Chapter 7: Simulation and co-simulation with other software

Chapter 7: Simulation and co-simulation with other software

7.1-Introduction

The Simulink platform supports co-simulation between components. In Simulink, co-simulation is between components with local solvers or involving simulation tools. For example, co-simulation can be an S-function implemented as a co-simulation gateway between Simulink and third-party tools or custom code, or a Functional Mockup Unit in co-simulation mode imported to Simulink.

In this chapter we will talk about the different software simulation like PSim, PSpice, Proteus,…etc.

7.2 - Simulation by PSim and Simulink-PSim co-simulation

PSim is simulation software specially designed for the analysis and design of power electronics and motor drives. It provides a powerful simulation and design environment for switch mode power supplies, analog/digital control, and electric motor drives.

PSim simulation tool is widely used among research scholars because of its easy to use, fast simulation and flexible control representation [13].

7.2.1 - Setting up Co-Simulation with Matlab/Simulink

PSim can perform co-simulation with Matlab/Simulink. When PSim is installed, if the Sim Coupler Module is enabled in the license, the installation will set up the co-simulation between PSim and Matlab/Simulink automatically. If you change the PSim folder name after PSim is installed, you must set up the co-simulation again by running Utilities >> SimCoupler Setup.

If you are running multiple versions of PSIM or Matlab/Simulink, to associate a specific version of PSim with a specific version of Matlab/Simulink, from that PSim version, run Utilities >> SimCoupler Setup, and select the desired Matlab/Simulink version.

To see how the co-simulation works, follow the steps below to simulate the sample circuit "TP1.slx":

1. From PSim, run Utilities >> SimCoupler Setup. This will set up PSim for the co-simulation. Note that this needs to be run only once.

2. Launch Matlab. From Matlab, launch Simulink.

3. In Simulink, load the file “TP1.slx” from the “examples\Simulink co- simulation” sub-folder in the PSim directory.

4. Double click on the SimCoupler block (the block with the orange color). Make sure that the path for the file “TP1_psim.sch” is correct. If it is not, click on the Browser button and locate the file in the “examples\Simulink co-simulation” sub-folder in the PSim directory.

5. In Simulink, start simulation. After simulation is completed, SIMVIEW will be launched to view waveforms.

To start, double click on the Pspice SV icon and Pspice schematic will open. The figure below is schematic

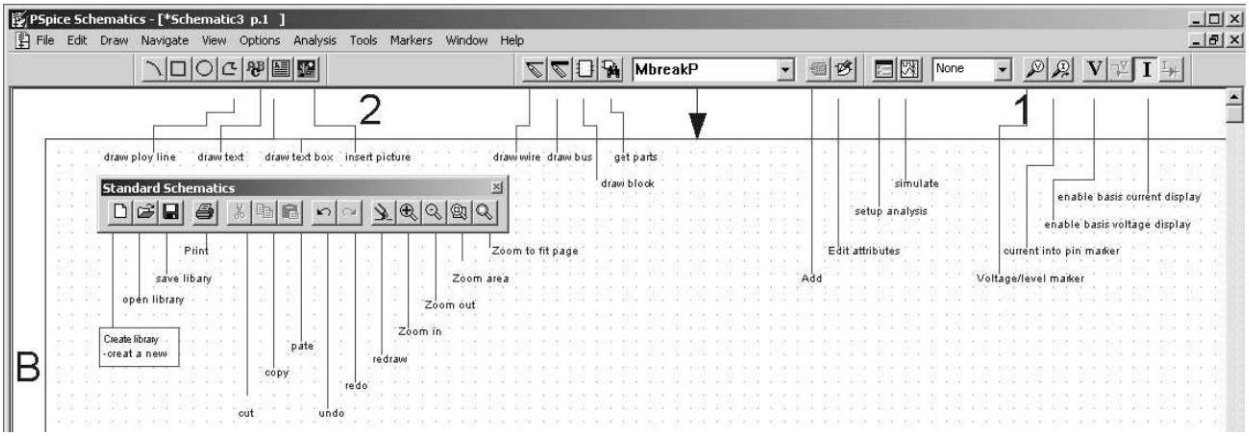

Figure 7.1 *PSpice Schematics*

Example:

7.3 - Simulation with other software

7.3.1 - Search for components

If we select a library, for example “eval”, we will appear in the “Part List” the list of components constituting this library. By choosing a component In particular, it can be viewed at the bottom right of the main window. Using the arrows direction of the keyboard, up and down, you can visualize all the components of this library [14],[15].

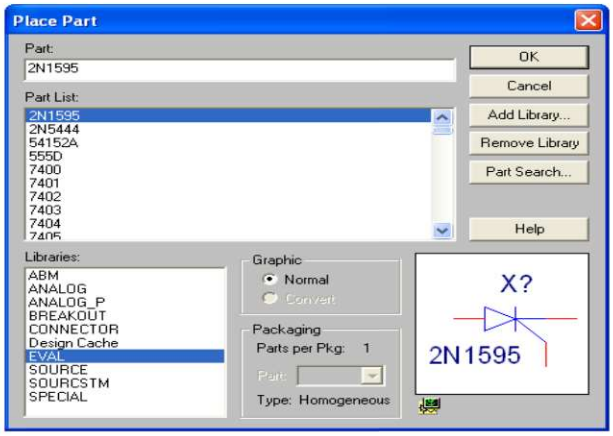

***Figure** 7.2 Symbol placement window*

We select the component, then “Ok”.

7.3.2 - Automatic search in a library

You can also launch an automatic search directly in the library chosen, by entering the “Part” field. You must use the * character to replace a part of the component name of which we are not certain.

we are looking for example for the 2n2222 transistor. Just mention “*22*” in the “Part” field, click on “Ok”. The research editor will find:

7.3.3 - Scheme entry

7.3.3.1 - Component Placement

Component placement is done by clicking on the icon (Place Part) or Place→Part, or keyboard shortcut “p”.

This opens the following window:

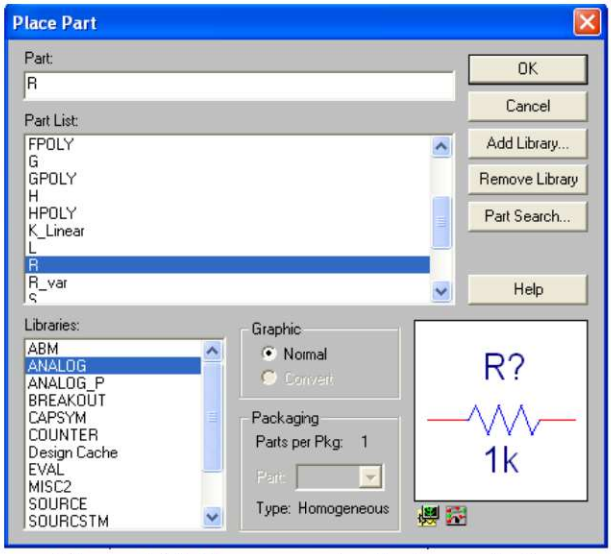

Figure 7.3 *Component selection window*

7.3.3.2 - Simulation

7.3.3.2.1 - Creation of a simulation profile

To carry out a simulation, you must define the parameters of the variable quantities [16].

That is to say the variation range, the calculation step, as well as other information which will be seen later, but we must first create a simulation profile by clicking on the icon (new simulation) or pspice→ new simulation profile, and indicate a name on the window which is displayed:

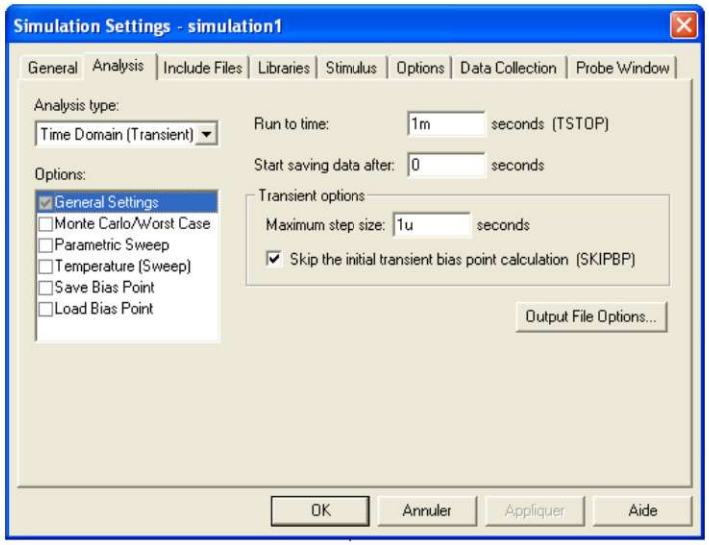

Figure 7.4 *Define the type of analysis and simulation parameters*

7.3.3.2.2 - Simulation types

7.3.3.2.2.1 - Temporal analysis "Time Domain (transient)"

A temporal analysis (transient or TRAN) corresponds to a simulation based on time. The following window explains the (Time Domain) option in detail:

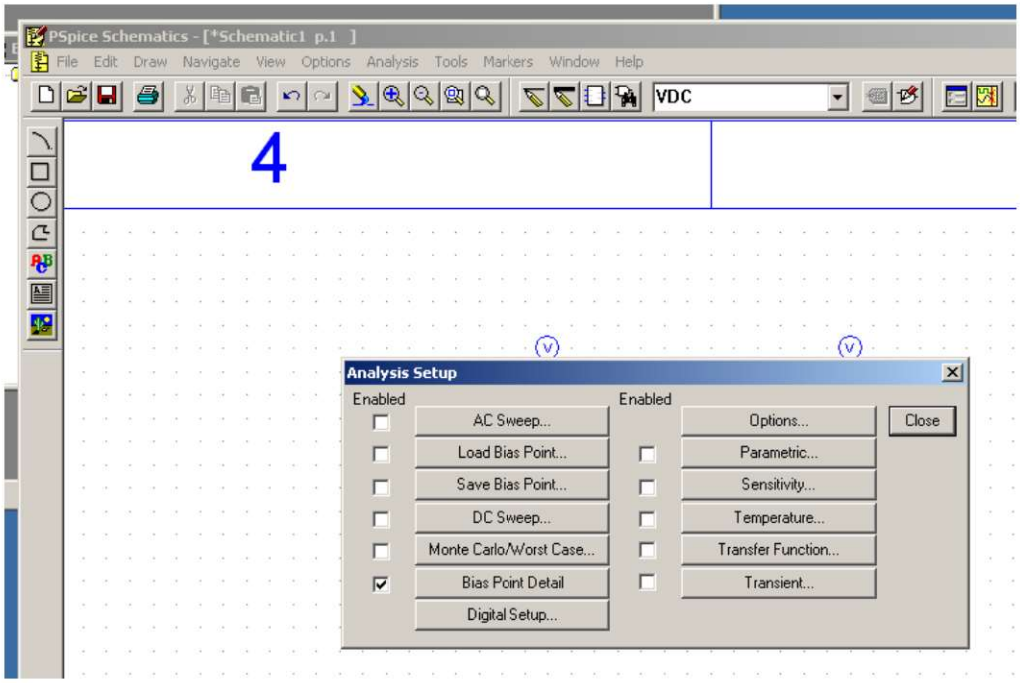

Figure 7.5 *Analysis setup*