# Creating a Symbol and Introduction to Parametric Sweep

Note that, you may use your own knowledge of LTSpice. This is just a general guideline.

Step 1: Design a CMOS inverter as you did in the last lab. But do not add any sources. Instead add pins. You can do that by clicking on 'net' , select port as 'input' or 'output'. Create VDD and gnd pins as input. It's also okay to not have a gnd pin for LTspice.
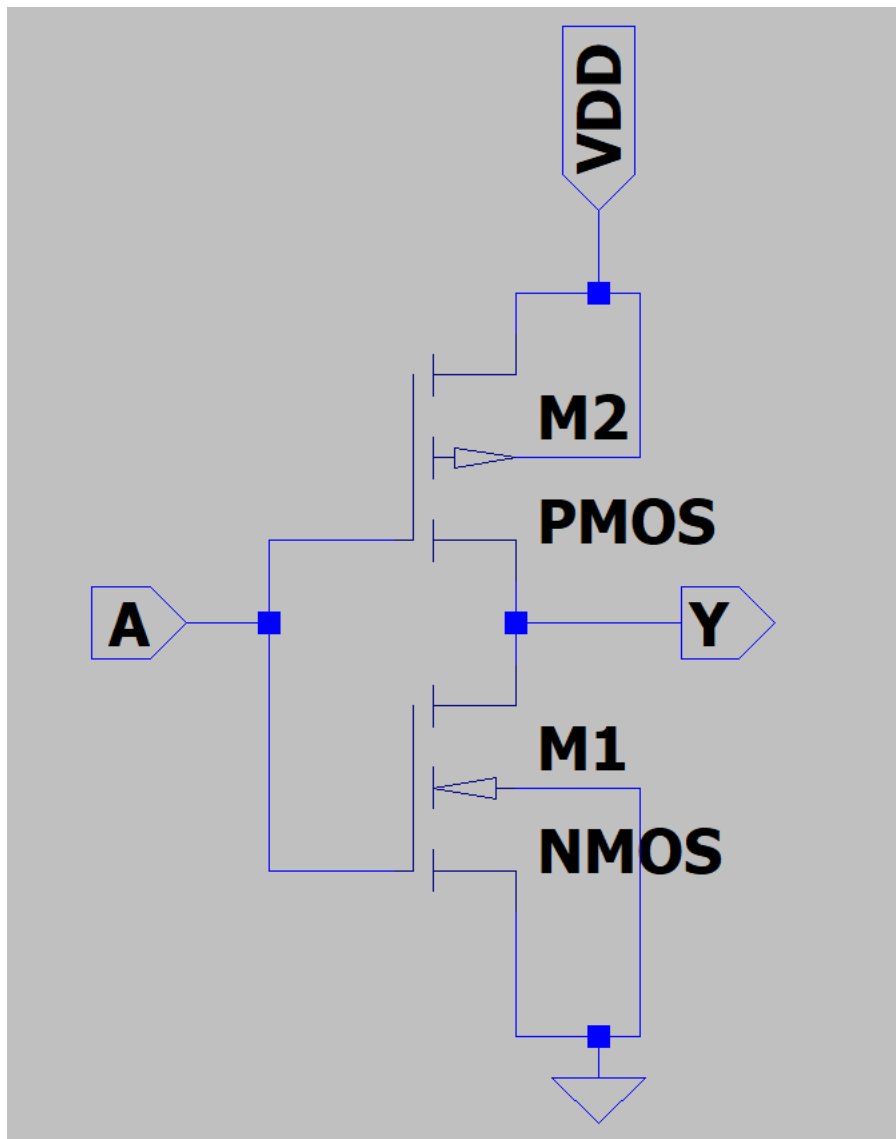
Step 2: Create a symbol. Click File—New Symbol. Draw your inverter symbol using Draw—Line and other options.

Step 3: Add pins just like your schematic using Edit—Add pin/port. Make sure you have the exact same pins as the schematic. Save your symbol with .asy extension and the same name as the schematic.

Step 4: Use your symbol in a schematic. Connect it to the power supply. Provide an input just like the first lab. Run transient analysis.

Step 5: Now connect a capacitor. In the capacitance box, put {C1}

After connecting everything, your circuit will look something like the following:
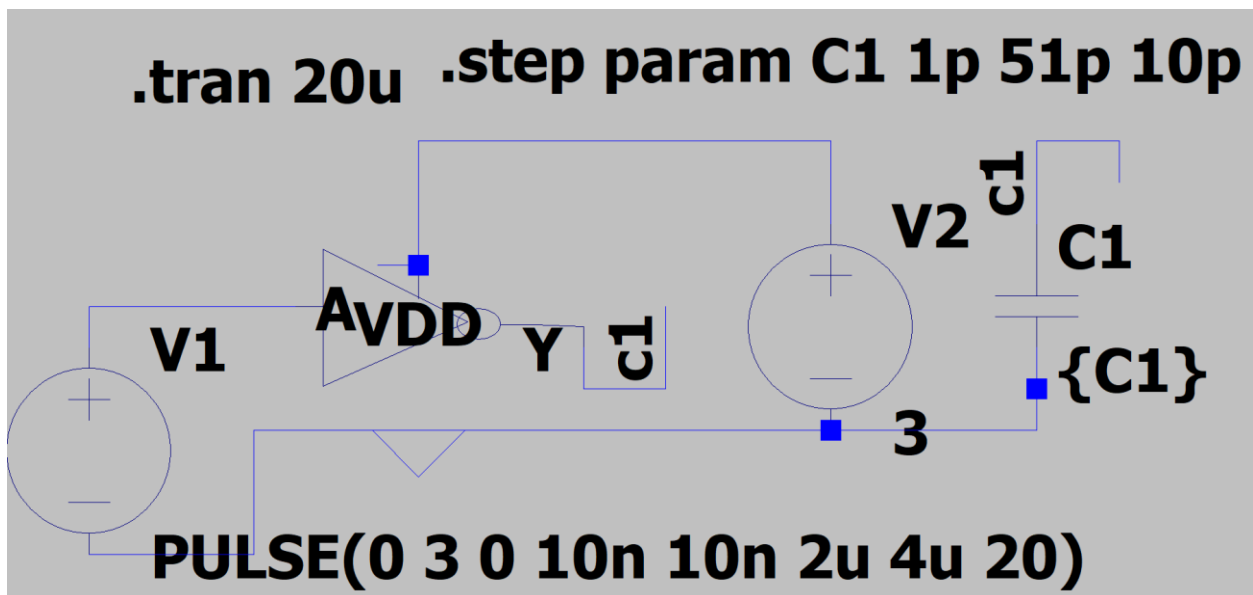
Step 6: Put the text: .step param C1 1p 51p 10p, then run transient analysis again.