

Analog Circuit Design I. – Circuit simulation of a CMOS inverter

In this lab, we will use LTspice XVII, a widely used industrial circuit design tool. Since it is freeware, it's free to download and use, and there's no upper limit of nodes, components, or even sub-circuits.

Main parts:

- a) Schematic design editor
- b) SPICE simulation engine
- c) waveform display

Run simulations:

- a) time-domain (transient)
- b) small-signal (AC)
- c) large-signal (DC)
- d) large-signal transfer (DC Transfer)
- e) operating point calculation
- f) noise

The software is also equipped with the features needed to design switching power supplies, which is the main application of this tool today. However, it is not suitable to design printed wiring boards (in a discrete case), and neither to create the physical layout of integrated circuits (layout), nor for logical simulations.

You can download the latest version of the software from the following link:

<https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>

Around the middle of the page, click on *Download for Windows 7, 8 and 10*. A 42 MB installer will start to download. Once it's done, install it on your computer in the usual way. If you have not changed the installation location, you will find the program in the c:\Program Files\LTC\LTspiceXVII\ folder along with other files (parts symbols, components, etc.) required for its operation. In order to design integrated circuits provided by Austria Microsystems (which are N and P channel MOS transistors), we need to add additional symbols and models to the contents of the lib folder. To do this, you need to download the AMS components from the EDU system.

After unpacking (or looking into .zip with Total Commander for example), you will find two folders. Inside the sym folder you can find an *AMScells* subfolder containing two .asy files, which are no less than hierarchical symbols (NMOS, PMOS) edited with a graphical editor. As you look into them with a text viewer (F3 in Total Commander), you can see that the symbol drawings are described in a unique format. The subfolder contains an AMSLev49.sub file which is no less than the description of the transistor model in SPICE language. Here, we are talking about the BSIM3 level 49 transistor description, which is a quite complex model with many parameters.

Copy the „ltspice” folder to the root of C: drive, and copy the „sym” folder C:\Users\USERNAME\Documents\LTspiceXVII\lib. IMPORTANT! DO NOT copy it to C:\Program Files\LTC\LTspiceXVII\ because it will not be visible for the program. In order to use the newly added components, you need to create an .include directive in the schematic, which will be discussed later.

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.

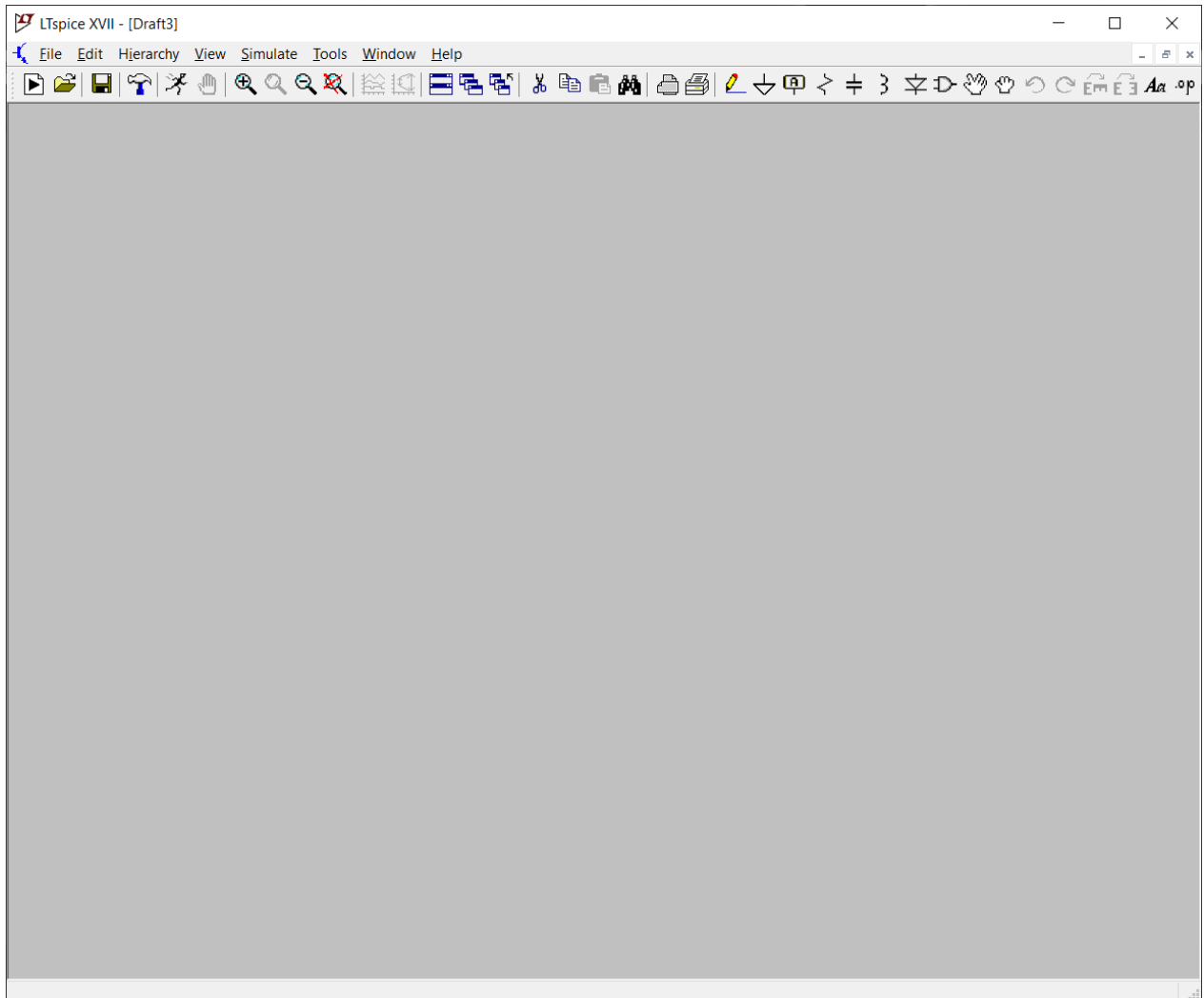


Figure 1. Schematic design window of LTspice

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).

Let's draw an inverter!

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, and **p4** component, and place one. (If you cannot find *AMScellsDigit* here, please try to copy the content of the .zip file into the right folder). Now go back to Edit – Component 'F2', select *AMScellsDigit* folder, and **n4** component, and place one (Fig. 2).

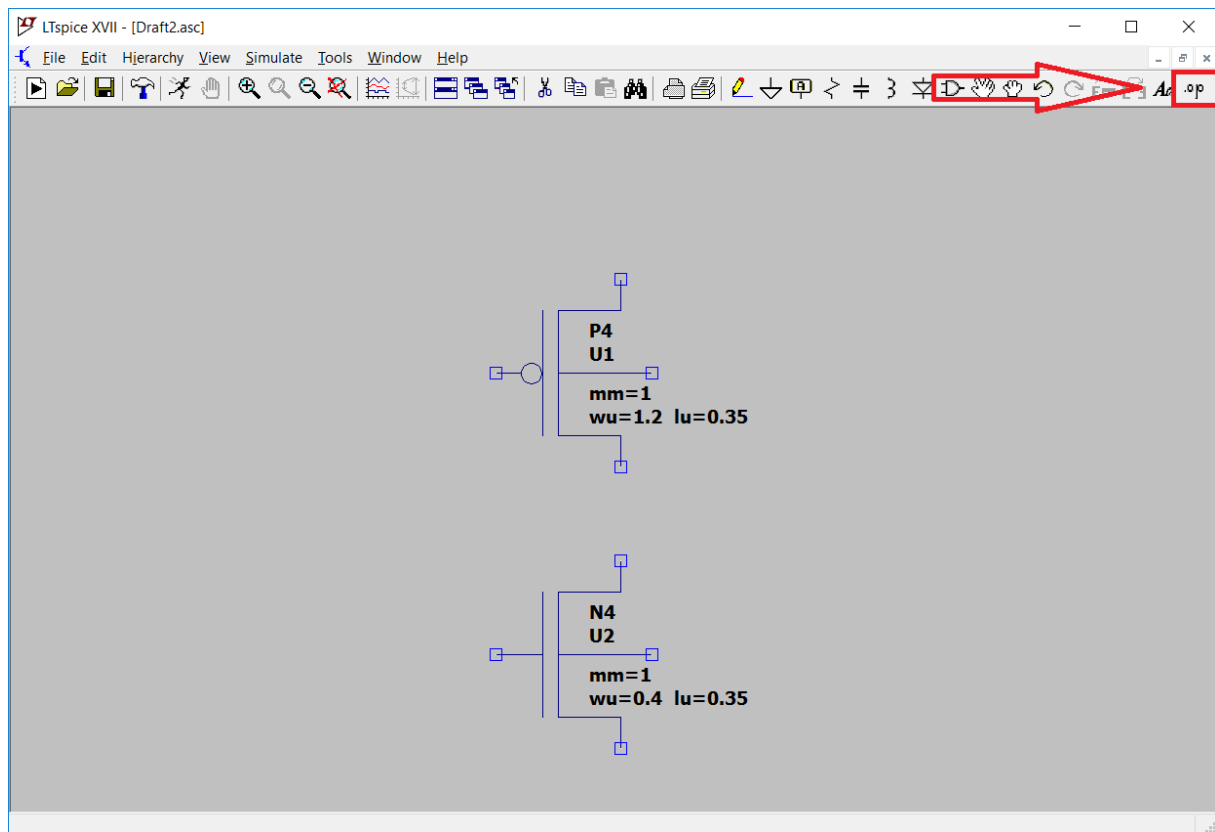


Figure 2. Placement of MOS transistors

Now we have to click on the .op button (indicated in Fig.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
```

And place it somewhere on the schematic (like in Fig.3.).

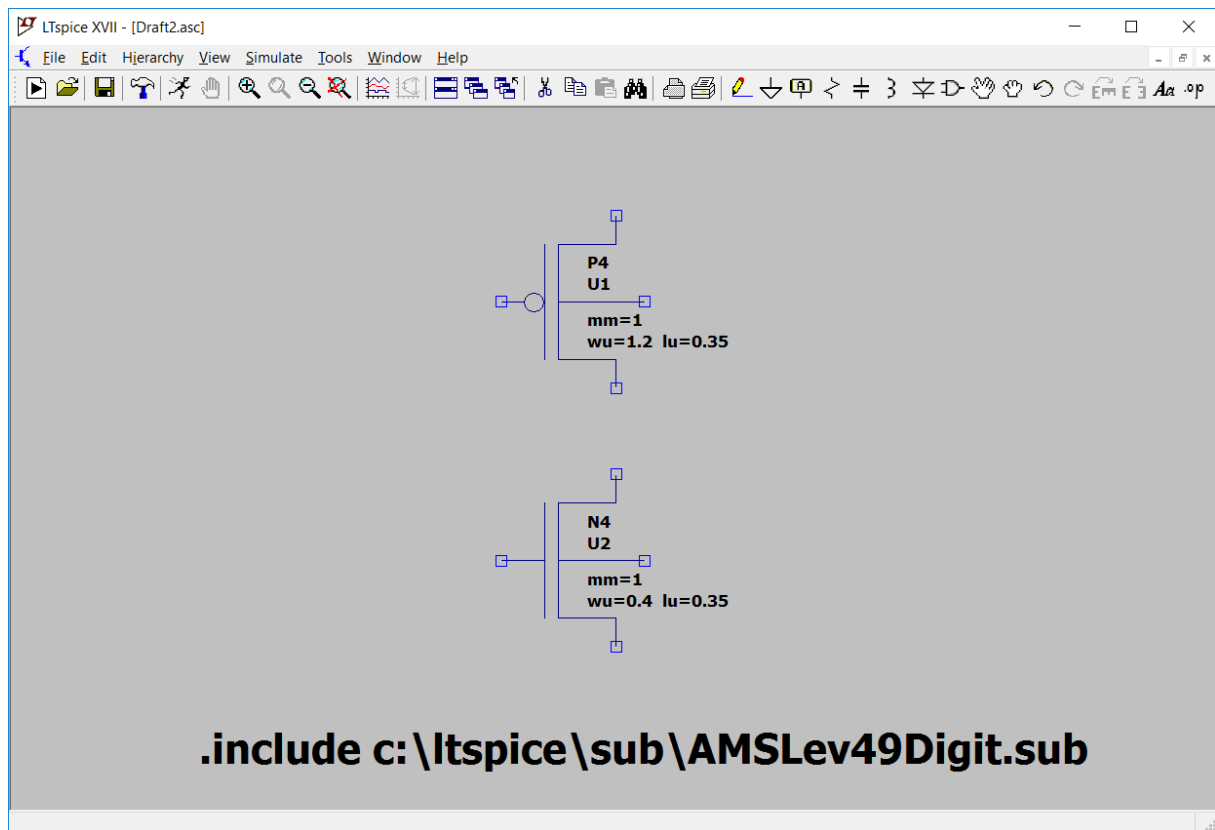


Figure 3. Add SPICE Directive

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.

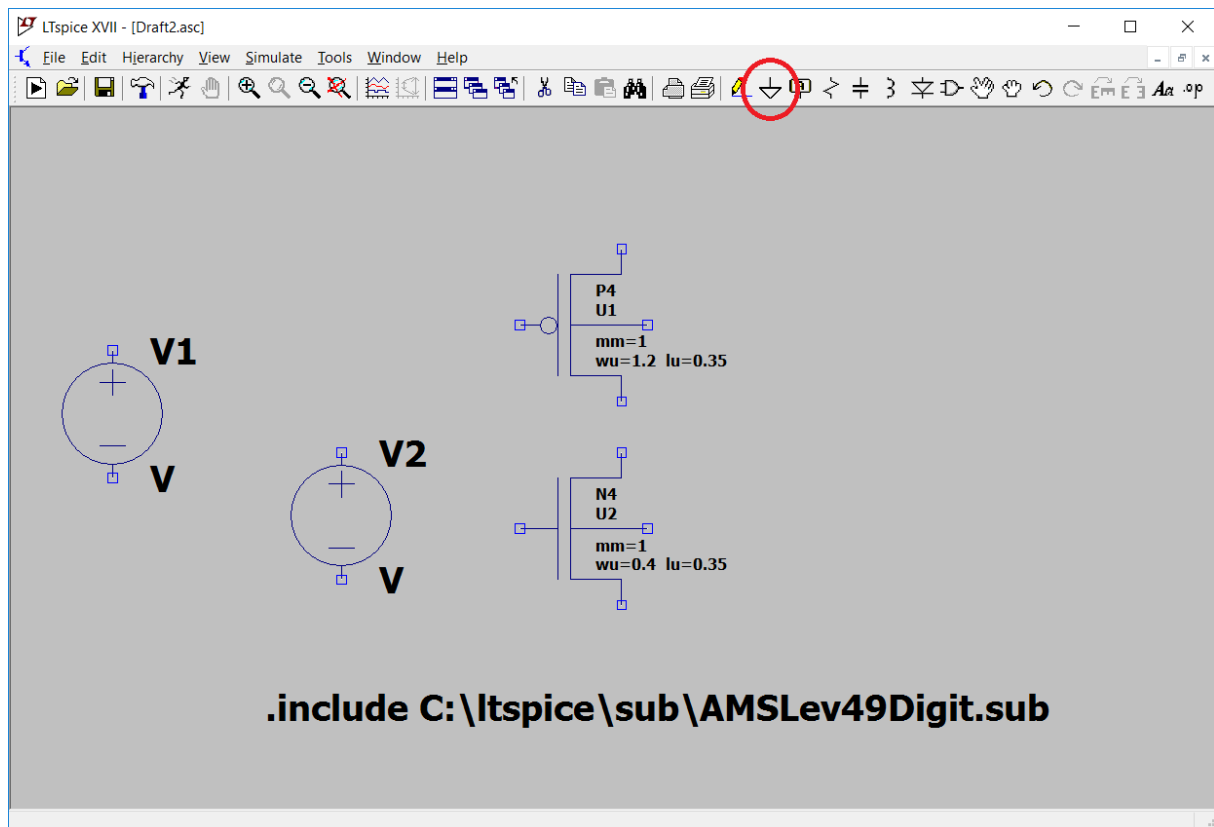

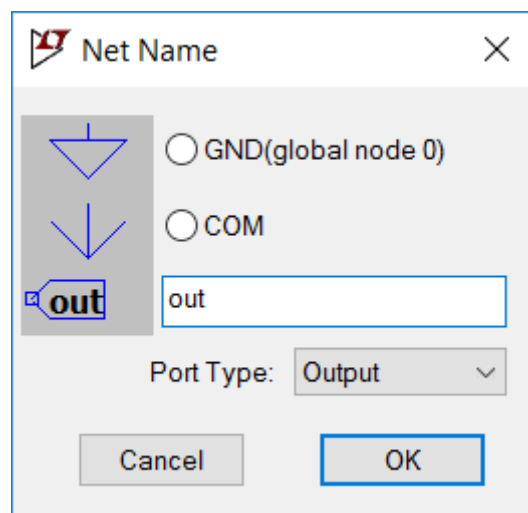


Figure 4. Insert a Ground

Now place three of Ground components (see Fig. 4.). Now we can add an output port to the circuit.

Select  button, and Set the Port Type to Output, and write *out* into the input field.



Now we can wire the circuit, so please use Edit – Draw wire 'F3' command to do it.

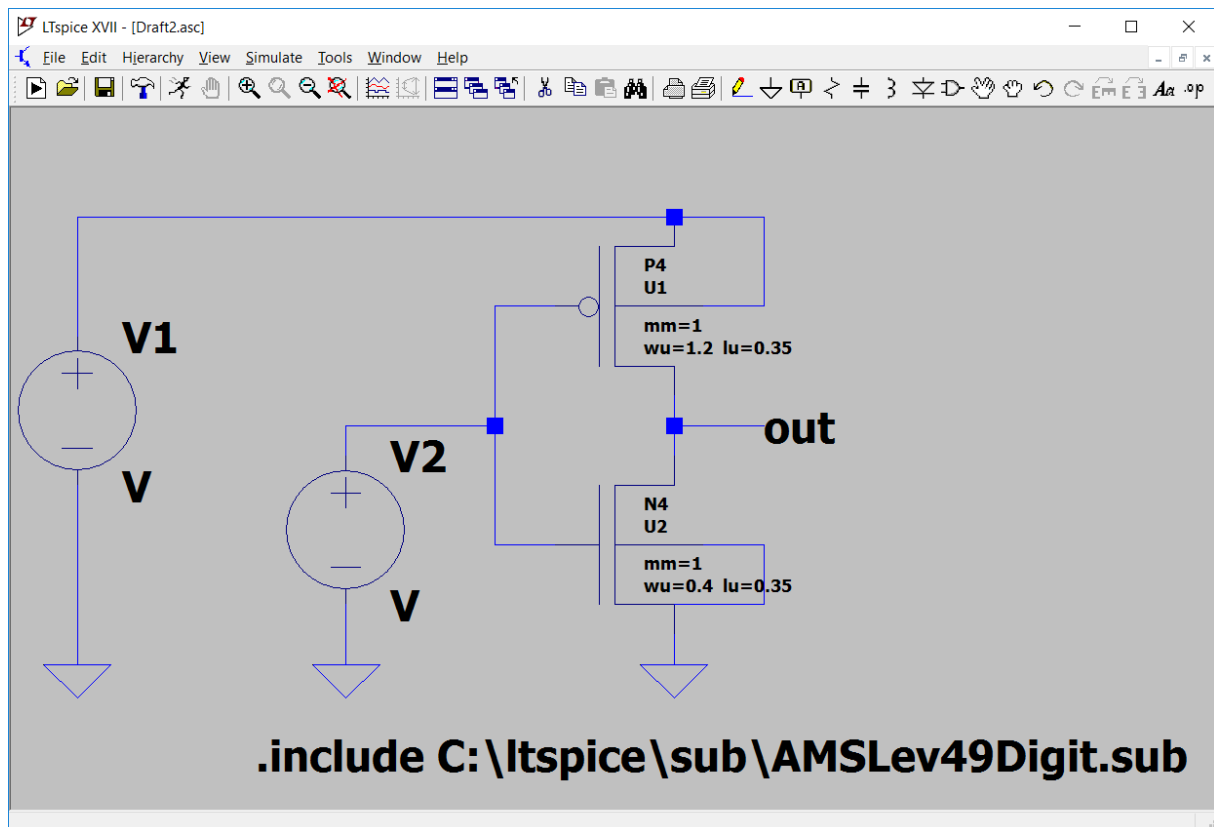


Figure 5. Wiring the schematic

Now we have to set the voltage of the voltage sources. We can do it by clicking the right mouse button on it. The power supply voltage has to be 5 V (see Fig.6.).

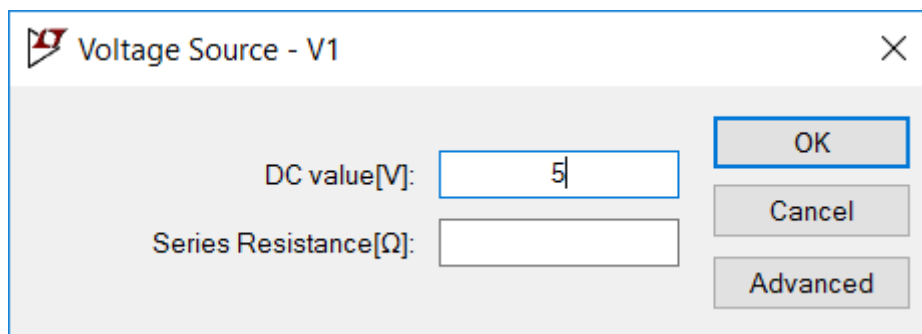
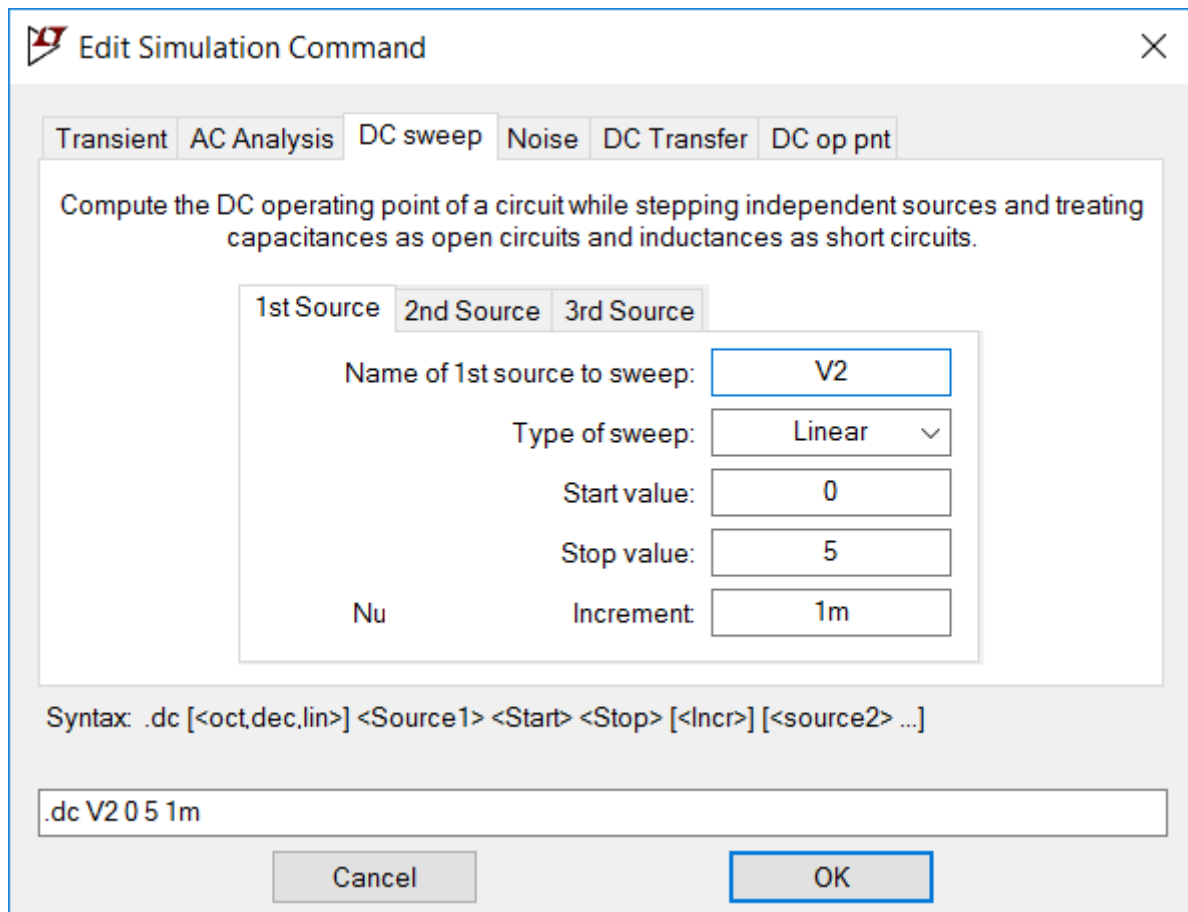


Figure 6. Power supply voltage

Set the other voltage source to 0.


DC Simulation

Now we are ready for the first simulation. Select Simulate – Edit Simulation Cmd, and choose DC sweep tab. Fill the input field, as it can be seen in Fig. 7.



The image shows a software window titled "Edit Simulation Command" with a close button (X) in the top right corner. It contains several tabs: "Transient", "AC Analysis", "DC sweep", "Noise", "DC Transfer", and "DC op pnt". The "DC sweep" tab is selected. Below the tabs, a text box explains: "Compute the DC operating point of a circuit while stepping independent sources and treating capacitances as open circuits and inductances as short circuits." Underneath this, there are three sub-tabs: "1st Source", "2nd Source", and "3rd Source". The "1st Source" sub-tab is active. It contains the following fields: "Name of 1st source to sweep:" with a text box containing "V2"; "Type of sweep:" with a dropdown menu showing "Linear"; "Start value:" with a text box containing "0"; "Stop value:" with a text box containing "5"; "Number of points to sweep (Npts):" with a text box containing "100"; and "Increment:" with a text box containing "1m". Below these fields, a "Syntax:" label is followed by the command format: ".dc [<oct,dec,lin>] <Source1> <Start> <Stop> [<Incr>] [<source2> ...]". At the bottom, there is a large text box containing the command ".dc V2 0 5 1m". Finally, there are "Cancel" and "OK" buttons at the bottom right.

Figure 7. DC sweep simulation settings.

If you are done, click on OK, and place the simulation command on the schematic. Select  button to perform the simulation. If everything is done, you can see an empty diagram like in Fig.8.

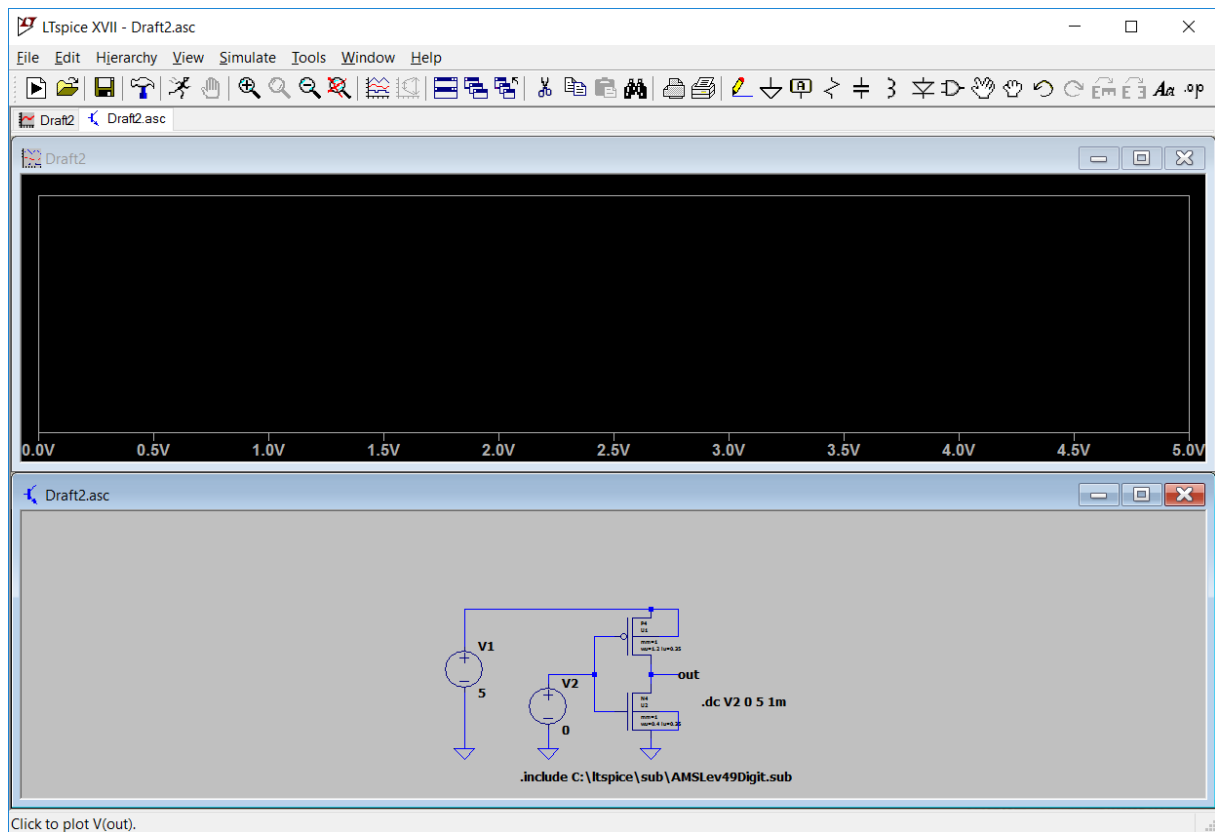


Figure 8. Simulation diagram without waveforms.

If you click on the input wire (the wire from the positive terminal of the V2 voltage source) and the output wire, you can get the DC transfer characteristics of the inverter. Please insert the screenshot of the waveforms. Read the threshold voltage (x value where the curves cross each other).

Transient simulation

First, we have to set the input source for the transient simulation. Right-click on the input voltage generator, then Advanced button, choose Pulse function, and set 200MHz, 50% duty cycle digital signal, which has a 10p second long rising and falling edge.

Independent Voltage Source - V2

Functions

☐ (none)

☒ PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)

☐ SINE(Voffset Vamp Freq Td Theta Phi Ncycles)

☐ EXP(V1 V2 Td1 Tau1 Td2 Tau2)

☐ SFFM(Voff Vamp Fcar MDI Fsig)

☐ PWL(t1 v1 t2 v2...)

☐ PWL FILE:

Vinitial[V]:

Von[V]:

Tdelay[s]:

Trise[s]:

Tfall[s]:

Ton[s]:

Tperiod[s]:

Ncycles:

Make this information visible on schematic: ☒

DC Value

DC value:

Make this information visible on schematic: ☒

Small signal AC analysis(AC)

AC Amplitude:

AC Phase:

Make this information visible on schematic: ☒

Parasitic Properties

Series Resistance[Ω]:

Parallel Capacitance[F]:

Make this information visible on schematic: ☒

Figure 9. Voltage source settings.

Go to Simulate, Edit Simulation Cmd. Turn of the DC simulation by inserting a % character at the beginning of the simulation command.

% .dc V2 0 5 1m

Choose *Transient* tab, set the Stop time to 10 nanoseconds, and the Maximum time step to 1p. (see Fig. 10)

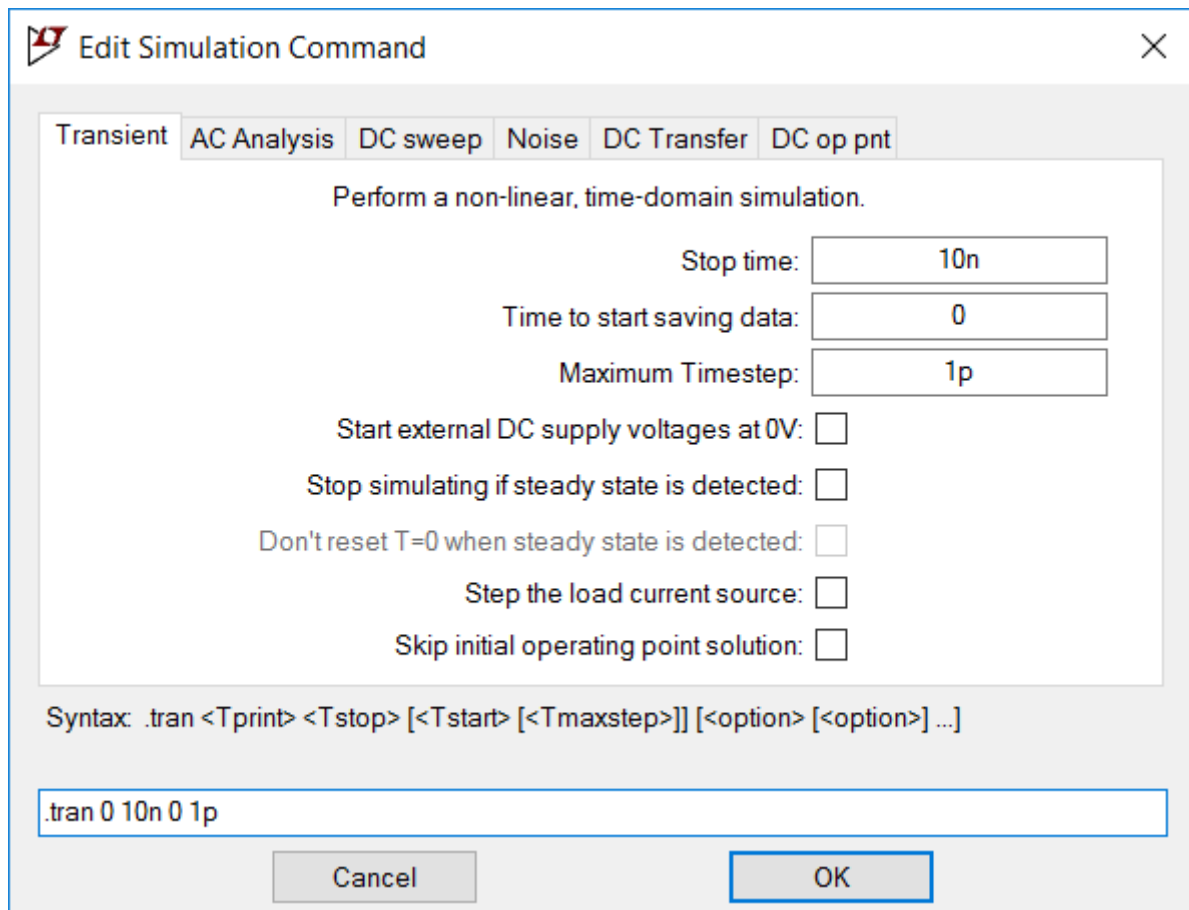


Figure 10. Transient simulation settings.

Now you can perform the simulation. If it is done, please click on input and output wires to see the waveforms.

We are going to measure the time delay for the rising edge and for the falling edge (the time difference between the 50% points). We need two cursors. To reach them, please right-click on **V(out)** and there choose 1st & 2nd (see Fig.11.). After clicking OK, you will have two cursors. You can move them when a '1' or '2' appears, and you can move them.

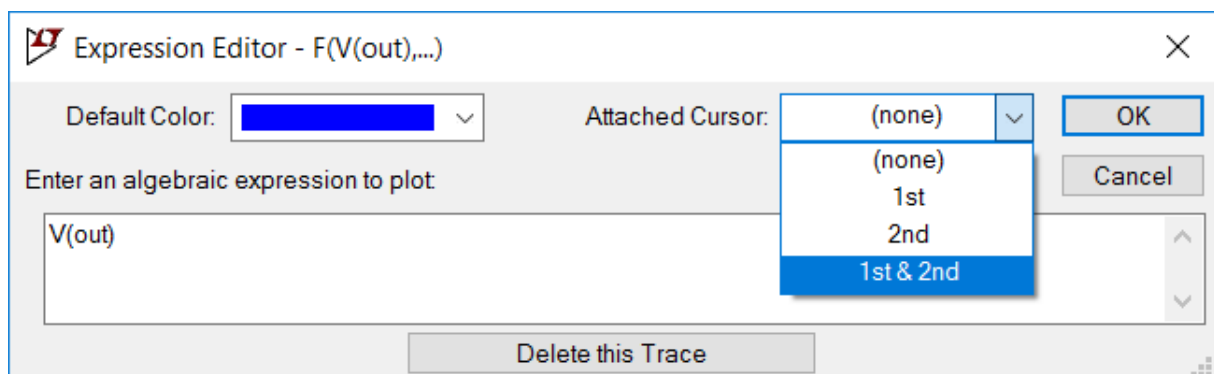


Figure 11. Adding cursors

Move one to the start, and the other to the 50% point of the output signal (2.5 V). In the draft window, you can read the time difference between the cursors (see Fig. 12.)

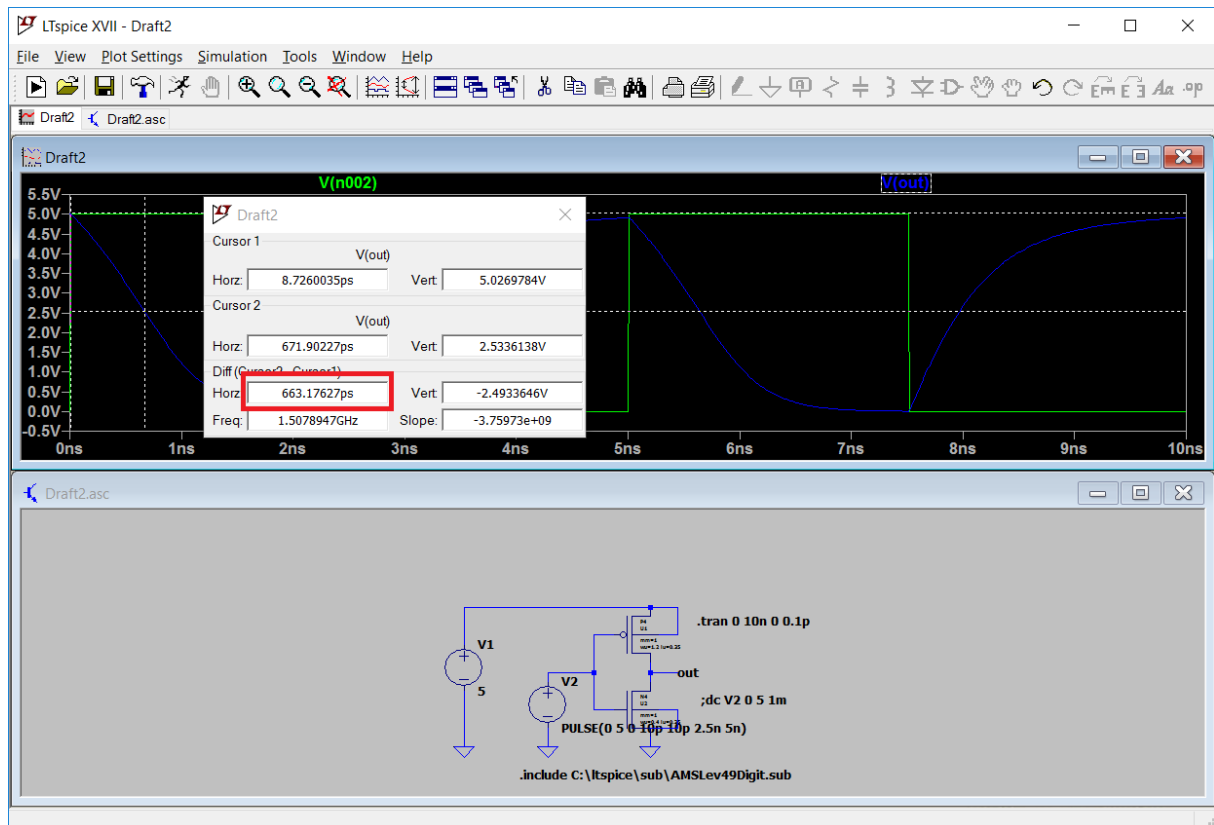


Figure 12. Reading the cursors

Please do the same for the rising edge of the output. Please include these results in the lab report.