# Analog Circuit Design III.
Circuit simulation of a Common Source Amplifier

In this lab, we will use LTspice XVII again with the AMS 350 nm integrated technology library.

Start the circuit design tool by clicking its icon on your desktop or by using the Start menu shortcut (path: "C:\Program Files\LTC\ LTspiceXVII\XVIIx64.exe").

Use the File - New Schematic command to create a new wiring diagram. Components, wiring and simulation commands can be placed here.
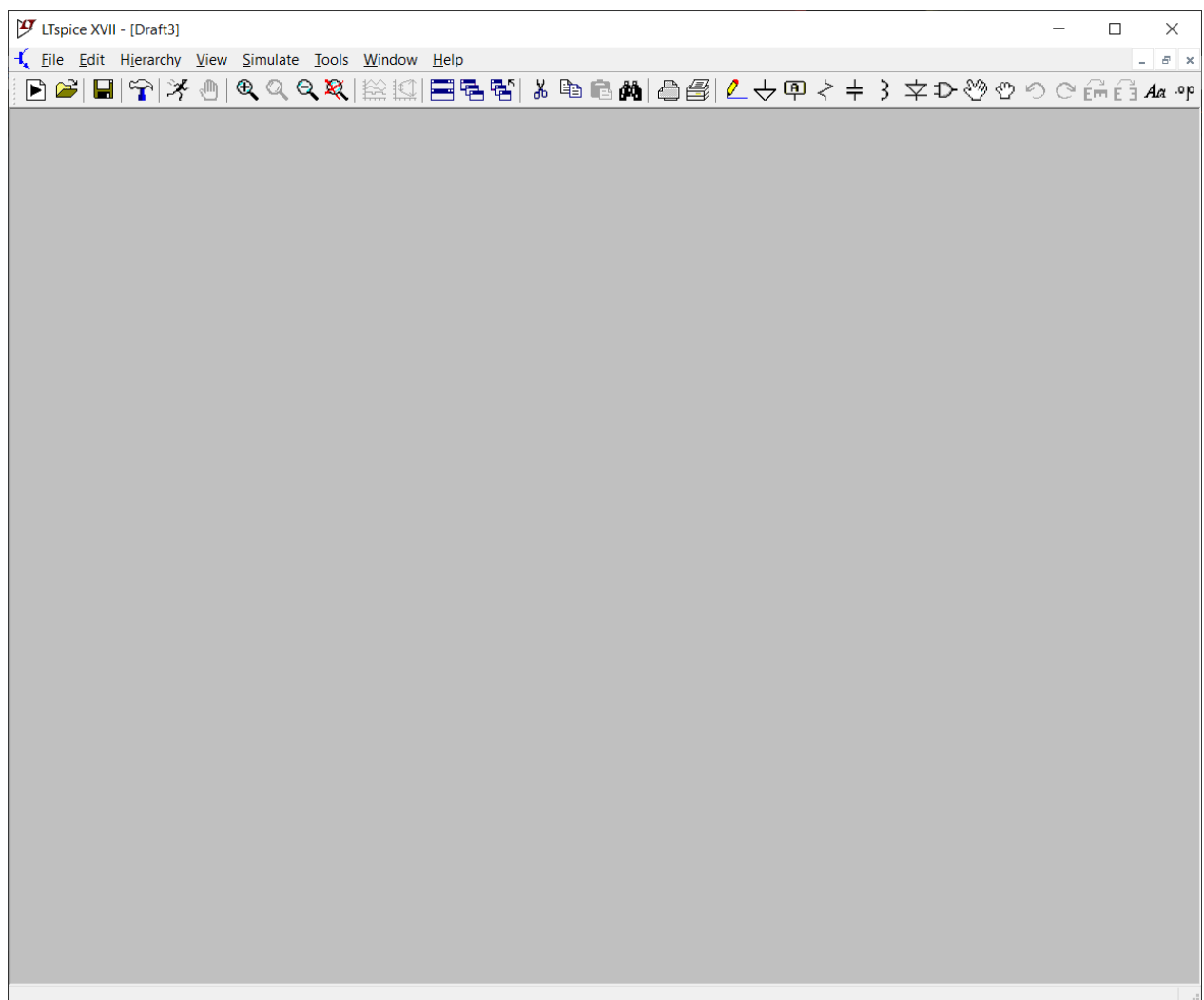
Figure 1. Schematic design window of LTspice

Click on the Edit drop-down menu to see a list of commands needed to create the schematic, with shortcuts enclosed in apostrophes. Some follow logical pattern (e.g. resistance - 'R'), but there are some interesting ones (e.g. wire - F3, undo (Ctrl + Z) - F9).

Simulation of a Common Source amplifier

Select Edit – Component 'F2', where we can select a component from the default folder (*C:\Users\USERNAME\Documents\LTspiceXVII\lib\sym*). The folders are in square brackets and they contain more components. Please choose *AMScellsDigit* folder, select **n4** component *and place it.* (If you cannot find *AMScellsDigit* here, please try to copy the content of the .zip file into the right folder).
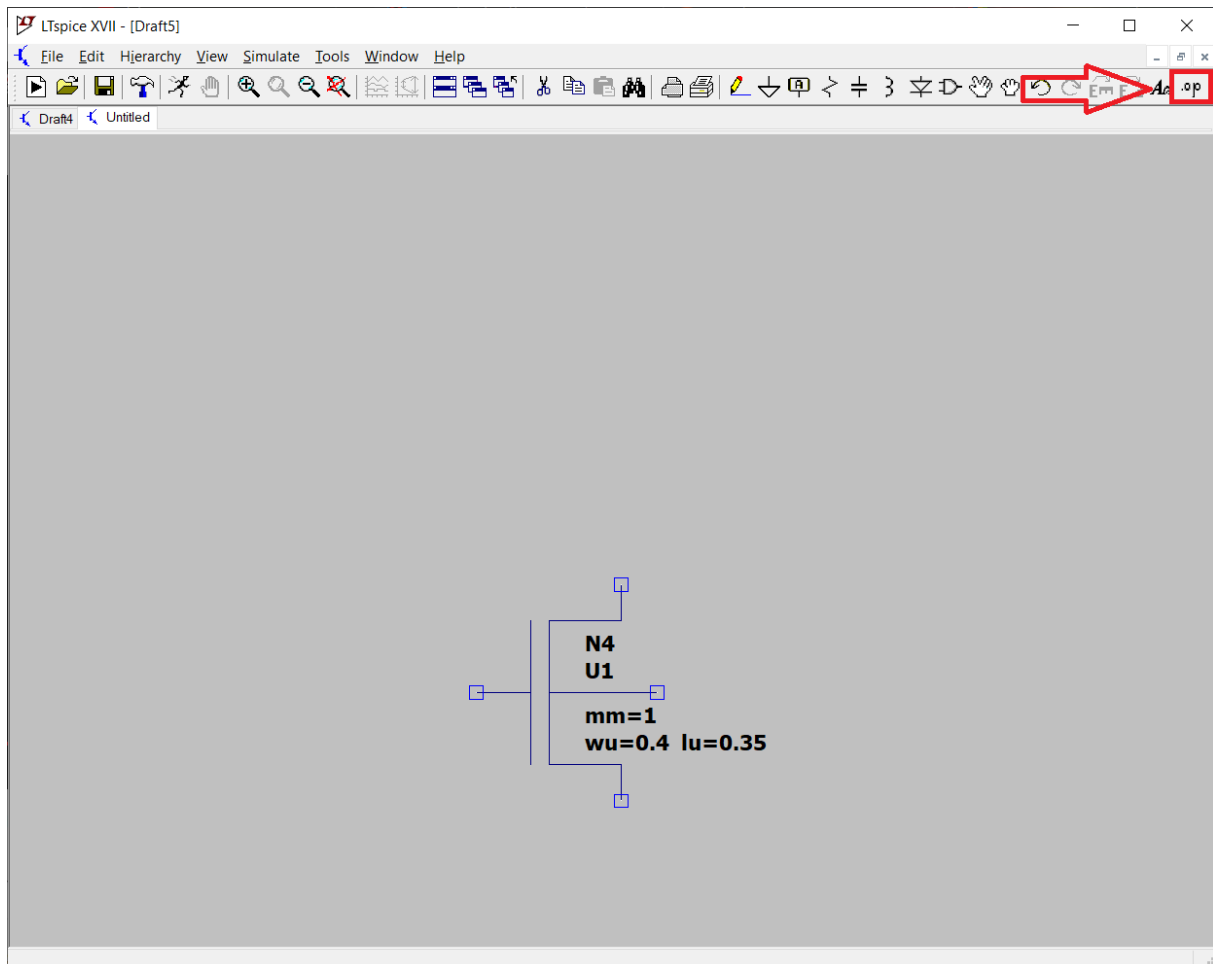

Figure 2. Placement of a MOS transistor

Now we have to click on the .op button (indicated in Figure 2) to create a SPICE Directive. Here we can define the path of the model file. Insert this line below:

```
.include c:\ltspice\sub\AMSLev49Digit.sub
```

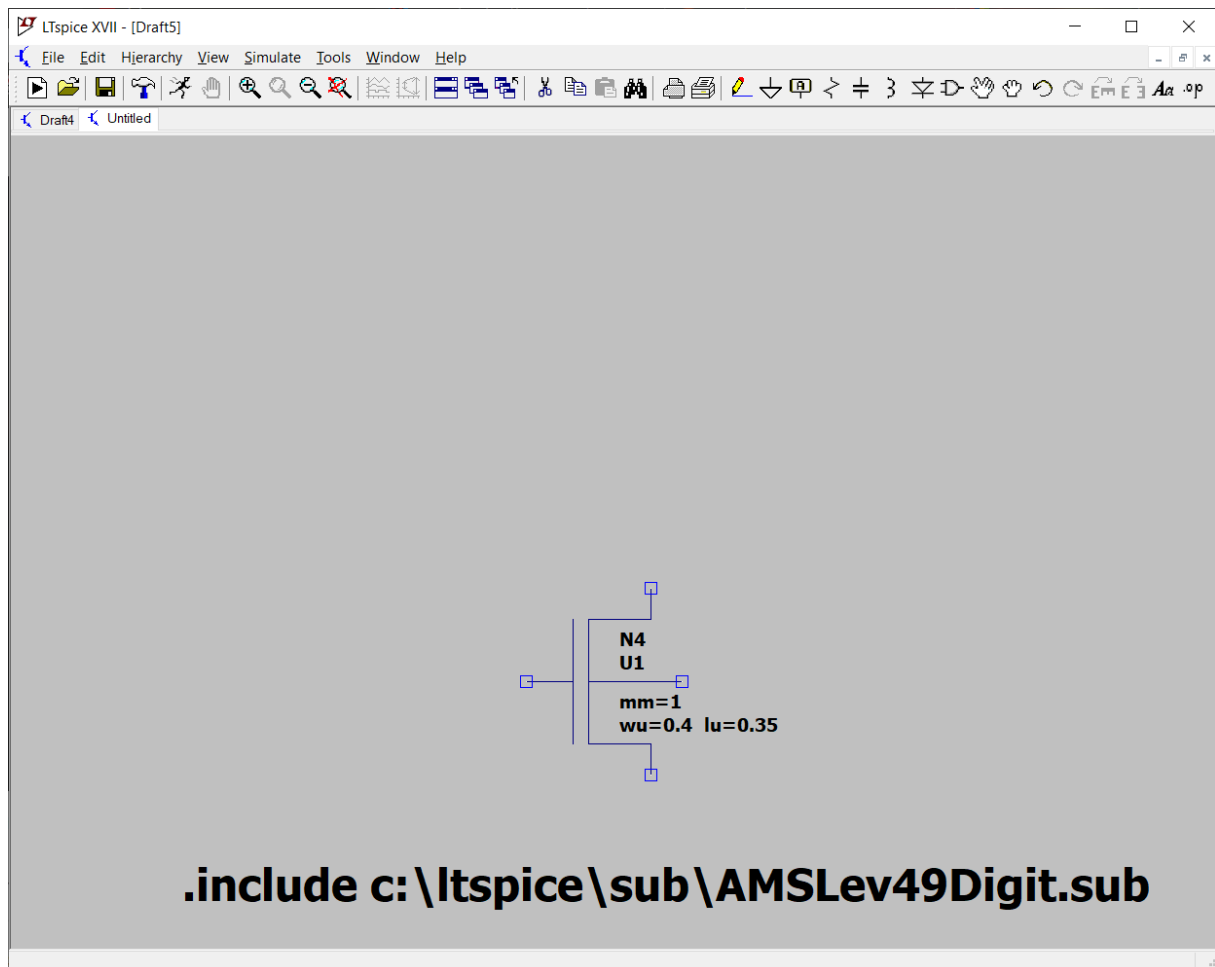Place it somewhere on the schematic (like in Figure 3).

Figure 3. Adding SPICE Directive

Now we have to insert two voltage sources, one for the power supply and one for the input.

From Edit – Component 'F2' choose 'voltage' (you might have to go back from a subfolder clicking on[..]) and place two of it.
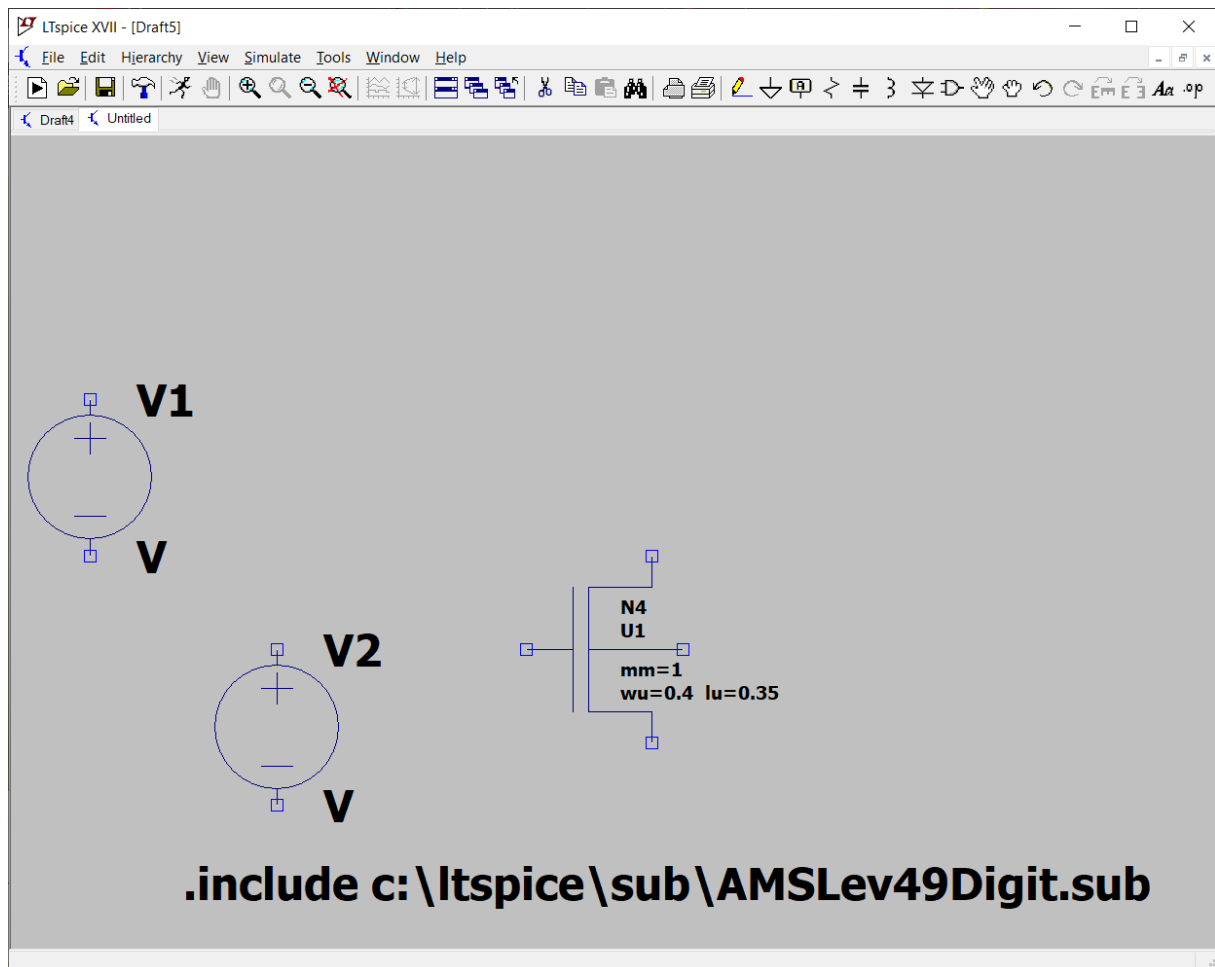
Figure 4. Insert the Grounds

Now place four Ground components (see Figure 5). After you finished it, add an output port to the circuit. Select ⒜ button, Set the Port Type to Output and write *out* into the input field.
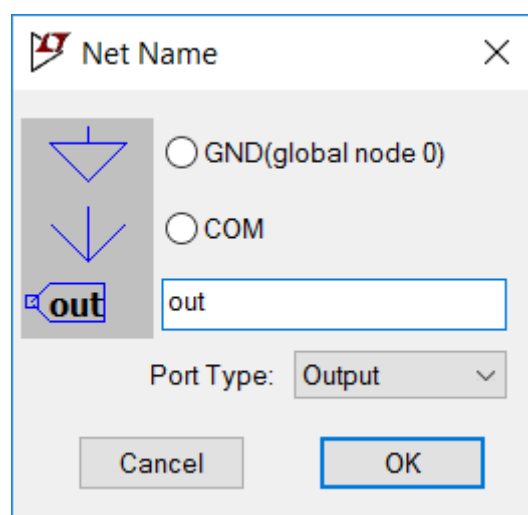

Figure 5. Naming the net

Add a resistor (F2 – res), and wire the circuitry using Edit – Draw wire 'F3' command, as it can be seen in Figure 6.
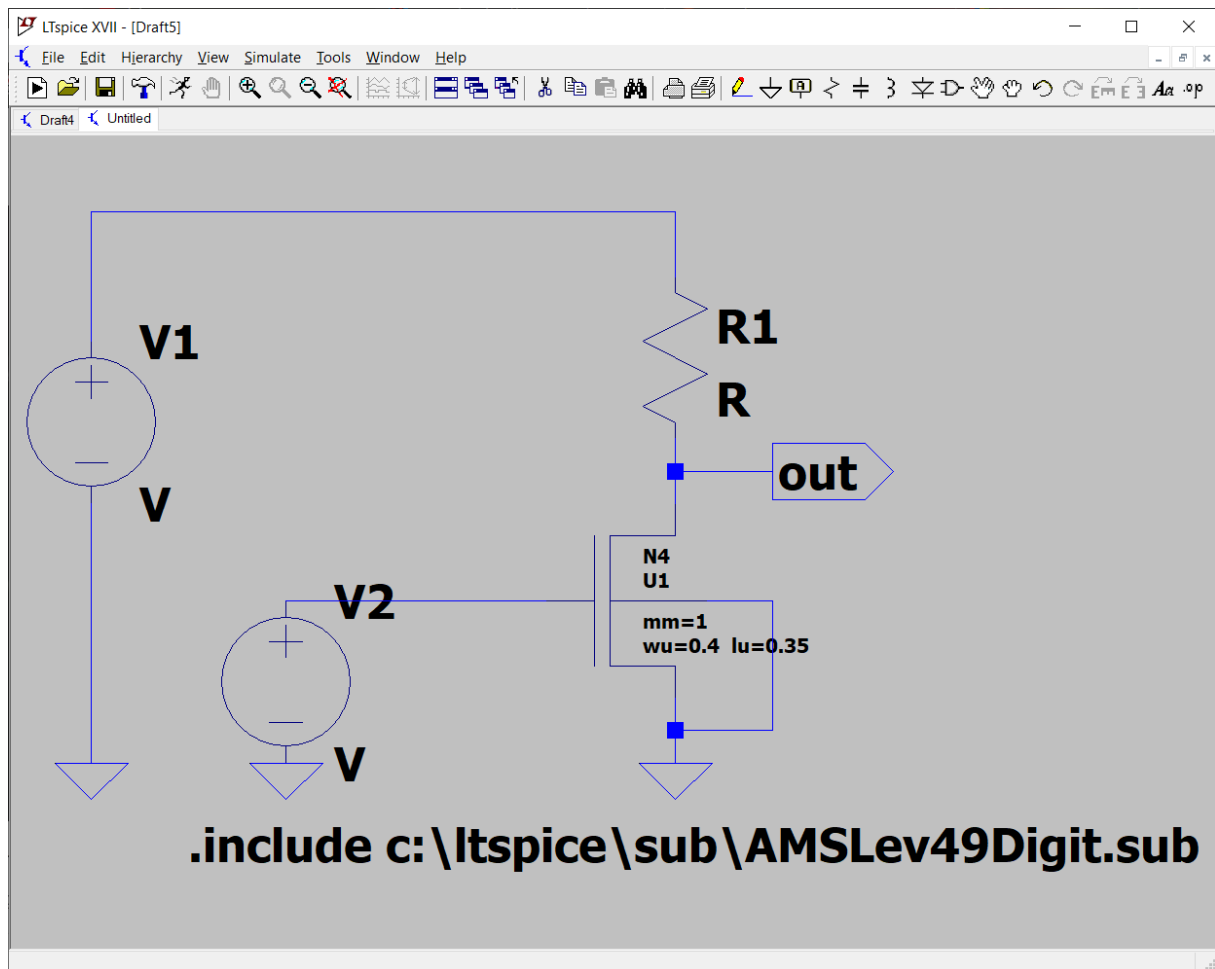
Figure 6. Wiring the schematic

Now we have to add a label for the input wire. To do this, please click on Label Net icon ![icon], type 'in' and place it.
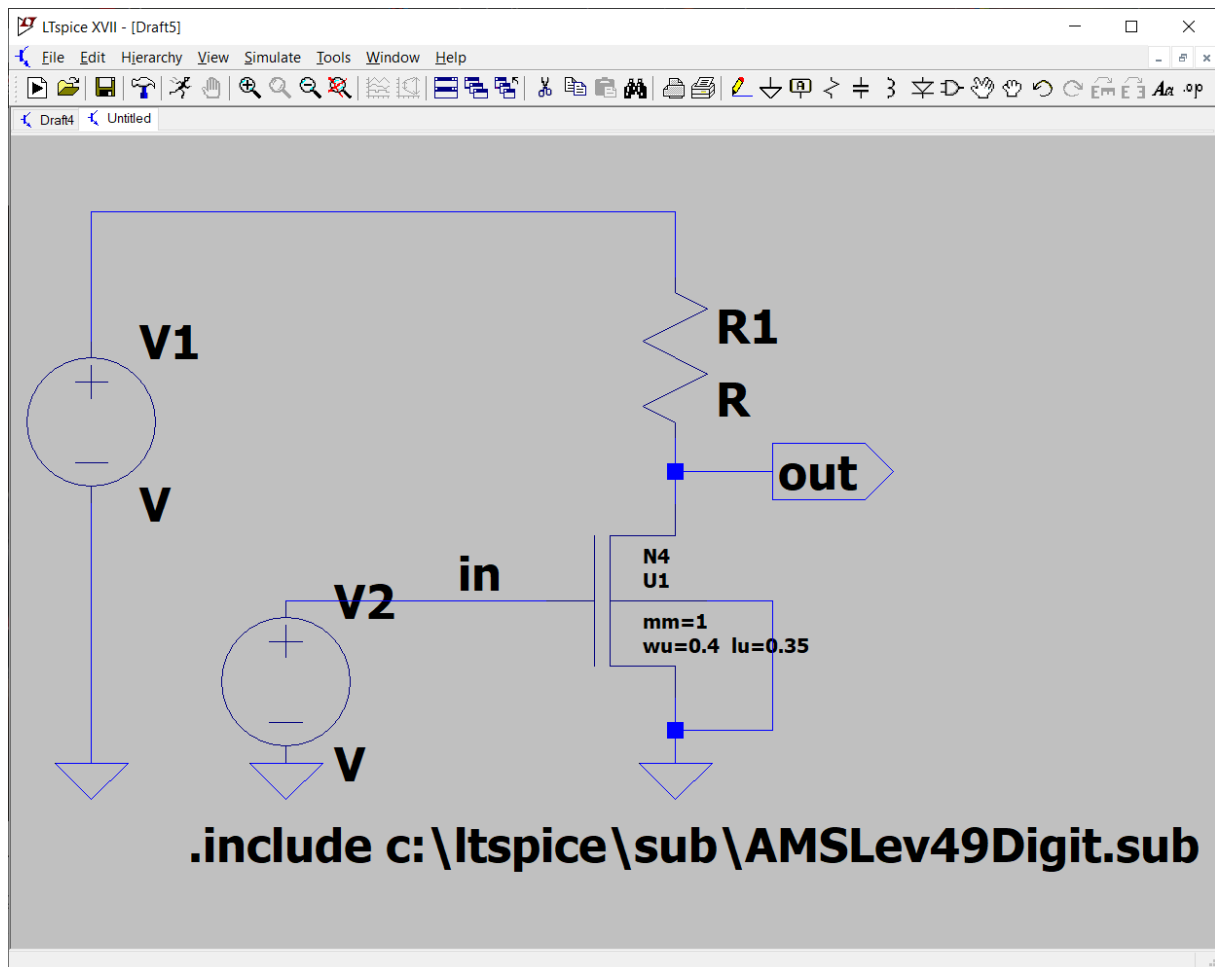
Figure 7. Wired schematic

Now we have to set the voltage of the voltage sources, the resistance of the resistor and the channel dimensions of the MOS transistor. We can do it by clicking the right mouse button on the selected component. The power supply voltage (V1) has to be 5 V (see Figure 8.).
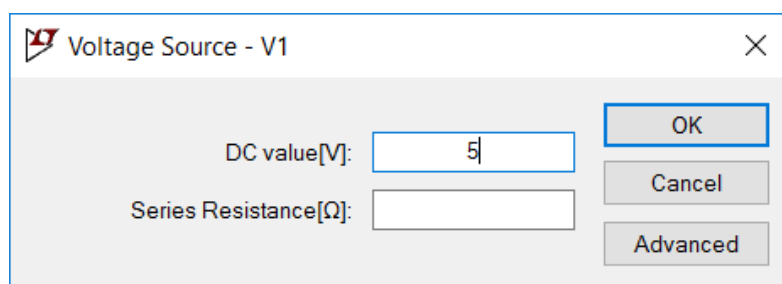

Figure 8. Power supply voltage

For V2, please set 0 Volts (we will modify it later), let R=10k and the channel dimension wu=1, lu=1 (see Figure 9).
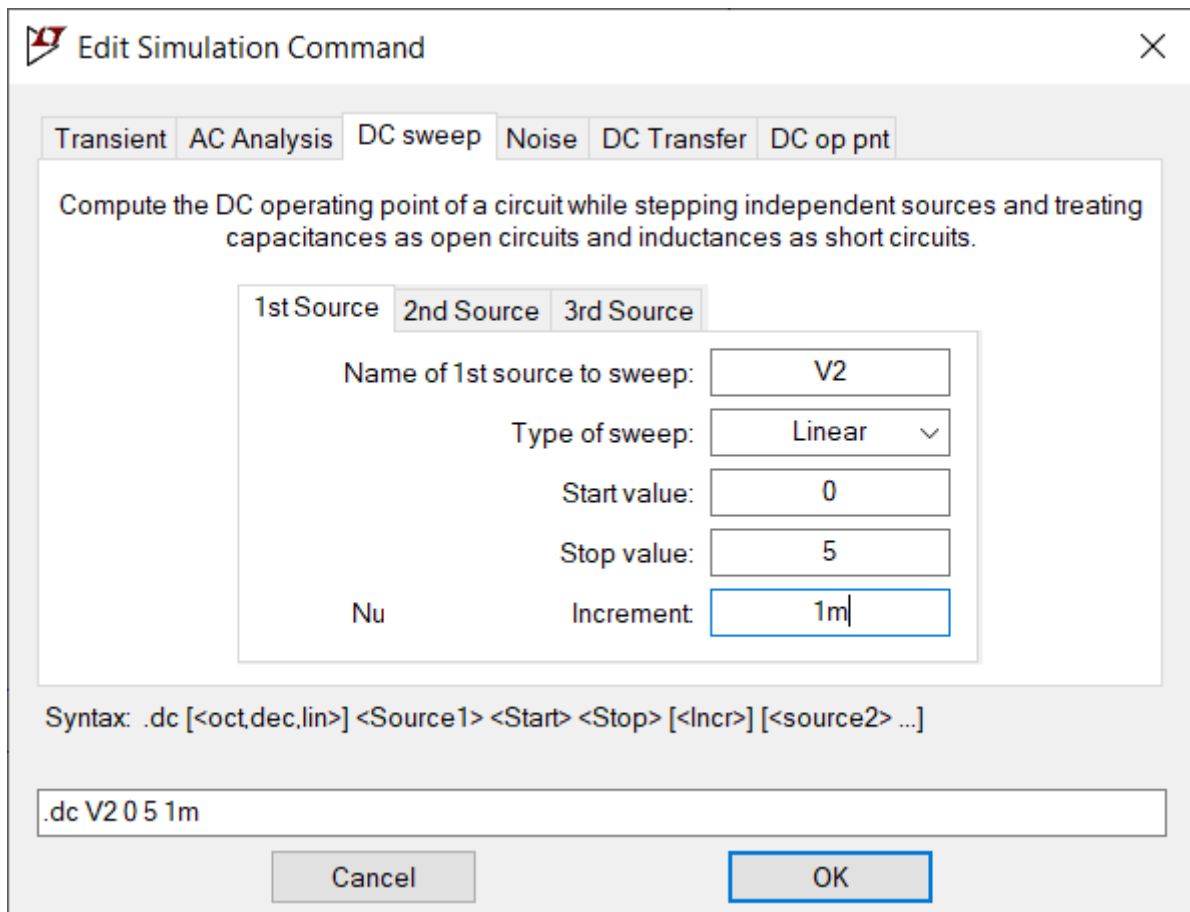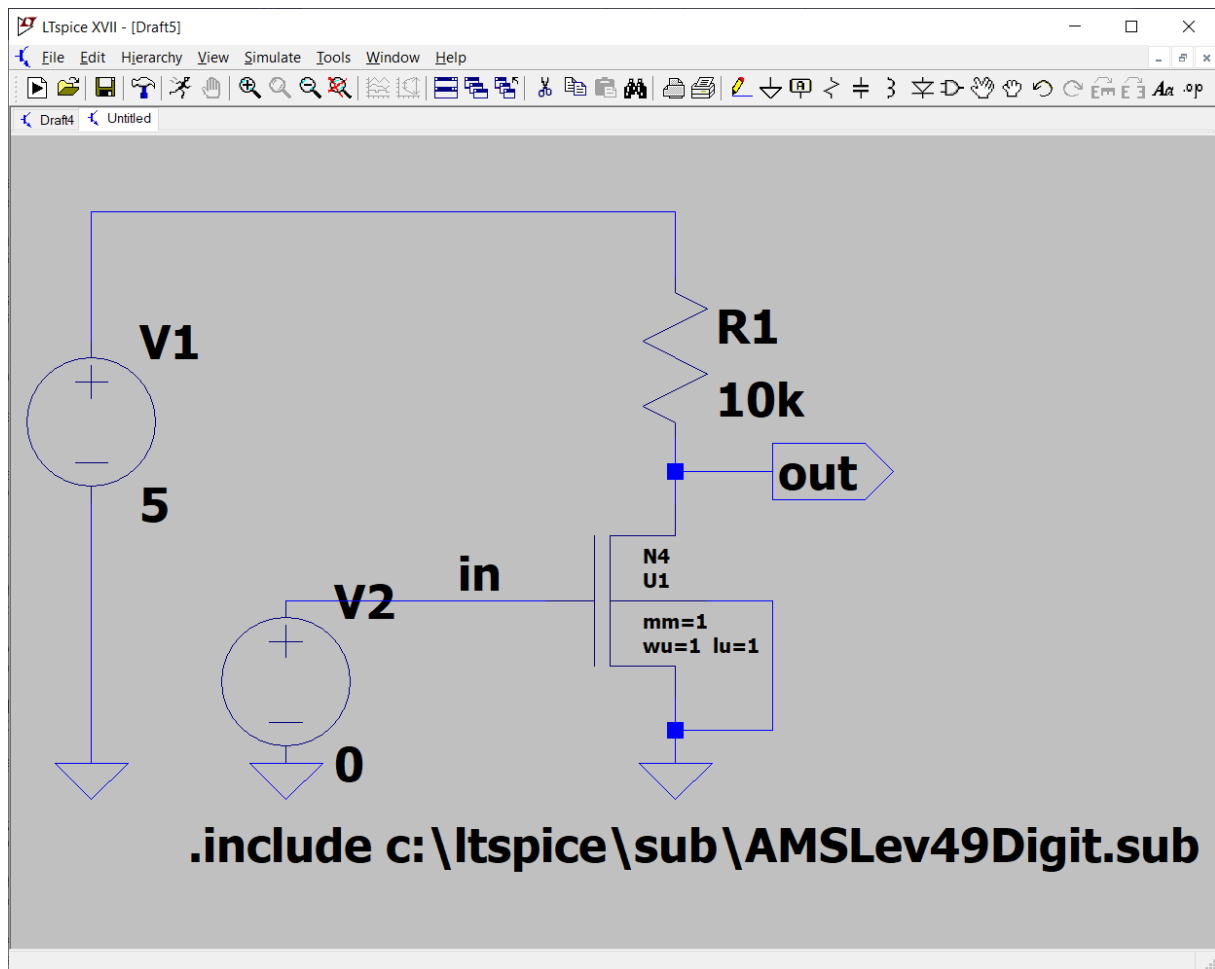
Figure 9. Schematics with component parameters

DC sweep simulation

Now we are ready for the simulation. Select Simulate – Edit Simulation Cmd, and choose DC sweep tab. Fill the input field, as it can be seen in Figure 10.

Figure 10. Simulation settings

If you are done, click on OK, and place the simulation command on the schematic. Select ![icon] button to perform the simulation. If everything is done, you can see an empty diagram like in Figure 11.
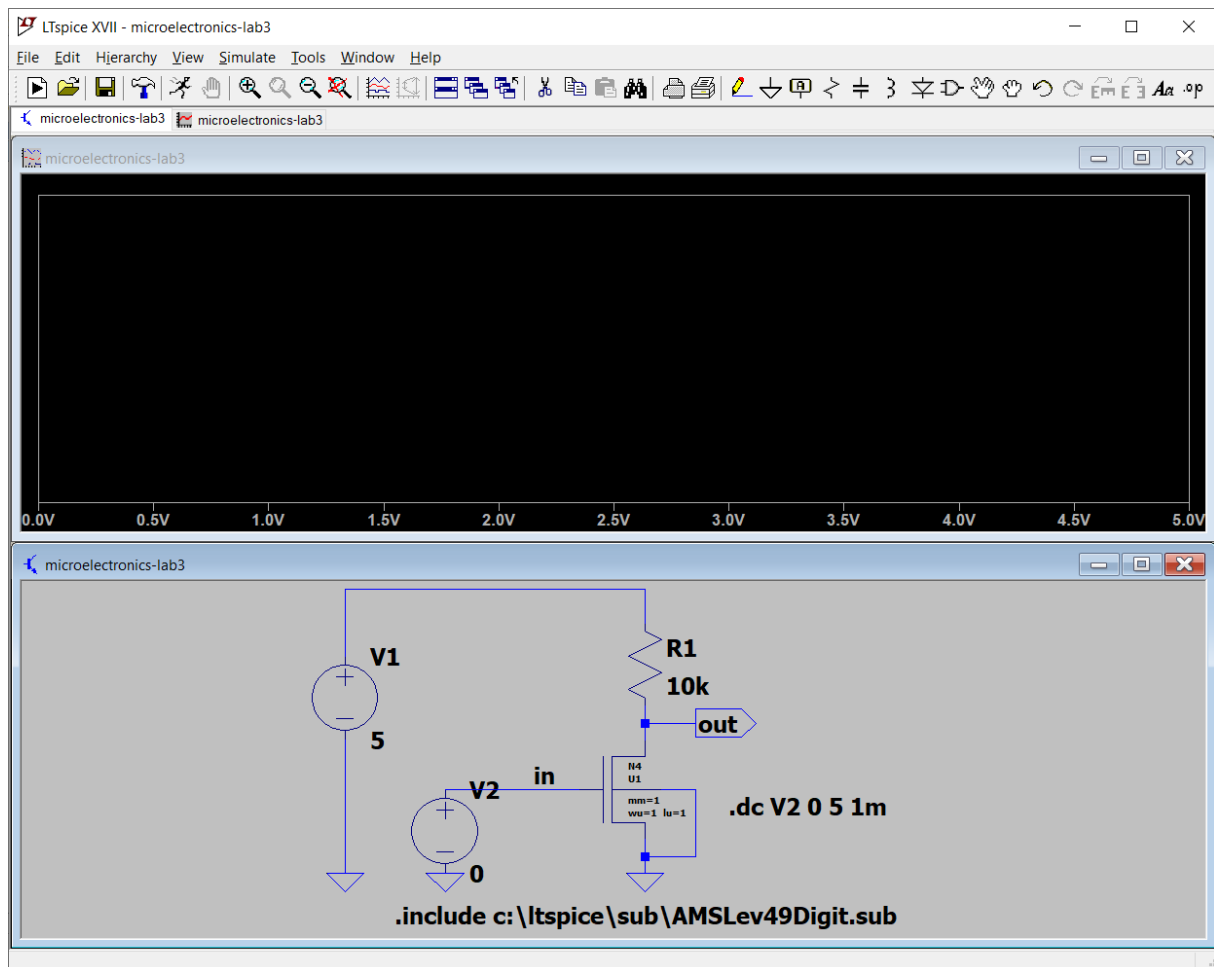
Figure 11. Simulation diagram without waveforms.

If you click on the input and output wires (the wire from the positive terminals of the V2 called 'in'), and the output wire. Please insert the graphs into the lab report.

As you can see, the output voltage cannot reach the 0 Volts, and the slope of the curve is not steep enough. There are two ways to make it better: 1) decreasing the drain current by increasing the resistance of the resistor, 2) increasing the transconductance of the transistor by increasing the width of the channel. Modify the resistance to 50k and the channel width (wu) to 10. Resimulate it. Compare the result with the previous one.

Finding the operating point

Use the cursor to find the operating point when the output node is at half of the power supply voltage (2.5 Volts). Include this value into the lab report, and DC value of V2 voltage source to this value.

AC simulation

First, please deactivate the DC sweep simulation by inserting a semicolon ';' to the beginning of the simulation command (right mouse click on the command to modify). Then, Simulate – Edit simulation cmd, select AC Analysis tab, and set the values as in Figure 12.

Figure 12. AC simulation settings.

After that, please right mouse click on V2 voltage source, then Advanced and set AC Amplitude to 1. Now please perform the simulation. Select the output port to get the Bode-plot. Please insert it to the lab report, and read the amplification of the amplifier.

Increasing the amplification of the amplifier

In this task, please try to increase the amplification of the amplifier. You can modify the resistance of the resistor and the channel dimensions but do not exceed the following limits:

1) The maximum resistance is 100k
2) The maximum transistor area (product of the 'wu' and 'lu') 1000
3) The minimum channel length is 1

Hint: if you modify at least one of the parameters listed above, the operating point will be different, so first, you have to perform a DC simulation to find the proper DC value of the V2 voltage source.

Include the Bode-plot into the lab report, and indicate the amplification of the amplifier.