

Chapter 7: Simulation and co-simulation with others software

Using PSIM

Launch the software from the desktop icon.

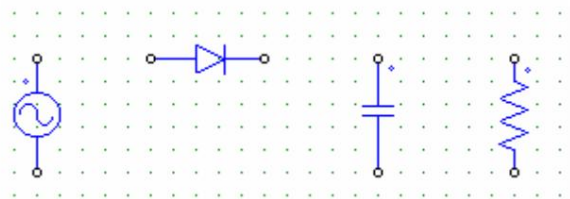


For a new job: **File**

New

Go to **File, Save As** to give it a name

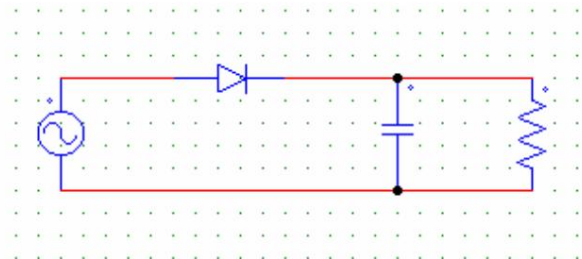
From the **elements** menu , place the following components:



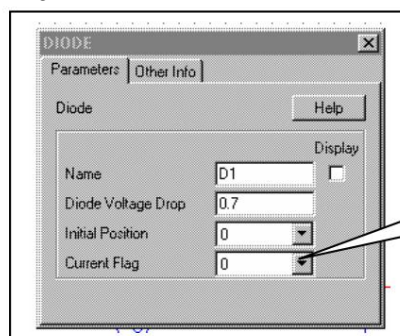
To rotate the components, right-click.

Pressing Esc changes the cursor into an arrow, allowing you to select a component to move or delete it. (Delete)

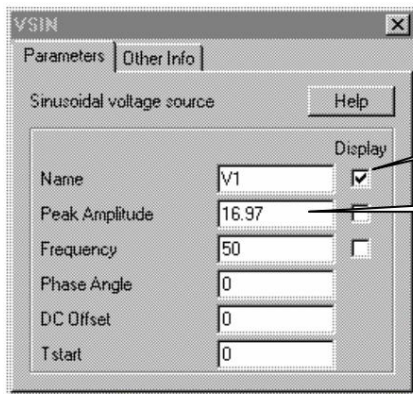
We perform the wiring, go to the **Edit menu, then Wire**



Values are assigned to components by double-clicking.



With a 1 on can view the current during the simulation



Option to display the name of the component that can be modified

Warning, value max., here 12V effective

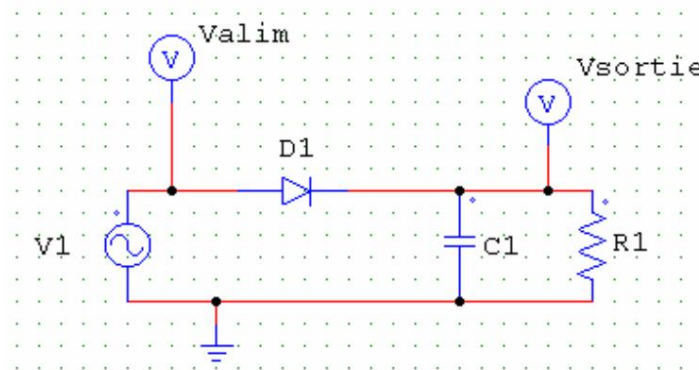
Choose a resistor of 100 Ω , a capacitor of 470 μF (47×10^{-5})

Next, the measuring devices are placed.

Menu **elements, other, probe, VP**

Making the connections: To

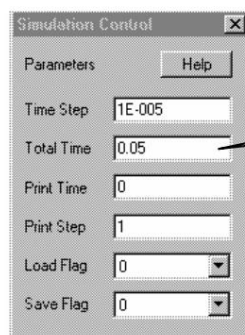
function correctly, the devices need a reference. Menu **items, other, probe, Ground**



Next, the simulation parameters must be specified.

Simulate menu , **simulation control**

Choosing the parameters here is tricky, because aberrant results can be obtained.

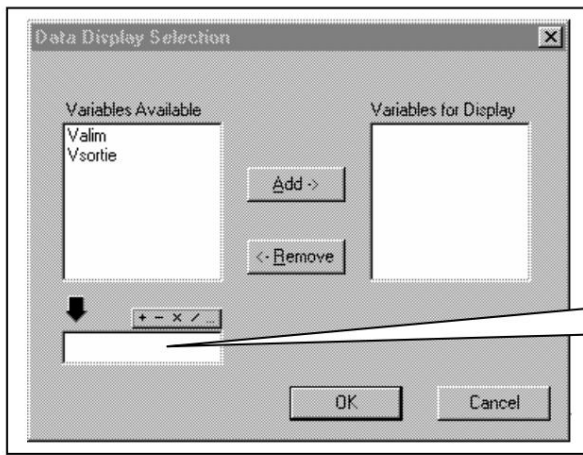


Simulation time

Launch the simulation with Menu **simulate, Run PSIM**

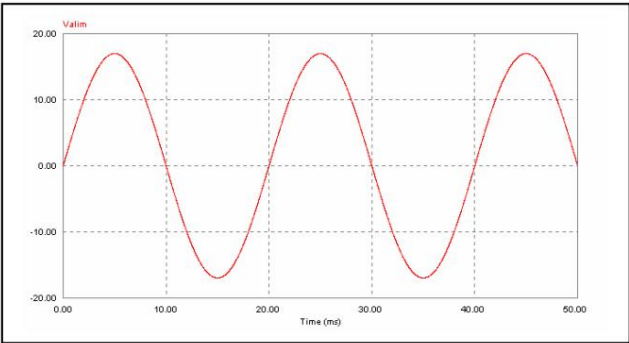
Select Valid

Then Add

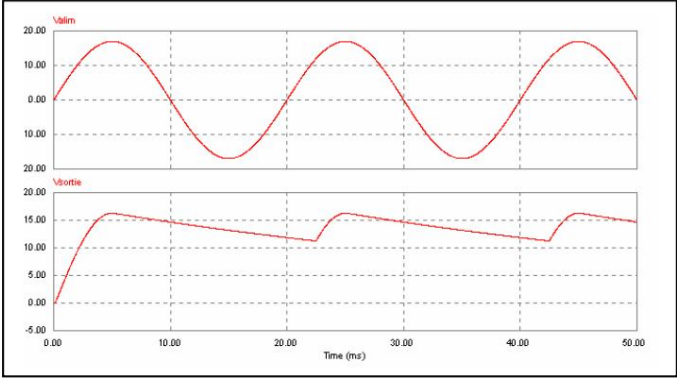


With the values obtained, we can perform mathematical operations For example, calculating the product $u \cdot i$

Then OK

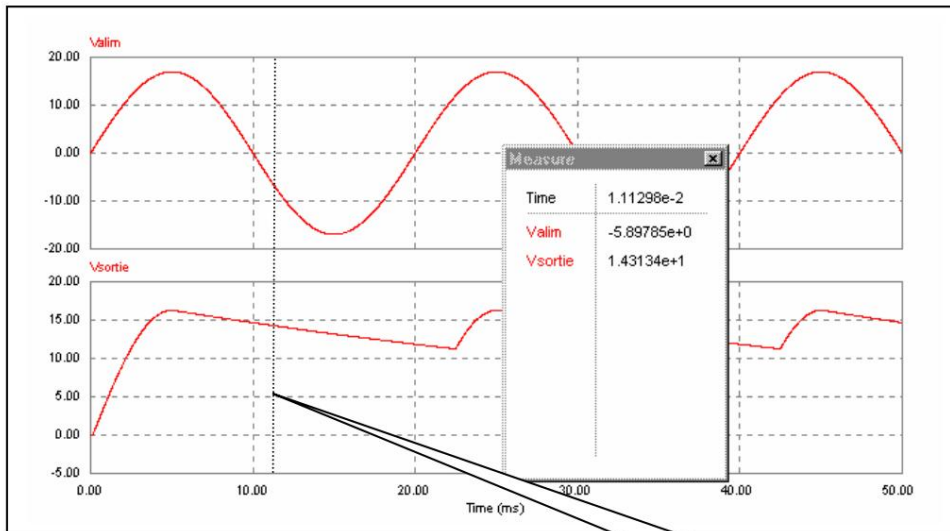


We add the second Voutput curve
Menu **Screen, Add screen**



We have the ability to carry out measurements

Menu **Measure, measure**



We can move the cursor to see the values displayed in the painting.

You can also print these curves: **File > Print**

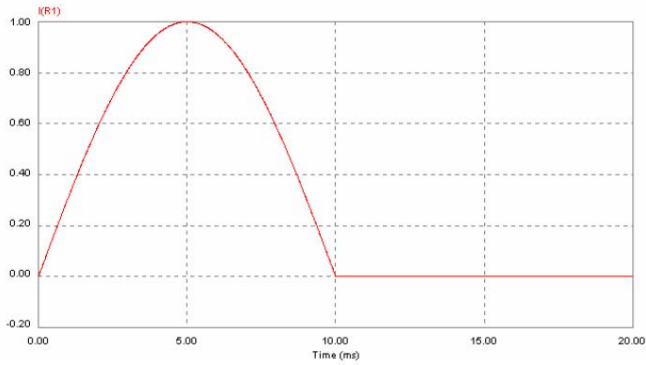
You can change the scales of the curves: **Axis menu**, then choose **X Axis** or **Y Axis**.

Consider the following diagram:



We will see how to perform different measurements on the current in the resistor, for example.

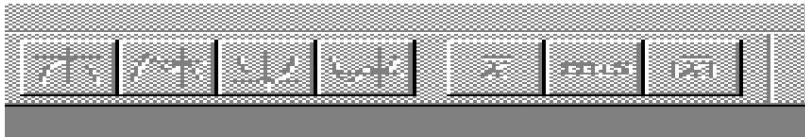
After launching the simulation, always use a whole number of periods to obtain results correct.



Menu **Measure, measure**

Then select I(R1) on the curve in the upper left corner

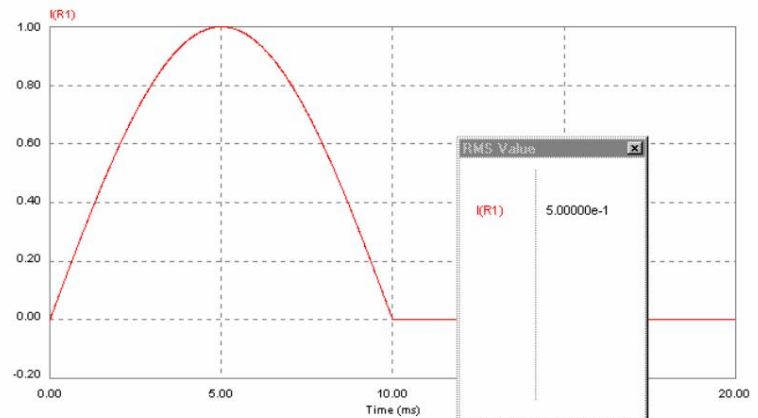
The following icons become active.



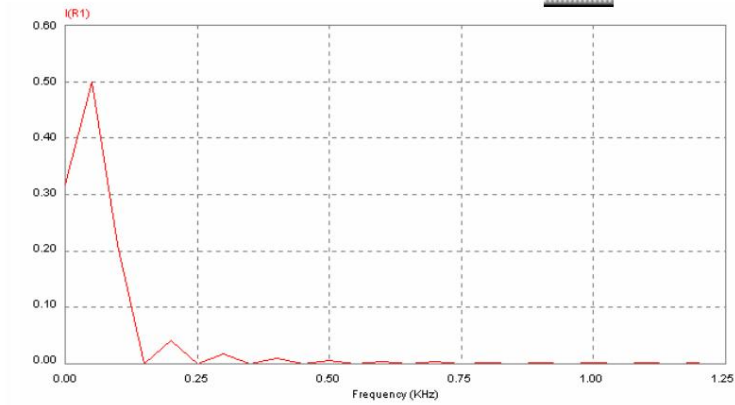
Max, Local Max, Min, Local Min, Average, RMS, Average of



measurement of the effective value for example



The FFT can also be performed using the FFT icon.



You can read the values (maximum values) of the harmonics by moving along the curve after activating the icon.



Use the zoom function to obtain the desired curve.



You can "paste" diagrams or curves into a document such as Word.

Select the part to copy with the arrow cursor, then go to **Edit, Copy to clipboard, choose Color or Black and White.**

Open the Word file and paste (it is advisable to paste into a frame to move the image easily later).

When launching the simulation, it is necessary to carefully consider the calculation step size to impose on the software. This depends on the nature of the circuit to be simulated. The software can modify this step size, and its new value is recorded in a... text file message.doc

For example:

The time step has been changed to:

.10E-02