How to use an imported Spice Model in LTSpice.

I found a Spice model (irf3205s.spi) for an NMOS transistor on the web. It is written as a subcircuit (not a .model). It is a text file which starts with .SUBCKT irf3205s 1 2 3. It has three pins in the order D G S. How do I integrate it into LTSpice?

Step 1. Save the file where LTSpice expects to find all other library and model files. (mine is in C:\Programs\LTC\lib\sub)

You need to create a new symbol which links to the new subcircuit. If you try to use the generic NMOS symbol that comes with LTSpice, it links to LTSpice's internal library of .models; not to an added subcircuit.

Rather than creating a new three-pin NMOS symbol from scratch, and since the subcircuit has the same DGS pin order, here is how to modify the supplied LTSpice NMOS.asy into a modified one that will work with a subcircuit instead of LTSpice's library method.

Step 2: Using Windows Explorer, find nmos.asy (mine is in C:\Programs\LTC\lib\sym)

Step 3: Make a copy and rename it to NMOSsub.asy. The new file should be in the same sub-directory as above.

Step 4: Open NMOSsub.asy with LTSpice . It should open in a Symbol Editor window.

Step 5: Select the delete tool (scissors), click on NMOS. This deletes the existing placement of the value attribute which is used to link to the internal library of models.

Step 6: Select Edit/Attributes/Attribute Window. Pick SpiceModel from the list, and drag it the same location as where the value (NMOS) attribute used to be. This is a new attribute placeholder for pointing to the subcircuit name.

Step 7: Select Edit/Attributes/Edit Attributes. Change Prefix from MN to X. Change SpiceModel from blank to subckt . Delete whatever is in value so that it is blank

Step 8: File/Save

You now have augmented the standard symbols with a new one which can be linked to any three-pin DGS NMOS subciruit you choose. You can now place this new component in a schematic

Step 9: In the schematic editor, Click on the nand gate tool, which should show you the list of standard components. Pan down to nmos and you will see nmos, nmos4, and the newly created NMOSsub. Select NMOSsub and place it in your schematic. . Do this just like you wanted to place a standard NMOS transistor.

Step 10: Click on subckt and change it to irf3205s.

Step 11: Add the .INC irf3205s.spi (or whatever file the actual subcircuit is contained in) directive to your schematic.

Now you should be able to simulate the schematic.