

There are several options for opamps in LTSpice. LTSpice has many default models for industry parts preloaded. Unfortunately the 741 is not one of them.

UniversalOpamp2 is a good compromise. The default values for are pretty close to the 741 values. You can change these values for a more accurate simulation by right clicking on the symbol and editing the parameters in the Value and SpiceLine fields.

The symbol Opamp is a good basic opamp model. It is the model I use in most of the inclass demonstrations. The advantage of this model over UniversalOpamp2 is that it doesn't require power connections. This makes the schematic simpler and less crowded which is nice for class demonstrations. Unfortunately, LTSpice is not setup to use this opamp by default. We have to do some customization.

The steps below will explain a couple of ways to modify LTSpice so that you can use the Opamp model and simulate any of the in class demonstrations. The files needed for this are in the Spice directory under My Courses.

- 1) Download the file opamp.sub into any directory. Opamp.sub is a generic opamp model.
- 2) In your schematic add an instance of opamp.

*Option #1:*

1. 1) Add a line to the schematic that calls the .sub file. What this should look like is illustrated in Figure 1.
1. Click on the “.op” command on the top toolbar
2. Your new command is a “spice directive” not “comment” and it should be *.lib path/opamp.sub* Path is the full pathname to where you have stored your file. You may omit the path part of the .lib command if you store the file in the default path location: *C:\Program Files\LTC\LTspiceIV\lib\sub*

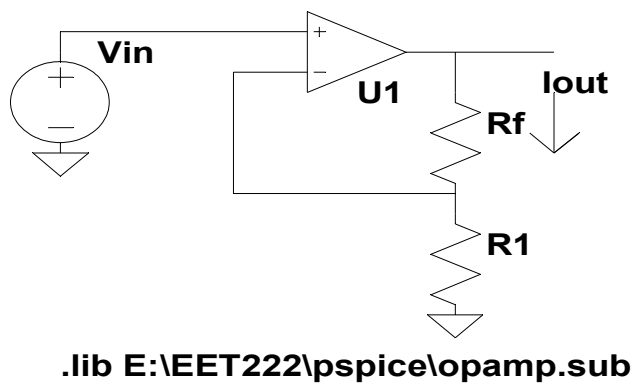


Figure 1: Sample schematic showing .op command to include opamp.sub

Option #2:

- 1) Store opamp.sub in the default subcircuit directory for LTSpice; *C:\Program Files\LTSpiceIV\lib\sub*
- 2) Now place a copy of the opamp component in your schematic. Right click on the component to bring up the window shown in Figure 2.

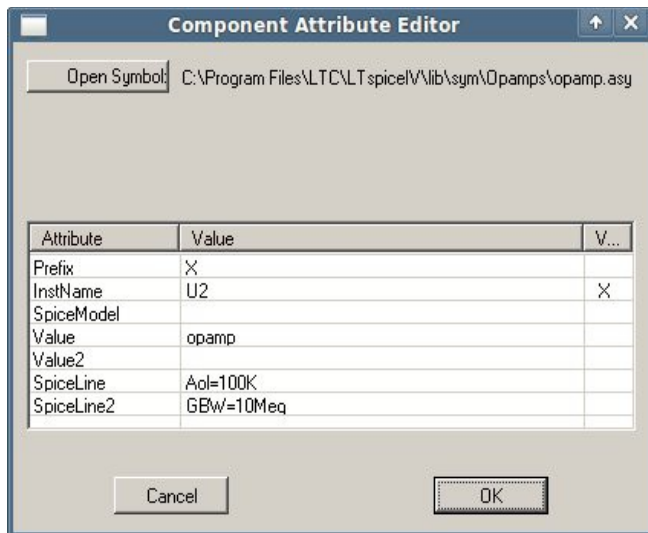


Figure 2: Component Attribute Editor Window

- 3) Click on the *Open Symbol* button. This will open a copy of the symbol in another window.
- 4) In the symbol window, select *Edit* → *Attributes*. You should see a window similar to Figure 3.

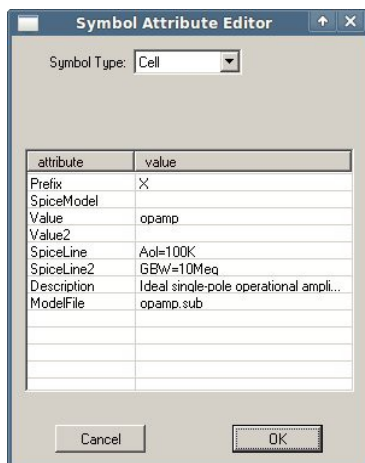


Figure 3: Attribute Editor Window

5) Make sure the *ModelFile* line is correct. Figure 3 shows the correct value for this line. The *ModelFile* will most likely be blank.

6) You should now be able to simulate using either Opamp or UniversalOpamp2. Neither case will require you to include and load any additional libraries or files

### *Final Comments*

\* An additional subcircuit file is included on the website. This file is *lm741.sub*. It is a model file downloaded from National Semiconductor's website. This file should be a more accurate model of the lm741. It includes input offset and more accurate slewing. In order to use this file you will need to modify the UniversalOpamp2 symbol and create an lm741 symbol. The following link explains how to go about doing that:

<http://jeastham.blogspot.com/2009/10/adding-lm741-op-amp-model-to-ltspice.html>

\* If you get an *Unknown Subcircuit* error, it most likely means that the .sub file is not specified properly. Either the path is incorrect in Option #1 or all the steps of Option #2 have not been set up properly. You should *View* → *Spice Netlist*. At the end of the file there should be a line *.lib opamp.sub* (or *UniversalOpamp2.sub*). This file should include a model for the opamp that you are simulating.