the microcontroller, it is not possible. For that purpose, we have to generate the hex code for your respective file.
In order to generate the hex code, right click on the 'Target 1' folder and select options for target 'Target 1'. Select the Output tab in the target 'Target 1' dialog box. Make sure Create Hex File option is checked and the HEX format should be HEX-80. Click OK.
Again rebuild your project by pressing F7. Your required hex file would have been generated with the same as your project in the Objects folder.
If you wish you can also view your hex code by using a notepad.



Step 8: Burning the hex code into 8051 microcontroller

In order to burn the hex code to your microcontroller, there are two ways which are specific to the device you are working with. Some devices, for example, P89V51 they have their own built-in boot loader and you can burn the hex code directly through the serial port. Mostly you will require a specific programmer for your 8051 microcontroller through which you can easily upload your hex code by connecting the programmer via normal USB port.

2. Flash Magic

Flash Magic is a PC tool for programming flash based microcontrollers from NXP using a serial or Ethernet protocol while in the target hardware.
Procedure for using Flash Magic:

1. Select the IC from Select Menu.

2. Select the COM Port. Check the device manager for detected Com port.

3. Select Baud rate from 9600-115200

4. Select None ISP Option.

5. Oscillator Freq 12.000000(12 MHz).

6. Check the Erase blocks used by Hex file option

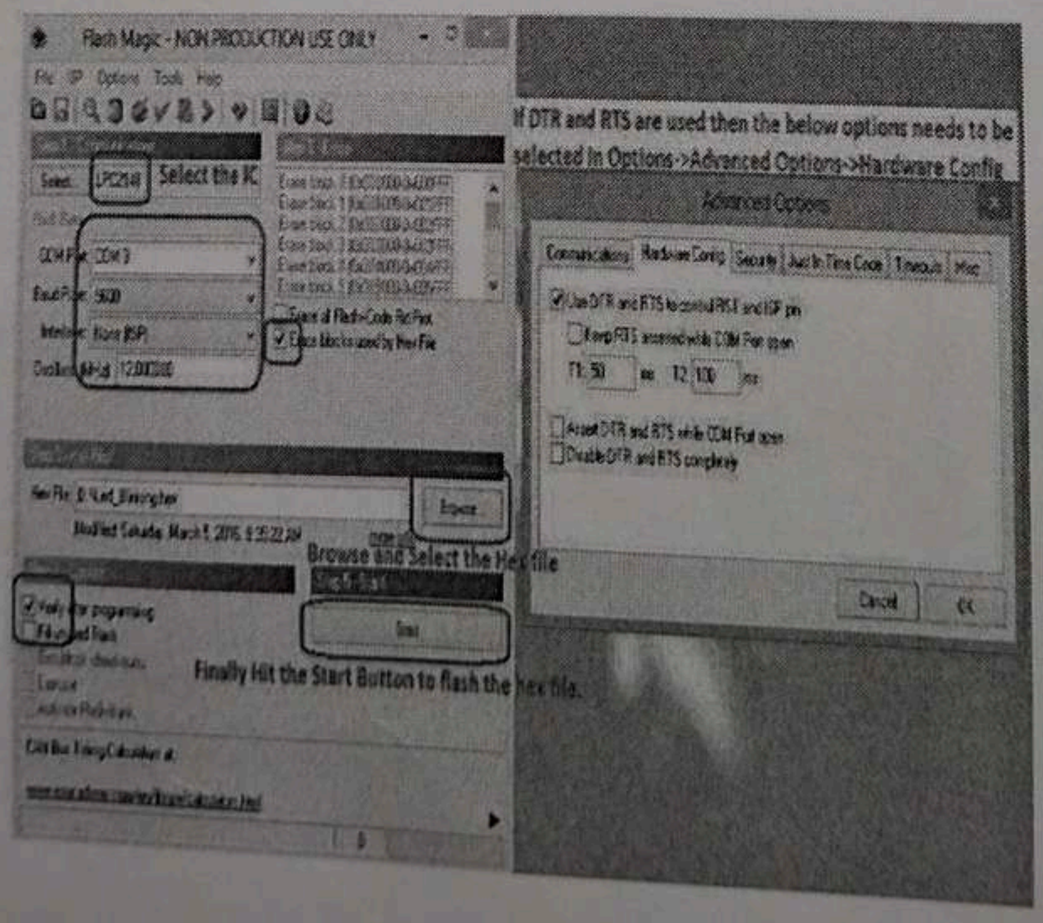
7. Browse and Select the hex file.

8. Check the Verify After Programming Option.

9. If DTR and RTS are used then go to Options->Advanced Options-> Hardware Configuration and select the Use DTR and RTS Option.

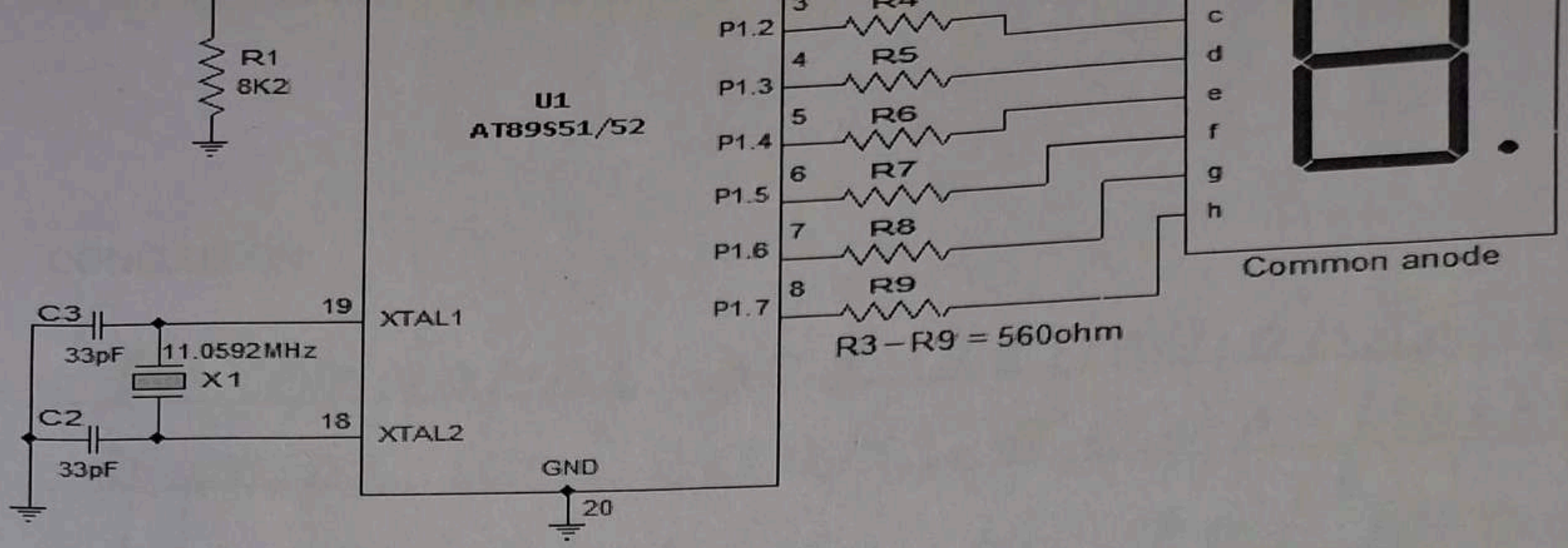
10. Hit the Start Button to flash the hex file.

11. Once the hex file is flashed, Reset the board. Now the controller should run your application code.



CONCLUSION:

the project is regarding to partial plate interfacing of LED's By using micro-controller kit power supply



Procedure:

1. Open the Keil µVision4 or 3 Software

2. Click on the project –> click on new µvision project –> type project name –> Click on Save

3. Select the Target from SST family controller –> click OK–> click YES on a copy STARTUP.A51 to Project

4. Click on File –> click on the NEW document –> Save this document with the filename.C extension.

5. Type C program and save using filename.c

6. After this in Project window Right-click on Source group 1–>click on Add existing file on source group 1–> click filename.asm–> Click on Add.

7. After this click on Project –> click Build Target(or pressF7 key) –> on Build the output window shows 0 errors and 0 warning then project is completely built if not then remove the error then build it.

8. Click on 'Project' –> Click on 'option for target' –> click on 'Output'–>Tick on 'create Hex file'–>click on OK –> click on build

9. The hex file store at the location of your project in the object folder.

10. Use SST flash software to burn hex file in SST89E516RD2 microcontroller trainer kit.

CONCLUSION:

for Ceneating and vizualication of segments we create a code first and then implement this on keil software & see output at flash.

Button Interfacing

Procedure:

1. Open MPLABX IDE and create a new project.

   File-> New Project->Microchip Embedded -> Standalone Project. Select device PIC18F4550. (simulator)

   Select compiler XC8.

   Project name ->

   Finish

   bootloader
   right click on name
   properties
   +cs linker
   Additional options
   codeoffset 800.

2. Create a C source file: Right click on the Source Files New -> C source file

   Give a name to the file.

3. Copy or type the source program in the C file created.

4. Save the file.

5. Right-click on the project name, and select "Clean and Build".

6. Open PIC Loader.

7. Press F3

8. Press Reset

9. Press F4

CONCLUSION :

In these practical we studied how to implement interfacing button relay (LED) & Buzzer.