In this video, you’ll learn how to simulate this simple circuit in Simscape. Get started by opening MATLAB.

You can create a new model from the Simulink start page.

To build the circuit, we need a DC voltage source. Double-click the canvas to search for a block.

You can see that there are several DC voltage source blocks. We only want to use blocks from the Simscape Foundation Library, so select that one.

The block looks a little small. You can use the mouse scroll wheel to zoom in on the canvas.

By default, this source provides 1V. Double click the voltage source to open the block parameters. Now you can adjust the voltage.

With the supply ready, let’s add the resistors. Again, select the resistor from the foundation library.

To rotate the resistor, use control R.

This is a 2 Ohm resistor.

To connect the circuit, click and drag a terminal to another terminal. You can also adjust the signal lines by clicking and dragging.

The circuit has another resistor in parallel. You can use the usual keyboard shortcuts to copy and paste the resistor: control c to copy and control v to paste.

With the connections complete, let’s run the model.

Uh oh, looks like there’s an issue. Simscape models require a solver configuration block to set parameters for the numerical solver. This block doesn’t affect the circuit topology, but is required for the simulation to run.

After connecting the solver configuration block, run the model again.

Looks like there’s another issue. This time, there’s an issue with the solvability of the equations. This happens because we haven’t assigned a reference node.

To fix this, add an electrical reference block.

With these additions, the simulation runs successfully. Now you’re ready to take measurements.