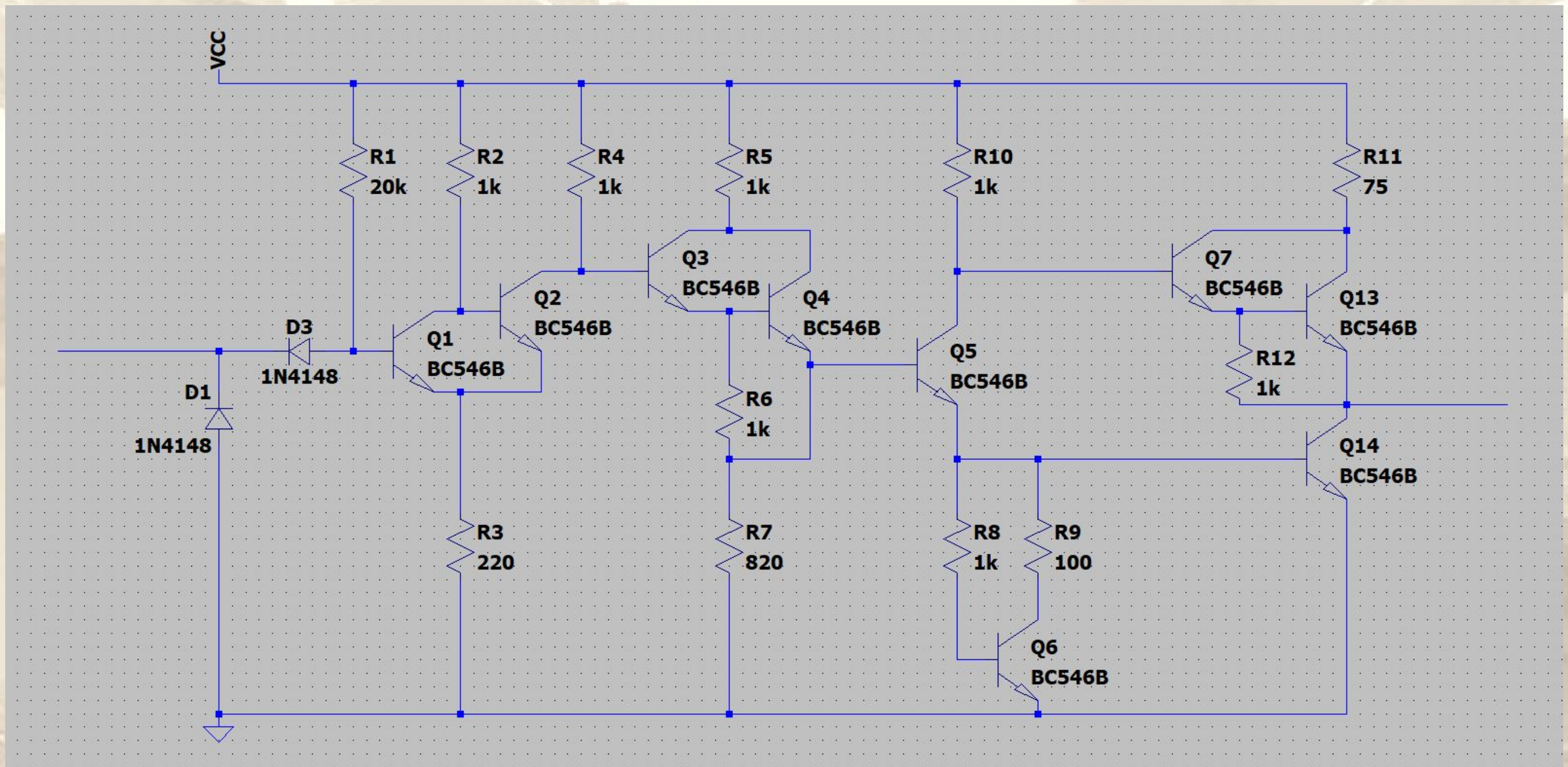


Drops of LTSpice



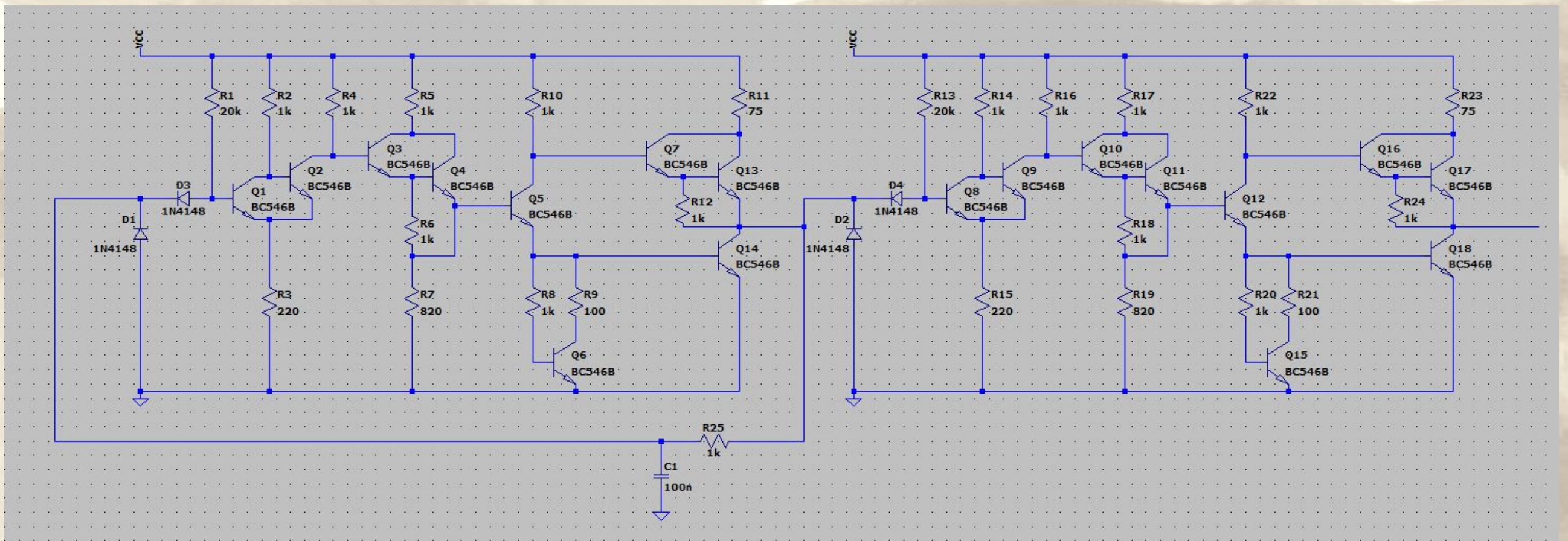
How to create subcircuits?

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



And you want to use it as a component.

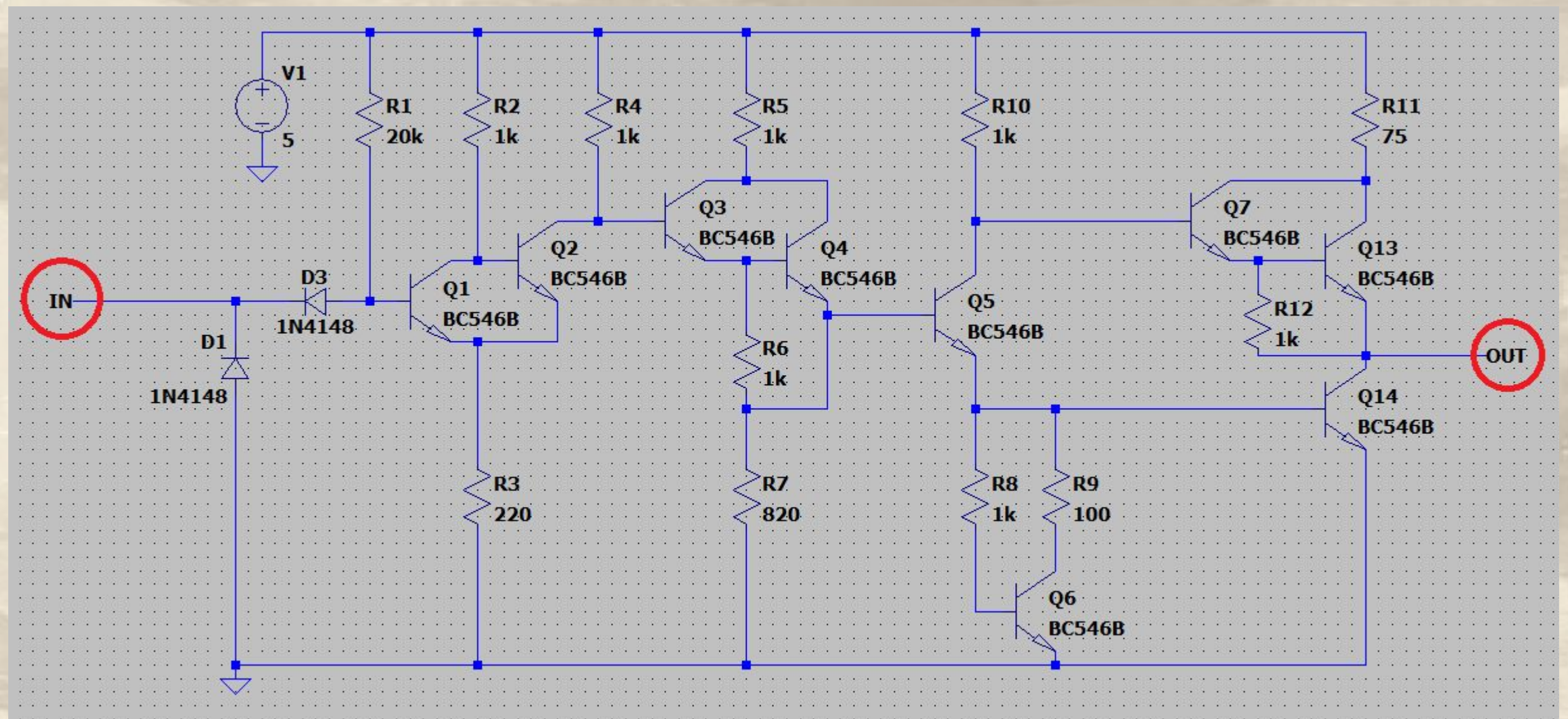
However, a bigger circuit like this can be complex to use or analyze.



Can we simplify?

Of course we can!

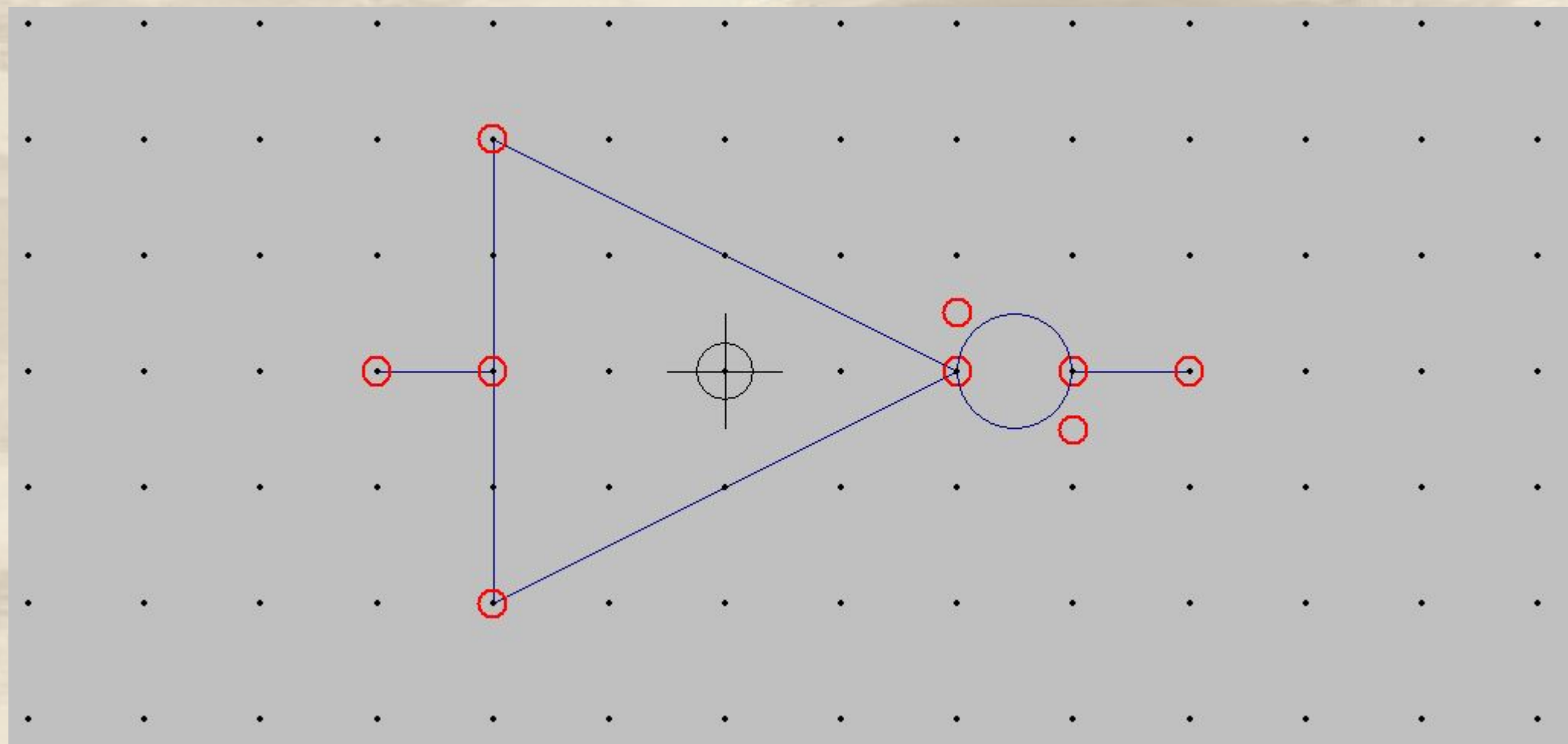
First, we need to define the Net Labels of the signals that will be exposed.



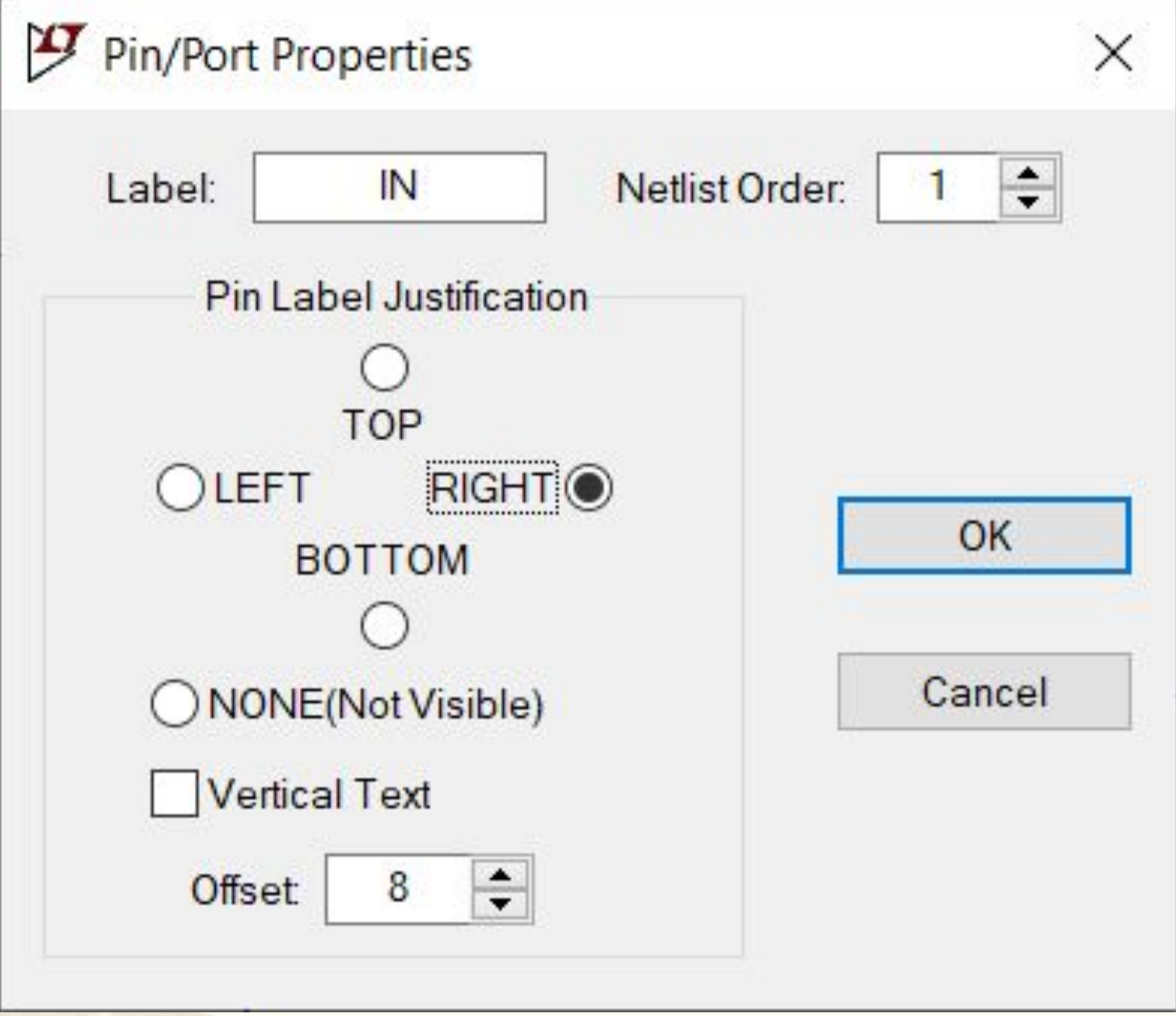
Save it as ASC file.

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.



The image shows a 'Pin/Port Properties' dialog box with a title bar containing a red lightning bolt icon and a close button. The dialog is divided into several sections. At the top, there is a 'Label' text box containing 'IN' and a 'Netlist Order' spinner box set to '1'. Below these is a 'Pin Label Justification' section containing five radio button options: 'TOP', 'LEFT', 'RIGHT' (which is selected and has a dotted border), 'BOTTOM', and 'NONE(Not Visible)'. Below the justification options is a 'Vertical Text' checkbox, which is currently unchecked. At the bottom of the dialog is an 'Offset' spinner box set to '8'. On the right side of the dialog, there are two buttons: 'OK' and 'Cancel'.

Pin/Port Properties

Label: IN Netlist Order: 1

Pin Label Justification

☐ TOP

☐ LEFT ☒ RIGHT

☐ BOTTOM

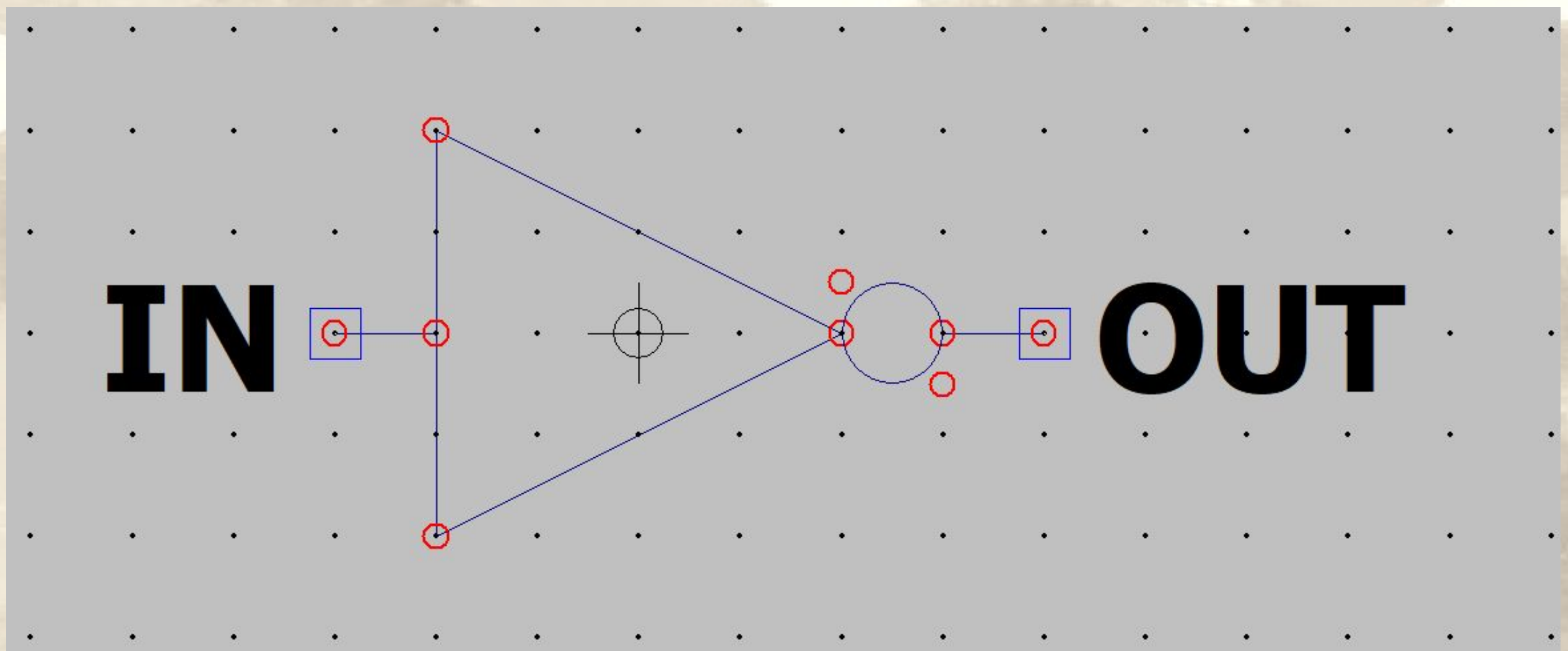
☐ NONE(Not Visible)

☐ Vertical Text

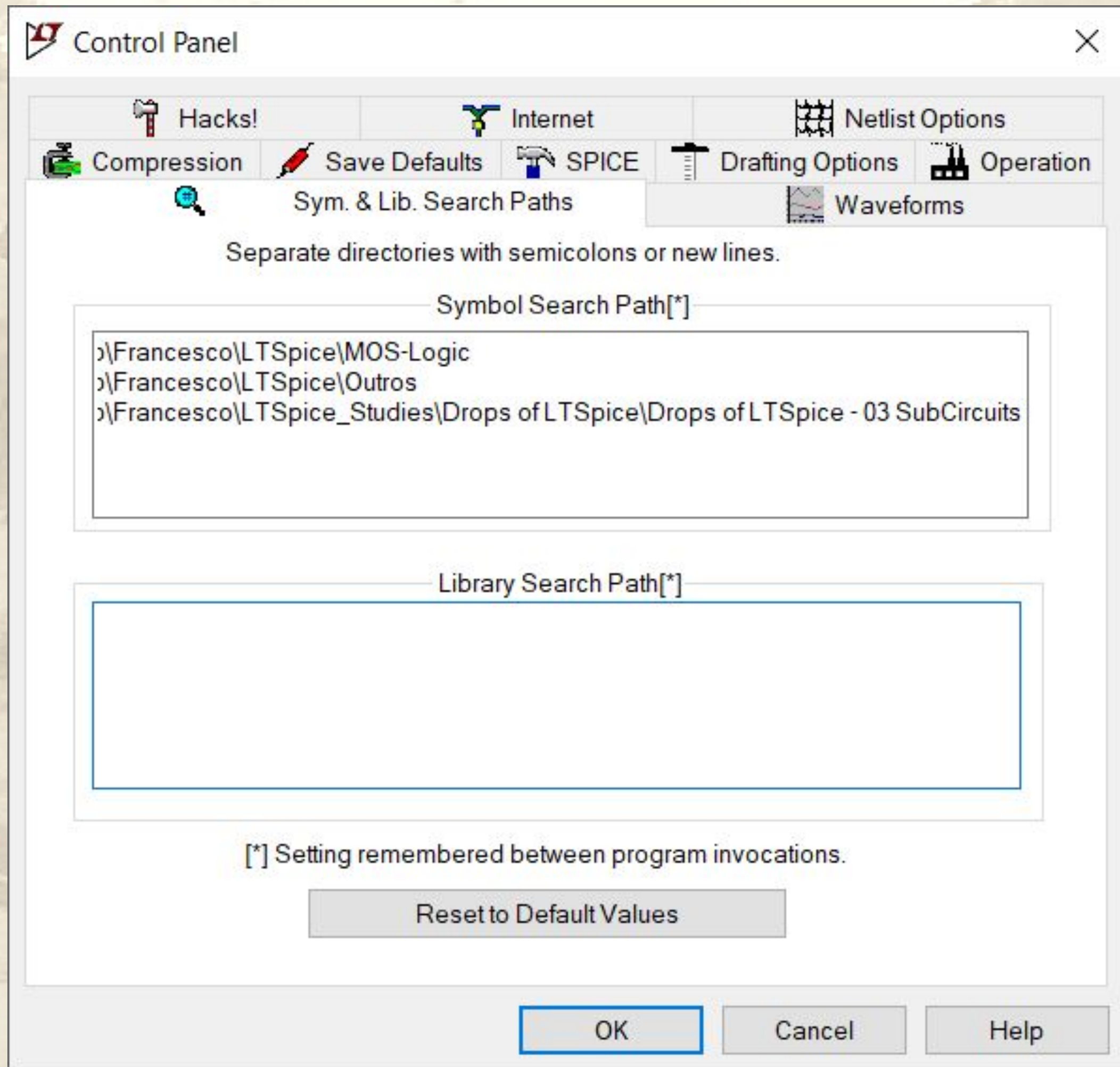
Offset: 8

OK Cancel

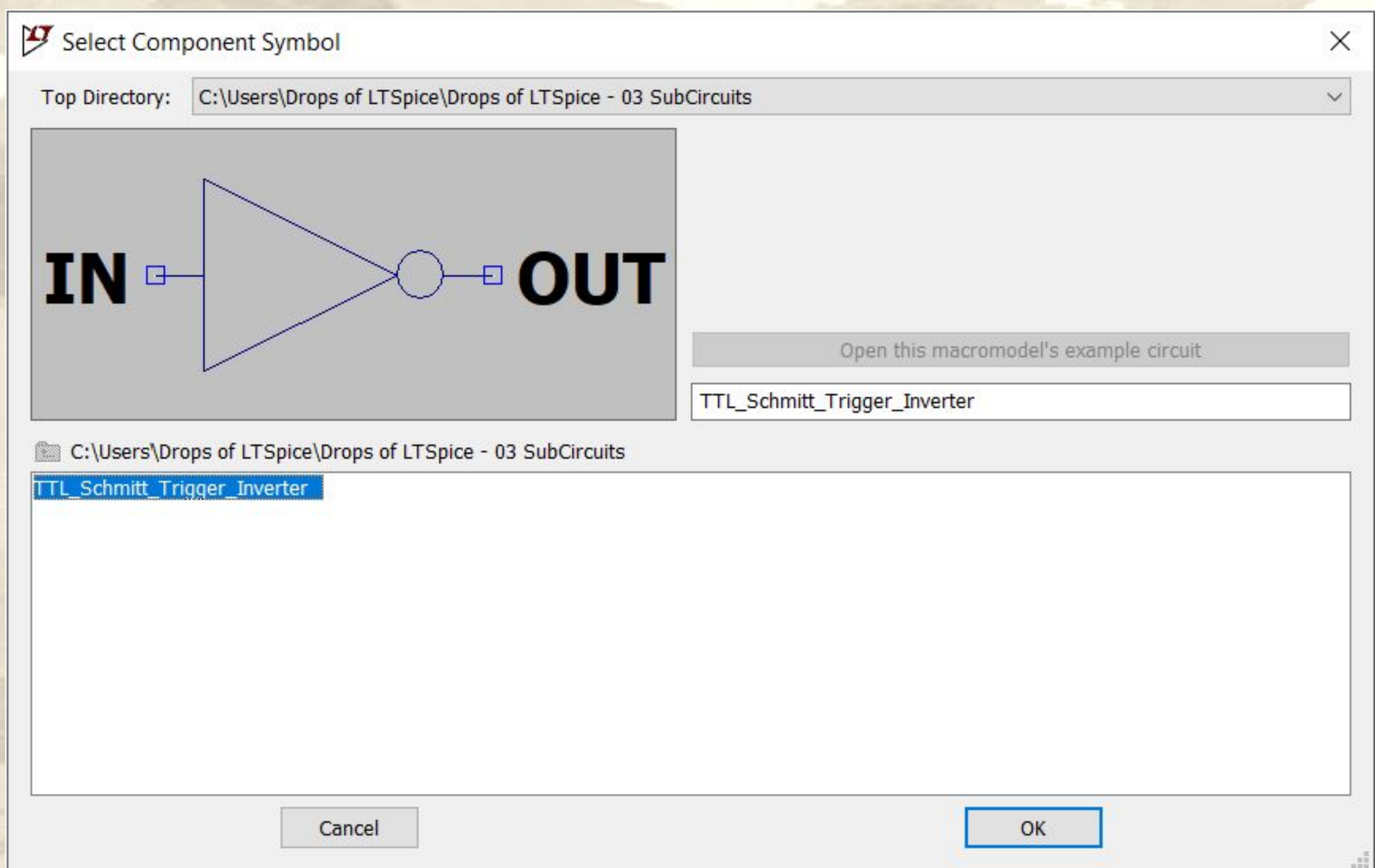
Save the ASY file with the same name. Your component is done!



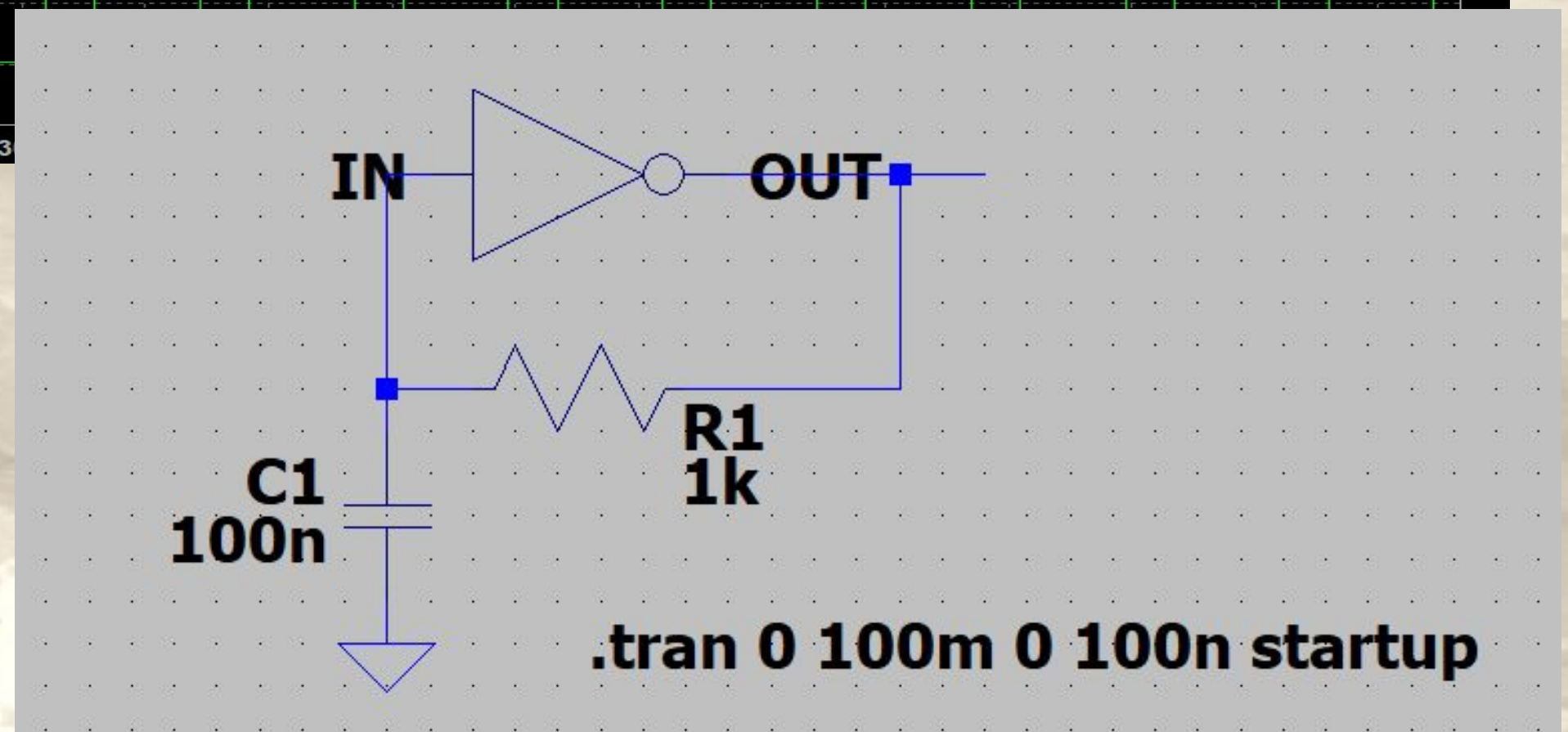
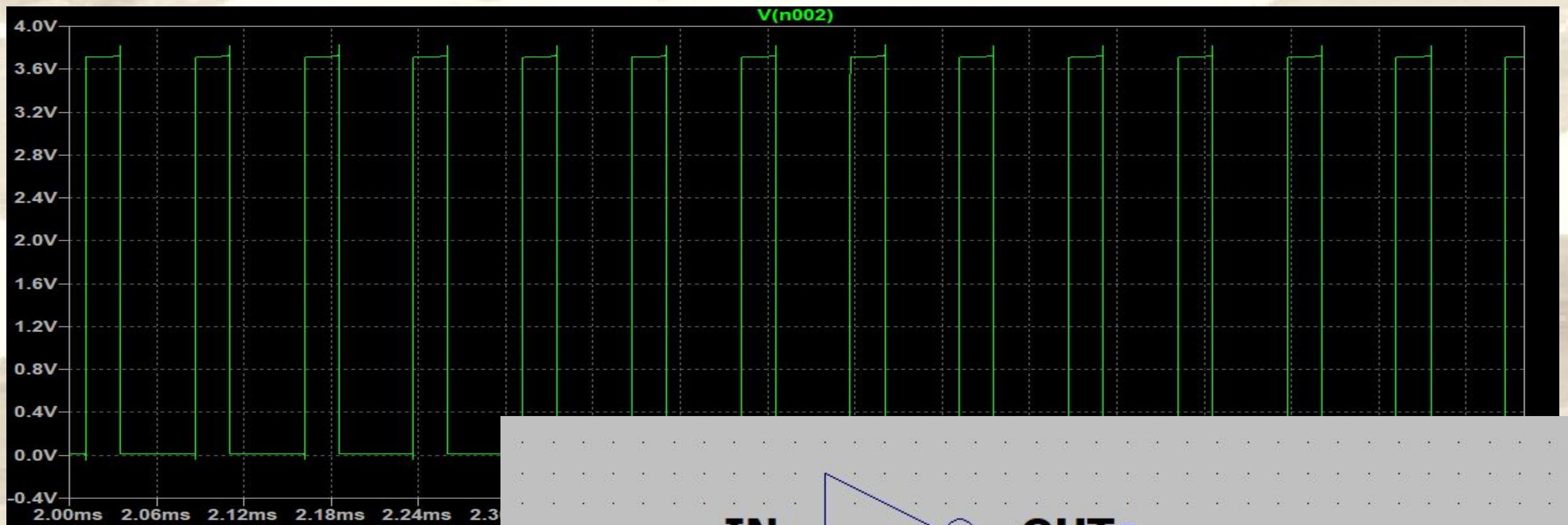
Now you need add the path to the files in the Control Panel.



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