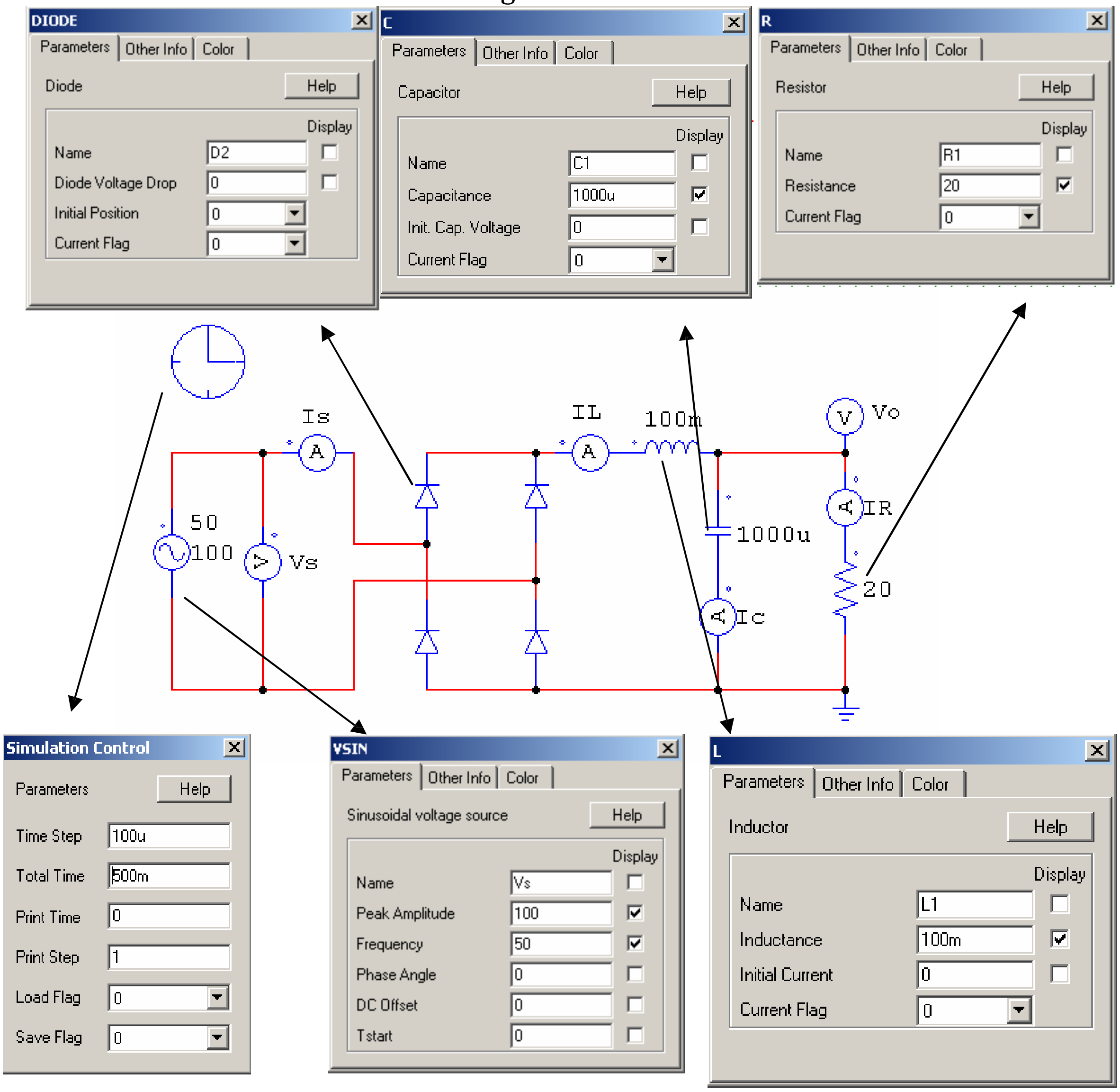
Roll. No:	Date:
Checked By:	Grade:

Power Electronics Laboratory-1

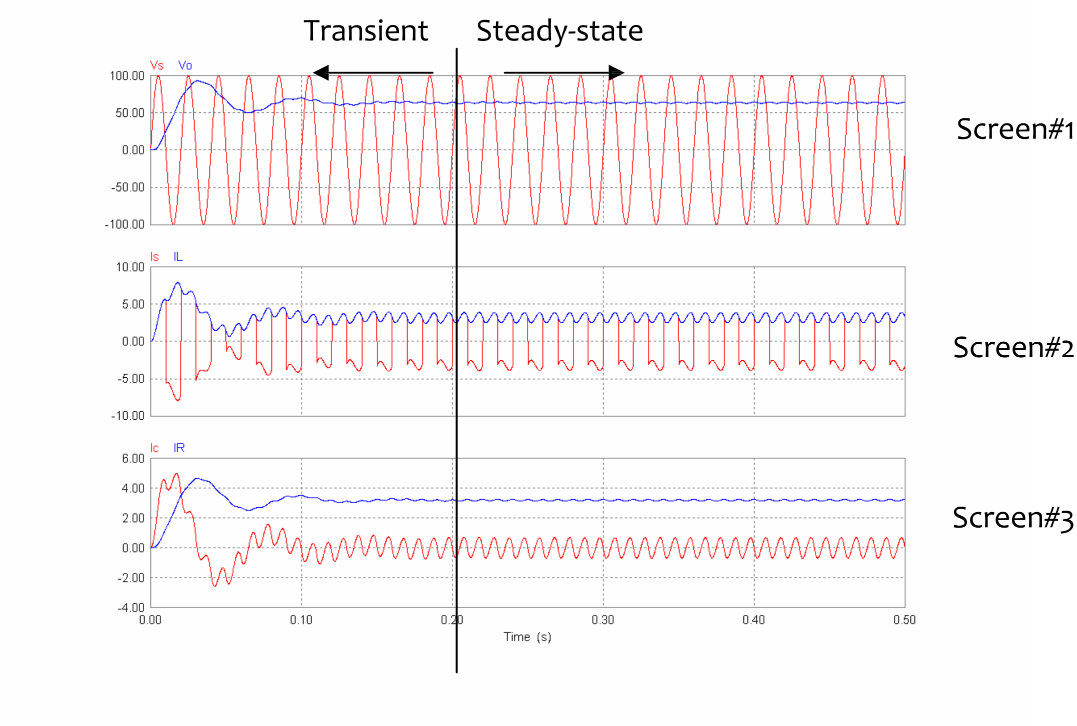
Introduction To PSIM

Create the circuit as shown below using PSIM software.

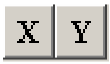
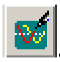


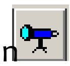
1. Run the simulation by selecting "Run Simulation" icon.
2. The window switches automatically to result screen after simulation is completed. Plot the waveforms of Vs and Vo on screen#1.
3. Add one more screen by selecting "Add Screen" icon. Plot Is and IL on screen#2 using "Add/delete curve" icon.
4. Add screen#3 to plot IR & Ic.

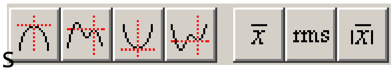
Now you should obtain a screen as shown in the figure below. The result is plotted from zero to 0.5s. In this circuit transient occurs at the beginning of simulation and reaches steady value at around 0.2s.



Measuring rms, average, min & max

All measurement must be performed at steady-state period of the simulation. Use axis setting icon  to show the steady period of the waveform. **It is very useful to show a complete cycle of waveform** when detail measurement is required such in FFT analysis. 3-5 cycles is preferred in most application. To return to the total simulation result, click on the Re-draw icon .

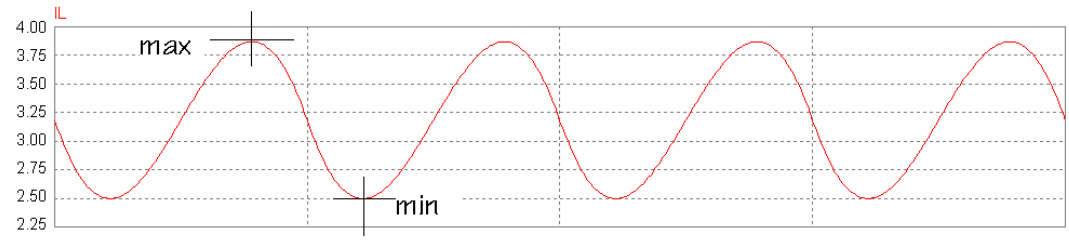
Click on the measurement icon  to activate measurement tools.

Use one of these icons  to measure average, rms, max, or min value of the waveform.

From the waveforms in simulation results, find the following values:

1. rms value of Vs and Is.
2. average value of Vo, IL and IR.
3. Peak value of Is.

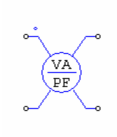
Measuring ripple



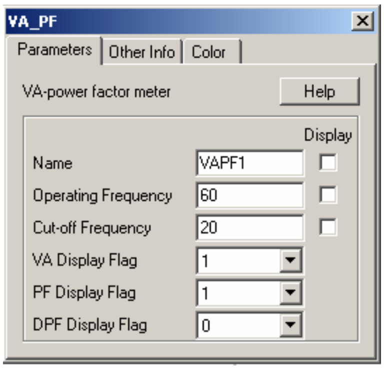
Obtain maximum & minimum values of waveform. The difference between max & min is the peak-peak ripple value. Find peak-peak ripple of Vo and IL from simulation.

Measuring power factor

PSIM provides pf meter in it library. It can be found in Element→Other→Probes→VAR/pf meter.



Insert this element between AC source and rectifier. The parameter of pf meter is as follow:



Operating frequency is set according to the frequency of ac voltage source.

Simulation Control Setting in PSIM

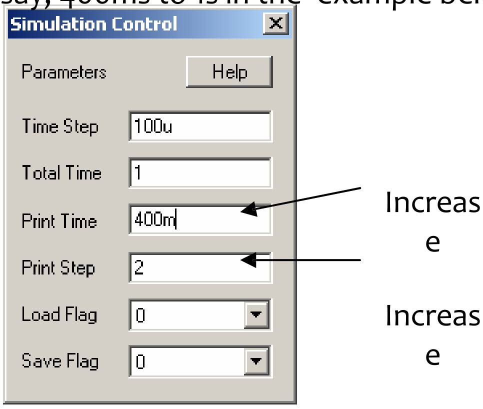
PSIM Demo version has limitation on:

1. Number of display/meter to maximum of 8.
2. Total result plotted on the screen is **limited to 6000 points.**

In this laboratory, all circuit has been designed to run with demo version. However, high resolution result is not possible. If the simulation result being plotted is more than 6000 points, a warning will be displayed.

Increasing the **Print step** in Simulation Control may overcome this problem with the price of lower resolution of the waveform.

Another way is by plotting the steady-state period of the waveform only, as long as transient period is not the aim of the analysis. This can be done by increasing **Print time**. Instead of from zero the result will be plotted from, let say, 400ms to 1s in the example below.



Example how to set Simulation Control parameters

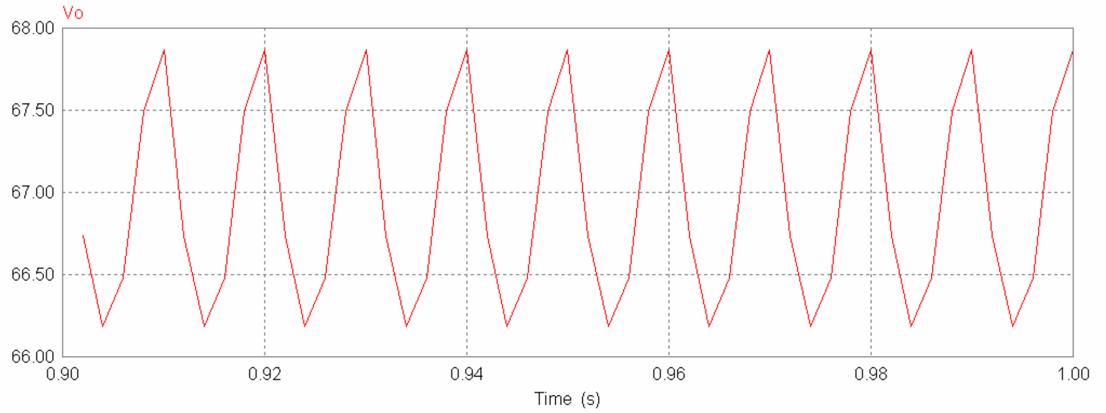
1. Identify the highest frequency of any sources in the circuit.
2. The maximum time step is <u>one tenth of it period by default</u>:

e.g. voltage source of 50Hz, gives 20ms period. **Time step** is 1/10 x 20ms = 2ms.

3. Estimate the time at which simulation reaches steady-state. For example 0.4s. Therefore 1s should give steady-state result. So the **Total time** is 1s.
4. Using maximum resolution **Print step** is set to 1.
5. Only showing the last 5 cycles of waveform by setting the **Print time** as:

Prin time = 1s – (5cy x 20ms) = 1000ms – 100ms = 900ms

The result is a coarse waveform.



Don't waste the 6000 points provided by PSIM. How?

6. Increase the resolution of simulation by reducing **Time step:**

Time step = 5 cycles / 6000 points = 100ms / 6000 = 16.66 us ~ round up to 20us.

Now the result has 100X higher in the resolution.