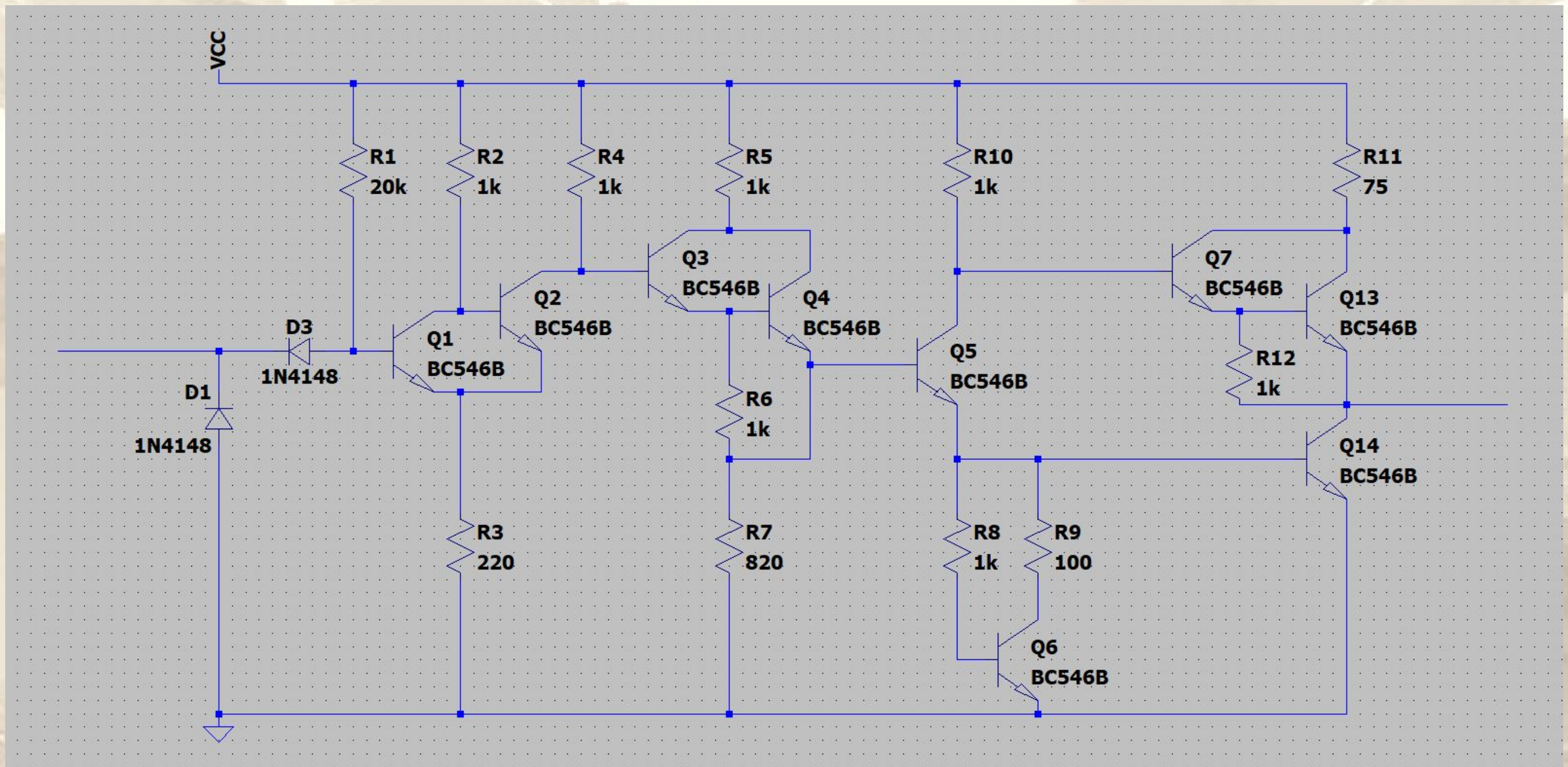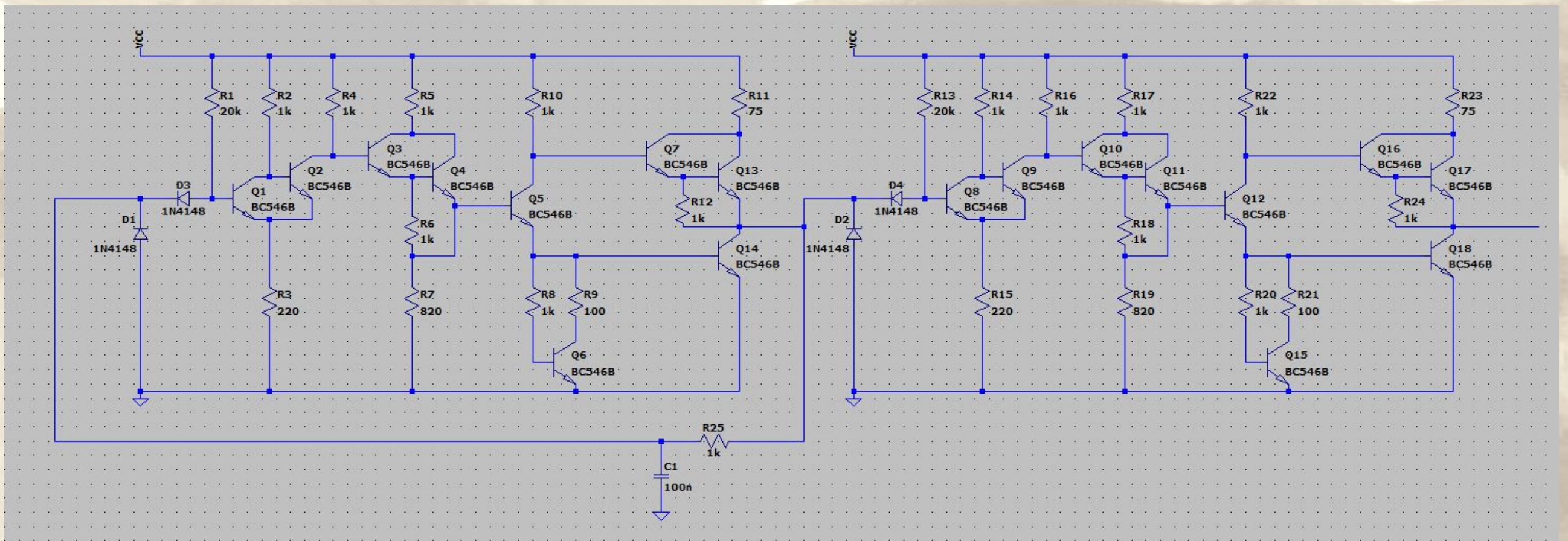# Drops of LTSpice

How to create subcircuits?

Let's imagine that you created a very nice Schmitt Trigger circuit.

And you want to use it as a component.
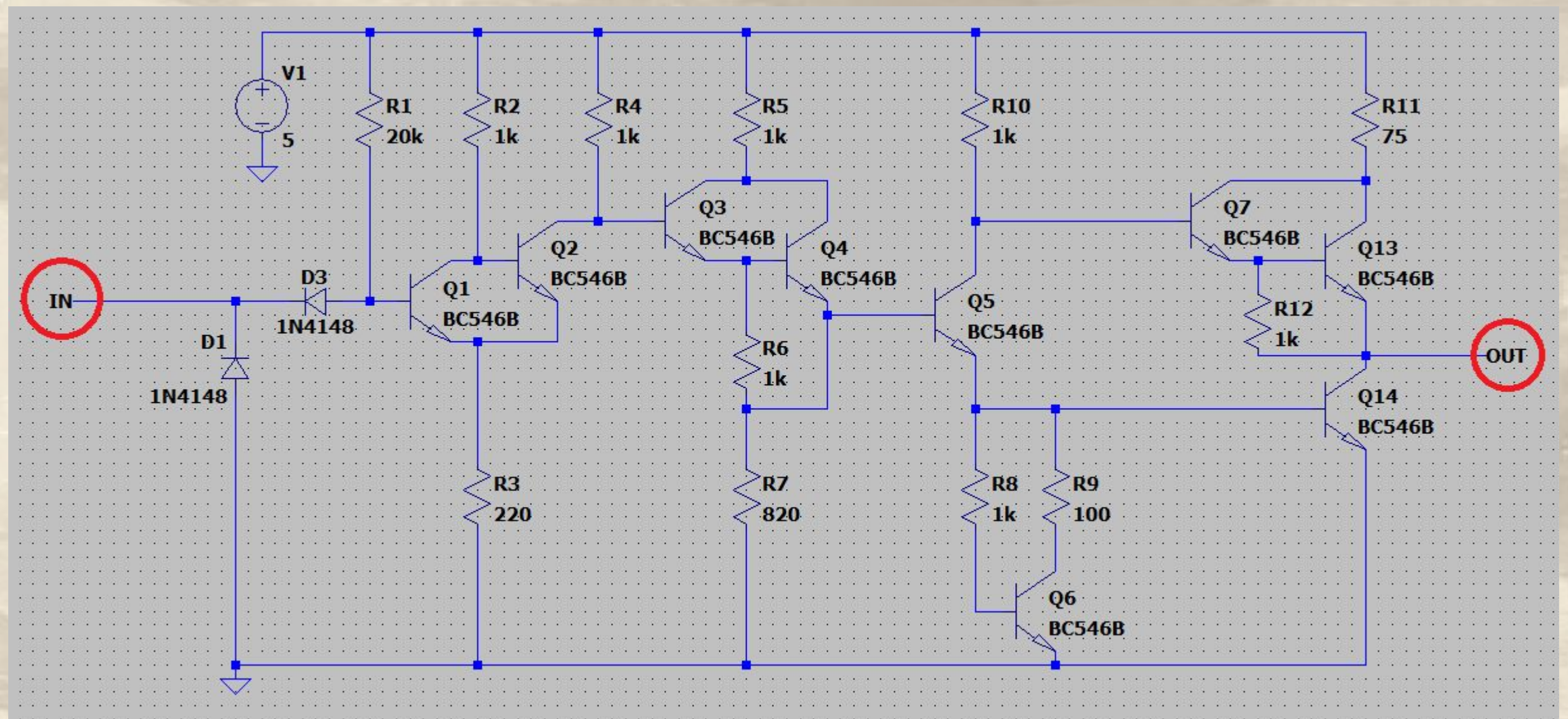
However, a bigger circuit like this can be complex to use or analyze.

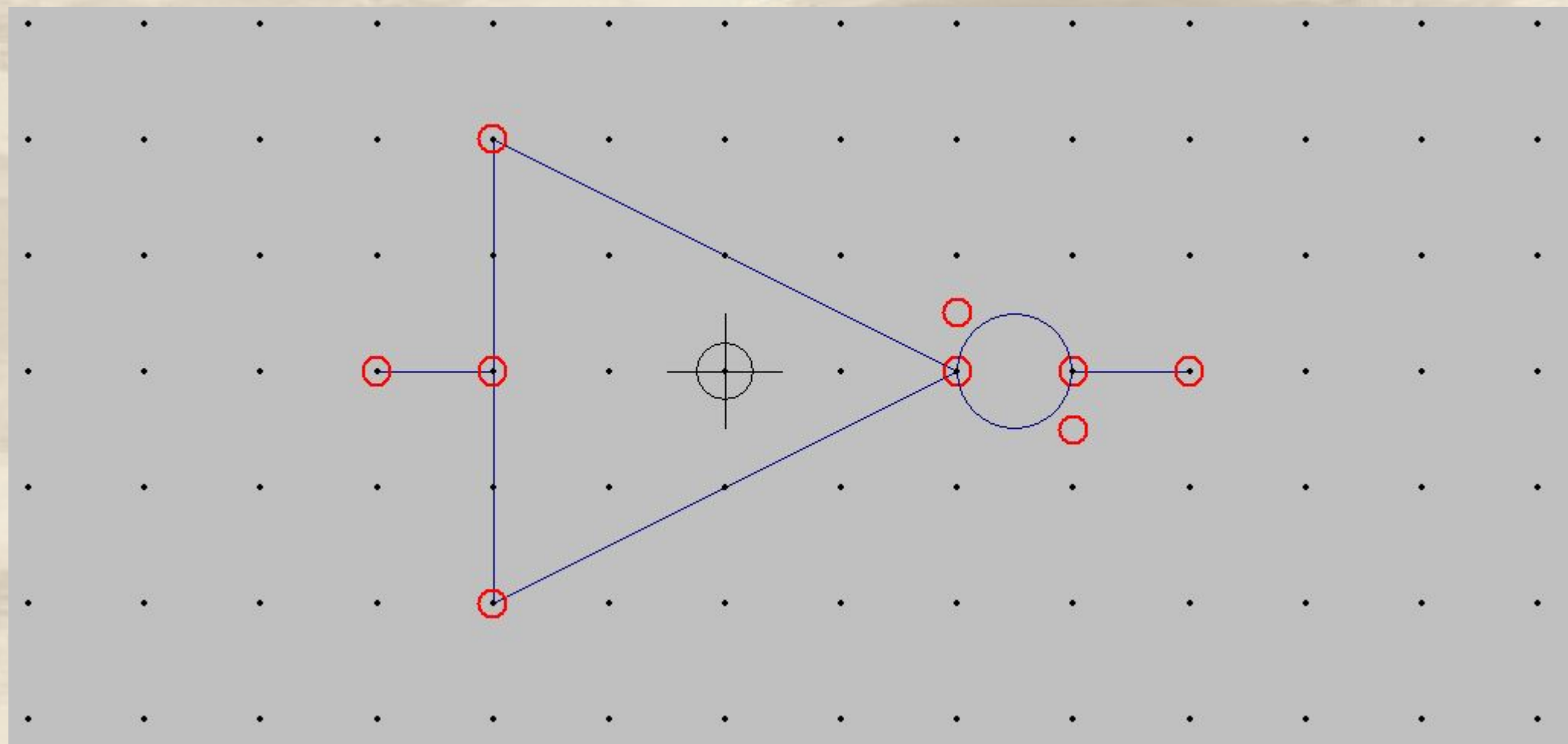Can we simplify?

Of course we can!

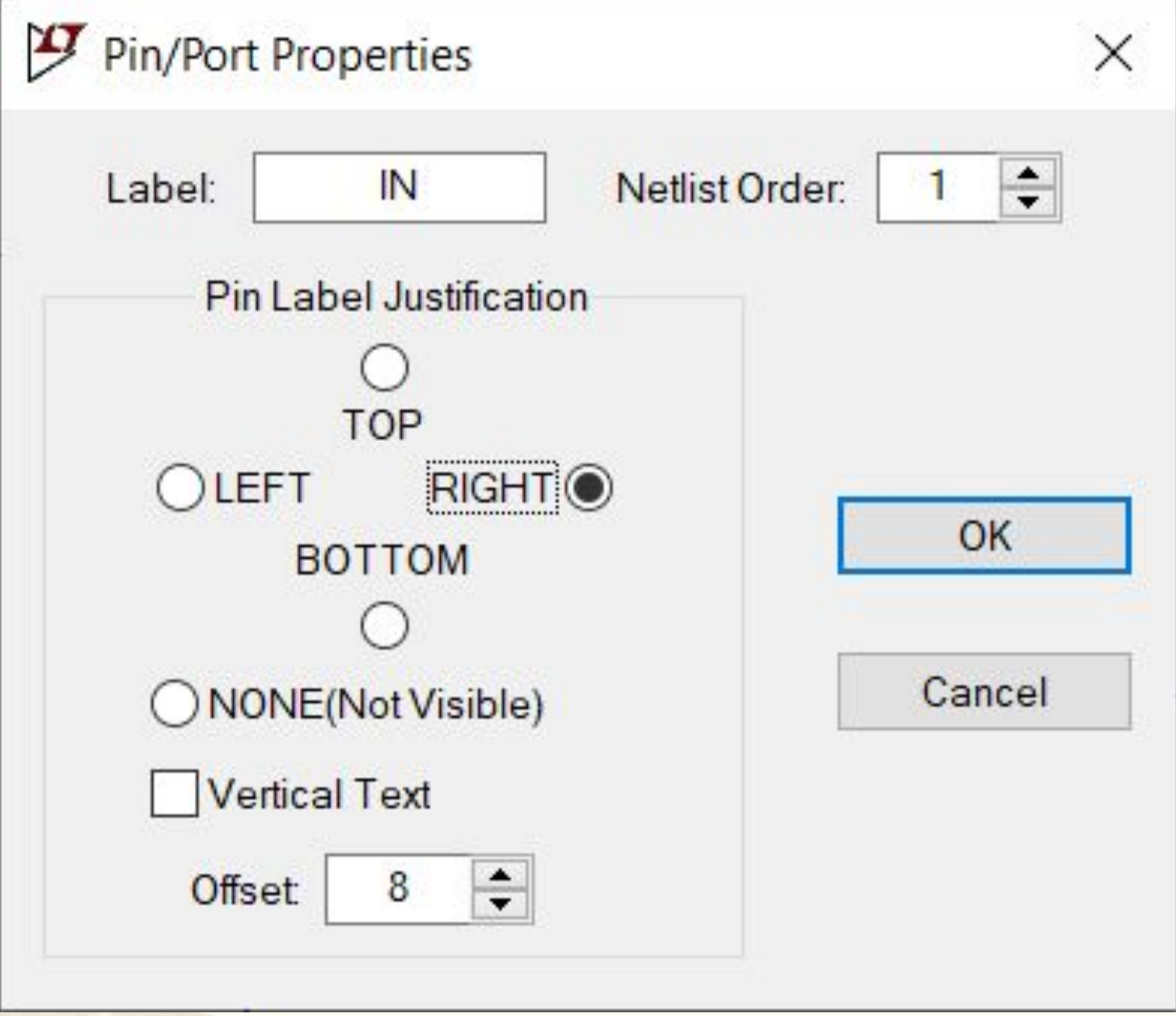## First, we need to define the Net Labels of the signals that will be exposed.

Save it as ASC file.

Now, let's create a symbol.

Use the Draw menu to add the lines to create your symbol.

In the Edit you have the Add Pin menu. You need respect the same Net Label that you define.
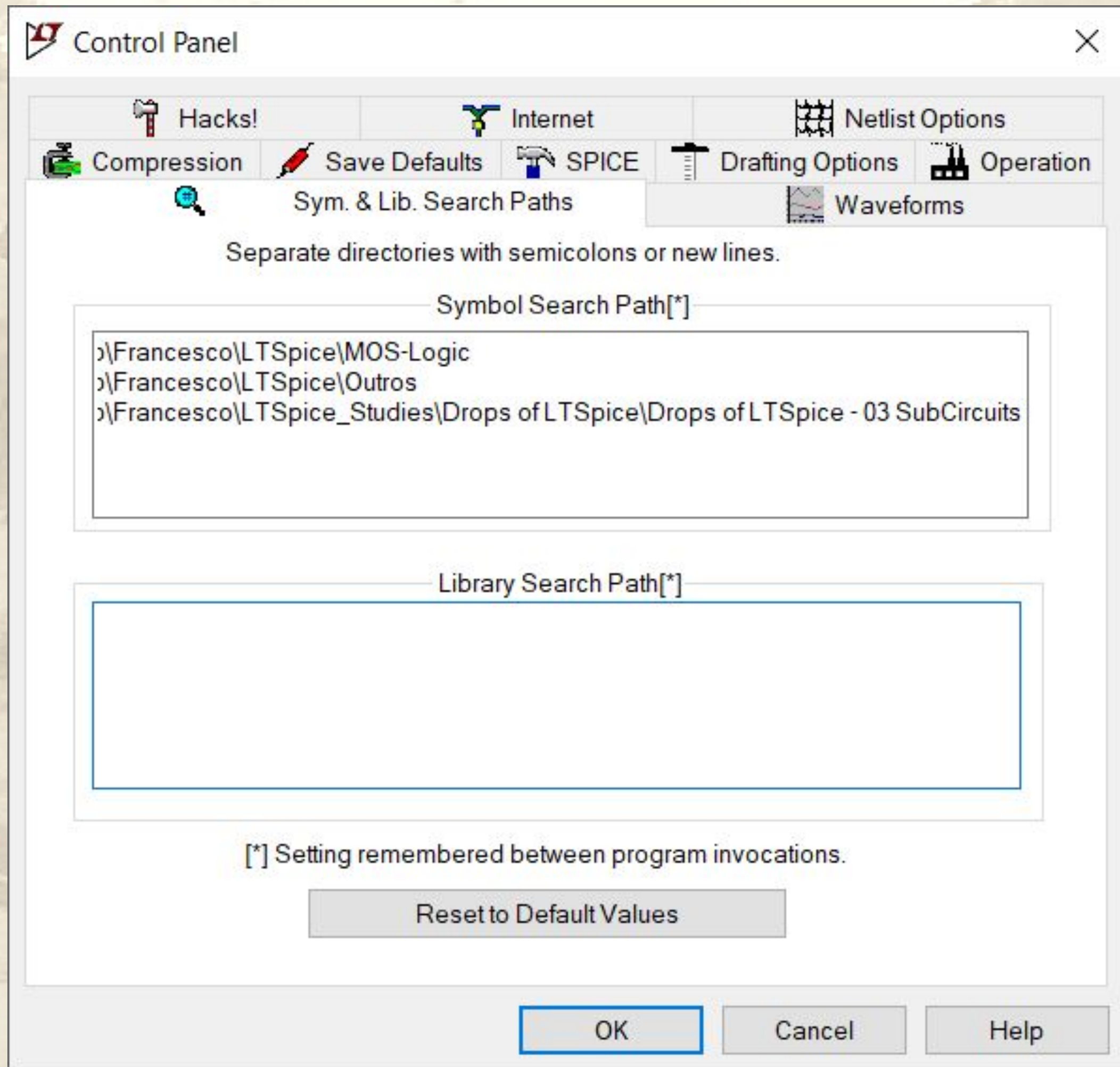
Save the ASY file with the same name. Your component is done!
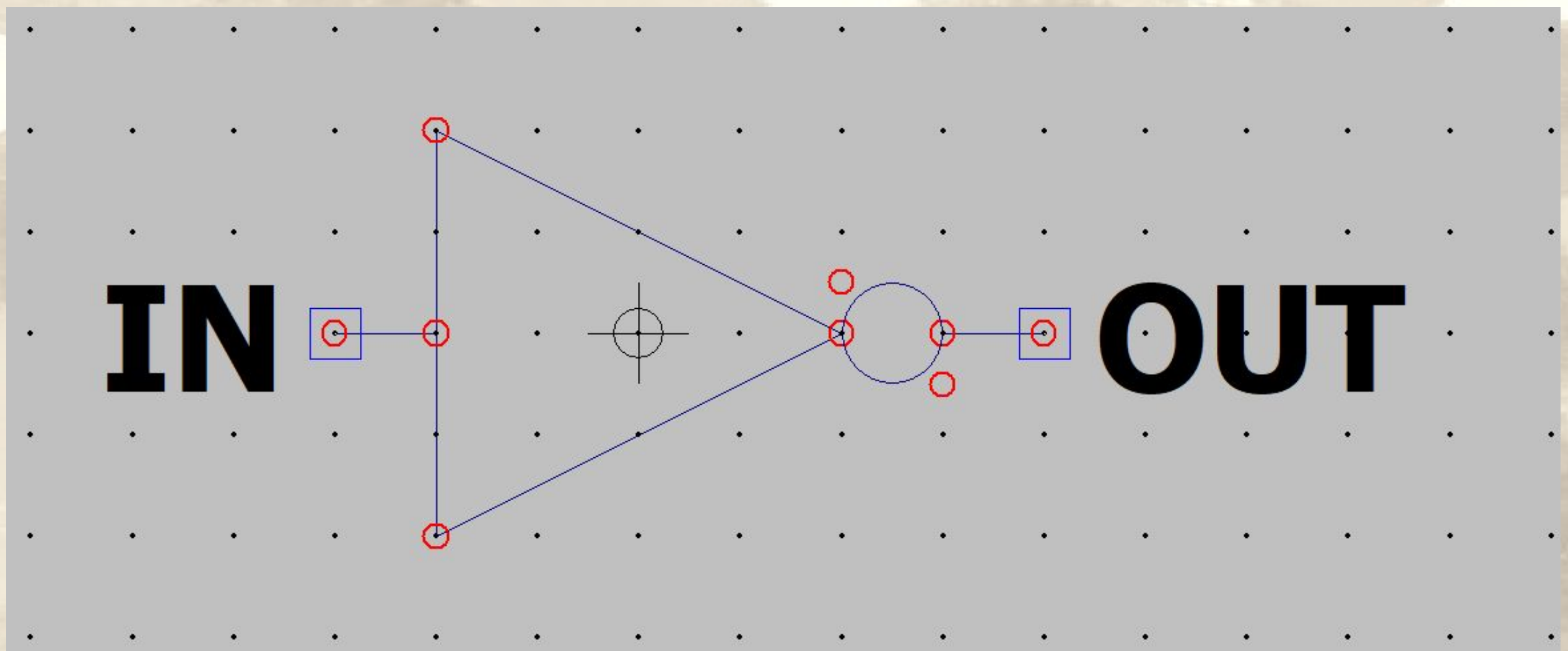
Now you need add the path to the files in the Control Panel.

Control Panel

Hacks!	Internet	Netlist Options
Compression	Save Defaults	SPICE	Drafting Options	Operation

Sym. & Lib. Search Paths | Waveforms

Separate directories with semicolons or new lines.

Symbol Search Path[*]

```
)\Francesco\LTSpice\MOS-Logic
)\Francesco\LTSpice\Outros
)\Francesco\LTSpice_Studies\Drops of LTSpice\Drops of LTSpice - 03 SubCircuits
```

Library Search Path[*]

[*] Setting remembered between program invocations.

Reset to Default Values

OK   Cancel   Help

Now, you can add your new component in the project. Notice that you need select the directory in the list.
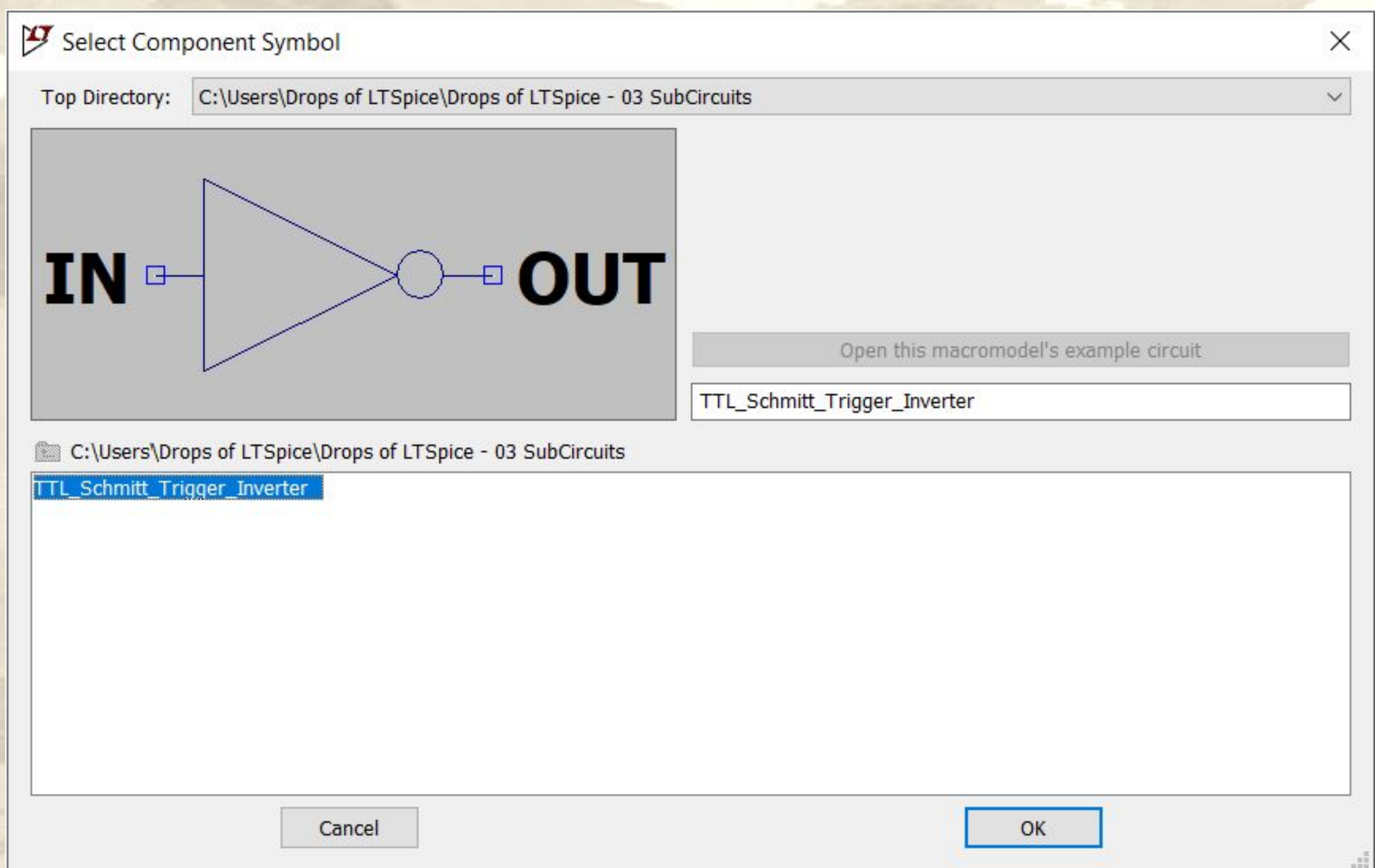
And look how simple is your simulation now!
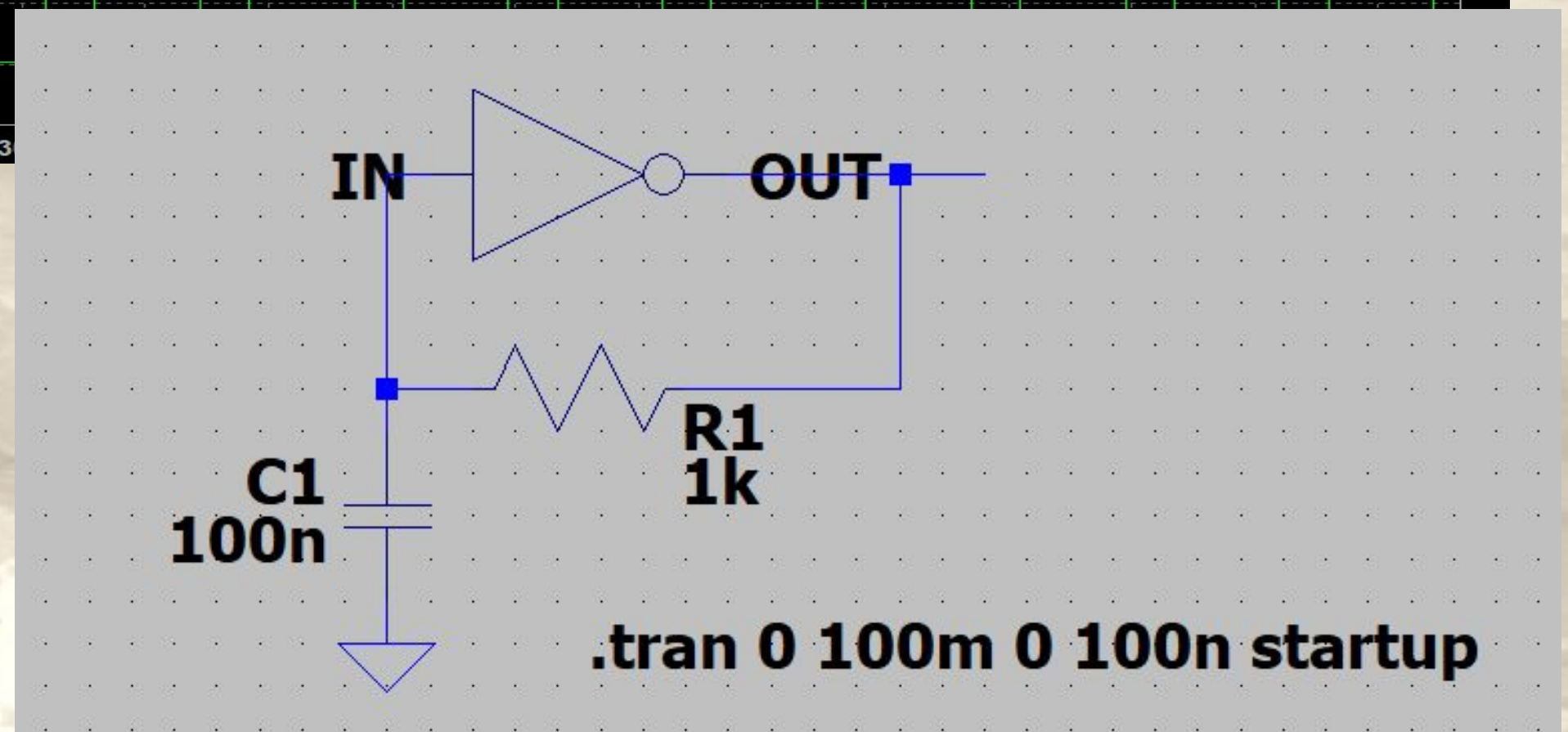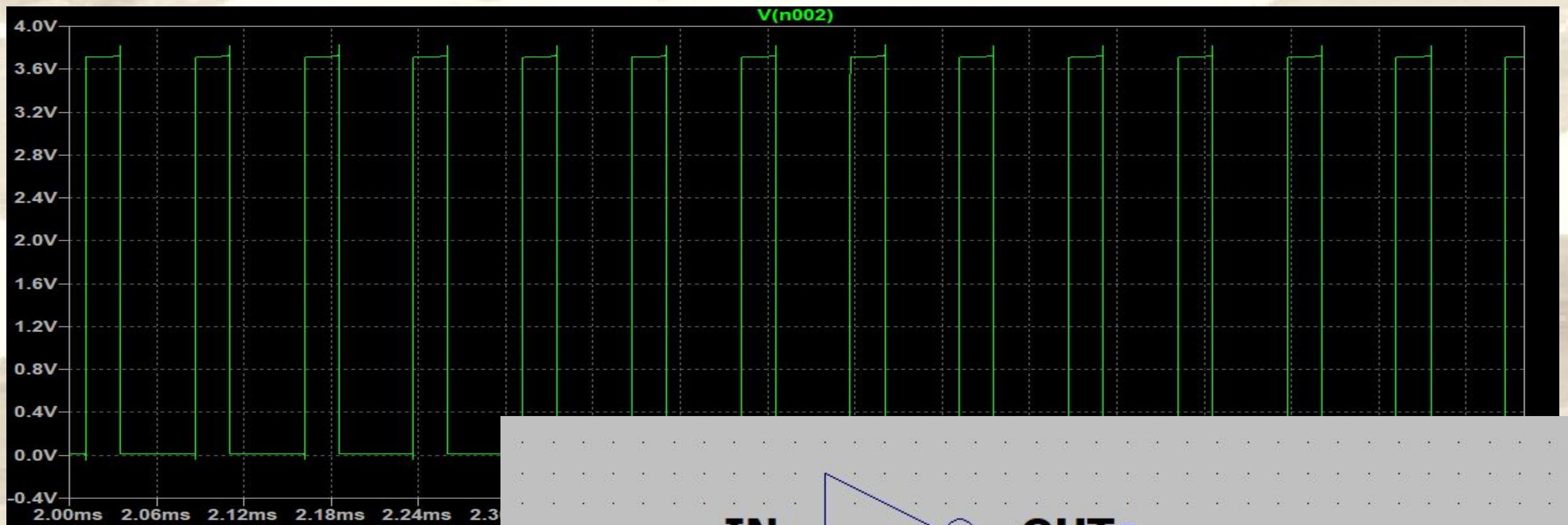
Francesco Sacco
linkedin.com/in/saccofrancesco