ELEC 330 Electronic Circuits I
Tutorial and Simulations for Micro-Cap IV
by
Adam Zielinski
(posted at: http://www.ece.uvic.ca/~adam/)

This manual is written for the Micro-Cap IV Electronic Circuit Analysis Program for Macintosh Computers. The PC Version of the program is available at: www.spectrum-soft.com

1. Drawing schematics with Micro-Cap IV

The tool Menu and component list are shown in Figure T1.

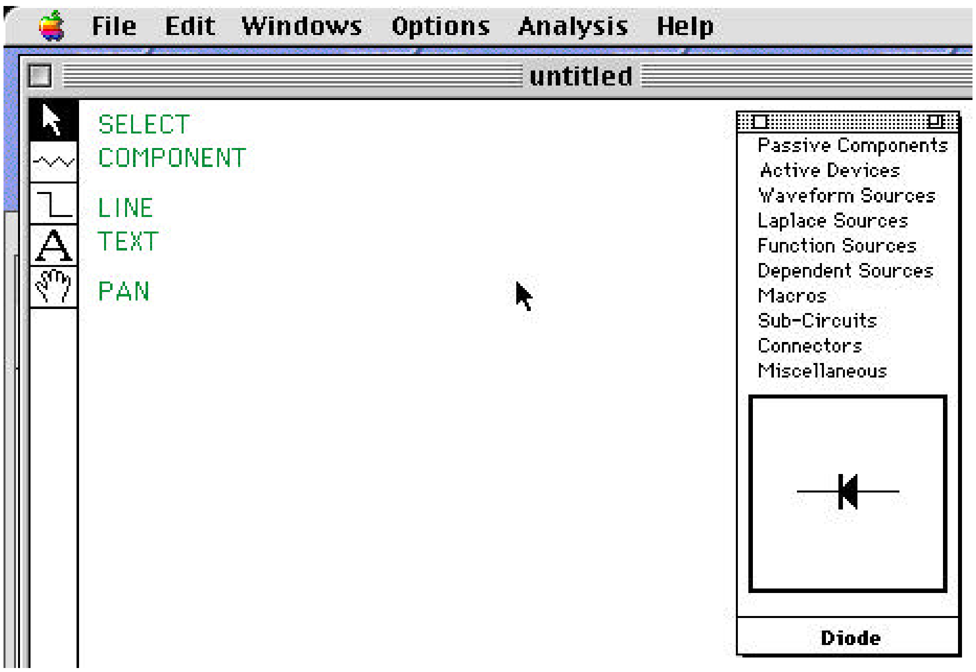


Figure T1. Tool Menu

The Tool Menu consists of five different tools:

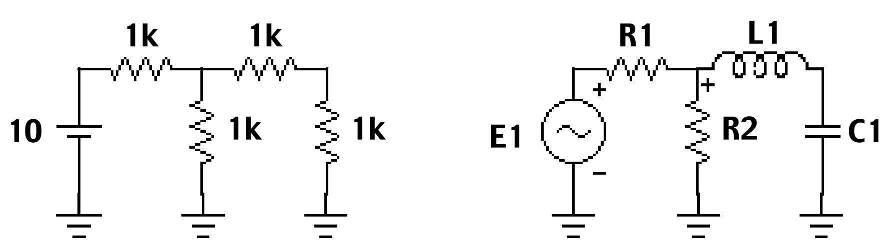
Select	The Select Tool allows selection of components, lines, text or regions to be highlighted for moving, deleting etc. This tool also enables editing of components by double clicking on them.
Component	The Component Tool allows placing the default component. The default component is selected from the Component floating menu.
Line	The Line Tool allows placing lines to connect components together.

<u>Text</u> The Text Tool allows placing DEFINE statements, MODEL statement and comments. The DEFINE and MODEL statements are instructions used in simulation to describe various characteristics of certain components.

Using the Tool menu, you can conveniently draw electronic schematics. Proceed as follows:

1. Open a new document
2. Under the Window menu, select Components
3. Select the desired component from the Components window.
4. Place it on the main drawing board. An outline of the component will appear on the sheet and a dialog box will open which may include a list of components (i.e. diodes).
5. Fill in the dialog box or select the component model from the list if appropriate. After clicking OK the component text will appear. Placing some components will also cause a MODEL statement to appear.
6. Rotate the component as desired. To do so select this component and press the “space” bar keeping the mouse pressed. Components can also be rotated during placement by holding the mouse button down while placing and pressing the spacebar.
7. Using Line tool, connect the components as needed.

<u>Exercise</u>:
Create the schematics shown in Figure T2a and 2b:



.MODEL E1 SIN (F=1MEG A=1 DC=0 PH=0 RS=1M RP=0 TAU=0 FS=0)

(a) (b)

Figure T2. Drawing of Electronic Circuits

The battery and the sine wave source are in Waveform Sources in the Component menu. To place the sine source E1, select the Sine Source and then click on the sheet where the Sine Source is to be placed. A dialog box as shown in Figure T3 will appear. Enter E1 as shown and click on OK. A second dialog box as shown in Figure T4 will then pop up, click on Yes as shown. This will generate the default MODEL statement shown in Figure T2 above.

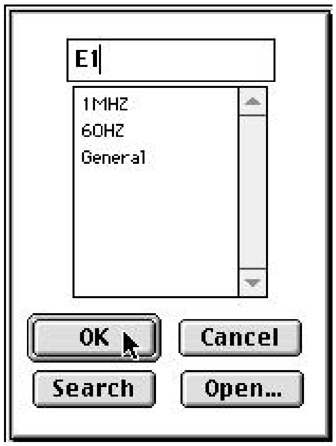


Figure T3 Place Component Dialog Box

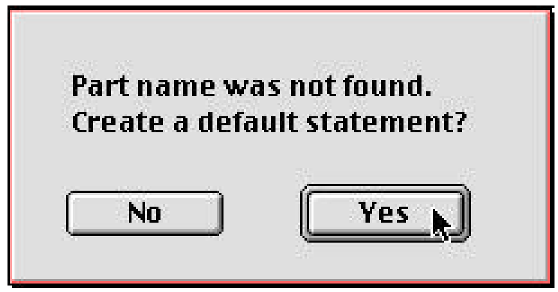


Figure T4 Create Default Statement Dialog Box

2. Copying and Printing

Graphs and schematics can be printed directly from MicroCap IV and copied to other documents. Claris Works has been provided on each Mac for writing Lab reports. Graphs and schematics can be copied to Claris Works documents by following the procedure below.

- Select File-> Save To Clipboard to copy the graphic (schematic or waveform) to the clipboard.
- Open Claris Works and select the Word Processor option or switch to the Claris Works document if one is already open. Click in the document where the graphic is to go, then select Edit -> Paste to place the graphic in the document.

Although schematics and waveforms can be directly printed and copied, other items such as Transient Analysis Limits dialog boxes (explained later) can not.

To copy and print these items use the procedure below.

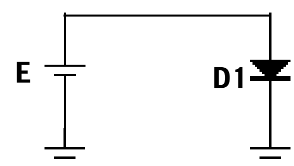
- Make sure the mouse pointer is off the graphic to be copied or it will be included in the image.
- Take a picture or snapshot of the screen by the **Apple-Shift-3** keys at the same time. You may hear the sound of a camera taking a picture.
- In the hard drive you should find a file named “Picture 1” or “Picture 2” etc. The Picture file with the largest number is the most current. Double click on the file to open it. It will open in Simple Text.

- Select the graphic by clicking on one corner of the graphic and dragging across it while holding the mouse button down. Copy the graphic to the clipboard (Apple-C).
- Paste the graphic in to a ClarisWorks document. Resize as needed.

Exercise:
Copy the schematics created in the previous exercise to a Claris Works Word Processor document using both the Copy to Clipboard and screen snapshot methods.

3. Diode Characteristics

Let's create the following circuit shown in Figure T5:



.MODEL D1 D (IS=10F RS=0 N=1 TT=0 CJO=0 VJ=1 M=500M EG=1.11 XTI=3 KF=0 AF=1 FC=500M BV=0 IBV=100P RL=0)

.DEFINE E 0

Figure T5 A circuit with a diode

The model for diode D1 was selected as a default model using the same procedure outlines in section 1. For the battery, a designator (E) is entered instead of a voltage. E is then defined by the statement: .DEFINE and its voltage set to E=0 volts. This allows components to be referenced in an Analysis Dialog Box such as the DC Analysis Dialog Box shown in figure T6.

A diode is a two-terminal device with a certain I-V characteristic. To obtain V-I characteristics of this diode we will use the DC analysis. Here is the applicable dialog box:

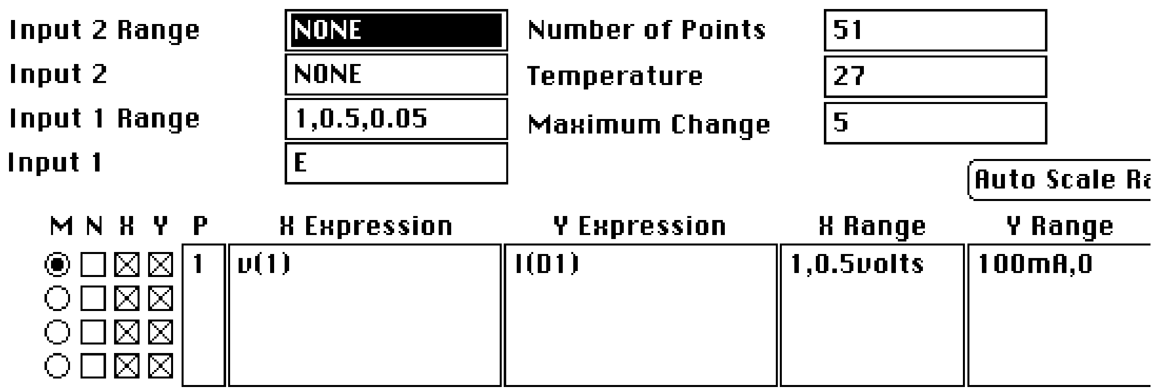


Figure T6 Dialog Box for DC analysis

In this analysis the applied voltage E (defined as Input 1) will be varied from maximum of 1 volt to minimum 0.5 volts at steps of 0.05. The corresponding diode current I (D1) is calculated and plotted on Y-axes for 0 to 100 mA range voltage applied to the diode and vs. X- axes. Here is the result:

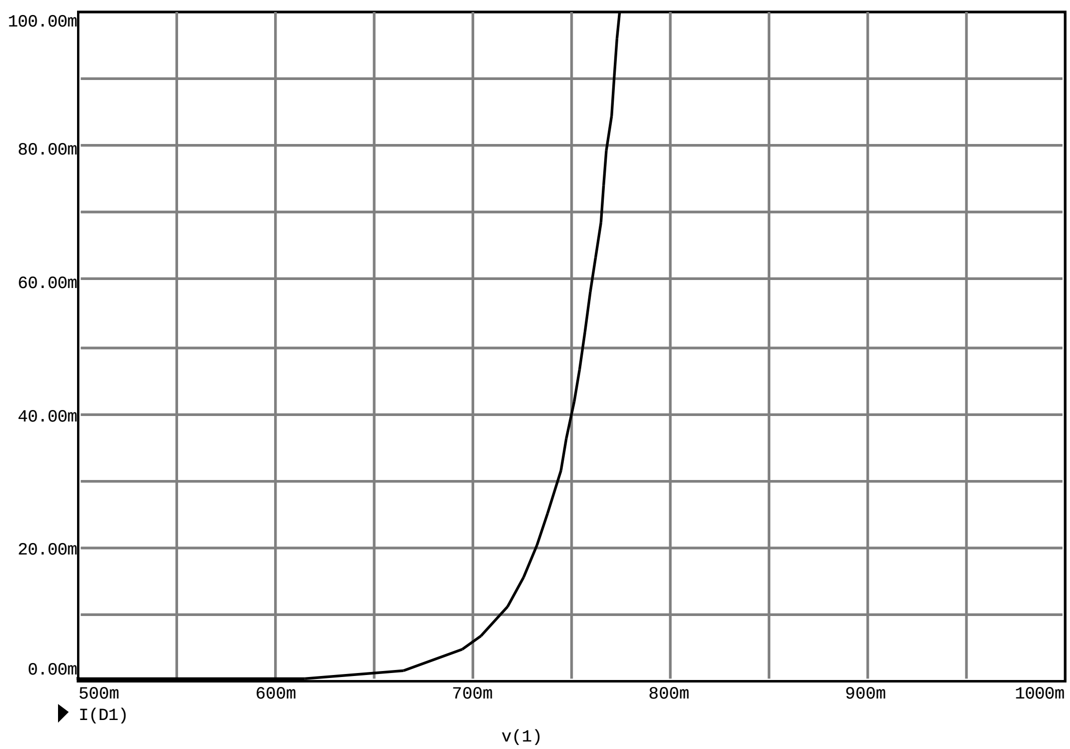


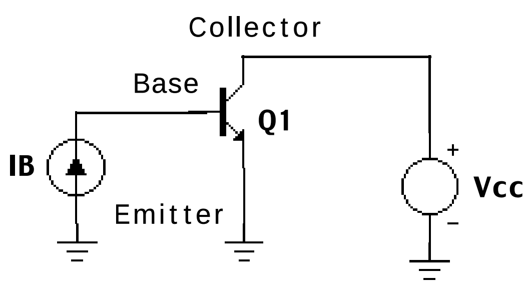
Figure T7 I-V characteristics of a diode.

Note: Simulation can be terminated any time by pressing "Apple" and "." keys. This was not necessary in this case as the simulation takes only a few seconds.

4. Bipolar Junction Transistor (BJT) characteristics.

A bipolar junction transistor (BJT) is a three-terminal device. One terminal is called <u>the base (B)</u> and is a controlling terminal. The other terminals, the <u>Collector (C)</u> and the <u>Emitter (E),</u> form a two-terminal device where the I-V characteristic is controlled by base current. The transistor is therefore characterized by family of I-V characteristics with the base current IB as the controlling parameter. DC analysis is used to generate such a family of transistor characteristics.

The transistor is an active device that requires a supply voltage Vcc to be applied between the collector and the emitter. Let's construct the following circuit using the default model for Q1:




```
.DEFINE IB 0mA
.DEFINE Vcc 0

.MODEL Q1 NPN (BF=100 BR=1 XTB=0 IS=0.1F EG=1.11 CJC=0 CJE=0
RB=0 RC=0 VAF=0 TF=0 TR=0 MJC=330M VJC=750M MJE=330M VJE=750M
CJS=0 VAR=0 NF=1 NR=1 ISE=0 ISC=0 IKF=0 IKR=0 NE=1.5 NC=2 RE=0
IRB=0 RBM=0 VTF=0 ITF=0 XTF=0 PTF=0 XCJC=1 VJS=750M MJS=0 XTI=3
KF=0 AF=1 FC=500M)
```

Figure T8 Circuit for DC analysis of BJT

We have to define base current IB generated by a current source and set it to 0 mA. Similarly, the voltage source Vcc is defined and set to 0 volts. The model for transistor Q1 is automatically generated by default.

After selecting DC analysis the dialog box shown in Figure T9 will appear.

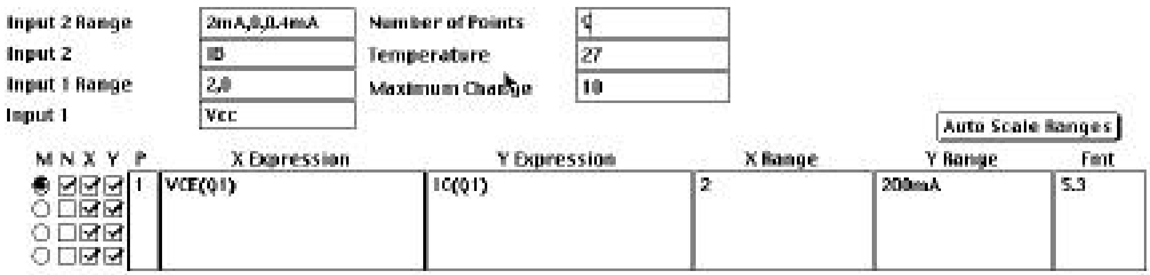


Figure T9 Limits table for DC analysis

Input 2 is defined as base current IB. It assumes values from 0 to 2 mA in steps of 0.4 mA - in a total of 6 steps. For each base current step, the Input 1 defined as Vcc voltage is swept from 0 volts to 2 volts. For a numerical display only, we can select the number of points for each Vcc sweep, say 9 points. The numerical results are generated if the column N in Figure T9 is selected. The results are shown in Table 1. V(in) = Vcc varies from 0 volts to 2 volts in 9 steps for each IB (shown here for IB=0mA and 0.4 mA only). The corresponding current IC is calculated and its value displayed.
Note: In order to paste this table to other applications, you have to save it as a text file.

Table T1. Numerical results of the simulation

Micro-Cap IV
DC Analysis Limits of Transistor Character
Date 12/26/00 Time 2:34 PM

Temperature= 27 Case= 1

v(in)	VCE(Q1) (V)	IC(Q1) (mA)
0.000	0.000	-0.000
0.250	0.250	0.000
0.500	0.500	0.000
0.750	0.750	0.000
1.000	1.000	0.000
1.250	1.250	0.000
1.500	1.500	0.000
1.750	1.750	0.000
2.000	2.000	0.000

Temperature= 27 Case= 1

v(in)	VCE(Q1) (V)	IC(Q1) (mA)
0.000	0.000	-0.396
0.250	0.250	39.724
0.500	0.500	40.000
0.750	0.750	40.000
1.000	1.000	40.000
1.250	1.250	40.000
1.500	1.500	40.000
1.750	1.750	40.000
2.000	2.000	40.000

The graphic representation of Table 1 is shown in Figure T10.

Note:
The text "IB = .. mA" has been added after simulation run.

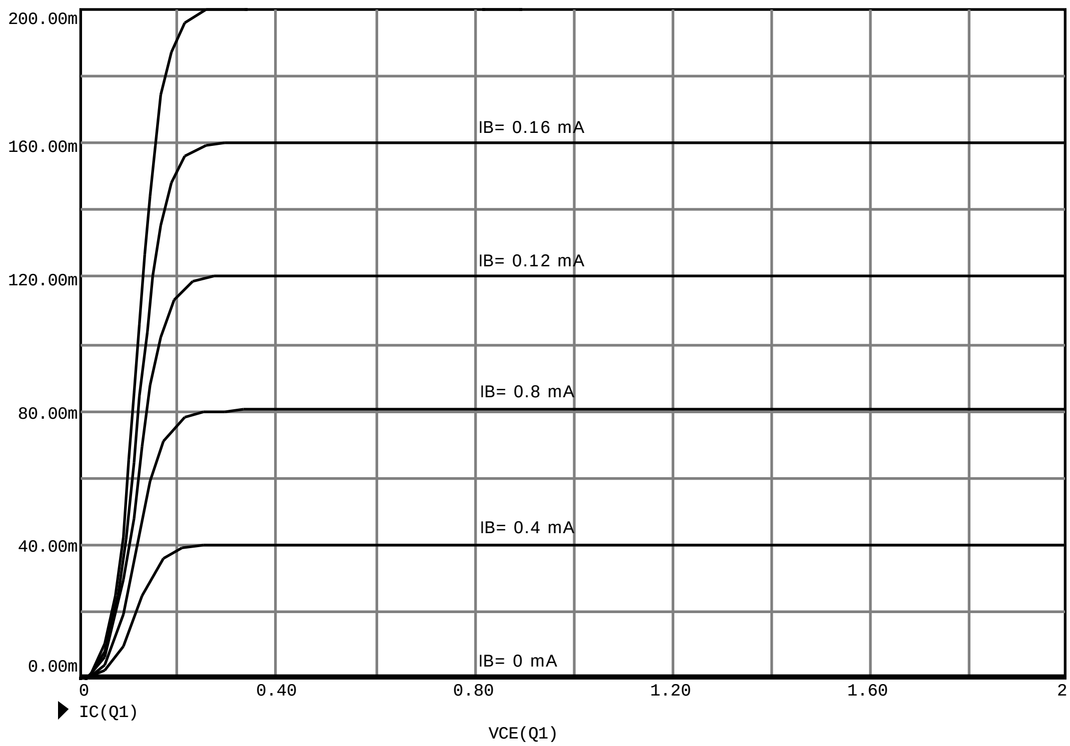


Figure T10 Transistor Characteristics

This default model represents a very idealized transistor.
Such plots are similar to that obtained by a curve tracer.

5. Transient Analysis.

Suppose we want find voltages at each node of Figure T2a. First we need to display the node numbers using Node Numbers under Option menu.

Note: If you want to copy (to clipboard) and paste the schematics to other applications, use the Text tool to place node numbers at each node (COPY does not copy them automatically). This is shown in Figure T11.

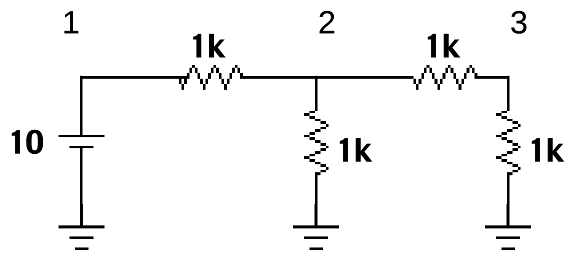


Figure T11 Circuit with a battery

You can then run a TRANSIENT analysis. This type of analysis simulates a scope and will generate voltage at each node v1, v2 and v3 vs. time. This is indicated by selection of T in Figure T12 limits box.

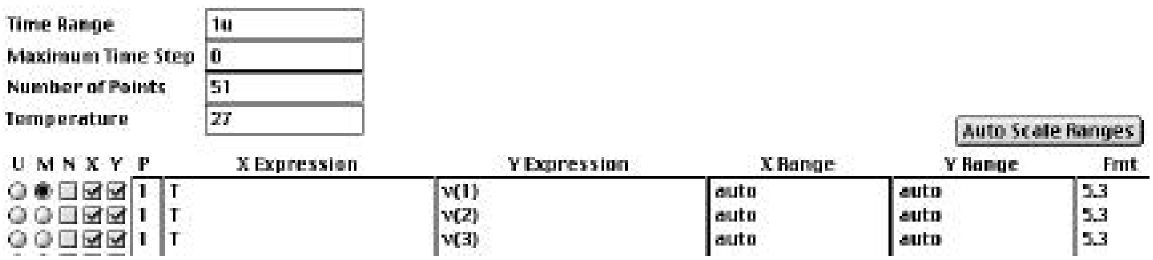
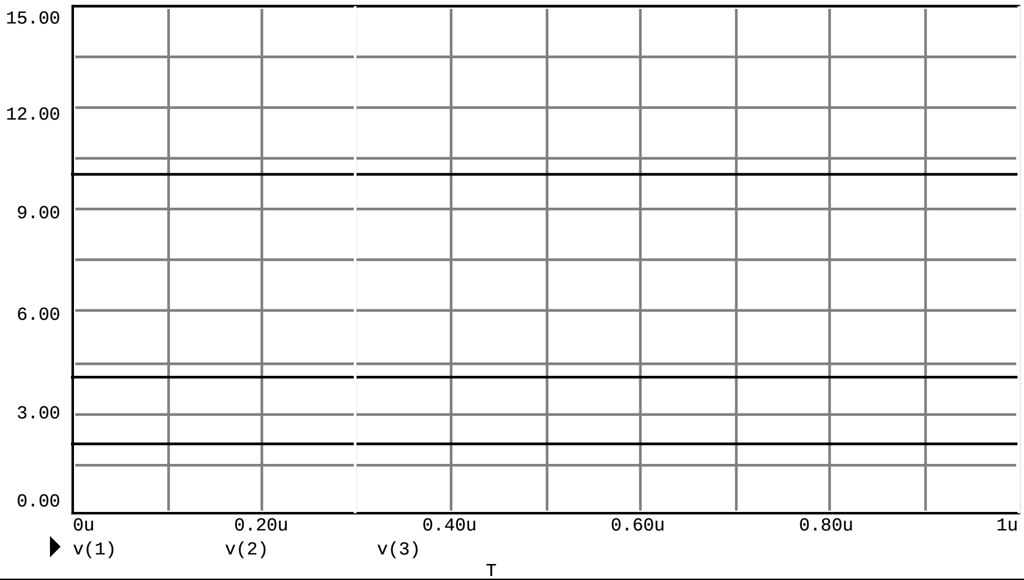


Figure T12 Dialog Box for Transient Analysis

Using Cursor mode under the Scope menu, you can generate the following plot shown in Figure T13.

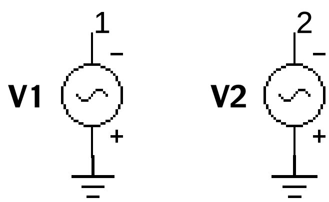


Expression	Left	Right	Delta	Slope
v(1)	10.000			

Model Parameters

Name	Parameter	Units	Default
F	Frequency	Hz	1e6
A	Amplitude	Volts	1
DC	DC level	Volts	0
PH	Phase	Radians	0
RS	Source resistance	Ohms	.001
RP	Repetition period of exponential	s	0
TAU	Exponential time constant	s	0

Select two sources and model them as in Figure T14.




```
.MODEL V1 SIN (F=100k A=1 DC=0 PH=0 RS=.001 RP=0 TAU=0 FS=0)
.MODEL V2 SIN (F=200k A=1 DC=0 PH=0 RS=.001 RP=0 TAU=0 FS=0)
```

Figure T14 Example of sinusoidal sources

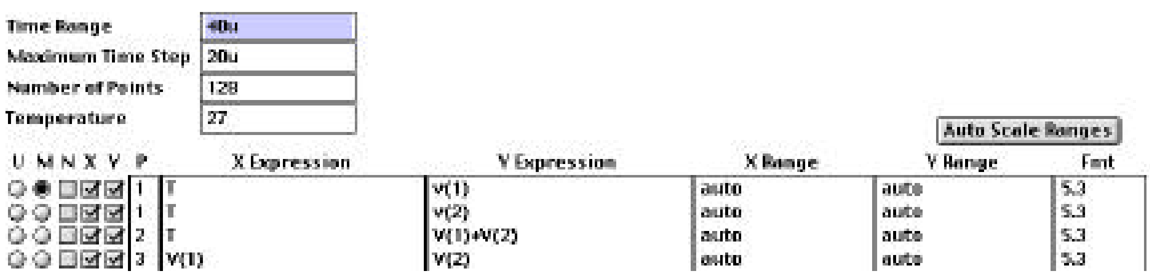


Figure T15 Dialog box for various displays

The entries in the limits box shown in Figure T15 are intended to produce three plots. One plot will display voltages waveform v (1) at node 1 and waveform v (2) at node 2. The second plot will show the sum of v (1)+ v (2). The third plot will plot v (1) vs. v (2). The last plot is similar to X-Y mode of the scope. In our case, it produces the so called Lissajous figure. The results are shown in Figure T16.

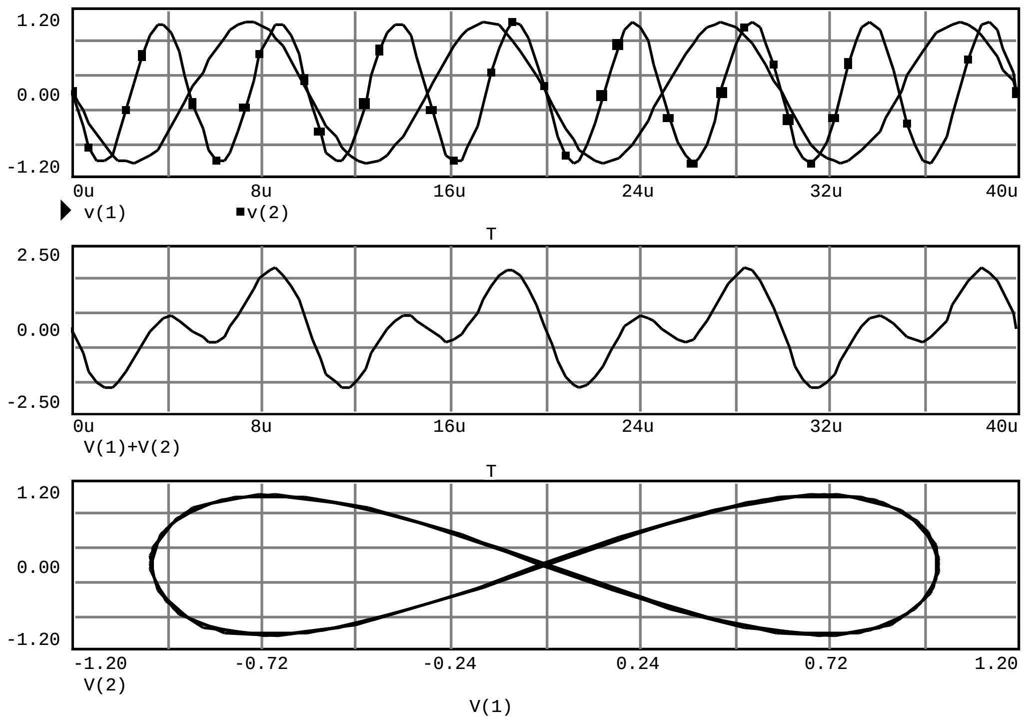
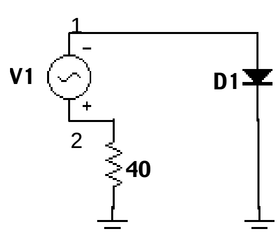


Figure T16 Various plots generated by sinusoidal sources

7. Curve Tracer

Here we will demonstrate simulation of a curve tracer. Construct the circuit shown in Figure T17. Instead of diode D1, you can insert any nonlinear two-terminal device to be investigated.



.MODEL D1 D (IS=10F RS=0 N=1 TT=0 CJO=0 VJ=1 M=500M EG=1.11 XTI=3 KF=0 AF=1 FC=500M BV=0 IBV=100P RL=0)

.MODEL V1 SIN (F=10 A=20)

Figure T17 Curve Tracer

A sinusoidal voltage is applied to the diode. The current through D1 is monitored as a voltage drop on a 40 ohms resistor. This resistor limits the maximum current through the diode (or any other nonlinear device).
We will plot the value of the diode current vs. the voltage applied to the diode. In order to get the right magnitude and the direction of this current we

have to convert the voltage at node 2 to a current through the diode as indicated in the dialog box in Figure T18.

Figure T18   Dialog box for curve tracer simulation

The simulation result is shown in Figure T19.

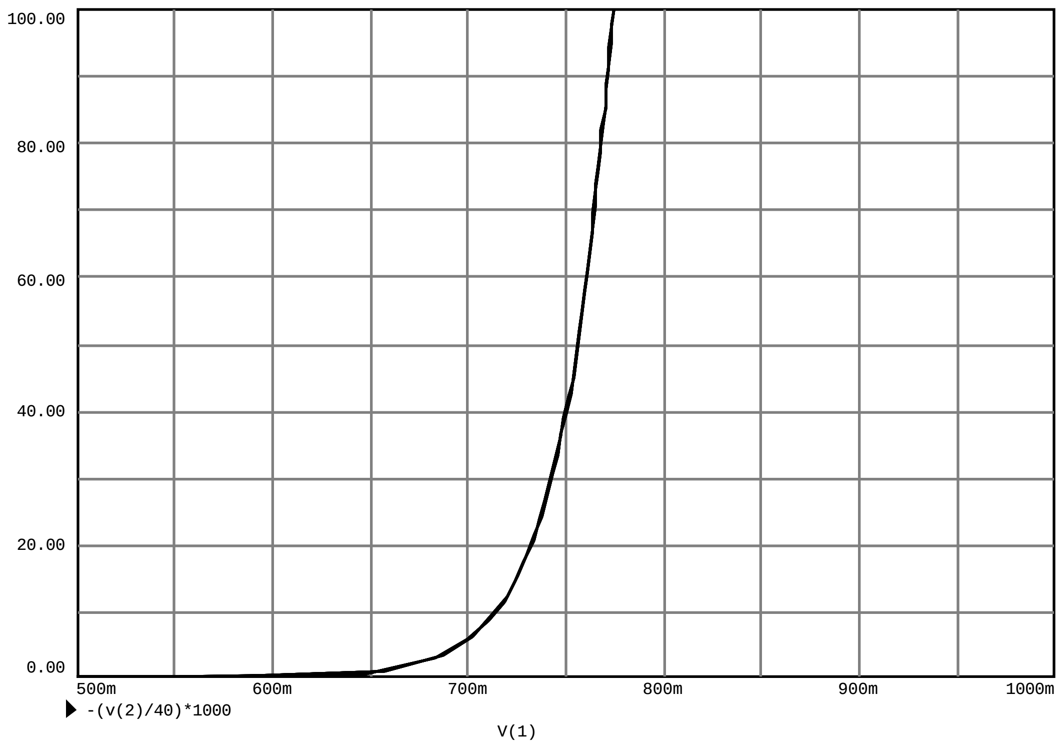


Figure T19.   Simulation results.

Note: In order to avoid transients within the diode, we used a very low sweeping frequency of 10 Hz. You can compare this result with that obtained earlier in Figure T6 for the same diode.

ELEC 330 – ELECTRONIC CIRCUITS I
SIMULATION #1
DIODES

1. Technical Information
   Several different types of diodes are available. For more information, open the file MODEL PPC to view some of the components. Locate a diode 1N4001.

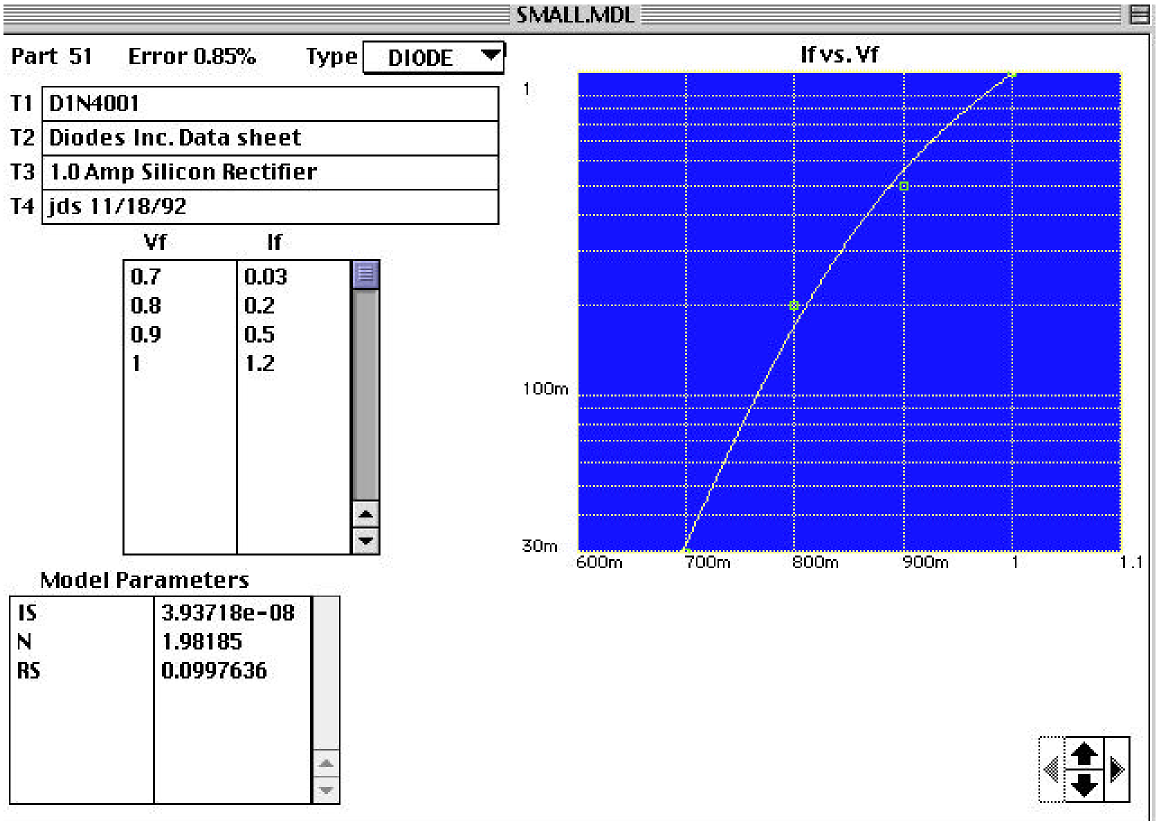


Figure 1.1. Information on diode 1N4001

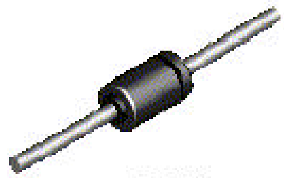
You can locate the manufacture and the designation of this diode as 1 A Silicon Rectifier. Your Laboratory manual contains some technical information on this diode.

Using a web search engine, search for 1N4001 and present some of the results. This is what can be found under:
http://www.fairchildsemi.com/pf/1N/1N4001.html

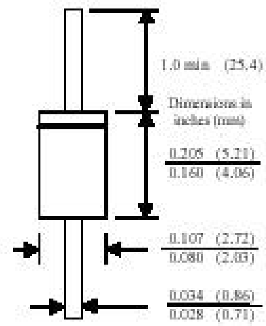
1N4001 - 1N4007

Features

- Low forward voltage drop.
- High surge current capability.



DO-41
COLOR BAND DENOTES CATHODE



1.0 Ampere General Purpose Rectifiers

Absolute Maximum Ratings* T_A = 25°C unless otherwise noted

Symbol	Parameter	Value	Units
I_O	Average Rectified Current .375 " lead length @ T_A = 75°C	1.0	A
$i_{f(surge)}$	Peak Forward Surge Current 8.3 ms single half-sine-wave Superimposed on rated load (JEDEC method)	30	A
P_D	Total Device Dissipation Derate above 25°C	2.5 20	W mW/°C
$R_{\theta JA}$	Thermal Resistance, Junction to Ambient	50	°C/W
T_{stg}	Storage Temperature Range	-55 to +175	°C
T_J	Operating Junction Temperature	-55 to +150	°C

*These ratings are limiting values above which the serviceability of any semiconductor device may be impaired.

Figure 1.2. Information on 1N4001 from the Web

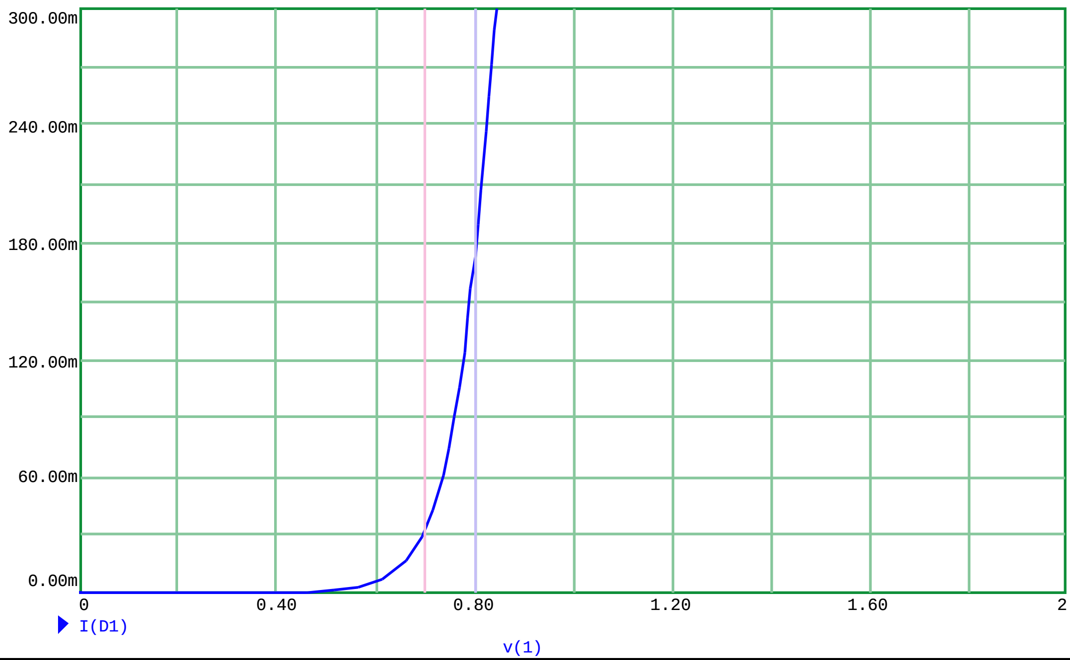
2. Diode characteristics.

.DEFINE E 0

.MODEL D1 D (IS=39.3718N RS=99.7636M N=1.98185 TT=4.76089N CJO=50.3997P VJ=408.892M M=386.137M BV=50 RL=10.0794MEG)

Figure 1.3. The circuit

Assemble the circuit as in Figure 1.3 and as in Tutorial point 2. The model for diode 1N4001 was selected; (open a suitable library if necessary) and use this model for diode D1. The result of the simulation is shown in Figure 1.4.



Expression	Left	Right	Delta	Slope
I(D1)	32.088m			

Assemble the circuit in Figure 1.5 and run the Transient Analysis (see Tutorial, point 4). After the simulation, display voltages at each node and write their values on the circuit using the text tool.

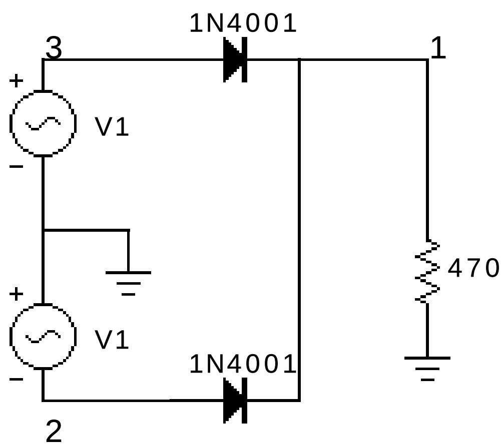
Calculate the node voltages using simplified model for the diode and compare them with the values obtained.

Answer:
Using the model for ideal diode with zero internal resistance and Vg = 0.7 volts, we will get across the 100 ohm resistor: 3.57 volts.

ELEC 330 – ELECTRONIC CIRCUITS I
SIMULATION #2
RECTIFIER

This simulation is part of preparation to the Laboratory Session #2

1. Construct the following full-wave rectifier

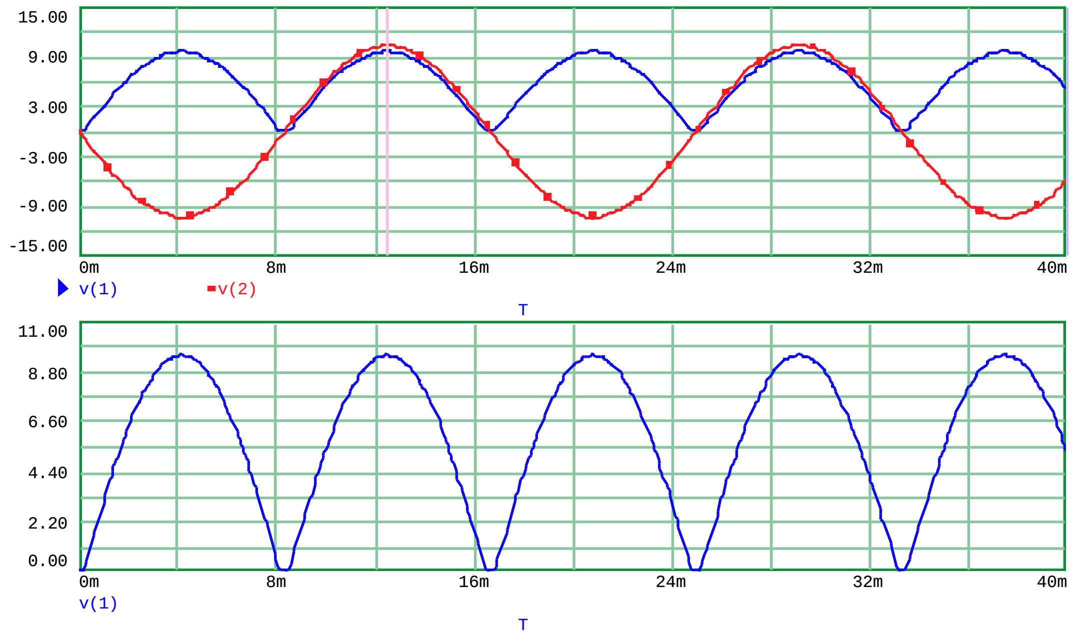


.MODEL V1 SIN (F=60 A=10.3)

.MODEL 1N4001 D (IS=10F RS=0 N=1 TT=0 CJO=0 VJ=1 M=500M EG=1.11 XTI=3 KF=0 AF=1 FC=500M BV=0 IBV=100P RL=10MEG)

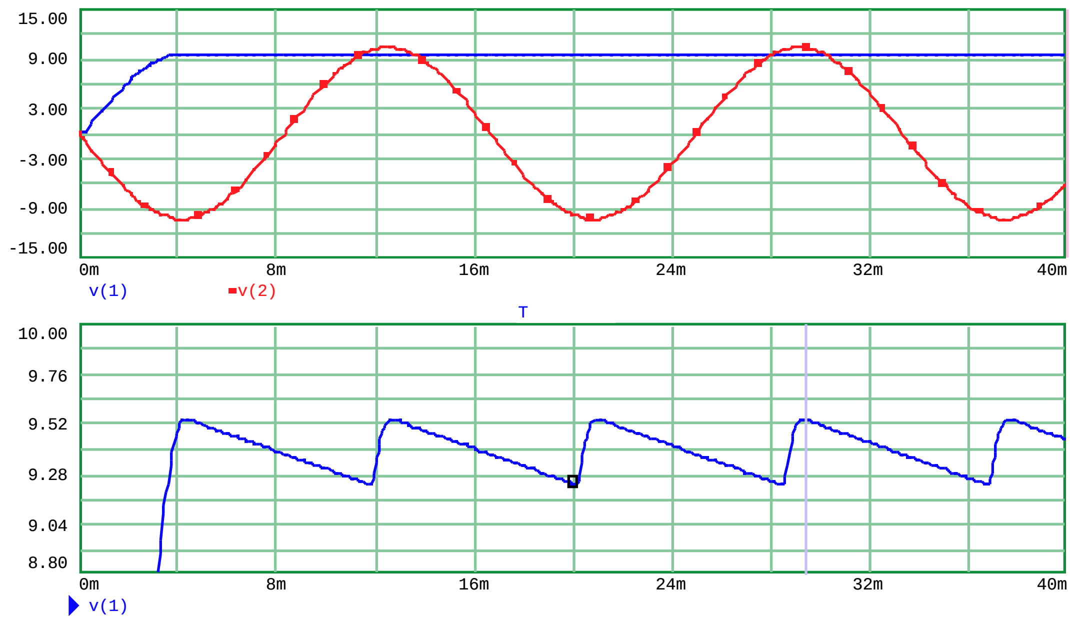
Figure 2.1 Rectifier

2. Perform Transient Analysis to obtain:



Expression	Left	Right	Delta	Slope
T	12.453m	40.000m	27.547m	1
v(1)	9.565	5.336	-4.229	-153.509
v(2)	10.298	-6.054	-16.352	-593.616

2. Connect the 470 uF filtering capacitor in parallel with the 470 ohm resistor. Perform Transient simulation to obtain:



Expression	Left	Right	Delta	Slope
v(1)	9.230			

ELEC 330 – ELECTRONIC CIRCUITS I

SIMULATION #3

TRANSISTOR, LOAD LINE AND BIASING CIRCUITS

This simulation is part of preparation to the Laboratory Session #3

1. Technical Information
   Several different types of transistors are available. For more information open the file MODEL PPC to view some of the components. Locate a transistor 2N3904 and find the information shown in Figure 3.1

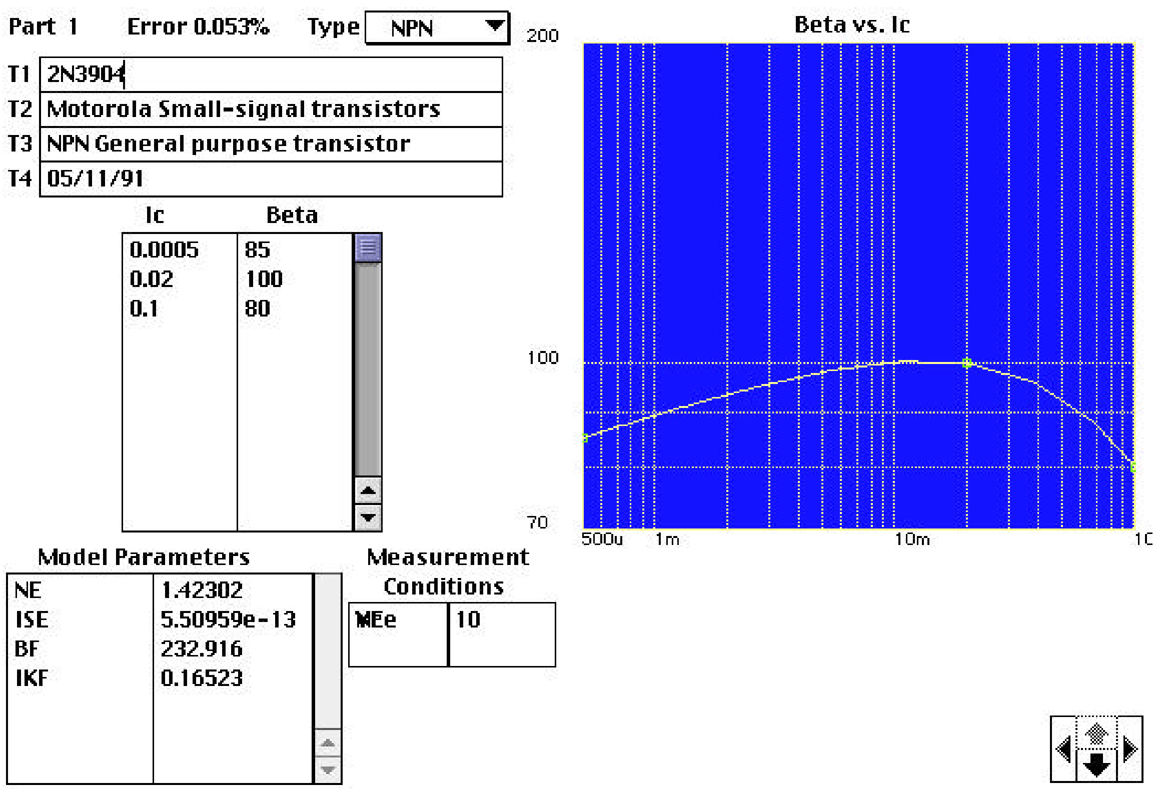
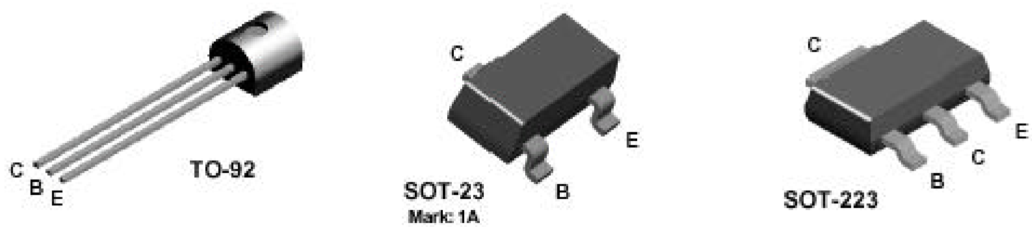


Figure 3.1 Information on transistor 2N3904

This transistor is used as a switch and general-purpose amplifier. The Laboratory Manual contains some technical information on this transistor. Using a web search engine, search for 2N3904 and present some of the results. This is what can be found under:
http://www.fairchildsemi.com/ds/2N/2N3904.pdf - see Figure 3.2

FAIRCHILD
SEMICONDUCTOR™

2N3904 MMBT3904 PZT3904



NPN General Purpose Amplifier

This device is designed as a general purpose amplifier and switch. The useful dynamic range extends to 100 mA as a switch and to 100 MHz as an amplifier. Sourced from Process 23.

Absolute Maximum Ratings*

T_A = 25°C unless otherwise noted

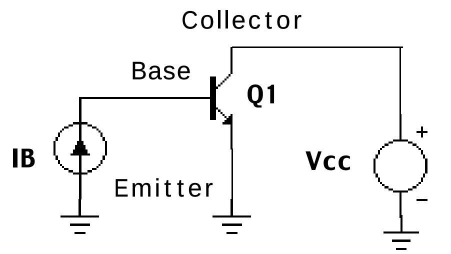
Symbol	Parameter	Value	Units
V_{CEO}	Collector-Emitter Voltage	40	V
V_{CBO}	Collector-Base Voltage	60	V
V_{EBO}	Emitter-Base Voltage	6.0	V
I_C	Collector Current - Continuous	200	mA
T_J, T_{stg}	Operating and Storage Junction Temperature Range	-55 to +150	°C

*These ratings are limiting values above which the serviceability of any semiconductor device may be impaired.

Figure 3.2 Information on 2N3904 from the Web

2. Transistor characteristics

Following Tutorial - point 3, assemble the circuit shown in Figure 3.3 to generate transistor characteristics. From the library select transistor 2N3905 and then, in the model description, replace 2N3905 by Q1.

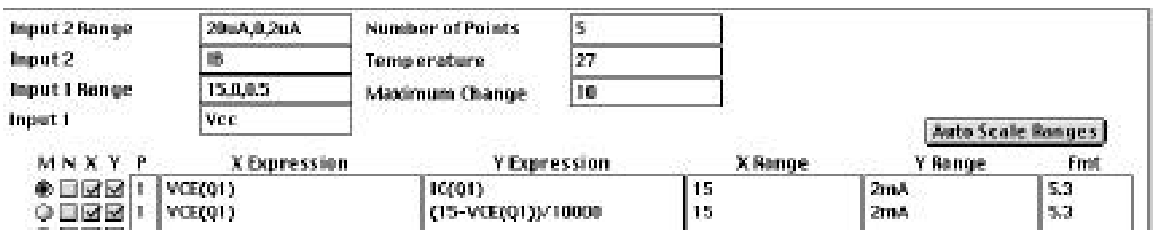


Note: Q1 Model is that of 2N3905

.DEFINE IB 0mA
.DEFINE Vcc 0

.MODEL Q1 NPN (BF=378.5 BR=2 IS=15.8478P CJC=3.62441P CJE=4.35493P
RC=1.00539U VAF=101.811 TF=666.564P TR=173.154N MJC=300M VJC=770.477M
MJE=403.042M VJE=1 NF=1.34506 ISE=61.1468P ISC=0.00155473F IKF=14.2815M
IKR=35.709 NE=2.02174 RE=1.10494 VTF=10 ITF=9.79838M XTF=499.979M
)

Figure 3. 3 Circuit to display transistor characteristics



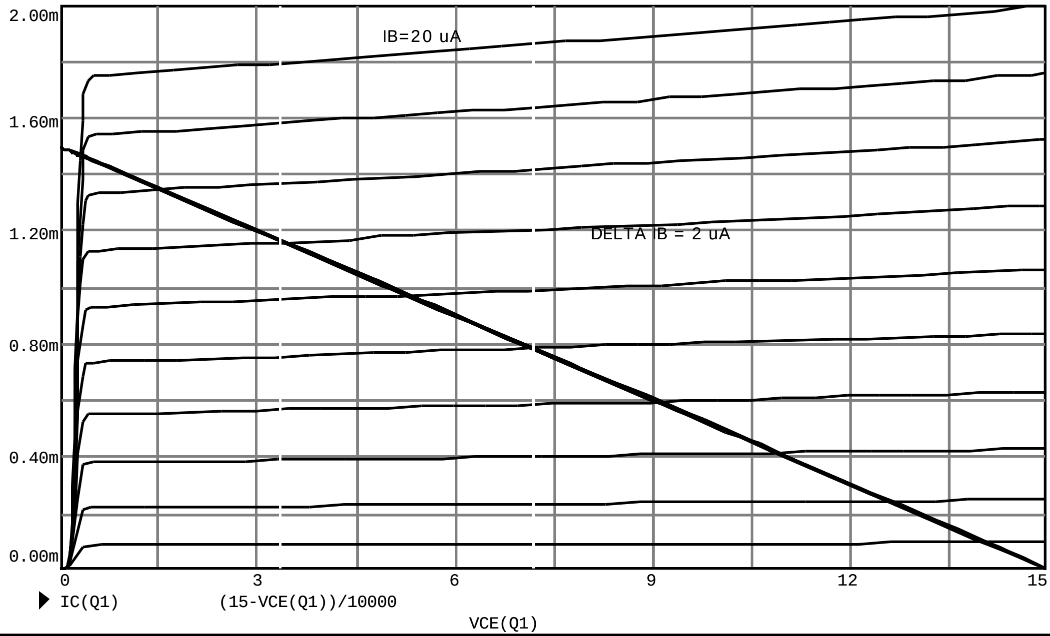
Input 2 Range	20uA,0,2uA	Number of Points	5
Input 2	IB	Temperature	27
Input 1 Range	15,0,0.5	Maximum Change	10
Input 1	Vcc		

Auto Scale Ranges

M	N	X	Y	P	X Expression	Y Expression	X Range	Y Range	Fmt
●	☐	☑	☑	1	VCE(Q1)	IC(Q1)	15	2mA	5.3
○	☐	☑	☑	1	VCE(Q1)	(15-VCE(Q1))/10000	15	2mA	5.3

Figure 3.4 Limit Box.

In the Limit Box of Figure 3.4 the base current is increased in 2 uA steps up to 20 uA. The second plot represents a dc-load line with load resistance of 10k and supply voltage of 15 volts. The results are presented in Figure 3.5.



Expression	Left	Right	Delta	Slope
IC(Q1)	0.000m			

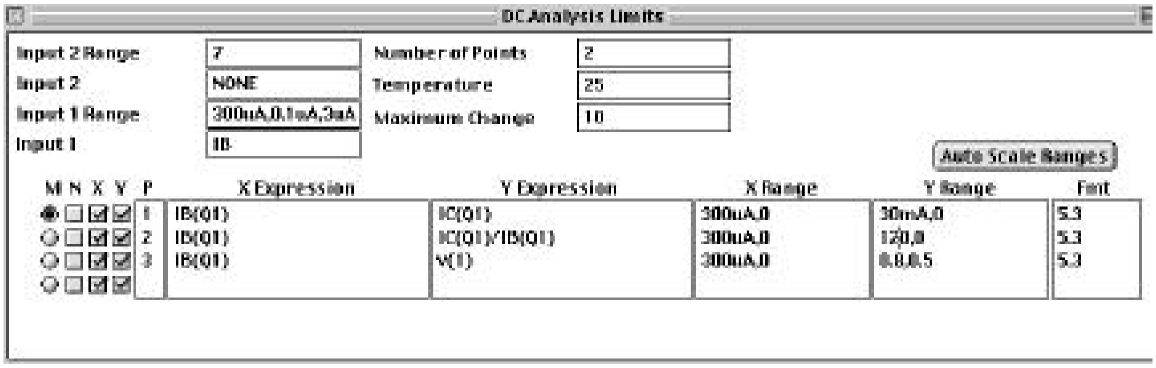


Figure 3.7 Limit Box.

We will vary the base current and observe collector current as well as base voltage v(1) as postulated in Figure 3.7. On plot 2 we plot current gain for the transistor. The results are shown in Figure 3. 8 (this Figure was imported using snap-shot)

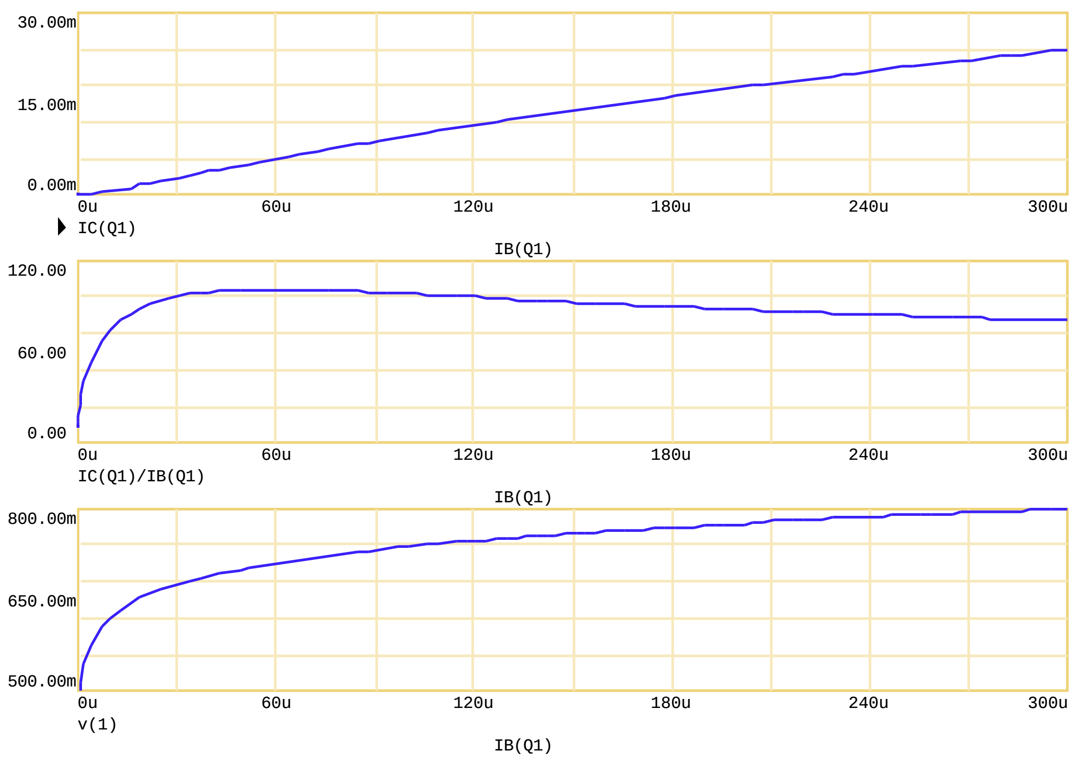


Figure 3.8 Current gain

a. Relate results of Figure 3.8 to that of Figure 3.1 and Figure 3.5.

b. Using this information, predict the voltages across the transistor if a collector resistor Rc =10 k added and Vcc=15 volts is applied. Compare your calculations with obtained results in Figure 3.5.

3. Biasing circuits.

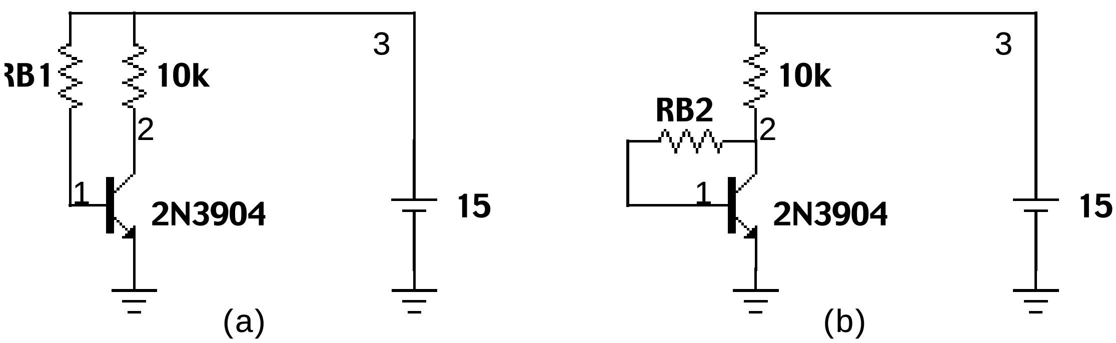
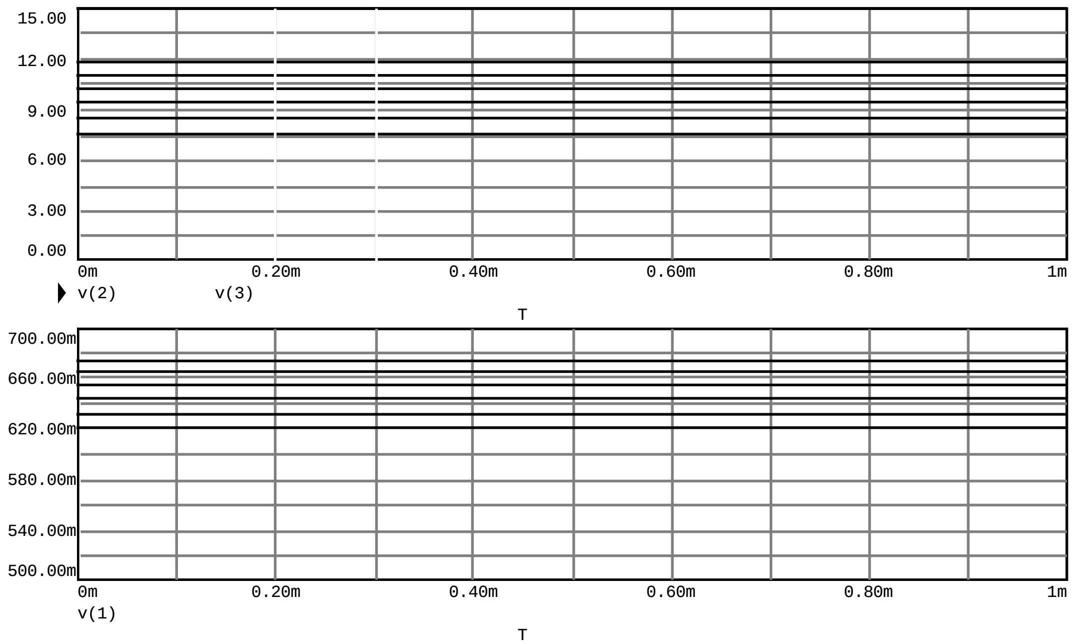
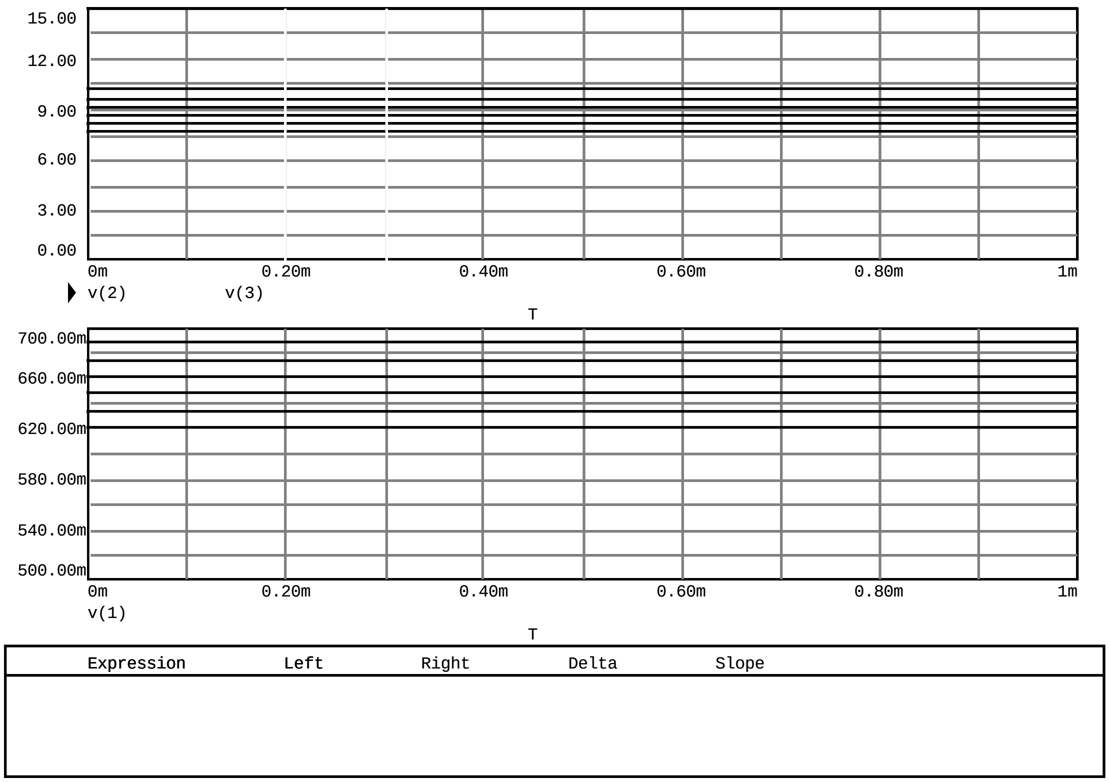


Figure 3.9 Biasing circuits

Shown in Figure 3.9 (a) and (b) are two biasing circuits. Find the value of the base resistor required to obtain 7.5 volts across the transistor (Vc =7.5 volts). Simulate both circuits and make a suitable adjustment to the calculated value of the base resistor. Run your simulation for temperature variation from 0 to 25 degrees in 5-degree increments and compare the results. The results are shown in Figure 3.9 (a) and Figure 3.9 (b)



Expression	Left	Right	Delta	Slope
T	0.200m	0.300m	0.101m	1
v(2)	11.725	11.724	-0.001	-6.599
v(3)	15.000	15.000	0.000	0

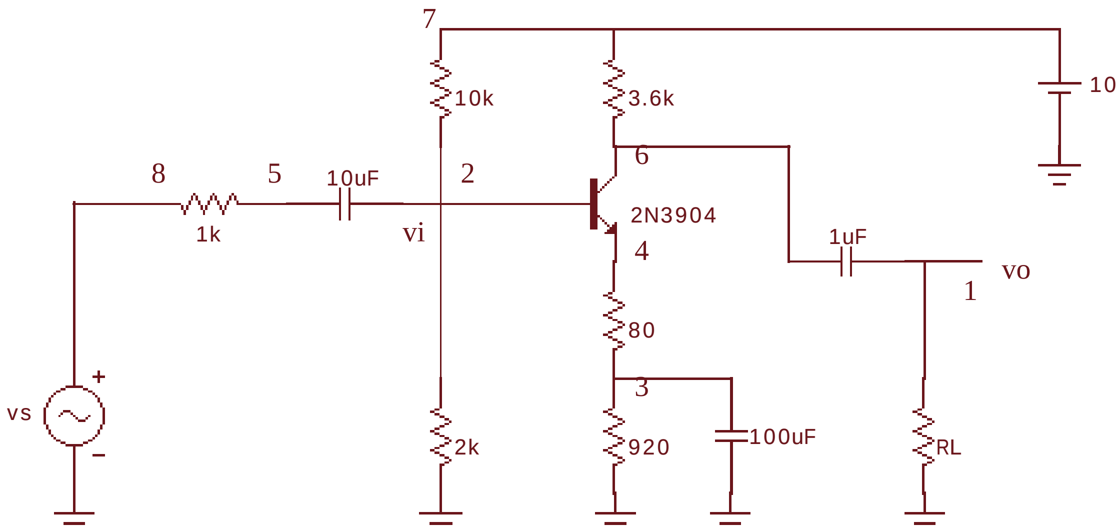

15.00
12.00
9.00
6.00
3.00
0.00
0m
0.20m
0.40m
0.60m
0.80m
1m
v(2)
v(3)
T
700.00m
660.00m
620.00m
580.00m
540.00m
500.00m
0m
0.20m
0.40m
0.60m
0.80m
1m
v(1)
T
Expression
Left
Right
Delta
Slope

ELEC 330 – ELECTRONIC CIRCUITS I

SIMULATION #4

AMPLIFIER

This simulation is part of preparation to the Laboratory Session #4



.MODEL vs SIN (F=1k A=10mV DC=0 PH=0 RS=1M RP=0 TAU=0 FS=0)
.MODEL 2N3904 NPN (BF=378.5 BR=2 IS=15.8478P CJC=3.62441P CJE=4.35493P RC=1.00539U VAF=101.811 TF=666.564P TR=173.154N MJC=300M VJC=770.477M MJE=403.042M VJE=1 NF=1.34506 ISE=61.1468P ISC=0.00155473F IKF=14.2815M IKR=35.709 NE=2.02174 RE=1.10494 VTF=10 ITF=9.79838M XTF=499.979M
)
.DEFINE RL 1MEG

Figure 4.1 Circuit of an Amplifier

In the circuit of Figure 4.1 we used SIN source with frequency 1kHz and internal resistance RS = 1 milli-ohm = 0.001 ohm. Note that the software uses incorrect symbol M for “milli-” and therefore MEG has to be used to denote Mega (rather then simply M). The transistor used was 2N3904 and has parameter BF=378 as the default value. Based on results from Simulation #4 we can assume the calculations
$b = b_{dc} = 100$. The load resistor RL has been defined and set to 1 MEG.

For the amplifier shown in Figure 4.1 calculate the following:

a. dc voltages at the base, collector and emitter.

b. The gain vo/vi with the load RL applied 9RL=1MEG can be considered an open circuit
c. Input resistance Ri
d. Output resistance Ro

Simulate the circuit, verify its operation and derive all calculated parameters.

Notes:

(a) The input resistance Ri can be evaluated based on loading effect at the input.
(b) The output resistance can be evaluated by changing the load RL from say 1MEG to 10k and observing the loading effect at the output.

The plots shown in Figure 4.2 contain sufficient information to perform the above tasks.

For Transient Run under Options use: select Operating Point. Under Stepping Option, select stepping RL from 1MEG to 10k by 980k. After simulation RUN use Cursor Mode and its menu to make a precise measurement. Note that the mouse button controls the left cursor and the button plus Shift key controls the right cursor. Experiment with different tools to find maximum and minimum values. After exiting Transient Analysis, select display voltages at each node. In this case: VB=1.64, VE=1.01 and VC=6.37 volts.

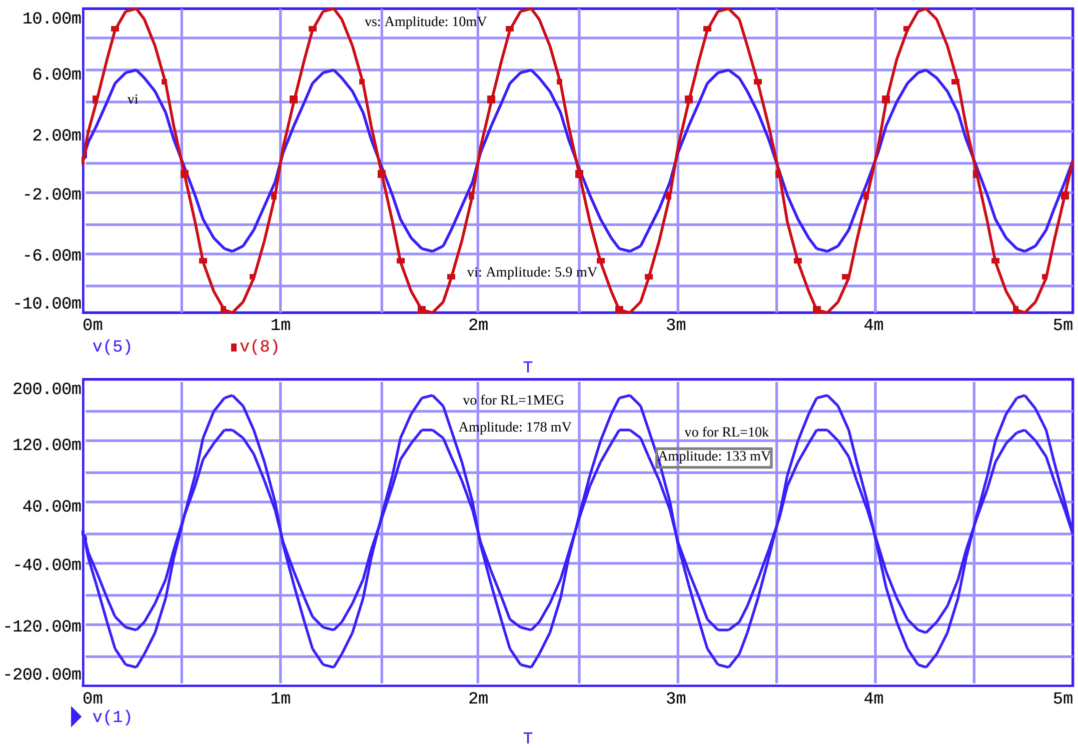


Figure 4.2 Transient Simulations

Report your findings in the Table 1.

Calculation of Ri, Ro, VS, VE, VC and Gain.

Table 4.1. Results of Simulation

Paramete r	Calculate d Value	Value obtained by Simulatio n)	Error (%)	Comments
VB (V)				
VE (V)				
VC (V)				
Ri (k)				
Ro (k)				
Gain (RL= 1MEG)				

ELEC 330 – ELECTRONIC CIRCUITS I

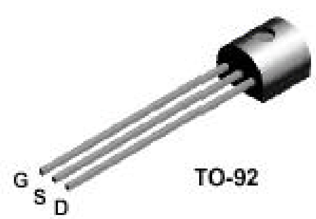
SIMULATION #5

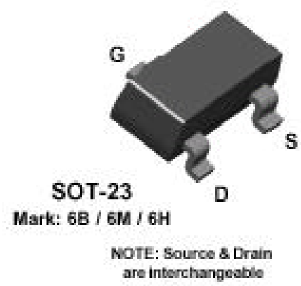
JUNCTION FIELD EFFECT TRANSISTOR

This simulation is part of preparation to the Laboratory Session #5

1. Technical Information
   Several different types of JFET transistors are available. For more information open the file MODEL HELP and locate a transistor 2N5486 to view data on this device. This transistor is used as a switch and general-purpose amplifier. The Laboratory Manual contains some technical information on this transistor. Using a web search engine, search for 2N5486 and present some of the results. Figure 5.1 shows some information obtained from:
   http://www.fairchildsemi.com/pf/2N/2N5486.html

2N5486 **MMBF5486**





N-Channel RF Amplifier

This device is designed primarily for electronic switching applications such as low On Resistance analog switching. Sourced from Process 50.

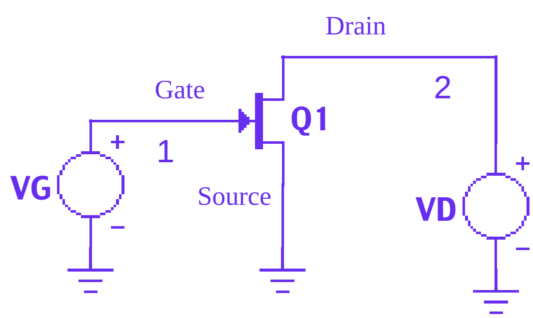
Absolute Maximum Ratings* TA = 25°C unless otherwise noted

Symbol	Parameter	Value	Units
V_{DG}	Drain-Gate Voltage	25	V
V_{GS}	Gate-Source Voltage	- 25	V
I_{GF}	Forward Gate Current	10	mA
T_J ,T_{stg}	Operating and Storage Junction Temperature Range	-55 to +150	°C

*These ratings are limiting values above which the serviceability of any semiconductor device may be impaired.

Figure 5.1 Information on 2N5486

2. Transistor characteristics
   Following Tutorial's point 3, assemble the circuit shown in Figure 5.2 to display transistor characteristics.



.DEFINE VG 0

.DEFINE VD 4

Note: Q1 Model is that of JFET 2N4586

.MODEL Q1 NJF (VTO=-4.42447 BETA=687.44U LAMBDA=20M CGS=2.50542P CGD=1.45913P PB=1.75991 KF=0.000281125F AF=497.042M)

Figure 5. 2 Circuit to display transistor characteristics

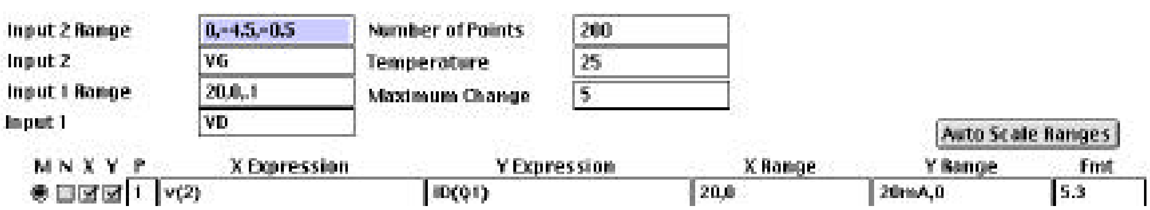


Figure 5.3 Limit Box.

The controlling parameter is gate voltage VG that will be stepped from 0 volts to –4.5 volts in –0.5 volt steps (in a total of 10 steps). This is set by the Limit Box of Figure 5.3. For each VG step, a voltage VD across the JFET is swept from 0 to 20 volts in 0.1 volts increments. The resulting family of characteristics is displayed in Figure 5.4

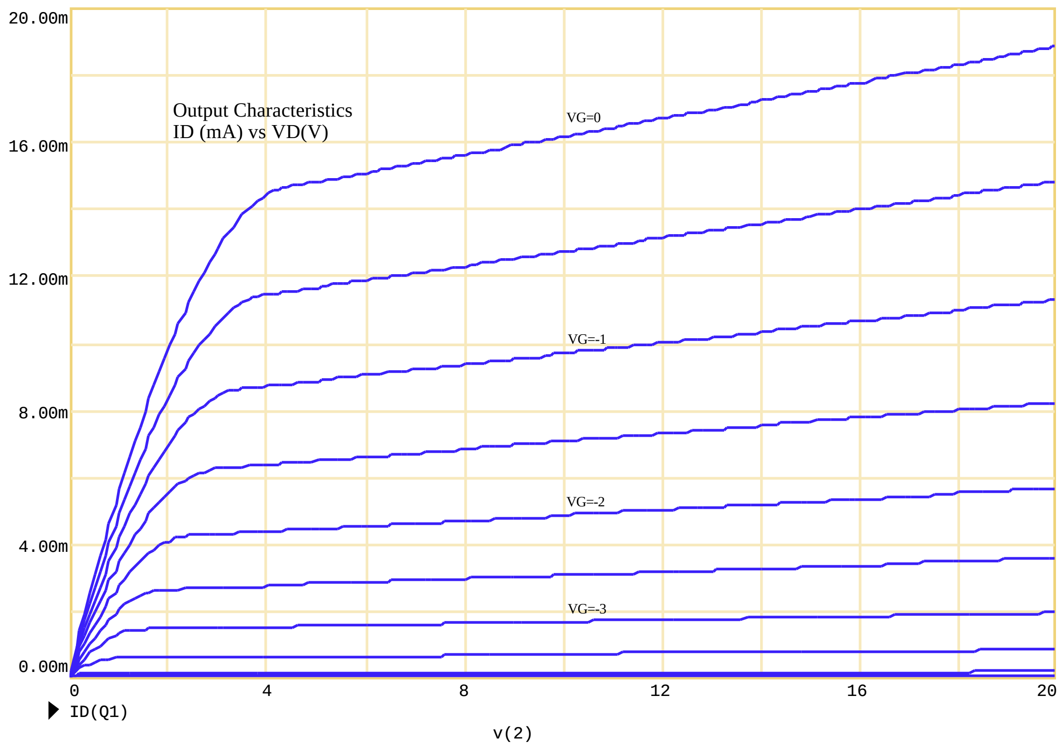


Figure 5. 4 JFET Output Characteristics

An interesting feature of JFET characteristics is that they extend to slight negative values of VD (say –100mV). At that region, ID becomes a linear function of VD with VG controlling its slope. This region is called ohmic region. Figure 5.5 shows the modified Limit Box and Figure 5.6 shows the results.

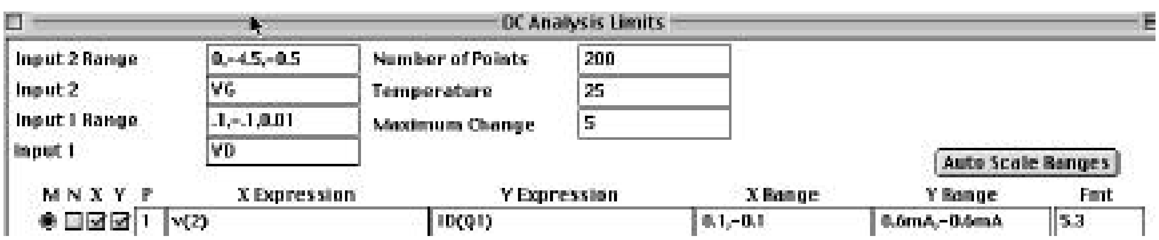


Figure 5.5 Limit Box

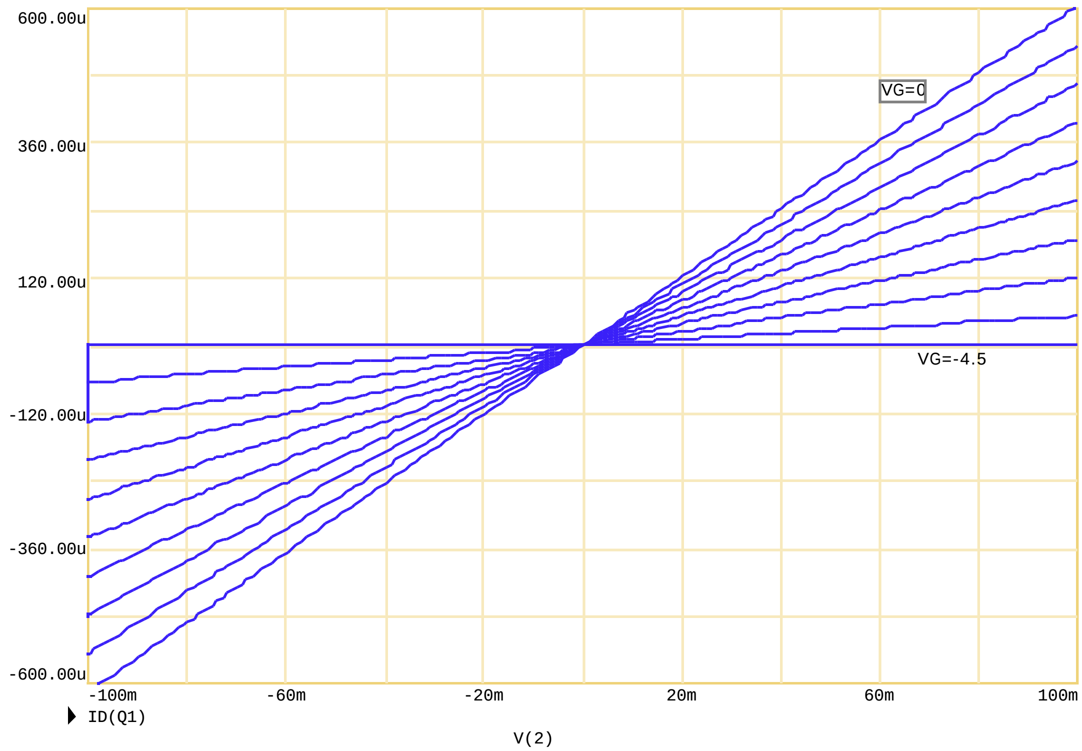


Figure 5.6 Ohmic region

For a fixed and sufficiently high VD > Vp, ID vs. VG is a parabolic curve. Figure 5.7 shows the Limit box and Figure 5.8 the results. Relate these results to those shown in Figure 5.4. Note that VD was set to 4 volts.

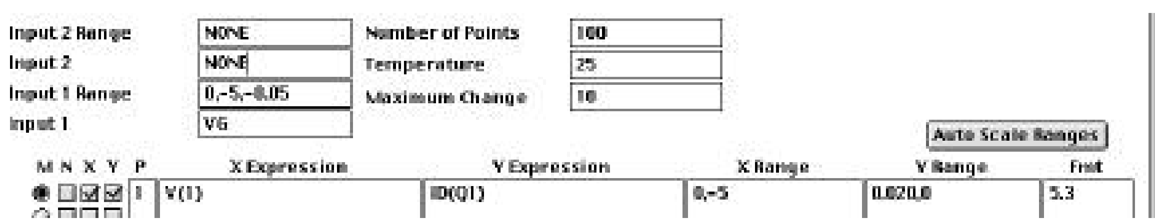


Figure 5.7 Limit Box.

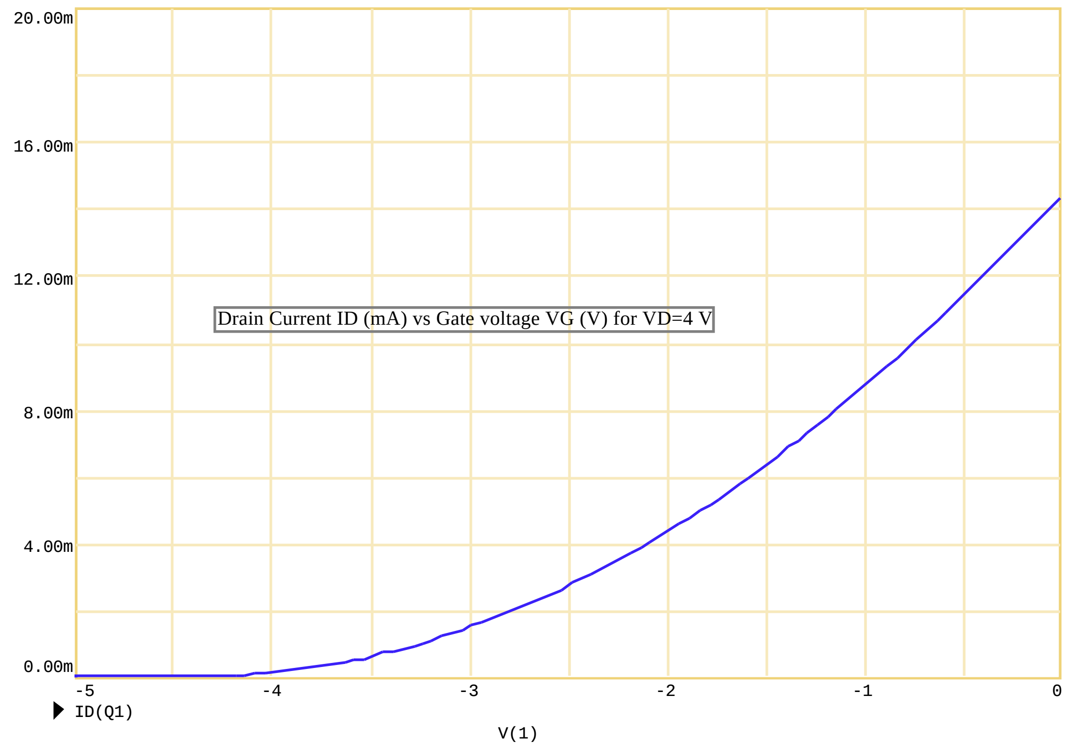
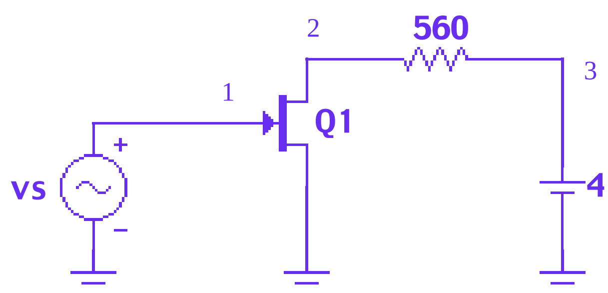


Figure 5.8 Input Characteristics

3. Parameter measurement



.MODEL vs SIN (F=1k A=.1 DC=0 PH=0 RS=1M RP=0 TAU=0 FS=0)

Note: Q1 Model is that of JFET 2N4586

.MODEL Q1 NJF (VTO=-4.42447 BETA=687.44U LAMBDA=20M CGS=2.50542P CGD=1.45913P PB=1.75991 KF=0.000281125F AF=497.042M)

Figure 5.9 Parameter Measurements

The circuit shown in Figure 5.9 can be used to measure basic parameters of JFET: I_{DSS} and V_p. The results are shown in Figure 5.10. Find these parameters.

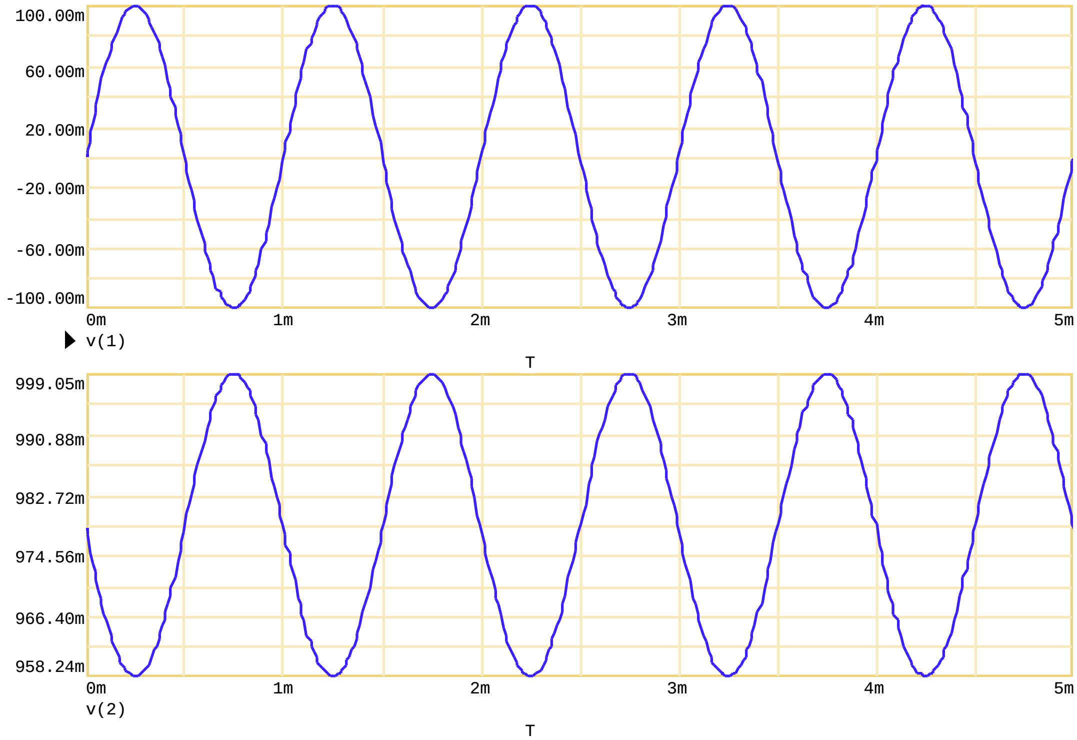


Figure 5.10 Simulation result