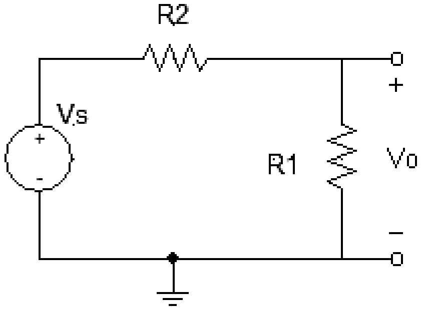
ECE 3254 PreLab 3 notes

Edited 09-09-2009

Voltage Divider



A voltage divider circuit consists of two series resistors, which form a voltage divider because the applied voltage is divided between the two resistors (in this case R1 and R2). The current through R1 (I_{R1}) and R2 (I_{R2}) can be found by dividing Vs by the sum of the resistors, or by solving a loop equation. Note that $I_{R1} = I_{R2}$ because there is only one loop.

$V_{R1} = I_{R1}*R1 = I_{R2}*R1$, $V_{R2} = I_{R1}*R2 = I_{R2}*R2$, or you can use Node Equations to find V_{R1} and V_{R2}.

For the Prelab, find the equation for V_{R1} in terms of Vs, R1, and R2. You will use this equation many times this semester, so be able to remember or derive it quickly.

Oscilloscope and Multimeter input impedance.

In our lab, each meter has about a 10MΩ input impedance (you can verufy this by using one meter as an ohmmeter, and connecting it to measure the resistance of another meter set to measure volts). This 10MΩ input will not significantly load low impedance circuits. When connected to a 1MΩ circuit, the meters will load the circuit somewhat, but accuracy will be acceptable. For a circuit with an impedance greater than 3MΩ, the meter loading effects will be significant.

The oscilloscope input impedance is 1MΩ in parallel with 14 pF.

At low frequencies (≤ 1MHz) with a circuit impedance ≤ 100kΩ, the scope will not load the circuit. As the circuit impedance or the frequency increase above 1Mhz or 100kΩ, the scope loads the circuit and accuracy suffers. We may increase the scope's frequency response and input impedance with a 10X probe, which multiples the scope impedance by 10 and reduces the parallel capacitance. This probe also acts as a /10 voltage divider.

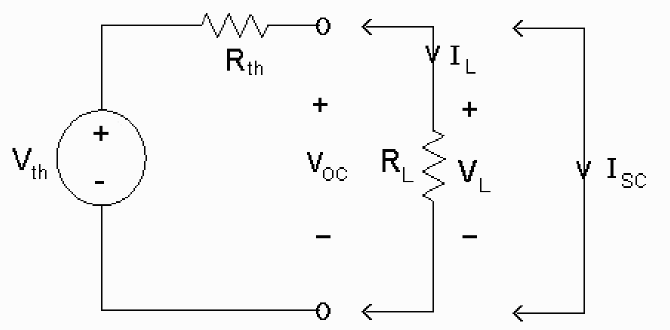
Thevenin Equivalent Circuit

Thevenin's Theorem: Any two terminal linear network composed of a combination of voltage sources, current sources, and resistors, is electrically equivalent to a single voltage source (Vth) and a single series resistor (Rth).

For single frequency AC circuits, Thevenin's theorem can also be applied to complex impedances. In the case of a complex source or load impedance, maximum power transfer occurs when Z_L equals the conjugate match of Z_{TH} ($R_{TH} = R_L$ and $j\omega X_{TH} = -j\omega X_L$).

The Thevenin equivalent is only valid at the load terminals. Internally, we can't say anything about the circuit from which the Thevenin equivalent was derived.

Any circuit can be represented as a “black box” Thevenin source. The Thevenin and Norton models provide a simple way to calculate of load voltage and current, and power for various loads. Instead of having to evaluate an entire circuit each time the load resistance is changed, you can use the simple Thevenin or Norton circuit.



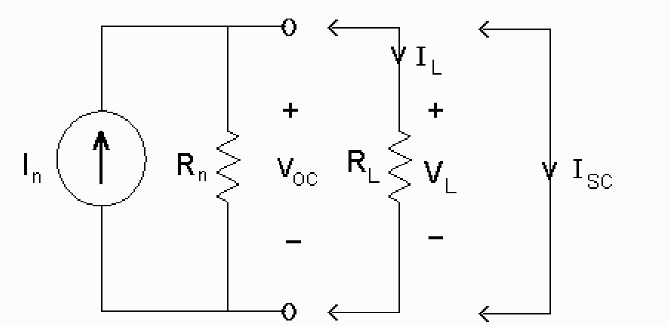
Thevenin Source

$V_{th} = V_{OC}$ (open circuit voltage)

$R_{th} = V_{th} / I_{SC}$ (short circuit current)

Given V_{OC}, V_L, R_L: $V_{th} = V_{OC}$, $I_L = V_L/R_L$,
$V_{Rth} = V_{OC} - V_L$, $R_{th} = V_{Rth} / I_L$

Given V_{th}, R_{th}, R_L: $I_L = V_{th}/(R_{th}+R_L)$, $V_L = R_L I_L$
or $V_L = V_{th}R_L/ R_{th}+R_L)$, $I_L = V_L/R_L$

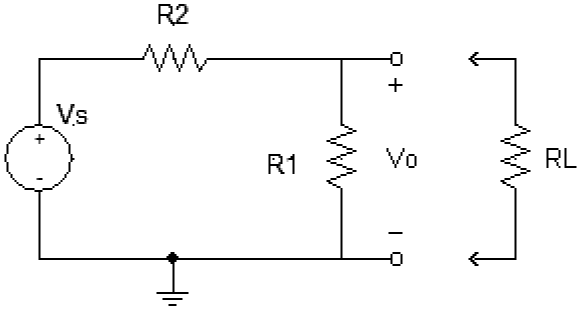


Norton Source

The Norton Source is interchangeable with the Thevenin source. $I_n = I_{SC}$ and $R_n = R_{th}$

$R_n = V_{OC}/I_n = V_{th}/I_{SC} = R_{th}$ and $V_{OC} = V_{th} = I_n R_n$

Thevenin Equivalent of a Voltage Divider



The voltage divider circuit produces an open circuit voltage (Voc) that can be found from your voltage divider equation. The short circuit current is easily found by shorting Vo (so Isc = Vs / R2 because R1 is shorted). Rth is then calculated by solving Rth = Voc / Isc. The result will be the equivalent of R1 || R2 (parallel connected).

Sometimes you can't obtain a short circuit current (perhaps the current would be too high for safety, or you do not have a meter that can make the measurement). Remember that the Thevenin source with a load resistor is just a voltage divider. If you can measure the voltage produced in a known load resistor, you can use the voltage divider equation to calculate Rth. Vth is, of course, still Voc.

Maximum Power Transfer

Maximum power transfer from the source to the load **occurs when $R_{TH} = R_L$**. Efficiency at maximum power transfer is 50% - the load and the source each dissipate half of the power produced.

When $R_L < R_{TH}$ - power transfer is lower, and efficiency is low because the source dissipates more power than the load.

When $R_L > R_{TH}$ - power transfer is lower, and efficiency is high because the load dissipates more power then the source.

A practical example of the relationship between R_L, R_S, power transfer, and efficiency can be found in the common stereo system audio amplifier. The output impedance of the amplifier is typically 2Ω or less. For our example amplifier, $V_{TH} = 20V_{RMS}$ and $R_{TH} = 2\Omega$.

With an 8Ω speaker, $P_{total} = 40W$, $P_{speaker} = 32W$, $P_{amp\ dissipation} = 8W$
With a 4Ω speaker, $P_{total} = 67W$, $P_{speaker} = 45$ W, $P_{amp\ dissipation} = 22W$ (the amp begins to get warm!).
With speakers paralleled for 2Ω, $P_{total} = 100W$, $P_{speaker} = 50$ W (WooHoo!), $P_{amp\ dissipation} = 50W$ The speakers get 50W, BUT the amplifier must dissipate the other 50W and many amplifiers do not have heat sinks capable of dissipating enough heat when a low impedance load is used.
With more speakers paralleled for 1Ω, $P_{total} = 133W$, $P_{speaker} = 44$ W, $P_{amp\ dissipation} = 89W$ (OUCH!). Not only has power transfer dropped, but efficiency is terrible and the amplifier will probably overheat.

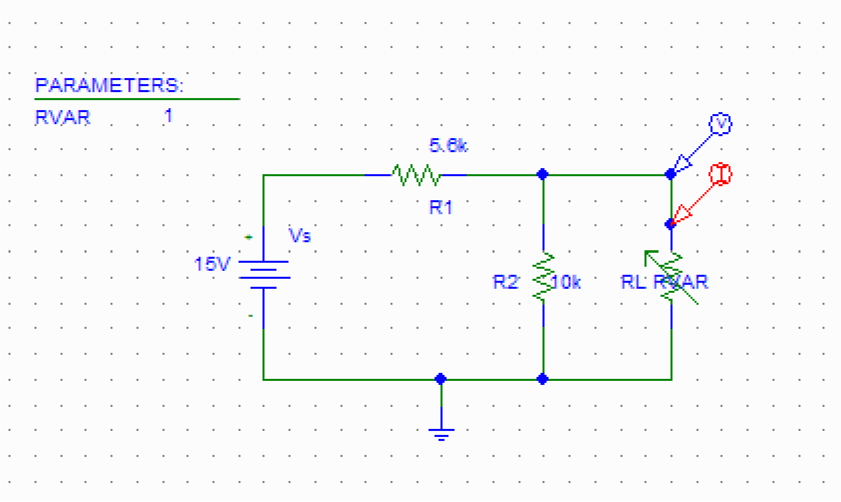
Pspice modeling using the Parametric Sweep

Begin with the voltage divider from Prelab question 2. Save the model as Lab 3 Thevenin or something similar.

Add a new variable resistor to use for RL
[Draw > Get New Part > R_var > Place & Close].
Hit the esc key after you have placed the var resistor.
Rotate the var resistor [Edit > Rotate], then drag it to where you want it.
Edit the Variable resistor to name it RL, and edit the component so that Value=RVAR, SET=1.
Connect (wire) RL in parallel with R1.

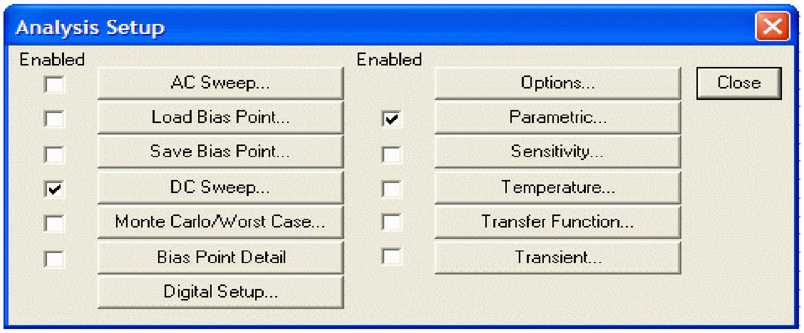
Move (or place) the Voltage and Current probes to measure V_{RL} and I_{RL}.
If you already have probes, click them and move them to measure I_{R1} and V_{R1}.
If you need probes, see Lab 1 about how to click the probes on the toolbar and place them.

Add a Parameter
[Draw > Get New Part > Param > Place & Close].
Hit the esc key after you have placed the Parameter somewhere on the schematic.
Click the Parameter and edit it so that Name1=RVAR, Value1=1.
Your schematic should look something like this:



Set up the analysis

[Analysis > Setup > check the boxes for DC Sweep and Parametric]

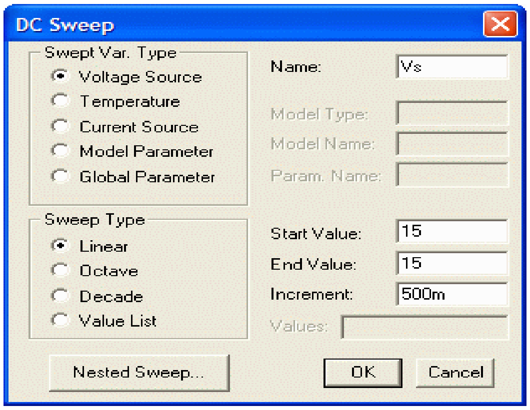


Analysis Setup

☐ AC Sweep...
☐ Load Bias Point...
☐ Save Bias Point...
☑ DC Sweep...
☐ Monte Carlo/Worst Case...
☐ Bias Point Detail
Digital Setup...
Options...
☑ Parametric...
☐ Sensitivity...
☐ Temperature...
☐ Transfer Function...
☐ Transient...

Click the DC sweep button and set these values (Note: this sets Vs to remain constant):

Set Var Type = Voltage Source,
Sweep Type = Linear,
Name = Vs
Start Value = 15
End Value = 15
Increment = 500m
> hit OK



DC Sweep

Swept Var. Type
● Voltage Source
○ Temperature
○ Current Source
○ Model Parameter
○ Global Parameter

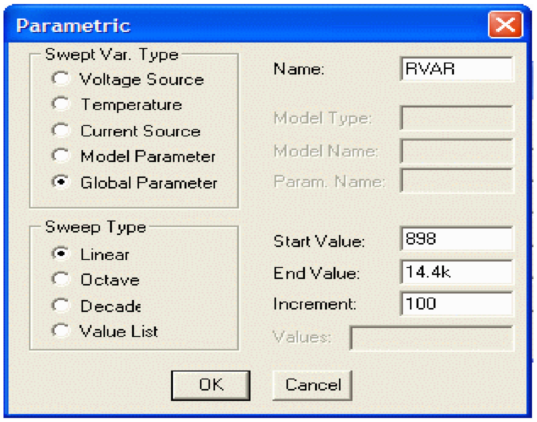
Name: Vs
Model Type:
Model Name:
Param. Name:

Sweep Type
● Linear
○ Octave
○ Decade
○ Value List

Start Value: 15
End Value: 15
Increment: 500m
Values:

Click the Parametric button and set these values
(Note: this sets RL to begin at (calculate and enter the value below) Rth/4, step in 100Ω intervals, and end at (calculate and enter the value below) Rth*4):

Swept Var Type = Global Parameter
Sweep Type = Linear,
Name = RVAR, Start Value = (Rth/4),
End Value = (Rth*4),
Increment = 100
> OK
> Close



Parametric

Swept Var. Type
○ Voltage Source
○ Temperature
○ Current Source
○ Model Parameter
● Global Parameter

Name: RVAR
Model Type:
Model Name:
Param. Name:

Sweep Type
● Linear
○ Octave
○ Decade
○ Value List

Start Value: 898
End Value: 14.4k
Increment: 100
Values:

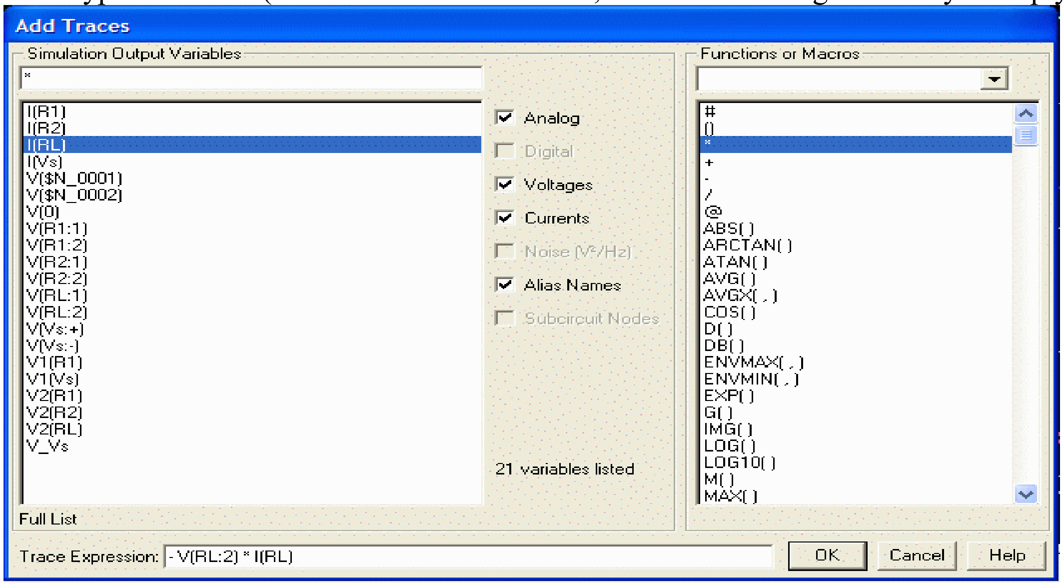
Run the Simulation

Hit the simulation button (or hit F11 or click Analysis > Simulate)
If your simulation had no errors, you should see V(RL:2) steadily increase and I(RL) near 0.
Troubleshoot your wiring and setup if you have errors.

Add a trace for P_{RL}

Click Trace > Add Trace >
In the "Simulation Output Variable" window: choose V(RL:2) (or your voltage probe is name),

Then type "*" after the voltage probe name (* is multiply, and P = V*I),
Then in the "Simulation Output Variable" window: choose I(RL)(or what your current probe is named),
Then type "*1000" (to make the trace readable, we are converting to mW by multiplying by 1000)



☑ Analog
☐ Digital
☑ Voltages
☑ Currents
☐ Noise (V²/Hz)
☑ Alias Names
☐ Subcircuit Nodes

21 variables listed

Trace Expression: - V(RL:2) * I(RL)

> OK. (You should see another trace. If it is negative, double click
the trace name and place a "-" in front of the expression and say OK).

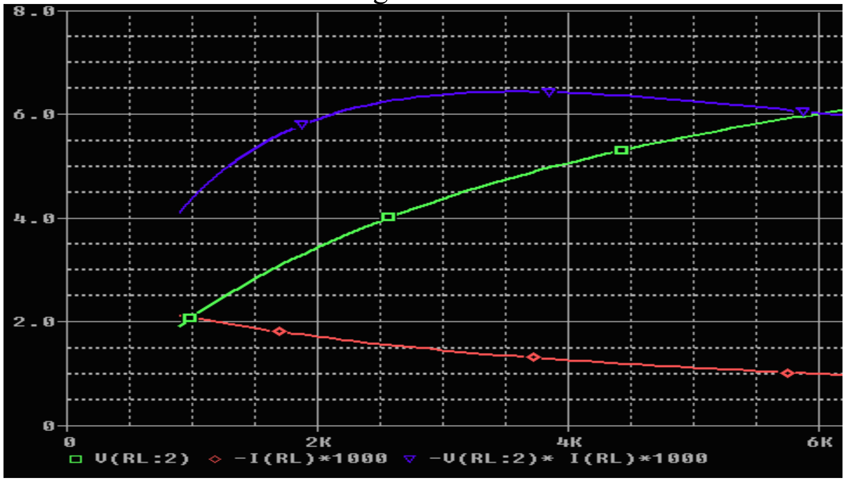
Change I(RL) to milliAmps

double click the -I(RL) trace identifier and change it to -I(RL)*1000
This will move the current trace up into visible range on the simulation display.
Multiplying the current trace by 1000 essentially changes the scale from A to mA.

Widen each trace for printing:

Click on each trace, then right click > properties > Width >
use drop down to select the second choice > OK

You should see now something like this:



Use the cursor to find and Label V_{RL}, I_{RL}, and P_{RL}.

Click the "Toggle Cursor" button on the toolbar , move the cursor to the desired value of RL
For each trace, Click the small square, diamond, or triangle to make a trace active, then click the "Mark Label" button on the toolbar .