

Experiment #8

Introduction to Proteus



Birzeit University

Faculty of Engineering and Technology

Department of Electrical and Computer Engineering

Objectives

The main objective of this experiment is getting familiar with Proteus tool by creating a project from scratch and simulating a simple circuit that contains an 8086 processor with a PPI.

Introduction

Proteus is a simulation and design software tool developed by **Labcenter Electronics** for Electrical and Electronic circuit design.

Creating a New Project

To start the software, click on the Start button and select Programs, Proteus 8 Professional and then the Proteus 8 application. The main application will then load and run and you will be presented with the Proteus home page.

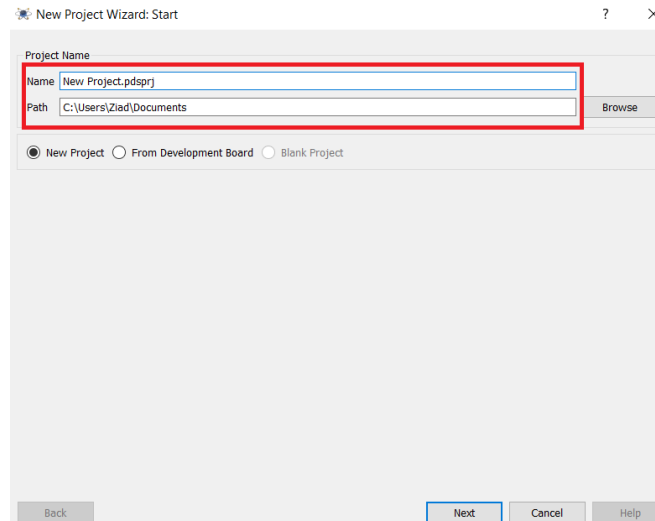


In order to create a schematic, we must first create a project as shown in the following steps:

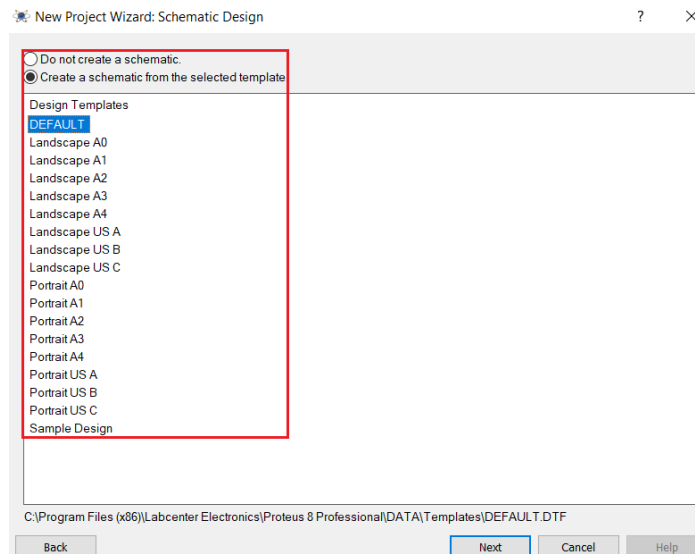
- 1- Start by pressing the new project button near the top of the home page in Proteus



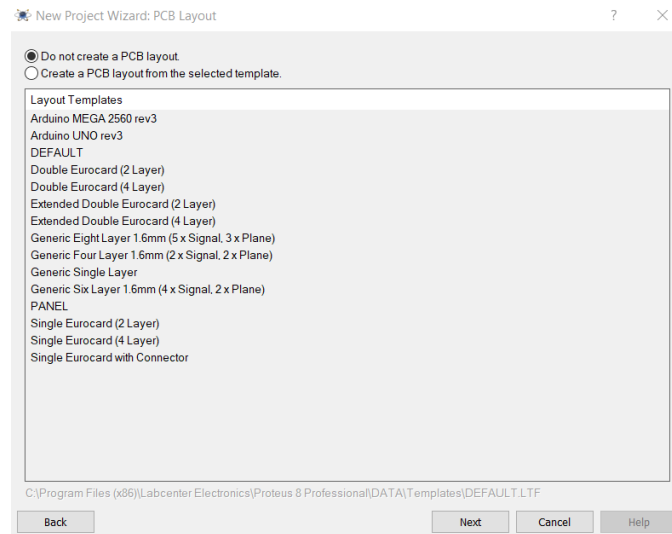
- 2- On the first page of the wizard specify a name and path for the project.



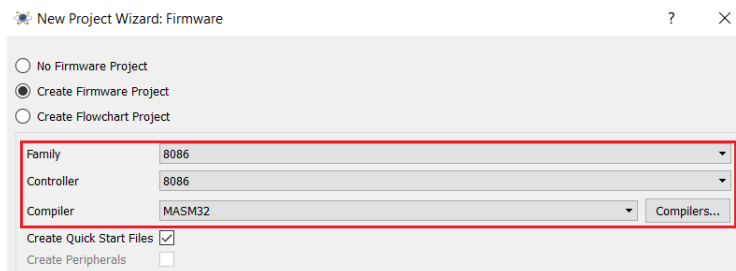
- 3- Since we need a schematic, we choose one of the provided templates. In most of the cases, we use the **DEFAULT** one.



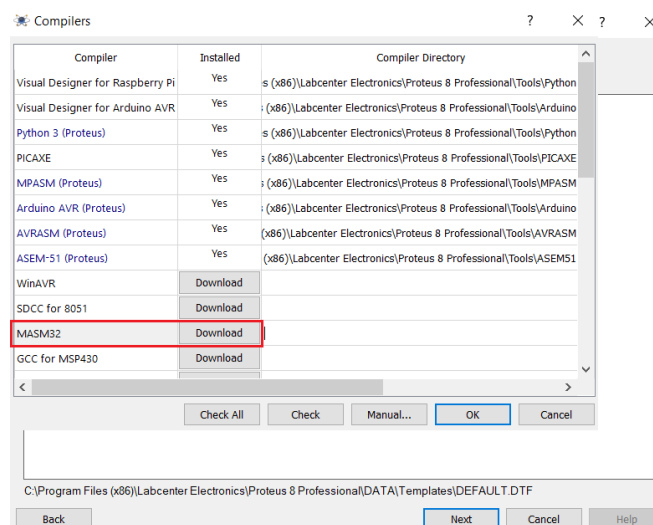
- 4- If a PCB layout is needed, you can use a specific template. If not, you can skip this step by choosing **Do not create a PCB layout**.



- 5- Since We will program an 8086 processor with a PPI using assembly language, we choose the following options.

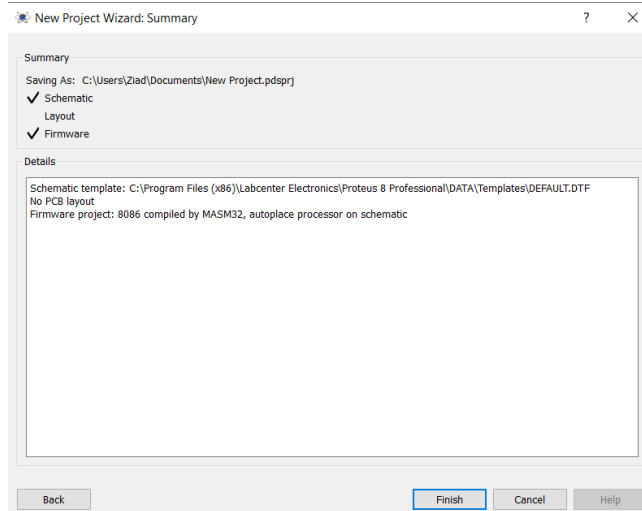


If MASM32 compiler is not configured, click on **compilers** button, the following window will appear:

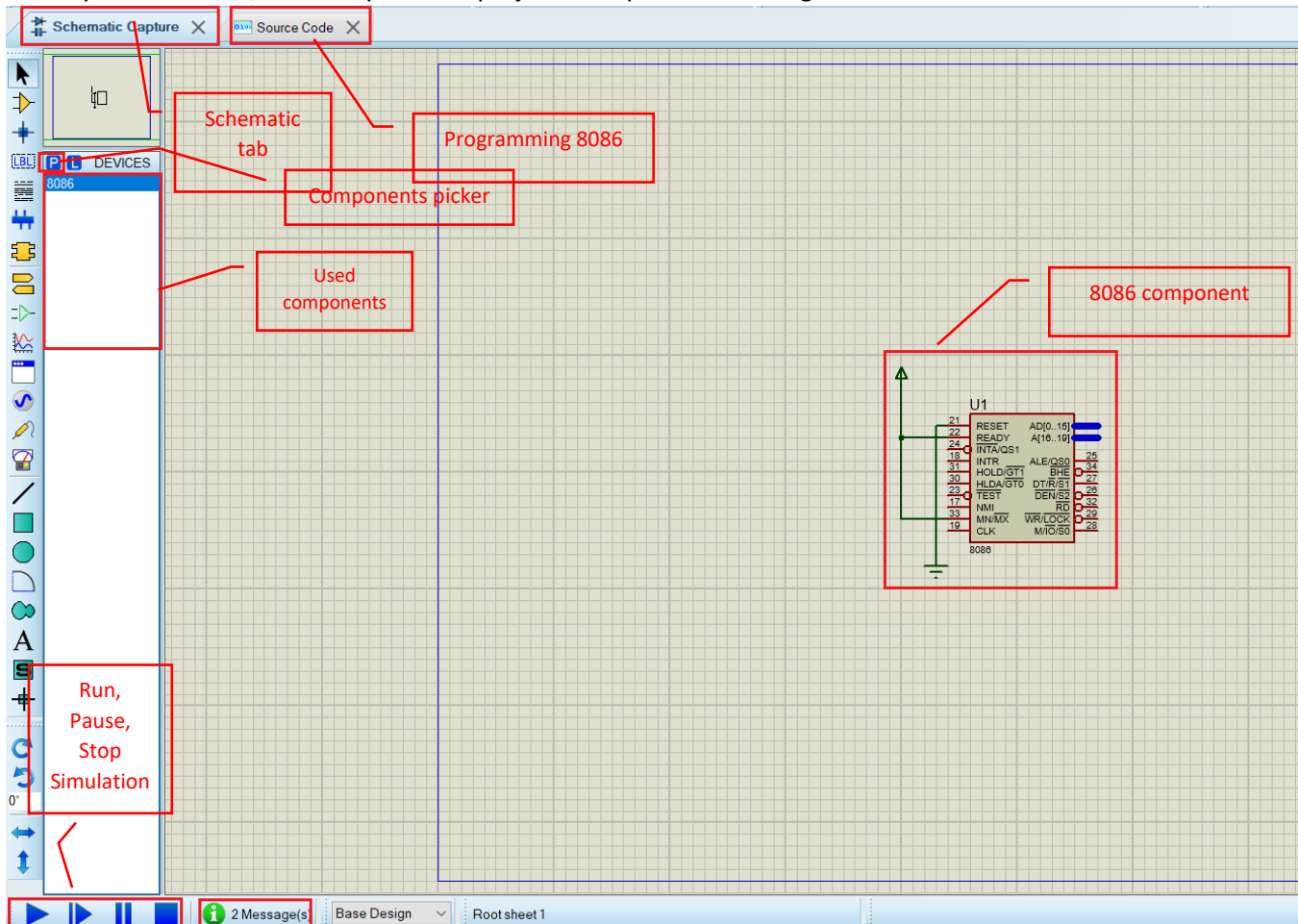


You can download it after making sure that you've started Proteus in **Admin mode**.

- 6- At the end, there will be a summary for the created project as following:



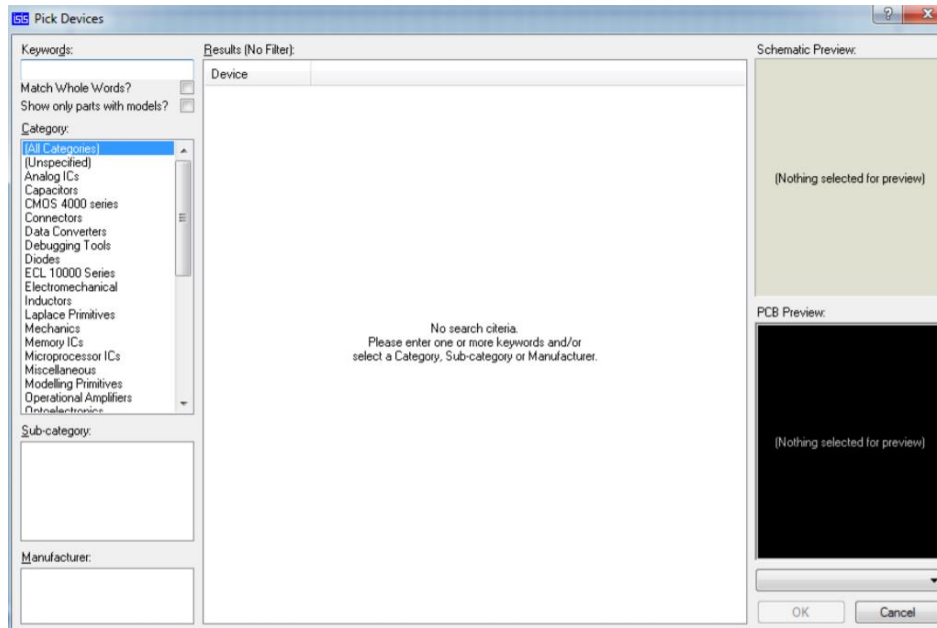
When you click **finish**, the newly created project will open as following:



After that, you're ready to implement your circuit by placing all the required components and route the wires.

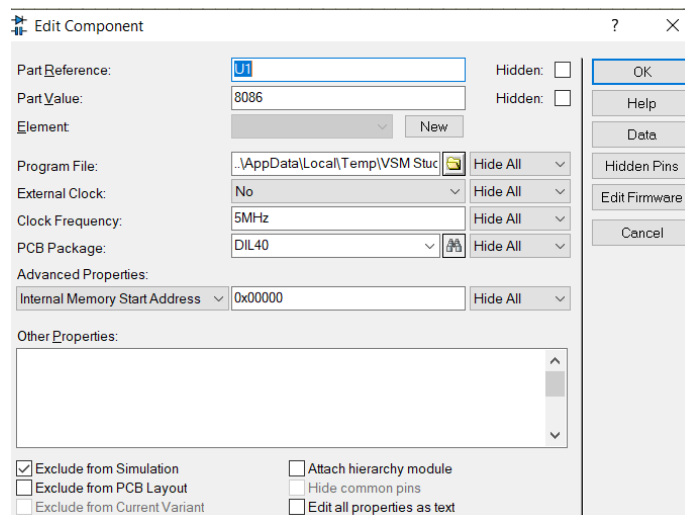
Components Picker

You can select your component From the **Library** menu select **Pick part** or from **object selector** window click on **P**. The following screen will be shown:



Select the components from categories or type the part name in Keywords text box. The selected components will appear in the devices list. Select the component and place it in the design sheet by left-click.

Double click on the component to edit the properties of the components and click on Ok. For example, if you double click on the 8086 component, you get the following screen:



Controlling simple DC motor using PPI

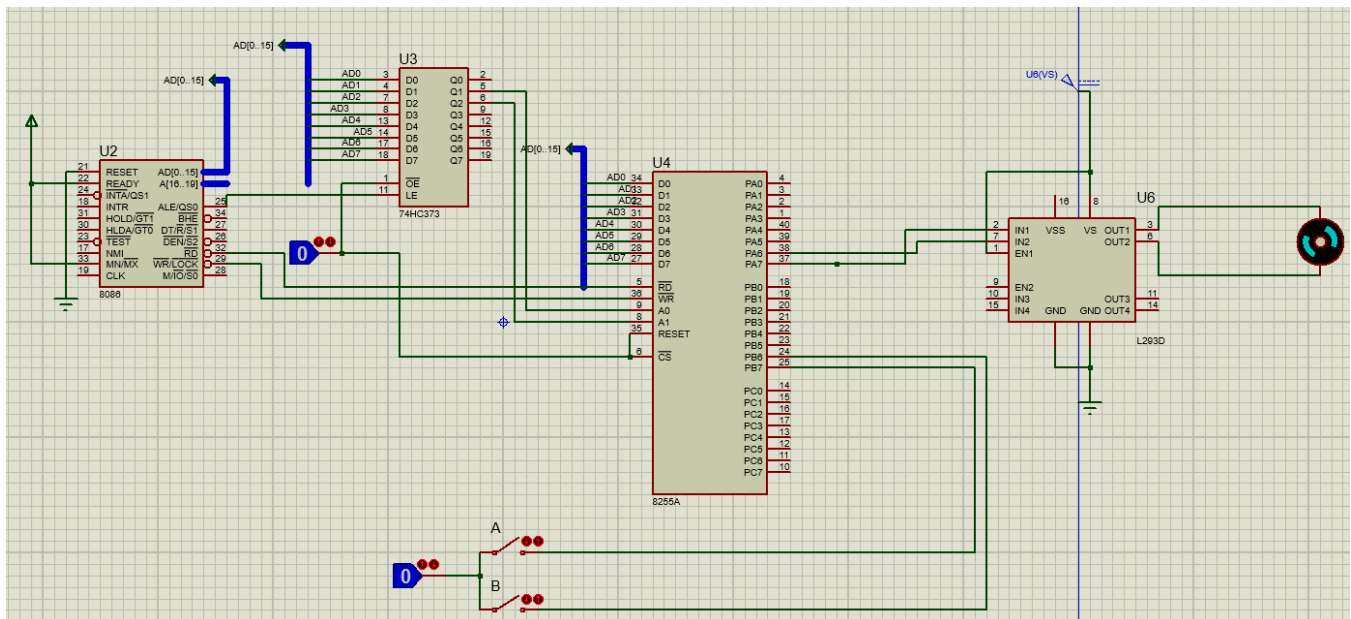
In this part, we control the rotation direction of the motor by using a PPI. We define **PORT B** as **input** while **PORT A** as **output**.

PB₆ & PB₇ will be used to read the switches values in order to determine the direction of rotation.

PA₆ & PA₇ will send the values to the driver to execute the rotation direction on the motor according to the values read from **PB₆ & PB₇**. **Required components**

- 8086 processor
- 8255A [PPI]
- MOTOR
- L293D [Driver]
- 74HC373
- SWITCH
- LOGICSTATE

I. Schematic



II. Code

```
CODE SEGMENT PUBLIC 'CODE'
ASSUME CS:CODE
START:
MOV DX, 0746H
MOV AL, 82H
PORTB EQU 0742H
PORTA EQU 0740H
PORTC EQU 0744H
OUT DX, AL
STT:
MOV DX, PORTB
IN AL, DX
MOV DX, PORTA
OUT DX, AL
MOV DX, PORTC
MOV BL, AL
NOT AL
OUT DX, AL
JMP STT
CODE ENDS
END START
```

III. Run Simulation

Run the simulation and try the following combinations on Switch A & B then determine the rotation direction of the motor.

A on, B off => Motor Direction: _____

A off, B on => Motor Direction: _____

References

- <https://labcenter.s3.amazonaws.com/downloads/Tutorials.pdf>