

[LTspice] Import New Components

Kirill Semenov

Version 1

Setup

First, create a folder where you want to save LTspice simulation files.

Then, create a new LTspice schematic and save it in the project folder.

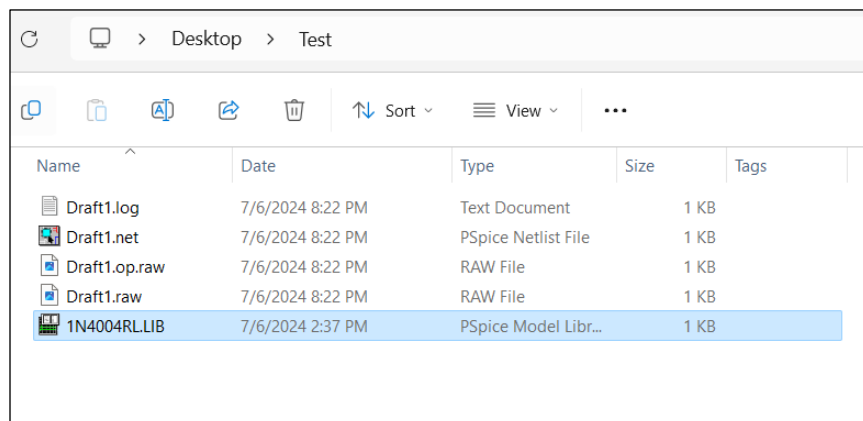
Download a file with a SPICE model (for example, the “1N4004RL.LIB” PSpice model). There are a few types of SPICE models: Spice2, Spice3, PSpice and more. All of them will work in LTspice, but PSpice is preferred because it is a newer format than others and it is usually more accurate and detailed.

Open 1N4004RL.LIB with any text editor. The value after “.MODEL” is the name of the diode model that we want to use. In this case, it is “D1n4004rl”.

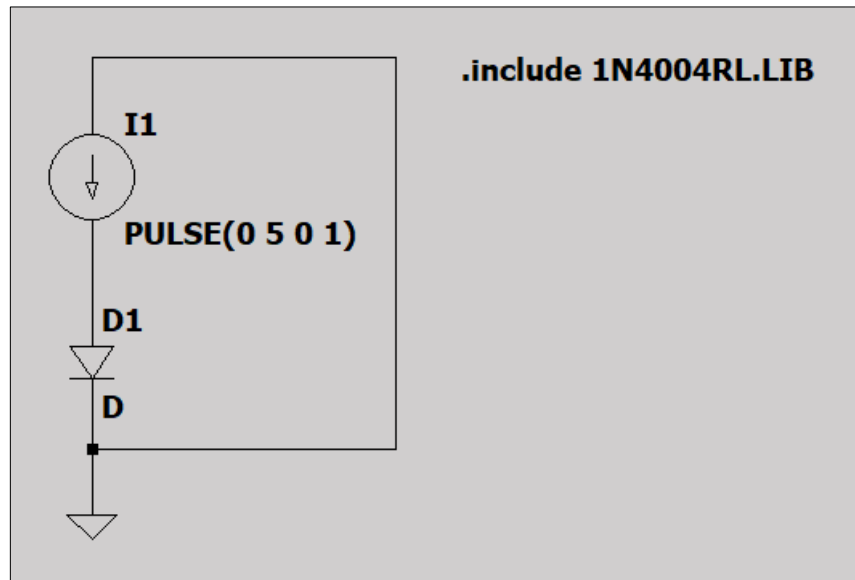
```
.MODEL D1n4004rl d
+IS=5.31656e-08 RS=0.0392384 N=2 EG=0.6
+XTI=0.05 BV=400 IBV=5e-08 CJO=1e-11
+VJ=0.7 M=0.5 FC=0.5 TT=1e-09
+KF=0 AF=1
* Model generated on October 12, 2003
* Model format: PSpice
```

Method 1

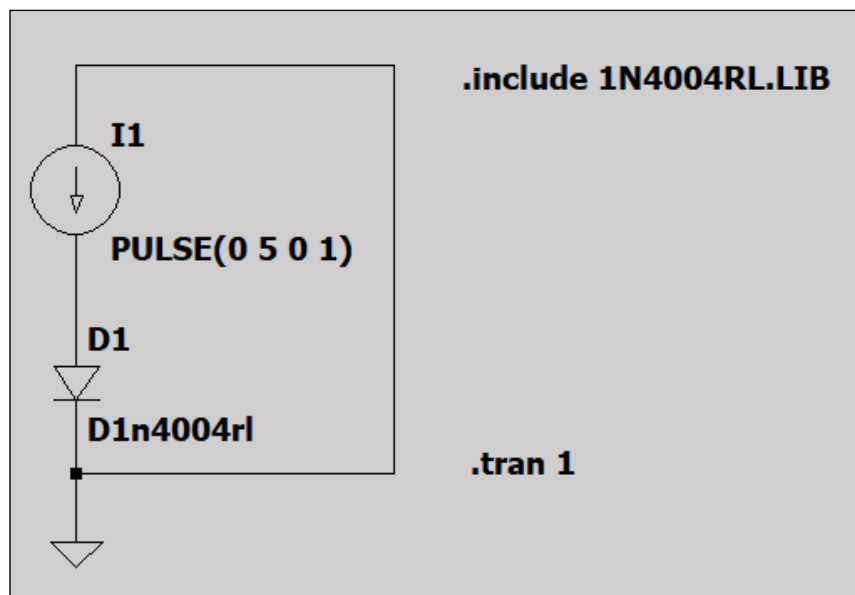
Save the file with the SPICE model in the project folder.



Open the LTspice project and add a SPICE directive: “.include 1N4004RL.LIB”.



Then change the value of the diode to the name of the model, which is “D1n4004rl”. You can right click on “D” next to the diode or CTRL + right-click on the diode.



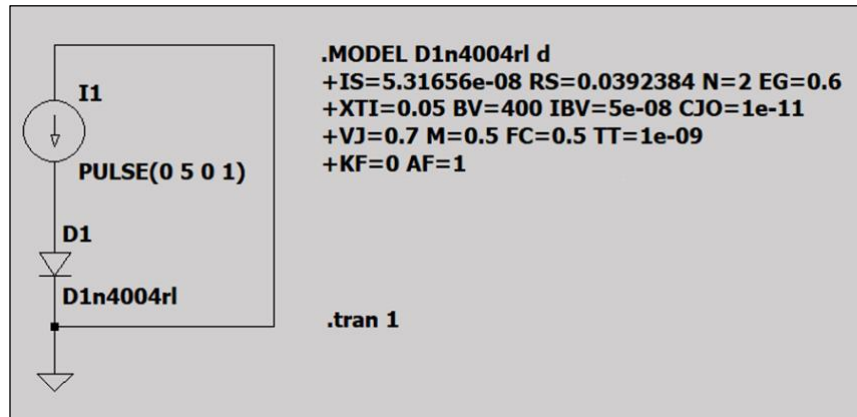
That’s it. Now we can use the new diode in the simulation.

Method 2

Open 1N4004RL.LIB with any text editor. Copy and paste the model as a SPICE directive into the LTspice schematic.

Change the value of the diode to the name of the model. We don't need to save 1N4004RL.LIB in the project folder, and we don't need to ".include" it.

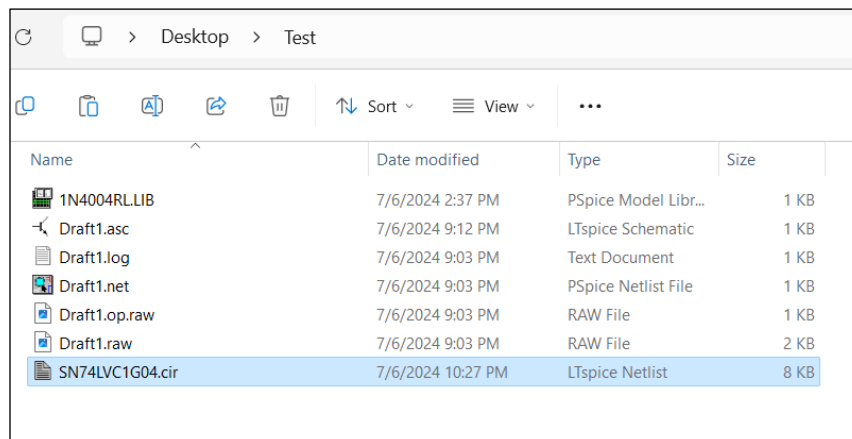
The entire model is now in the LTspice schematic.



Method 3

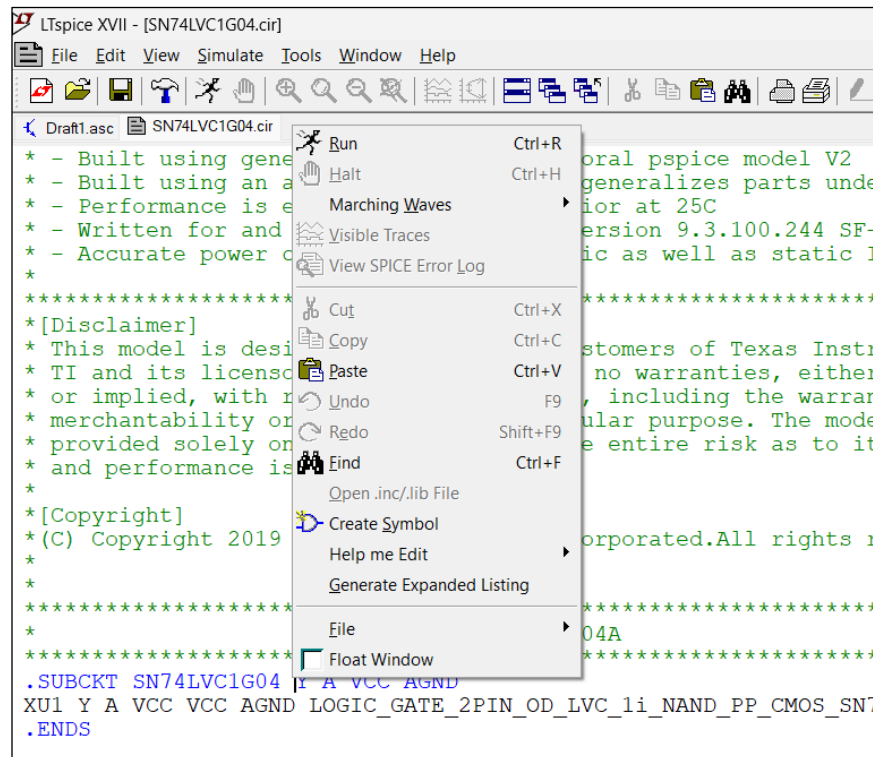
This method is for complex models. These models are defined with the ".SUBCKT" and usually consist of a few simple models that start with ".MODEL".

Download the PSpice model of SN74LVC1G04 Inverting Schmitt trigger and save it in the project folder. In this case, it is "SN74LVC1G04.cir".



Open your LTspice project. Then, go to File > Open. Change the file type to “All Files” and select the file with the SPICE model in the project folder.

Right-click on the part name next to the “.SUBCKT” and select “Create symbol”. LTspice will ask you to automatically create a symbol. Click “Yes”.

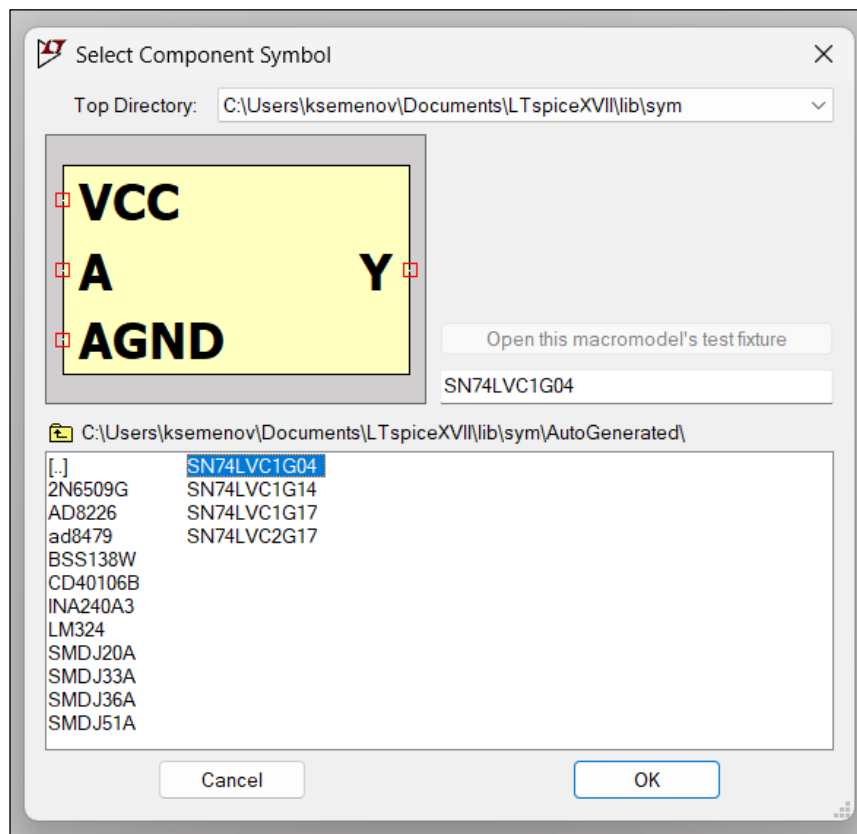
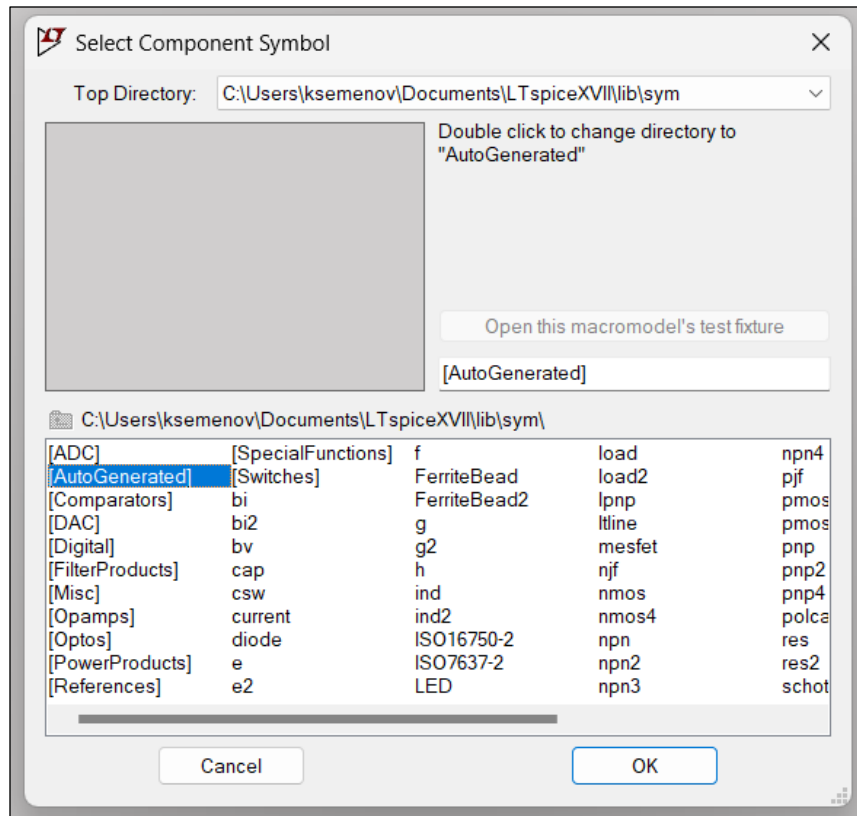


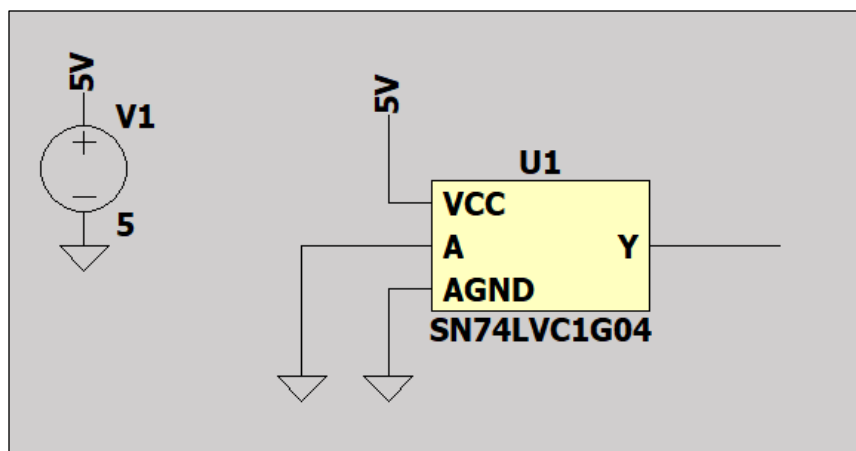
A new symbol will be autogenerated. You can change the shape of the component and move the pins. Save the changes and close the symbol editor window.

Open your LTspice schematic and press “F2” to place a component.

All newly autogenerated symbols by default are saved in:
C:\Users\<your_username>\Documents\LTspiceXVII\lib\sym\AutoGenerated.

Find our new component and place it.



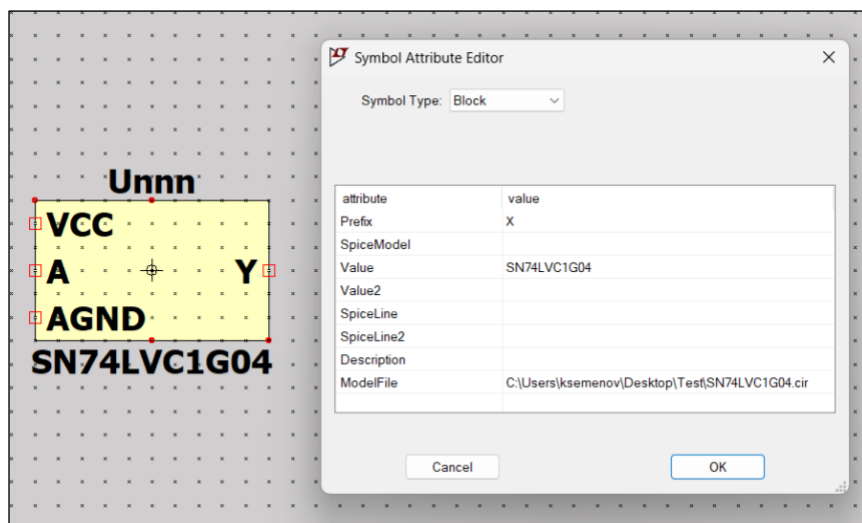


Simulate the circuit and make sure it works.

This symbol is just a visual representation of the component. All its electrical parameters are contained in the SPICE file that we downloaded and saved in the project folder. The symbol is linked to the model, and LTspice will automatically include the models in the background for all components.

Right-click on the component and then click on “Open symbol”. A new window will open where we can view and modify the symbol.

Then go to Edit > Attributes > Edit Attributes.



ModelFile field shows the path to the file with the SPICE model. Right now, it displays the full path to the project folder on my computer. If you want to use the newly imported models only on your computer, you can save all of them in one folder. For example, it could be the default LTspice folder in the Documents folder: **C:\Users\<your_username>\Documents\LTspiceXVII\lib\sub.**

Sometimes we need to share our simulation file with others, and they might not have all the models on their computers. I think the best solution in this case is to save all newly imported models in the project folder.

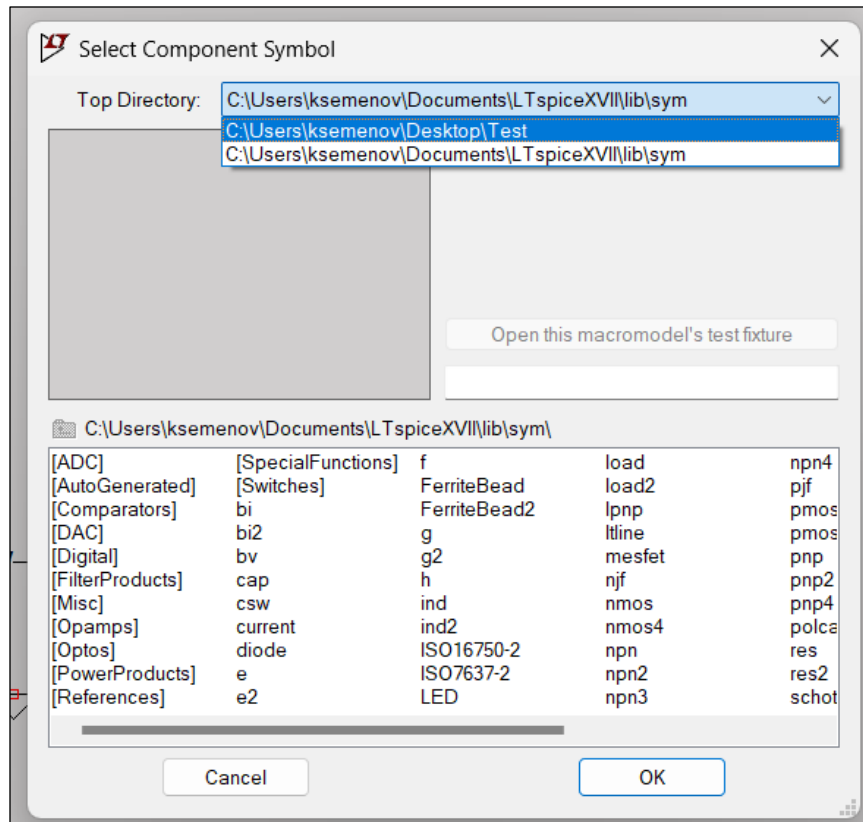
Right now, the ModelFile field shows the full path to the project folder, and if we move or rename the project folder, the path will point to the wrong location. We can delete the beginning of the path and leave only “SN74LVC1G04.cir”.

Now, it is a relative path with respect to the project folder. LTspice will likely try to find the file in its internal folders and then search in the project folder. Save the edits and go back to the schematic. You need to delete the component and place it again because it still remembers the old settings.

At this point, our symbol is located in: **C:\Users\<your_username>\Documents\LTspiceXVII\lib\sym\AutoGenerated** and is linked to the SPICE model in the project folder.

The next step is to move the symbol to the project folder. Go to the folder where the symbol is saved. It should be in: **C:\Users\<your_username>\Documents\LTspiceXVII\lib\sym\AutoGenerated**. Cut and paste the “SN74LVC1G04.asy” file into the project folder.

Then, go to the LTspice schematic and delete the component. Press “F2” to open the component selection window. Select the project folder in the “Top directory” field because our symbol lives in the project folder now. Choose our component and place it in the schematic.



Finally, the symbol is saved in the project folder and is linked to the SPICE model in the same folder. Now, you can send the project folder to anyone, and they can run the simulation without manually importing the missing models of the components.

