

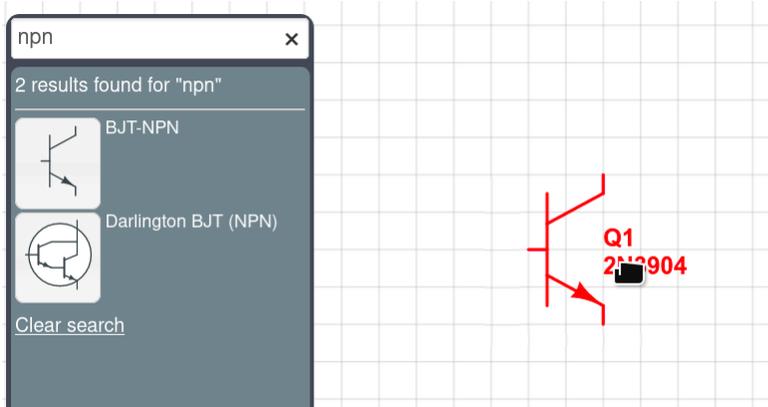
From: CircuitLab Academy
Sent: Wed, 15 Nov 2023 05:56:23 +0000
To: Metha Kerdkeaw
Subject: Input and output impedance made effortless

Hi Metha,

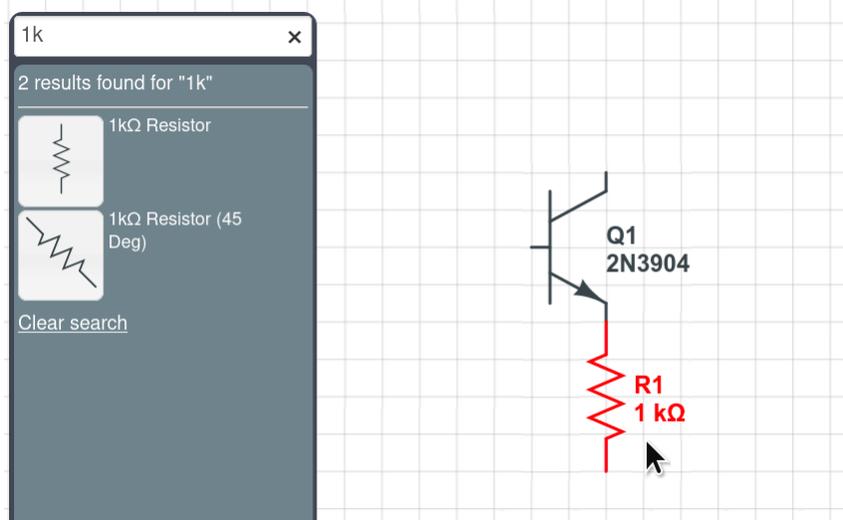
Calculating a circuit's input and output impedance by hand can be tricky, but CircuitLab makes it easy to check your work. Flip through the screenshots below to see how we can quickly simulate the Thevenin equivalent resistance looking into the input and output of an amplifier.

First, let's draw a BJT emitter follower circuit.

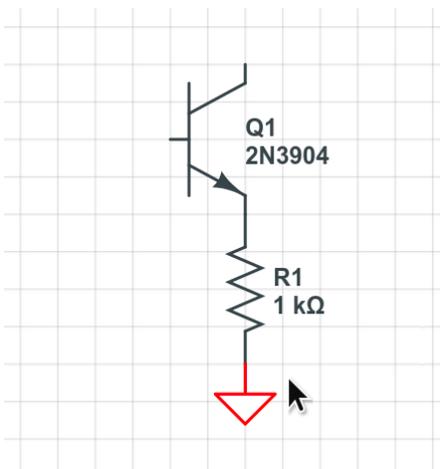
Press / (**forward slash**) to begin a toolbox search and type "NPN". Then, **click and drag** an NPN BJT from the toolbox to your circuit:



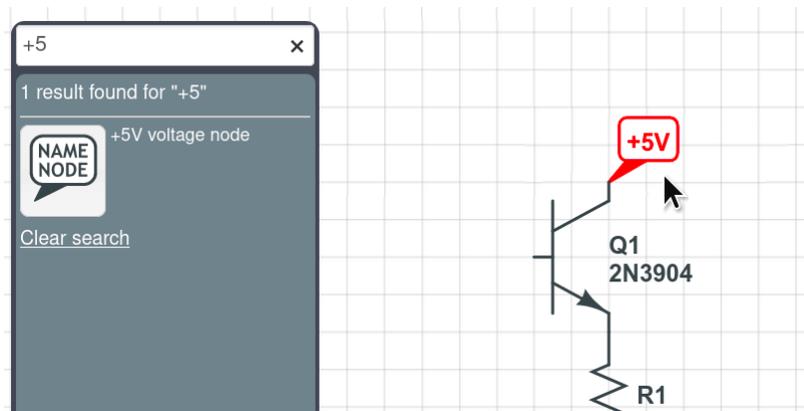
Press / (**forward slash**) to search again and type "1k". Then **click and drag** the 1K resistor from the toolbox to Q1's emitter:



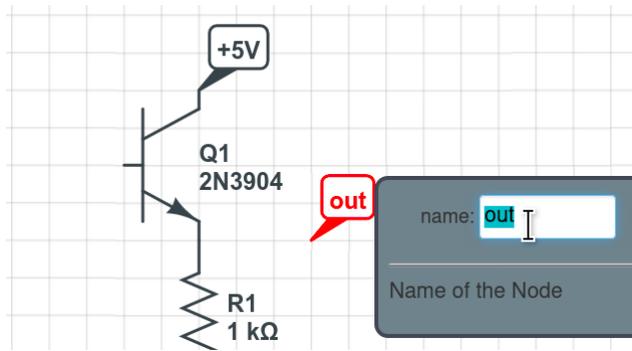
Press **G** and **click** to insert a ground node at the bottom of the resistor:



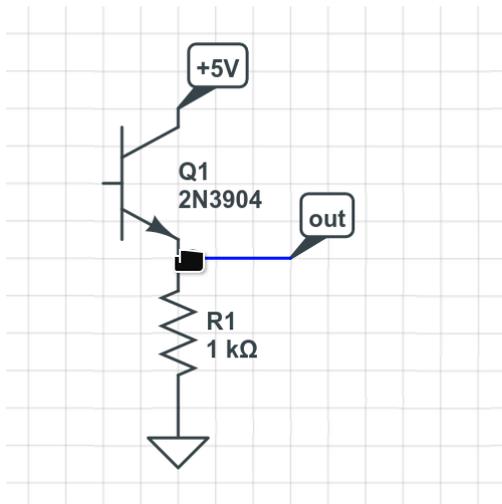
Press **/** (forward slash) and search for "+5", and then **drag** the +5V voltage node to Q₁'s collector:



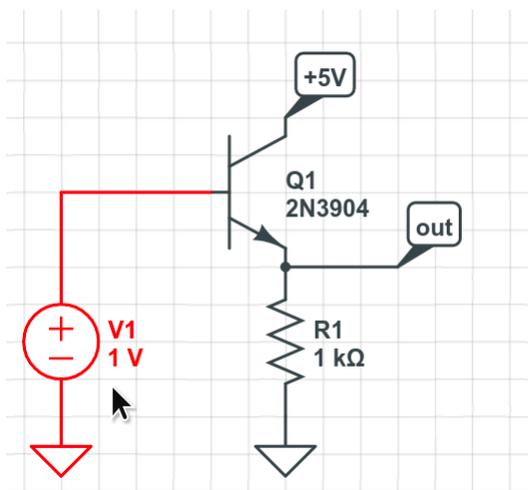
Press **N** and **click** to insert a node name at Q₁'s emitter. **Double-click** and name it "out":



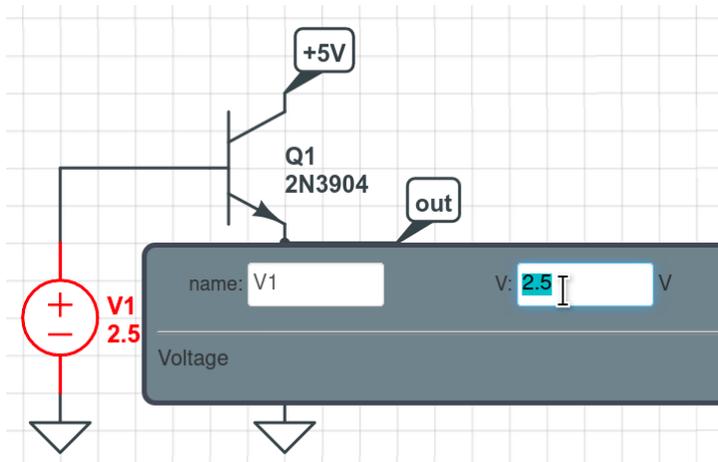
Drag a wire from the node name to Q1's emitter:



Drag a Voltage Source to the schematic. Then press **G** and **click** to insert a ground node, and finally, **drag** to wire the positive terminal to Q1's base:

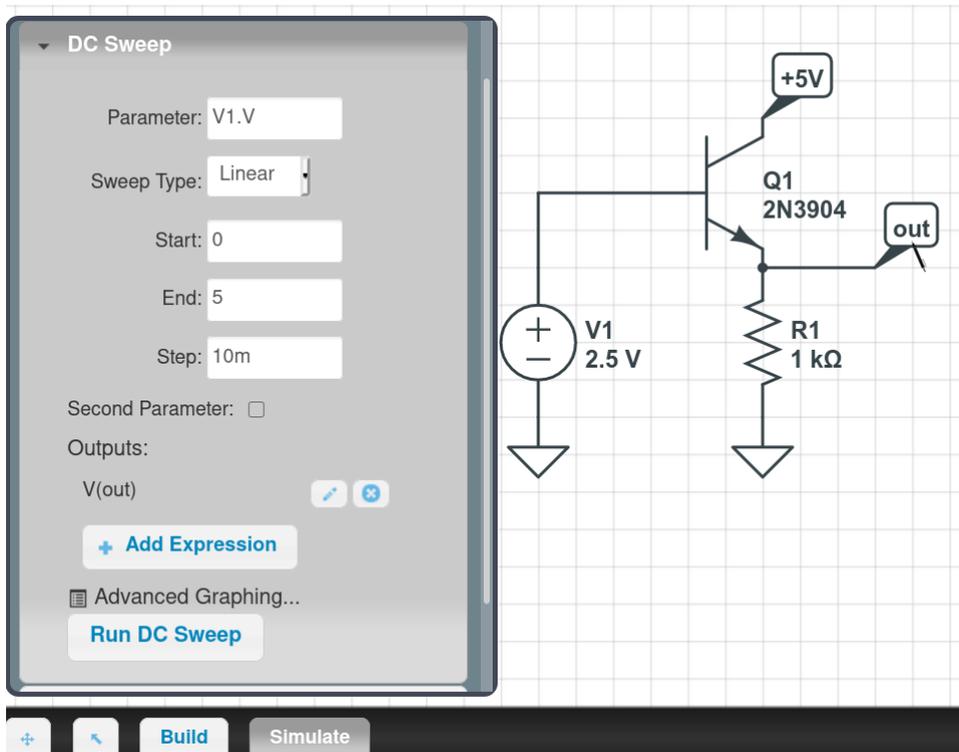


Double-click V₁ and change its voltage to 2.5, the middle of our range:

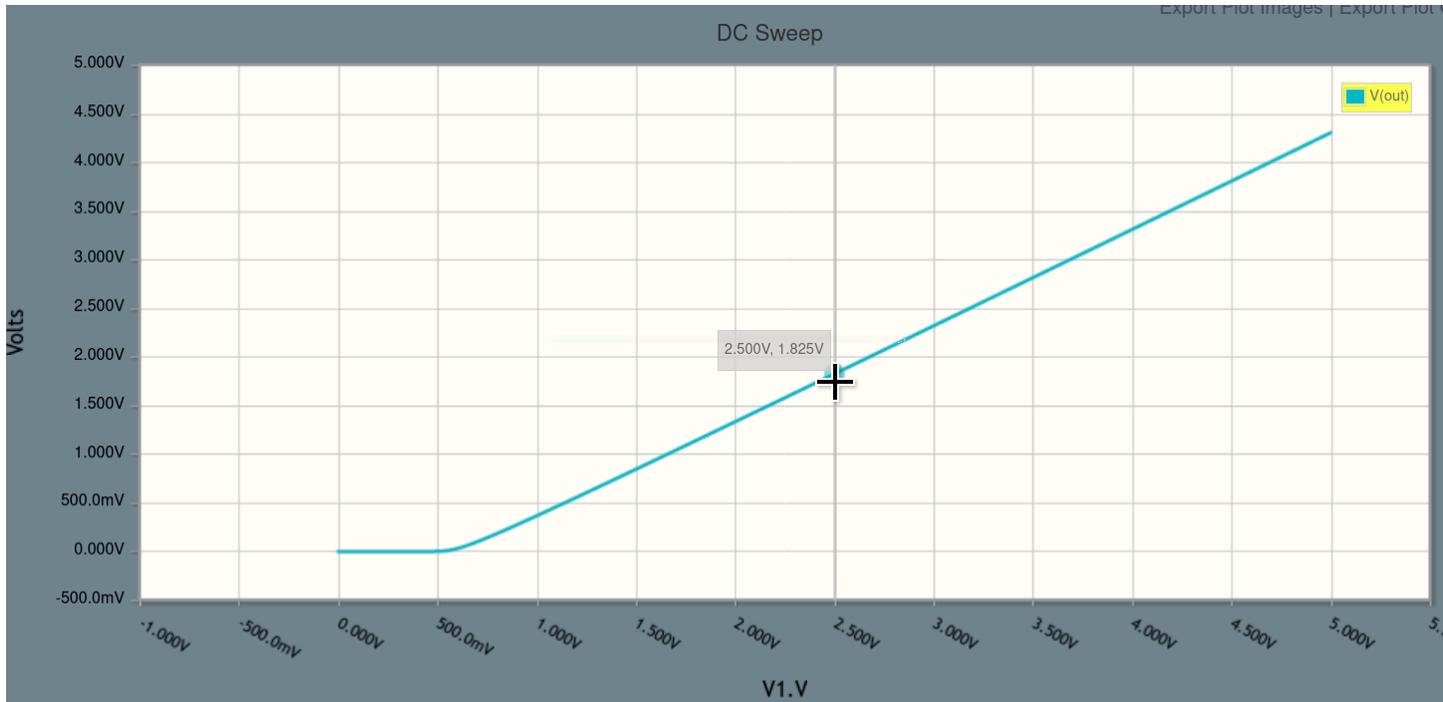


Before we look at impedance calculations, let's make sure the emitter follower is working properly.

Click **Simulate** at the bottom of the window, then expand the **DC Sweep** tab and choose "V1.V" as the **Parameter**, with a **Start** of 0, **End** of 5, and **Step** of 10m as shown. Click the "out" node name to add V(out) to the **Outputs** list:

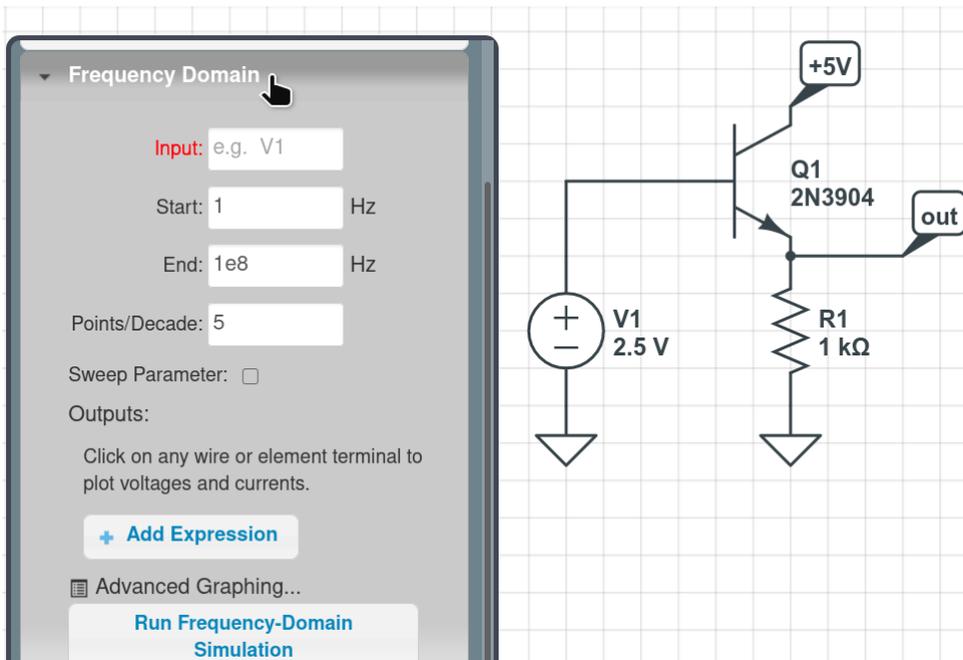


Click **Run DC Sweep**. A plot window appears:

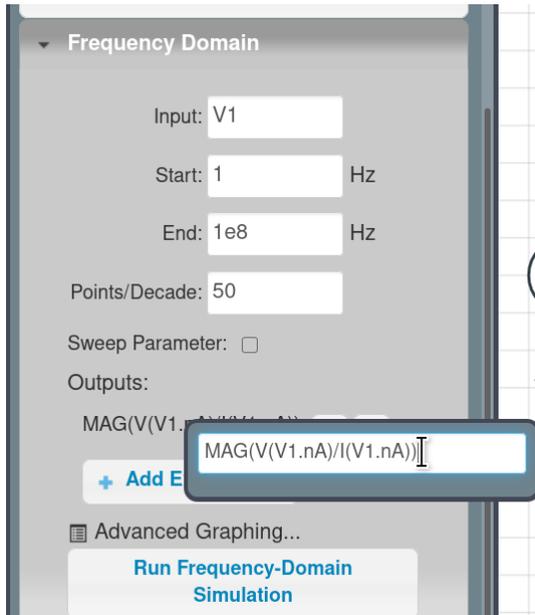


Perfect: we can see that over most of the range there's a 1:1 slope between input and output voltage, as we'd expect for an emitter follower.

We're ready to find the input impedance. Click **Hide** on the plot window, and in the simulation settings, click to open up the **Frequency Domain** tab:

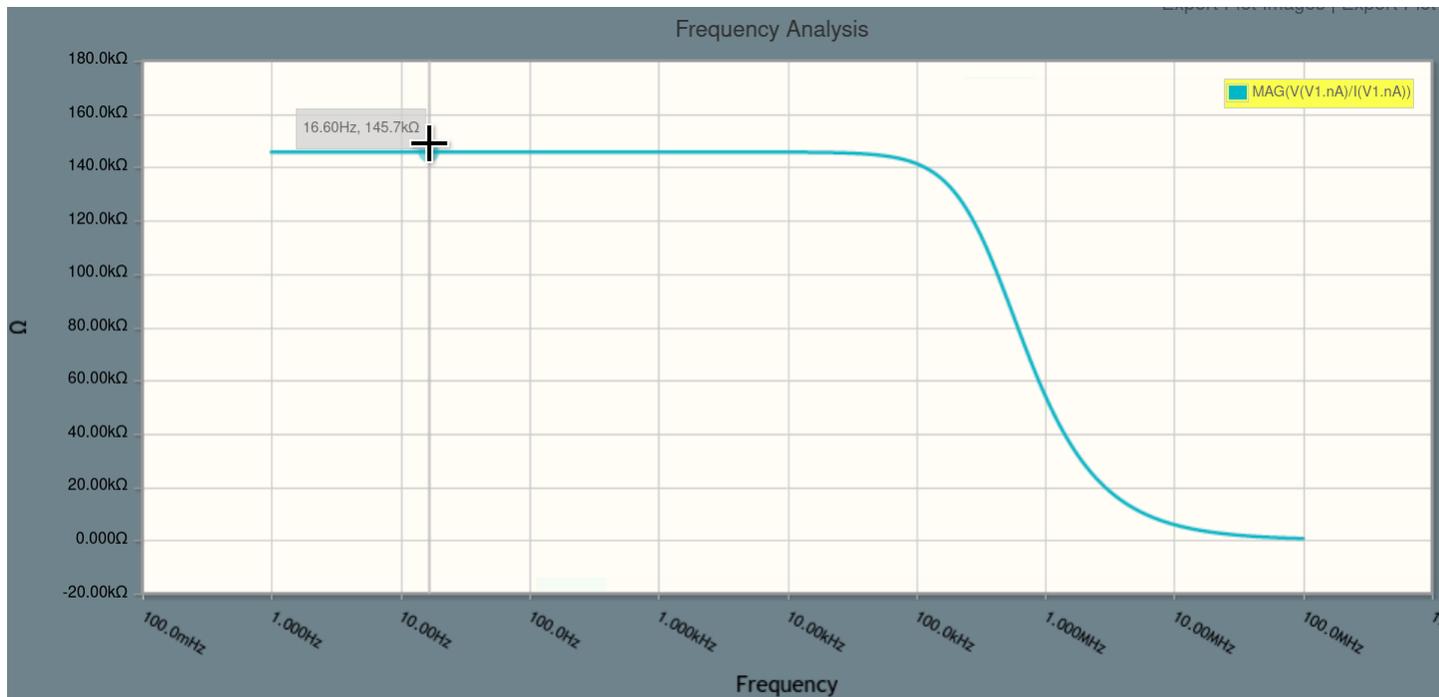


Set the **Input** source to V_I , and increase **Points/Decade** to 50. Click **+ Add Expression** and type in " $\text{MAG}(V(V_{I.nA})/I(V_{I.nA}))$ ":



This is the expression for the input impedance. The simulator is testing small variations in the input source V_I , and then we're asking it to compute the small-signal voltage divided by the small-signal current (from V_I into the base of Q_I) that results. The **MAG** tells the simulator that we're only interested in the magnitude, not the phase, of this relationship.

Click **Run Frequency-Domain Simulation**. A plot window appears. **Hover** over the trace:

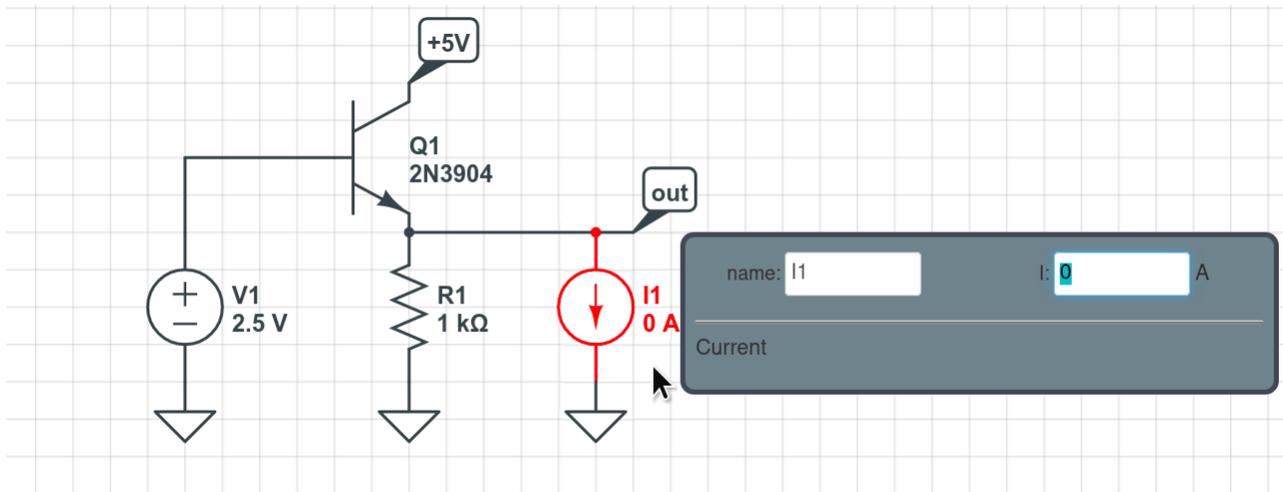


At DC and low frequencies, the input impedance of our circuit is 145.7 kilohms. This input impedance is relatively high and means that the emitter follower does not present much load to whatever is driving it.

What about output impedance?

To calculate output impedance, we need to make one modification to the circuit, adding a test current source.

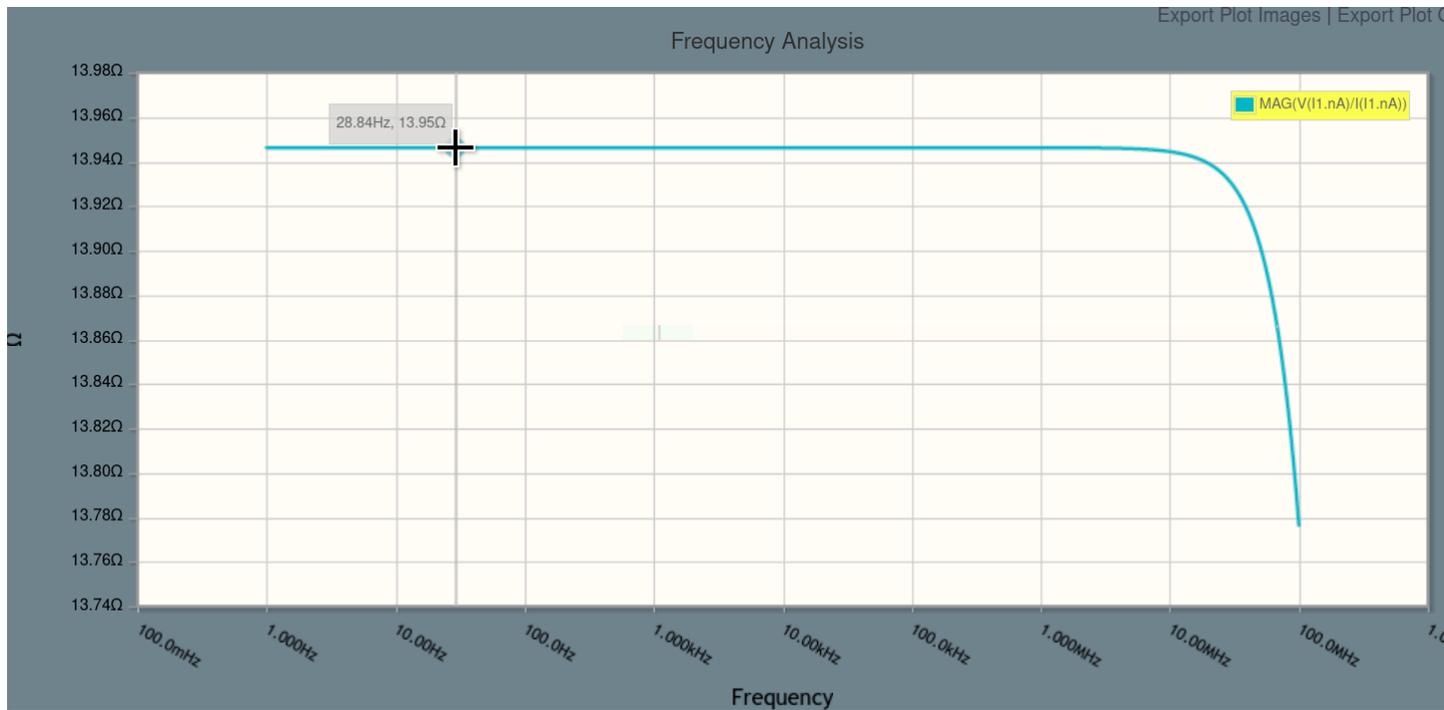
Click **Build** at the bottom of the window to return to build mode. **Drag** a current source from the toolbox to the schematic, and connect it to the "out" node. Press **G** and **click** to add a ground. Finally, **double-click** I1 and set its current to be 0:



Note that we need I1's current to be 0 because we don't want to affect the DC operating point of the simulation. The Frequency Domain simulation is always looking at a small-signal model of the circuit linearized about the DC operating point.

Click **Simulate** to reopen the simulation settings window. We're going to make two changes. First, change **Input** source to I1. Second, click the pencil icon in the **Outputs** list to edit the expression to be "MAG(V(I1.nA)/I(I1.nA))" (replacing V1 with I1 twice):

Click **Run Frequency-Domain Simulation**. A plot window appears. **Hover** over the curve:



We can see that the output impedance is about 14 ohms. This is relatively low, telling us that the emitter follower can drive a substantial output load without its output being compromised.

The emitter follower, with the simplicity of just one BJT and one resistor, achieves a ratio of over 10000 between its output and input impedance.

CircuitLab's advanced simulation engine makes it easy to find the small-signal impedances looking into any terminal. In fact, it makes it incredibly easy to see how these parameters vary when we change the circuit a bit.

Click to focus the simulation settings window, and click to check the **Sweep Parameter** box. Enter "R1.R" as the **Parameter**, with a Linear sweep from a **Start** of 1k to **End** of 10k, with **Step** of 1k:

Points/Decade: 50

Sweep Parameter:

Parameter: R1.R

Sweep Type: Linear

Start: 1k

End: 10k

Step: 1k

Outputs:

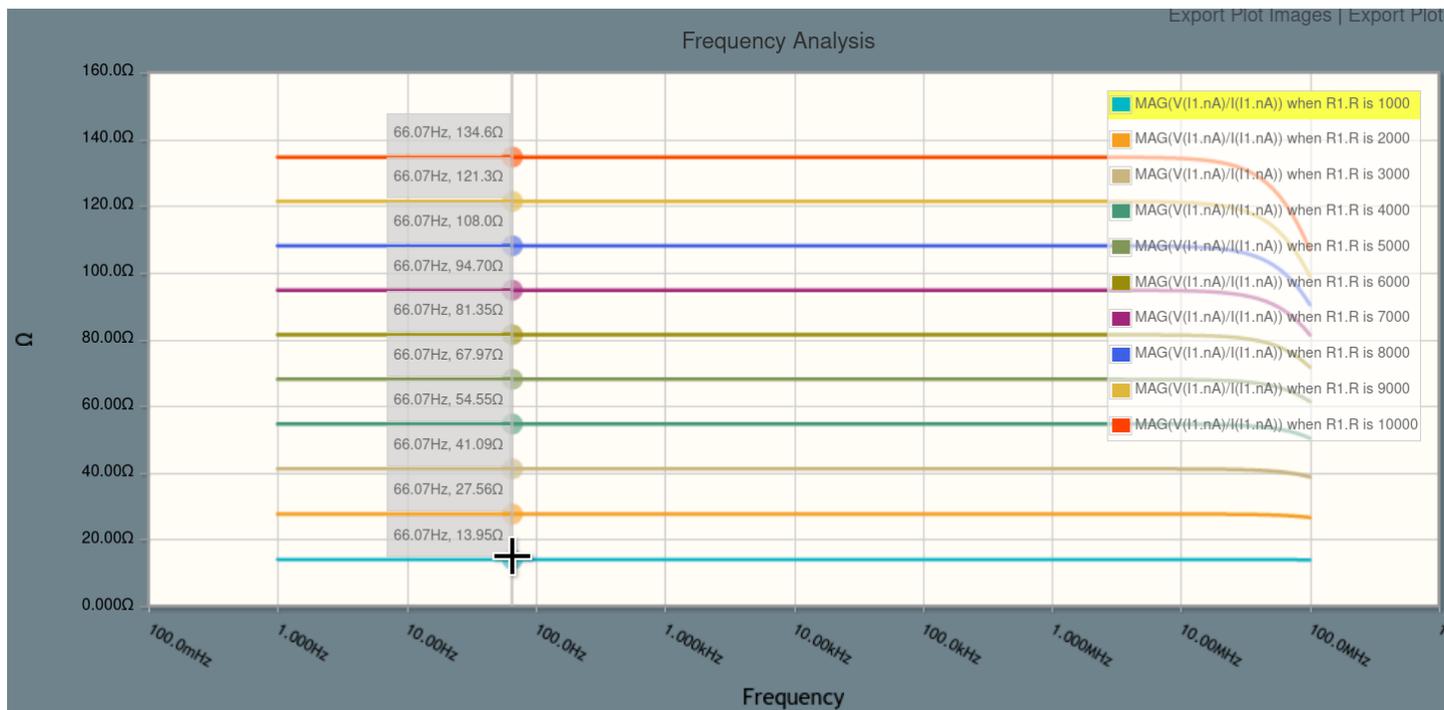
MAG(V(I1.nA)/I(I1.nA))

+ Add Expression

Advanced Graphing...

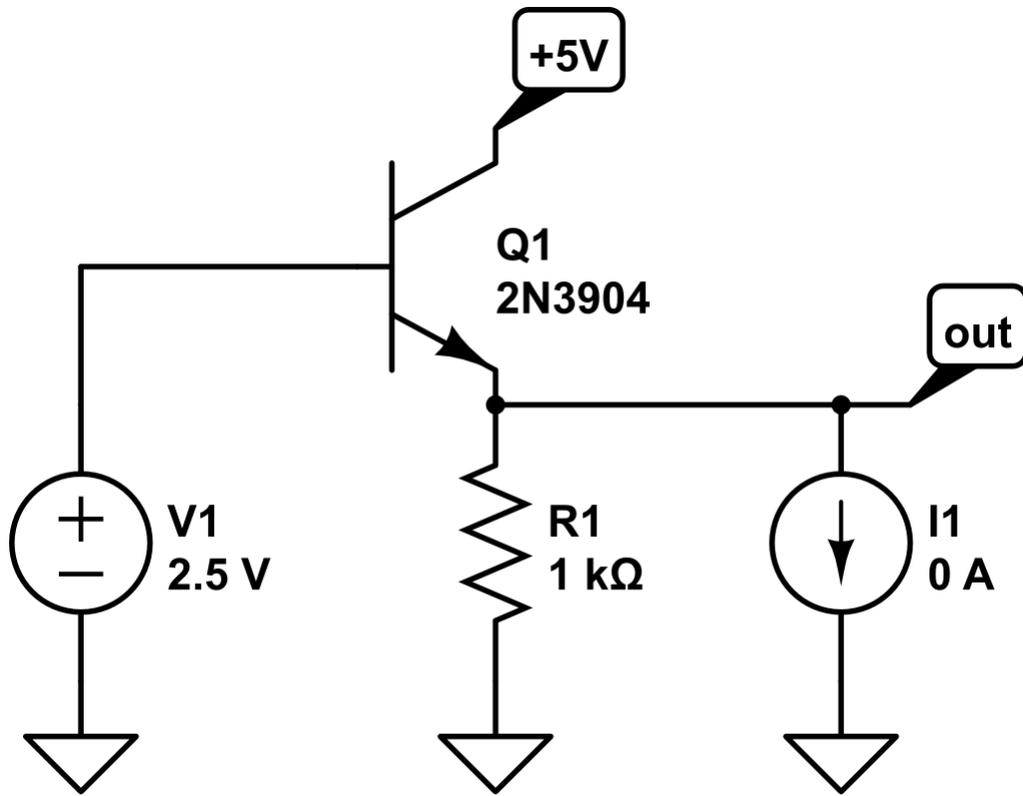
Run Frequency-Domain Simulation

Click **Run Frequency-Domain Simulation**. A plot window appears:



In just a few clicks, we've now calculated 10 different output impedances, depending on our choice for resistor R1. This can help us, for example, to choose a value for R1 where the output impedance is low enough for our needs, but high enough not to burn power (reducing battery life) unnecessarily.

That's it! Click below to [open the final circuit](#), or [try it yourself from scratch](#) (recommended).



[Unsubscribe from CircuitLab Academy onboarding emails](#)