

Making First Project Using LadderWORK

OBJECTIVE: MAKING AN LED BLINK CONNECTED AT PORT PIN OF GENERAL 8051 BASED MCU

ASSUMPTION:

- ✓ DOWNLOADED LADDERWORK EVALUATION SOFTWARE

PREPARATION:

- ✓ OPEN A NEW PROJET WITH NAME "BLINK"

WRITTEN BY: AJAY BHARGAV

DATED: 21ST Nov, 2006

REVISION CODE: 1.0

Now we are ready to start with the first step to LadderWORK software.

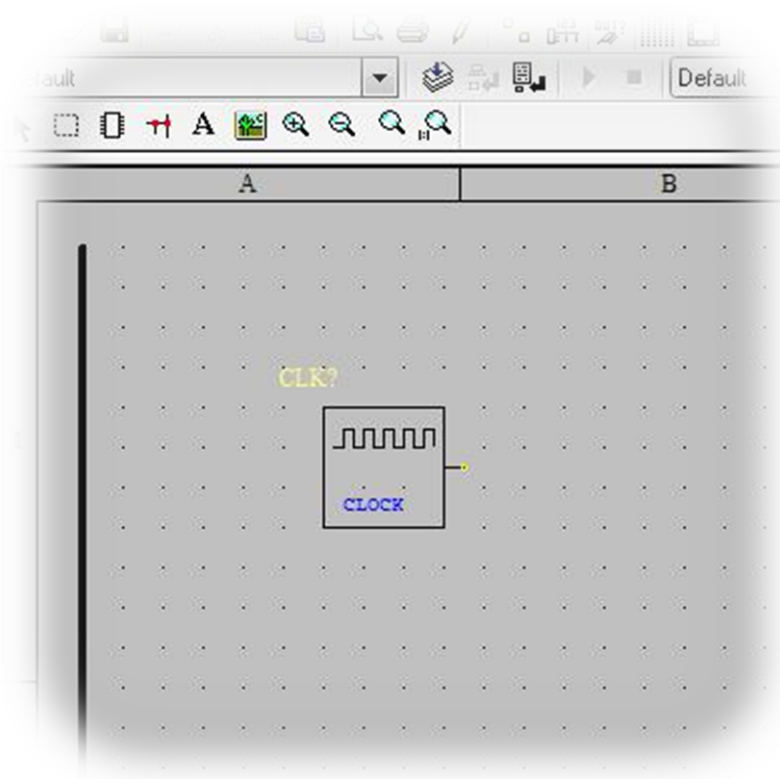
Step1: Adding Components:

In ladderWORK IDE you see a component bar on the right side, which has a number of components to select from.

As we are going to make an LED blink so there must be something which can give u a square wave output. That means “A Clock”.

In the component bar select the clock component and u see component attached to your cursor. Now place the component on your worksheet wherever you want.

So you now have something like this on your worksheet.



Step 2: Editing System Configuration:

Before we setup the clock parameters we first need to set the system configuration.

Go to Option menu → System Edit

Now you see the System I/O Module Editor Dialogue.

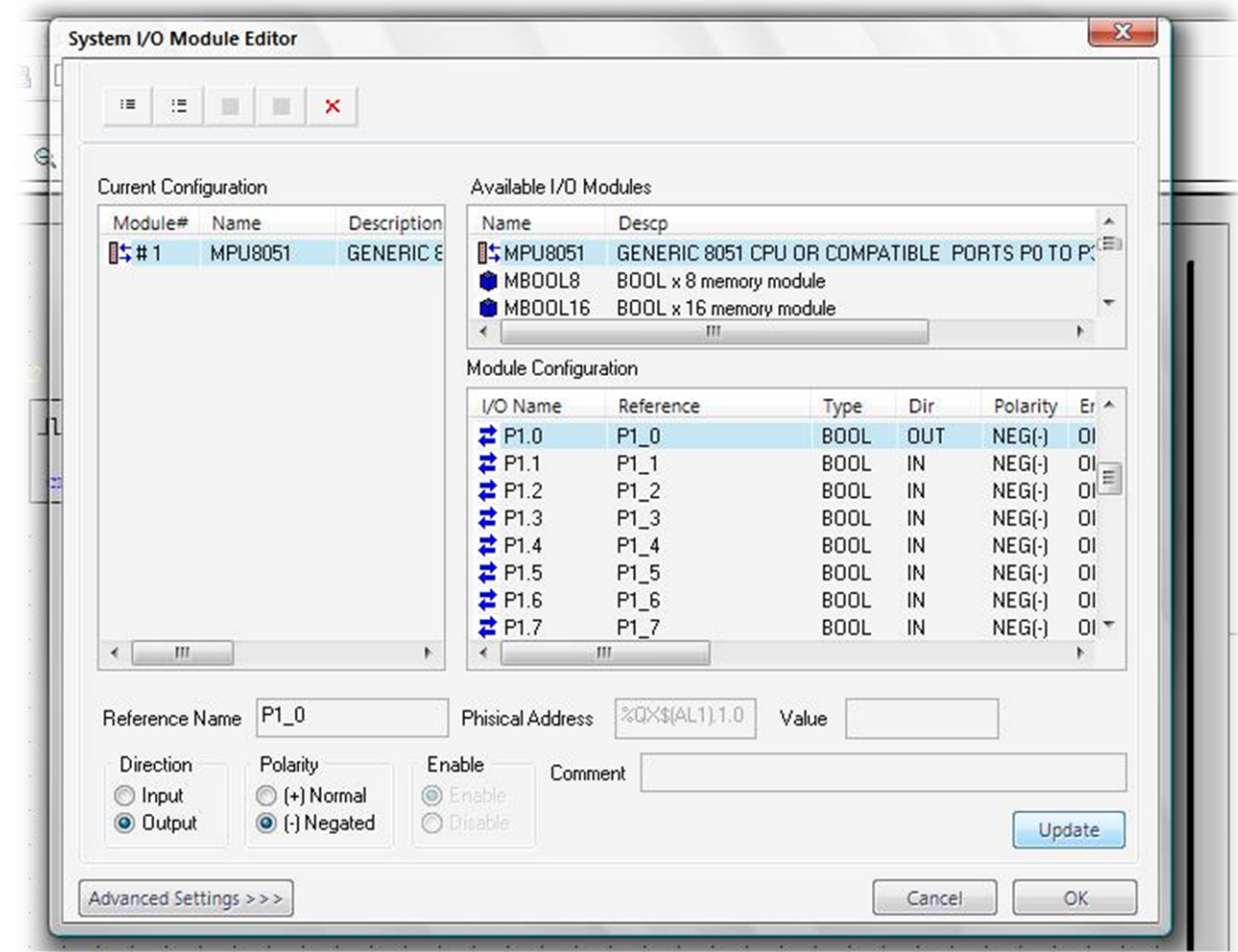
Here we define the configuration for the MCU we are going to use and also configure the port where we are going to blink our LED.

In our case we will select the "MPU8051"

For selecting, Double click on the MPU8051 for general 8051 CPU core with P0 to P3.

So now you have MPU8051 in your current configuration section.

Now selecting that module you can edit the Module configuration. Here you can define the ports to be Input or Output, Logic High or Logic low etc.

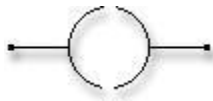


As we are going to use Port1 Pin 0 for our LED. Select P1_0 from the module configuration block. Change the direction of the port pin to Output and click update button.

Now you have configured the system for your LED. Click ok and exit from the system I/O module editor.

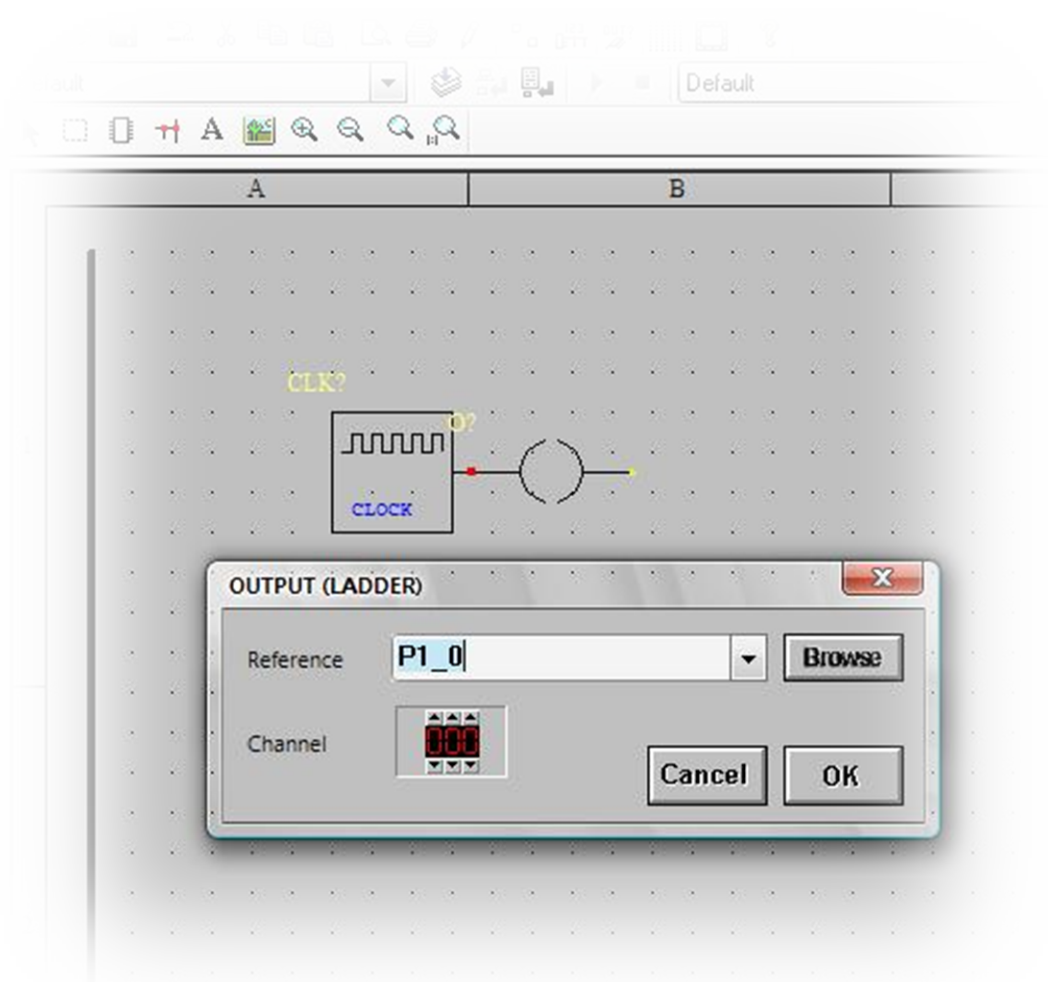
Step 3: Adding Output Port:

To add an output port select the output from the right component bar and add it in the worksheet.



now added another component in your worksheet. Double-Click on the output component and change its Parameters.

You will see a window like this



From the Drop down reference box, select the port where u want the output be active. As in our case its P1.0 so select P1_0 from the drop down.

Click ok and now connect the other pin of the output component to ground. Selecting ground component from the right component bar.

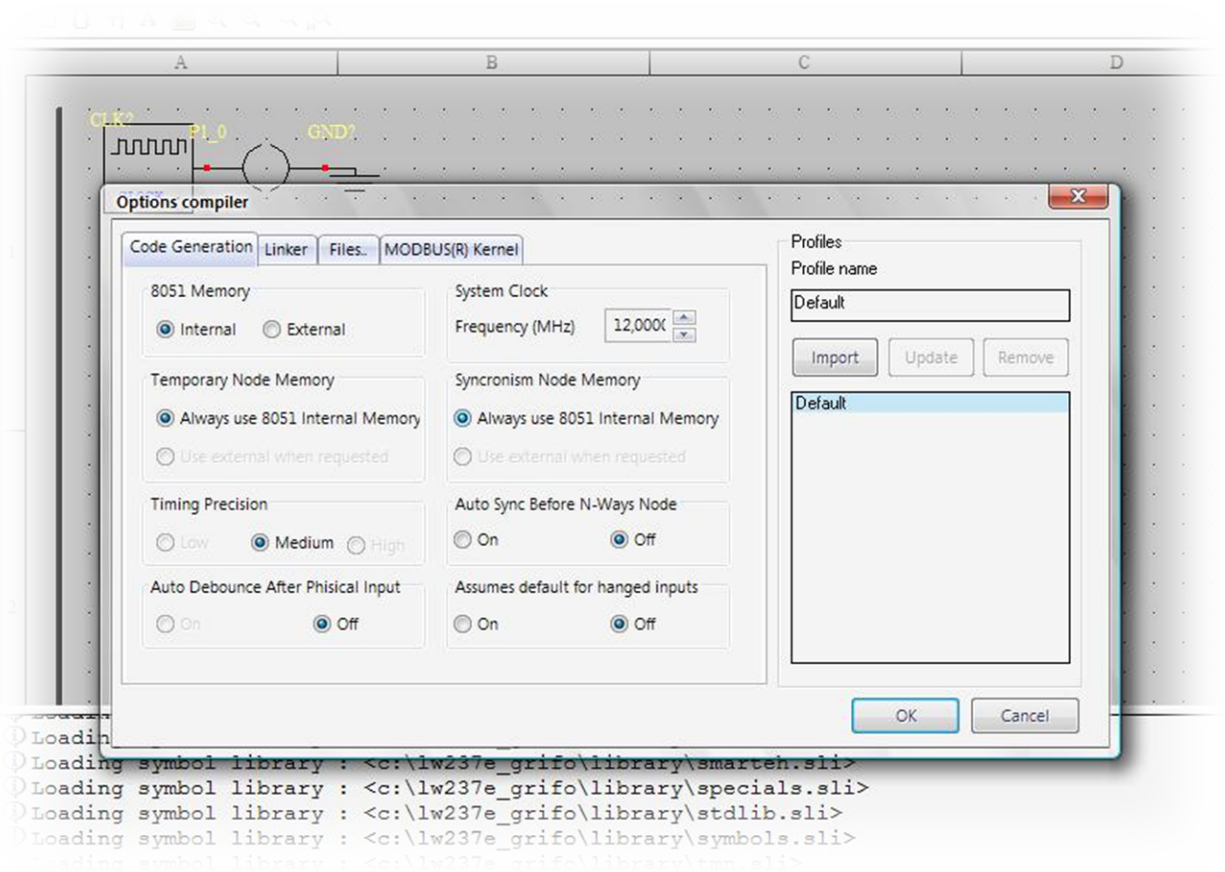
Set the frequency of clock by double clicking the clock component. Enter the value of frequency you want. Then click ok.

Now our design is complete.

Step 4: Setting Compiler option:

Go to Option menu → Compiler

Now to get the hex file ready we need to compile and build our Ladder Design. For that, First we need to configure the compiler options, to tell the compiler how to MAP the code and data segment.



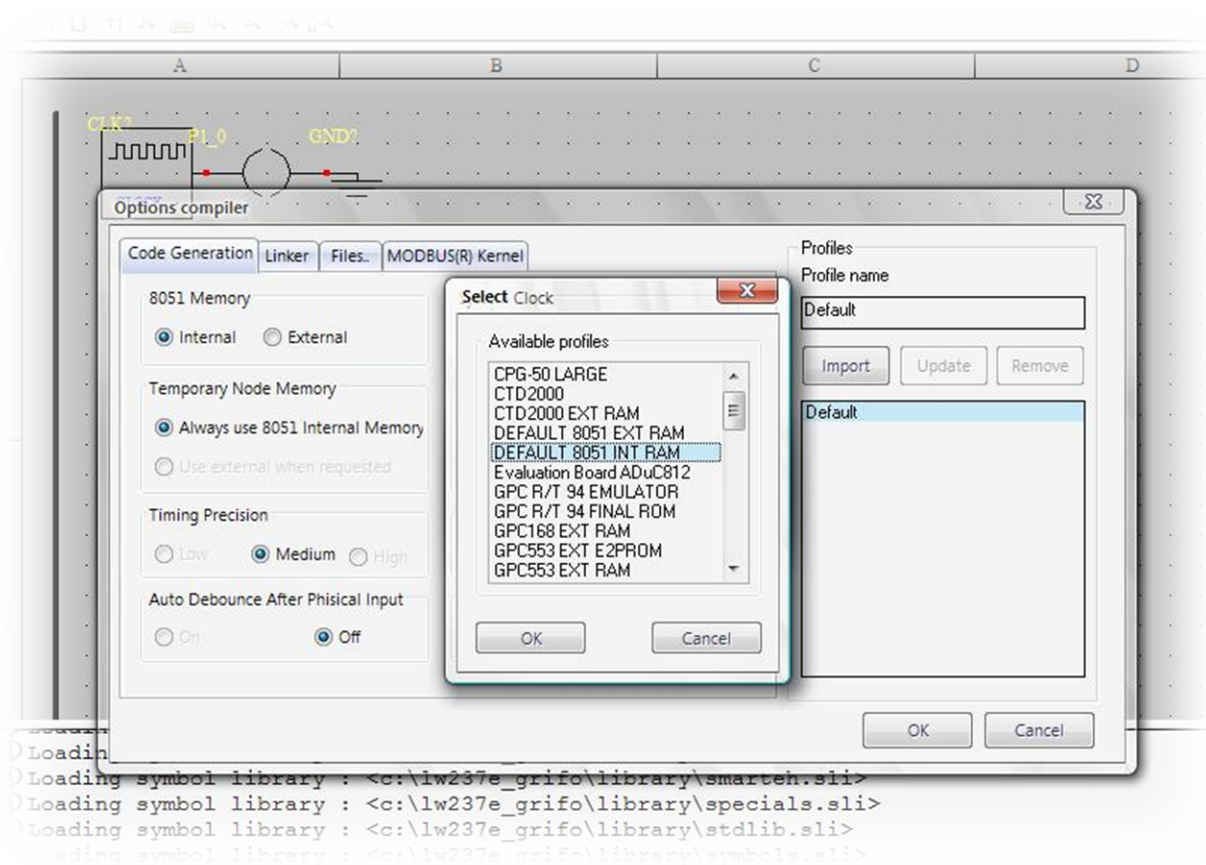
Set the configuration according to your board. We don't need much ram for our project so select the following configuration for our project:

- ✓ 8051 Memory: Internal
- ✓ Temporary Node Memory: Always use 8051 Internal Memory
- ✓ System Clock: Set what you are using on your board (12.000MHz in our case)

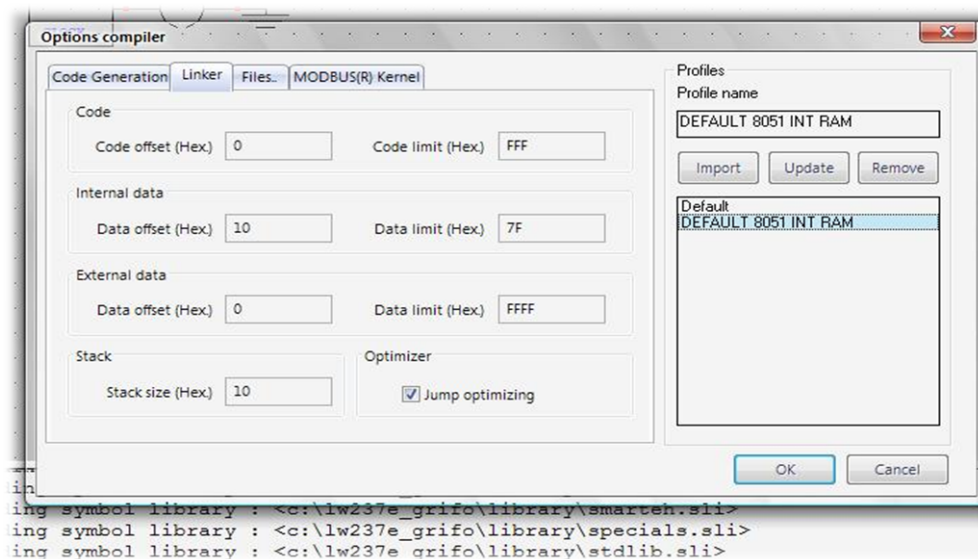
Now next we need to import profile for our project. In the Profiles block.

Click on the "Import" button. You will see a Select profile Dialogue box.

Select profile "Default 8051 INT RAM" from the Available profiles and click ok button to exit.



Now lets configure the linker MAP. Click on linker tab in same Compiler option window and set the linker configuration, How you want to map your output code against the memory.



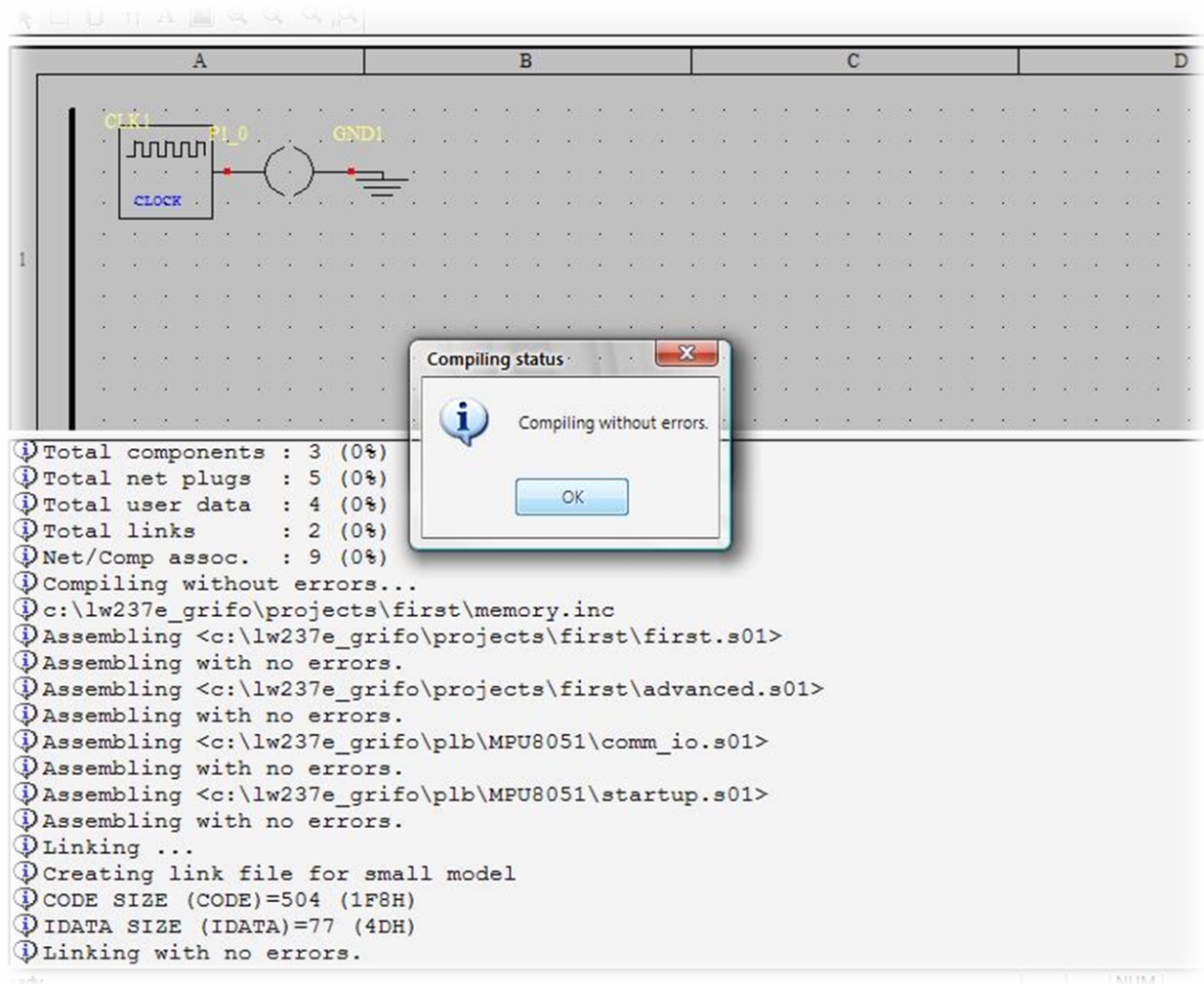
After this click ok. To exit the compiler configuration window. Now we are ready to build our project.

Step 5: Building Project:

To build the project

Go to Build → Compile

After the successful compilation of the design u see this prompt.



In the project directory where we have created our project you will find Blink.HEX file.

Load the hex file on to your target board and Run!