In this tutorial, we will examine the use of the DC sweep analysis, which is handy for looking at transfer characteristics for inverters and similar circuits. To illustrate the method, we will simulate a simple inverter using a BJT loaded with a resistor.

This tutorial is written with the assumption that you know how to do all of the basic things in PSPICE: starting a project, adding parts to a circuit, wiring a circuit together, using probes, and setting up an using a simulation profile.

**Simulation with default values**

Build the circuit shown below. Use the “QbreakN” model for the BJT. Edit the BJT description so that $\beta_F = 125$. (Or pick your own favorite value.) Recall that to change the current gain for a transistor, we must click on the transistor to select it and then right-click and choose “Edit PSPICE model…” from the pop-up menu. Add $BF=125$ to the text that shows up in the window and save the changes. (See the earlier “BJTs in a simple DC circuit” tutorial.)

It is not necessary to set a value for the VS voltage source. Its value will be changed incrementally as part of the DC sweep.

Finally, attach a voltage probe at the collector of the transistor.

Create a new simulation profile and choose the “DC sweep” simulation option from the drop down menu. On the right-side of the dialog box are spaces to enter the name of the source that will be changed, the starting and ending voltages, and the size of the steps in the sweep. The dialog is shown below. For this case, we will sweep the input voltage from 0 to 5 V in 0.05 V increments for a total of 101 simulation points.
Run the simulation. If everything was set up correctly, the plot window should open, displaying the voltage transfer characteristic ($v_{CE}$ vs $v_S$) for the circuit.

As discussed in class, the BJT is off when $V_S < 0.7$ V (approximately) and the output it pulled high. As the transistor is turned on, it operates in its forward active region and the output continually drops as $V_S$ increases. Eventually the output drops to the saturation level of the transistor.
Below is a second simulation using the same circuit with the resistor value has been increased to 10 kΩ. It is qualitatively very similar, except the transition region is steeper.