

PW ADVANCED POWER ELECTRONICS

PW N°0

Initiation to matlab/simulink Using the Power Electronics Toolbox

General objectives :

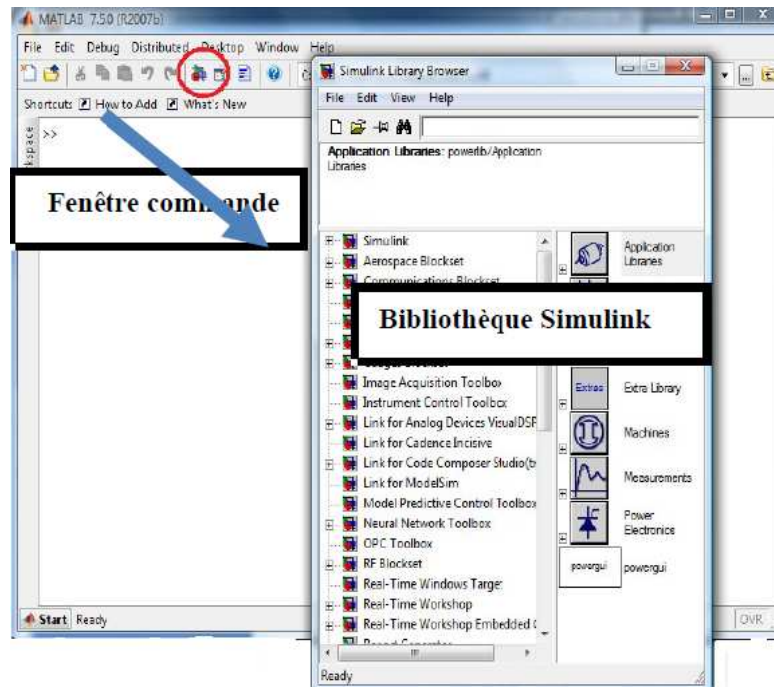
Through this practical work you will be able to:

- understand the design stages with matlab/simulink.
- use the electronic power toolbox: SimPowerSystems.
- configure a power component.
- draw the characteristic $V=f(I)$ of a diode.

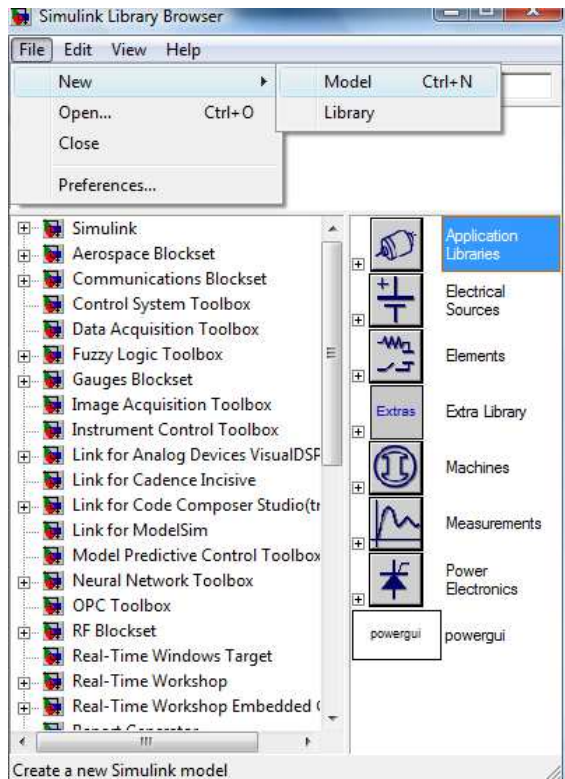
I- Design of a single-wave rectifier with matlab/simulink

1. Launch matlab by clicking on the icon located in the desktop; the command window is displayed.

2. On the taskbar, click on the icon  , the simulink library is displayed



PW ADVANCED POWER ELECTRONICS



3. To open a file, click on File ⇒ New ⇒ model (you can click on which is located in the taskbar of the Simulink library window)

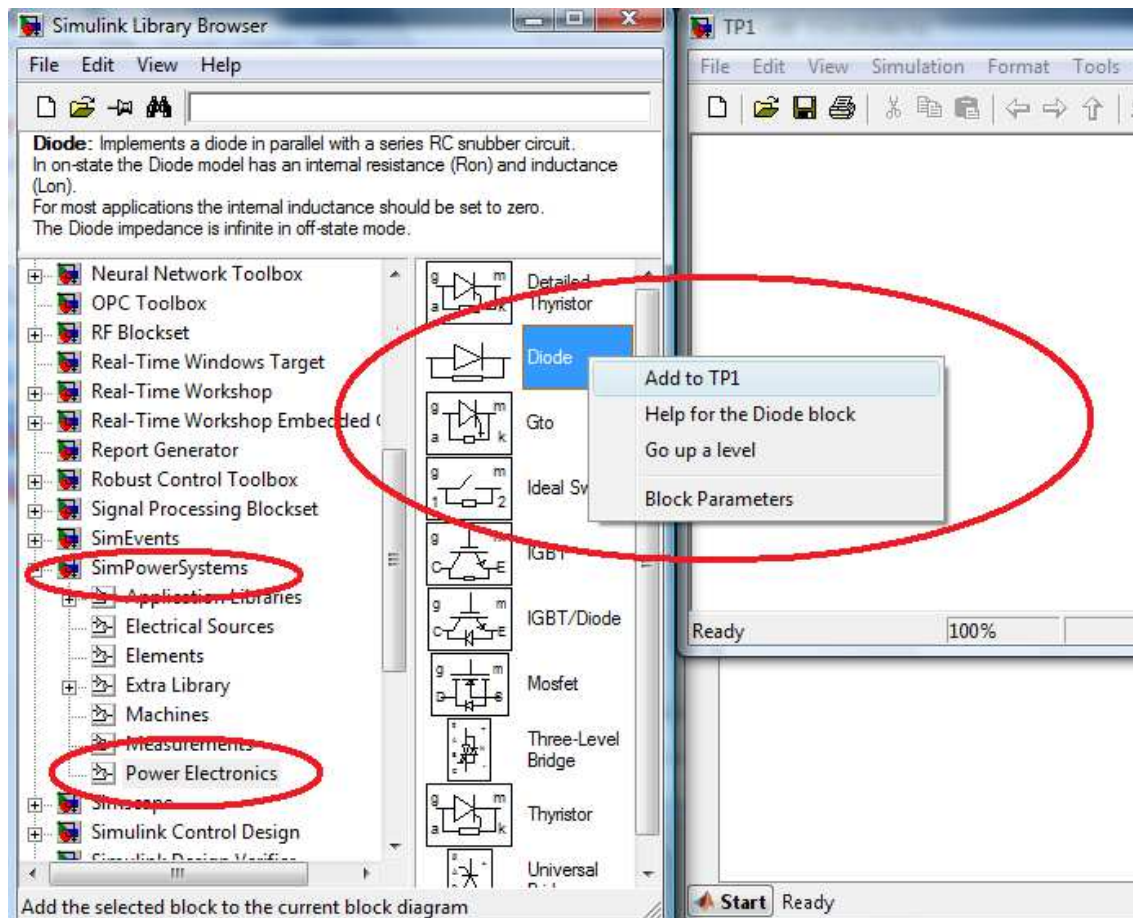
4. Save under the name TP1.



PW ADVANCED POWER ELECTRONICS

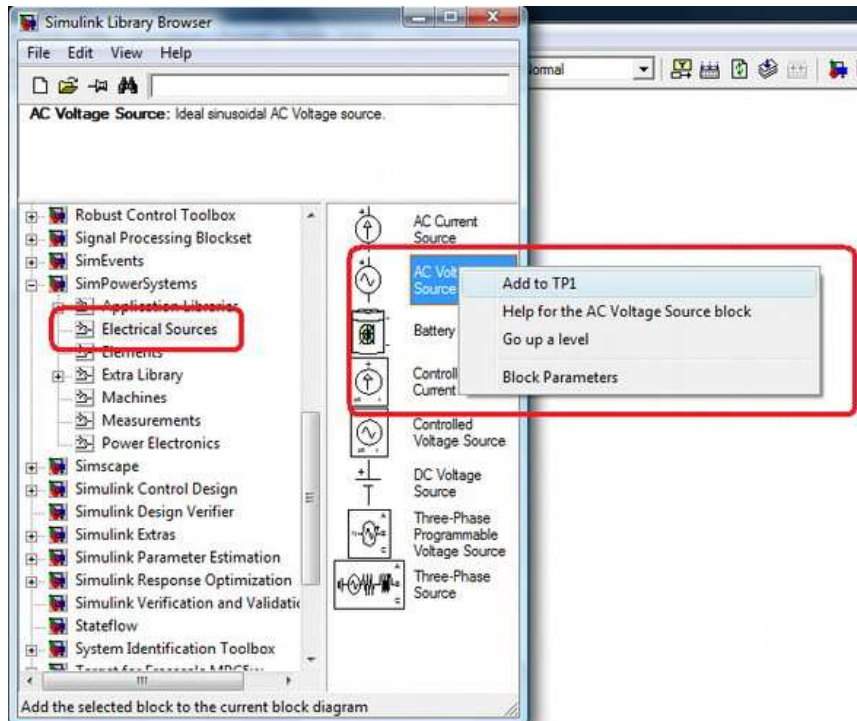
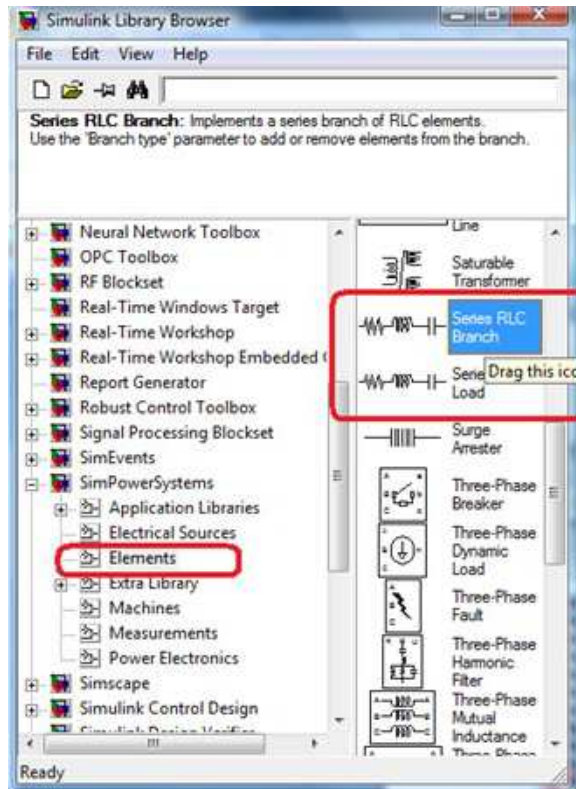
5. Use the scroll bar to locate **SimPowerSystems**; Click on it then choose **Power Electronic** ⇒ **Diode**.

6. Click on the right mouse button then **Add to TP1** (you can drag and drop the diode)





PW ADVANCED POWER ELECTRONICS

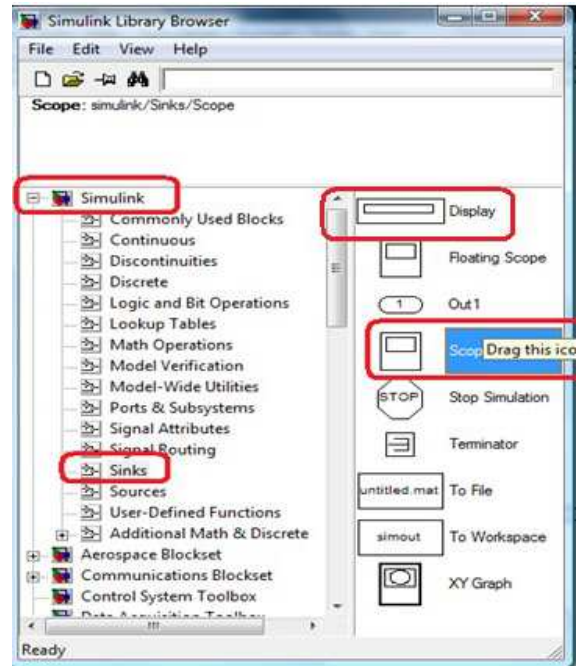
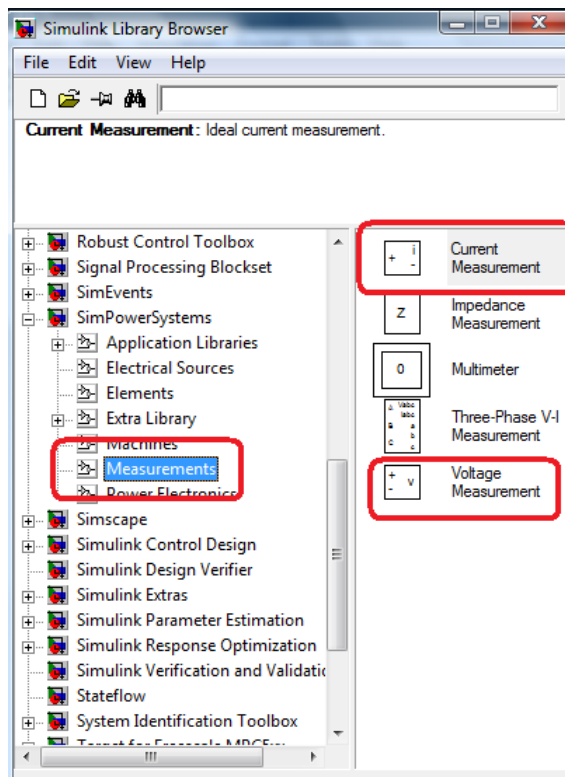




PW ADVANCED POWER ELECTRONICS


7. The diode is displayed; double click on the component, **the parameter block** is displayed;
8. To add resistance; repeat step 5 by choosing **Series RLC Load**
9. Double click on the component, **the parameter block** is displayed; Choose $R=15\Omega$
10. To add a voltage source, in **SimPowerSystems**; choose **Electrical sources** \Rightarrow **AC voltage source**.
11. Click on the right mouse button then **Add to TP1**
12. Double click on the component, the parameter block is displayed; enter the following values:
 - Amplitude =100.
 - Phase=0.
 - Frequency=60Hz.
13. To add measuring devices, in **SimPowerSystems**; choose **Measurements** \Rightarrow **AC current Measurements** then **voltage Measurements**.
14. Click the right mouse button then **Add to TP1**.

PW ADVANCED POWER ELECTRONICS



15. Use the scroll bar to locate **Simulink**; Click on it then choose **Sinks** ⇒ **Scope**.

16. Click the right mouse button then **Add to TP1**.

17. Double click on the device then on the icon  and enter 3.

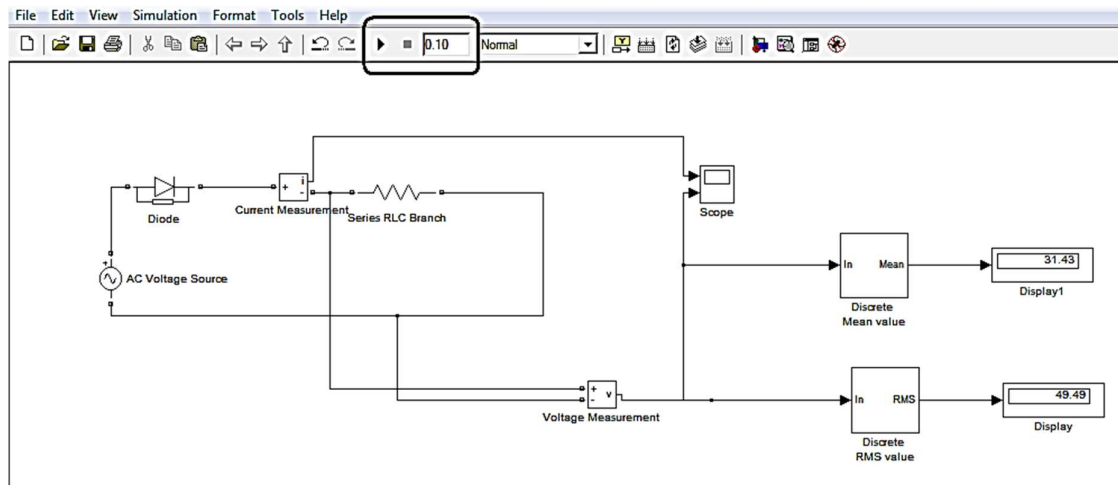
18. Use the scroll bar to locate **Simulink**; Click on it then choose **Sinks** ⇒ **Display**.

19. Click the right mouse button then **Add to TP1**.

20. To display the average and effective value, click on **SimPowerSystems**; choose **Extra Library** ⇒ **Discrete Measurements** ⇒ **Discrete Mean value and Discrete RMS value**.

21. Connect all components as follows :

PW ADVANCED POWER ELECTRONICS

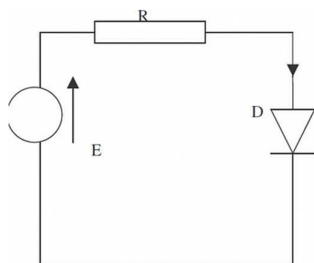


22. Before running the simulation for a time of 0.1 sec, choose the ODE23TB solver in the Simulation parameter \Rightarrow parameter \Rightarrow Solver \Rightarrow ODE23TB.

23. Choose a diode with a threshold voltage of 400 and visualize the signals.

II-Handling

Carry out the following assembly:



With $R = 50\Omega$

Complete the following table:

E	-5	-4	-3	-2	-1	0	0.5	1	1.5	2	2.5	3	3.5	4
Vd														
id														

Draw the characteristic of the diode: $i_d = f(V_d)$

Deduce from this characteristic the threshold voltage of the diode (V_0) and the dynamic resistance (r) in the on state.