

October 10<sup>th</sup>, 2011

VERSION: 1.0

## acs\_triacs 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 acs\_triacs\_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 acs\_triacs\_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.

```
*****
*                                     *
*               ACST PSpice Models   *
*                                     *
*****
* This ACST model simulates:
* -IGT (the same for all quadrants) maximum of the specification
* -IL positive and negative maximum of specification
* -IH positive and negative typical values
* -VCL positive and negative typical values (behavior overvoltage protection)
* -IBO positive and negative typical values (behavior overvoltage protection)
* -(dI/dt)C and (dV/dt)C parameters are simulated if those constraints
*   exceed very highly the specified limits.
* -Power dissipation is realistic
*
* All these parameters are constant, and don't vary neither with temperature
* nor other parameters.
*
* For a correct ACST behavior, the "Maximum step size" must be below
* or equal 20μs.
*****
```