Setting Initial Conditions for HSPICE Simulations

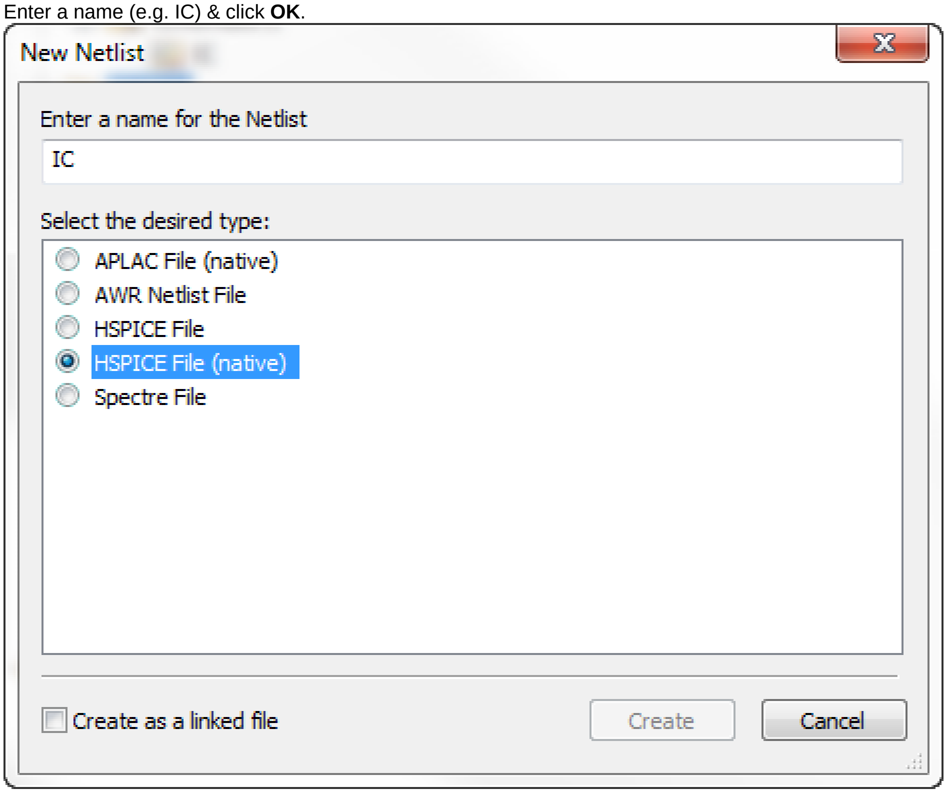
Problem

I need to set the initial HSPICE simulation voltage (voltage at time=0) for one or more nodes in the circuit

Solution

The native HSPICE netlist model "IC" sets the initial (time=0) voltage of a node to the desired value for HSPICE transient simulations. Step by step instructions:

1. Right-click **Netlists**
2. Select **New Netlist**
3. Check the box for **HSPICE file (native)**.
4. Enter a name (e.g. IC) & click **OK**.

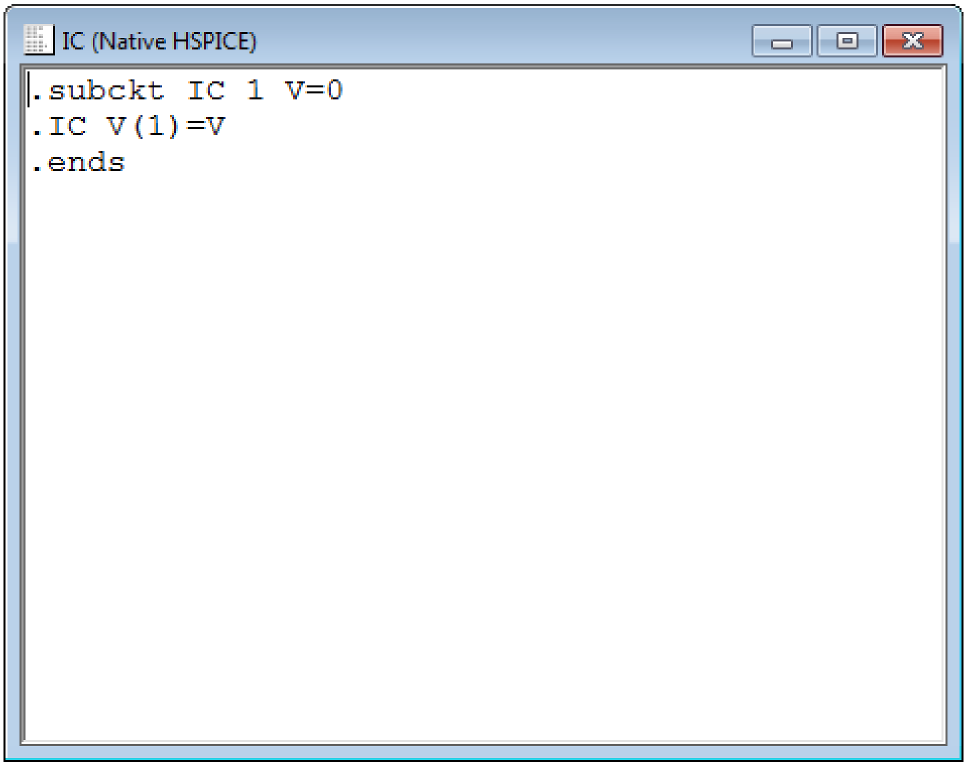


5. Copy the text of the HSPICE netlist in this project into the window

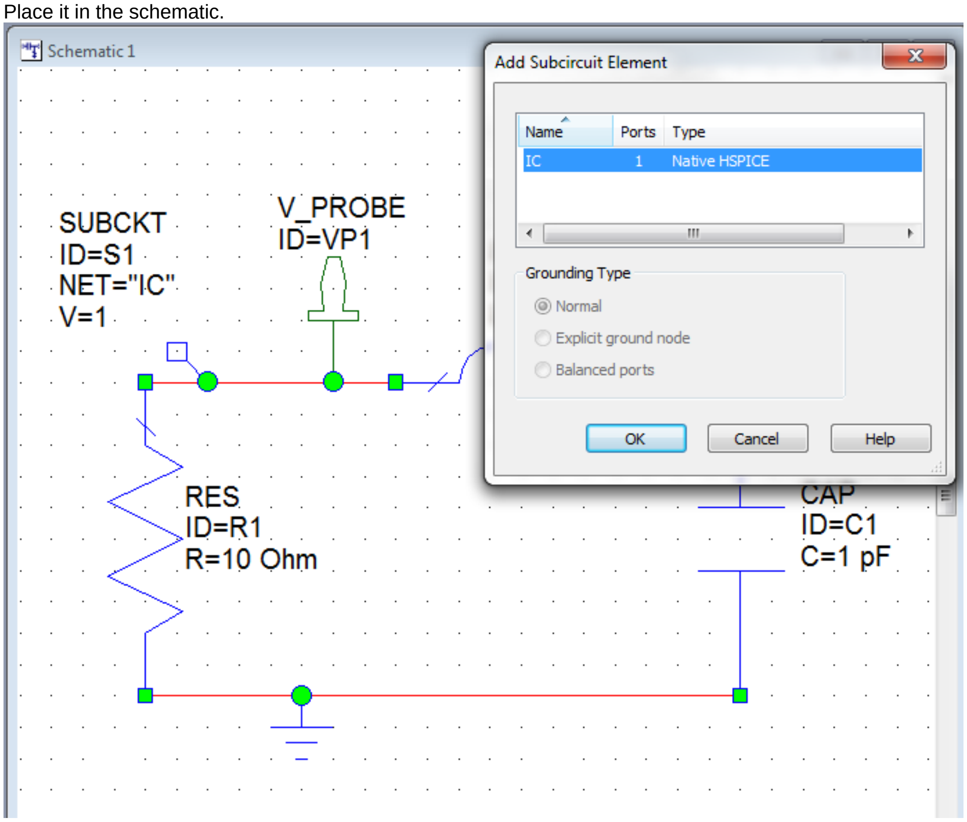
```
.subckt IC 1 V=0

.IC V(1)=V

.ends
```

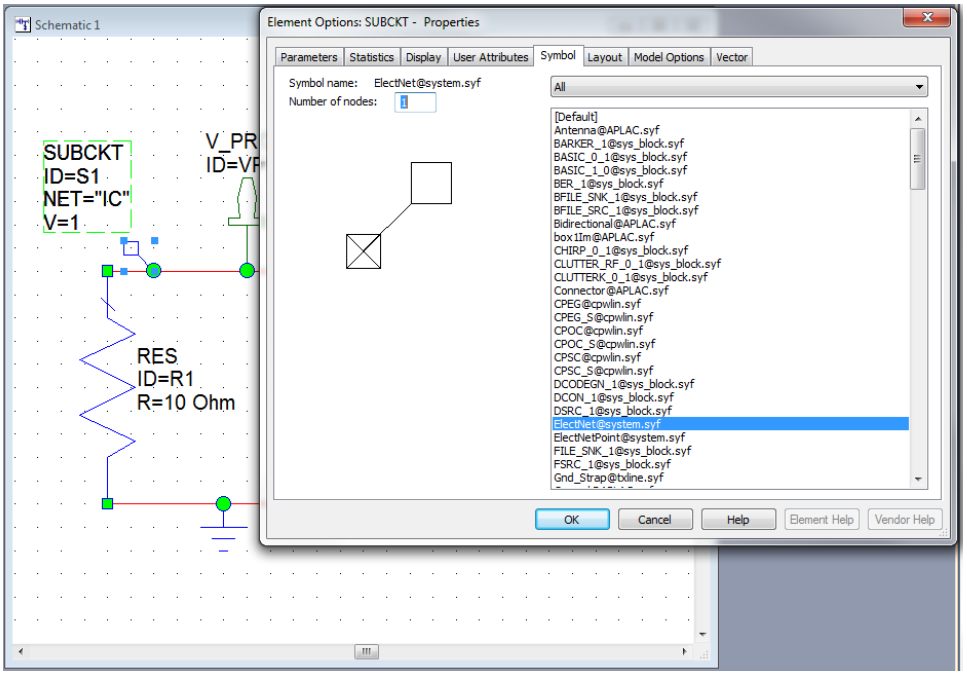


6. Close the window.
7. Click on the schematic window
8. Select **Draw > Add Subcircuit**
9. Select your IC model.
10. Place it in the schematic.



11. Double click it.
12. Select the **Symbol** tab.

13. You can use the default or the custom symbol. In this example we choose **ElectNet@system.syf** symbol because it's relatively smaller than the others



14. Click **OK**.
15. Connect to the desired node.
16. Set the **V** parameter to the desired voltage.
17. This can be simulated with HSPICE only so use **HSPICE Trans** as the simulator