DIAYLA UNIVERSITY
COLLAGE OF ENGINEERING
DEPARTMENT OF ELECTRICAL
POWER &MACHINES ENGINEERING

PS pics Simulation

SEMINAR PREPARED BY
ASSIST. LECTURER:
HUSHAM IDAN HUSSEIN

What is PSpice??

PS pice :is a Power Simulation Program for Integrated Circuits Emphasis

❑Simulates the behavior of electronic circuits

– Allows designers to evaluate designs without actually building them.

– Major savings in both $$$ and time!

– Designer know what to expect when you build your circuit

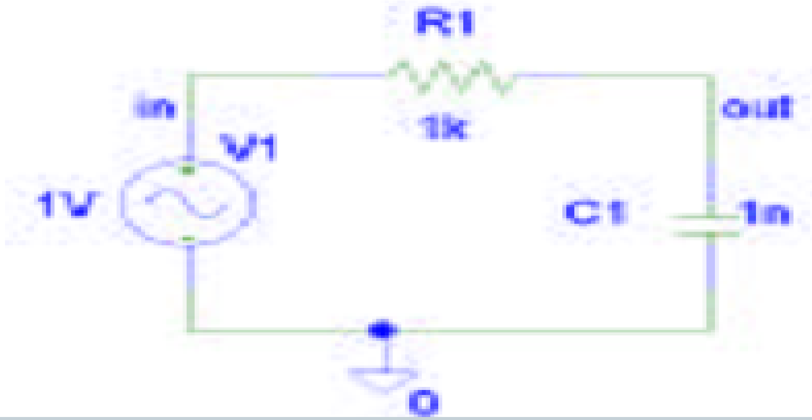
Many types of PSpice

- HSPICE
- Spectre , Spectre RF(Cadence)
- Eldo (Mentor Graphics)
- And many, many more…

Five Most Common Semiconductor Devices:

diodes, BJTs, JFETs, MESFETs, and MOSFETs.

How does it work?



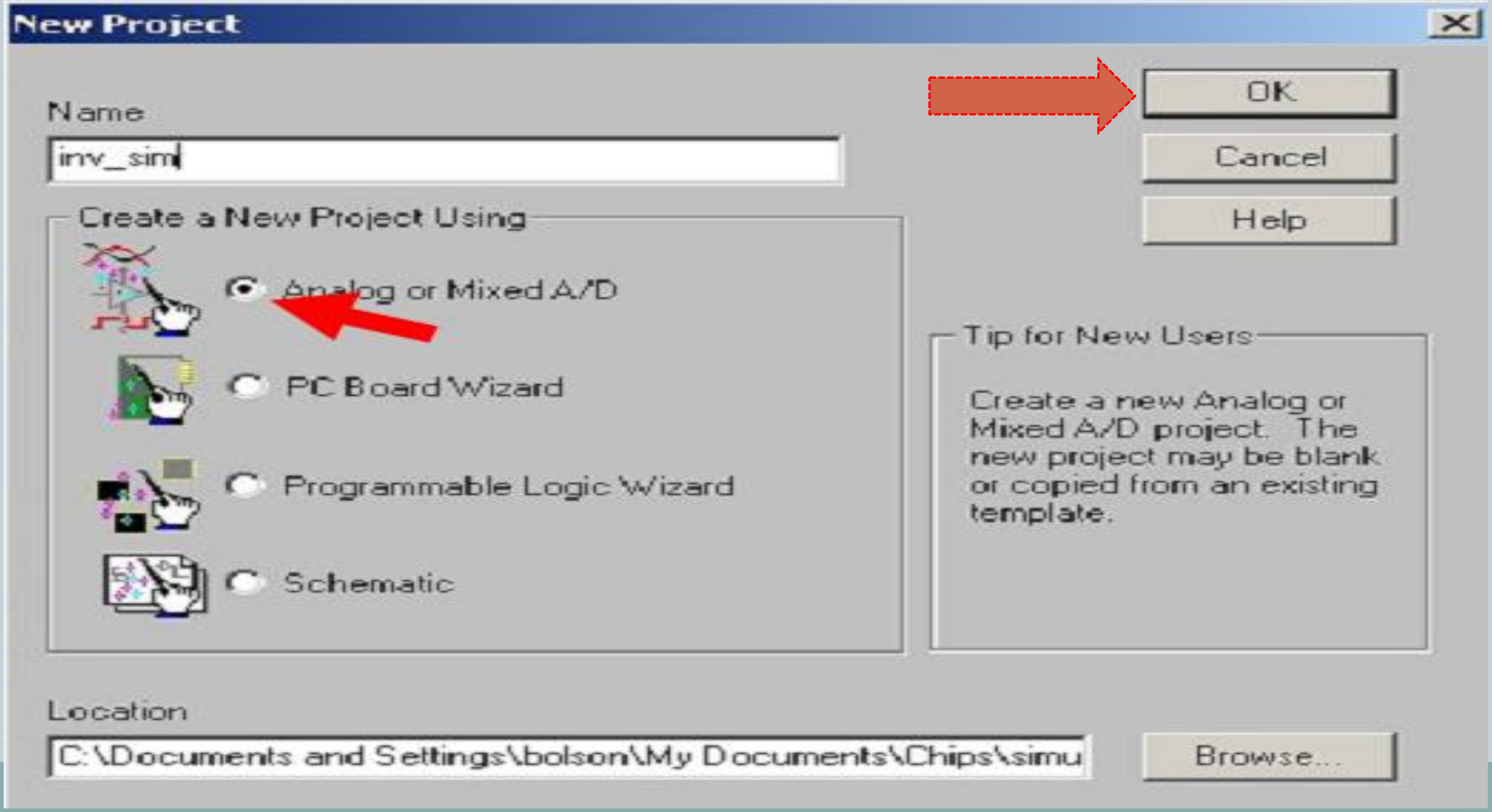

```
V1  in  0  1V
R1  in  out  1k
C1  out  0  1n
```

What can you do with it?

- Frequency response; Bode Plots
- Transient Response
- DC sweeps, component value sweeps
- DC bias point
- Temperature analysis
- Noise Analysis

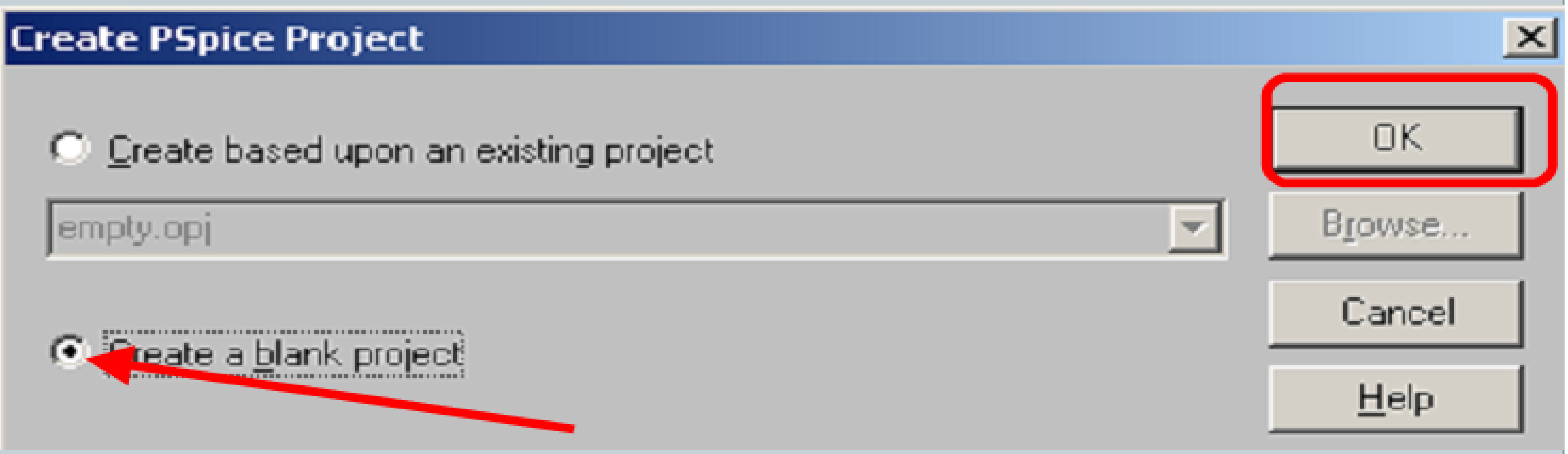
Get starting with PSpics

1- from the start menu click ps pice capture
2- At this point you will see a large window. From the toolbar at the top select: File->New->Project as show in the pic. Below:



3-You will see the Create PSpice menu:

- Select : Create blank project. As shown below
- Hit <OK>

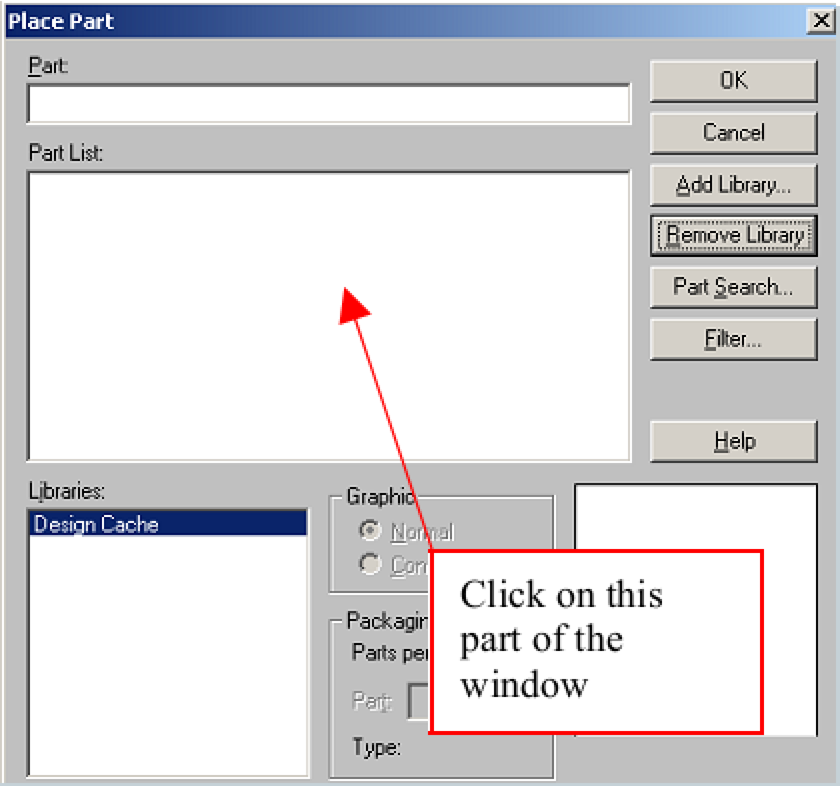


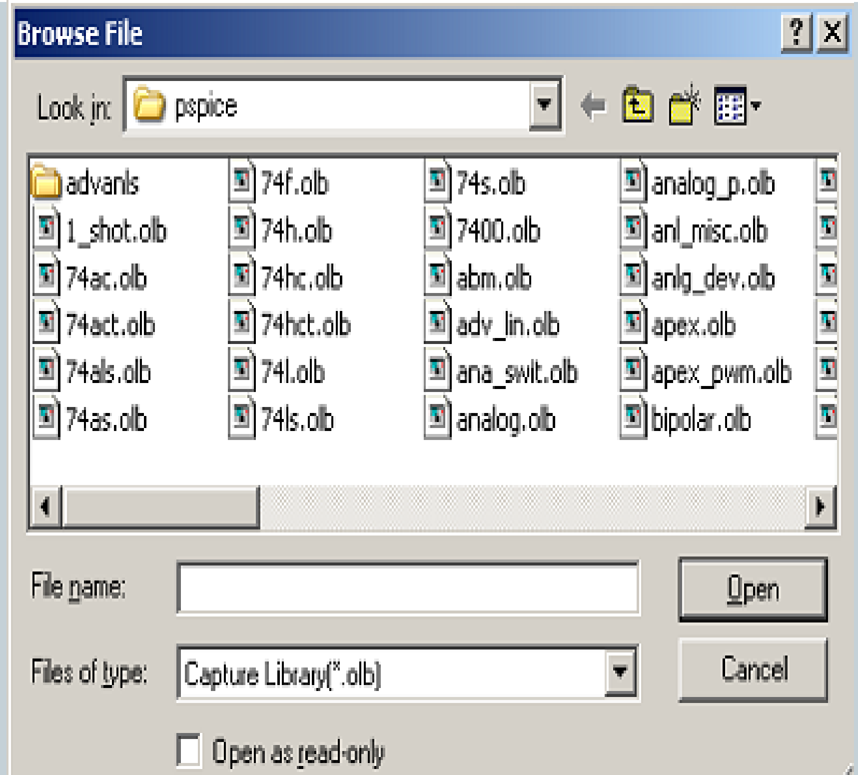
You are now ready to add parts

4- First click you mouse on the screen to activate the toolbars

5-When you first launch PSpice the libraries containing the parts may not be loaded . If this is the case you need to add them. If have libraries loaded skip the Adding Libraries section that follows.
Adding Libraries

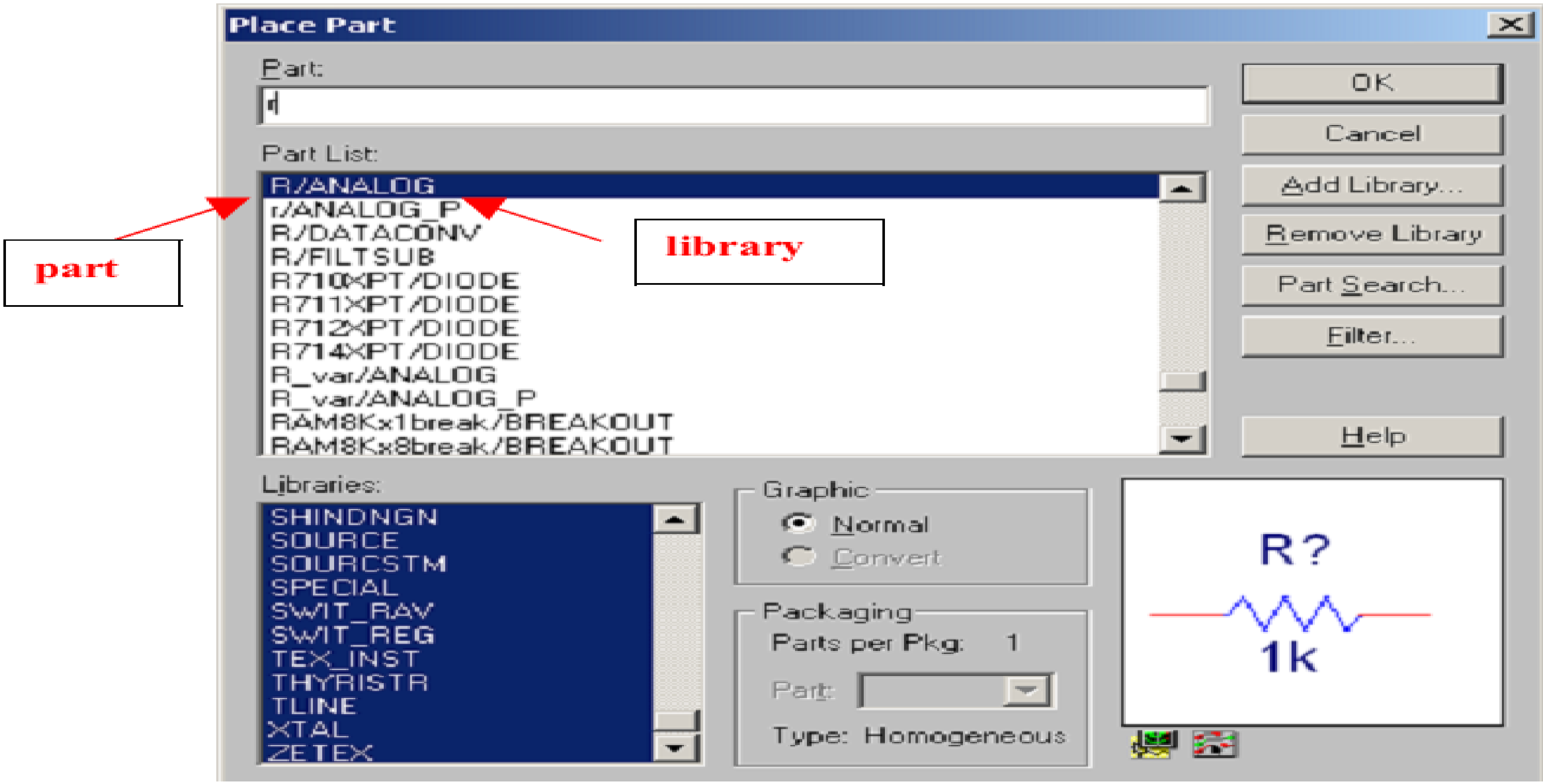
- In the menu below hit <Add Library>
- You will see Browse File menu.
- You can select all the libraries at once in the Browse File menu by: clicking your mouse on the window in the position indicate and then hitting <ctrl> a
- Too add all the selected libraries hit <Open> in the Browse File window as shown below


Place Part
Part:
Part List:
OK
Cancel
Add Library...
Remove Library
Part Search...
Filter...
Help
Libraries:
Design Cache
Graphic
Normal
Packagin
Parts per
Part:
Type:
Click on this part of the window


Browse File
Look in:
pspice
advanls
1_shot.olb
74ac.olb
74act.olb
74als.olb
74as.olb
74f.olb
74h.olb
74hc.olb
74hct.olb
74l.olb
74ls.olb
74s.olb
7400.olb
abm.olb
adv_lin.olb
ana_swit.olb
analog.olb
analog_p.olb
anl_misc.olb
anlg_dev.olb
apex.olb
apex_pwm.olb
bipolar.olb
File name:
Files of type:
Capture Library(*.olb)
Open
Cancel
Open as read-only

Place of part

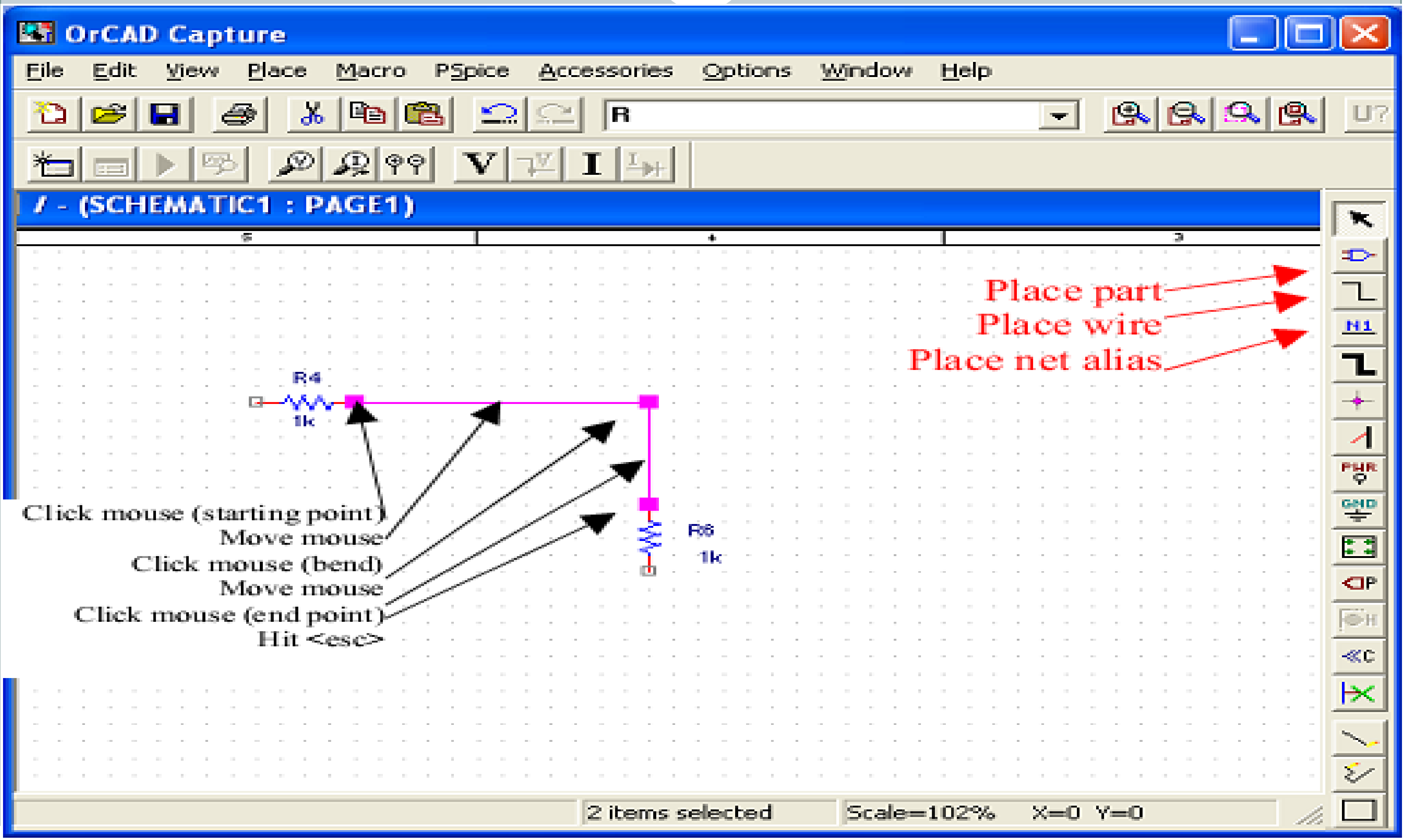
6-Enter the name of the part that you wish to add by scrolling or typing it directly. and Hit <OK>



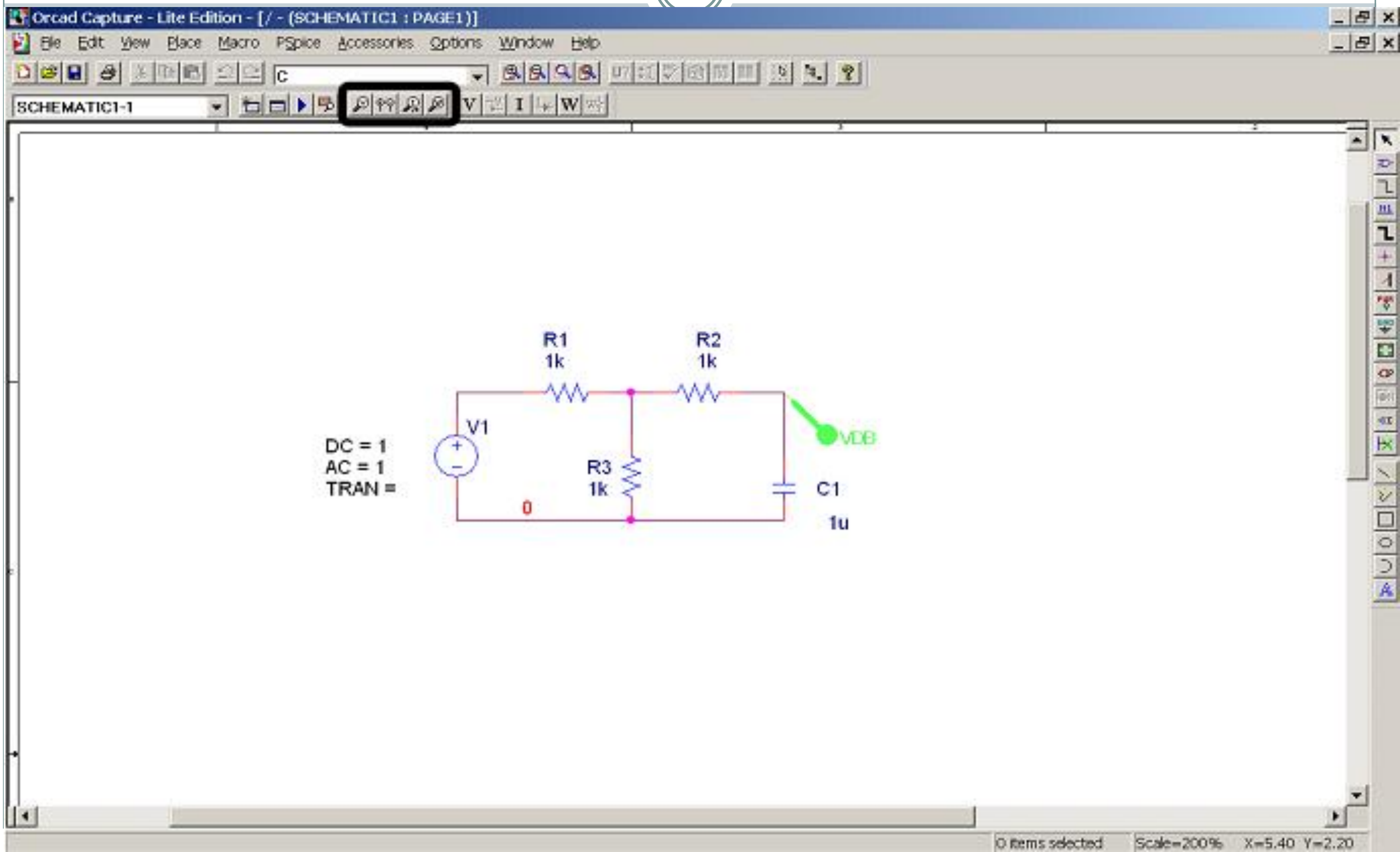
Some parts that you might want to use

Vdc	DC (constant) voltage source (default value is 0V You might want to change this)
Vsrc	Voltage source (for DC, AC and Transient simulations)
Idc	DC (constant) current source (default value is 0A. You might want to change this)
Isrc	Current source (for DC, AC and Transient simulations)
R	Resistor (default value is 1K)
C	Capacitor (default value is 1nF)
E	Voltage controlled voltage source
H	Current controlled Voltage Source
G	Voltage controlled Current source
F	Current controlled Current Source
Q2N2222	BJT frequently used for 220 and 320

Build your System

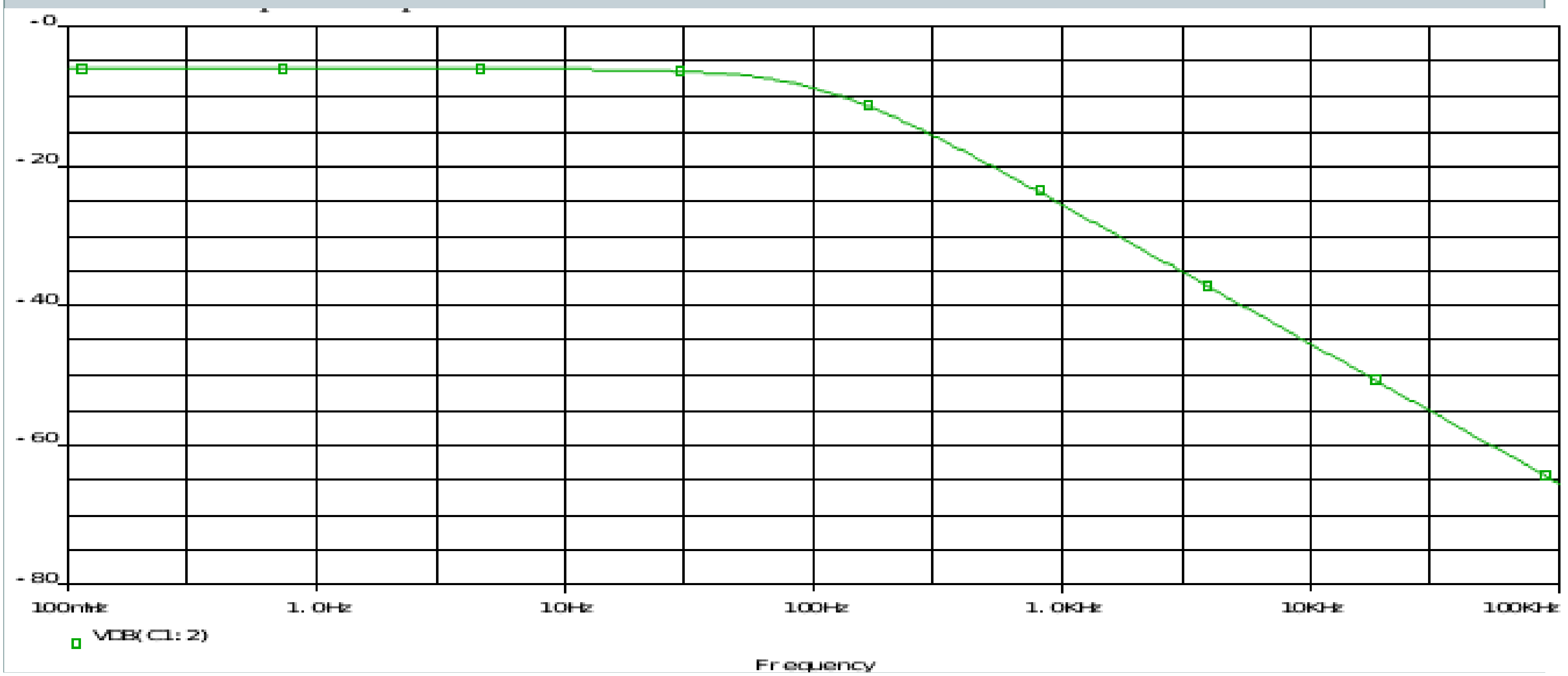


EXAMPLE

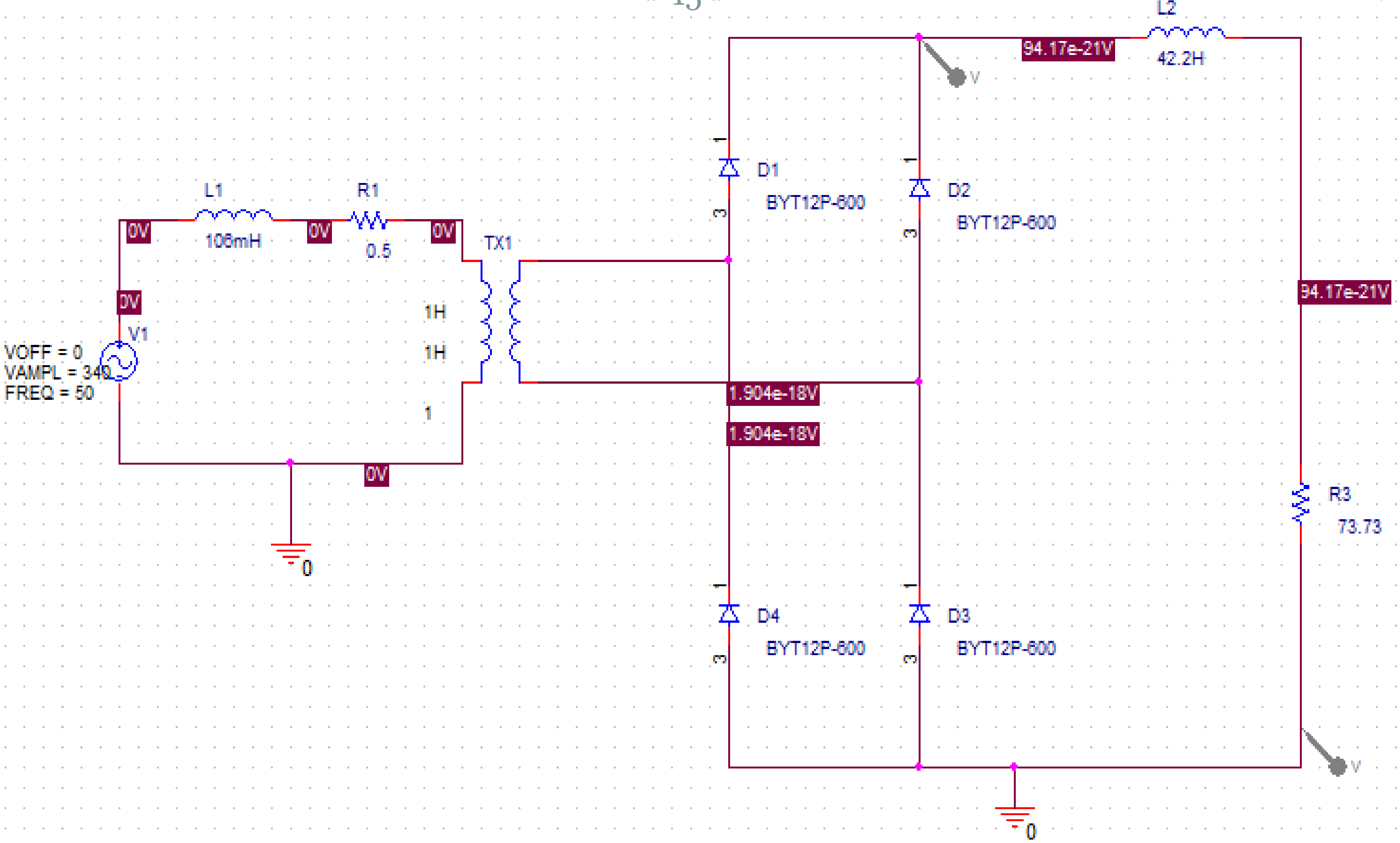


Orcad Capture - Lite Edition - [/ - (SCHEMATIC1 : PAGE1)]
File Edit View Place Macro PSpice Accessories Options Window Help
SCHEMATIC1-1
R1
1k
R2
1k
V1
DC = 1
AC = 1
TRAN =
R3
1k
C1
1u
0
VDB
0 items selected
Scale=200%
X=5.40 Y=2.20

This marker will produce plots like this one:



My own example



I will use Ps pice simulator to re explain all the previous steps .