PSPICE Tutorial

Outline

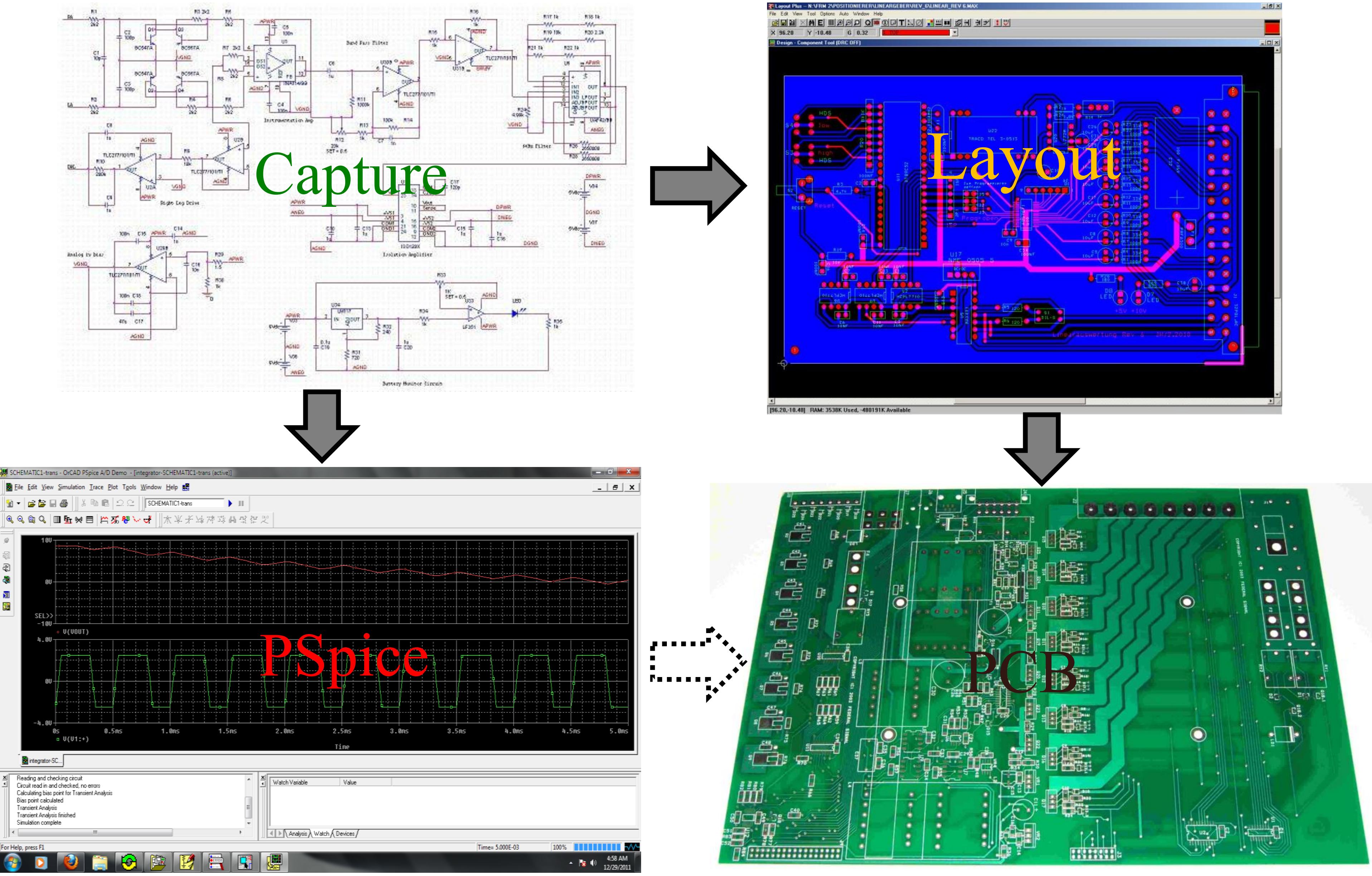
- Introduction
- Installation
- Prepare a circuit for simulation
- Simulation using PSPICE
- A typical example

Before we begin

- Reference: https://engineering.purdue.edu/~ee255d3/readings.html

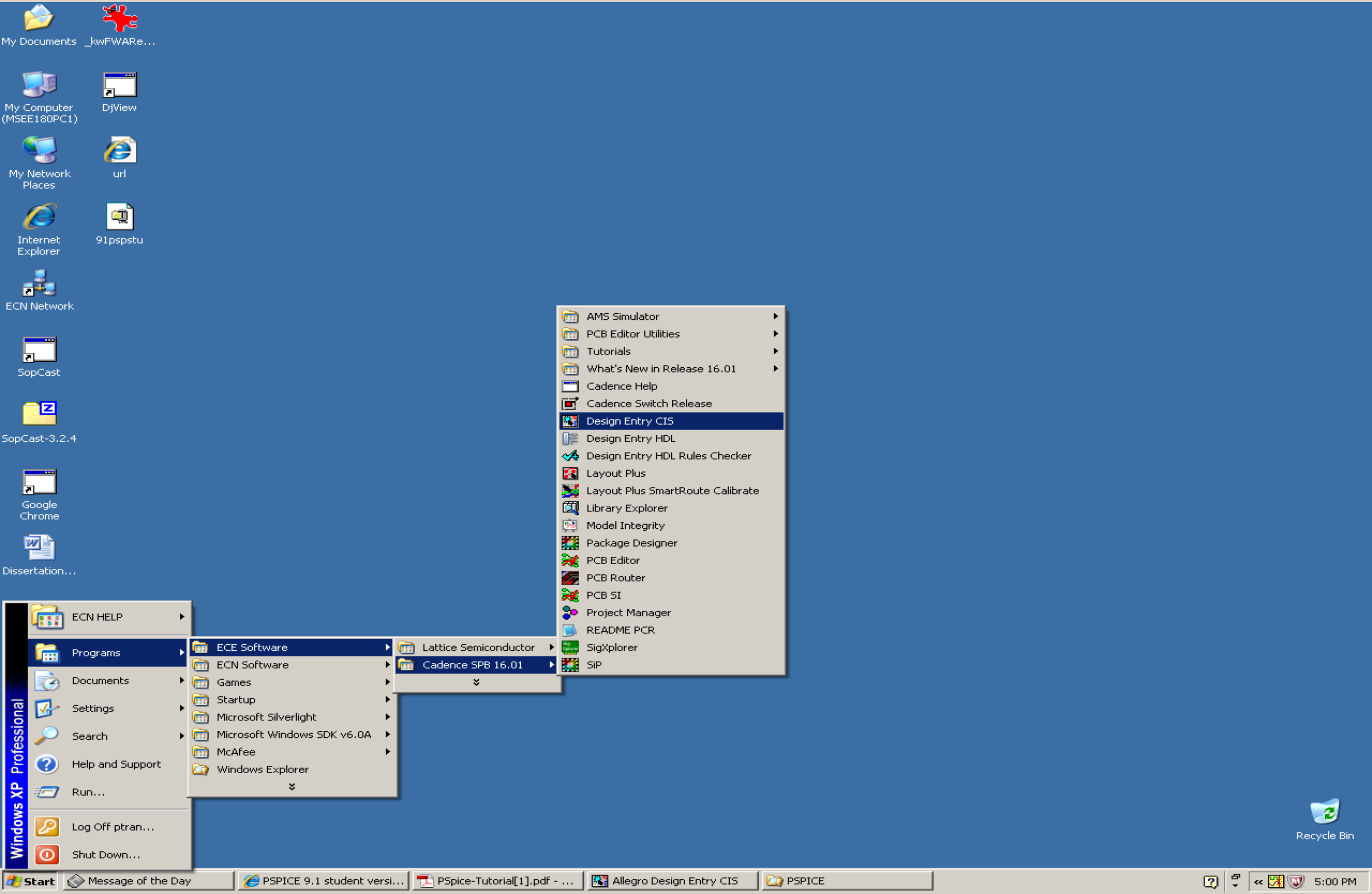
Introduction

- SPICE – Simulation Program with Integrated Circuits Emphasis
- Developed by University of California at Berkeley in 1970s.
- A simulation program that models the behavior of a circuit containing analog or mixed A/D devices, used to test and refine your design before implementing on hardware (PCB).
- PSPICE is the most prominent commercial version of SPICE, initially developed by MicroSim (1984), but now owned by Cadence Design System. Pspice is now a component of the OrCAD® Product Family (including Capture CIS, PCB Editor, Pspice, Layout Plus ...)


Capture
Layout
PSpice
PCB

Installation

- Almost every computers in ECN labs are equipped with the standard version of PSPICE, a product of Cadence.
- You can also download the PSPICE student version here.
- To install the student version: Unzip the downloaded file, run setup.exe and follow the instructions on the screen. *Note: you should close all other Windows programs (especially web browsers) before installing PSPICE.*
- To run PSPICE on ECN computers: go to Start > Programs > ECN Software > Cadence SPB 16.01 > Design Entry CIS.


My Documents
_kwFWARe...
My Computer (MSEE180PC1)
DjView
My Network Places
url
Internet Explorer
91pspstu
ECN Network
SopCast
SopCast-3.2.4
Google Chrome
Dissertation...
AMS Simulator
PCB Editor Utilities
Tutorials
What's New in Release 16.01
Cadence Help
Cadence Switch Release
Design Entry CIS
Design Entry HDL
Design Entry HDL Rules Checker
Layout Plus
Layout Plus SmartRoute Calibrate
Library Explorer
Model Integrity
Package Designer
PCB Editor
PCB Router
PCB SI
Project Manager
README PCR
SigXplorer
SiP
ECN HELP
Programs
Documents
Settings
Search
Help and Support
Run...
Log Off ptran...
Shut Down...
Windows XP Professional
ECE Software
ECN Software
Games
Startup
Microsoft Silverlight
Microsoft Windows SDK v6.0A
McAfee
Windows Explorer
Lattice Semiconductor
Cadence SPB 16.01
Recycle Bin
Start
Message of the Day
PSPICE 9.1 student versi...
PSpice-Tutorial[1].pdf - ...
Allegro Design Entry CIS
PSPICE
5:00 PM

Prepare a circuit for simulation

• To simulation your design, you need to provide Pspice with the following information:

1. the parts in your circuit and how they are connected → schematic
2. what analyses you want to run → simulation profile
3. and the simulation models that correspond to the parts in your circuits → part library.
4. The stimulus definitions to test with → stimulus editor

• Two ways to describe your circuit:

- By scripts (write an input file *.cir)
- By drawing schematic

(Today I mostly focus on the latter).

Simulation

- Bias point details
- DC sweep analysis
- AC sweep analysis
- Transient analysis
- Frequency response
- And more ...

Examples

- Clipper.
- Transistor amplifier

Example 1: 2-diode clipper

- Bias point.
- DC sweep analysis
- Transient analysis
- AC sweep analysis

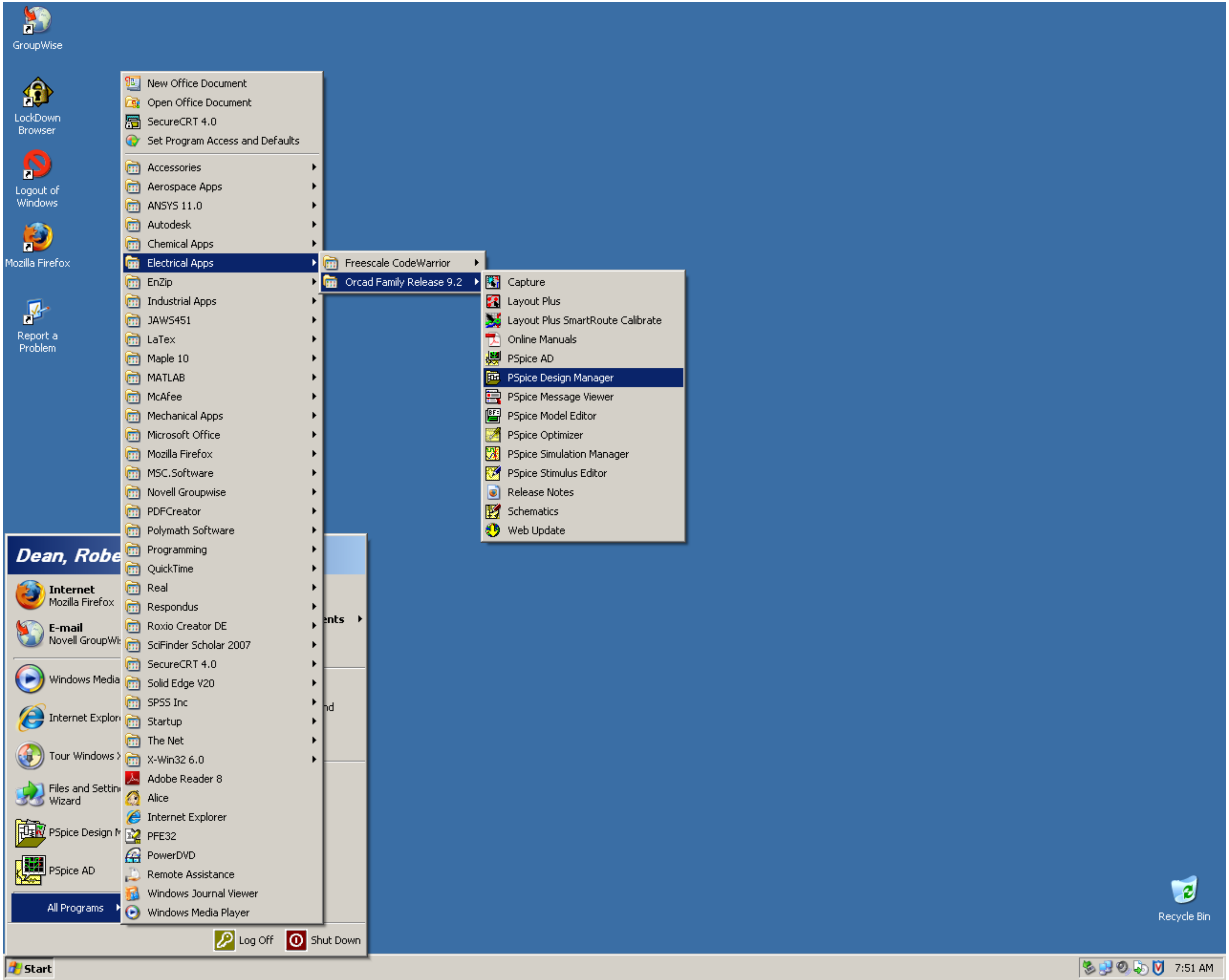
Student version

- Some features have been limited in the student version.
- There may be a little change in the user interface when creating a circuit for simulation, please refer to the slides 14-43 of this file for step-by-step simulation procedure using Pspice student version.

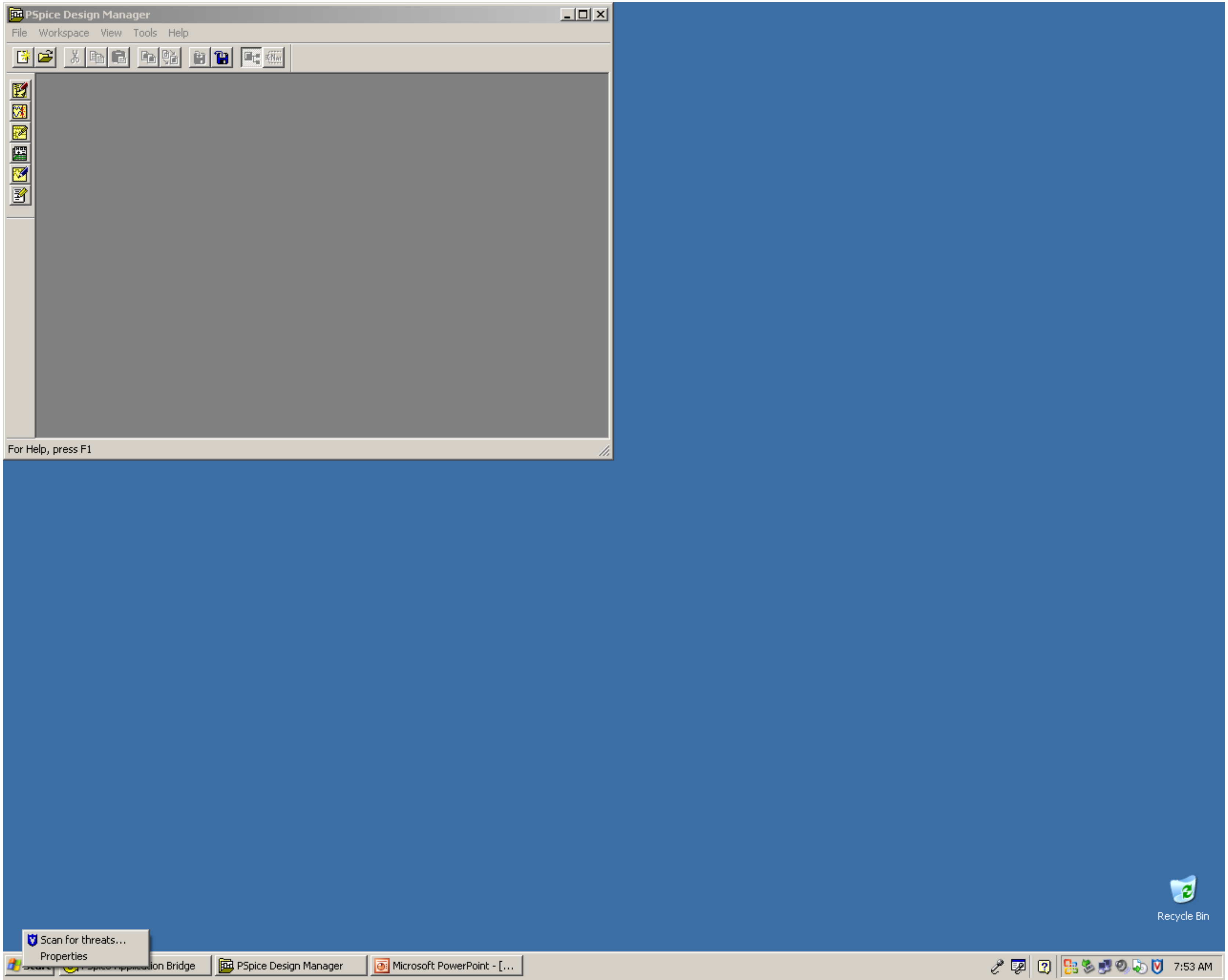
Thank You!

Question and Answer

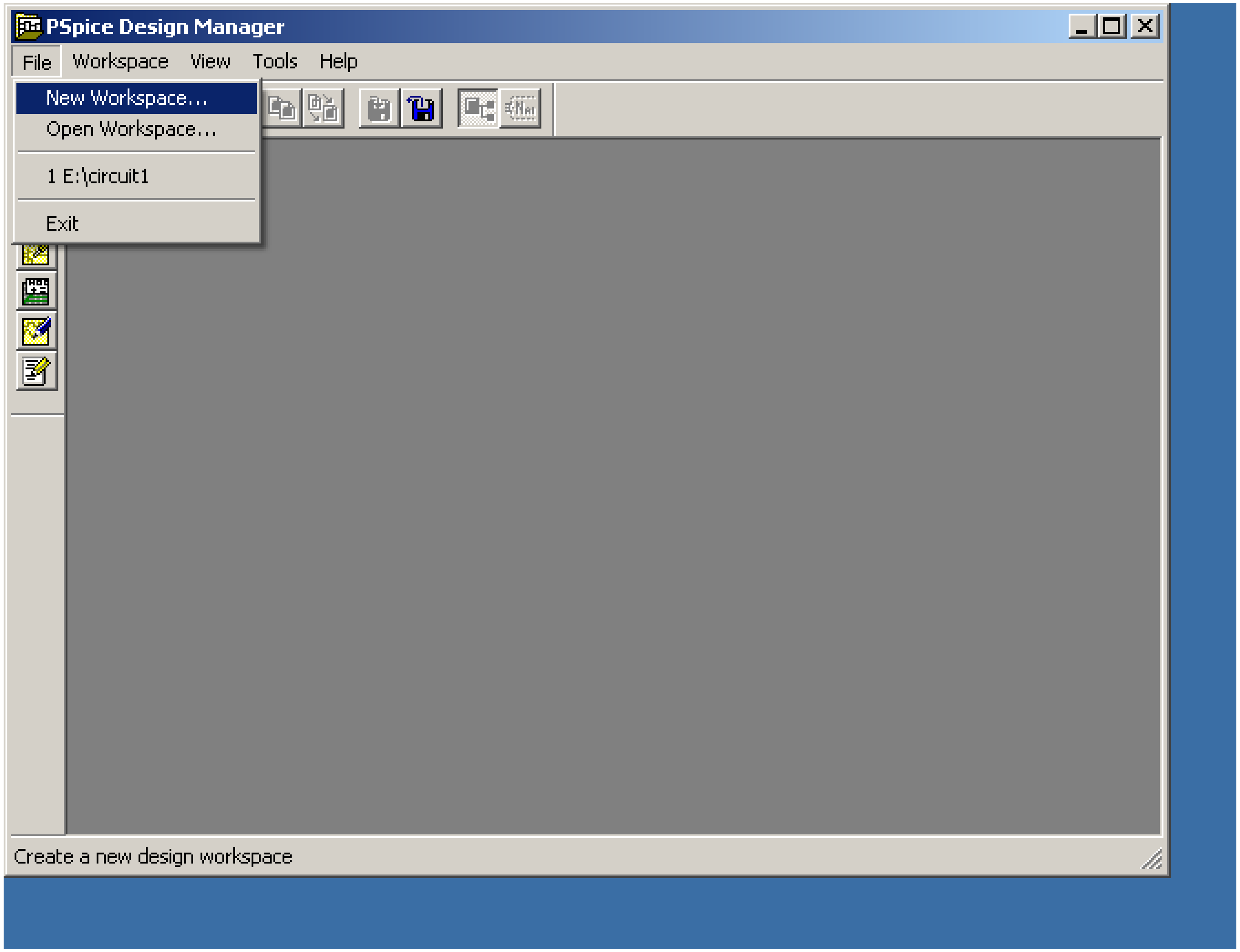
Go to: Start: All Programs: Electrical Apps: Orcad Family… : PSpice Design Manager



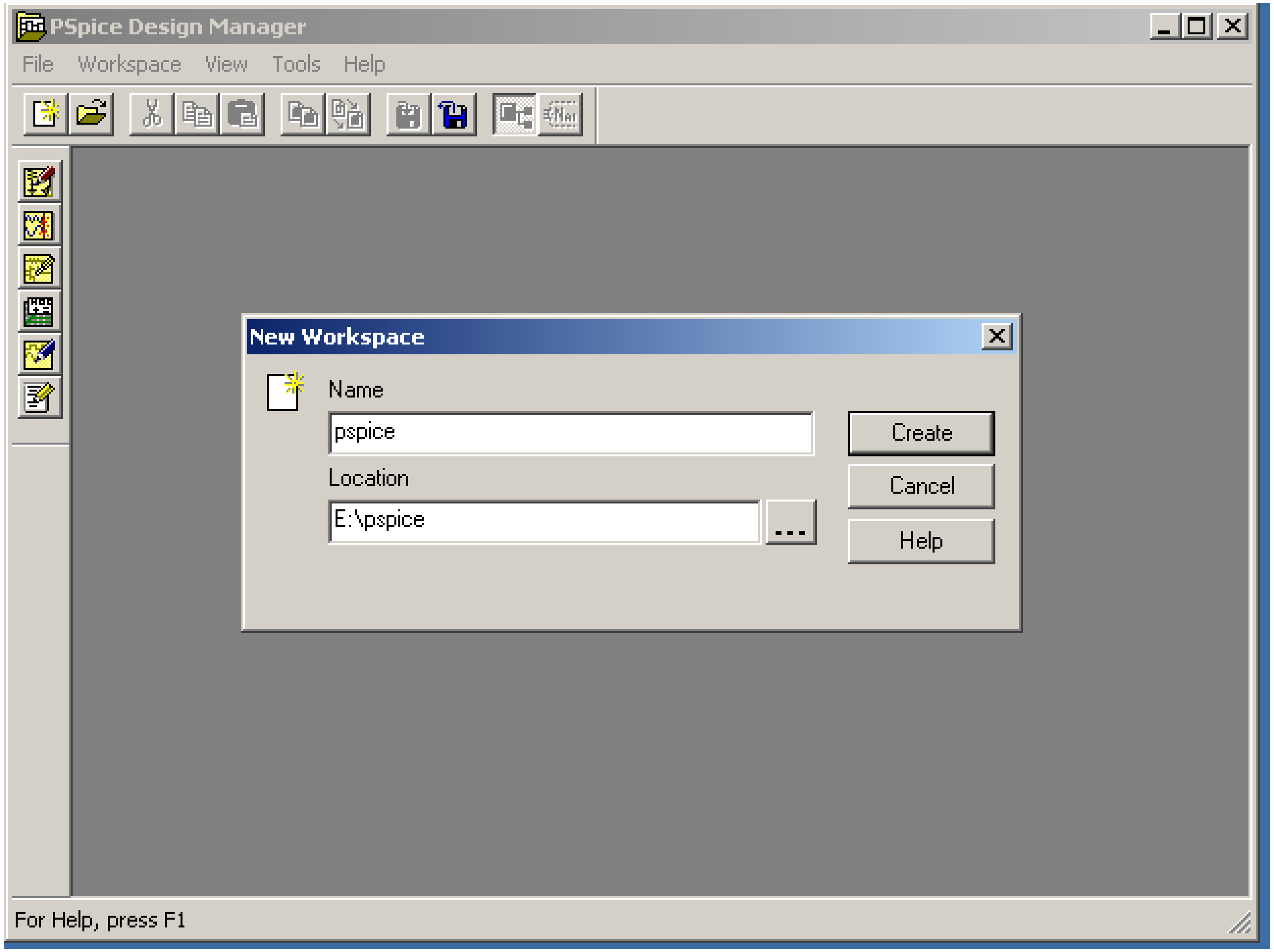
It will look like this:



Click on: File : New Workspace

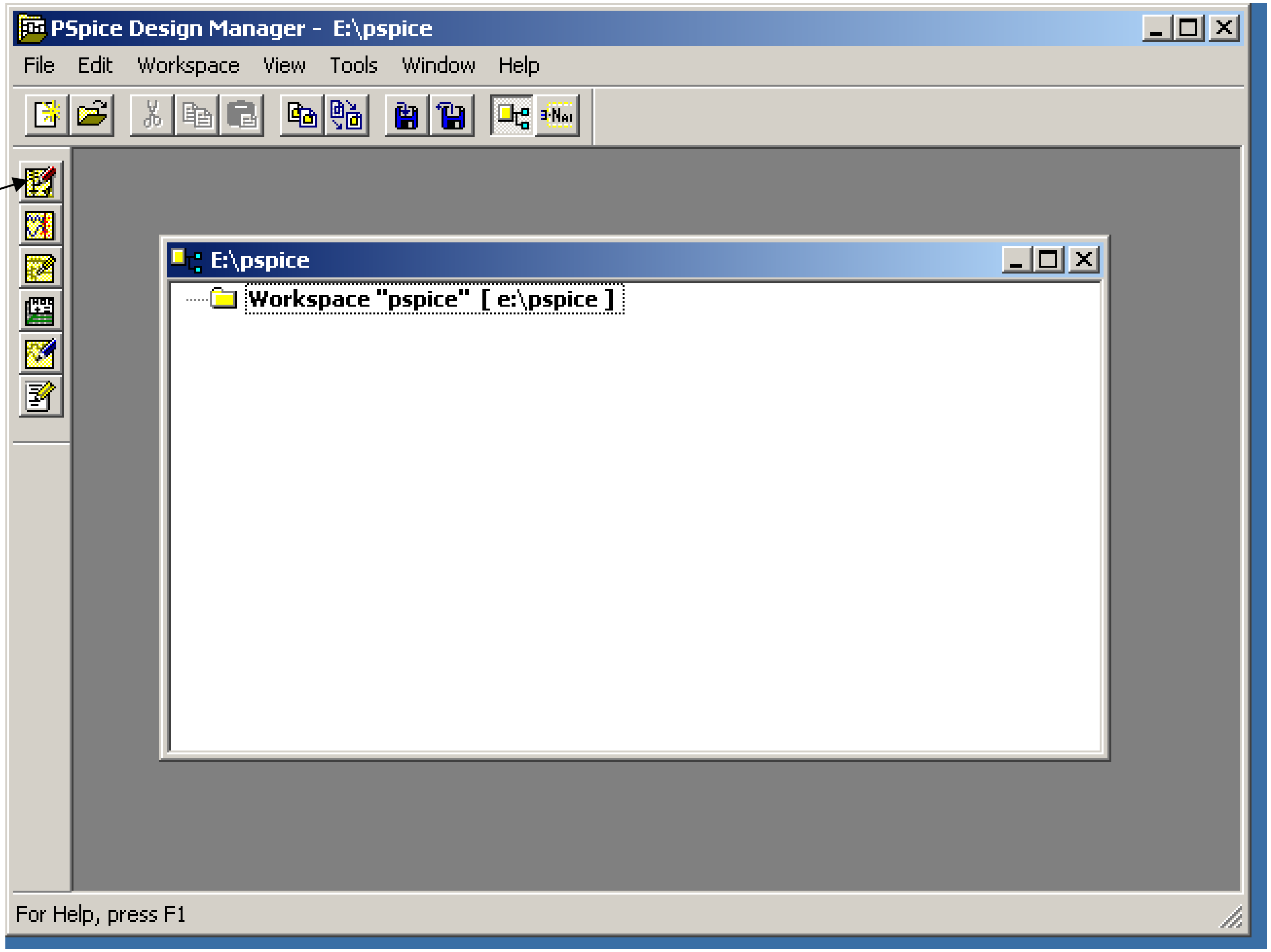


Give your New Workspace a Names and Location

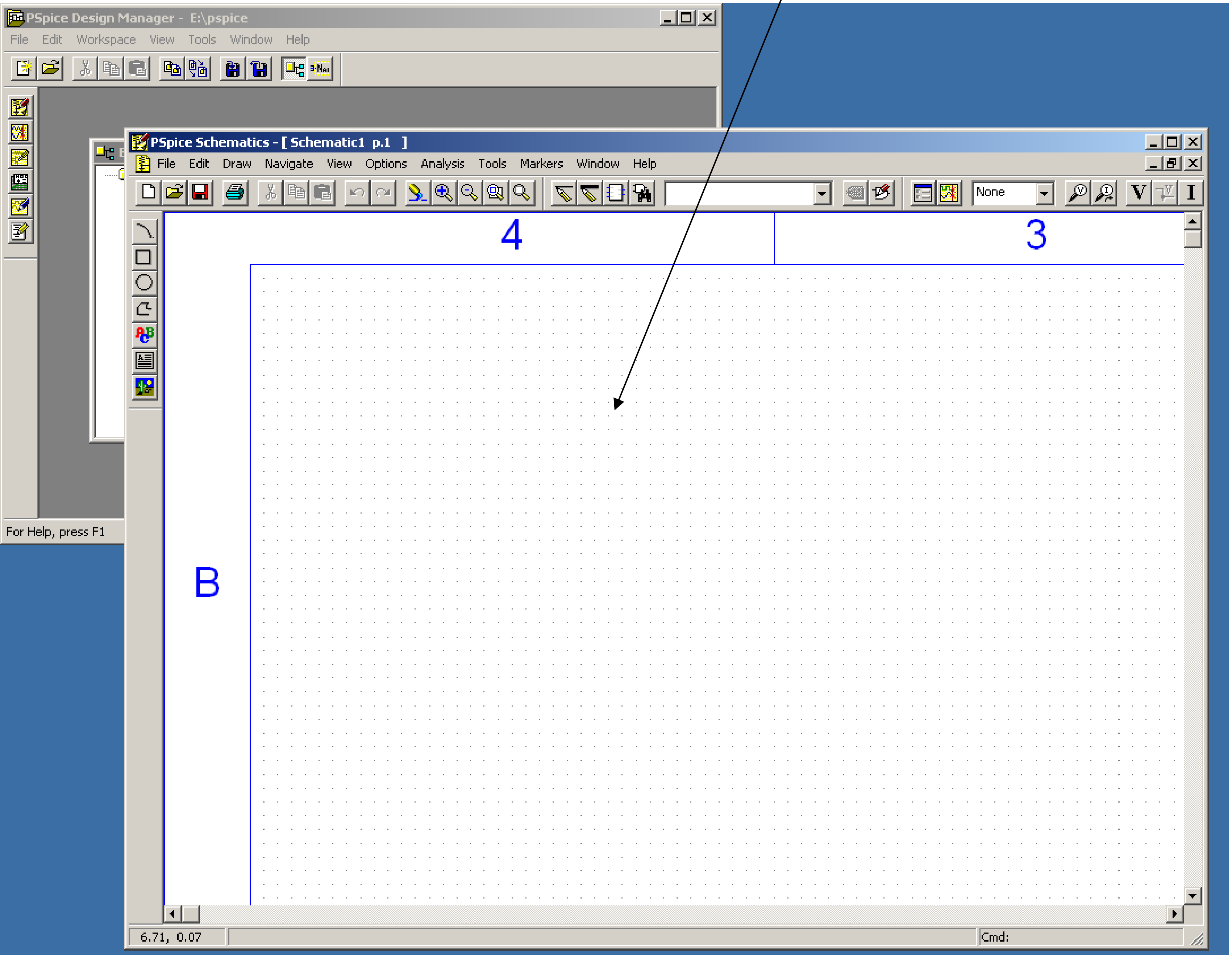


It will then look like this:

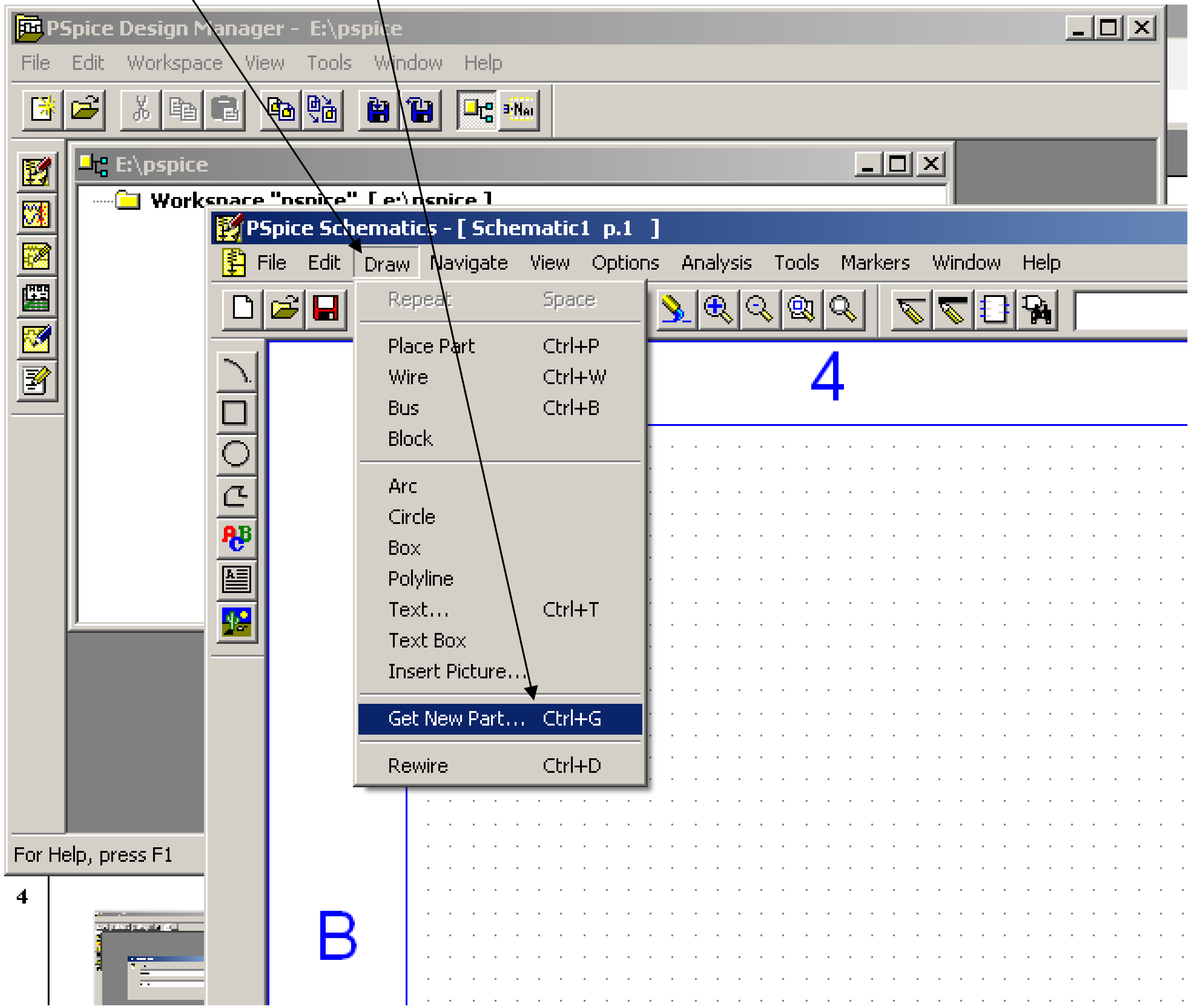
Then click here



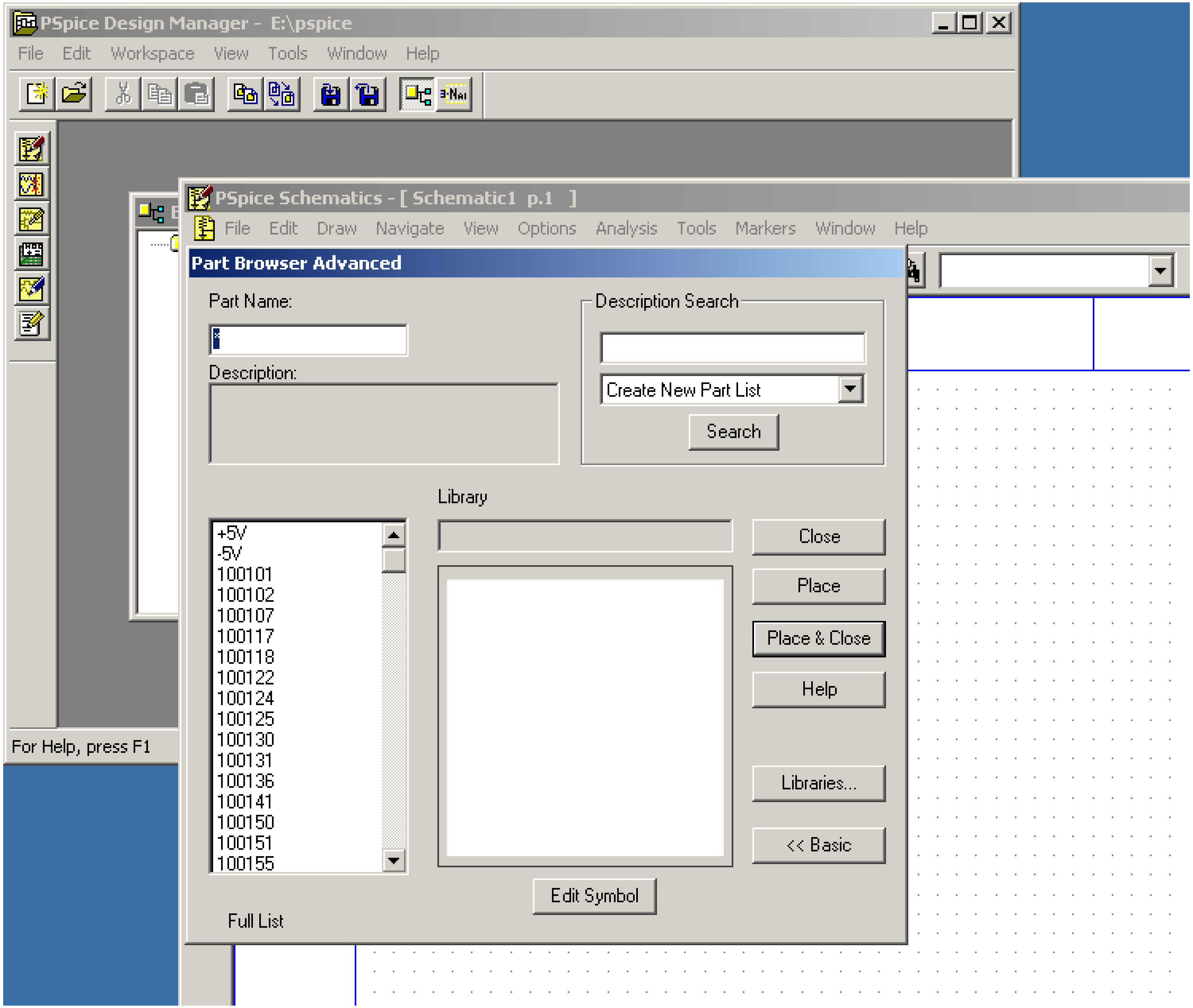
An area to draw your circuit will open up



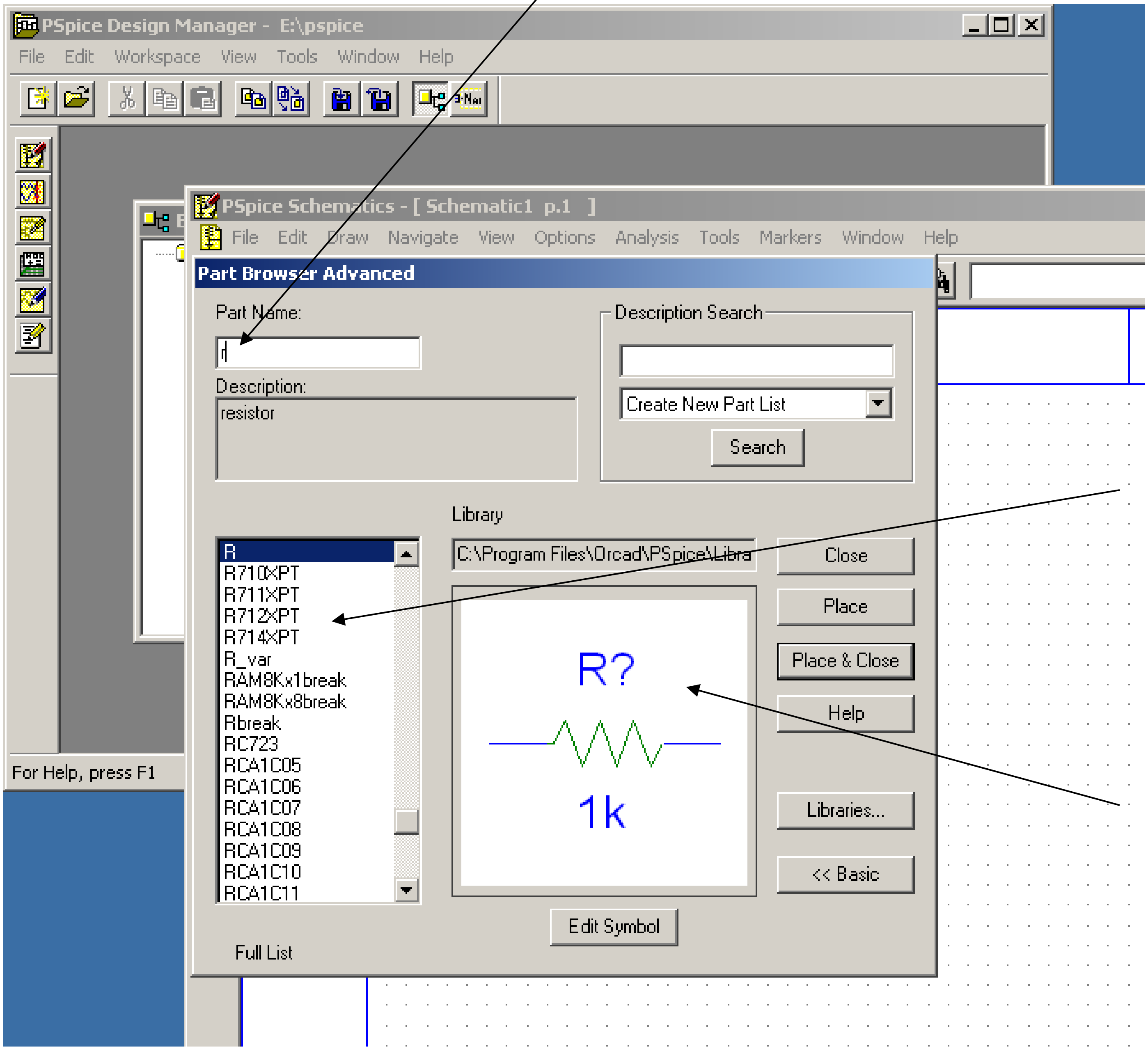
Click on Draw: Get New Part



This window will then open up



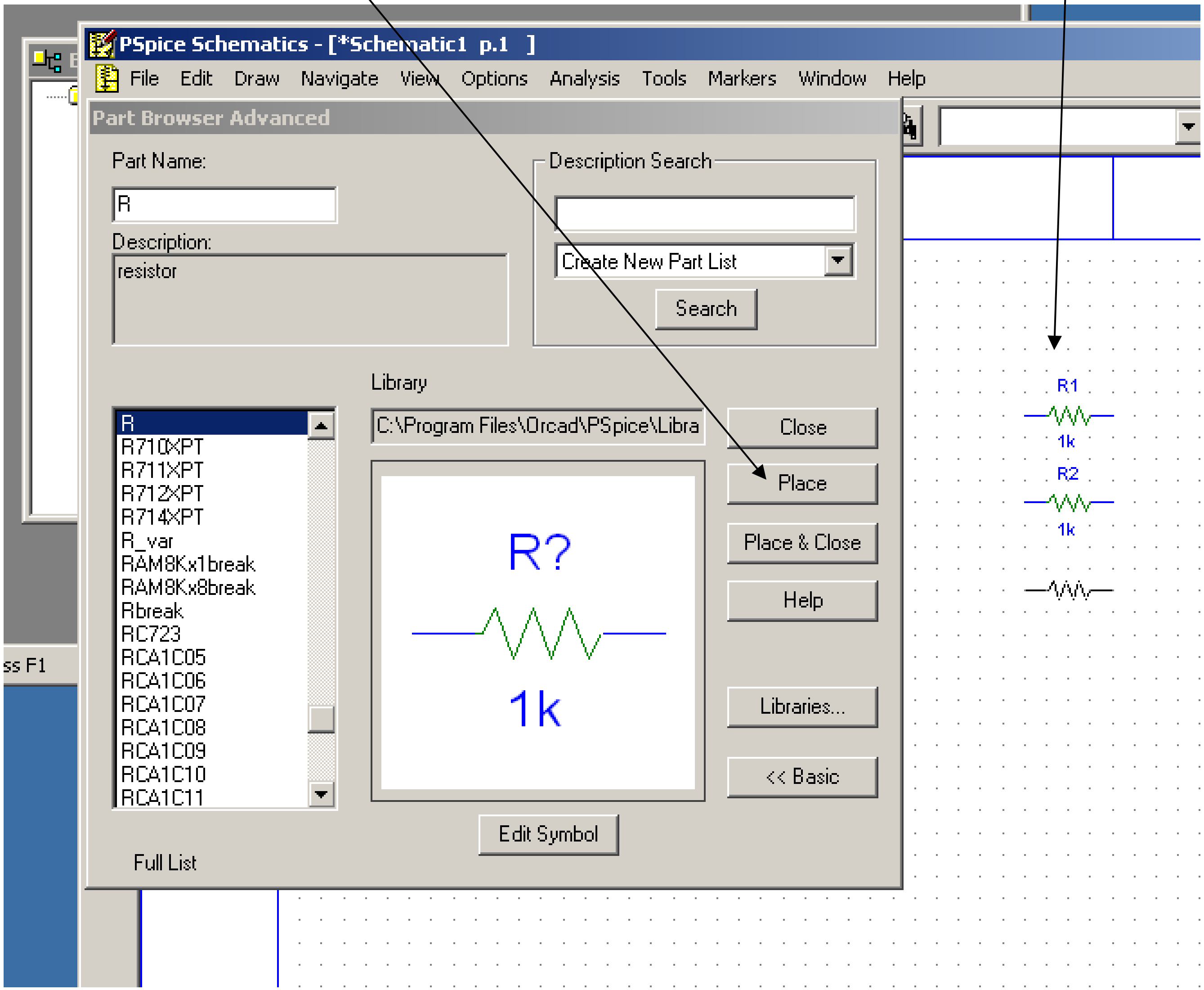
Enter a part name: R, C, L, etc…



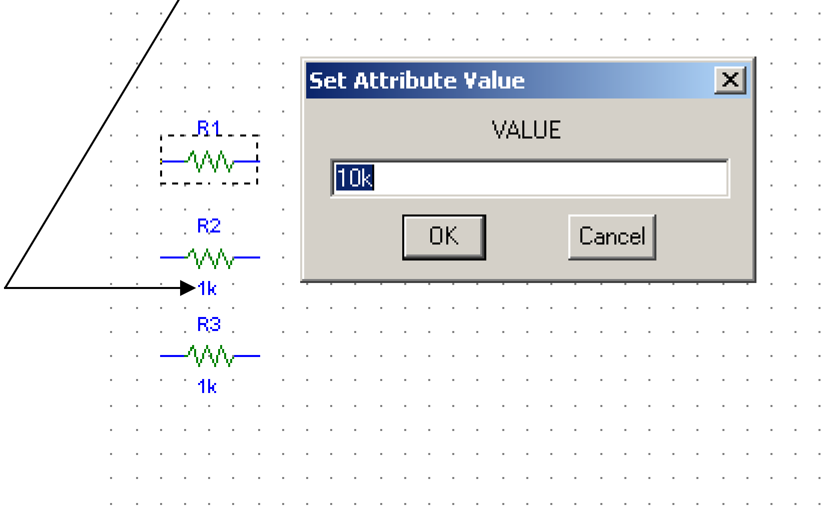
All related parts will list here: select the one you want

It's symbol will then appear here

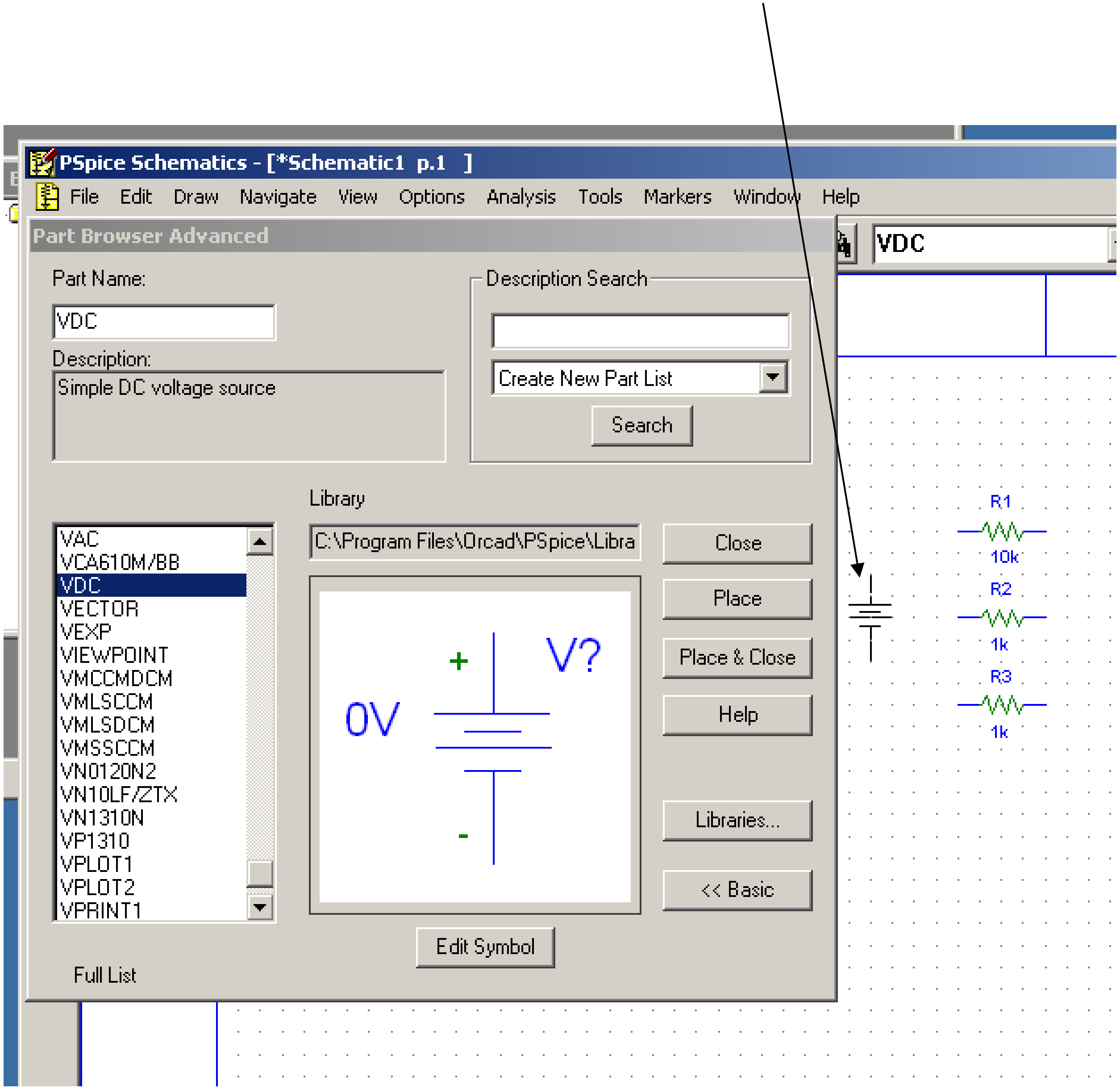
Click Place and put the part(s) where you want it or them



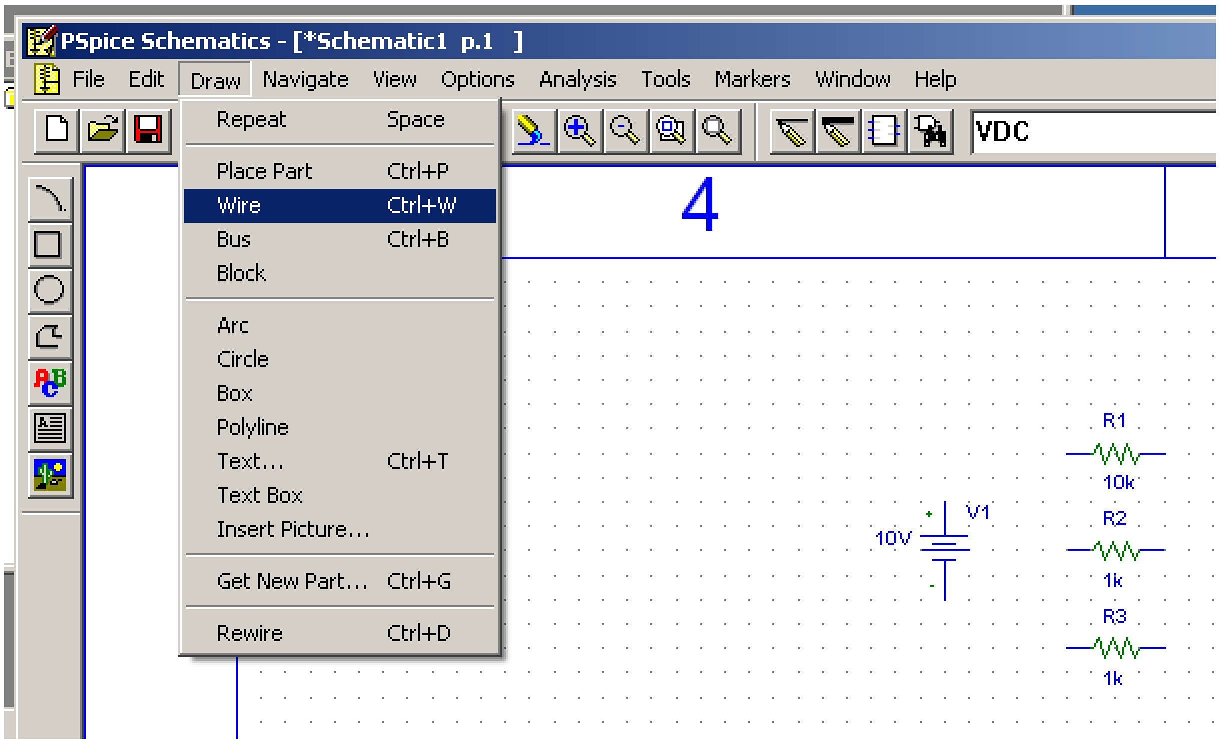
Click on the part value to change its value:



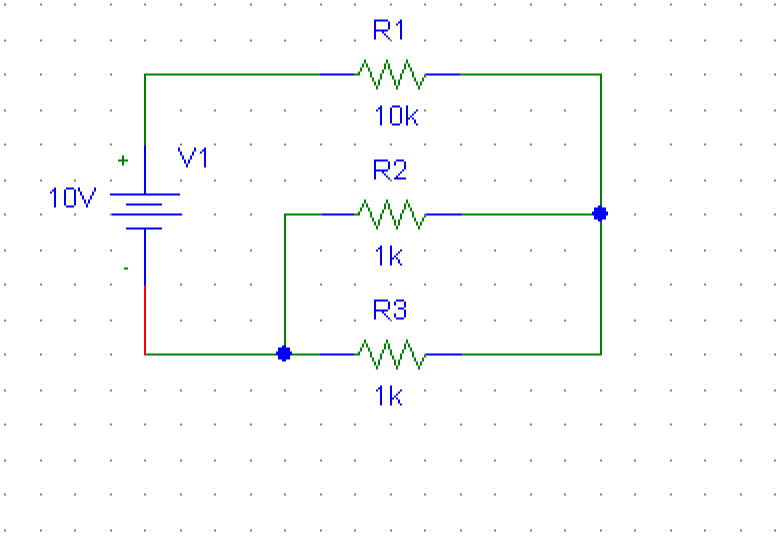
Then select a DC voltage supply and place it



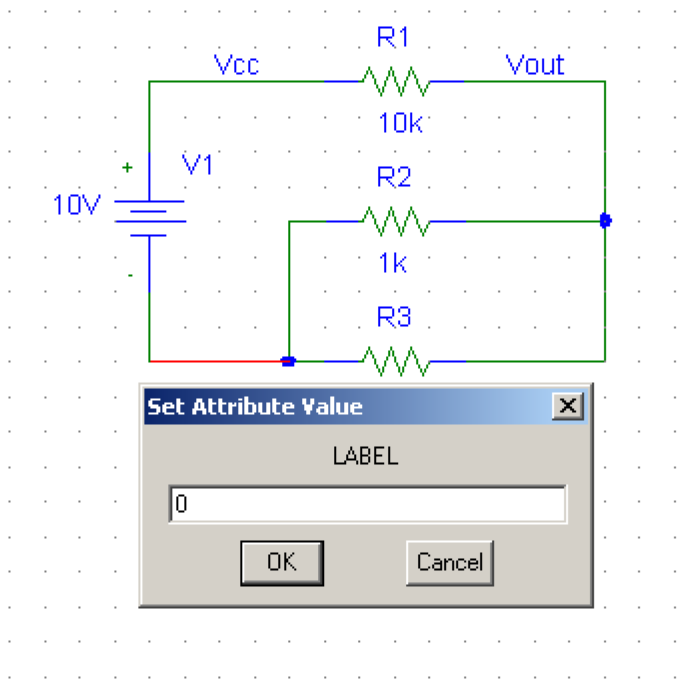
Draw the wires to connect all the parts



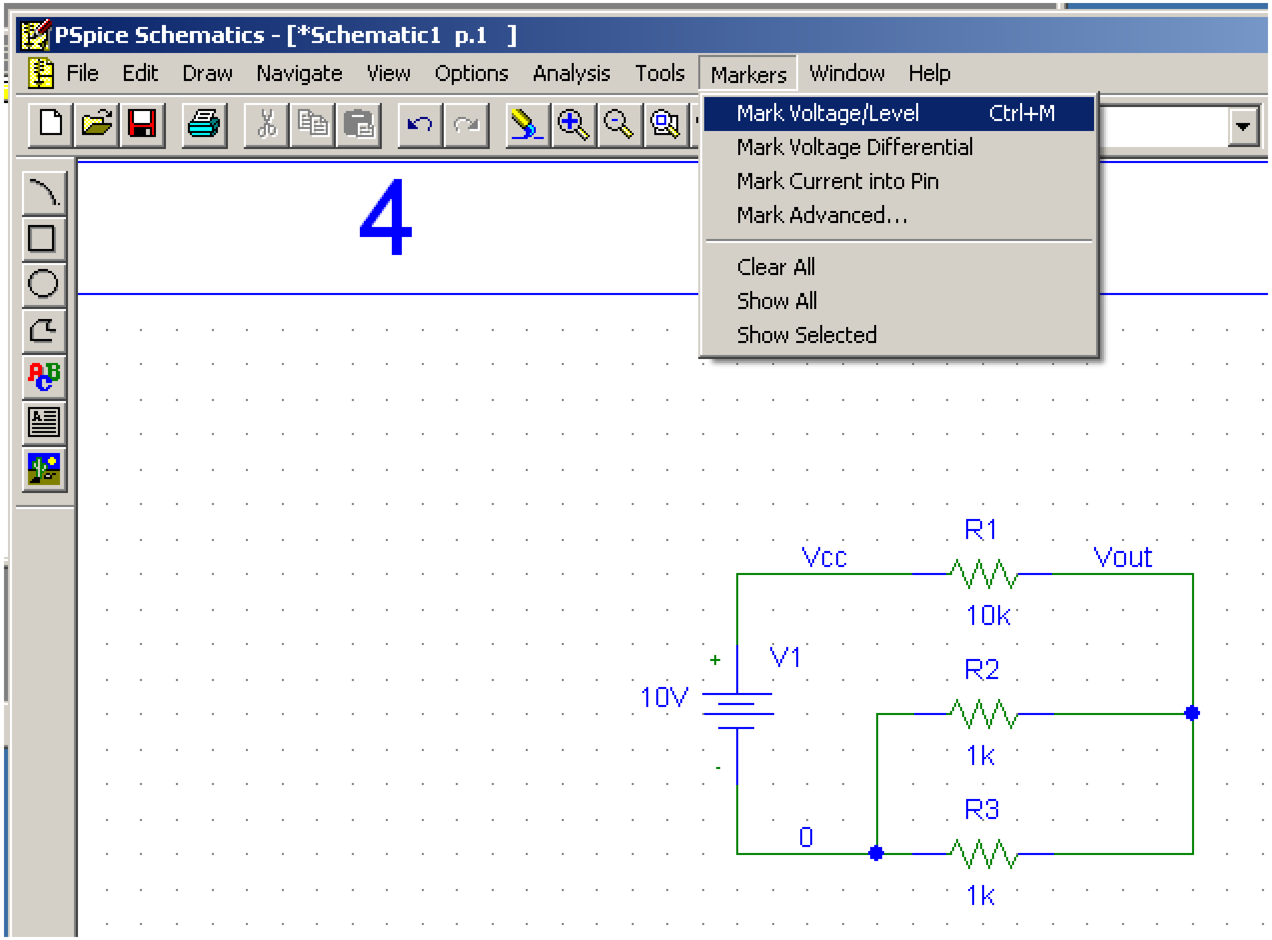
Your circuit will look something like this:



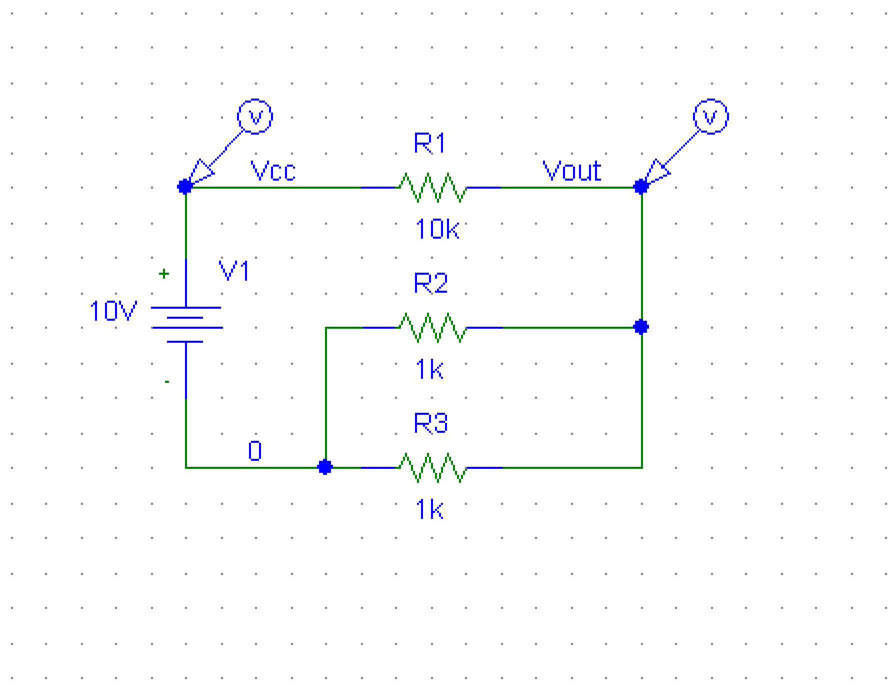
Click on the wires to label the circuit nodes: call the ground wire “0”



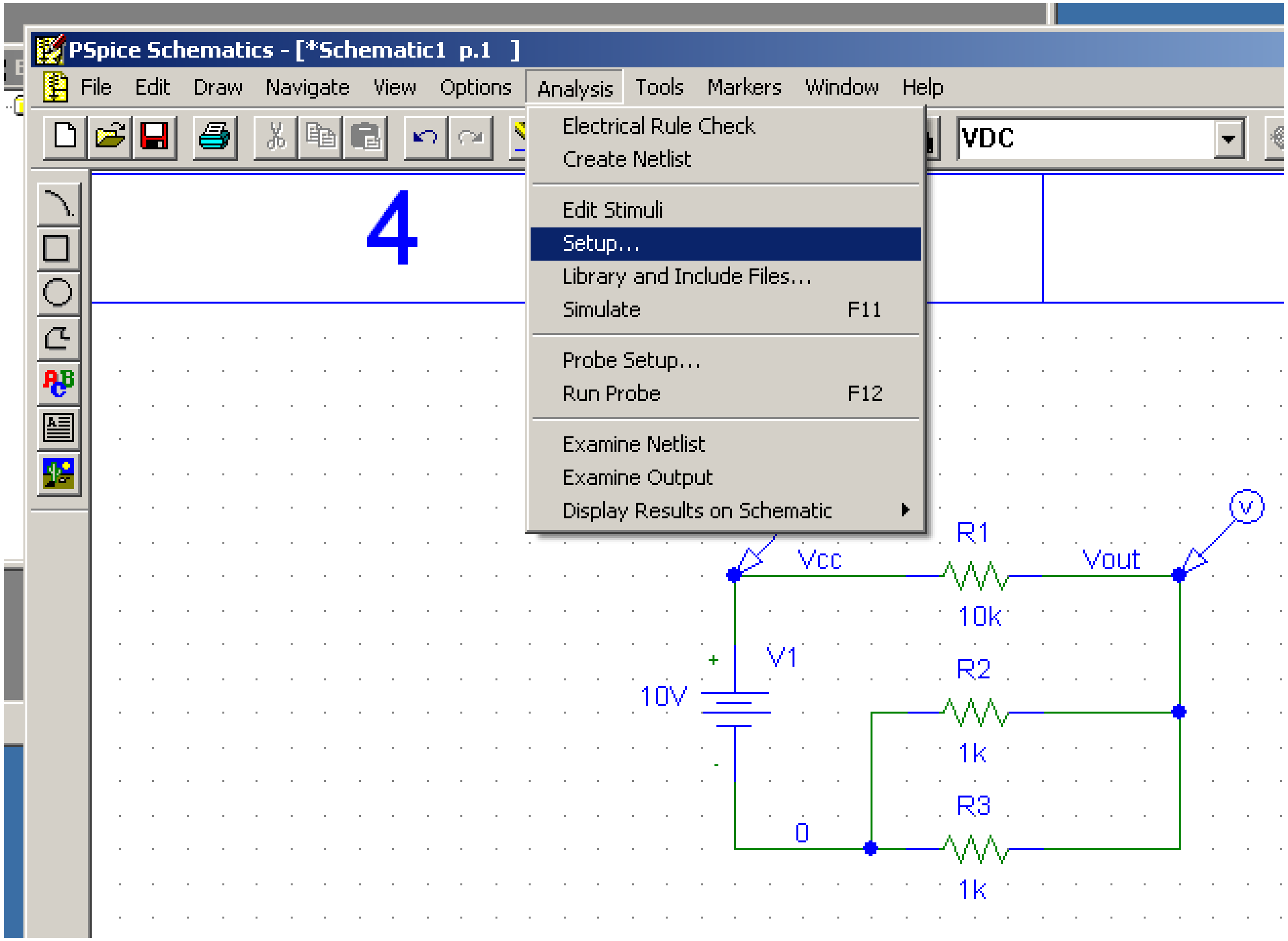
Now mark the circuit nodes of interest with Voltage Level Markers



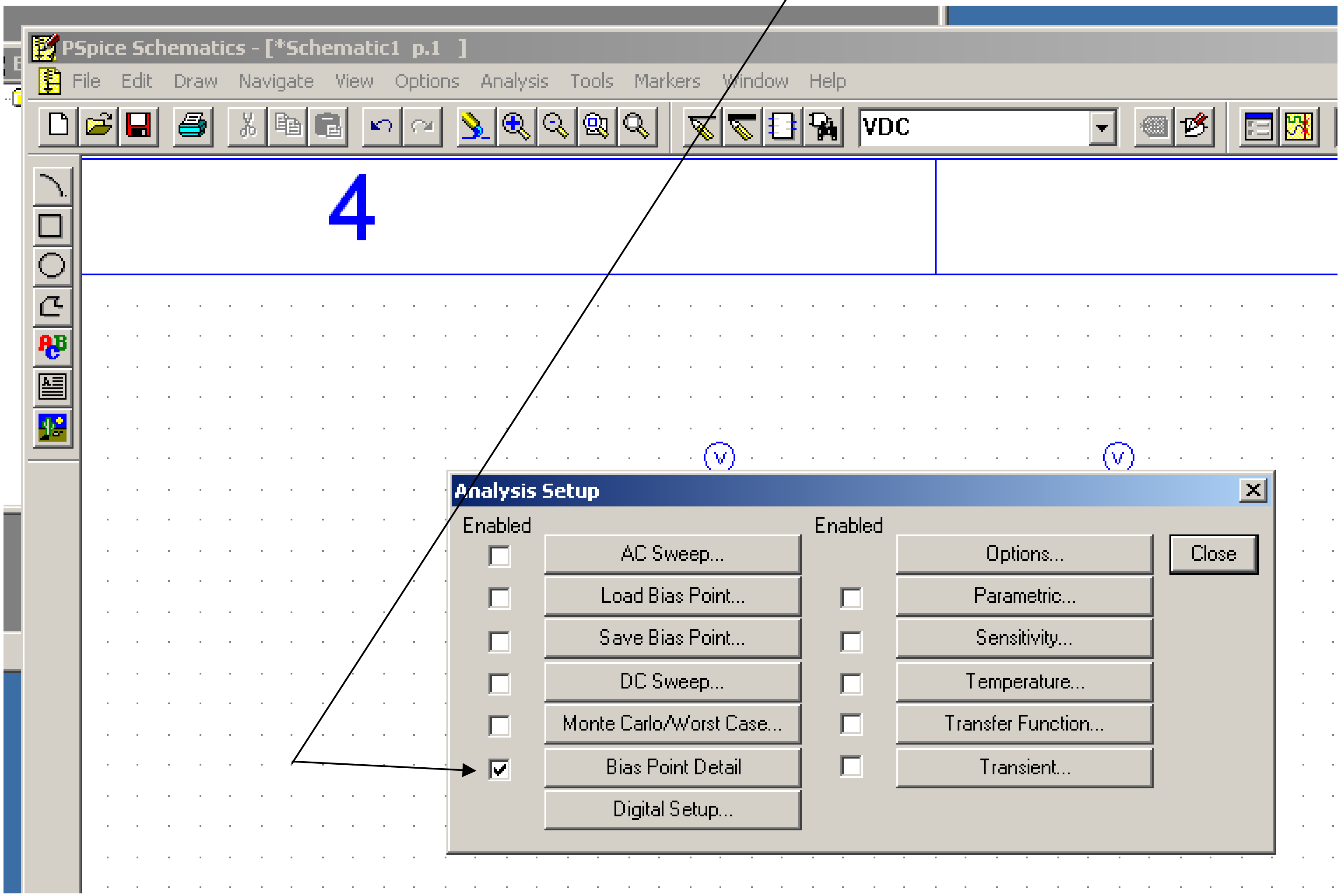
Your circuit will look something like this:



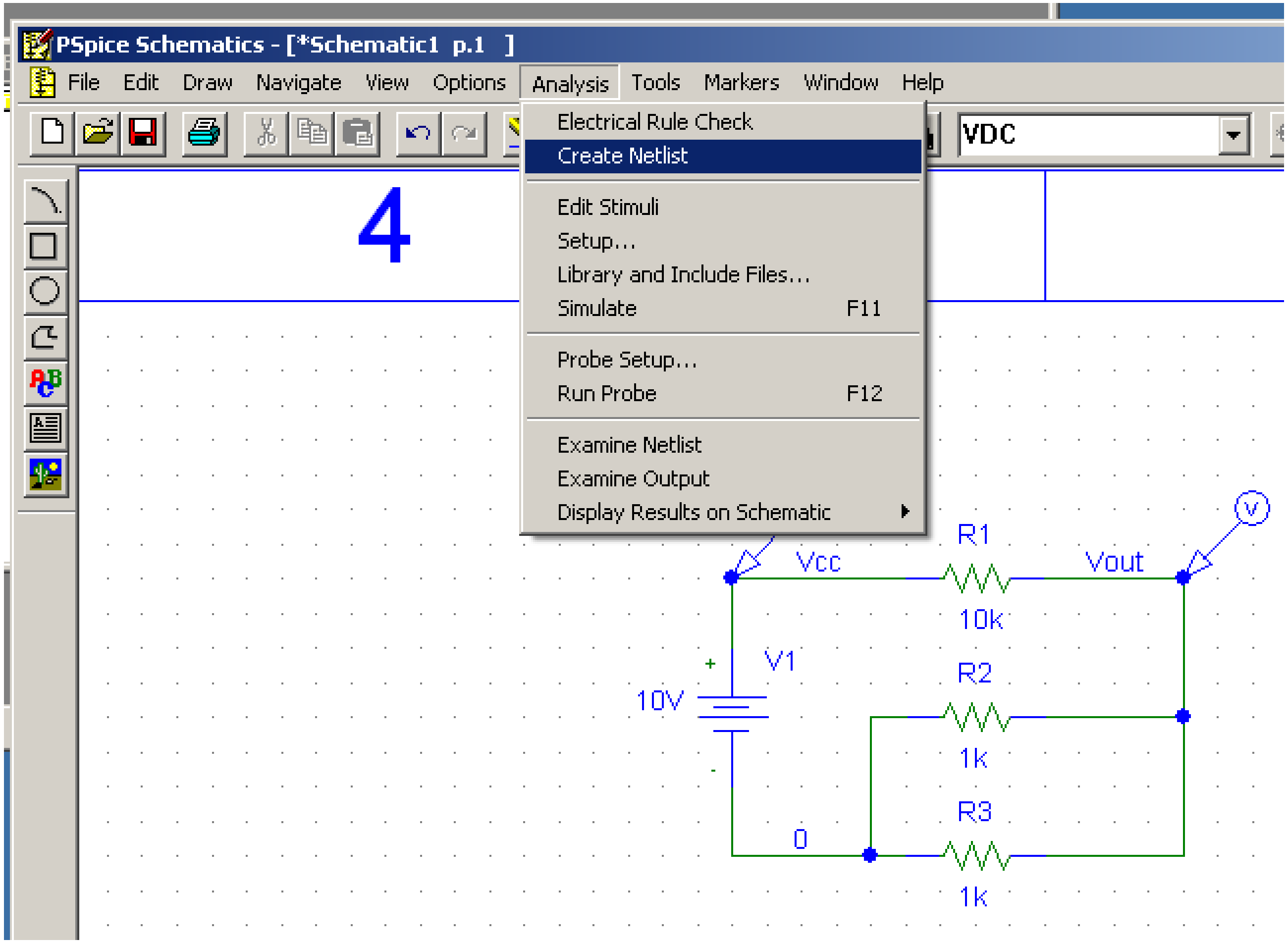
Click on Analysis: Setup



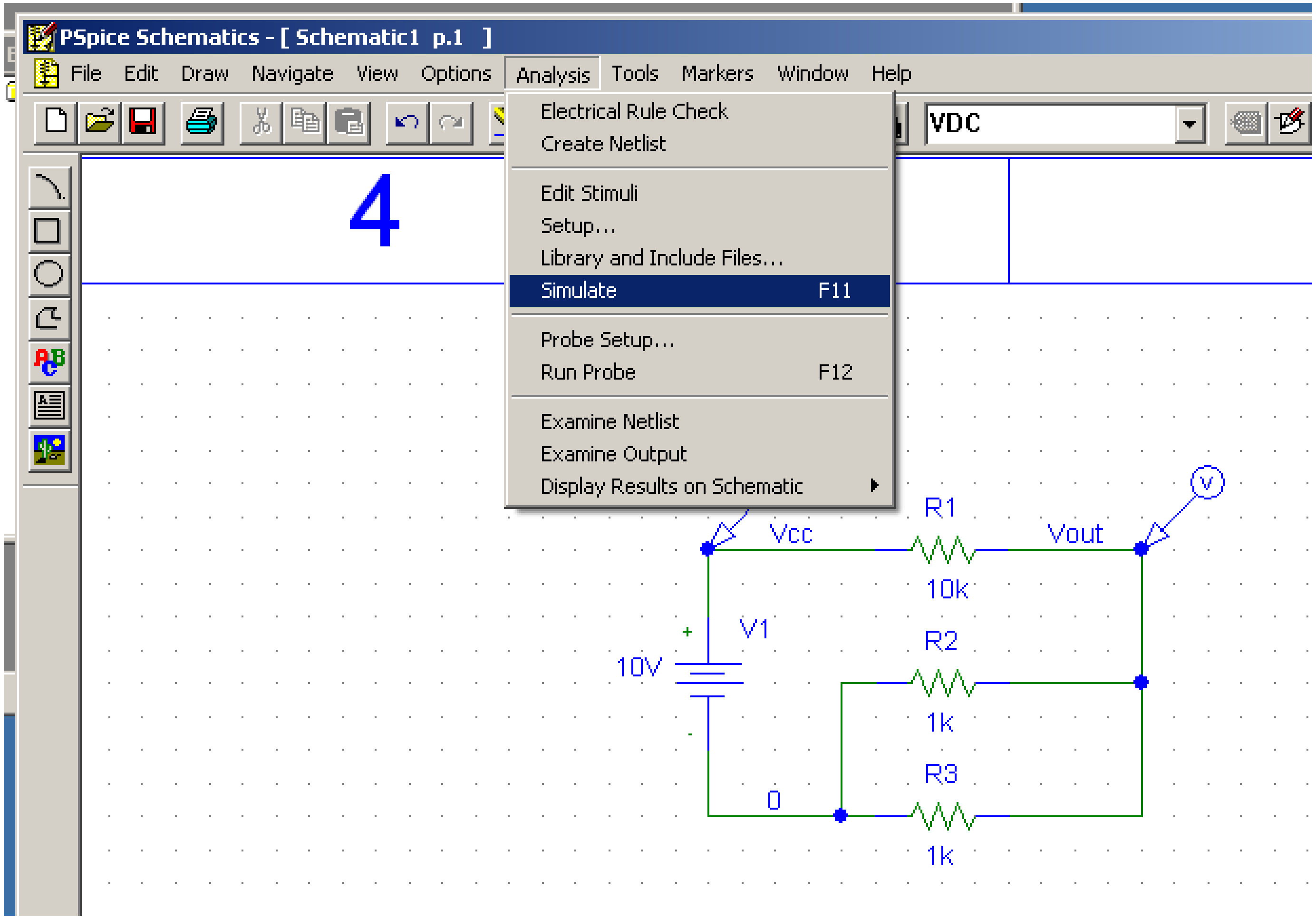
For DC circuit analysis, enable: “Bias Point Detail”



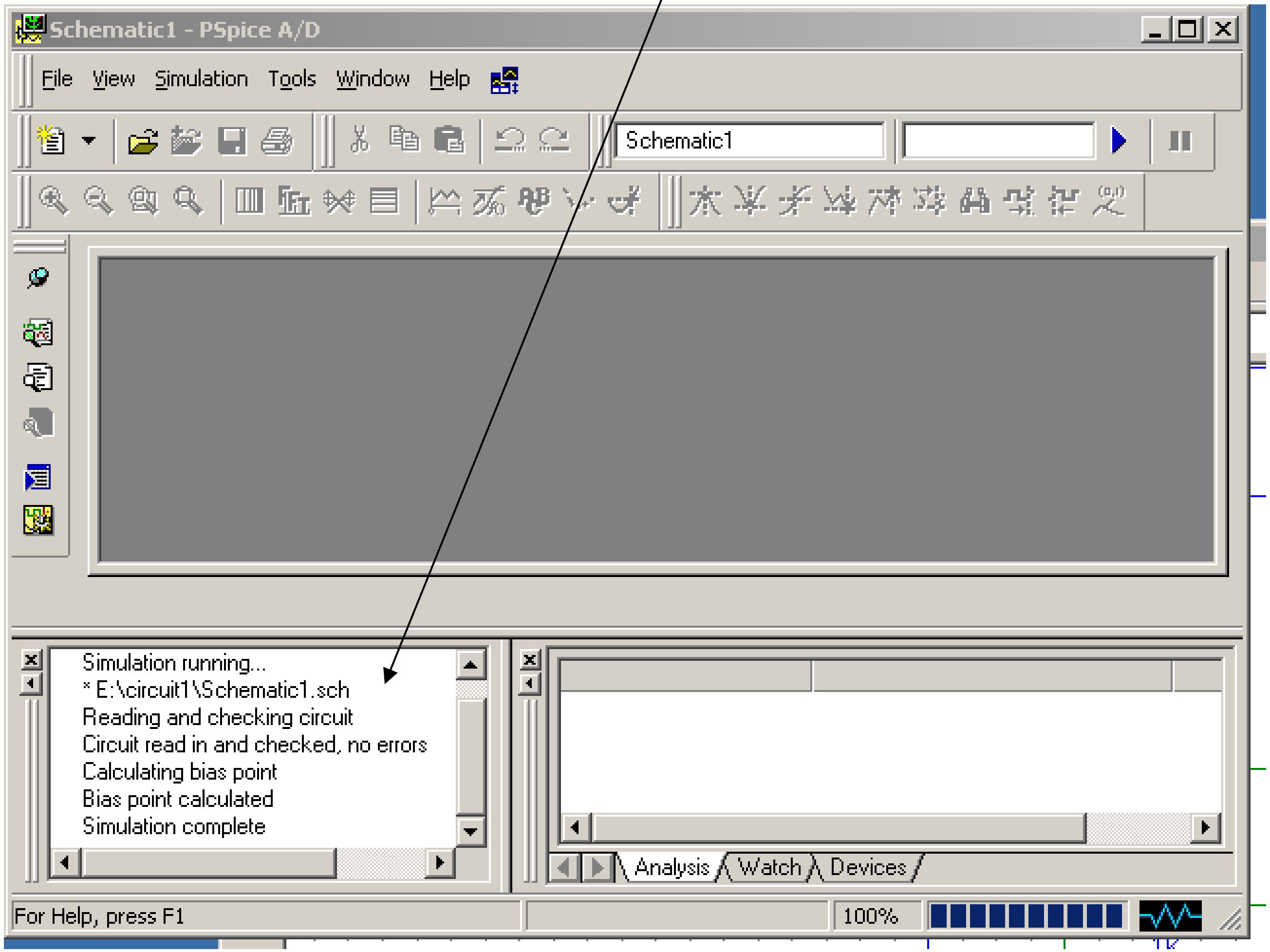
Select Analysis: Create Netlist (correct circuit layout if any errors found)



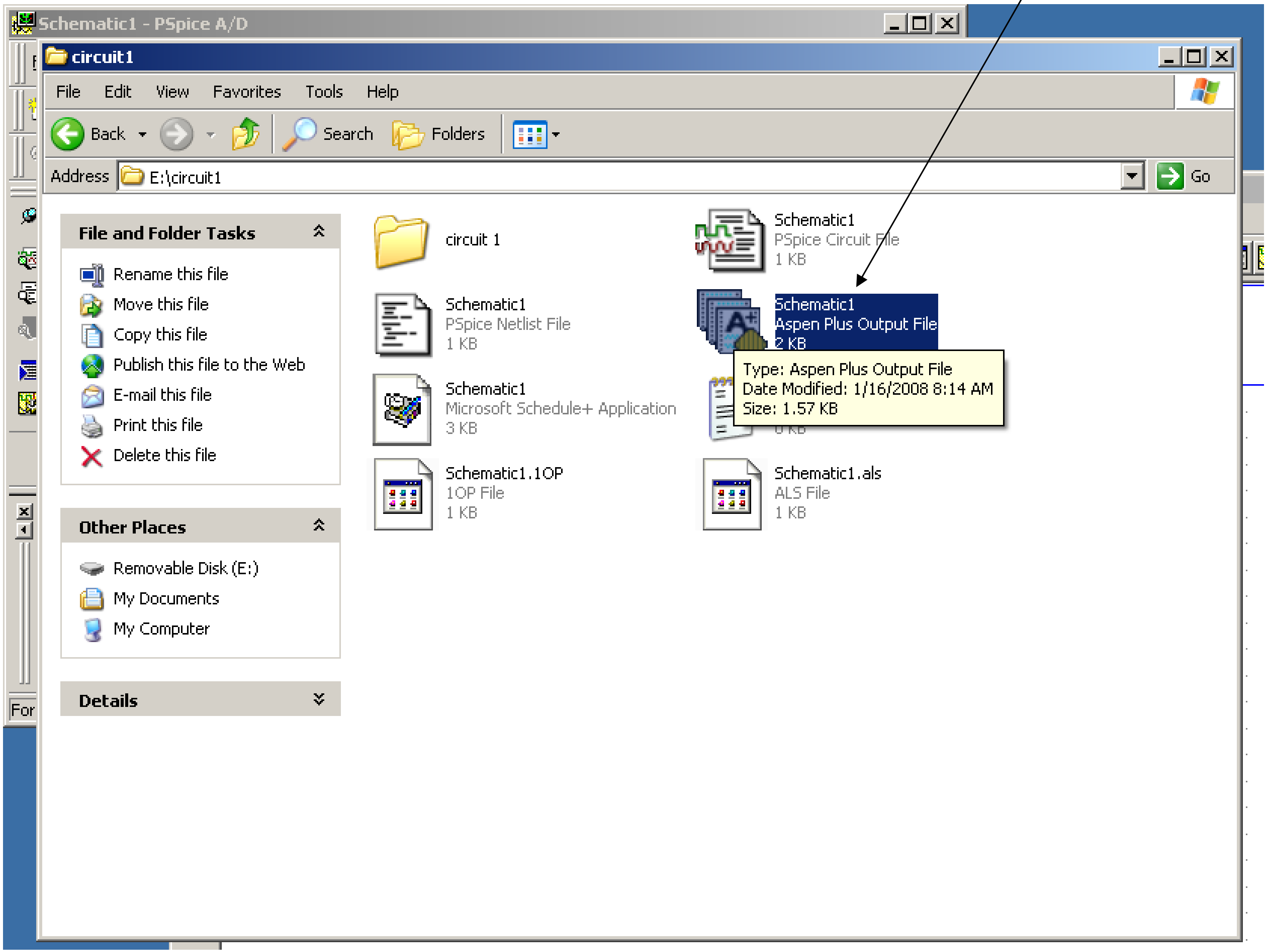
Select Analysis: Simulate to analyze the circuit



This window tells you how the simulation went



Go to your workspace directory and click on the Output File



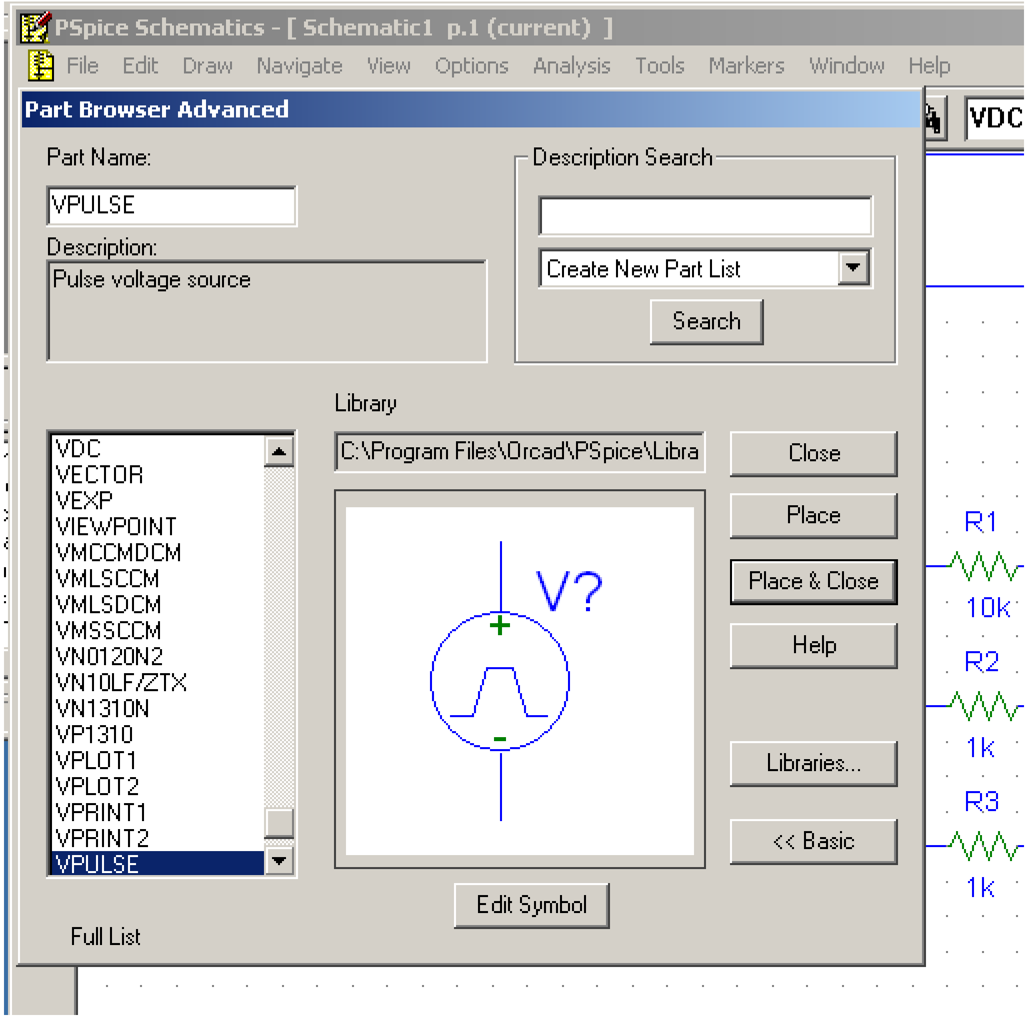
Schematic1 - Notepad

File Edit Format View Help

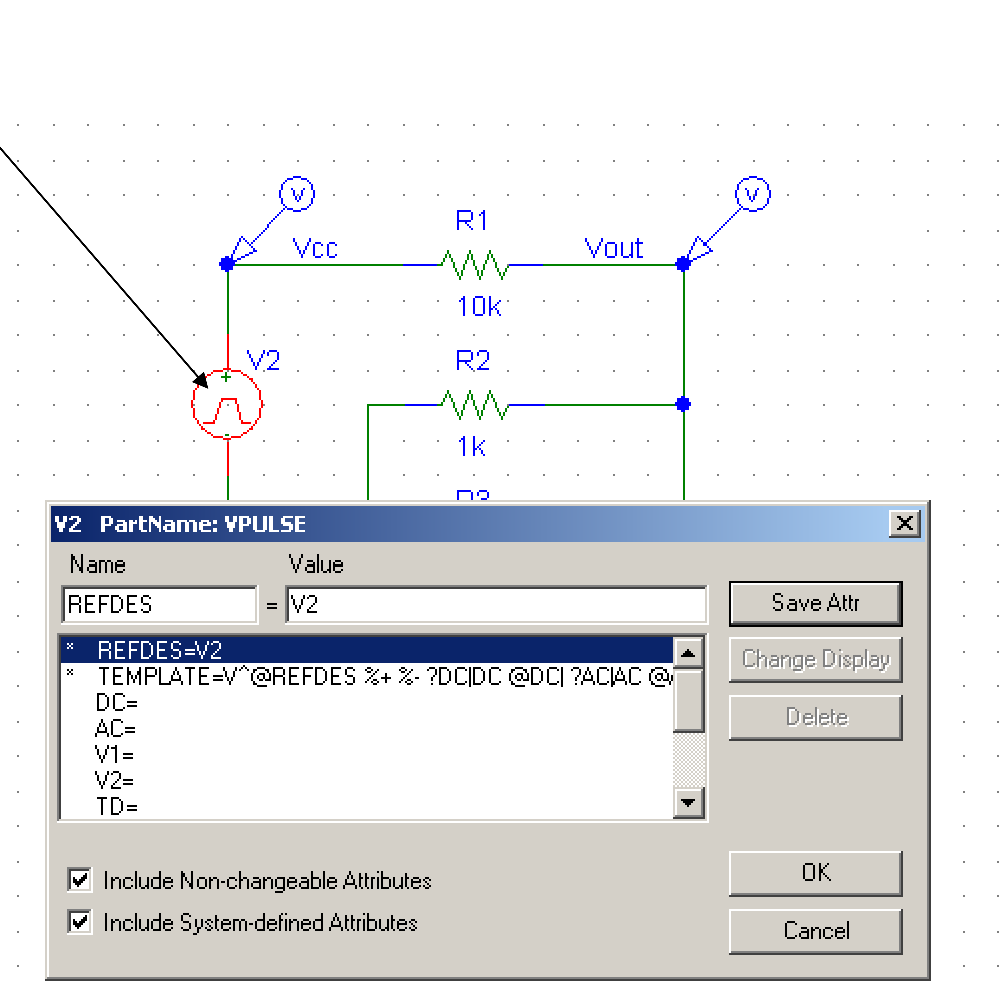
```
** Analysis setup **
.OP


* From [PSPICE NETLIST] section of C:\Program Files\orcad\PSpice\PSpice.ini:
.lib "nom.lib"

.INC "Schematic1.net"



**** INCLUDING Schematic1.net ****
* Schematics Netlist *



R_R1         Vcc Vout  10k
R_R2         0 Vout  1k
R_R3         0 Vout  1k
V_V1         Vcc 0 10V

**** RESUMING Schematic1.cir ****
.PROBE V(*) I(*) W(*) D(*) NOISE(*)


.END
▯
**** 01/16/08 08:14:24 ********* PSpice 9.2 (Mar 2000) ******** ID# 1108003738
 * E:\circuit1\Schematic1.sch


 ****     SMALL SIGNAL BIAS SOLUTION       TEMPERATURE =   27.000 DEG C


******************************************************************************



 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE


(  Vcc)   10.0000  ( Vout)     .4762




    VOLTAGE SOURCE CURRENTS
    NAME          CURRENT

    V_V1         -9.524E-04
```

Scroll down to you find the voltages for the nodes you marked in the circuit. The voltages are referenced to node “0.”

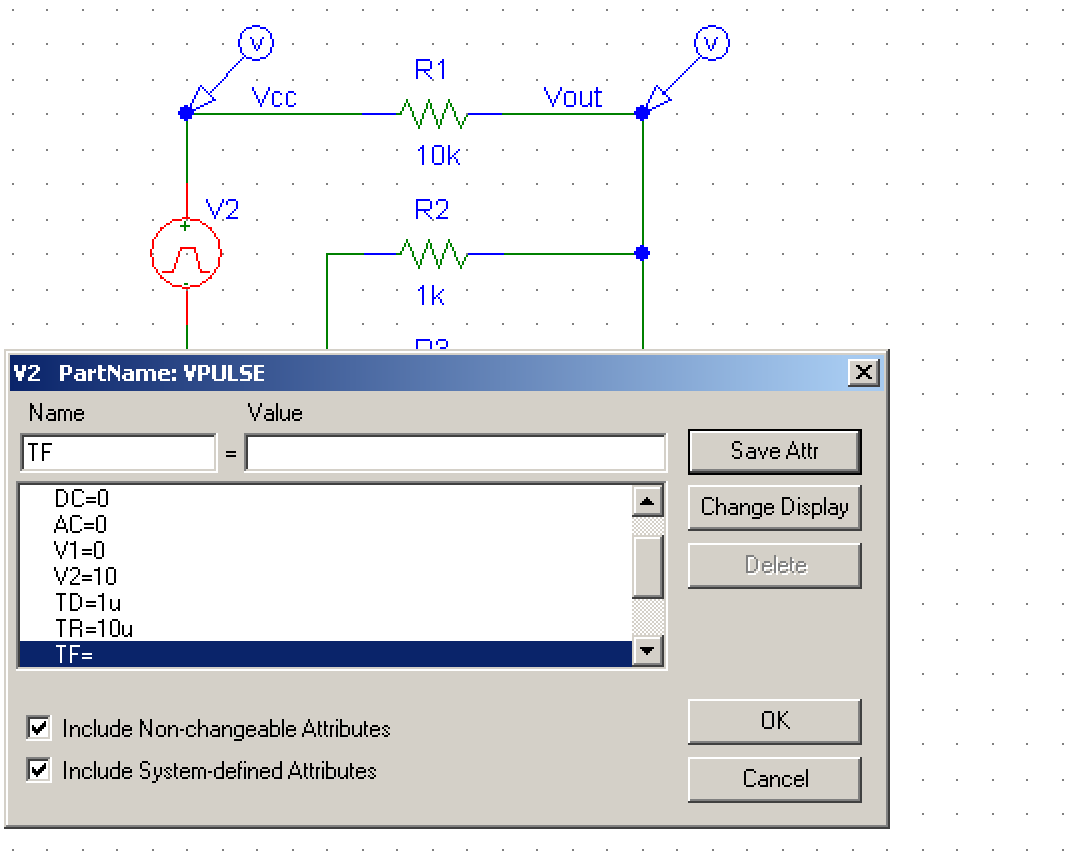
This time, replace the DC voltage source with a time varying voltage source, such as VPULSE



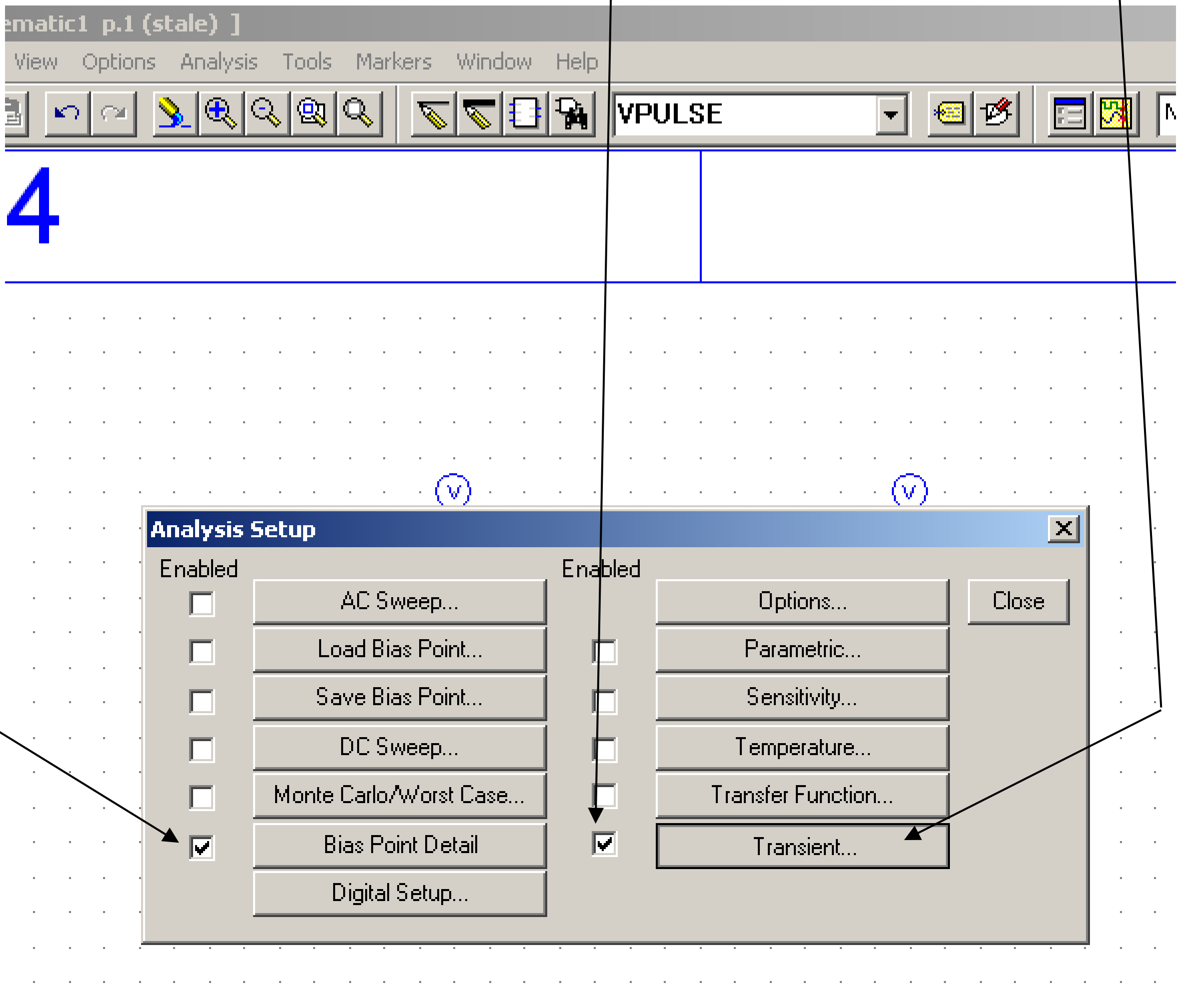
Put it in the circuit and click on it to select it’s parameters



Here I selected the constant DC and AC values to be zero, the initial voltage level (V1) to be zero, the final voltage level (V2) to be 10V, the time delay (TD) to be 1us and the rise time (TR) to be 10us. I left the other options blank

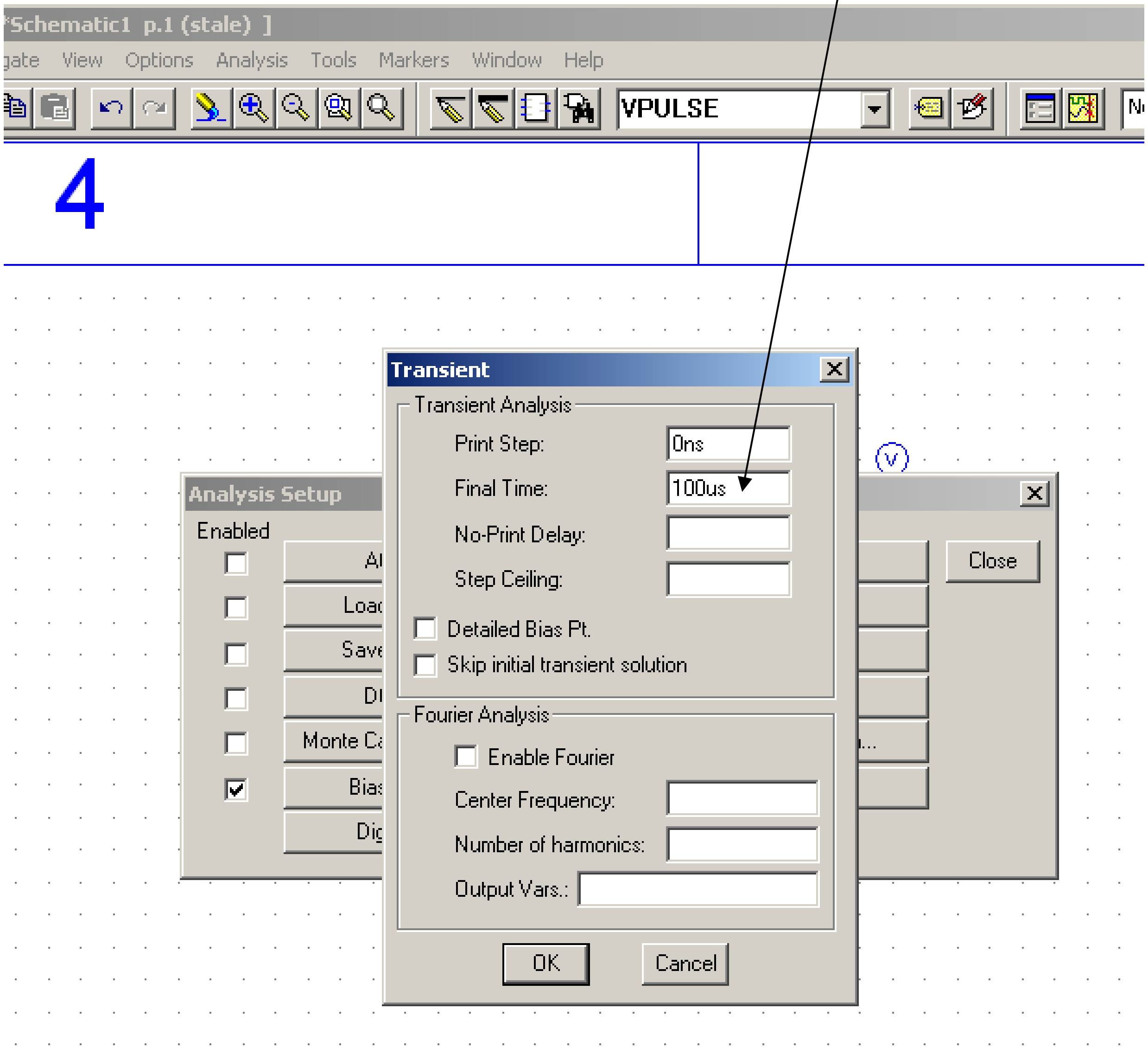


Under Analysis Setup, enable “Transient…” and click on it



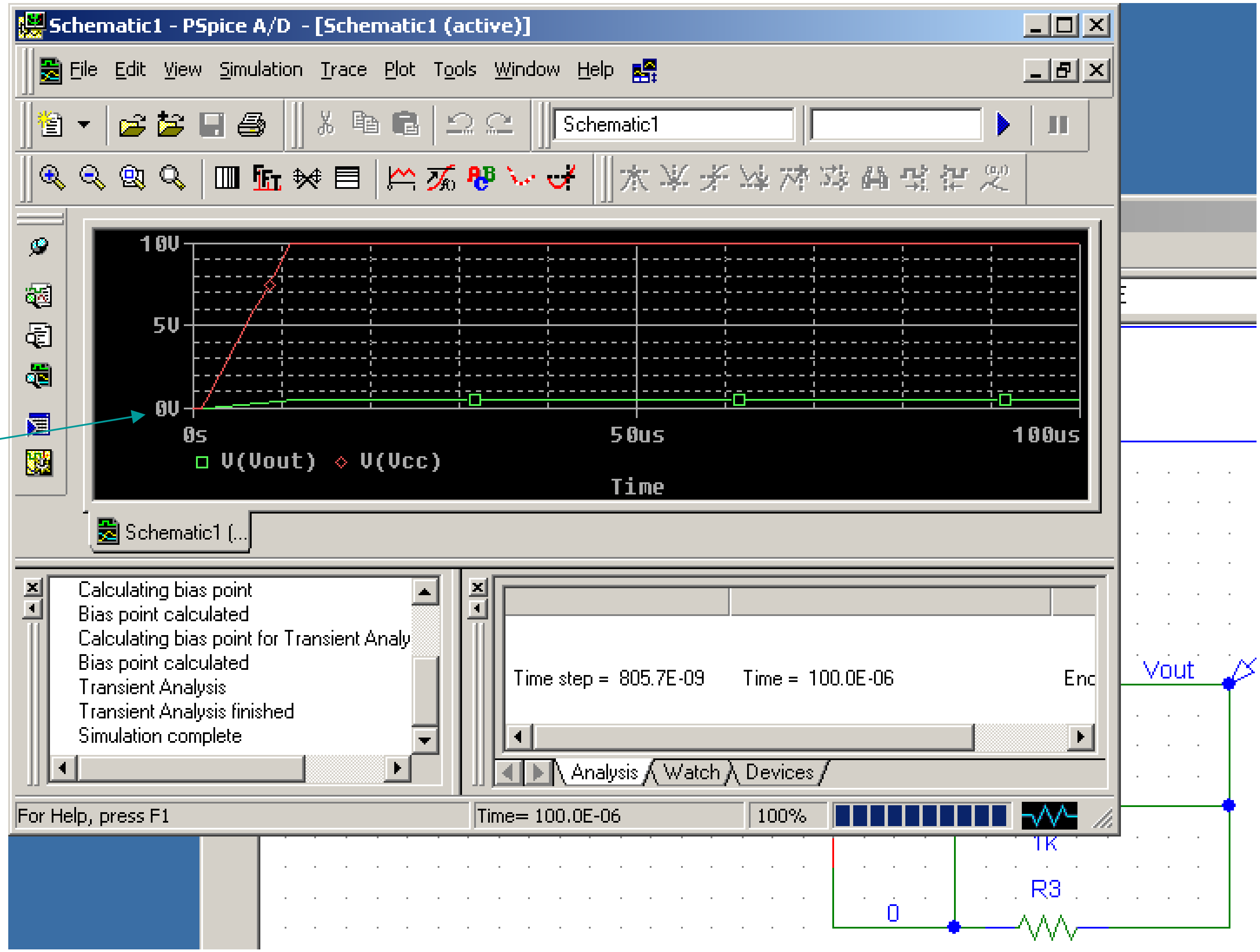
Keep the “Bias Point Detail” enabled too

Set the Transient Analysis Final Time to 100us



Run the Simulation and these result will appear:

A plot of the voltages at all the marked nodes over the transient analysis time (0s to 100us)



Play with this window to adjust display time and what signals are displayed