

Instructions to Simulate the Ring Oscillator in 5Spice

The aim of this document is to give a short tutorial to simulate TTL based ring oscillators. Each step will be analyzed in details.

Step 1: Adding your subcircuit into your library

You will be supplied with the regarding subcircuit file called "ttlinv.fam". Place it under the directory of

"C:\Program Files\5Spice Analysis\Library\SubCircuits\AnalogDev "

as an example. Then from Tools menu, Rebuild Spice Library. Now our subcircuit will be added.

Step 2: Defining your Subcircuit

Start by insert a dummy subcircuit into your schematic by selecting the "Subcircuit, 2-100 pins" icon in rightmost column (under the op-amp like shaped icon menu). Then similar to what you have already done while inserting different models for your transistors, select "TTLINVERTER" as shown in Figure 1.

Step3: Build your Circuit

For starters, you may build a similar circuit as given in Figure 2. Here, pay attention to the port definitions of our subcircuit.

Port 1 => Input Signal Port 2 => Power (Vcc) Port 3 => Output Port 4 => Ground

Step4: Initial Conditions

To start the ringing oscillations, we should set some of our nodes to prespecified voltages. In our case, let us set the initial voltage of our output node to 5Volts. To do this, put an Initial Conditions (IC) flag to the regarding node. The IC flag is under the TP_y menu at the rightmost. Edit the flag such that it has 5Volts at t=0 as given in Figure 3. While plotting the transient response of the circuit, do not forget to check the initial conditions box. This is a critical step.

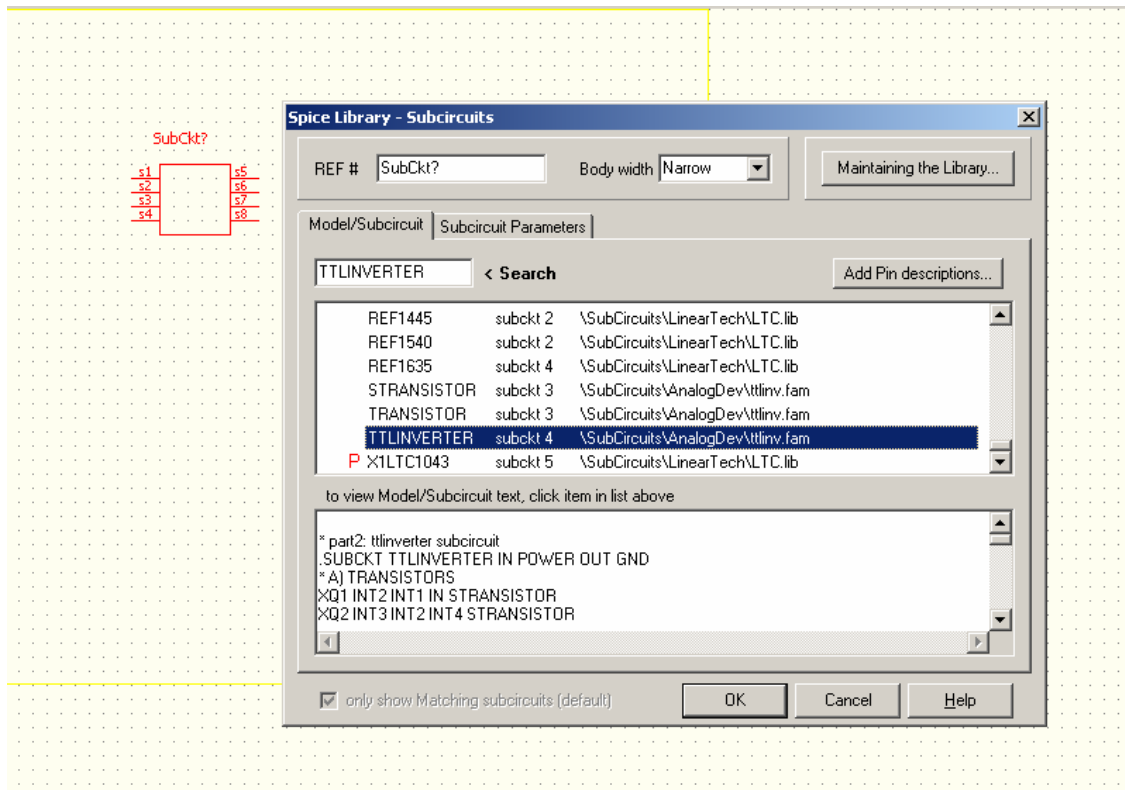


Figure 1

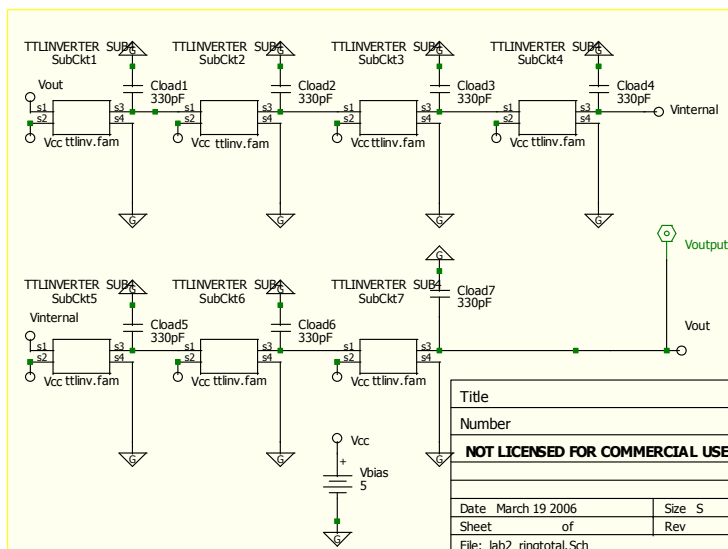


Figure 2

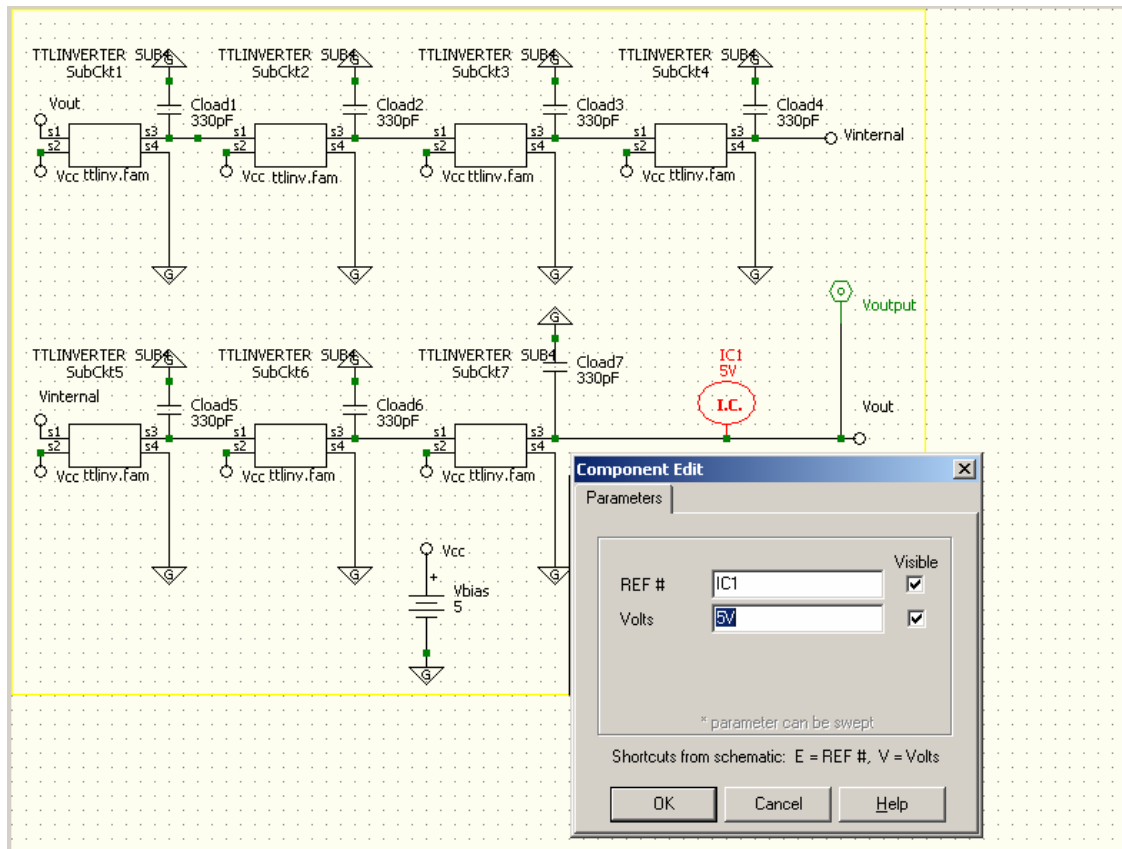


Figure 3

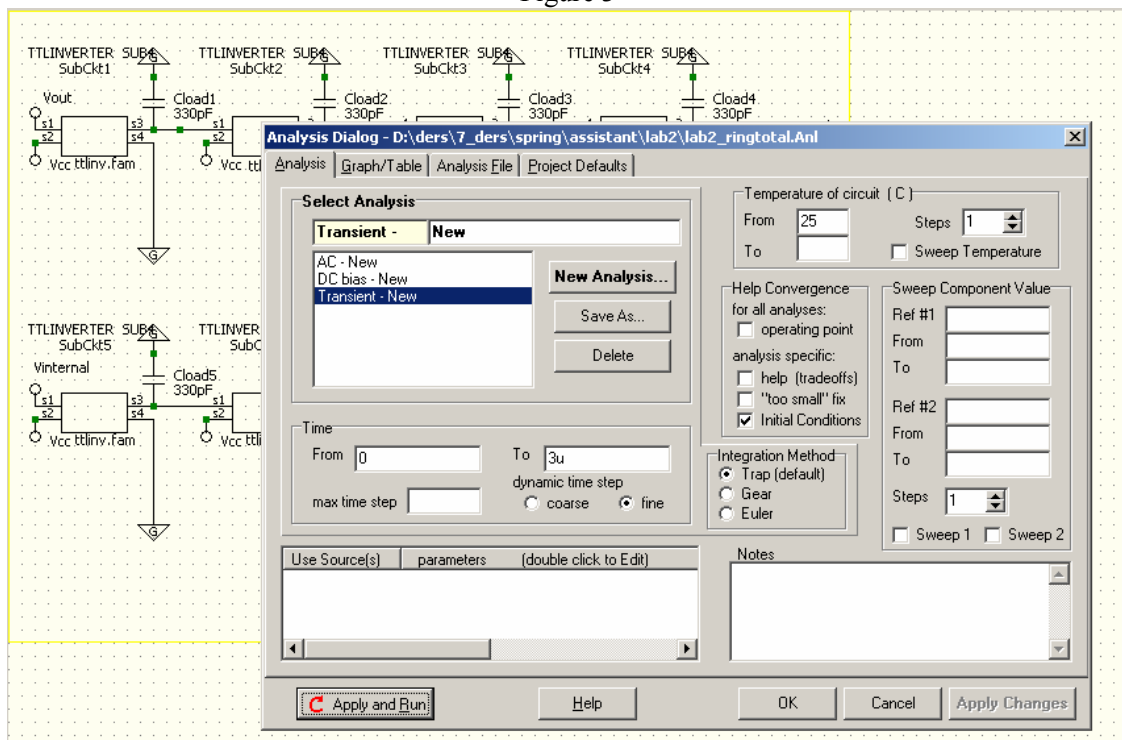


Figure 4