



PSIM SIMULATION SOFTWARE

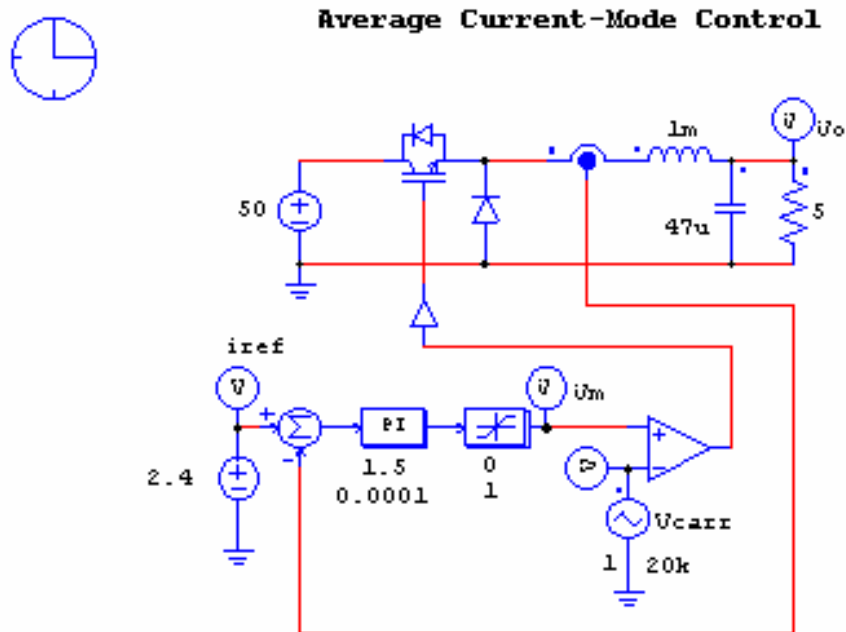
TUTORIAL

How to use the Simcoupler

The objective of this tutorial is to help you use the Simcoupler module to realise co-simulation between PSIM 6.0 and Matlab/Simulink*.

We are going to realise a circuit with the power part implemented in PSIM and the control part implemented in Simulink.

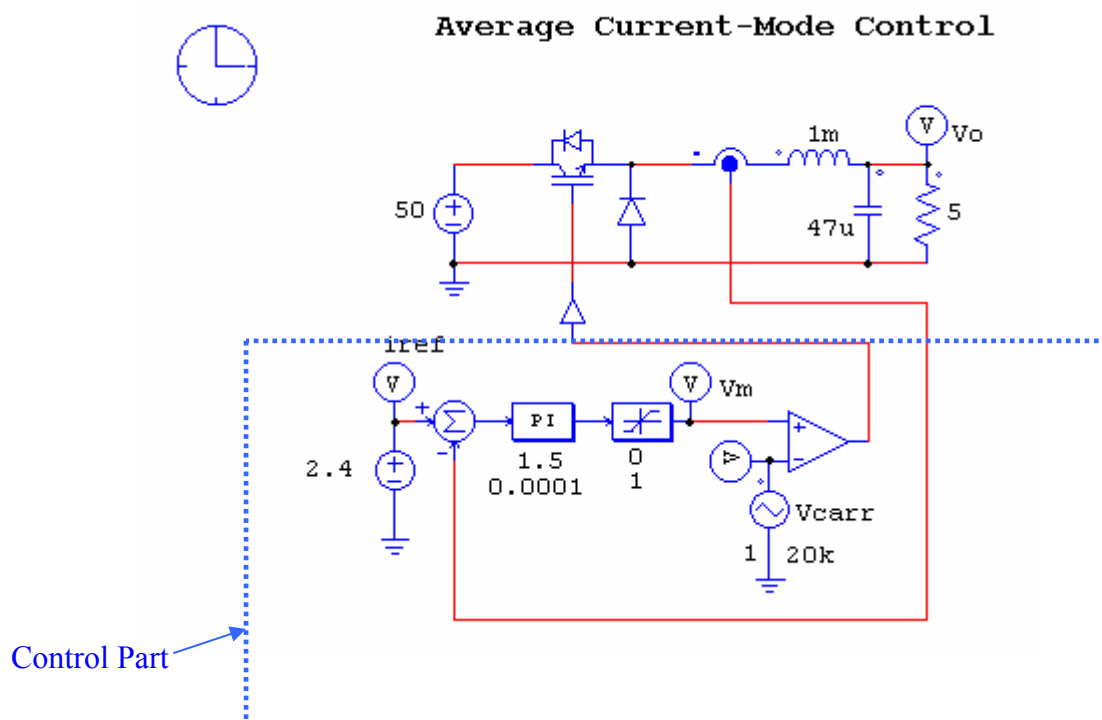
The example circuit is the following (this example can be run with the demo version) :



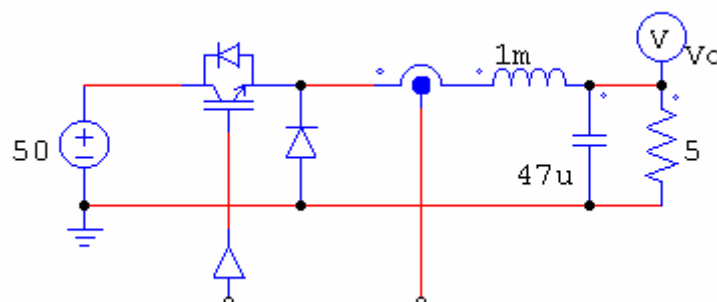
* : Matlab and Simulink are registered trademarks of MathWorks, Inc

Here are after the different steps to realise co-simulation :

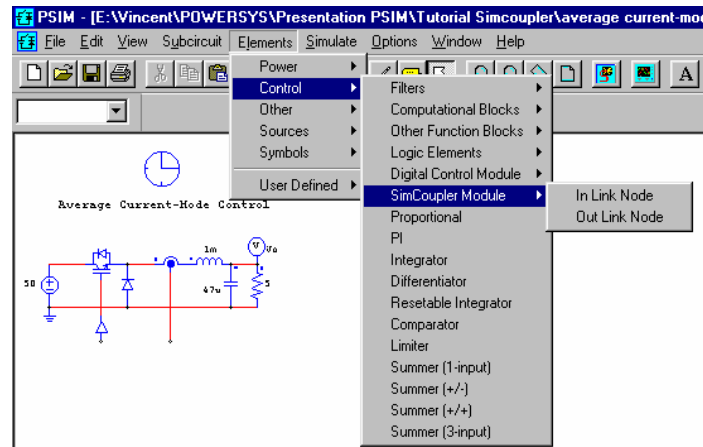
1. Open your PSIM version.
2. Open the Psim file “average current-mode.sch”.
3. Save the file in a directory (for example in E:\Test\) and give for example the name “average_current_mode_psim.sch”.
4. Modify the precedent file by deleting the control part :



5. You now have the following circuit :

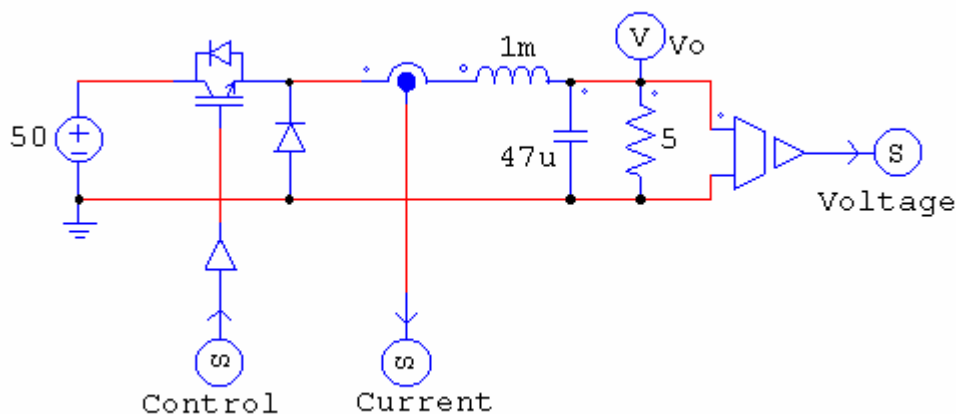


6. Go to the Menu “Elements/Control/SimCoupler Module/



Then connect one OUT SLINK node to the current sensor output and rename it as “Current” ; connect one IN SLINK node to the switch controller, and rename it as “Control”. We will add another OUT SLINK to measure the resistance voltage. Rename it as “Voltage”(the demo version has a limit of one OUT and IN SLINK, then do not add the second OUT SLINK if you only have this version).

An In Link Node receives a value from Simulink and an Out Link Node sends the value to Simulink



7. If we have more than one OUT SLINK node, we need to arrange the order of the link nodes. Go to the Simulate Menu, and select Arrange SLINK Nodes (Image 1). A dialog window will appear (Image 2). Arrange the order of the SLINK_IN nodes and SLINK_OUT nodes to be the same as how the input/output ports would appear in the SimCoupler model block in Simulink (the order of the ports is from the top to the bottom).

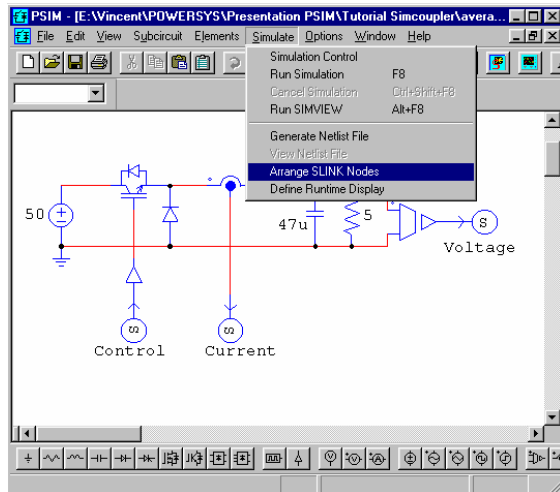


Image 1

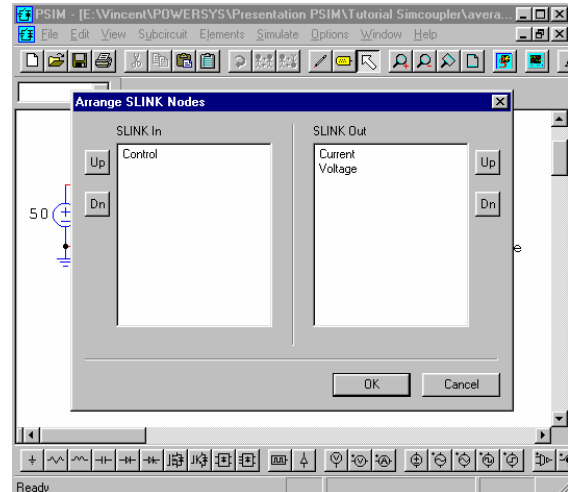
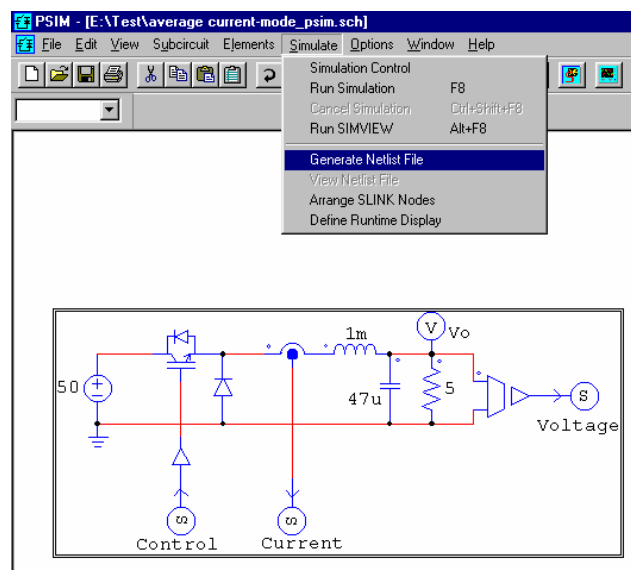


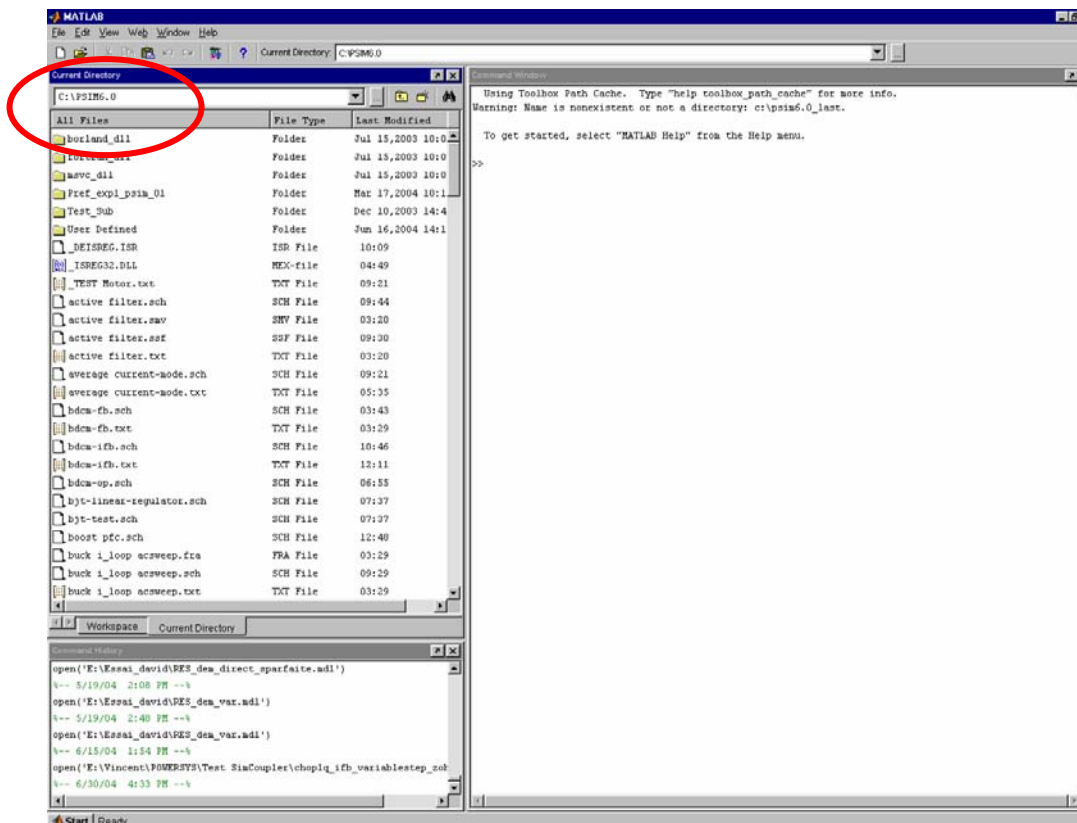
Image 2

8. Go to the Simulate Menu, and select Generate Netlist File. A netlist file with the .cct extension will be generated and saved under the same directory as the schematic file. In the simulation control we have a time step equal to 2u. In this example, we worked in the directory "E:\Test\". Then the netlist file name and path will be "E:\Test\average_current_mode_psim.cct".

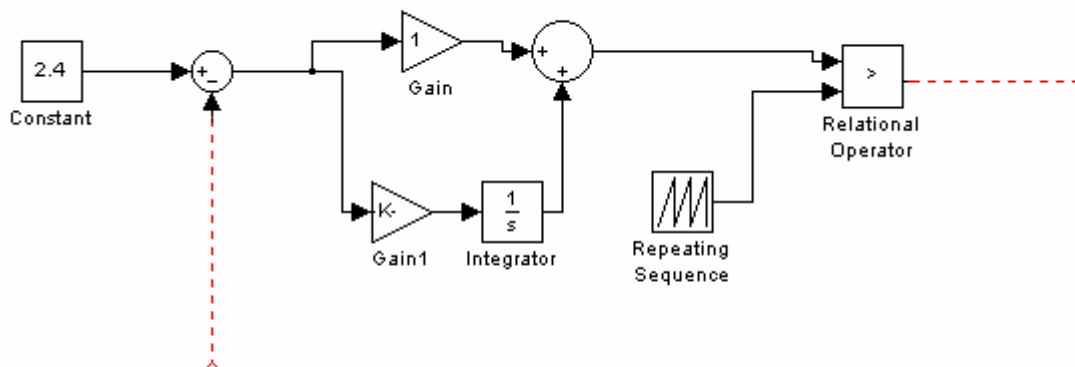


At this Step we have finished the PSIM part.

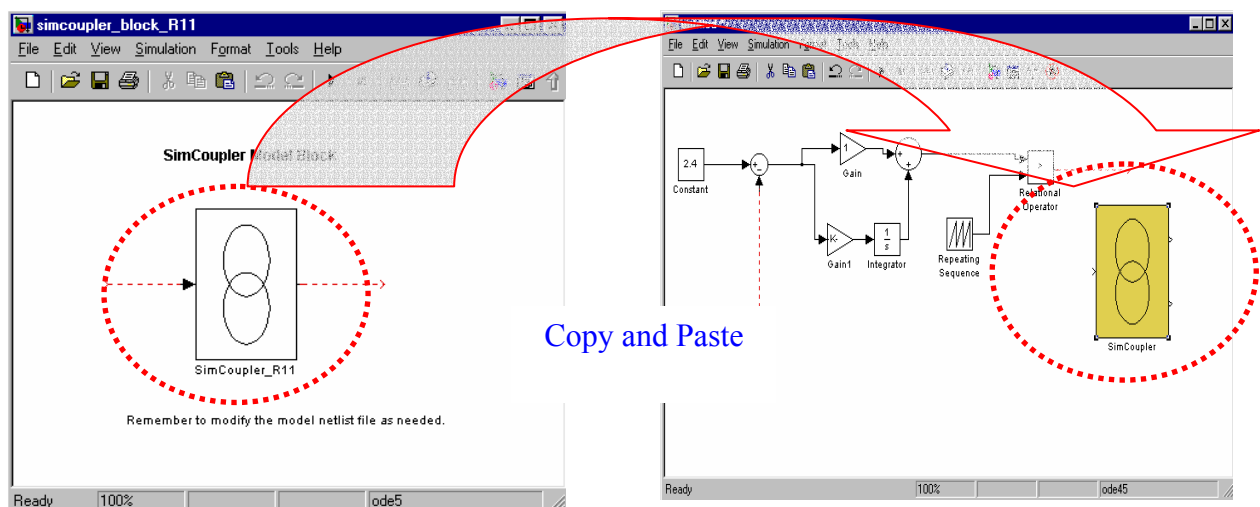
9. Start Matlab.
10. Change the working directory to the PSIM directory. If PSIM is installed in the directory "C:\PSIM6.0", change the directory to "C:\PSIM6.0".



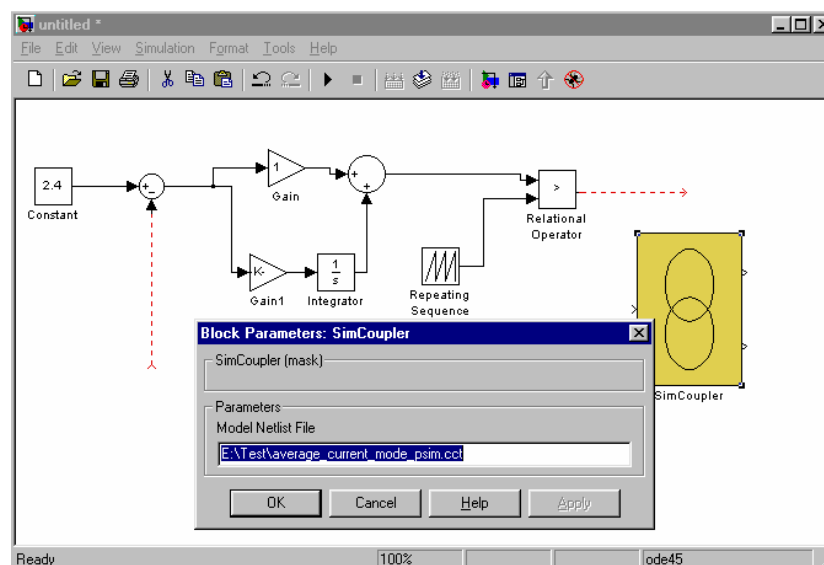
11. Launch Simulink and create a new file corresponding to the control part we have deleted in PSIM at step 4.



12. Save the file in a directory (for example in E:\Test\)) and give the name “average_current_mode_simulink.mdl”.
13. In Simulink go to the menu “File /open” and open the file “SimCoupler_Block_R11.mdl” that stores the SimCoupler model block which is in the PSIM directory. Copy and paste the SimCoupler model block into the example file.

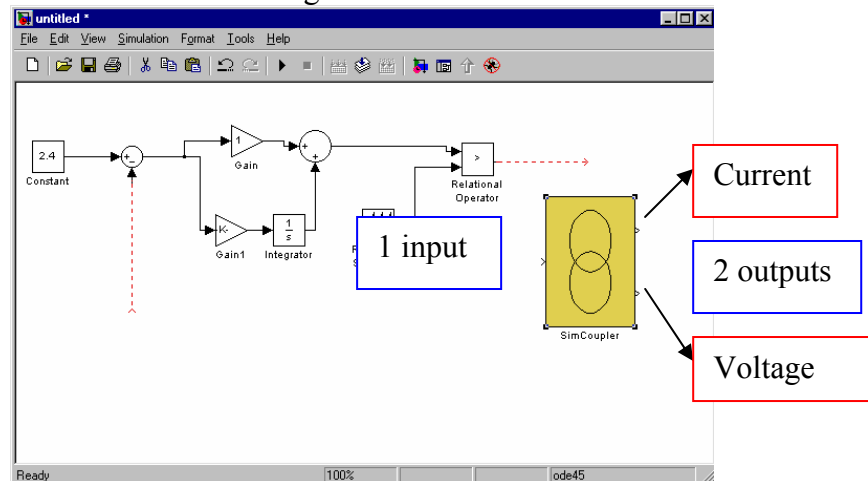


14. Double click on the SimCoupler block, and enter the name and the location of the PSIM netlist (E:\Test\average_current_mode_psim.cct), and click on Apply.



If you have more than one IN and OUT SLINK, the number of input and output ports for the SimCoupler model block will automatically match those defined in the PSIM netlist. In this case, there will be 1 input port and 2 output ports (except if you have the demo version and if you did not add the second OUT SLINK).

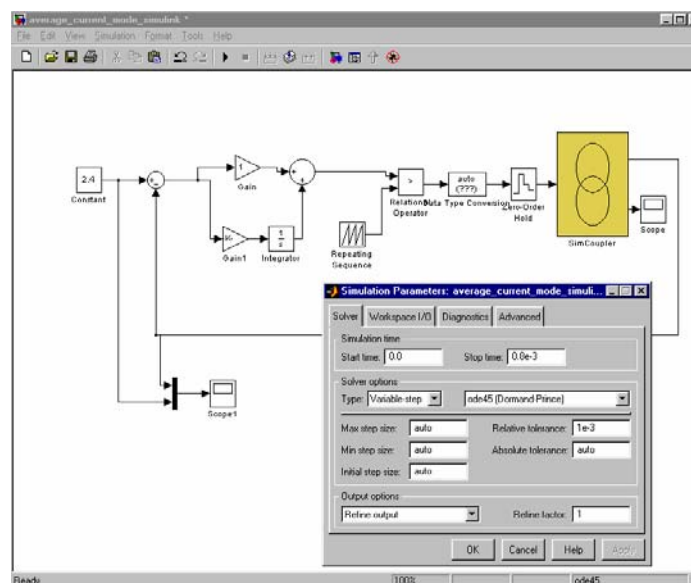
The order of the output ports had been decided at step 7. In this example we have from the top to the bottom the “Current” and the “Voltage”.



If the number of link nodes in the netlist is changed later, go to the Edit menu in Simulink and choose Update Diagram. This will update the model block ports.

15. Finish the connection for the Simulink part : go to the Simulation menu and select Simulation Parameters. Under Solver Options, set the Type to “Fixed-step”. Set the step size to be the same as or close to PSIM’s time step.

In this case, we used a variable step with a zero-order-hold set to 2us (see additional remarks next page).



16. The setup of the co-simulation is now complete. Go to Simulink and start the simulation.

Additional remarks :

- When the SimCoupler model block is used in a feedback system in Simulink, the SimCoupler model block may be part of an algebraic loop. Some versions of Matlab/Simulink can not solve a system containing algebraic loops, while others can solve the system but with degraded performance. To break an algebraic loop, place a memory block at each output of the SimCoupler model block. The memory block introduces one integration time step delay.
- Certain restriction is imposed on the selection of the solver type and the time step in Simulink when performing the PSIM-Matlab/Simulink co-simulation. Simulink must be set up to have the Solver Type as Fixed-step with the time step the same or close to the PSIM time step, or if the Solver Type is Variable-step, a zero-order-hold must be used with the sample time the same or close to the PSIM time step.
- Because Simulink and PSIM can have different time steps, it is not a good idea to exchange data if the data are of logic signal type (0 and 1).