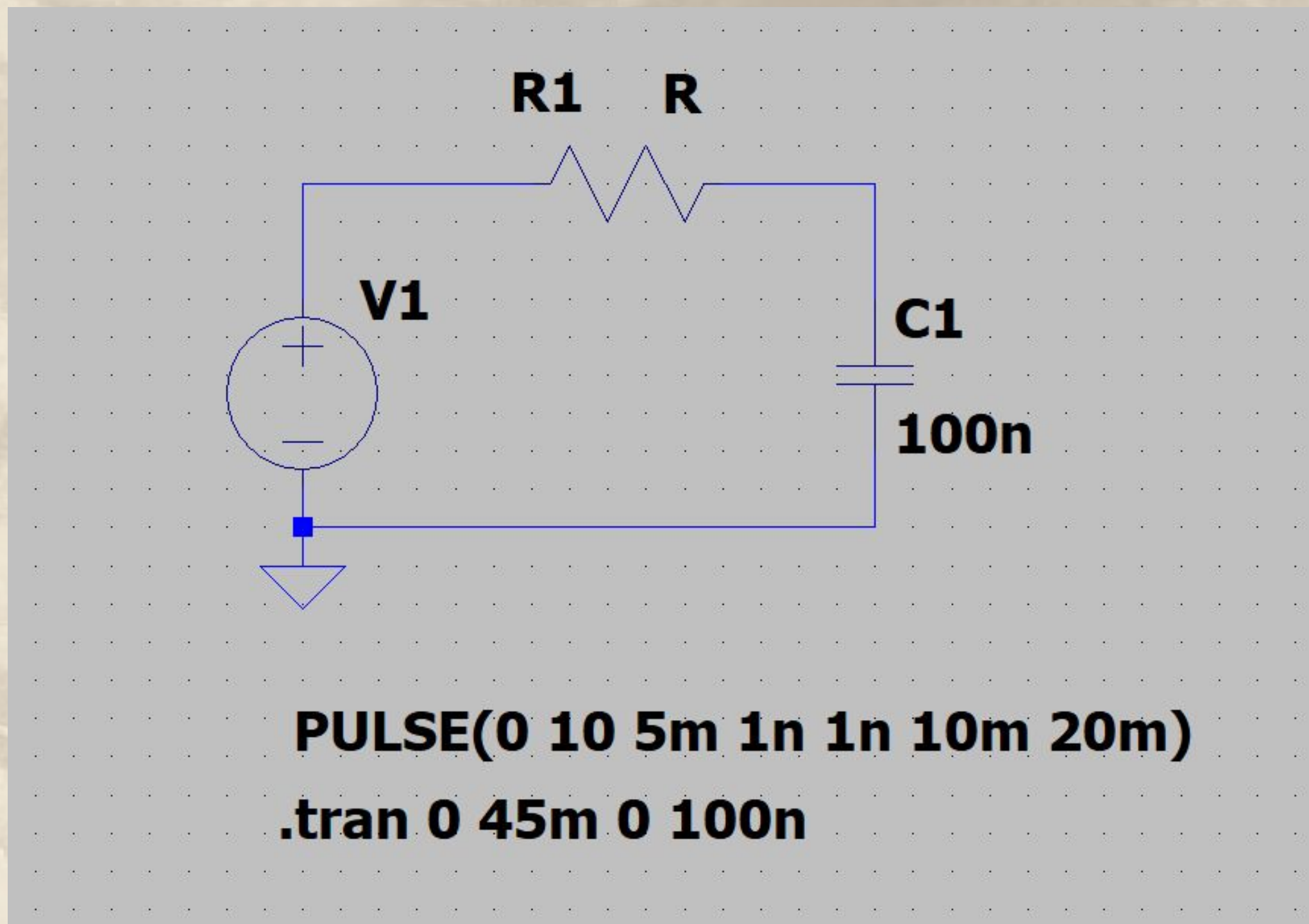


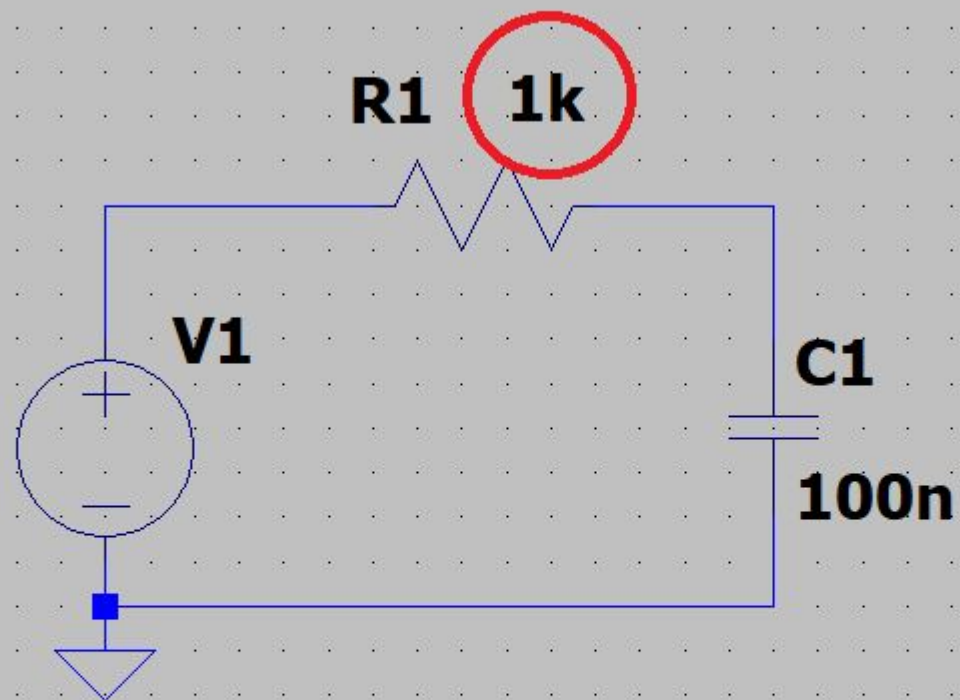
Drops of LTSpice



How to repeat analysis with
different values?

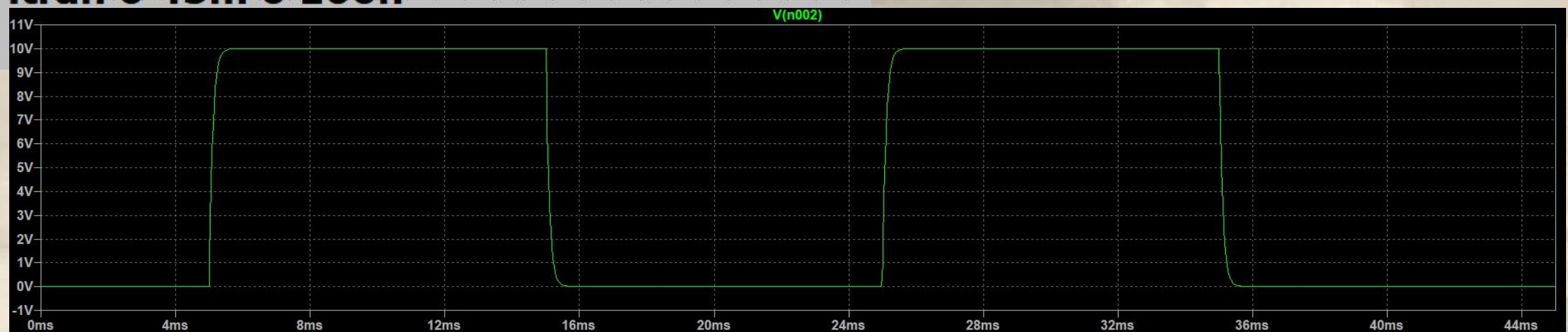
Let's imagine we have this circuit, and we'd like to test different values for R.





PULSE(0 10 5m 1n 1n 10m 20m)

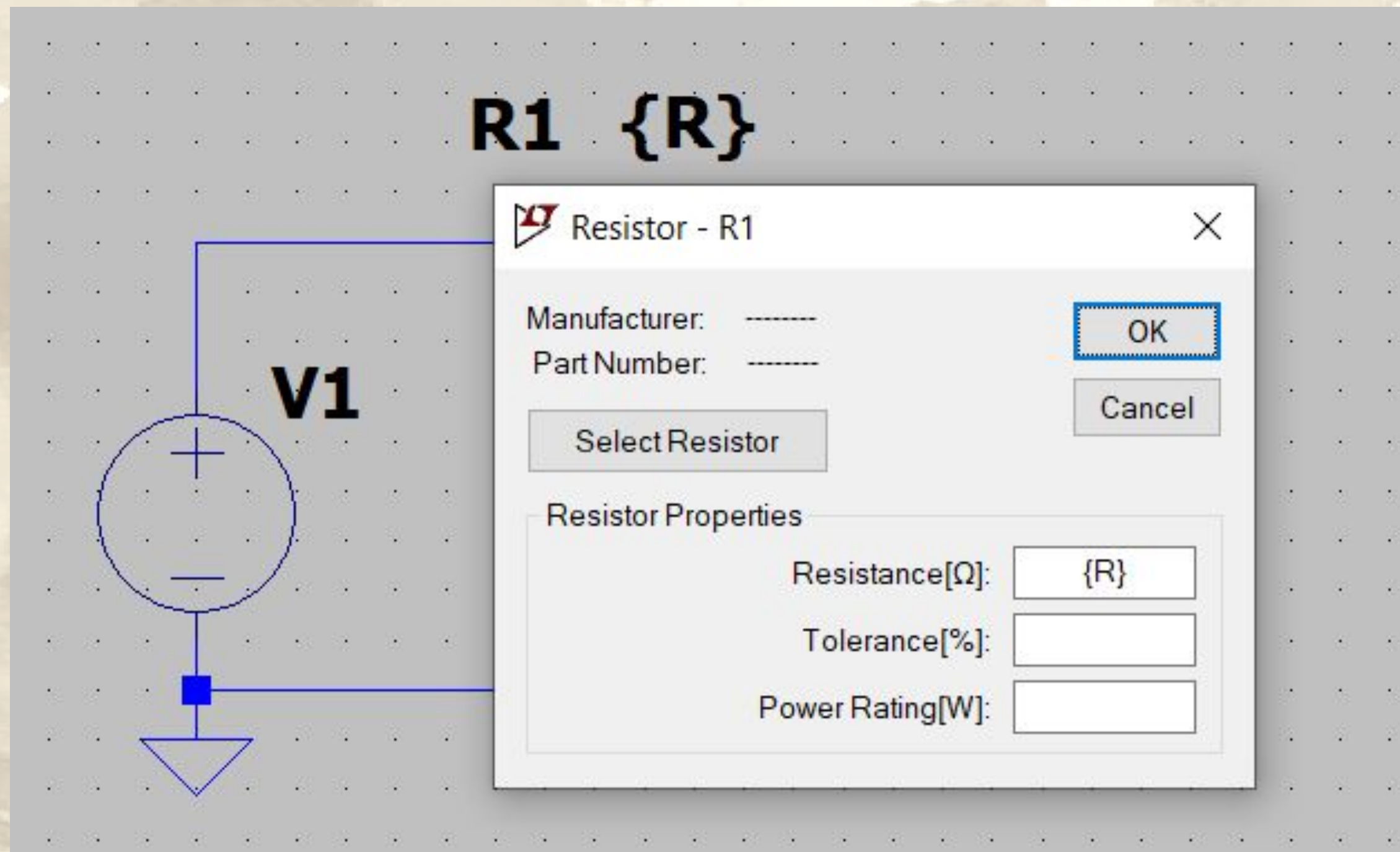
.tran 0 45m 0 100n



One option is to set each test value manually.

But we can do better!

Change the value of the component for a variable, for example {R}.



The curly bracket is necessary.

After, you should add this directive.

Edit Text on the Schematic:

How to netlist this text

☐ Comment

☒ SPICE directive

Justification

Left

☐ Vertical Text

Font Size

1.5(default)

OK

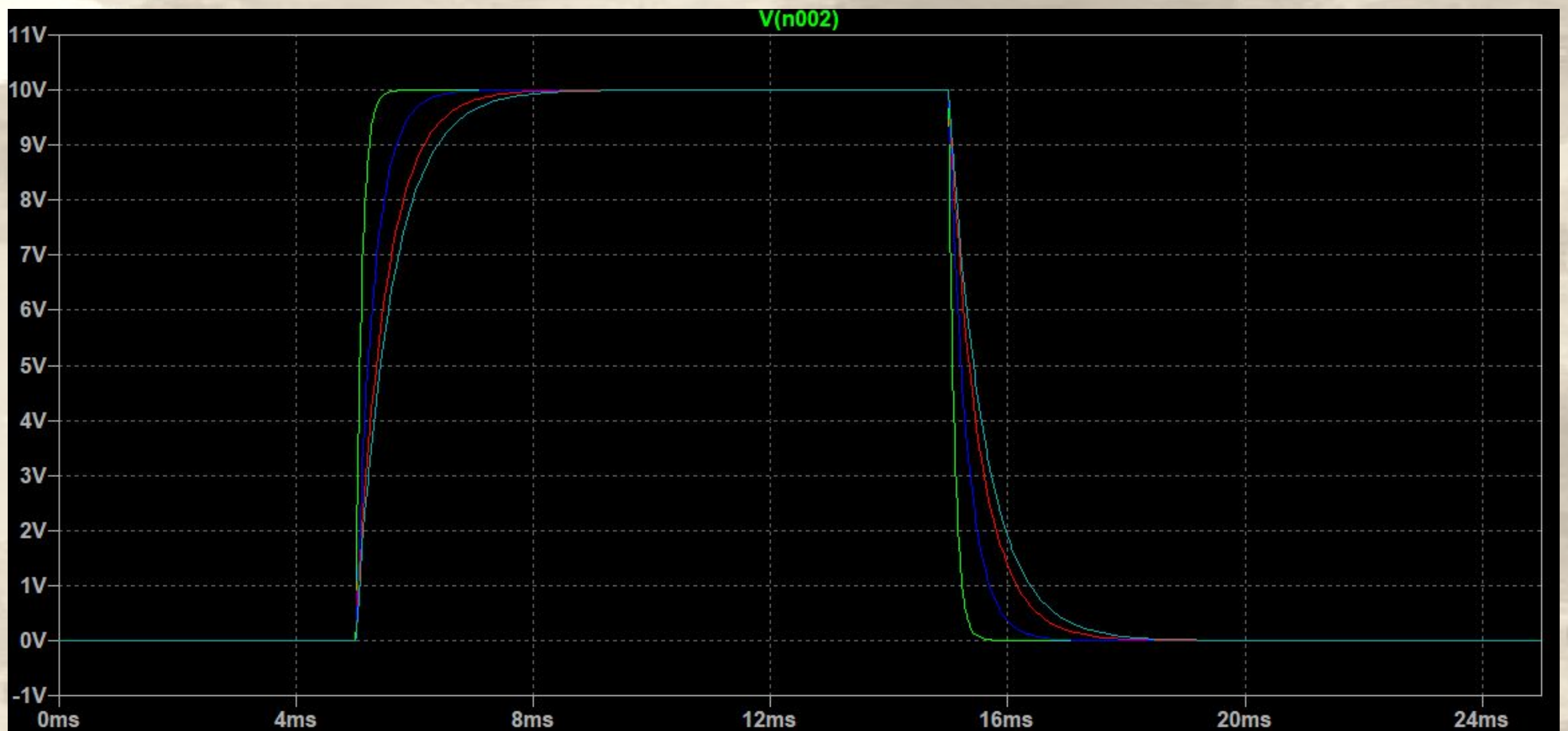
Cancel

.step param R 1k 6k 2k

Type Ctrl-M to start a new line.

It'll change the R value from 1k Ω to 6k Ω with steps of 2k Ω

Once you run, the LTSpice will test all values for you.



Francesco Sacco
[linkedin.com/in/saccofrancesco](https://www.linkedin.com/in/saccofrancesco)