

# NXP RFpower Model Library Manual for Microwave Office

## Version 20120420

### Contents

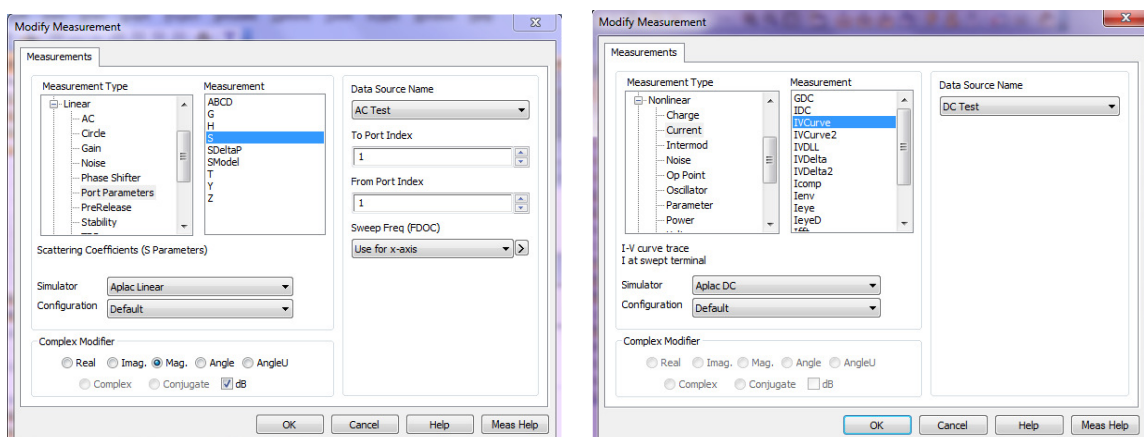
- Model library release notes
- How to install the NXP RFpower Model Library and add it to your design project
- How to install stand-alone NXP LDMOS Models and add them to your design project
- How to insert NXP LDMOS and GaN Models into your design
- How to correctly use NXP LDMOS and GaN Models with a Thermal Node.
- Advise on MWO APLAC Harmonic Balance Simulator Settings.

### Model Library Release Notes

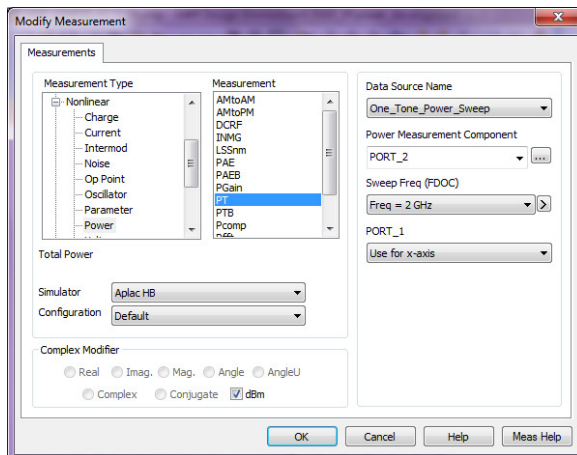
The NXP RFpower Model Library (version V07p0) for Microwave Office (MWO) is suited for use with MWO version 9.02 and higher and with the APLAC simulator only. This simulator is the recommended default simulator engine within MWO from now onwards, especially for non-linear devices in large-signal applications. In addition, this library contains from Version V04p0 release onwards, a new basic model for the LDMOS models, which provides you with more accurate simulation results.

Due to these changes some points have to be taken into account:

- To add models of the NXP RFpower Model Library version V07p0 to a circuit containing models from the NXP LDMOS Model Library version V03p0 and older, you have to replace the old models with new ones.
- Use APLAC Simulators for all simulations.

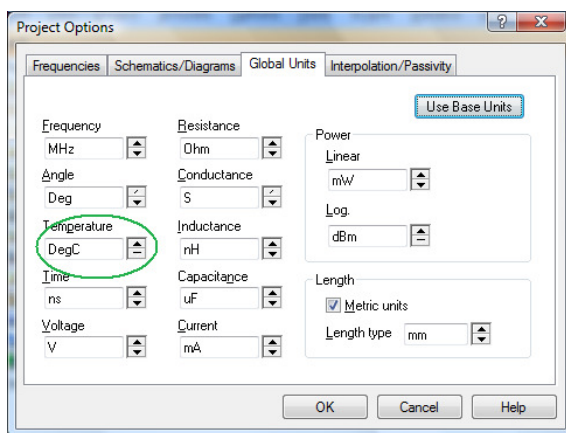


Use APLAC Simulators for all simulations (cont.)



- iii. This library uses the global variable `_TEMP` as Simulation Temperature. Ensure to use Project Options => Global Units Temperature: DegC. This global temperature may be assigned a value in the Global Definitions window.

Use the instance parameter “DTA” to change the specific device temperature relative to `_TEMP`.



## How to install the NXP RFpower Model Library and add it to your design project

- i. The NXP RFpower Model Library for Microwave Office (MWO) is installed in the folder named:  
"\$AWR\Foundry\NXP\RFpower";  
where  
"\$AWR" is the directory where the AWR software is installed (typically  
"C:\Program Files\AWR").

Before installing a new version of the NXP RFpower library, delete any previous installation folder of the library or its predecessor NXP LDMOS and all of its sub-folders.

- ii. Create a directory called "\$AWR\Foundry\NXP" in case it does not already exist and unzip the file "NXP\_RFpower\_Lib\_Vxxpx.zip" into that directory.  
You should see a new directory "\$AWR\Foundry\NXP\RFpower".
- iii. Start MWO from the Windows "Start" menu => "All Programs" => "AWR...", or from the default shortcut on your desktop.
- iv. Either start a new MWO project and load the installed library or add the installed library to an existing MWO project:
  - a. To start a new MWO project, go to the menu "File" => "New With Library" => "Browse...". Browse to the directory "\$AWR\Foundry\NXP\RFpower" to find "NXP\_RFpower.ini". Select it and press "Open". The NXP RFpower Model Library will be loaded into the new MWO project.
  - b. To add the NXP Library to an existing project, go to the menu "Project", select "Process Library" => "Add/Remove Library...". Press the "Add" button, then open "NXP\_RFpower.ini". Press "OK" and the NXP RFpower library will be added to your project file.
- v. The next time you start MWO, go to the menu "File" => "New With Library" and you should see "NXP\_RFpower" in the list.
- vi. The NXP Library will appear under the element browser "Elements" => "Libraries". Click on the "+" sign to expand the folder "Libraries" and you should see the folder "NXP-RFpower". Expand the folder to see all of the parts.

## How to install stand-alone NXP LDMOS Models and add them to your design project

- i. NXP LDMOS models for Microwave Office (MWO) are installed in folders named:  
"\$AWR\Foundry\NXP\<LDM\_type>";

where

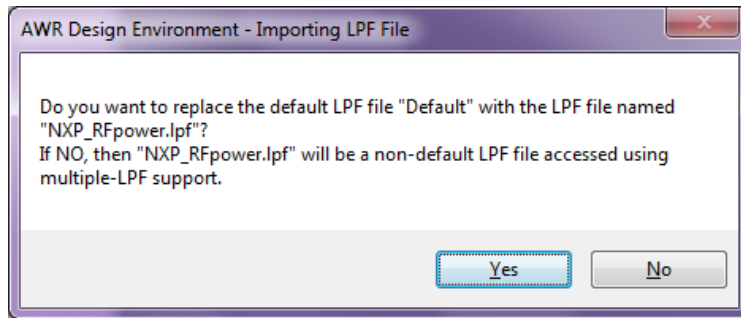
"\$AWR" is the directory where the AWR software is installed (typically  
"C:\Program Files\AWR"),  
"<LDM\_type>" is the unique LDMOS type included in each library.

Before installing a recent version of a <LDM\_type> library, delete any previous installation folder for that type and all of its sub-folders.

- ii. Create a directory called "\$AWR\Foundry\NXP" in case it does not already exist and unzip the file "NXP\_<LDM\_type>.zip" into the \$AWR directory.  
You should see a new directory "\$AWR\Foundry\NXP\<LDM\_type>".
- iii. Start MWO from the Windows "Start" menu => "All Programs" => "AWR...", or from the default shortcut on your desktop.
- iv. Either start a new MWO project and load the installed library or add the installed library to an existing MWO project:
  - a. To start a new MWO project, go to the menu "File" => "New With Library" => "Browse...". Browse to the directory "\$AWR\Foundry\NXP\<LDM\_type>" to find "NXP\_LDM\_type.ini". Select it and press "Open". The NXP LDMOS Model will be loaded into the new MWO project.
  - b. To add the NXP LDMOS Model to an existing project, go to the menu "Project", select "Process Library" => "Add/Remove Library...". Press the "Add" button, then open "NXP\_<LDM\_type>.ini". Press "OK" and the NXP LDMOS Model will be added to your project file.
- v. The next time you start MWO, go to the menu "File" => "New With Library" and you should see "NXP\_<LDM\_type>" in the list.
- vi. The NXP Library will appear under the element browser "Elements" => "Libraries". Click on the "+" sign to expand the folder "Libraries" and you should see the folder "NXP-<LDM\_type>". Expand the folder to see the symbol in the window below.

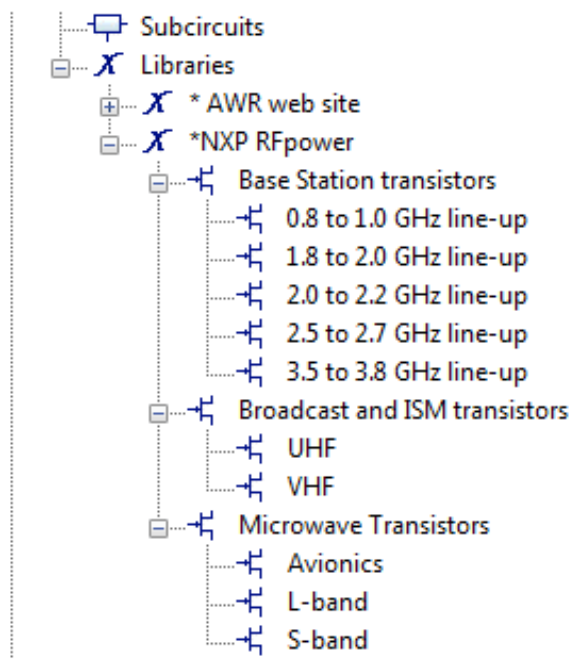
**Note:** The Layer Process file **NXP\_RFpower.lpf** is used with the devices package drawings. It's settings do not effect the circuit simulations as long as Global Units for Temperature are in DegC

“Yes” or “No” can be selected when the following message appears on adding the NXP Library to an existing project.



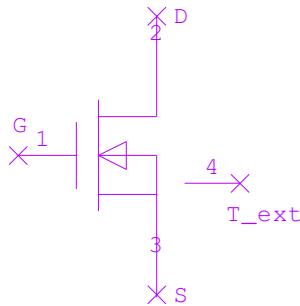
## How to insert NXP LDMOS and GaN Models into your design

- i. Browse to the element browser "Elements" => "Libraries". Click on the "+" sign to expand the folder "Libraries" and you should see the folder "NXP-RFpower". Expand the folder to see either all of the parts in case of the complete library or the stand-alone transistor.
- ii. Select the transistor type you want to add to your circuit and drag and drop it to your schematic.



## How to correctly use NXP LDMOS and GaN Models with a Thermal Node.

```
SUBCKT
ID=S1
NET="NXP_BLF7G22_130"
DTA=0
Zth_Enable=1
```



The “T\_ext” node is the thermal node of the device and must be connected to the ground either directly or through a RC-parallel network which may be used to describe an additional or alternative external thermal networks.

The “Zth\_Enable” parameter allows enabling or disabling the default thermal network of the device. This parameter can be used in combination with an external thermal network to reproduce the following situations:

1. Zth\_Enable set to 0, T\_ext node connected to ground directly: Isothermal simulations.
2. Zth\_Enable set to 0, T\_ext node connected to ground through a RC-parallel network: ET simulations with the external RC-parallel as thermal network.
3. Zth\_Enable set to 1, T\_ext node connected to ground directly: ET simulations with the default device thermal network.

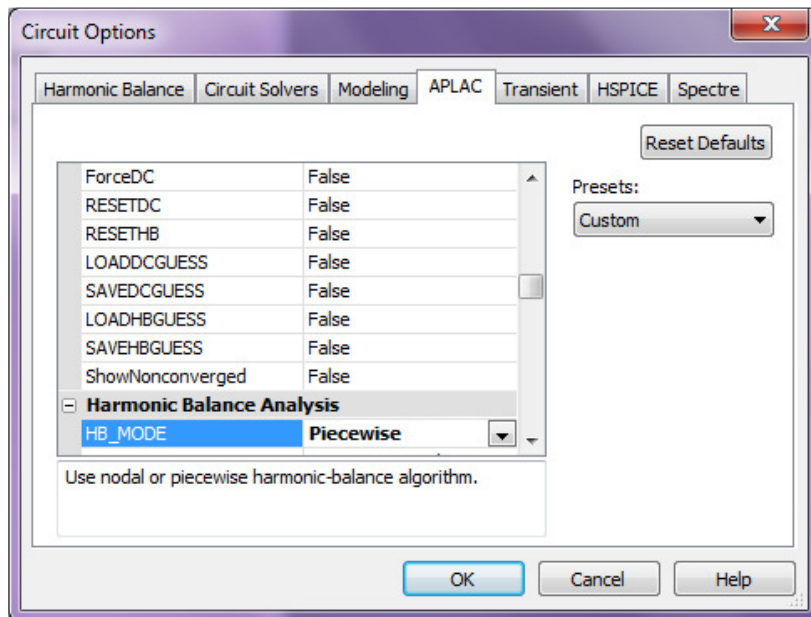
2. These components might not be suitable to support time-domain simulations such as transient simulations or transient-assisted HB simulations. Please contact NXP Semiconductors if this constitutes a problem.

3. Failing to connect the “T\_ext” node to the ground either directly or through a RC-parallel network will cause the non-convergence of simulations.

4. Zth\_Enable set to 1, T\_ext node connected to ground through a RC-parallel network: ET simulations with the series of the external RC-parallel and the device default as thermal network.

## Advise on APLAC Harmonic Balance Simulator Settings.

The Harmonic Balance analysis algorithm mode setting is found under the APLAC tab in the circuit options window.



HB\_MODE: HB analysis algorithm.

It is advisable to experiment with both modes to find the optimum simulation time for your circuit design.