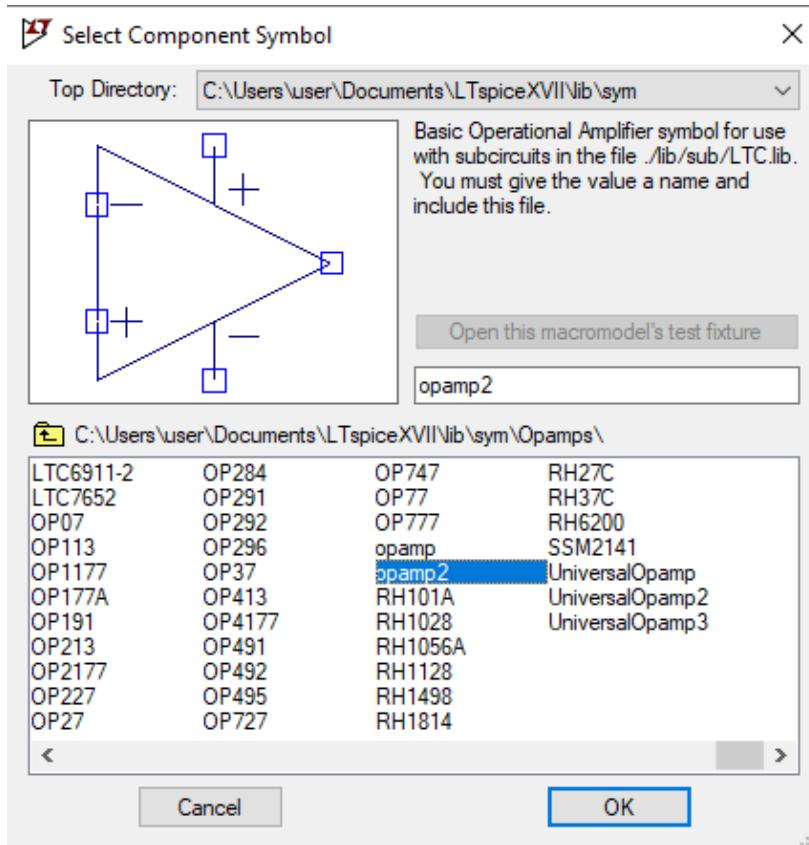


## Steps to install the opamp spice model

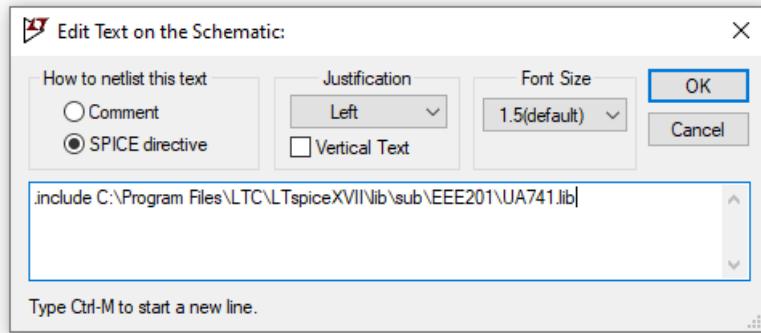
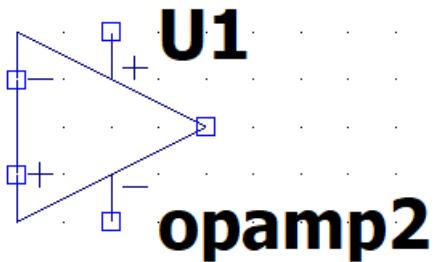
1. Open a new schematic in LT spice and click on *Component* to select the components you want to use. Under components click on the folder labelled *Opamps* and search for a component labelled *opamp2* and select it and drop it on the schematic.



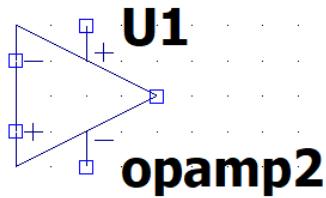
2. Click on the icon called Spice Directive which is circled in the figure shown below. A popup box appears.



Copy the file directory name of the file you labelled as **UA741.lib** and paste it in the popup box in LT spice as **.include <file directory>** . This means you want to include the spice model of the opamp in your schematic. An example of how this is supposed to look like is shown below.

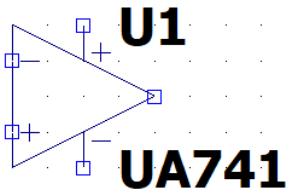


After writing this you can click OK. You can drop the text anywhere on the schematic and it should look like this now.



**.include C:\Program Files\LTC\LTspiceXVII\lib\sub\EEE201\UA741.lib**

3. Right click on the opamp and change its *Value* to UA741 and the final schematic should look like this:



**.include C:\Program Files\LTC\LTspiceXVII\lib\sub\EEE201\UA741.lib**

**NB:** Its is very important to remember that opamps must be biased else your circuits will not work. You can bias your opamps as shown below and take note of the polarities of the biasing voltage sources.

