October 10th, 2011
VERSION: 1.0

diacs_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 diacs_symbols.olb.

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 diacs_pspice.lib.

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.


```
*******************************************************************
*                 DIAC PSpice Models                            *
*******************************************************************
* This DIAC model simulates:
*tr:     Rise time (in µs)
*V_{BO}:  Break over voltage
*Δ V:  Dynamic breakover voltage
*I_{BO}:   Breakover current
*All these parameters are constant, and don't vary neither with temperature
*nor other parameters.
*
*Simulated parameters models different from given value:
*simulated I_{BO} depends on Maximum step size; this step must be > 1 us to have
*simulated  I_{BO} < maximum I_{BO} of specification
*******************************************************************
```