

Adding a Model of the 741 Op Amp to LTSPICE

Step 1) Create a new folder in the LTSPICE subcircuit directory. It is probably something like C:\Program Files\LTC\LTspiceIV\lib\sub on your computer. For simplicity, I called it BEE215. So you should end up with something like:

```
C:\Program Files\LTC\LTspiceIV\lib\sub\BEE215
```

Step 2) Create a new folder in the LTSPICE symbol directory. It is probably something like C:\Program Files\LTC\LTspiceIV\lib\sym on your computer. Also call this folder BEE215. So you should end up with something like:

```
C:\Program Files\LTC\LTspiceIV\lib\sym\BEE215
```

Step 3) Find a spice model for the 741 op amp (usually with a .MOD file extension). A first guess at a good source would be the manufacturers website, and sure enough, National Semiconductor has a model file available for the LM741. This particular model can be found here:

```
http://www.national.com/models/spice/LM/LM741.MOD
```

Download this small file and copy it to the sub\BEE215 directory you created in step 1. So this folder should now contain a file called LM741.MOD.

```
C:\Program Files\LTC\LTspiceIV\lib\sub\BEE215\LM741.MOD
```

Step 4) Now for a slightly tricky part. Open up the LM741.MOD file you downloaded with a text editor such as Wordpad. Find a line in the file that should read: .SUBCKT LM741/NS 1 2 99 50 28

Edit the text LM741/NS to read LM741 (remove the /NS)

The line should now read: .SUBCKT LM741 1 2 99 50 28

Save the file into the same directory as LM741.lib. You can delete the LM741.MOD file now if you want to.

LTSPICE subcircuit files need the .lib extension.

Step 5) Now we need a symbol for the spice model of the 741 op amp we downloaded. You can create your own symbols from scratch, but it is easier in this case to just copy an existing symbol of a similar component and modify it. The symbol for the generic part opamp2 in the LTSPICE symbol directory will do nicely. On your computer it is probably located at:

C:\Program Files\LTC\LTspiceI\lib\sym\Opamps\opamp2.asy

Copy this file to the symbol directory you created in step 2 and rename it to LM741.asy. This folder should now contain:

C:\Program Files\LTC\LTspiceI\lib\sym\BEE215\LM741.asy

(note that we are using the symbol directory created in step 2 and not the subcircuit directory created in step 1)

Step 6) Start up LTSPICE and select File->open. Change the "Files of type:" to Symbols (*.asy) and navigate to the sym\BEE215 directory you created in step 2. Open the file LM741.asy that should now be in this directory. You should now have an exploded drawing of a basic op amp on your screen.

Step 7) From the LTSPICE main menu bar, choose Edit->Attributes->Edit Attributes (or just press Ctrl A). This should bring up a Symbol Attribute Editor window. On the third line, change the Value attribute from opamp2 to LM741. On the eighth line (ModelFile attribute), you need to type in the name of the model file LTSPICE will use for the LM741 symbol. This is the file you downloaded, edited and saved as LM741.lib in step 4. Specify the directory (BEE215) and the filename (LM741.lib) by adding the text BEE215\LM741.lib to the ModelFile attribute line. Do not include a "\" before the BEE215. You can also change the Description attribute on line 7 to whatever you want, or leave it as is. From the LTSPICE main menu bar, choose File->Save. Close all LTSPICE windows.

Step 8) Your new component should now be in a new component subdirectory called BEE215. Start LTSPICE and select new schematic. Choose component and click the BEE215 directory. You should be able to select a LM741 op amp and place it on the schematic. Done!