

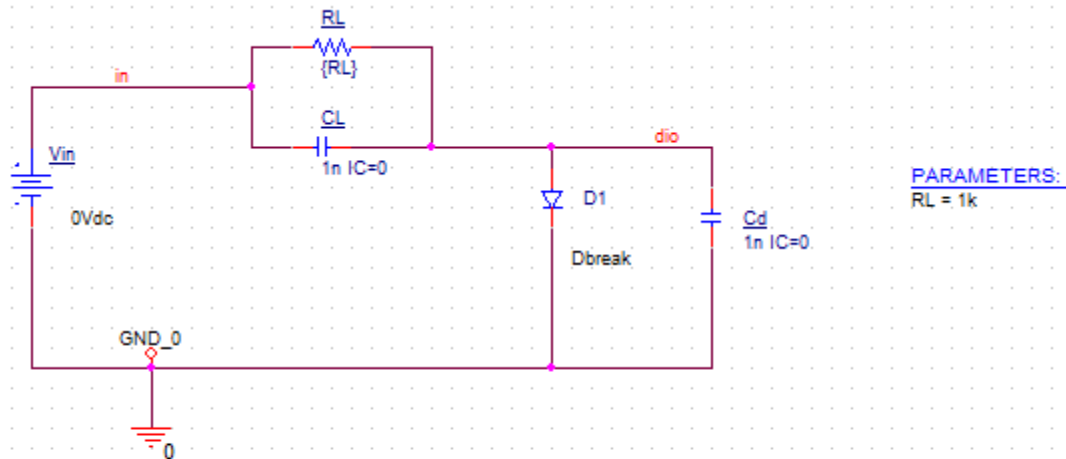
HW 1

1.

(60 points, Diode bias & Spice transient analysis)

Set up the following circuit in Spice. Then do a DC run on V_{in} (about 0 to 3V)

along with a Parametric run to find R_L to give a diode current of 1mA.

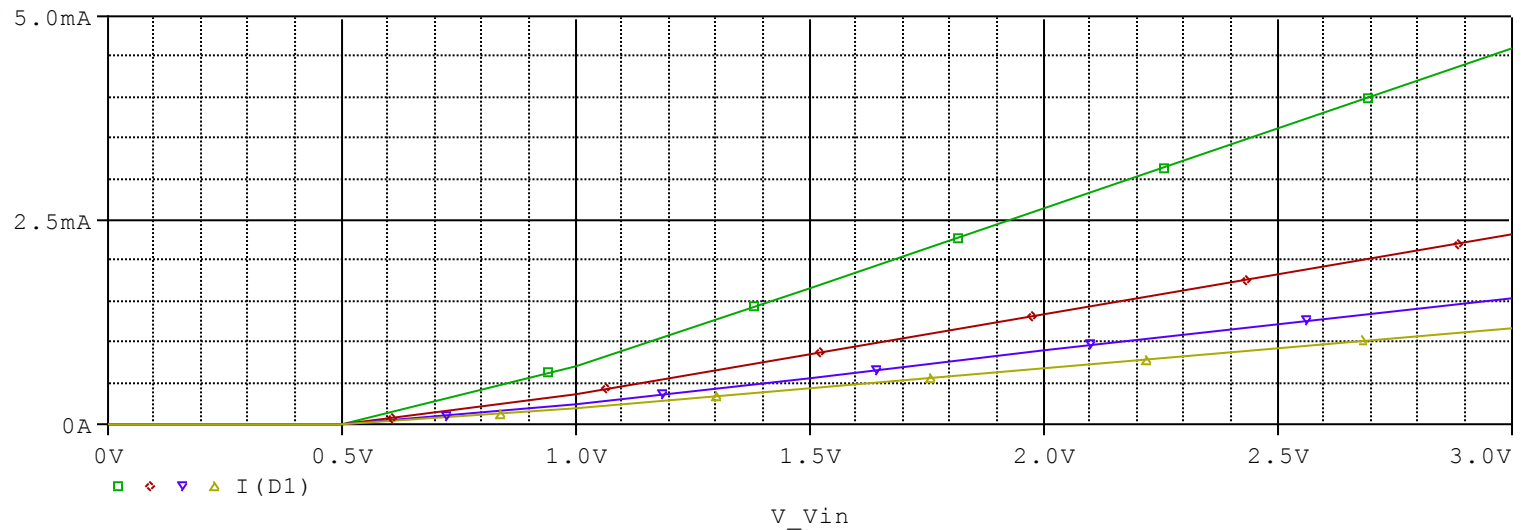


Copy and paste the GND_0 circle in order to connect the Analog GND to the appropriate wires in your circuit.

$IC = 0$ means that each capacitor has an initial condition of $V_{\text{capacitor}} = 0$

DC Sweep of V_{in} (0 to 3V, 0.5V step)
and Parametric Sweep of R_L (500 to 2k ohms, 500 ohms step)

The plot below traces the diode current: $I(D1)$



The legend at the bottom left lists the curve colors in the order they were executed (e.g. green is $R_L=500$ ohms, red is $R_L=1000$ ohms, etc.)

a)

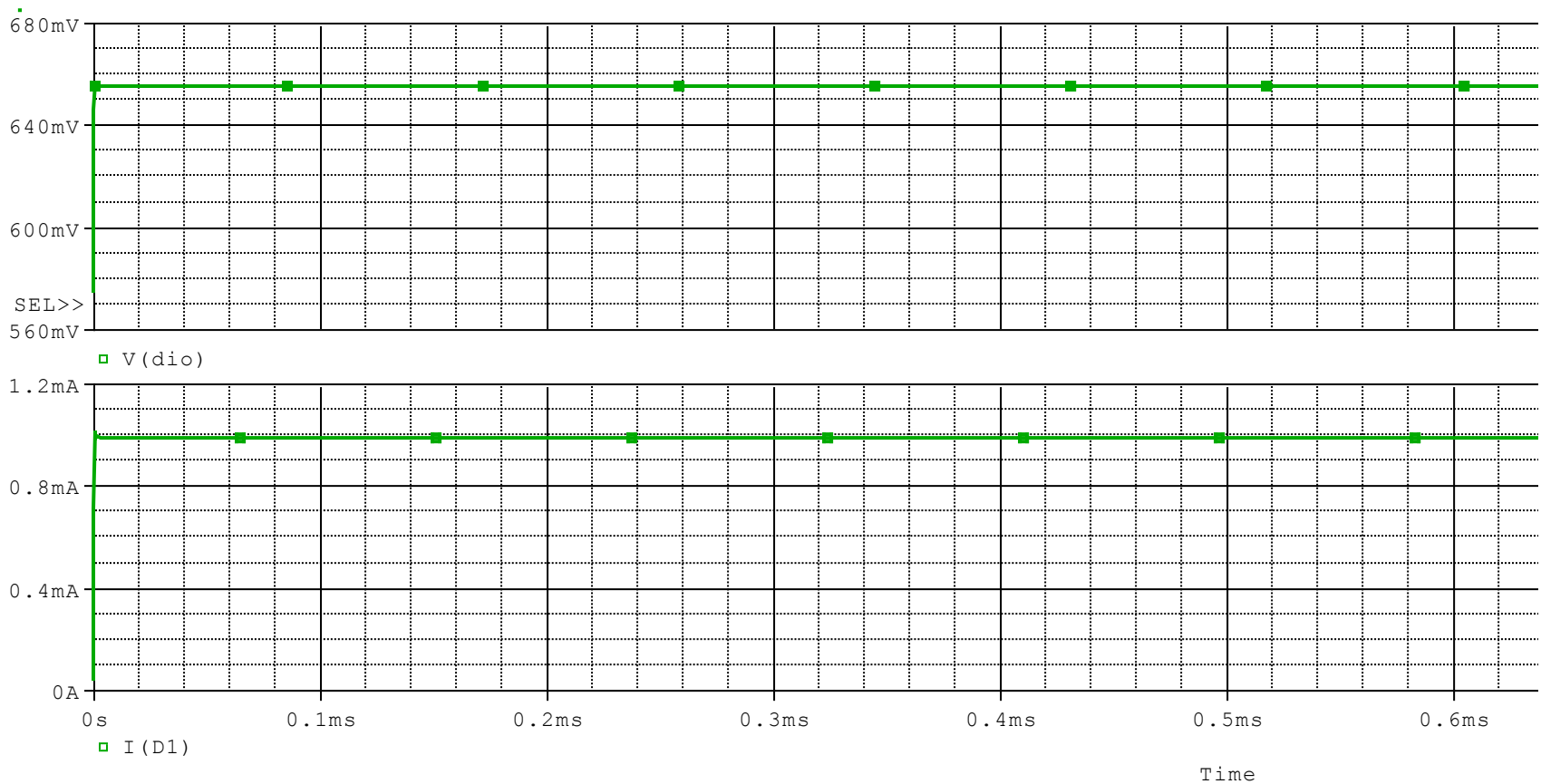
Record 1) the resistance R_d of the resistor R_L for the diode current to be 1mA, 2) the diode voltage, V_{dio} , at this diode current Q point, 3) the value of V_{in} for this Q point and 4) the conductance (slope), g_d , of the diode current vs diode voltage at this Q point.

A few different plot points would have worked to get 1 mA through the diode. I choose this one:

$$R_d = 500 \text{ ohms}$$

$$V_{in} = 1.15V$$

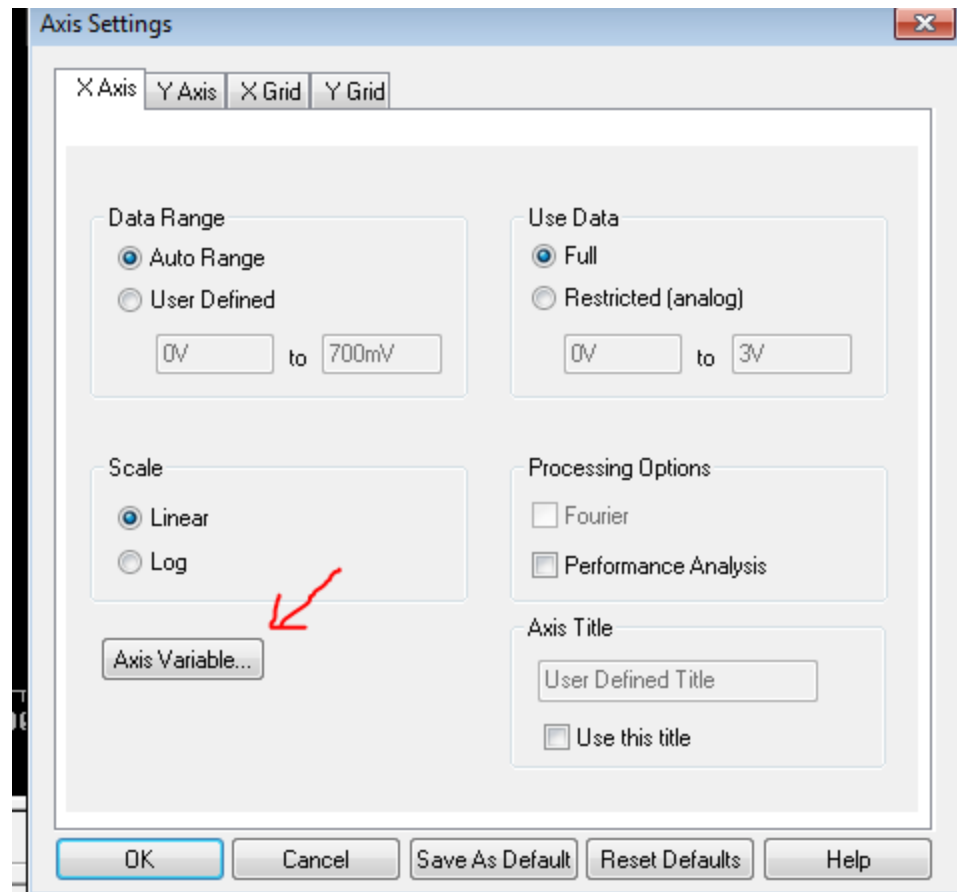
Below is the time response of the circuit with V_{in} and R_d now selected. The bottom plot verifies that the current through the diode is indeed 1 mA. The top plot shows that V_{dio} for this current is about 655 mV.



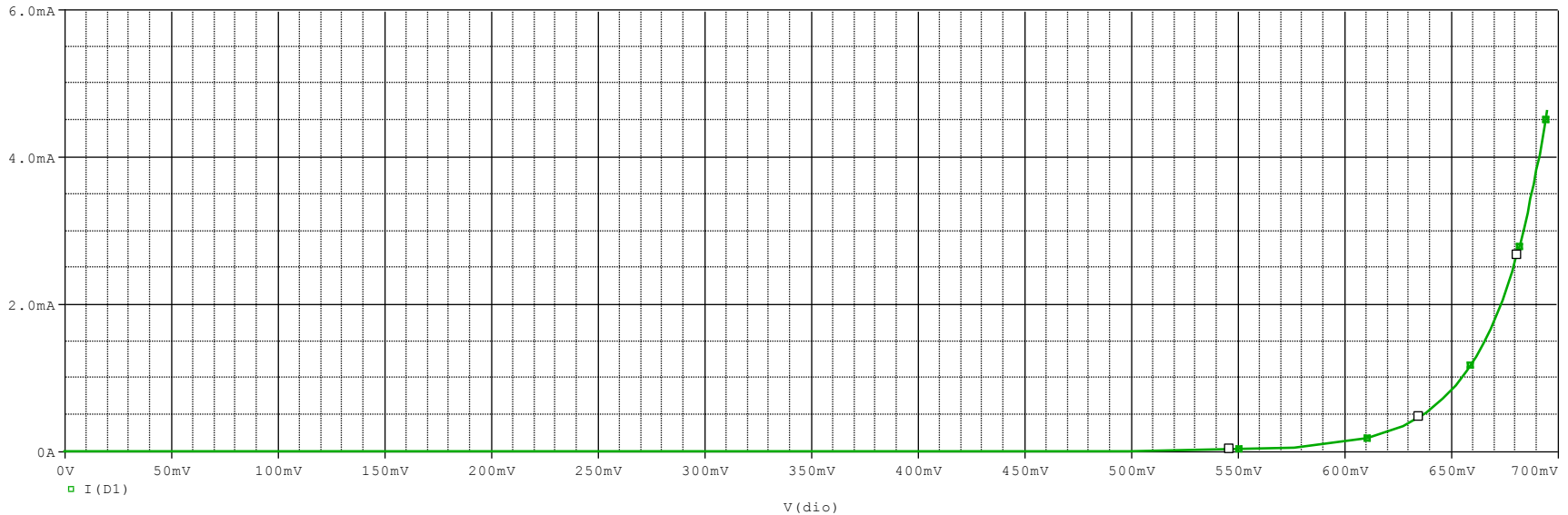
To get the conductance (g_d) of the diode around that Q-point, we need to plot the i - v characteristic of the diode.

Run a DC sweep on V_{in} . I chose to sweep from 0V to 3V, with 0.1V increments.

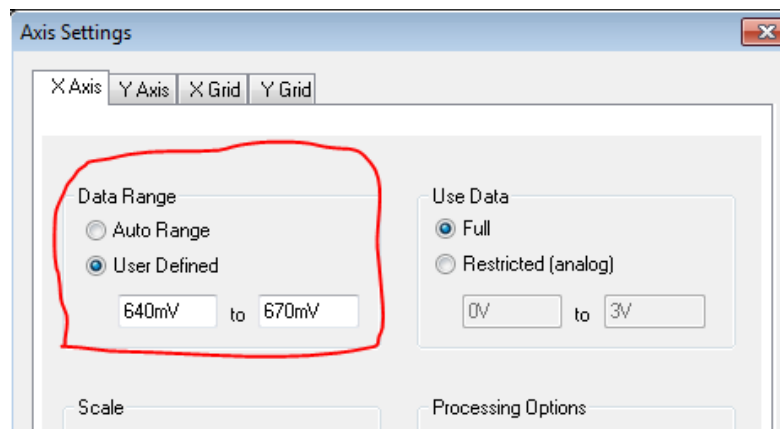
After the run is finished, in the plot window, click on Plot -> Axis Settings. Change the x-axis variable to $V(\text{dio})$ (see below).



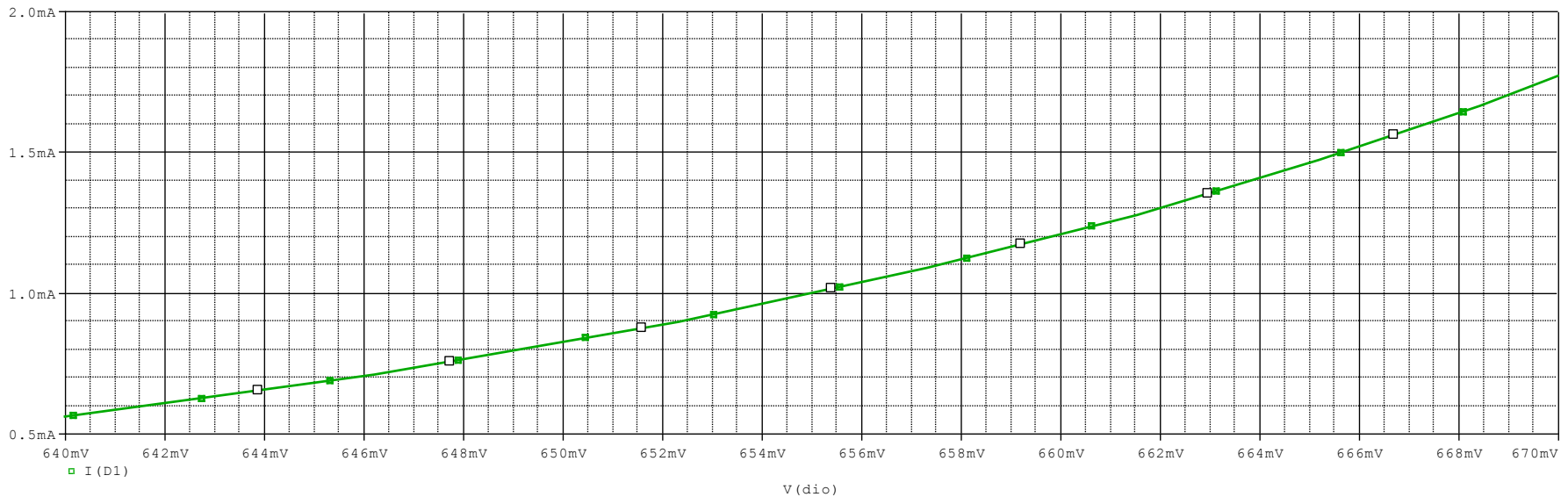
Then add a trace for the diode current I(D1). The resulting plot should look like this:



We are interested in the slope around 655 mV, so we can zoom in on that region. Go back to axis settings and set a user defined range for the x-axis:



Now the plot should look like this:



You may have used a different method to get the slope (conductance). I just used a simple linear approximation with two points.

Choose two points close to 655mV to calculate the slope. I chose:

$$v_1 = 654 \text{ mV}, \quad i_1 = 0.961 \text{ mA}$$

$$v_2 = 656 \text{ mV}, \quad i_2 = 1.036 \text{ mA}$$

This gives a conductance $g_d = 0.0375$ Siemens (A/V)

b)

Change the model for the diode via Edit PSpice Model using the model for the 1N4007 from the course web page and repeat part a).

Comment on any differences.

Default Dbreak model:

```
.model Dbreak D Is=1e-14 Cjo=.1pF Rs=.1
```

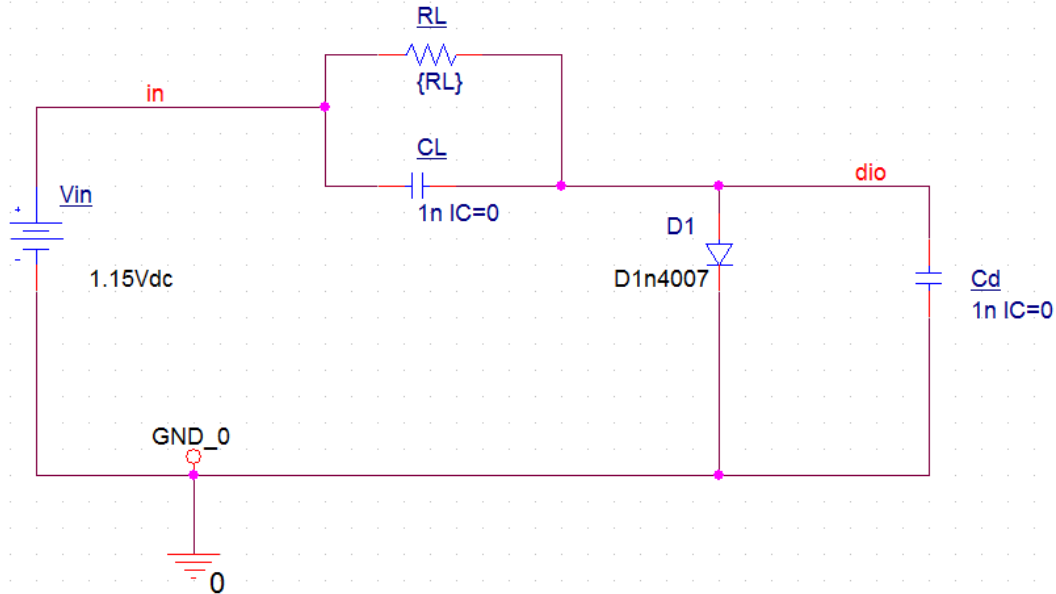
1N4007 model:

```
.MODEL D1n4007 d  
+IS=7.02767e-09 RS=0.0341512 N=1.80803 EG=1.05743  
+XTI=5 BV=1000 IBV=5e-08 CJO=1e-11  
+VJ=0.7 M=0.5 FC=0.5 TT=1e-07  
+KF=0 AF=1
```

The general form for a PSPICE diode model is given below. The first 4 terms (which are missing in both examples above) have information that is alternatively specified by the schematic you drew. A new line must begin with a “+” character. As you can see, the 1N4007 model has many more parameters specified.

```
d[name] [anode] [cathode] [modelname] .model ([modelname] d  
[parmtr1=x] [parmtr2=y] . . .)
```

The new circuit should look like this, with a D1n4007 label.

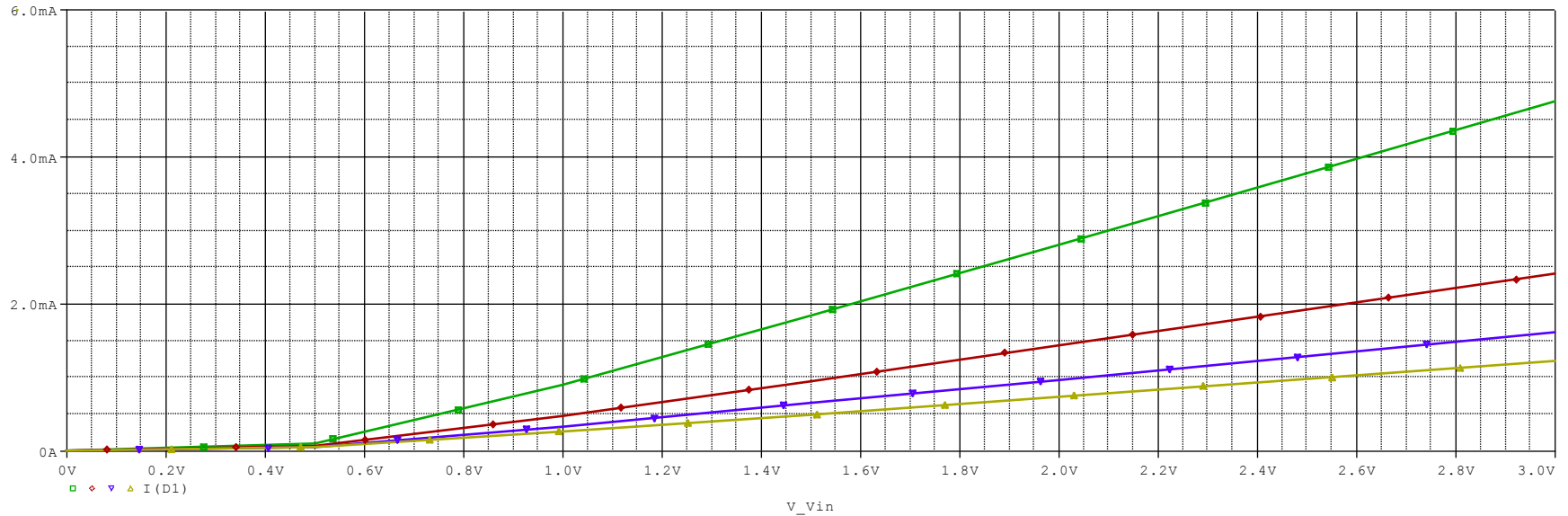


PARAMETERS:
RL = 500

Copy and paste the GND_0 circle in order to connect the Analog GND to the appropriate wires in your circuit.

DC Sweep of V_{in} (0 to 3V, 0.5V step)
and Parametric Sweep of R_L (500 to 2k ohms, 500 ohms step)

The plot below traces the diode current: $I(D1)$

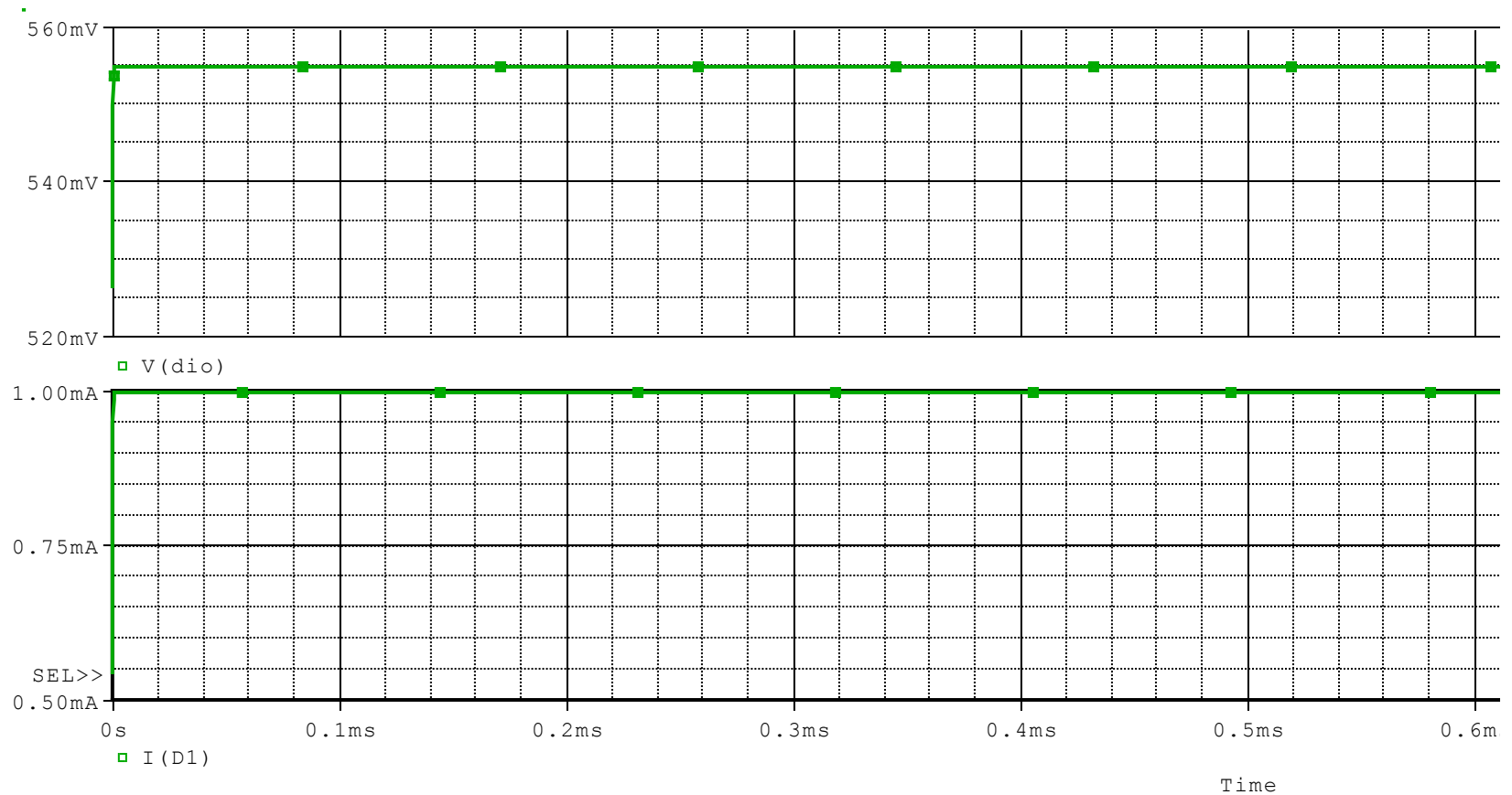


This time I chose to use:

$R_d = 500$ ohms

$V_{in} = 1.053V$

Below is the time response of the circuit with new V_{in} and R_d . The bottom plot verifies that the current through the diode is indeed 1 mA. The top plot shows that V_{dio} for this current is about 555 mV.



I used the same method as on Slide 6 to find the conductance (g_d) of this new diode.

$$v_1 = 554 \text{ mV}, \quad i_1 = 0.986 \text{ mA}$$

$$v_2 = 556 \text{ mV}, \quad i_2 = 1.027 \text{ mA}$$

$$g_d = 0.0205 \text{ Siemens}$$

The D1n4007 diode has a conductance that is about half that of dbreak model for their respective q-points at 1mA. Also note that the D1n4007 has a much shallower rise in current with respect to voltage. Where the dbreak diode rises to 4.5mA from 0 mA in about 100 mV, the D1n4007 experiences the same rise over a range of almost 200 mV.

c)

Make $R_L = R_d$, the value found for the above Q point.

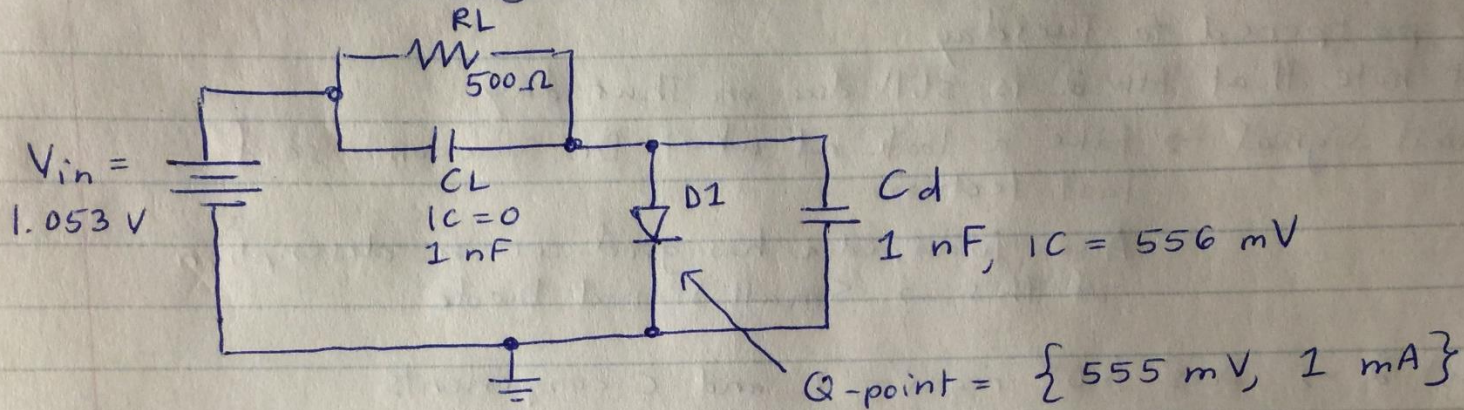
Set up the small signal differential equation when the initial value of the capacitor C_d is 1mV greater than the Q point voltage.

Do a transient analysis for about 50uSec in PSpice with that capacitor IC value. Submit your transient analysis curves of anode voltage and diode current.

The derivation of the differential equation is on the next two slides. Note that your values for R_L and V_{in} may differ. I solved for the voltage across the diode, you may have solved for a different variable.

Small-Signal Differential Equation

9/09/15



We know these circuit characteristics:

Resistor - $V = IR$

Capacitor - $V = \frac{1}{C} \int I dt + V_0$ -- or -- $I = C \frac{dV}{dt}$

Diode (Small-Signal Approximation) - $g_D \sim I_Q / V_T = 1/r_D$

We know relationships among components from Kirchhoff's Voltage and current laws.

$$V_{in} = V_{CL} + V_{Cd}$$

$$V_{CL} = V_{RL} = i_{RL} R_L$$

$$V_{Cd} = V_{D1} = i_{D1} r_D$$

NOTE:

$$C_L = C_d = C$$

$$V_{in} = V_{CL} + V_{Cd}$$

$$V_{CL} = V_{RL} = i_{RL} R_L$$

$$V_{Cd} = V_{D1} = i_{D1} r_D$$

NOTE:

$$C_L = C_d = C$$

$$i_{RL} + i_{CL} = i_{D1} + i_{Cd} \rightarrow i_{RL} = V_{RL}/R_L, i_{D1} = V_{D1}/r_D$$

$$\frac{V_{RL}}{R_L} + C \frac{dV_{CL}}{dt} = \frac{V_{D1}}{r_D} + C \frac{dV_{Cd}}{dt}$$

$$\frac{V_{CL}}{R_L} + C \frac{dV_{CL}}{dt} = \frac{V_{D1}}{r_D} + C \frac{dV_{D1}}{dt}$$

$$V_{CL} = V_{in} - V_{Cd} = V_{in} - V_{D1} \rightarrow \frac{dV_{CL}}{dt} = 0 - \frac{dV_{D1}}{dt} = - \frac{dV_{D1}}{dt}$$

$$\frac{V_{in} - V_{D1}}{R_L} + C \left(\frac{-dV_{D1}}{dt} \right) = \frac{V_{D1}}{r_D} + C \frac{dV_{D1}}{dt}$$

$$\frac{V_{in} - V_{D1}}{R_L} = \frac{V_{D1}}{r_D} + 2C \frac{dV_{D1}}{dt}$$

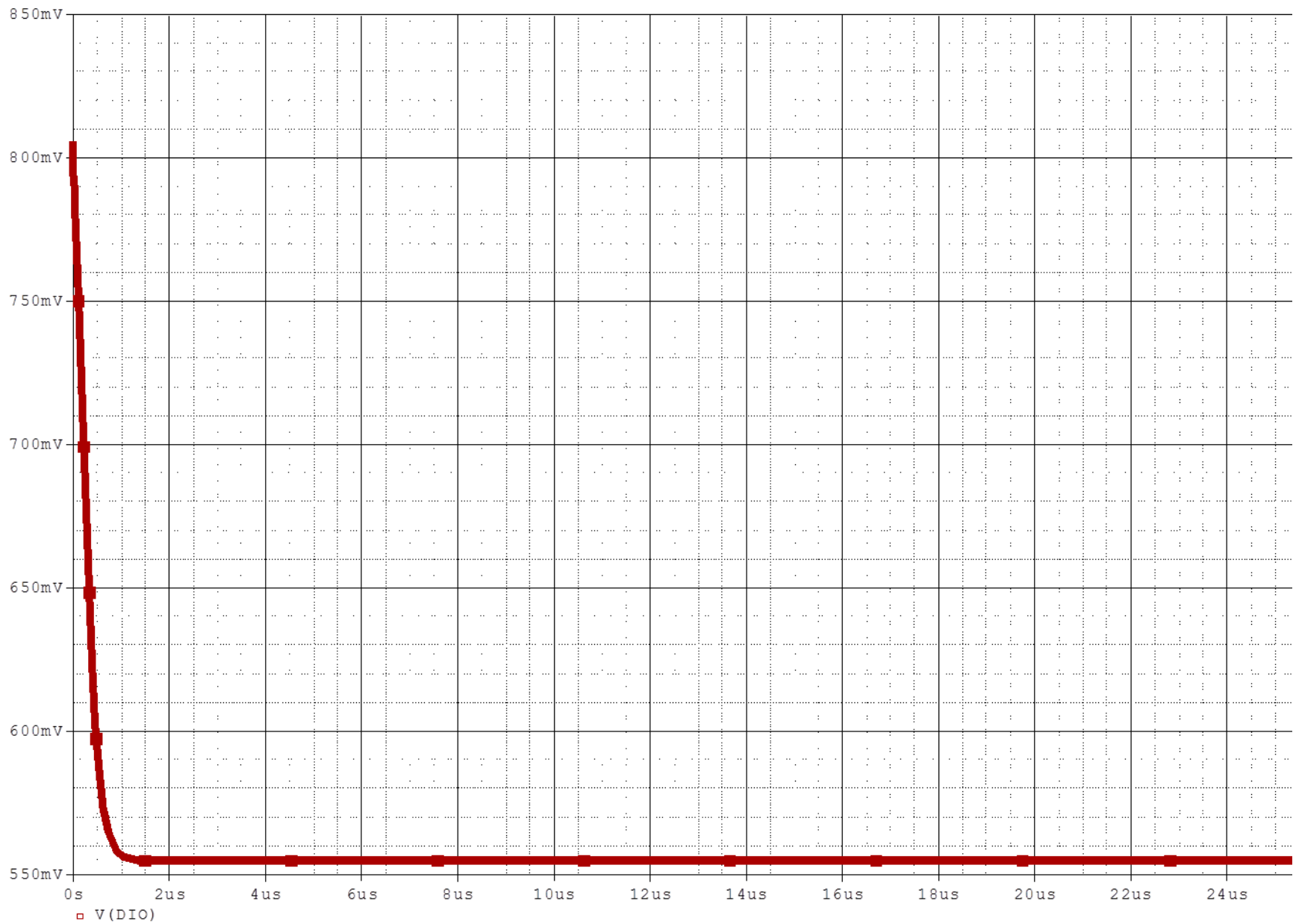
$$V_{Cd}(0) = V_{D1}(0) = 556 \text{ mV}$$

$$V_{in} = 1.053 \text{ V}$$

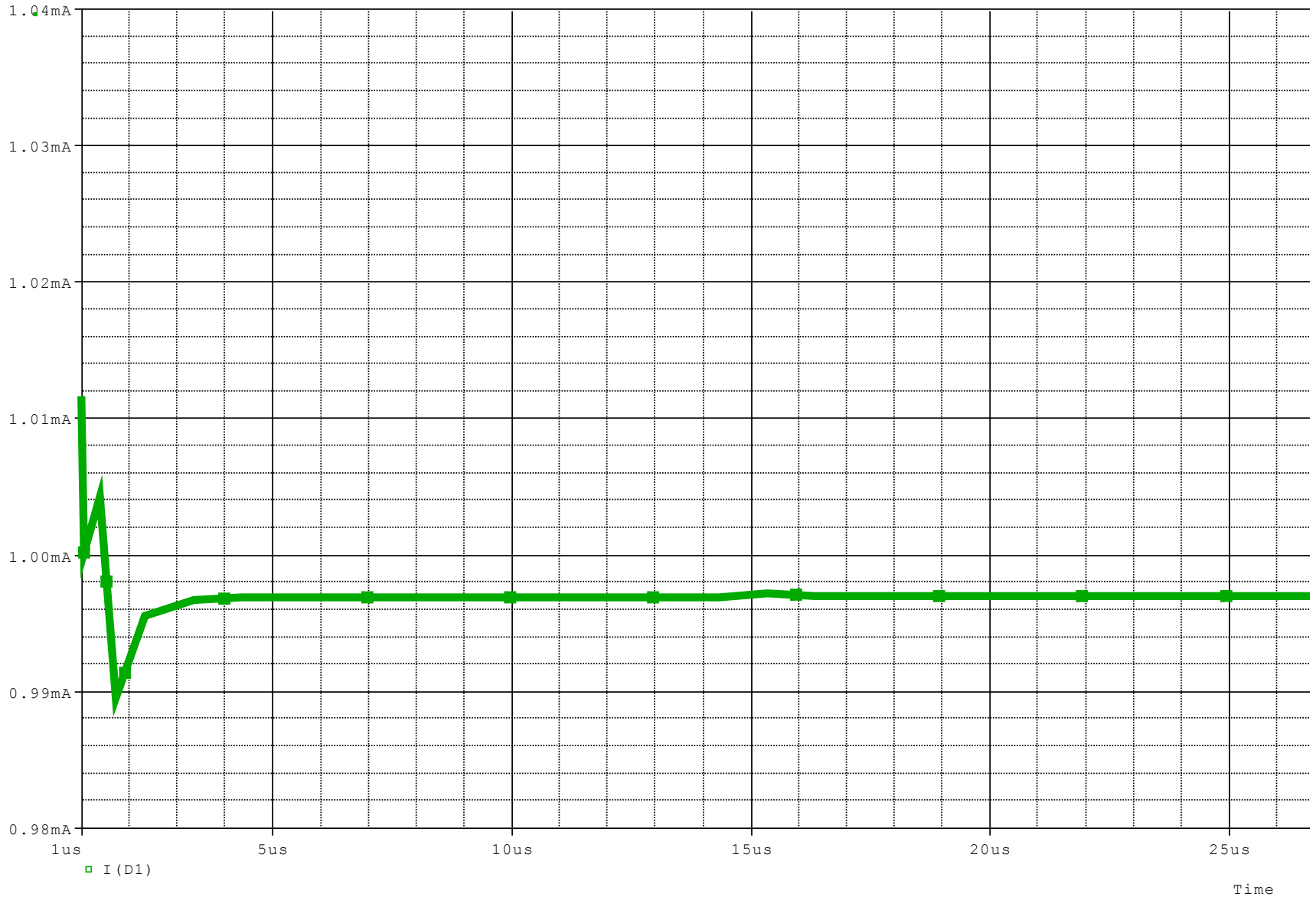
$$C = 1 \text{ nF}$$

$$r_D = 25 \text{ mV} / 1 \text{ mA} = 25 \Omega$$

$$R_L = 500 \Omega$$

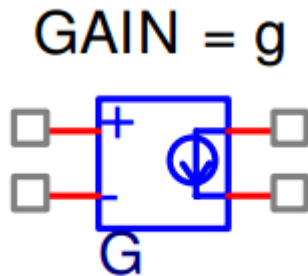


Time



2. (40 points, RLC circuits) Show that with the Spice G component one can obtain a negative resistance. Draw the resulting circuit, including a ground on both sides of the G component, and give the value of the negative resistance.

First off, this is the G/ANALOG component:



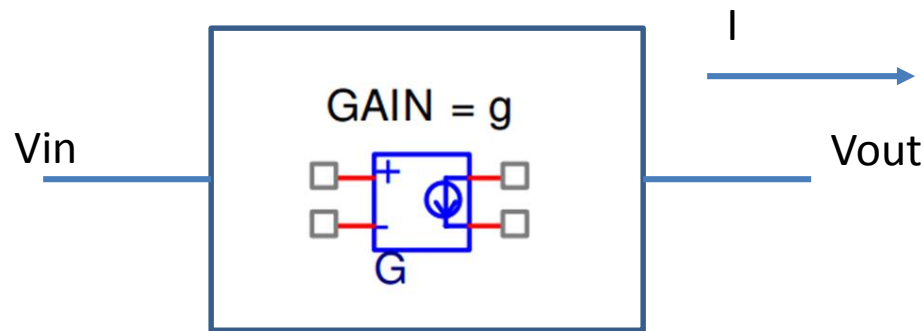
It is a Voltage-Controlled Current Source (VCCS), which means that the current that flows through the two terminals on the right (in the direction of the arrow) is proportional to the voltage across the terminals on the left, multiplied by the gain G .

What you want to do for this problem is set up a black box component that contains this G component inside, whose external voltage and current characteristics resemble negative resistance.

A component exhibits negative resistance when an increase in voltage across its terminals results in a decrease in current flowing through it.

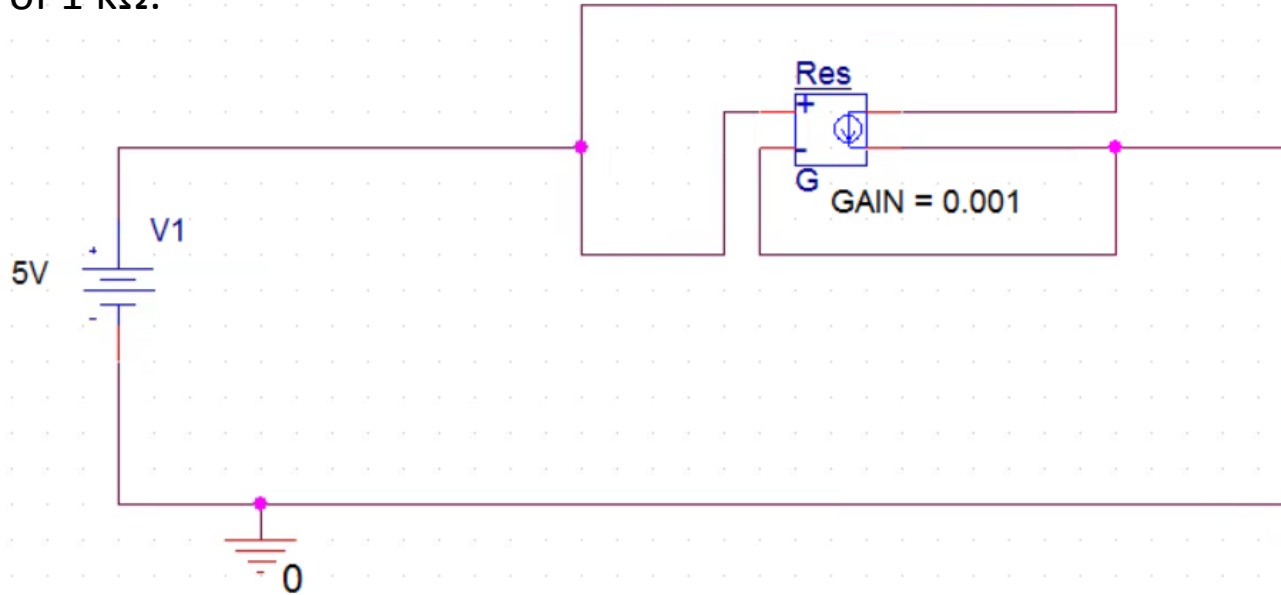
Where $R = V/I$, this basically means that voltage and current have opposite signs. Power = IV , so this would also mean that the component has negative power: it is adding power into the circuit (like a power source).

Let's start by creating a blackbox device that resembles an ordinary resistor

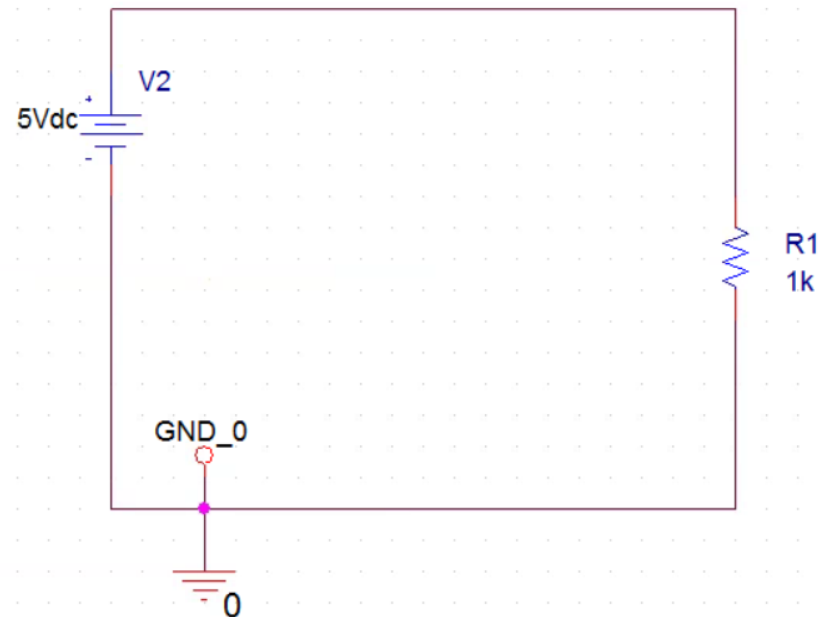


Notice that if you connect the terminals of the G component in the right way, then the gain $G = I/(V_{out}-V_{in})$, which is the conductance. So the resistance of this black box is $R = 1/G$.

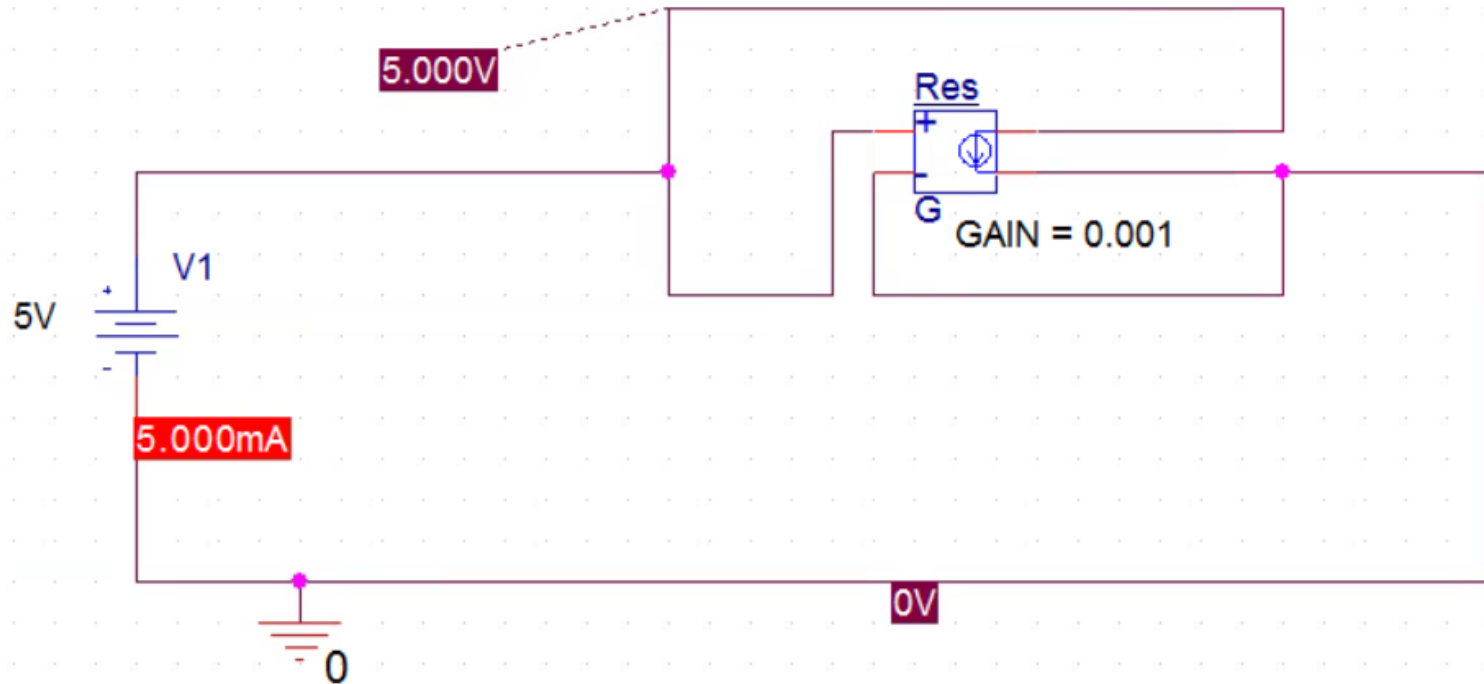
This is the circuit I created. The gain is $1/1000$, which corresponds to a resistance of $1\text{ k}\Omega$.



The circuit's behavior is identical to this one to the right:

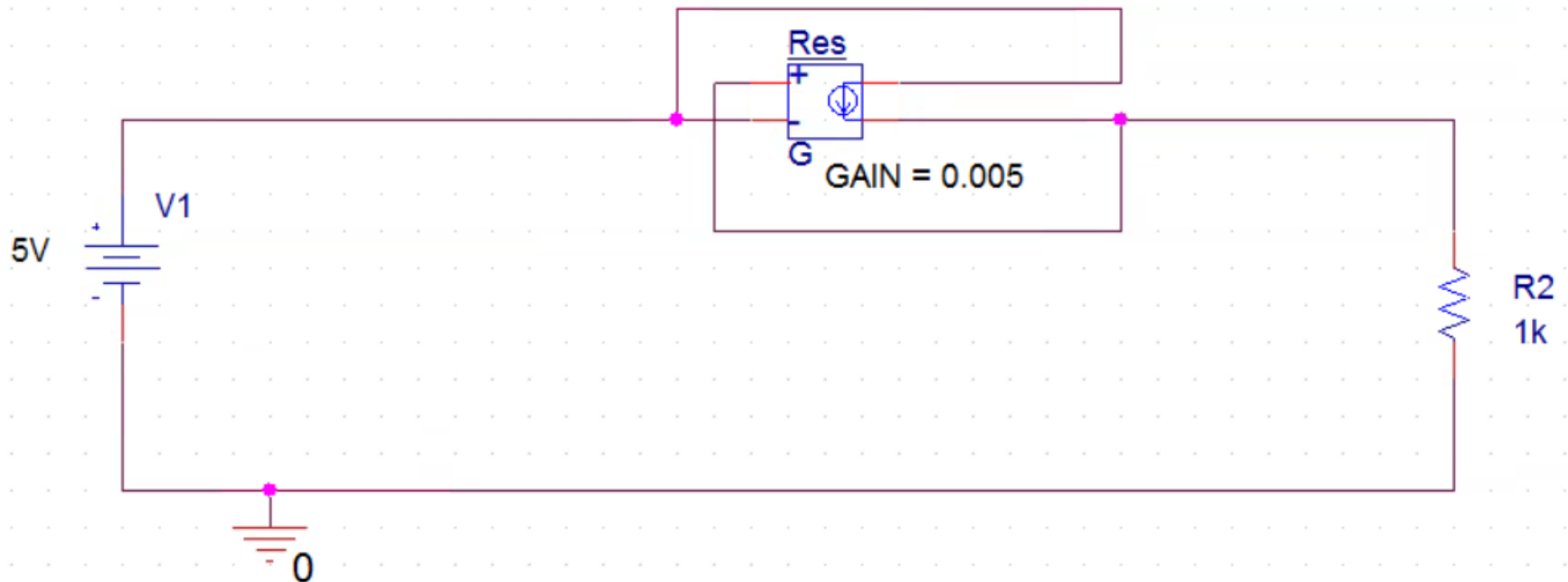


And sure enough, the voltage across the black box resistor drops from 5V to 0V, and the current running through the circuit is $5V/1000\Omega = 5mA$.



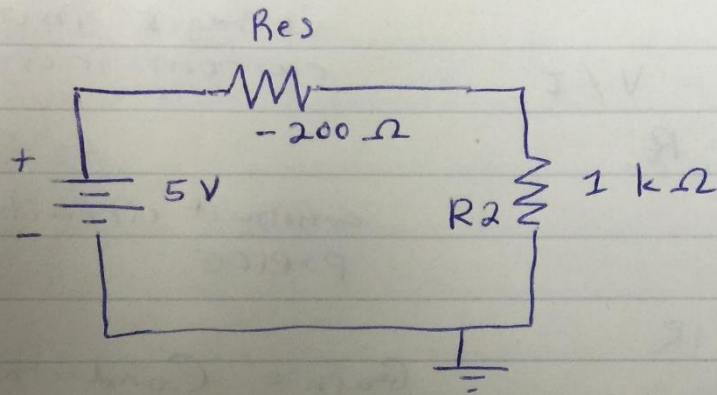
Now, turning this black box component into a negative resistor is very easy. Just switch the polarity of the voltage terminals or the current terminals on the G component, to get a negative relationship between voltage and current ($R = -V/I$). I chose to switch the voltage terminals.

This is what my new test circuit looks like:



I added another resistor (so the circuit does not only have power sources), and I changed the gain to 0.005. So the resistance value of the black box component is now $R = -1/G = -200\Omega$.

This circuit would be expected to behave like this:



$$V_{in} = V_{Res} + V_{R2}$$

$$5V = IR_{res} + IR_2$$

$$5V = I(-200\Omega + 1000\Omega)$$

$$I = \frac{5V}{800\Omega}$$

$$I = 6.25 \text{ mA}$$

$$V_{R2} = V_{in} - V_{Res}$$

$$= 5V - (6.25 \text{ mA})(-200\Omega)$$

$$= 5V + 1.25V$$

$$= 6.25V$$

Res increases the maximum voltage of the circuit, acting as a power source.

And sure enough, it does. The circuit hops up to 6.25V after the negative resistor, and the current through the circuit is 6.25mA.

