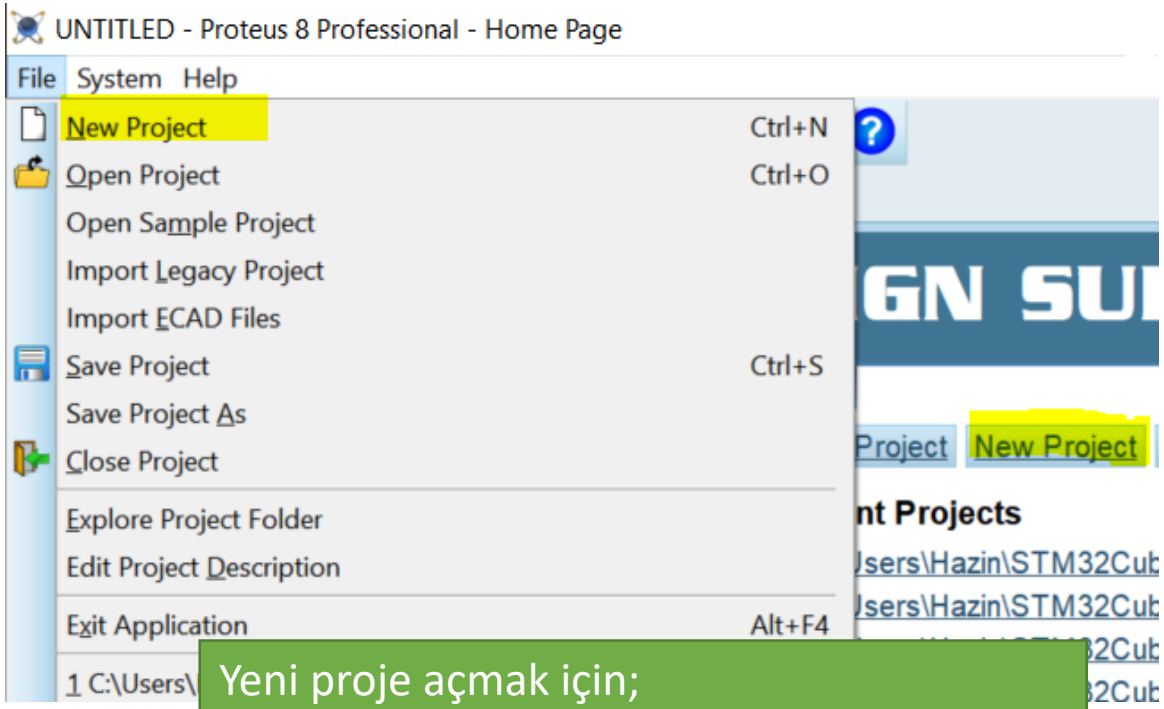
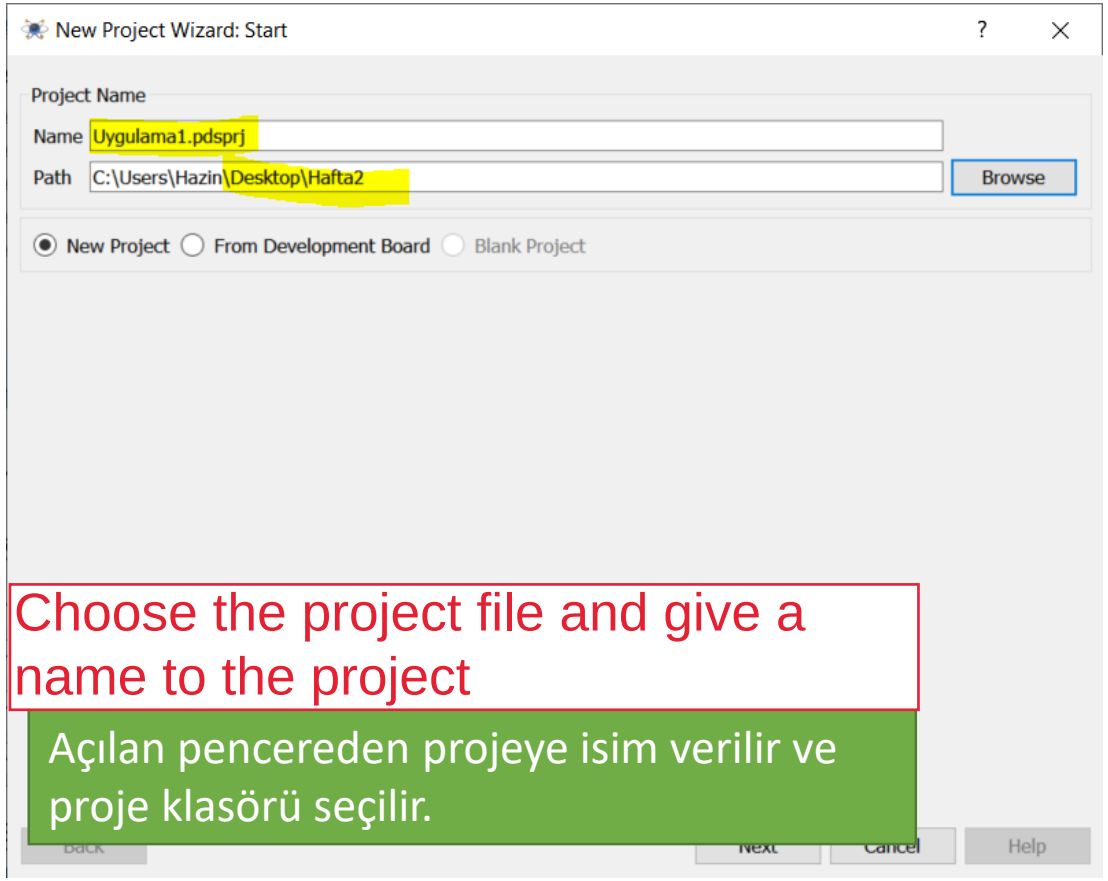
Proteus 8.10

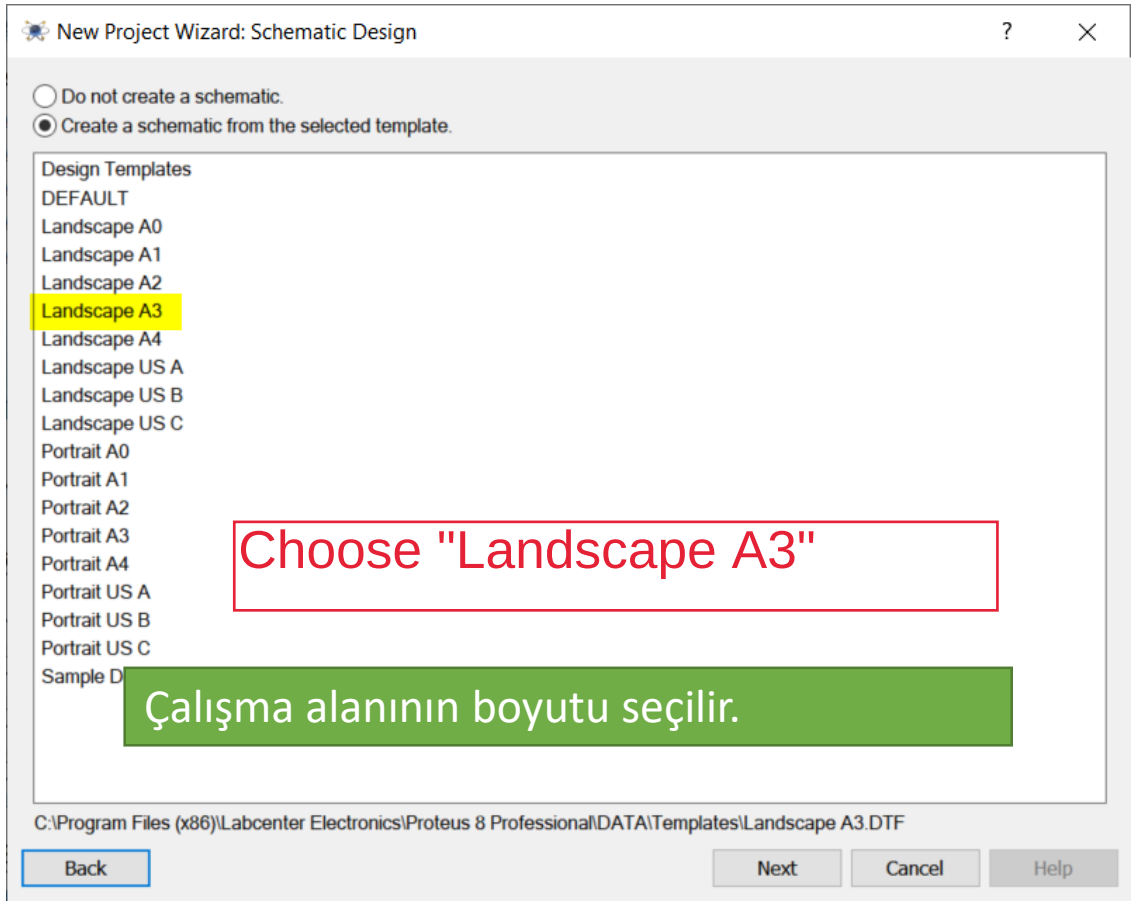


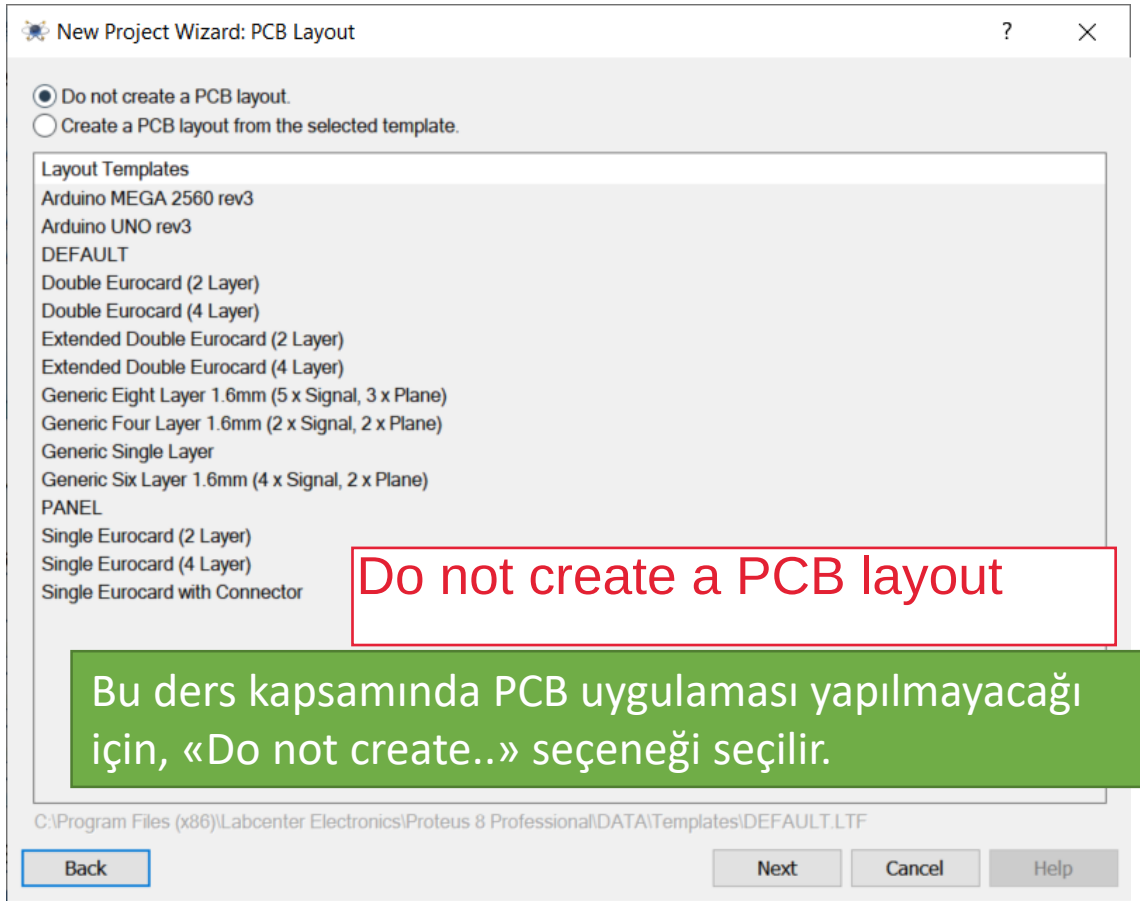
Proteus 8.10


UNTITLED - Proteus 8 Professional - Home Page
File System Help
New Project Ctrl+N
Open Project Ctrl+O
Open Sample Project
Import Legacy Project
Import ECAD Files
Save Project Ctrl+S
Save Project As
Close Project
Explore Project Folder
Edit Project Description
Exit Application Alt+F4
1 C:\Users\
Project
New Project
nt Projects
Yeni proje açmak için;
File-> New Project
To open new project;
File-> new project

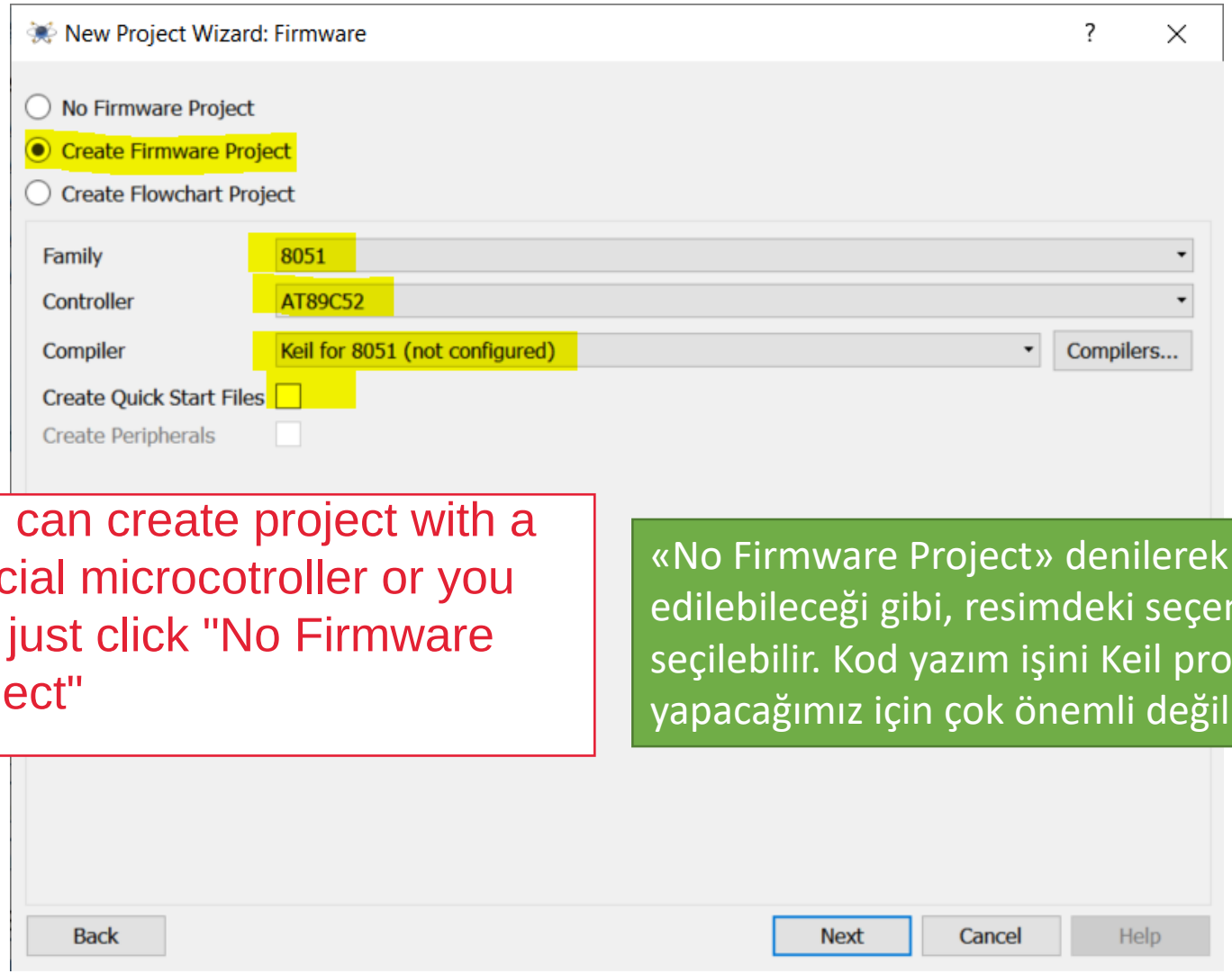

New Project Wizard: Start
Project Name
Name Uygulama1.pdsprj
Path C:\Users\Hazin\Desktop\Hafta2
Browse
New Project
From Development Board
Blank Project
Choose the project file and give a name to the project
Açılan pencereden projeye isim verilir ve proje klasörü seçilir.
Back
Next
Cancel
Help

Proteus 8.10


New Project Wizard: Schematic Design
Do not create a schematic.
Create a schematic from the selected template.
Design Templates
DEFAULT
Landscape A0
Landscape A1
Landscape A2
Landscape A3
Landscape A4
Landscape US A
Landscape US B
Landscape US C
Portrait A0
Portrait A1
Portrait A2
Portrait A3
Portrait A4
Portrait US A
Portrait US B
Portrait US C
Sample D
Choose "Landscape A3"
Çalışma alanının boyutu seçilir.
C:\Program Files (x86)\Labcenter Electronics\Proteus 8 Professional\DATA\Templates\Landscape A3.DTF
Back
Next
Cancel
Help


New Project Wizard: PCB Layout
Do not create a PCB layout.
Create a PCB layout from the selected template.
Layout Templates
Arduino MEGA 2560 rev3
Arduino UNO rev3
DEFAULT
Double Eurocard (2 Layer)
Double Eurocard (4 Layer)
Extended Double Eurocard (2 Layer)
Extended Double Eurocard (4 Layer)
Generic Eight Layer 1.6mm (5 x Signal, 3 x Plane)
Generic Four Layer 1.6mm (2 x Signal, 2 x Plane)
Generic Single Layer
Generic Six Layer 1.6mm (4 x Signal, 2 x Plane)
PANEL
Single Eurocard (2 Layer)
Single Eurocard (4 Layer)
Single Eurocard with Connector
Do not create a PCB layout
Bu ders kapsamında PCB uygulaması yapılmayacağı için, «Do not create..» seçeneği seçilir.
C:\Program Files (x86)\Labcenter Electronics\Proteus 8 Professional\DATA\Templates\DEFAULT.LTF
Back
Next
Cancel
Help

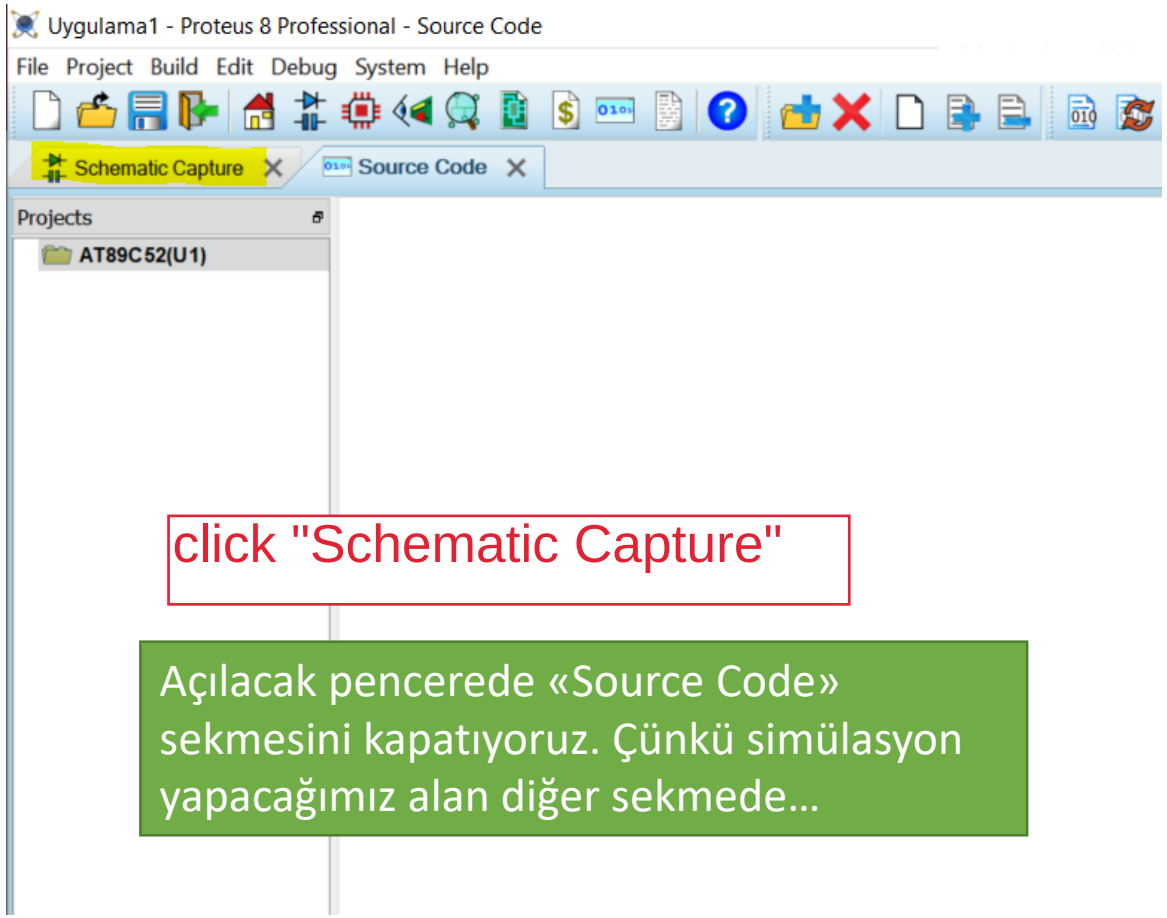
Proteus 8.10



You can create project with a special microcotroller or you can just click "No Firmware Project"

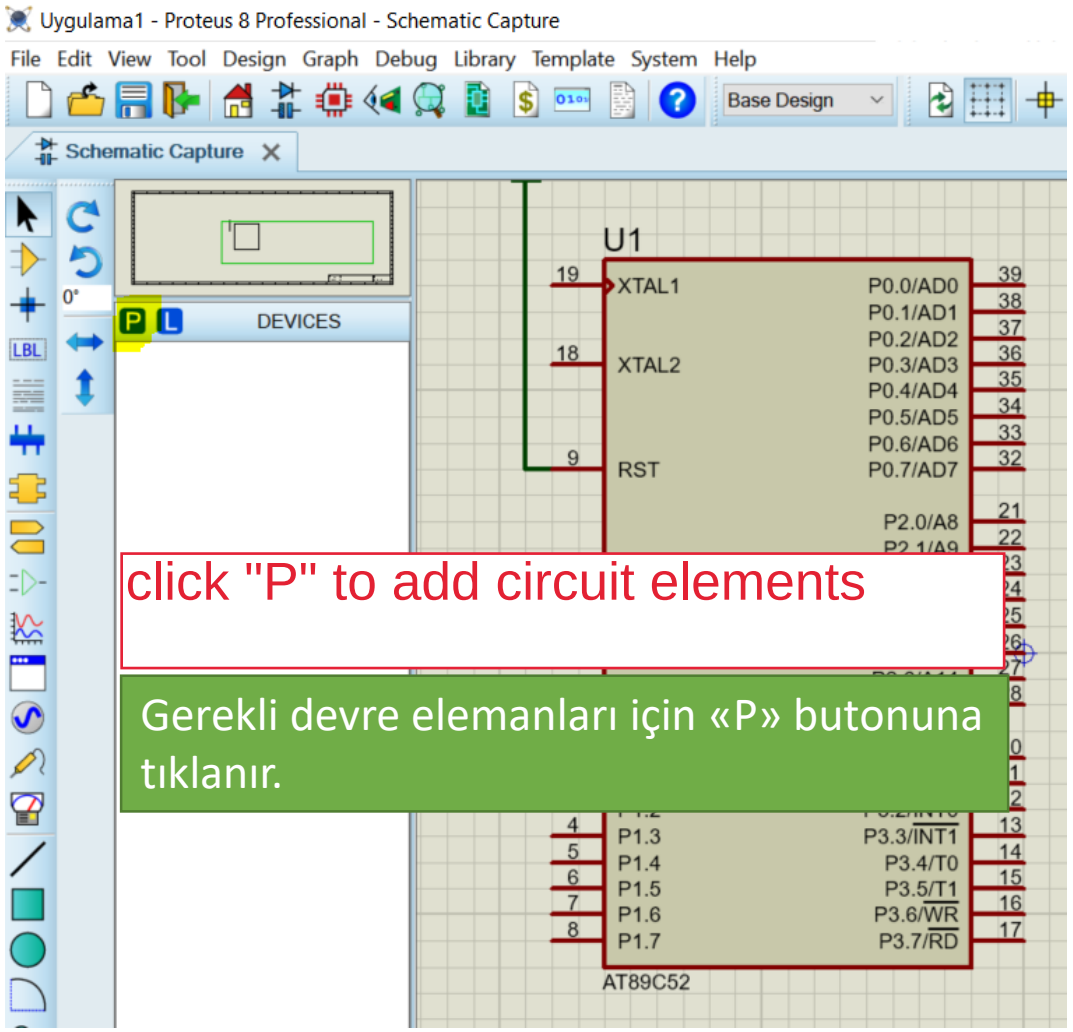
«No Firmware Project» denilerek devam edilebileceği gibi, resimdeki seçenekler de seçilebilir. Kod yazım işini Keil programında yapacağımız için çok önemli değil.

Proteus 8.10



click "Schematic Capture"

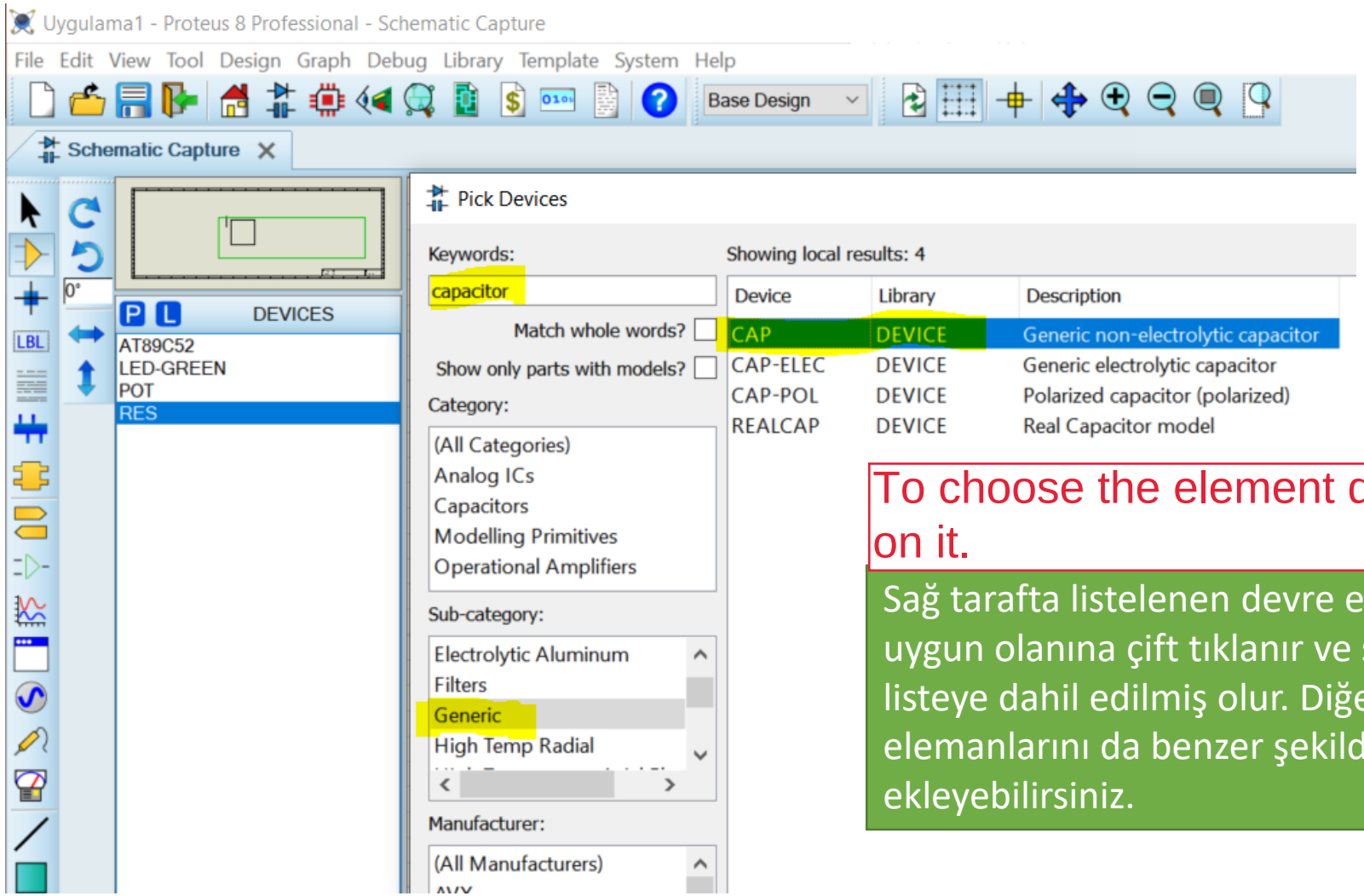
Açılacak pencerede «Source Code» sekmesini kapatıyoruz. Çünkü simülasyon yapacağımız alan diğer sekmede…



click "P" to add circuit elements

Gerekli devre elemanları için «P» butonuna tıklanır.

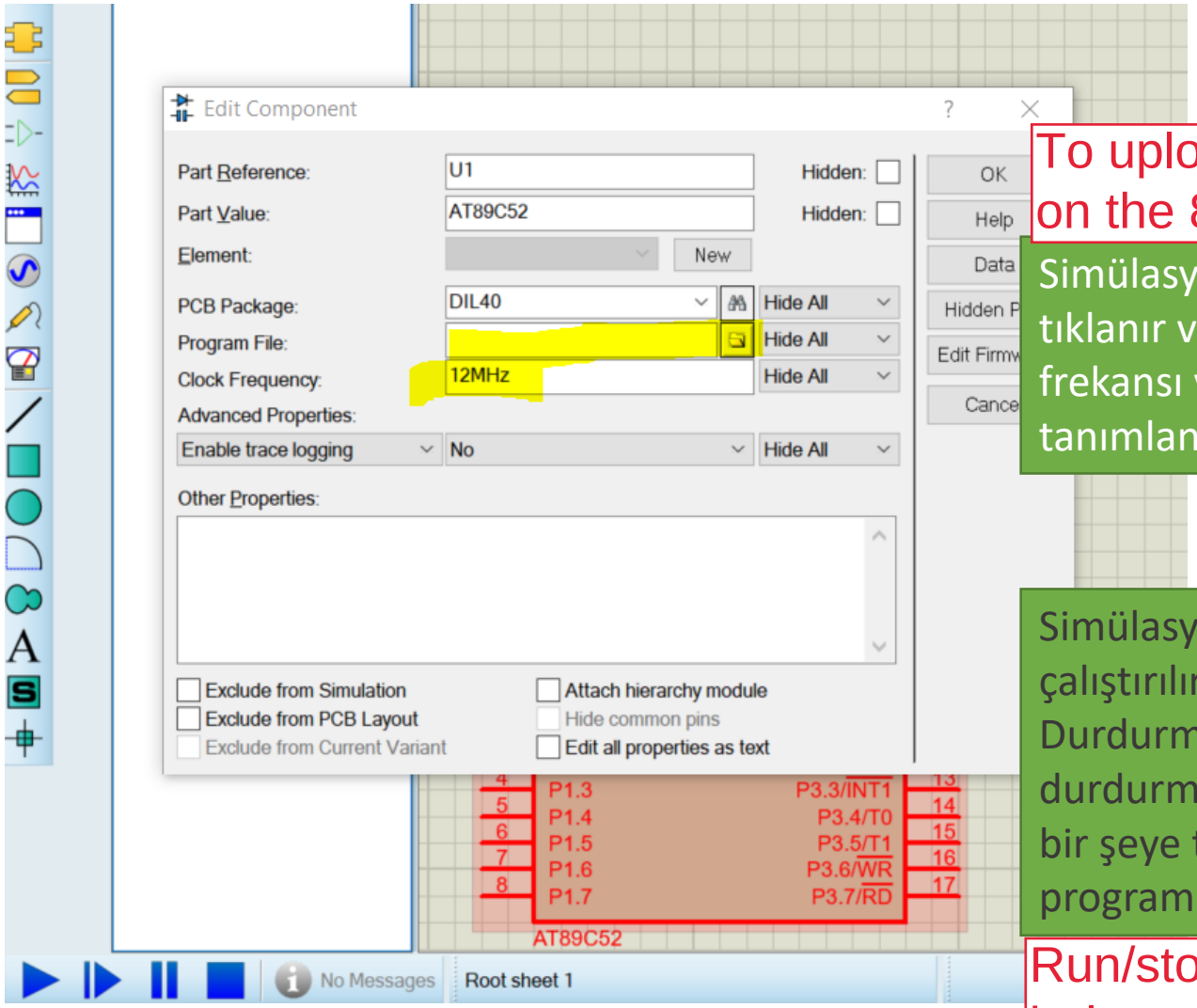
Proteus 8.10



To choose the element double click on it.

Sağ tarafta listelenen devre elemanlarından uygun olanına çift tıklanır ve sol taraftaki listeye dahil edilmiş olur. Diğer devre elemanlarını da benzer şekilde arayıp, ekleyebilirsiniz.

Proteus 8.10



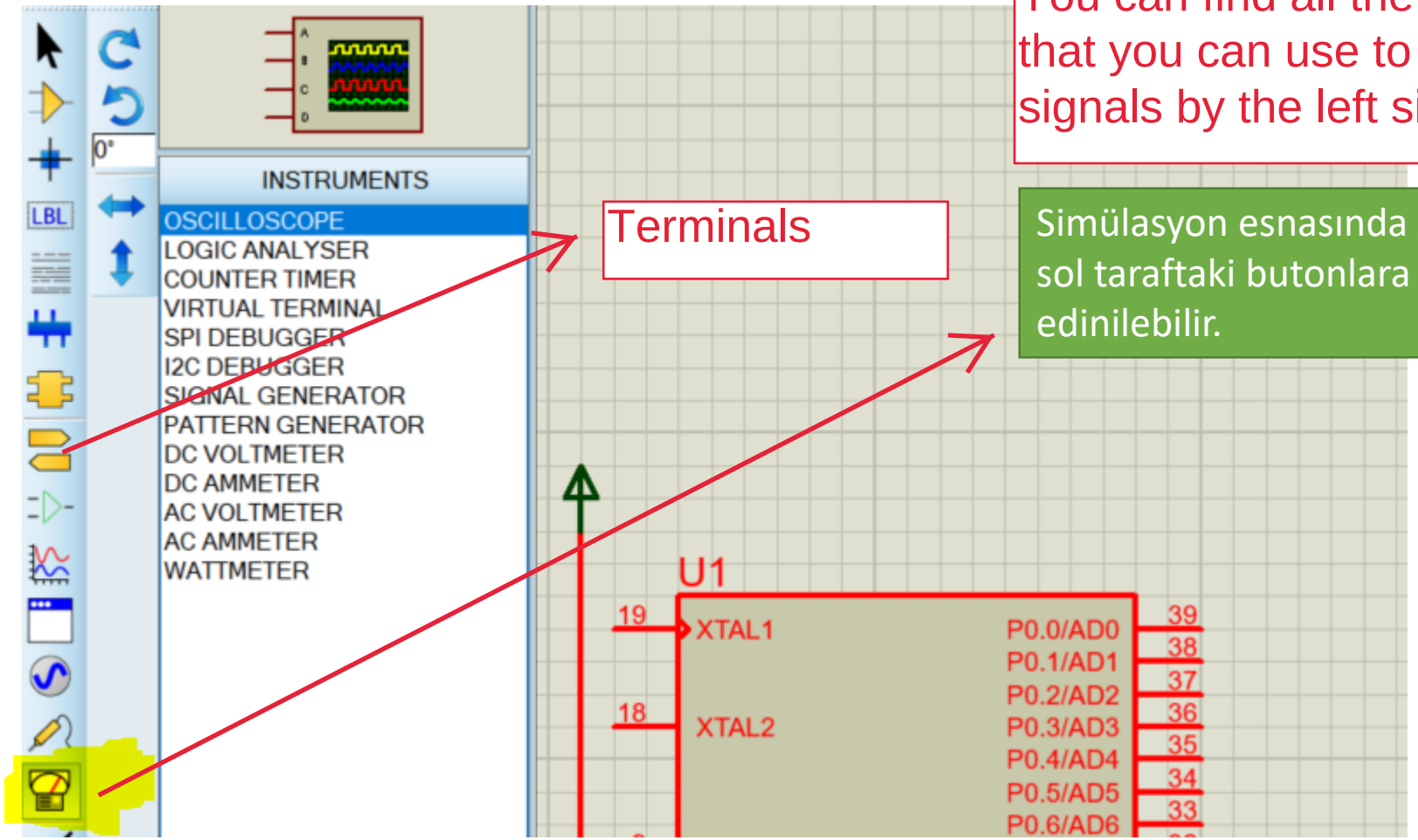
To upload the .hex file double click on the 8051 microcontroller

Simülasyonu çalıştırmadan önce 8051'e çift tıklanır ve açılacak pencerede, çalışılan saat frekansı ve yazılan kodun HEX dosyası tanımlanır.

Simülasyon sol alttaki «Run» butonu ile çalıştırılır, «Stop» butonu ile de durdurulur. Durdurma butonuna basıldığında, durdurma işlemi gerçekleşmeden herhangi bir şeye tıklamamalısınız. Aksi halde program hata verip kapanabilir.

Run/stop buttons are on the left below side.

Proteus 8.10


INSTRUMENTS
OSCILLOSCOPE
LOGIC ANALYSER
COUNTER TIMER
VIRTUAL TERMINAL
SPI DEBUGGER
I2C DEBUGGER
SIGNAL GENERATOR
PATTERN GENERATOR
DC VOLTMETER
DC AMMETER
AC VOLTMETER
AC AMMETER
WATTMETER
Terminals
You can find all the other devices that you can use to measure/shov signals by the left side buttons
Simülasyon esnasında gerekli bazı cihazlar sol taraftaki butonlara tıklanarak edinilebilir.
U1
19
XTAL1
18
XTAL2
P0.0/AD0
P0.1/AD1
P0.2/AD2
P0.3/AD3
P0.4/AD4
P0.5/AD5
P0.6/AD6
39
38
37
36
35
34
33