January 10th, 2014
VERSION: 2.0

Standard_sensitive_scr PSpice models PC/WINDOWS INSTALLATION STEPS
----

1. Copy all the files to the ORCAD\CAPTURE\LIBRARY\PSICE directory.
2. Run ORCAD Schematics program.
3. Select the PART... option in the PLACE menu.
4. Click on ADD LIBRARY... button.
5. Search and select standard_sensitive_scr_symbols.olb file.
6. Press the OPEN button. The symbols will be automatically loaded.
7. Press CANCEL button.
8. Now, select EDIT SIMULATION PROFILE option in the PSPICE menu.
9. Select LIBRARY option in Configuration files menu.
10. Click BROWSE... button
11. Search and select the standard_sensitive_scr_pspice.lib file.
12. Press the OPEN button.
13. Press the ADD AS GLOBAL button.
14. Press the OK button.
15. Congratulations, you are now ready to use your new STMicroelectronics model library.

****
* Thyristor PSpice Models *
****
* This thyristor model simulates:
* -I_{GT} MAX of the specification
* -I_L Typ of the specification ($I_{L\_Spice} = I_H / 2$)
* -I_H Typ of the specification ($I_{H\_Spice} = I_H / 3$)
* -V_{DRM}
* -V_{RRM}
* -Power dissipation is realistic and correspond to a typical SCR
*
* All these parameters are constant, and don't vary neither with temperature
* nor other parameters.
*
* For a correct SCR behavior, the "Maximum step size" must be below or equal 20µs.
****