Circuit Simulation using LTspice

In this lab, we will use LTspice XVII, a widely used industrial circuit design tool. Since it is freeware, it is free to download and use, and there is 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 Mac 10.9+*. A 116.9 MB installer will start to download. Once it is done, install it on your computer.

Installing LTspice on Mac:

After downloading the installation file to your computer, Launch the **.dmg** installer, a window will appear, drag & drop *LTspice* into *Applications*.
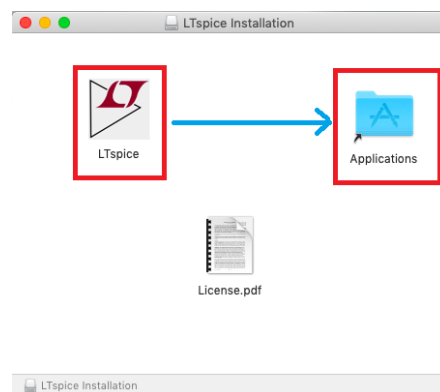

A window will appear showing the the copying progress of 'Ltspice.app' to 'Applications'.

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. 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. You can see that the symbol drawings are described in a unique format. The subfolder contains an AMSLev49Digit.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.
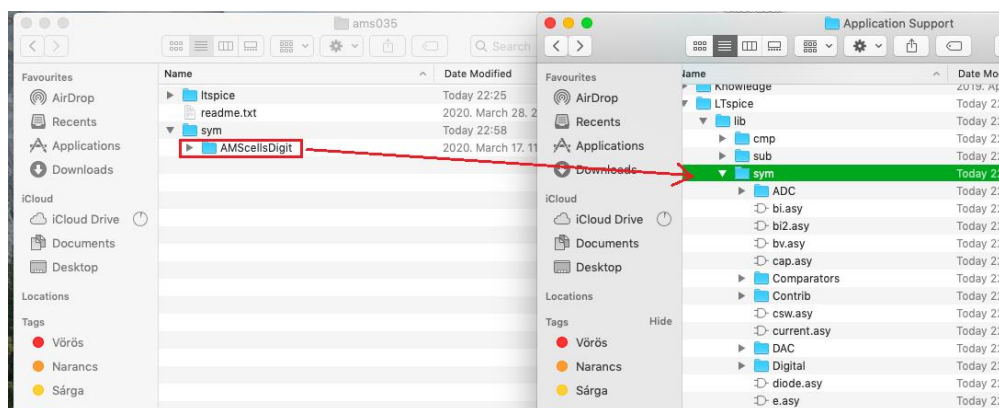
Once the installation is finshed, open a new *Finder* window (⌘+N) and go to your *home* folder. Press ⌘+Shift+. [dot] to show hidden files.

Navigate to *Library > Application Support > LTspice > lib*

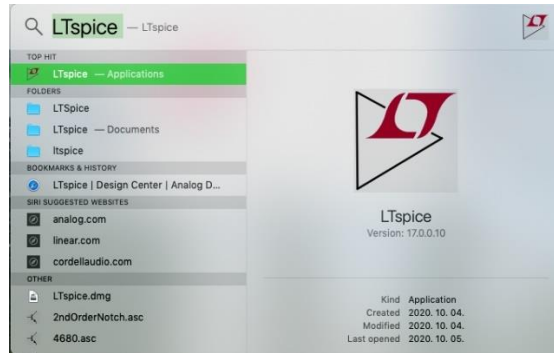Drag & drop the file from *ams035/ltspice/sub* to the *'sub'* folder of your Machintos HD.
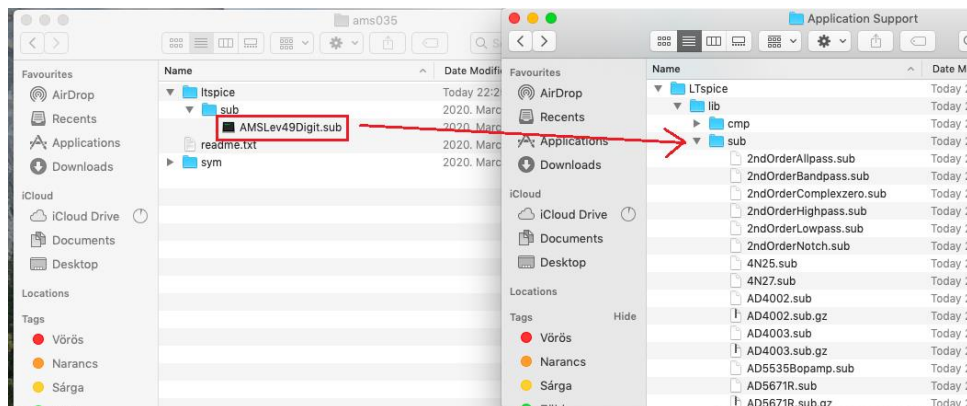
Repeat the step for the *'sym'* folder.

Press ⌘+Shift+. [dot] again to hide system folders.

Now use the *spotlight* search (⌘+Space) to launch Ltspice, type in „ltspice", hit **Return** and the software will launch.

A window called **'Welcome to LTspice'** appears everytime you launch the software. In here you can choose between opening an already existing schematics or to start a new blank one from the scratch, other options also exist. Choose *'Start a new, blank Schematic'*.
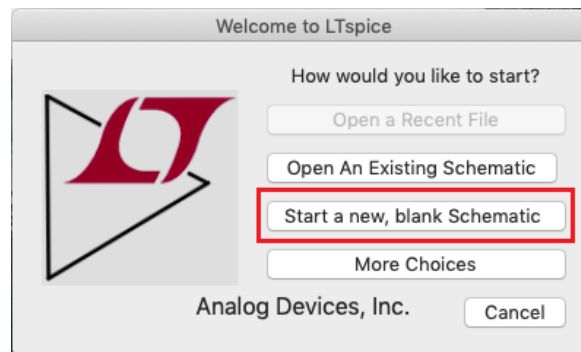

LTspice on Mac environment has different style than on Windows. For instance, on Mac, there is no menu bar on the top of the schematic workplace. By right click anywhere in the empty area of your schematic workplace, you can browse the options and tools that LTspice offers for the user (adding components, simulation,…,etc.)
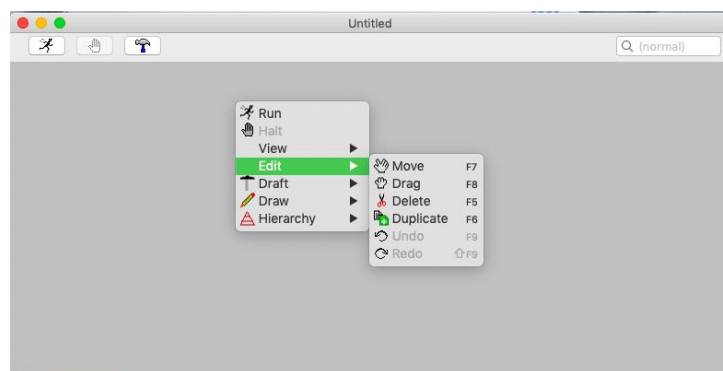

Drawing CMOS Inverter Schematic:

Right click on the empty workplace, and navigate to *Draft > Component* (for simplicity you can directly press **F2** on your keyboard).
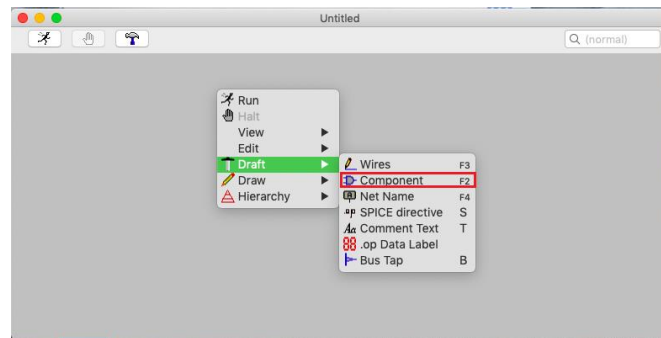
From the components library, we need to insert the NMOS Transistor, you can navigate to it from the folder *AMScellsDigit > n4*, or a faster way is to type **n4** in the search field. Choose **n4** and press *OK*. Now you can place your component anywhere in the workplace. Repeat the proceadure to add the **p4** component (the PMOS transisitor).
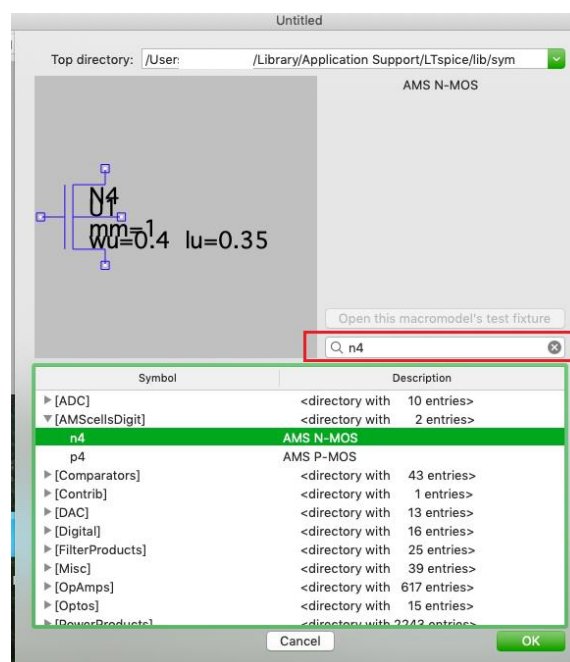

Now we need to add two voltage sources to our schematic. Navigate to *Draft > Component* and type in the serach field: **voltage** and click *OK*. Place two volage sources to the scematic. You can rearrange your components using the option *Drag* from the *Edit* menu.
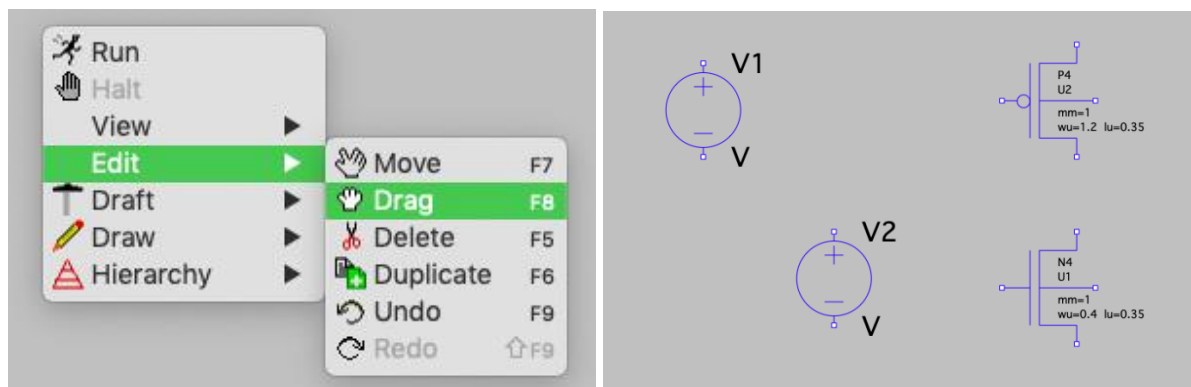
For the Ground (**GND**) terminal, choose the option *Net Name* from the *Draft* menu (or press **F4** on your keyboard). Choose the option GND(global node 0) and hit *OK*. (Place 3 GND terminals).
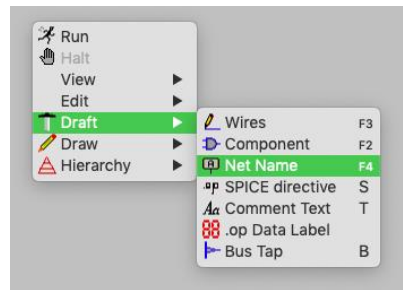
Next is to add an output terminal, again from *Draft > Net Name*, call the port **'out'** and choose the type to be *Output*.
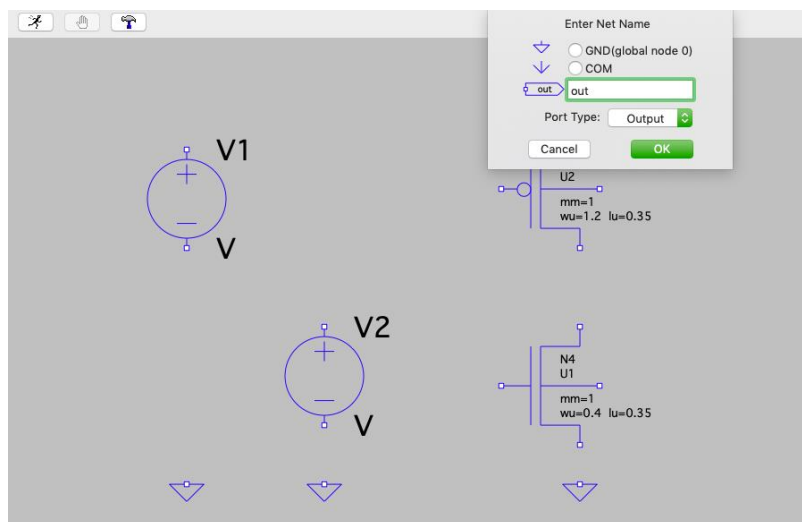

Right click on the voltage sources to set their DC value. Set **V1** to be **5V** and **V2** to **0**, use the option *Wires* in the *Draft* menu (or press **F3**) to connect the components as following:
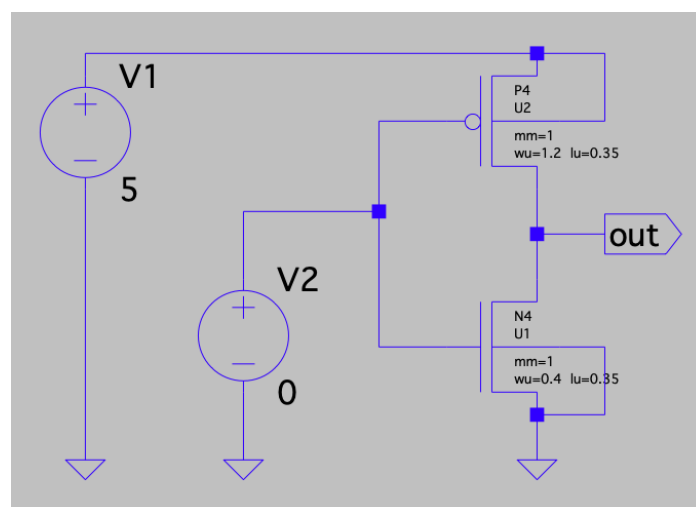
Now we have to click on ⬛ SPICE directive S from the *Draft* menu (or press **S**). Here we can define the path of the model file. Insert this line below:
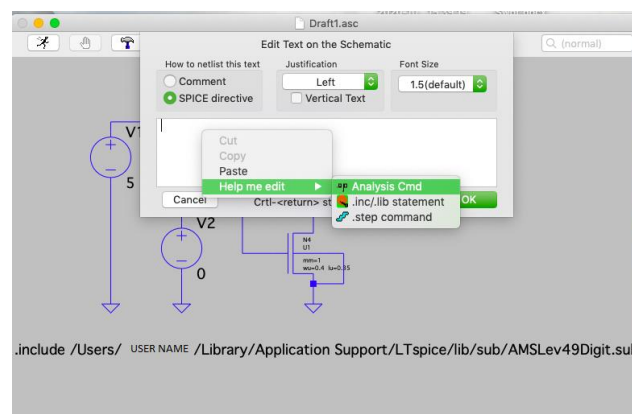
.include /Users/[YOUR USERNAME]/Library/Application Support/LTspice/lib/sub/AMSLev49Digit.sub

And place it somewhere on the schematic.
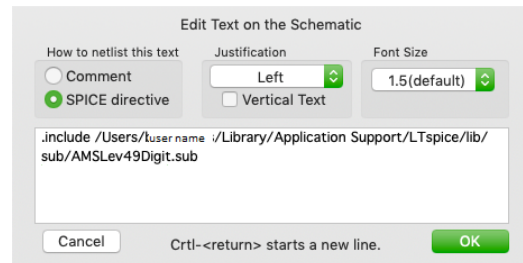
DC Simulation

click on ⬛ SPICE directive S from the *Draft* menu (or press **S**), right click on the text field and choose *Analysis Cmd*. Now the simulation settings menu will appear.

Choose the simulation type **DC Sweep**, fill the simulation settings as in the figure below. Click on **OK**.