

PC/WINDOWS INSTALLATION STEPS

1. Unzip file " ST_POWER_SCHOTTKY_V10.LIB" to the directory "OrCAD\tools\pspice\library\".
2. Unzip file " ST_POWER_SCHOTTKY_V10.OLB" to the directory "OrCAD\tools\capture\library\pspice\".
3. Run ORCAD Schematics program and open your design's file or create a new design.
4. Select "PART"... option in the "PLACE" menu.
5. Click on "ADD LIBRARY"... button or "ALT+A".
6. Search and select the file " ST_POWER_SCHOTTKY_V10.OLB".
7. Press the "OPEN"... button. The symbols will be automatically loaded.
8. Now, select "EDIT SIMULATION PROFILE"... option in the "PSPICE" menu or select "New Simulation Profile".
9. Select the Tab "Configuration Files" and Category "Library".
10. Click BROWSE... button
11. Search and select the file " ST_POWER_SCHOTTKY_V10.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 library STMicroelectronics.