How to Install and use the National Semiconductor LM741 OpAmp Subcircuit Model in LTspice Netlists

Let's suppose you want to simulate the circuit shown in Figure 1, with a realistic (non-ideal) model for the LM741 operational amplifier.
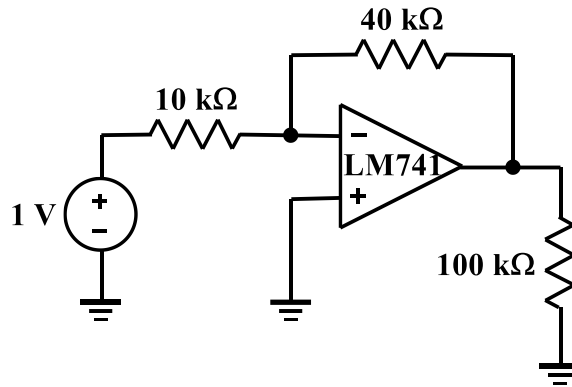

Figure 1

LTspice doesn't include a model for the LM741 operational amplifier, so I'll start by doing an internet search for "LM741 SPICE model". Several possibilities are found. Among them is a link to the Texas Instruments website for a model that was developed by National Semiconductor, as shown in Figure 2. (National Semiconductor became the "Silicon Valley" division of Texas Instruments in 2011.) On the TI site it's called a PSpice model, but that doesn't matter because LTspice and PSpice use the same structure and syntax for such models, as do virtually all SPICE-based simulators.

Figure 2

Clicking on this link takes you to http://www.ti.com/product/LM741/toolssoftware (See Figure 3.), where you can download a compressed file named **snom211.zip** that contains the file we really want – **LM741.MOD** – a SPICE model file.

Figure 3

Download and extract/unzip the file. Note that once it's unzipped, it's a SPICE model file with extension MOD (See Figure 4.).
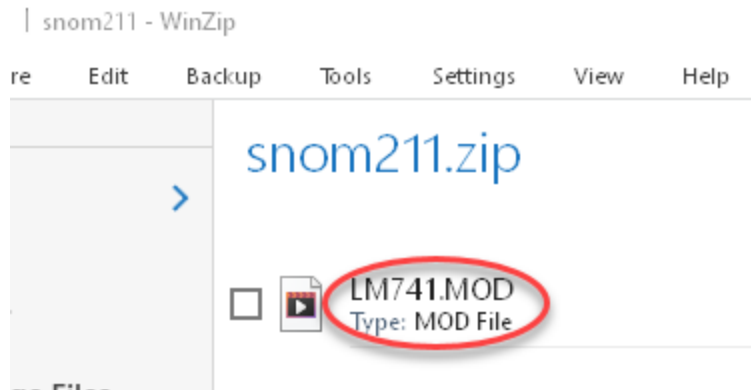

Figure 4

Now you can examine its content – it's a text file, so you can use Notepad – and you'll see that the model is defined as a SPICE subcircuit called LM741/NS (See Figure 5.).

Figure 5

Since it's a subcircuit description, copy the file **LM741.MOD** to the folder **…\Documents\LTspiceXVII\lib\sub** on your computer.

Now you've made the LM741 opamp a permanent part of your LTspice component library, and you're ready to create a netlist and do the simulation. First, label everything as you normally would – my labels are shown in Figure 6 – and then write the netlist required for the simulation. Note how I have added two voltage sources so as to bias the opamp properly, but they are not shown connected directly to the device in order to make the diagram neater than might be otherwise expected. The fact that the node numbers for the bias sources and the terminal connections on the opamp itself are the same means that LTspice will interpret them as being physically connected together.
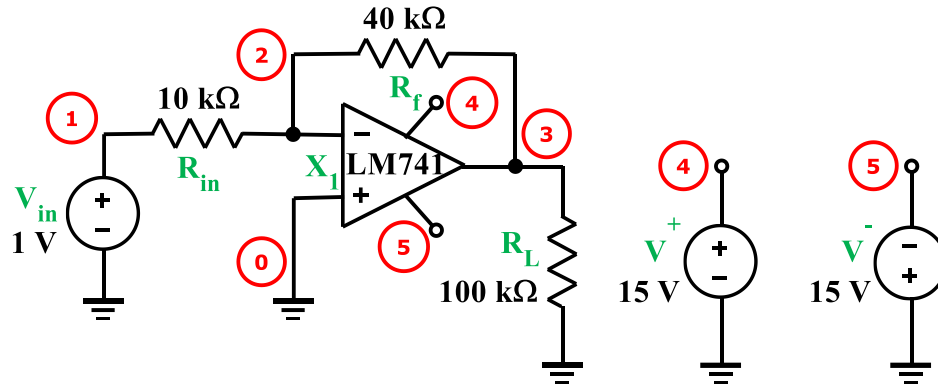
Figure 6

My netlist is shown in Figure 7.

```
* Q:\LTspice\My Stuff\LM741 OpAmp Inverter.cir
.include LM741.MOD
V+      4       0       DC      15
V-      5       0       DC      -15
X1      0       2       4       5       3       LM741/NS
Vin     1       0       DC      1
Rf      2       3       40k
Rin     1       2       10k
RL      3       0       100k
.op
.end
```

Figure 7

Note:
1. Inclusion of the ".include LM741.MOD" directive to tell LTspice where to look for the LM741 subcircuit model, and
2. The opamp requires five node connections, as shown in Figure 8.

Figure 8

The corresponding output is shown in Figure 9.

```
        --- Operating Point ---

V(4):           15              voltage
V(5):           -15             voltage
V(2):           0.00102008      voltage
V(3):           -3.99082        voltage
V(1):           1               voltage
I(Rl):          -3.99082e-05    device_current
I(Rin):         9.9898e-05      device_current
I(Rf):          9.97961e-05     device_current
I(Vin):         -9.9898e-05     device_current
I(V-):          0.00179686      device_current
I(V+):          -0.001657       device_current
Ix(1:1):        5.77868e-08     subckt_current
Ix(1:2):        1.01929e-07     subckt_current
Ix(1:99):       0.001657        subckt_current
Ix(1:50):       -0.00179686     subckt_current
Ix(1:28):       0.000139704     subckt_current
```

Figure 9

As mentioned earlier, now that you have the LM741 model installed, you'll have it available to use in other simulations at any time in the future.

This method of installation will work for virtually any device model you may download (or develop yourself). Most manufacturers have such models available for download from their website.

FYI, the complete content of the LM741.MOD file is shown in Figure 10.

You might also find the LT-wiki *Components Library and Circuits* useful for other applications.

```
*////////////////////////////////////////////////////////////////////
* (C) National Semiconductor, Inc.
* Models developed and under copyright by:
* National Semiconductor, Inc.

*////////////////////////////////////////////////////////////////////
* Legal Notice: This material is intended for free software support.
* The file may be copied, and distributed; however, reselling the
*  material is illegal

*////////////////////////////////////////////////////////////////////
* For ordering or technical information on these models, contact:
* National Semiconductor's Customer Response Center
*             7:00 A.M.--7:00 P.M.  U.S. Central Time
*                       (800) 272-9959
* For Applications support, contact the Internet address:
*   amps-apps@galaxy.nsc.com
```

```
*/////////////////////////////////////////////////////
*LM741 OPERATIONAL AMPLIFIER MACRO-MODEL
*/////////////////////////////////////////////////////
*
* connections:      non-inverting input
*                   |   inverting input
*                   |   |   positive power supply
*                   |   |   |   negative power supply
*                   |   |   |   |   output
*                   |   |   |   |   |
*                   |   |   |   |   |
.SUBCKT LM741/NS    1   2  99  50  28
*
*Features:
*Improved performance over industry standards
*Plug-in replacement for LM709,LM201,MC1439,748
*Input and output overload protection
*
***************INPUT STAGE**************
*
IOS 2 1 20N
*^Input offset current
R1 1 3 250K
R2 3 2 250K
I1 4 50 100U
R3 5 99 517
R4 6 99 517
Q1 5 2 4 QX
Q2 6 7 4 QX
*Fp2=2.55 MHz
C4 5 6 60.3614P
*
***********COMMON MODE EFFECT***********
*
I2 99 50 1.6MA
*^Quiescent supply current
EOS 7 1 POLY(1) 16 49 1E-3 1
*Input offset voltage.^
R8 99 49 40K
R9 49 50 40K
*
*********OUTPUT VOLTAGE LIMITING********
V2 99 8 1.63
D1 9 8 DX
D2 10 9 DX
V3 10 50 1.63
*
**************SECOND STAGE**************
*
EH 99 98 99 49 1
G1 98 9 5 6 2.1E-3
*Fp1=5 Hz
R5 98 9 95.493MEG
C3 98 9 333.33P
*
***************POLE STAGE***************
```

```
*
*Fp=30 MHz
G3 98 15 9 49 1E-6
R12 98 15 1MEG
C5 98 15 5.3052E-15
*
*********COMMON-MODE ZERO STAGE*********
*
*Fpcm=300 Hz
G4 98 16 3 49 3.1623E-8
L2 98 17 530.5M
R13 17 16 1K
*
**************OUTPUT STAGE**************
*
F6 50 99 POLY(1) V6 450U 1
E1 99 23 99 15 1
R16 24 23 25
D5 26 24 DX
V6 26 22 0.65V
R17 23 25 25
D6 25 27 DX
V7 22 27 0.65V
V5 22 21 0.18V
D4 21 15 DX
V4 20 22 0.18V
D3 15 20 DX
L3 22 28 100P
RL3 22 28 100K
*
**************MODELS USED**************
*
.MODEL DX D(IS=1E-15)
.MODEL QX NPN(BF=625)
*
.ENDS
*$
```

Figure 10