EEM 206
ELECTRIC CIRCUITS LABORATORY-II

EXPERIMENT BOOKLET

Prepared By:
T. Özge ÖZDİNÇ ONUR, Rıfat HACIOĞLU

BULENT ECEVIT UNIVERSITY
ENGINEERING FACULTY
ELECTRICAL-ELECTRONICS ENGINEERING DEPARTMENT

ZONGULDAK, 2024

INTRODUCTION

This booklet contains second class students of Electrical-Electronics Engineering Circuit Analysis-2 lecture notes. The subjects are:

- Frequency, Amplitude and Phase Measurement for Basic AC /RC Circuits
- Basic AC RC and RL Circuits
- Serial AC RLC Circuits
- Parallel RLC Circuits
- Power in AC Circuits
- Application of Transformer
- Balanced Three-Phase Y-Y Connected Circuits
- Balanced Three-Phase Δ-Δ Connected Circuits
- Passive Filters

Students must prepare the reports by obeying the rules. So, for the reports:

- There must be a covering letter and it must be contain name of the lecture, experiment number, the number of the group who prepared the report and their signatures.
- There must a purpose of the report and the study must be denoted clearly.
- The circuit realized during the experiment must be simulated by using PSpice and the obtained results must be given.
- The obtained results must be commented in the conclusion section.

GOOD LUCK ...

GENEL KURALLAR

- **Mazeretsiz olarak deneye devam şartını yerine getirmeyenlere DZ notu verilecektir.**
- **Deneyler gruplar şeklinde yapılacaktır.**
- **Deneyler süresi içinde bitirilmek zorundadır. Bu nedenle öğrencinin deney içeriğini dikkate alarak zaman yönetimi yapması gerekir.**
- **Deney ön hazırlıkları ilgili deneyin başında yapılması istenen kısımdır. O hafta yapılacak olan deneyin ön çalışması deneye gelmeden önce her grup üyesi tarafından ayrı olarak hazırlanmalıdır. Deney ön hazırlığı, deneylerde yer alan "PRELIMINARY STUDY" kısmında istenilenleri içermelidir. *Deneylere ön çalışma hazırlamadan gelen ya da deneyde kullanılacak malzemeleri getirmeyen öğrenciler deneyi yapamayacaklardır.***
- **Deney raporlarını her grup sadece kendi tecrübelerini kullanarak yazmalıdır. Başka bir grubun deney sonuçlarını veya başka kaynaklardan alınmış çıktıları getirmemelidir. Bu durumda, *deney rapor notu sıfır verilecektir*.**
- **Rapor zımbalanmalıdır, ayrı bir dosya kullanılmamalıdır.**
- **Raporda kurulan devreler ve kullanılan elemanlar detaylı bir şekilde verilmelidir. Tüm ölçüm ve çizimlerde kullanılan birimler mutlaka yazılmalıdır.**
- **Raporlarda bilimsel olarak anlamlı düzgün bir dil kullanılmalıdır.**
- **Hazırlanan deney raporu, belirtilen tarihte laboratuar dersinin başında teslim edilebilecektir. *Ders saatinden ya da belirtilen tarihten ve saatten sonra getirilen raporlar teslim alınmayacaktır*.**

CONTENTS

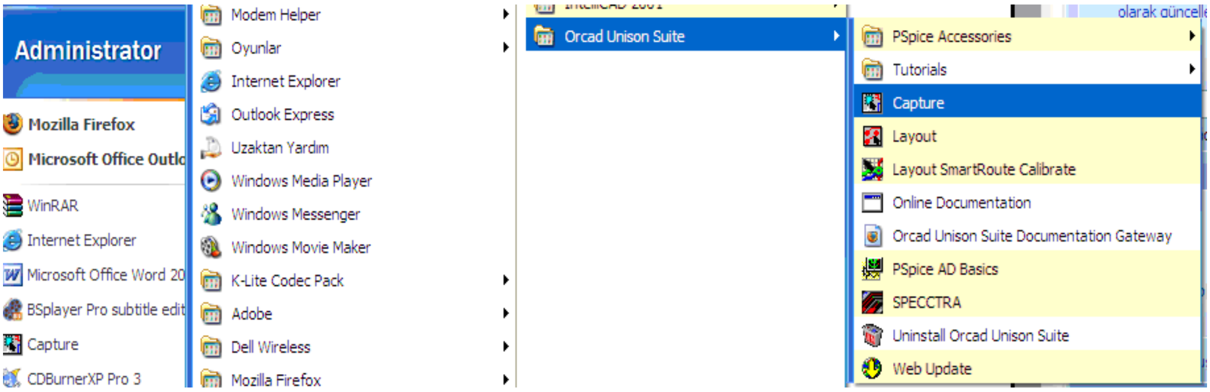
Page No

What is PSPICE ?

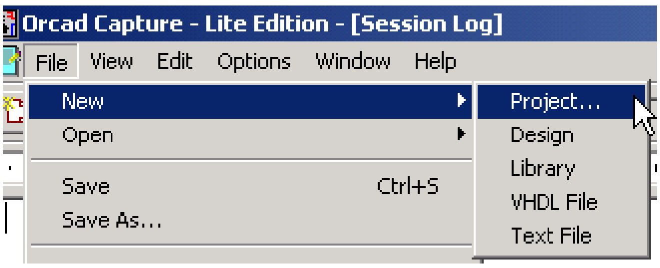
SPICE (***S***imulation ***P***rogram for ***I***ntegrated ***C***ircuits ***E***mphasis) is a program for simulating electronics circuits in the computer environment. As for PSPICE is a computer aided design and simulation program of Cadence/Orcad corporation works on Windows base.

Starting PSPICE

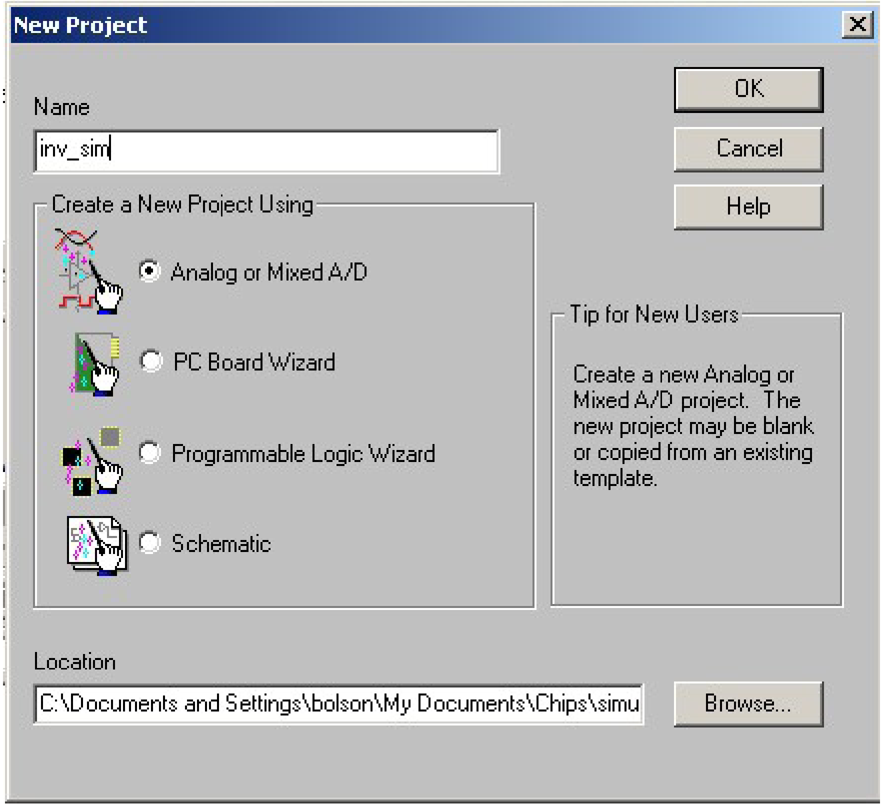
- Choose Programs>Orcad>Capture .



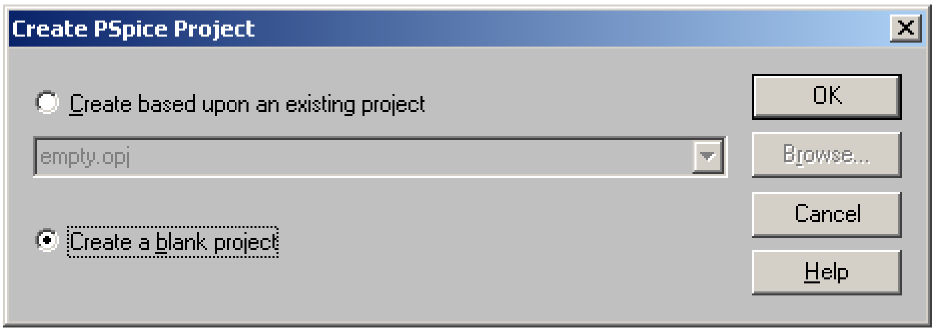
- Choose File>New>Project.



- Then choose "Analog or Mixed A/D" option from the opening window and give a name for your Project. It is suggested that **not to use** Turkish characters for your projects' names. Then click on "Browse" buton for choosing the place where your designs will be saved and choose a proper file.

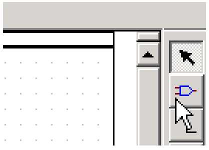


- Choose the "Create a blank project" and click on OK button.

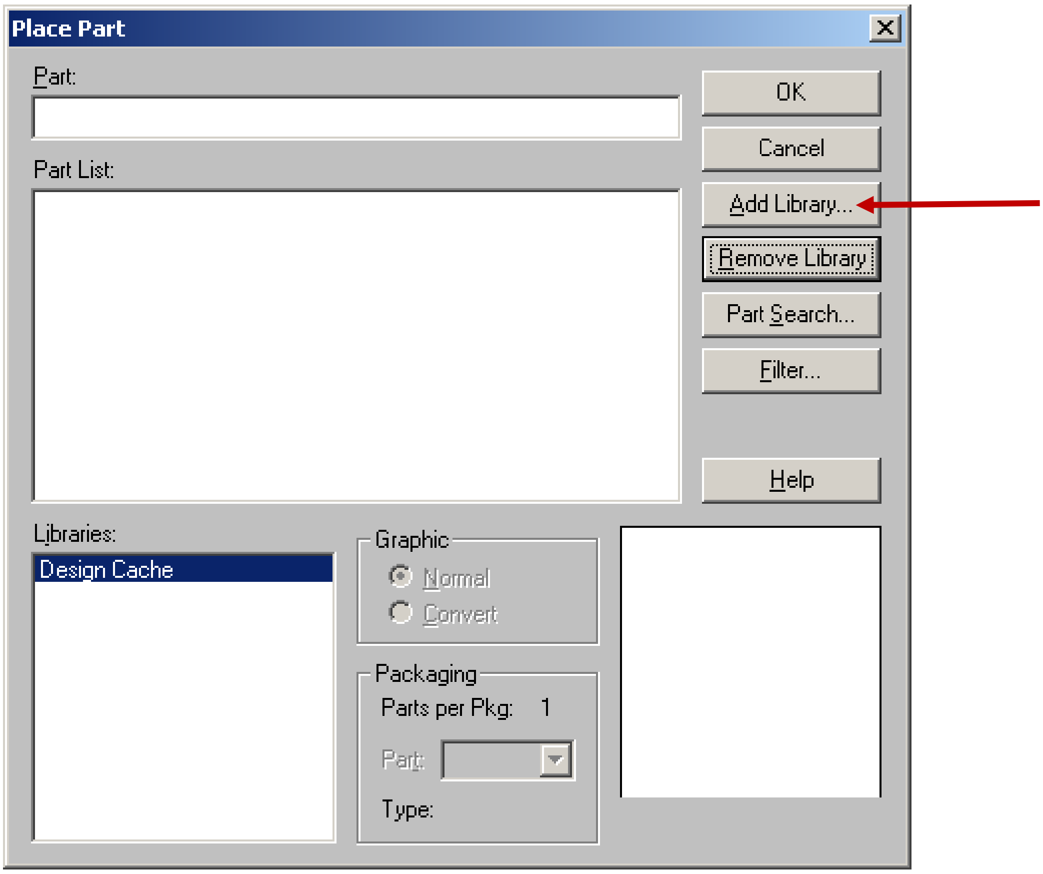


DRAWING SCHEMATIC

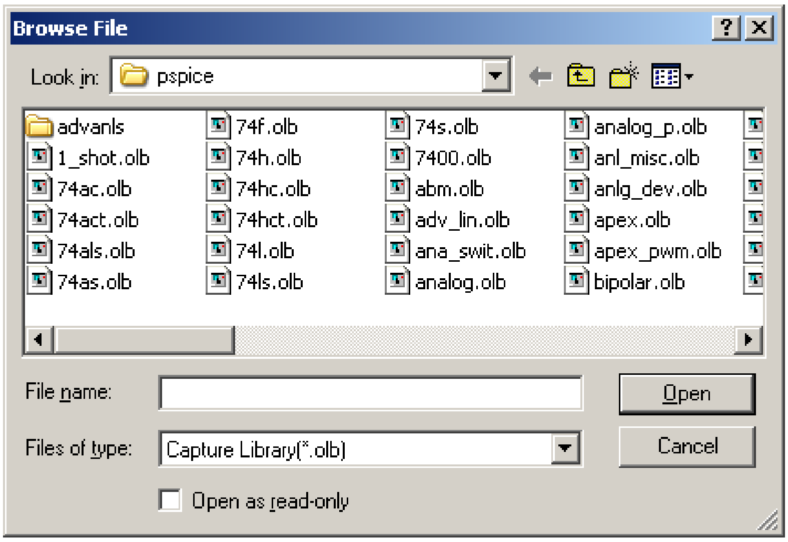
- For activating Toolbars, click on anywhere on the design screen.
- By using **"I"** and **"O"** keys on the keyboard, you can zoom in or zoom out to the design screen.
- Keystroke to the "**P**" on the keyboard and choose the icon given in the figure from the toolbar on the left.



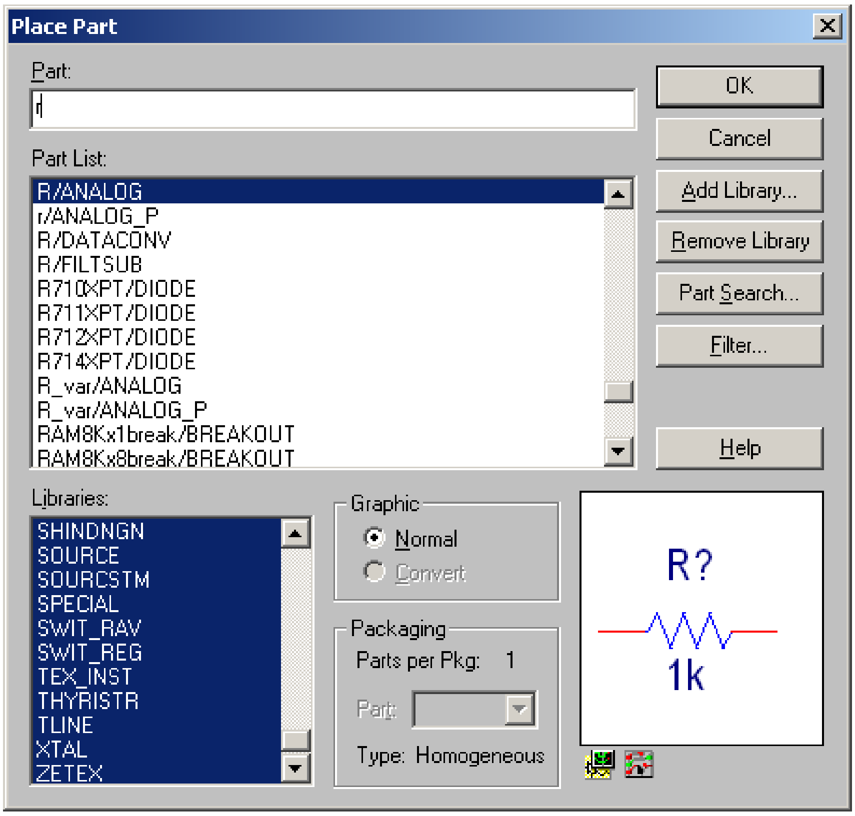
- Click on the "Add Library" button.



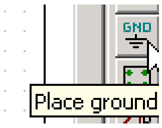
- Choose the libraries from the Pspice file, for choosing several elements please keystroke on the ctrl at the same time. (for the basic elements for example Resistor, Capasitor choose Analog.olb file and choose source.olb file for the sources.)

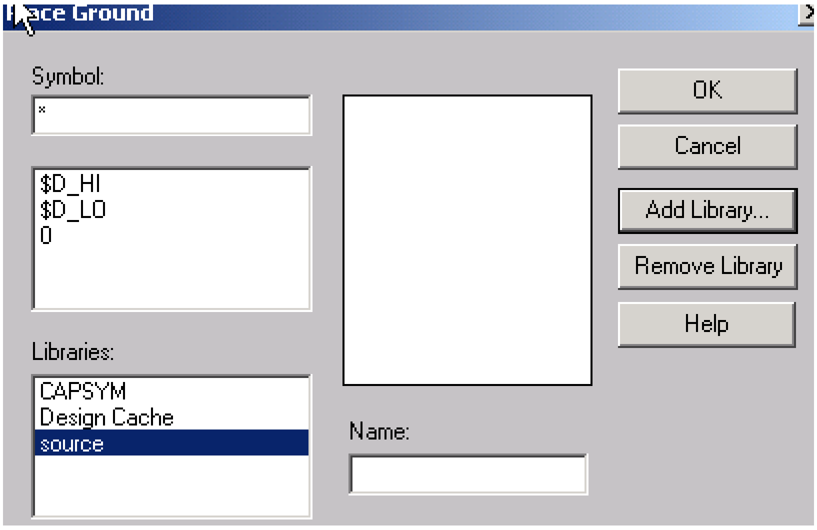


- For adding the element we want, choose the element and then click on OK.



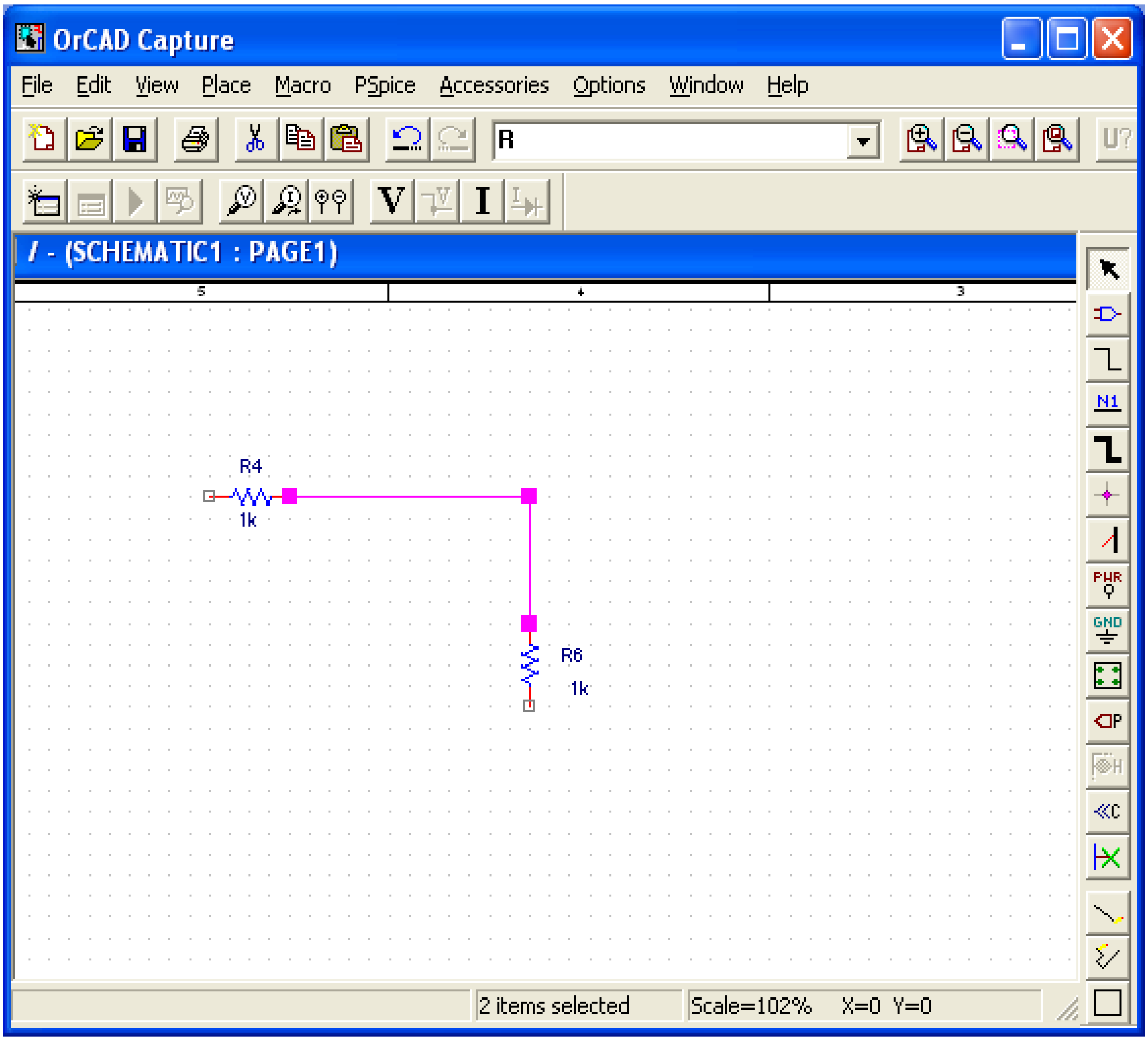
- For adding the elements such as current and voltage sources, choosing shortcut **“G”** buton and choosing the “ground” icon from the toolbar on the left is enough. Add **“CAPSYM”** ve **“Source”** libraries by using “Add Library” button on the opening window.



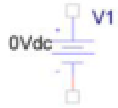
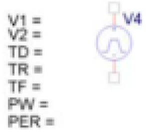
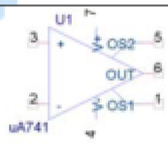
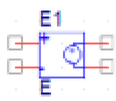


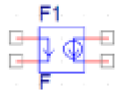
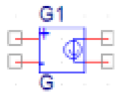
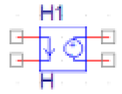
<u>IMPORTANT:</u> The name of the ground part must be "0" for Pspice.

- After adding the parts, by using **"w"** shortcut on the keyboard or cable icon on the toolbar on the right side, you can piece with the parts. Cables can be named with "Net Alias".

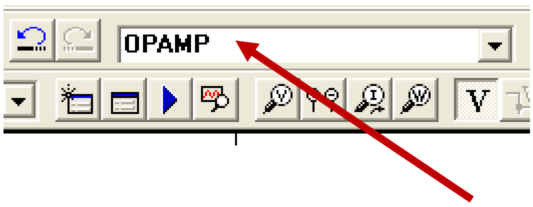

OrCAD Capture
File Edit View Place Macro PSpice Accessories Options Window Help
R
/ - (SCHEMATIC1 : PAGE1)
R4
1k
R6
1k
2 items selected
Scale=102% X=0 Y=0

FREQUENTLY USED PARTS

ELEMENT	ELEMENT NAME/LIBRARY	SYMBOL
DC Voltage Source	VDC / Source	
DC Current Source	IDC /Source	
AC Voltage Source	VAC / SOURCE	
Sine Wave Source	VSIN / SOURCE	
Triangle Save Source Square Wave Source	VPULSE / SOURCE	
Ground (reference voltage)	0 / SOURCE	
Resistor	R / ANALOG	
Capacitor	C / ANALOG	
Inductor	L / ANALOG	
741 OpAmp	uA741 / EVAL	
Diode	D1N4148 / EVAL	
Zener Diode	D1N5232 / EVAL	
npn Bipolar Junction Transistor	Q2N3904 / BIPOLAR	
Voltage Source with E Voltage-Control	E/ANALOG	

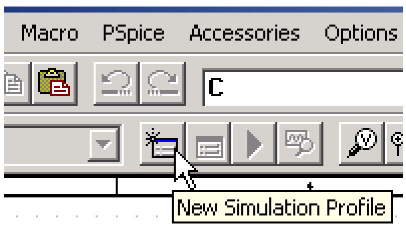
Current Source with F Current Control	F/ANALOG	
Current Source with G Voltage- Control	G/ANALOG	
Voltage Source with H Current-Control	H/ANALOG	

- You can obtain the parts on the list or all the parts easily by writing on the following box if their librariesare added.

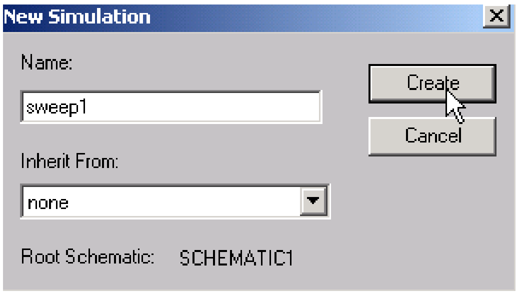


SIMULATION AND GRAPHIC ADJUSTMENTS

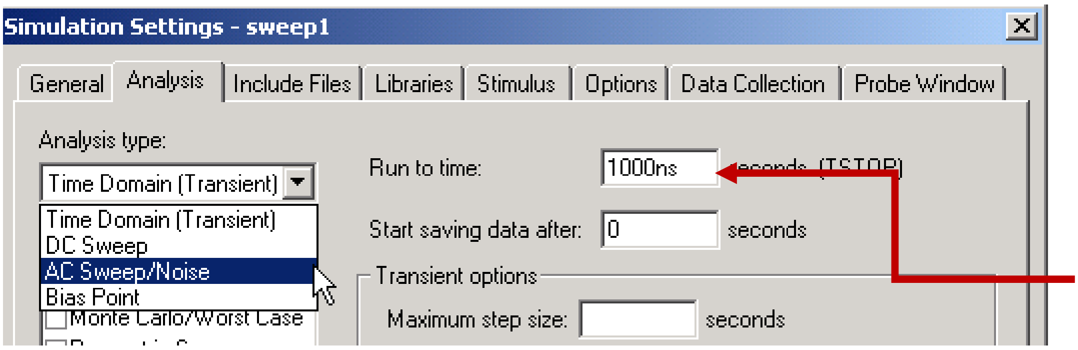
- It is necessary doing simulation adjustments after the desired circuit are set up. For this, it is enough to click on the simulation buton shown in the following.



- By using "New Simulation" window, you can give any name for your simulation. Do not use Turkish characters.

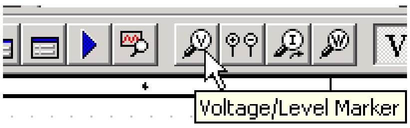


- Choose the proper analysis type by using “Analysis Type” list in the “Analysis” tab.

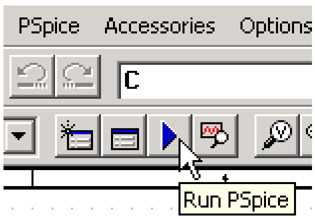


Enter the time value which determines what time the simulation will take.

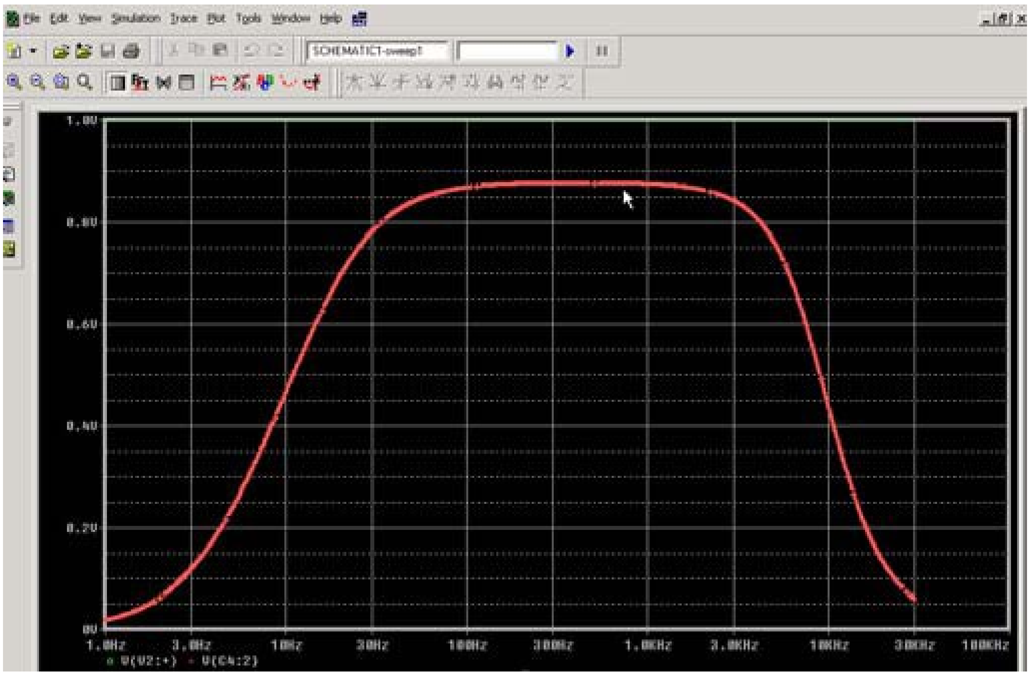
- The nodes wanted to be watched are signed by using “Voltage/Level Marker” .

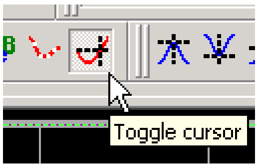


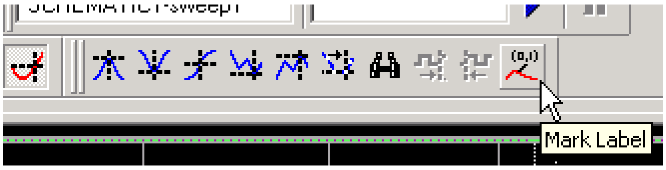
- Simulation is started by clicking on the Run Spice.



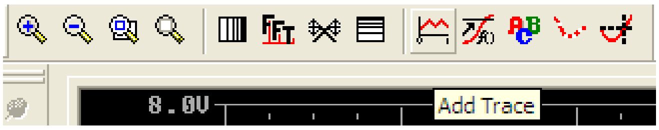
- For signing the wanted points in the graphic, you must find the point by using “toogle cursor” and then you can sign it with “mark” .



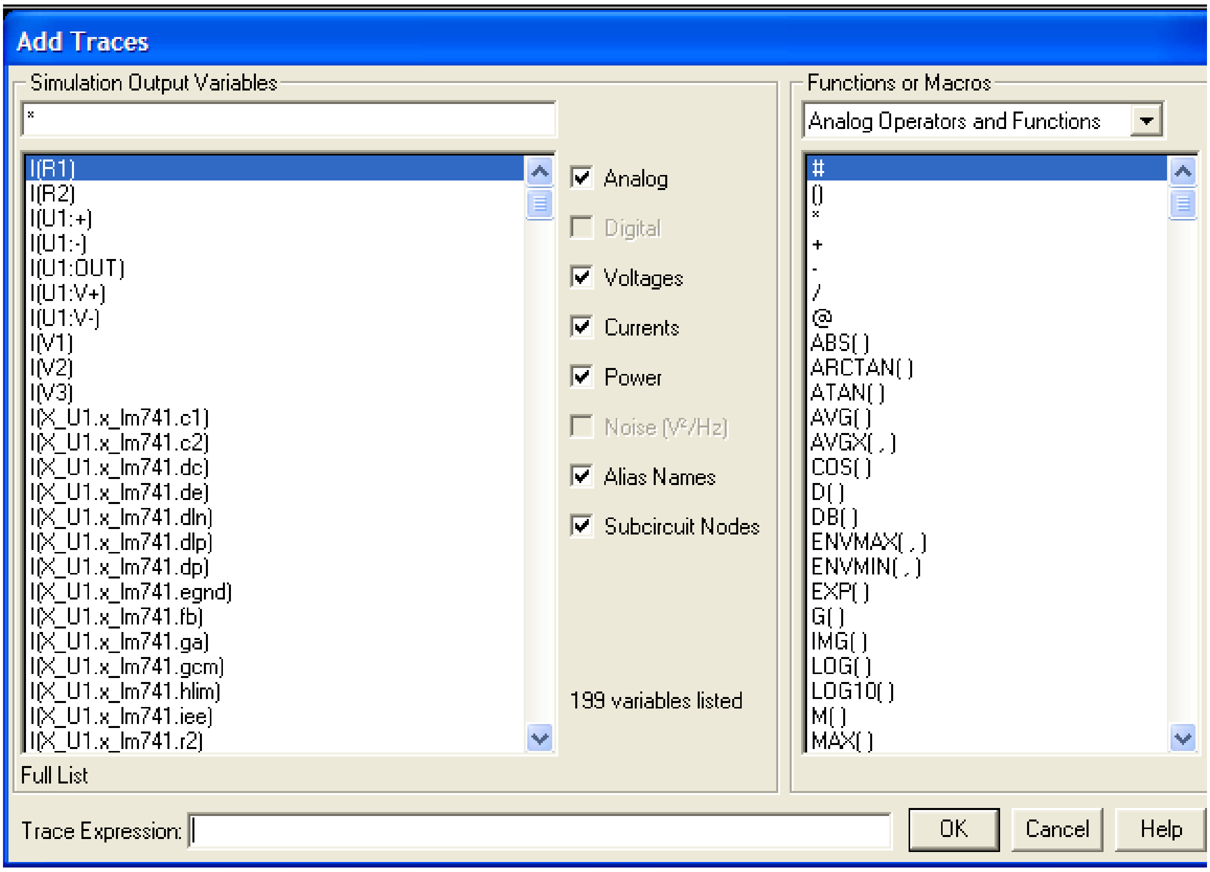




- Also, you can make mathematical operations with the variables in the simulation by using “Add Trace” buton and you can transfer the results on the graphic.



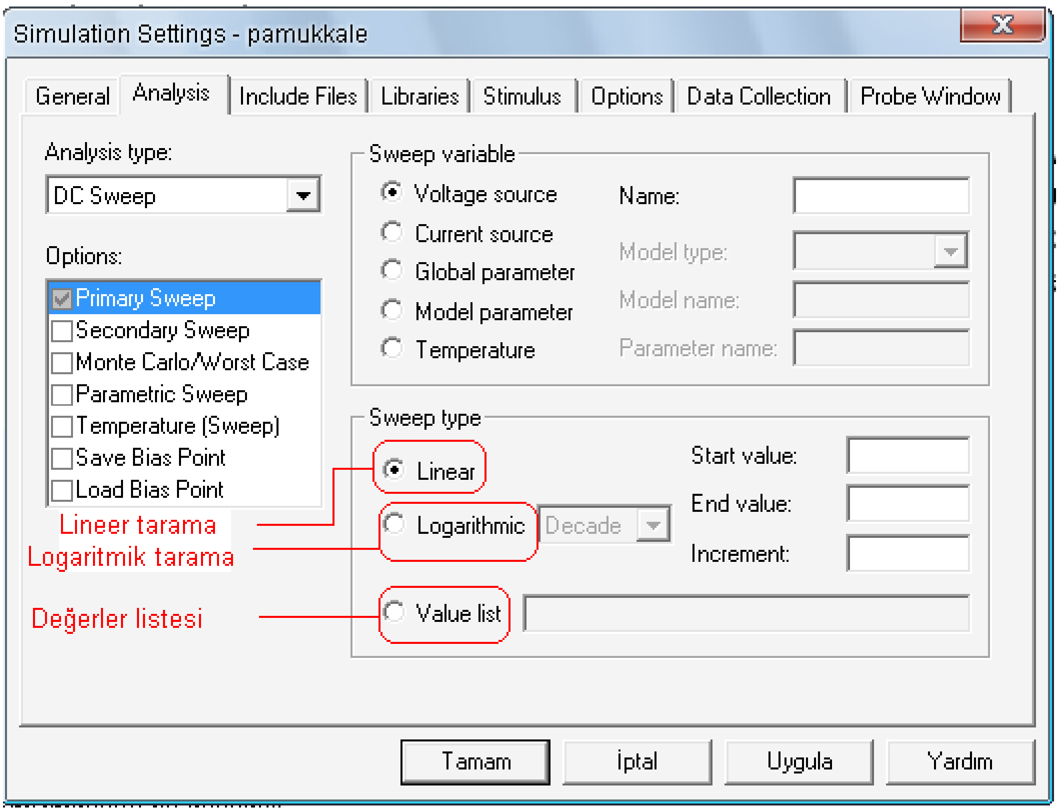
- After you clicked on the “Add Trace” button, on the opening window you can see the variables on the left list and mathematical functions on the right list. For example, if you want to write the sum of the values of the currents on the R1 resistor and R2 resistor on the graphic, you must write “I(R1) + I(R2)” on the “Trace expression” area and click OK.



PSPICE A/D ANALYSIS TYPES AND OPTIONS

1. DC Sweep Analyse:

You can realize the simulation by making the current and voltage values constant in the circuit. Also, you can make simulations when there are current and voltage changings between certain two values. For **DC** analysis, you can make browsing as from initial value up to final value step by step. For every input voltage value, circuit is analysed and results are saved. The **DC** transfer characteristics of the circuit is extracted. Scanning can be linear, logarithmic or as part of certain values. Also you can realize, Secondary **DC Sweep** Analysis, **Monte Carlo**/Worst Case Analyse, **Parametric** Analysis, **Bias Point** Analysis as choices of **DC** analysis.

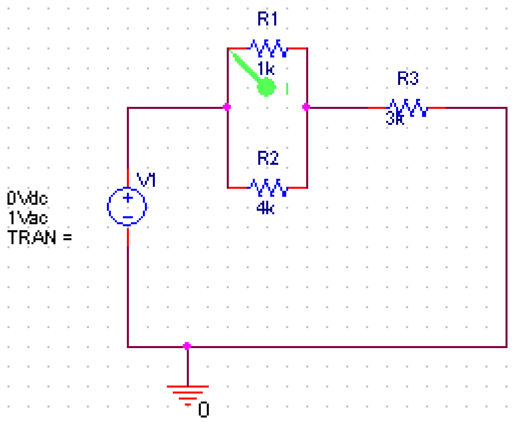


If it is containing multiple source DC scanning it is used **VSRC,** otherwise VDC is used for voltage. Similarly, **IDC** is used for current and if it contains multiple source DC scanning **ISRC** is used.

1.1. Primary Sweep:

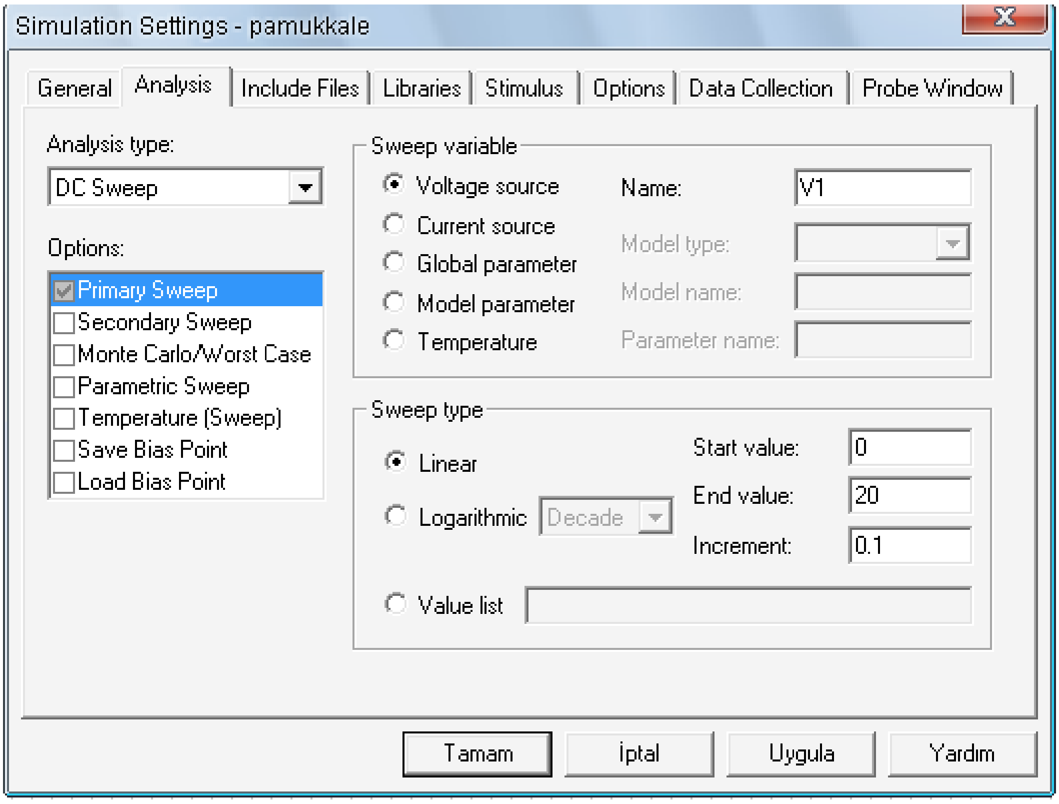
We can tell this by using an example:

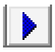
Let's scan V1 source between 0-20 V with 0,1 V value intervals and observe the changes on R1:

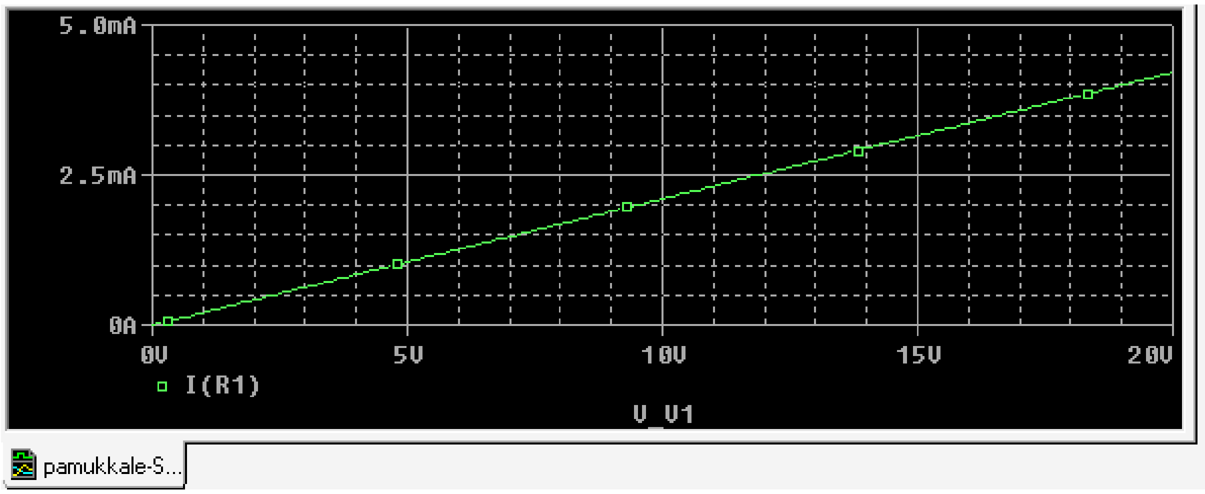


- After set up the circuit, click on the **Edit Simulation Settings**  button and choose the **DC Sweep** choice.

- Then, choose **Voltage Source** on **Sweep Variable** and write the source name on the **Name** part.
- Choose **Linear** from the **Sweep Type** part and enter the initial, final and increase values. Click OK.

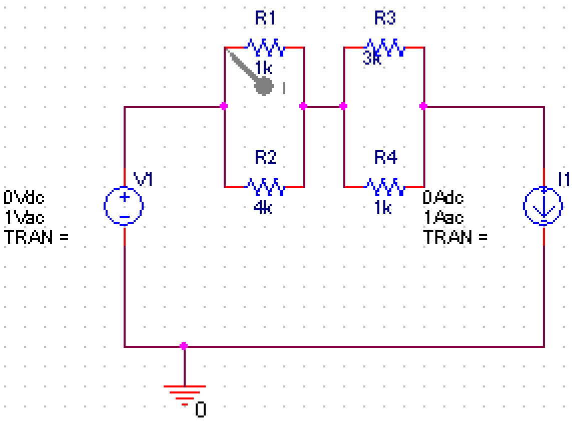


- Circuit is simulated by clickin on the **Run Pspice** . The current change of R1 can be seen on the graphic. If the scattered source is current source instead of the voltage source, it must be used **Isrc**. You can add the graphics you want by using **Add Trace.**

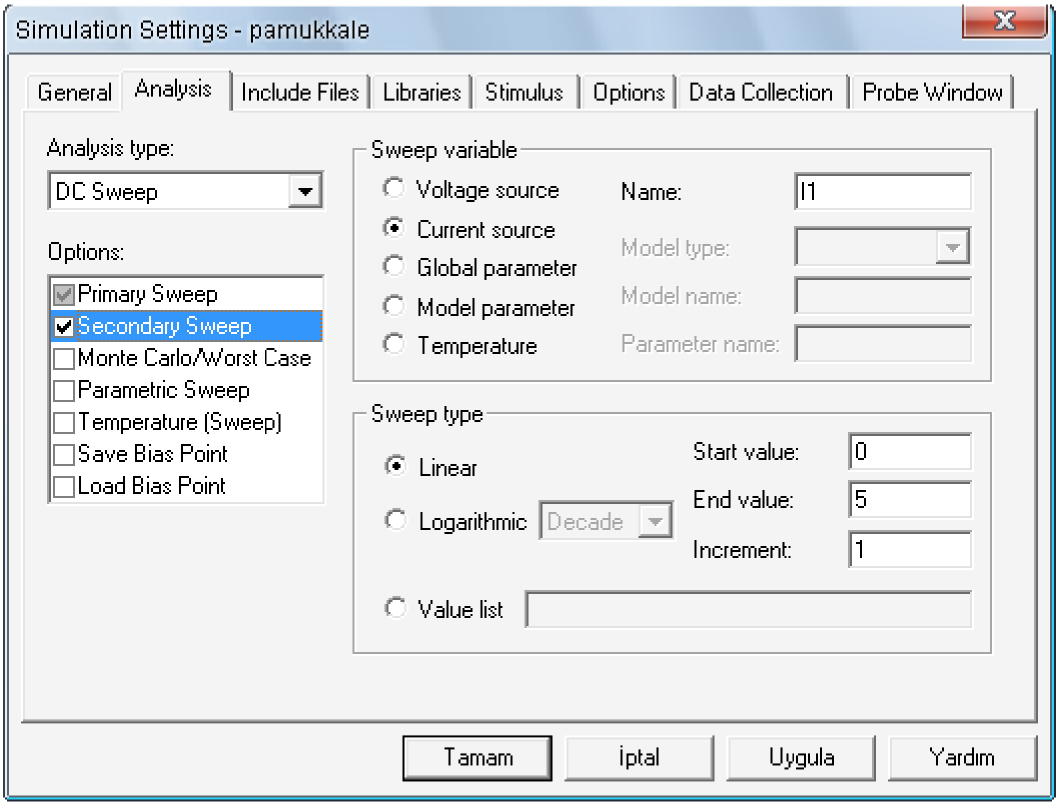


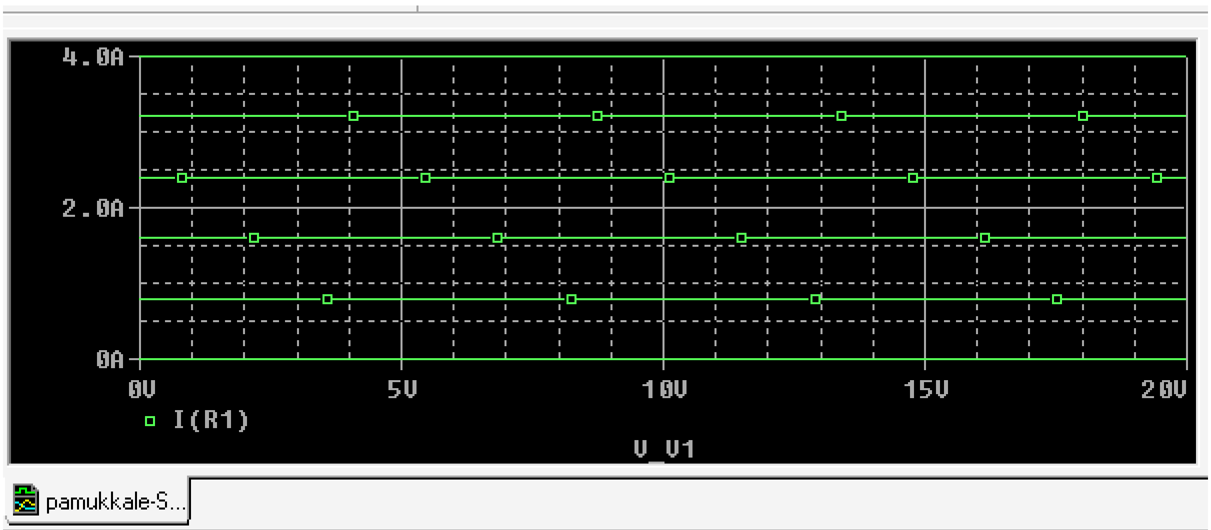
1.2. *Secondary Sweep:*

Let’s give an example. Let’s scan the current and voltage source values together. Scan the **V1** source between 0–20 V with 0,1V value intervals and **I1** source 0-5A with 1A intervals. Observe the current changes of **R1**.



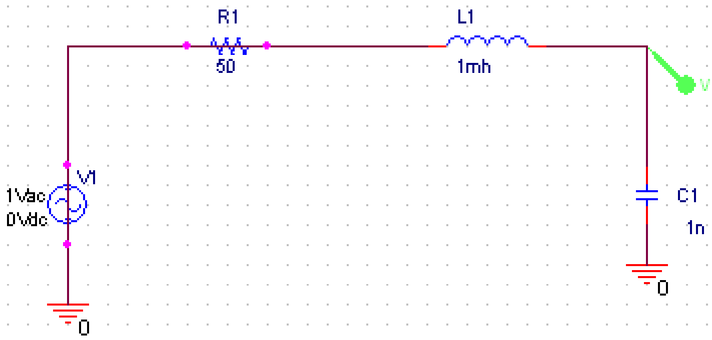
- After you set up the circuit, in addition to the steps of primary sweep, choose the **Secondary Sweep** in the **Simulation Settings** window.
- When click on the **Secondary Sweep** and choose **Current Source** from the **Sweep Variable.** Then, write the source name on the **Name** section. Choose **Linear** from the **Sweep Type** part and enter the *initial*, *final* and *increase* values.
- **OK** is selected and exit there. By clicking on the **Run Pspice**  buton, simulation is realized and current changes of **R1** can be seen on the graphic. By using **Add Trace,** the desired graphics can be added.

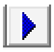


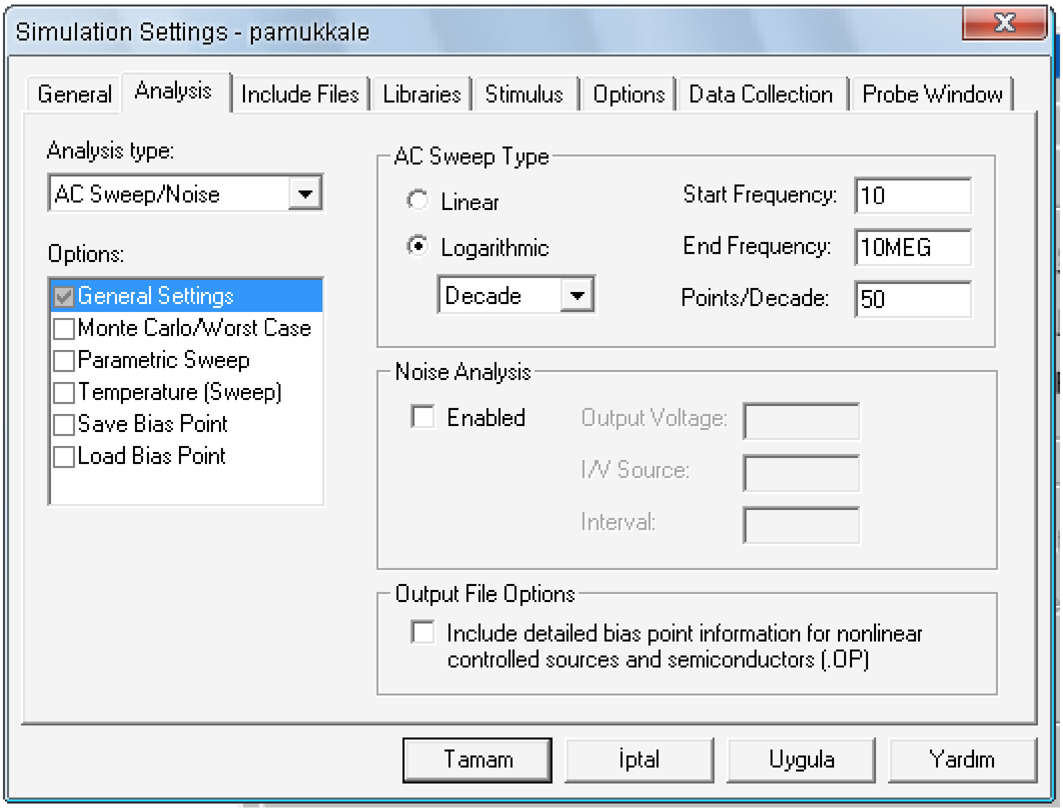


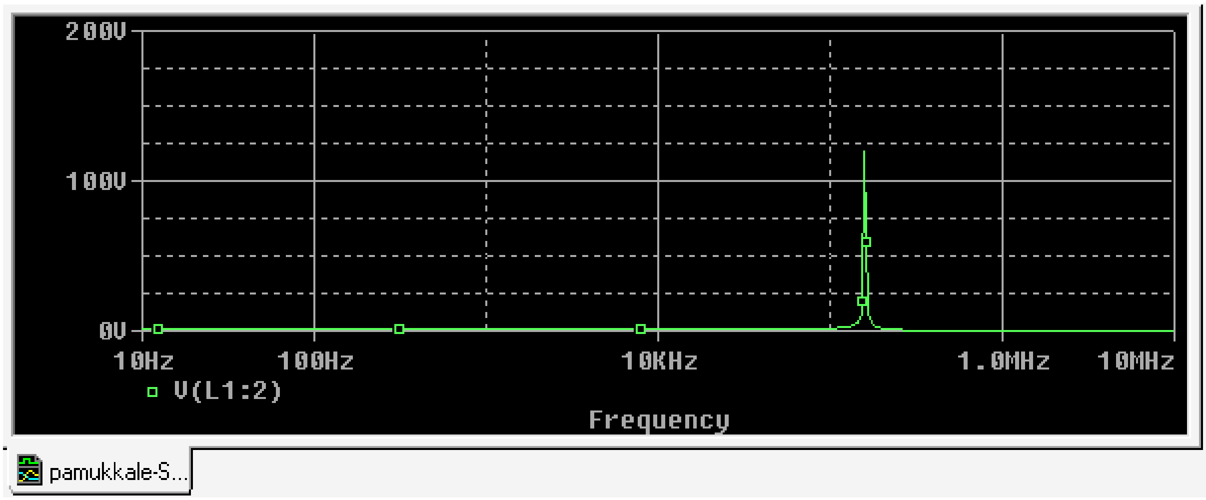
2. AC Sweep/Noise Analyses:

The frequency response of the circuit can be extracted or changing of impedance with frequency by using **AC** scanning. Also, as options of **AC** analysis, Monte Carlo/Worst Case Analyses, Parametric Analysis, Bias Point Analysis can be done. For voltage in **AC** scanning **VAC** is used and for multiple sources VSRC is used. Similarly, in **AC** scanning **IAC** is used for current and for multiple source **ISRC** is used. Let's investigate **AC** analysis on a **RLC** circuit:



- After set up the circuit, choose **AC Sweep** choice on **Simulation Settings** window for **AC** scanning. Choose OK. Simulation is realized by clicking on the **Run Pspice**  button.



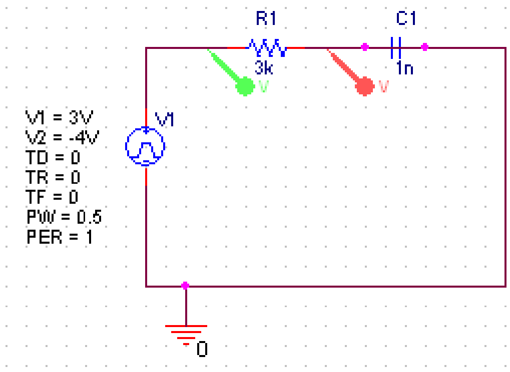


- Voltage value on the capacitor changes according to the frequency on the graphic. This variation can be seen by using **Voltage Marker** placed on the graphic**.** Here frequency change is between 10Hz and 10MHz.
- For **Noise** analysis**,** choose the analizi için **Enabled** choice on the **Noise Analysis** section. Write the output voltage that you want to measure on the **Output Voltage** choice.
- Write a free input voltage or current that will be a input value for the measuring on the **I/V Source** section. Write frequency interval on the **Interval** choice.

3. Time Domain (Transient) & Fourier Transform Analyses:

If you want to see the changing of the variables in the circuit according to the time, this simulation mode must be used. At the end of the simulation, values of the variables are obtained as a function of time. Time domain analysis always starts from **t=0** and finals at the **Tstop** value with the steps given by user. By using time domain analysis, responses of the circuits to the input signals like sinusoidals or pulse can be examined (e.g rectifier, clipper, amplifier). As options of **Time Domain** (**Transient**) analysis, Monte Carlo/Worst Case Anaysis, Parametric Analysis, Bias Point Analysis can be done.

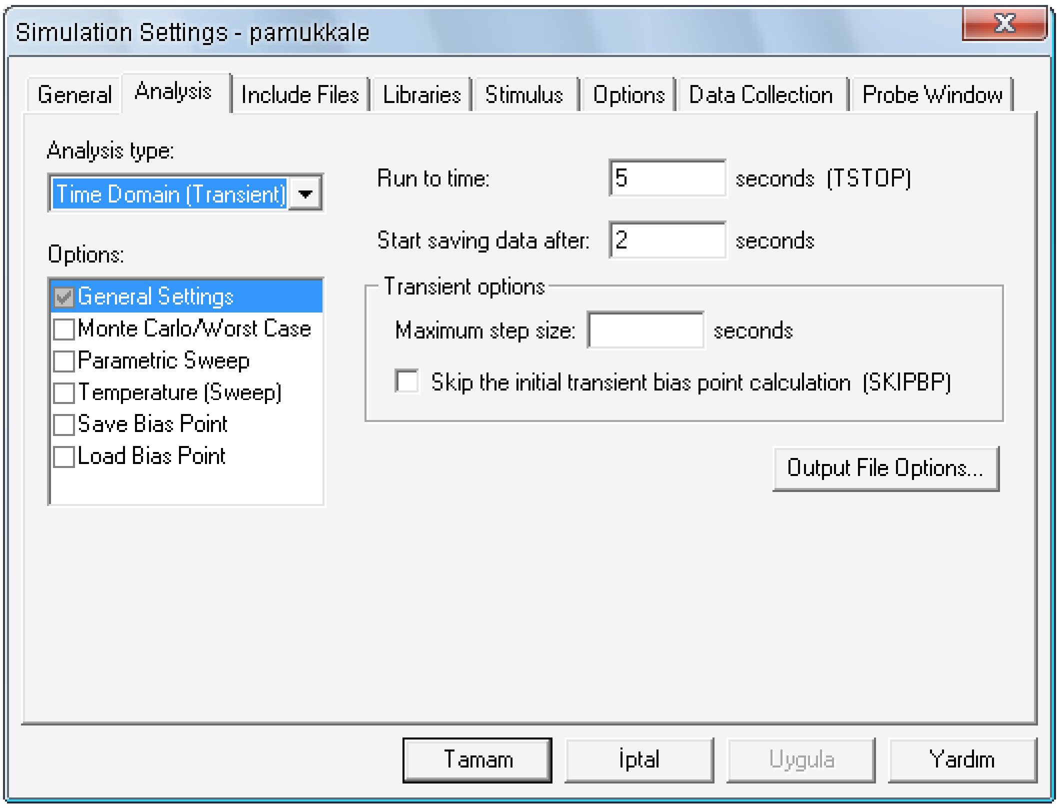
In the **Time Domain** anaylsis; ***for voltage:*** VSRC, VEXP, VPULSE, VPWL, VPWL_RE_FOREVER, VPWL_F_RE_FOREVER, VPWL_N_TIMES, VPWL_F_N_TIMES, VSFFM, VSIN sources, ***for current:*** ISRC, IEXP, IPULSE, IPWL, IPWL_RE_FOREVER, IPWL_F_RE_FOREVER, IPWL_N_TIMES, IPWL_F_N_TIMES, ISFFM, ISIN sources can be used.



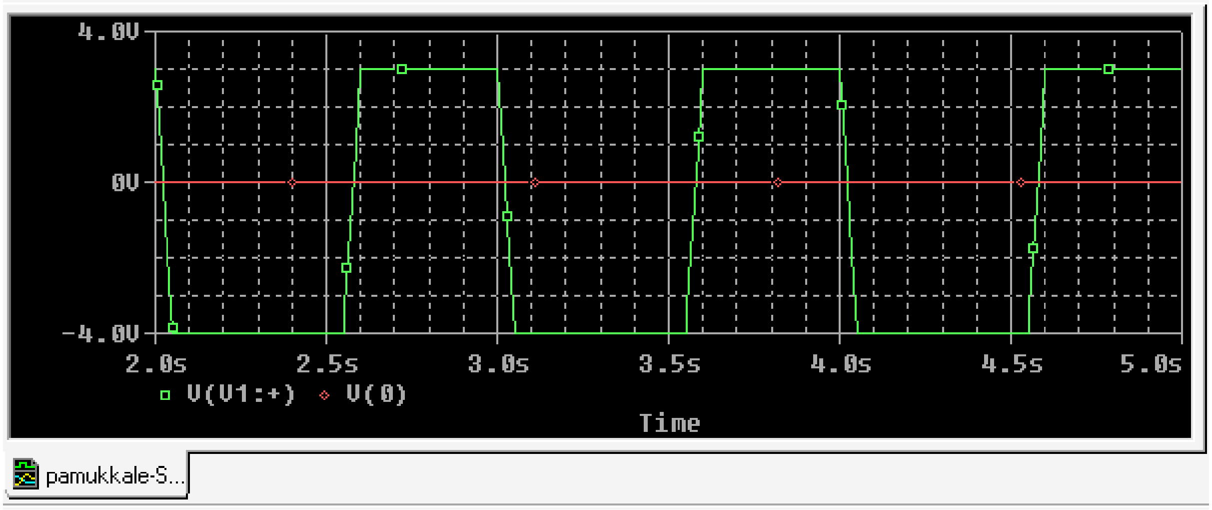
Let’s examine Time Domain analysis for a **RC circuit**.

- After set up the circuit, **Analysis type** is chosen as **Time Domain** on the **Simulation Settings** section.
- Then, for determining simulation time, write the duration of simulation in terms of seconds on **Run to Time**.

You can determine when the data will be taken after the simulation started by using **Start saving data after**.



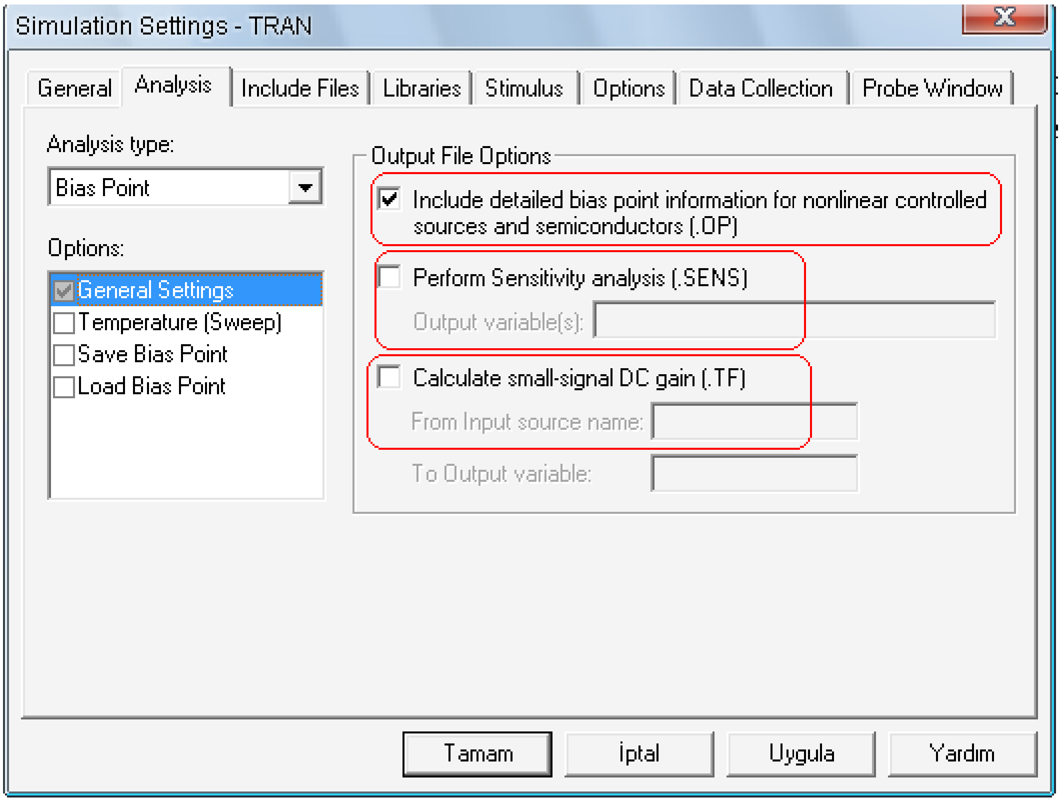
- Click OK and exit. Start simulation by clicking on **Run Pspice**  button.



You can write the maximum allowable time step size on the part of **Transient Options** of **Maximum Step Size**. Also, you can see **Fourier analysis option** in the **Time Domain** (**Transient**) Analysis type of **Output Files icon. Fourier Analysis** calculates the **DC** and **Fourier** components of Transient analysis.

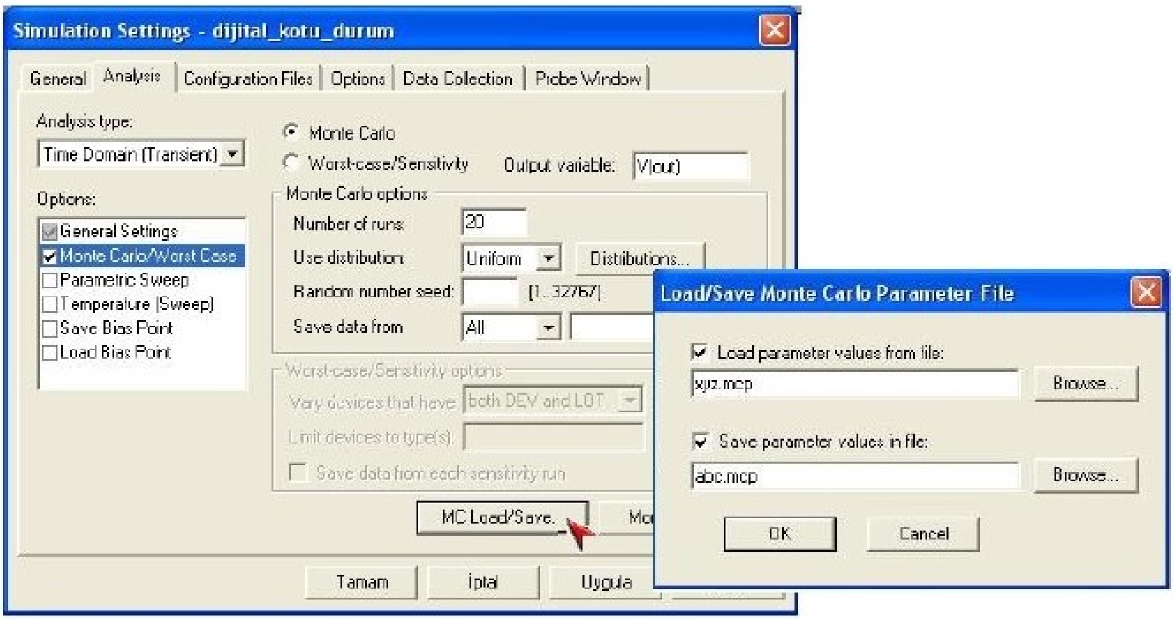
4. Bias Point (Working Point) Analysis:

Bias point saves the detailed Bas point data fort he simulation exit files. Also, as an option of Bias Point Analysis, it can realize temperature analysis. The reports data fort he exit files are: It lists all of the analog and digital point voltages, and it shows the currents between all of the voltage sources and their powers as you can see in the **Output Files** section. It realizes sensitivity analysis. It lists small signal parameters for all of the elements.



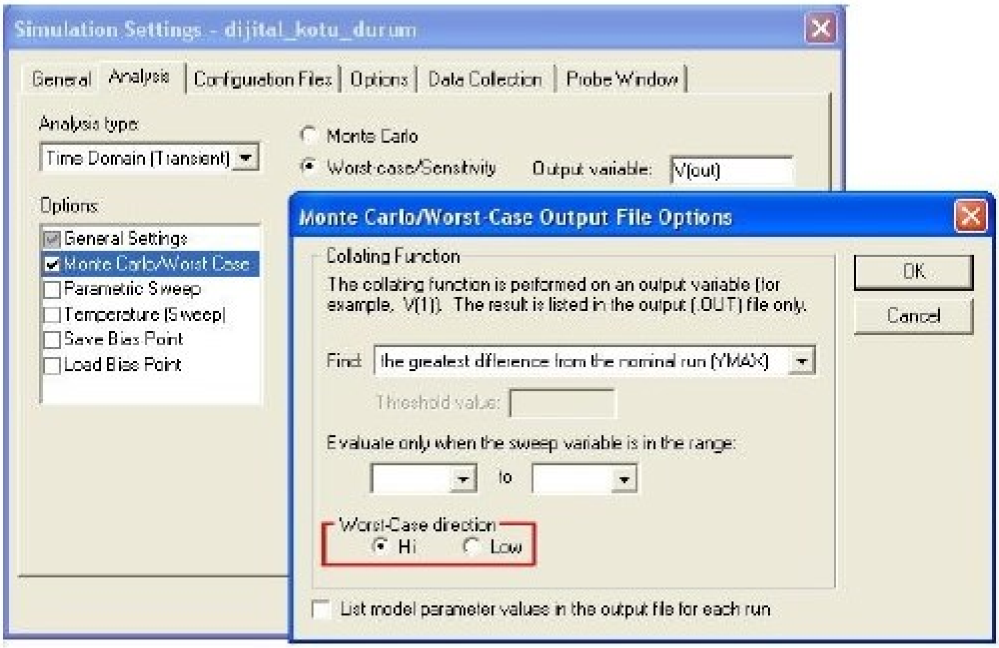
5. Monte Carlo Analysis

It calculates the response of the circuit by changing the values of elements randomly which are described in a certain interval. You can run this analysis for defined number, it uses nominal values for first run and variable values for the others.



6. *Worst Case Analysis*

It calculates the worst result for the maximum or minimum values of the elements with the closest value to the nominal. Addition to the circuit definition, it can determined that what is the ‘’worst’’ definition and parameter tolerations. Tolerations data are defined in .MODEL definition, "worst" definition is defined on the 'Simulation Settings' window for Pspice model.

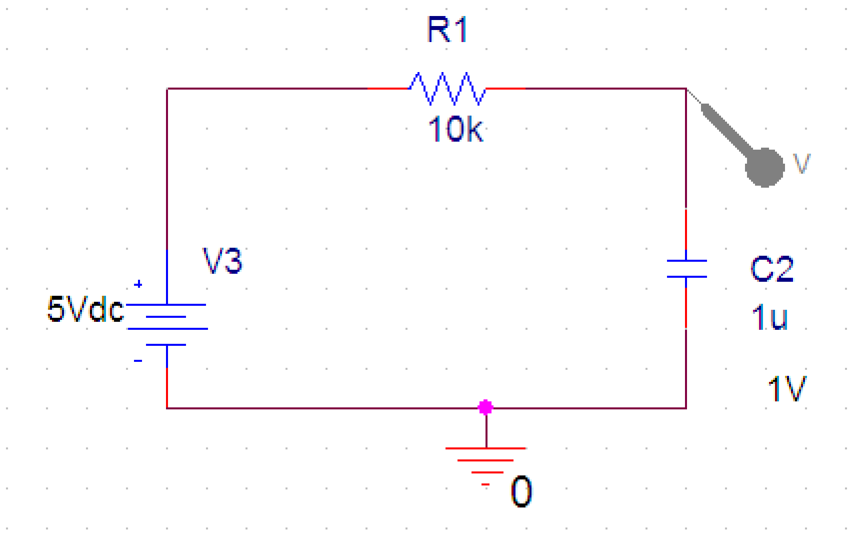


EXAMPLES

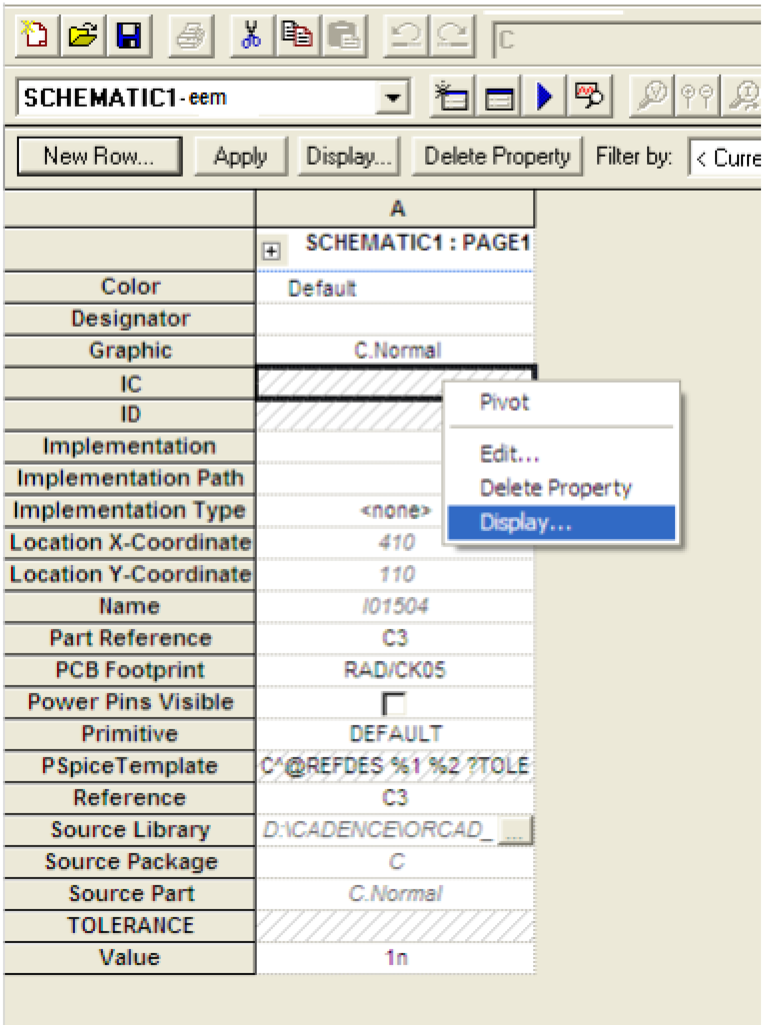
- ***RC Circuit***

Fort his example, a basic RC circuit will be impelemented.

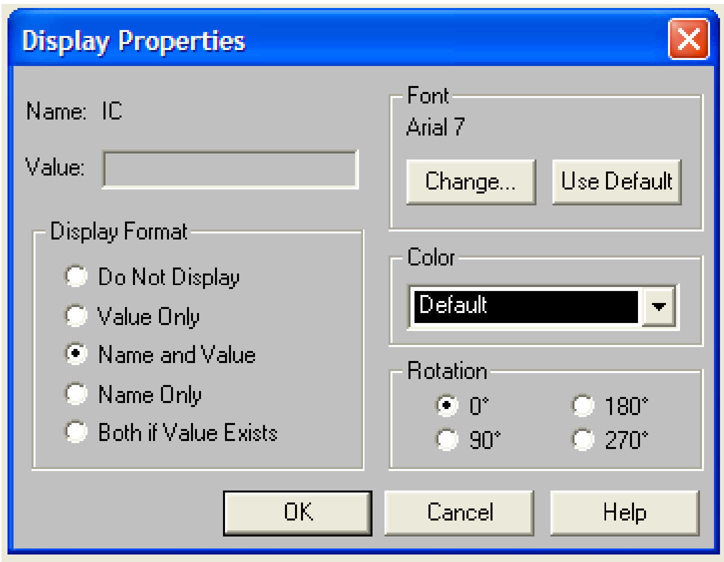
- Set up the circuit in the following by considering the knowledge given before.



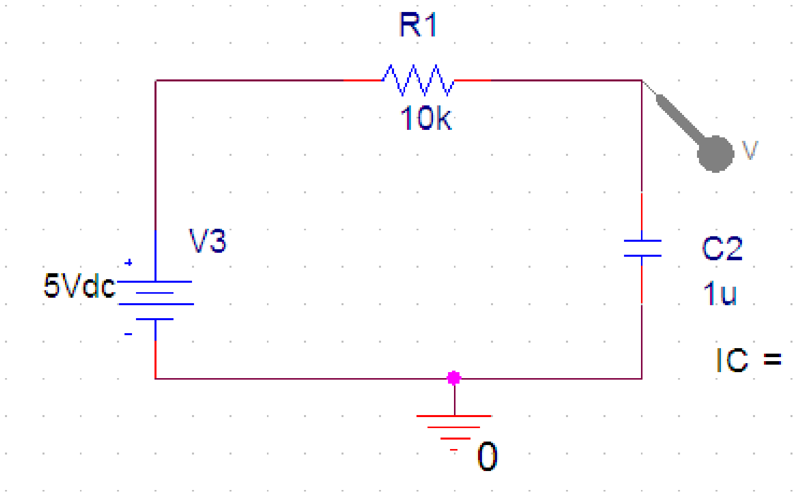
- Capacitor has an initial value for t=0 time. For entering this value give as 1 Volt click on the capacitor two times, so "Property Editor" is opened.
- For assigning a initial value with "Property Editor", find "IC" value and right click on the free space opposite of it and click on the "Display" choice.



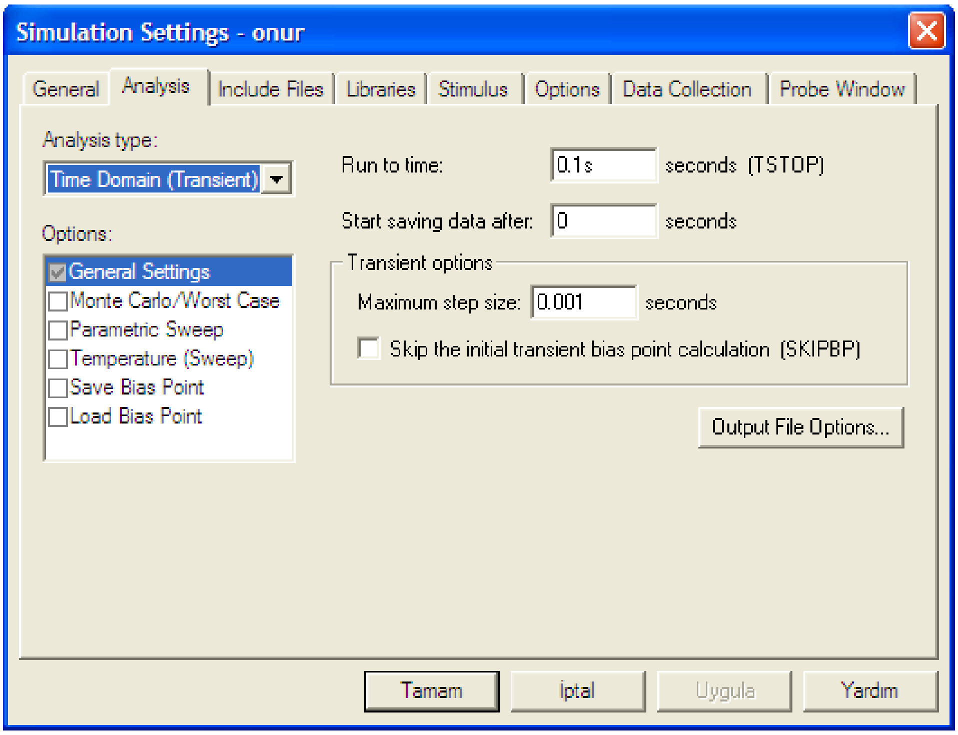
- After marking "Name and Value" choice in the incoming window, then confirm it by using OK buton.



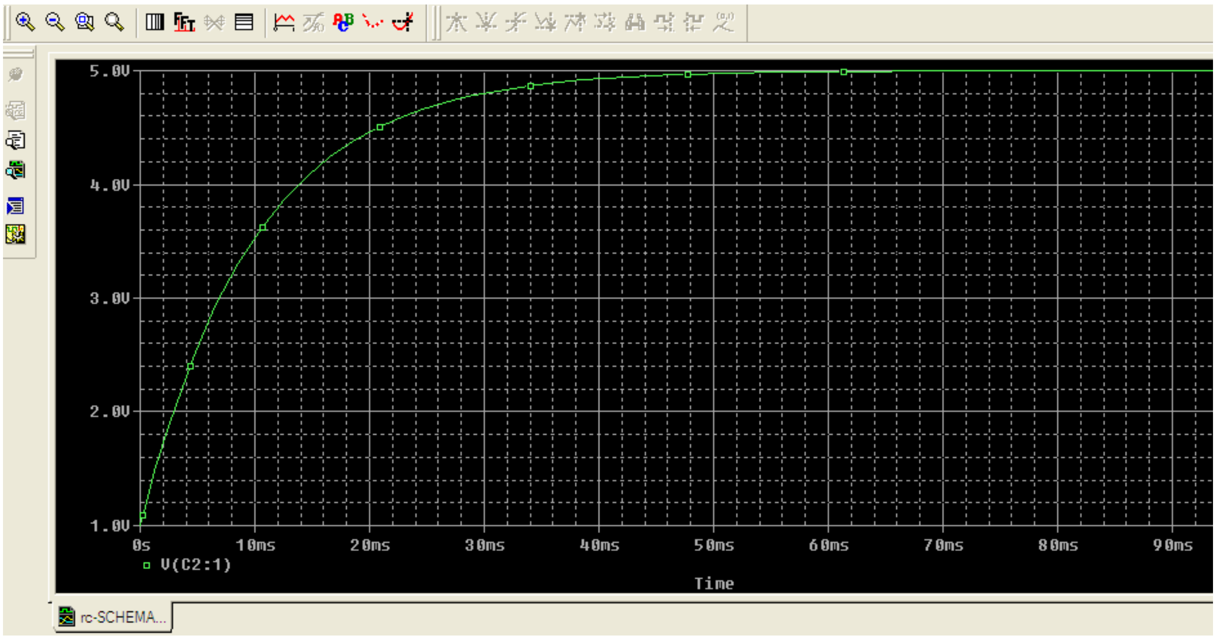
- After these steps, you can see the circuit as given in the following. By clicking on "IC" two times, you can write the desired 1V value.



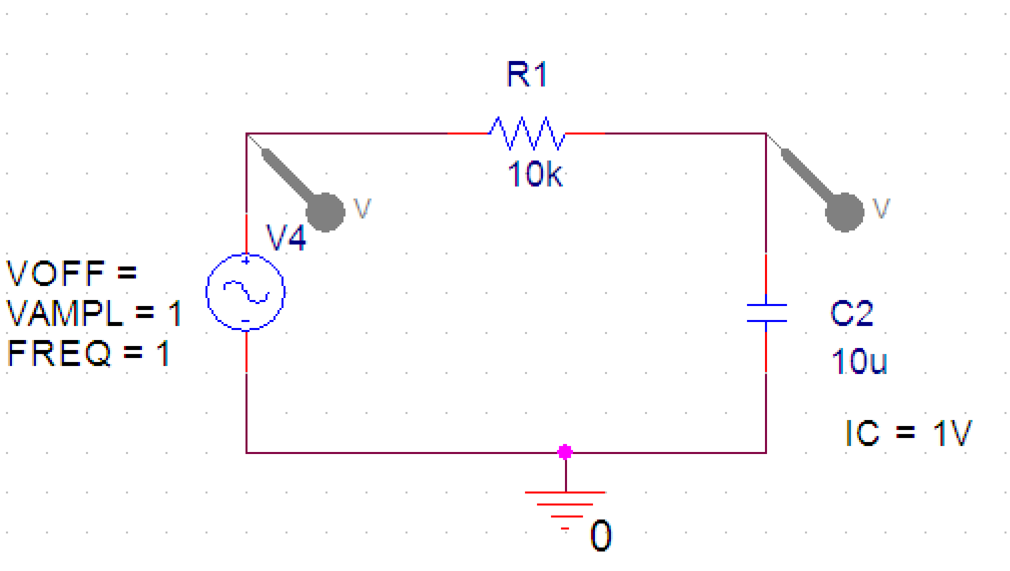
- Simulation adjustments must be as seen below.



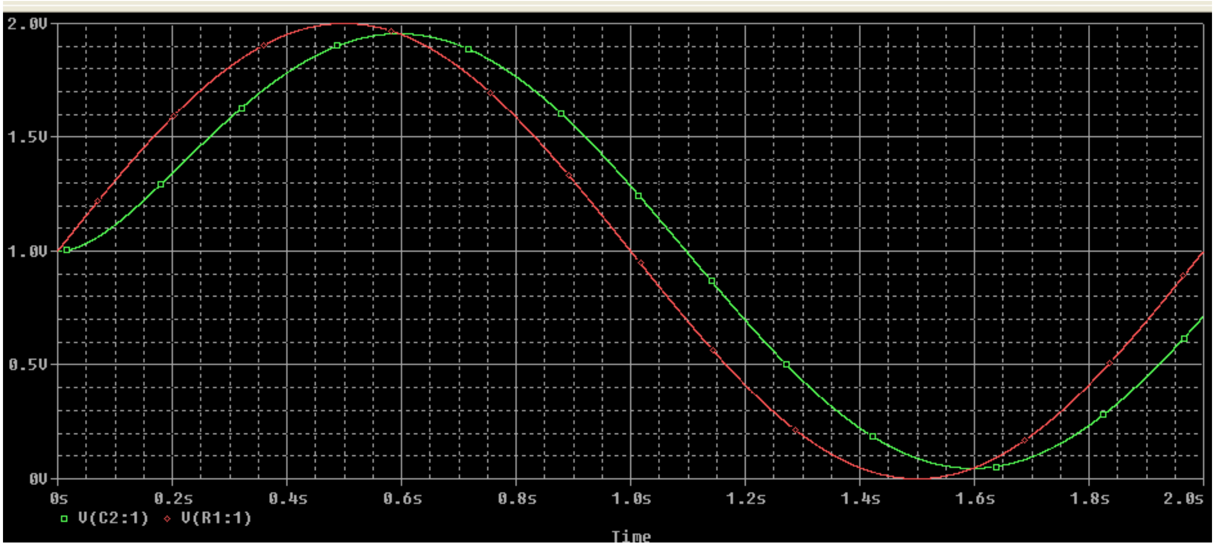
- After yo urun the simulation, you can obtain the graphic given in the following.



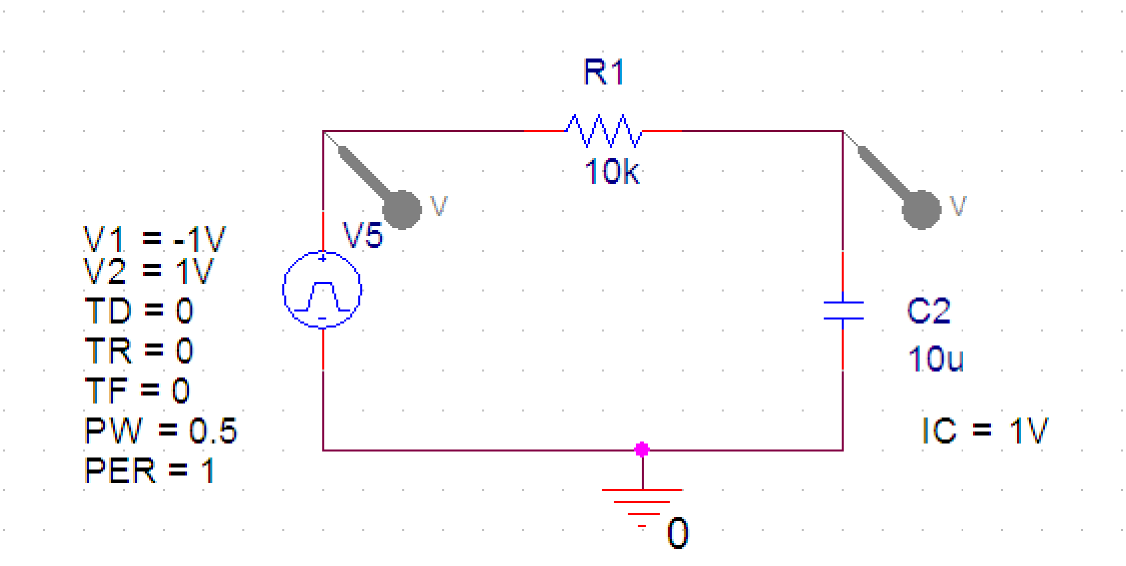
- Replace the "Vdc" source with "Vsin" in the same circuit and adjust the value of capacitor as "**10u**". Place a "Voltage Marker" on the source for observing input source. The values of Vsin source as given below.



- If you adjust the "Run time" for 2 seconds in the simulation adjustments the graphic in the following can be obtained. Here, V(C2:1) is the value of capacitor voltage and V(R1:1) is the voltage value of input source.



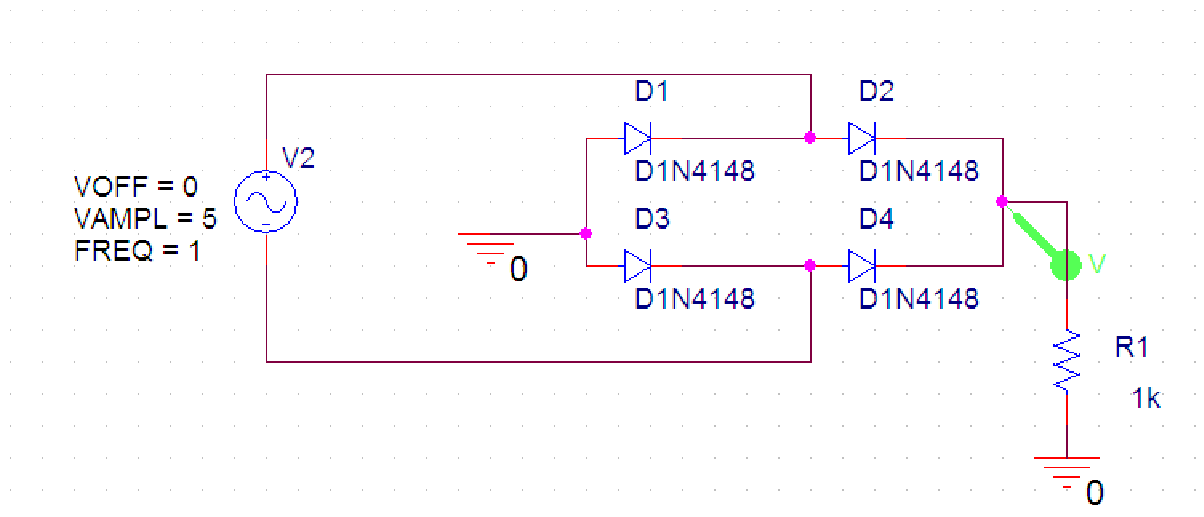
- Finally, by replacing “Vsin” source with “Vpulse” source, we will be able to observe the behaviour of RC circuit under square pulse. After adding “Vpulse” source, adjustments must be as shown below.



- You can obtain this graphic when yo urun the simulation.

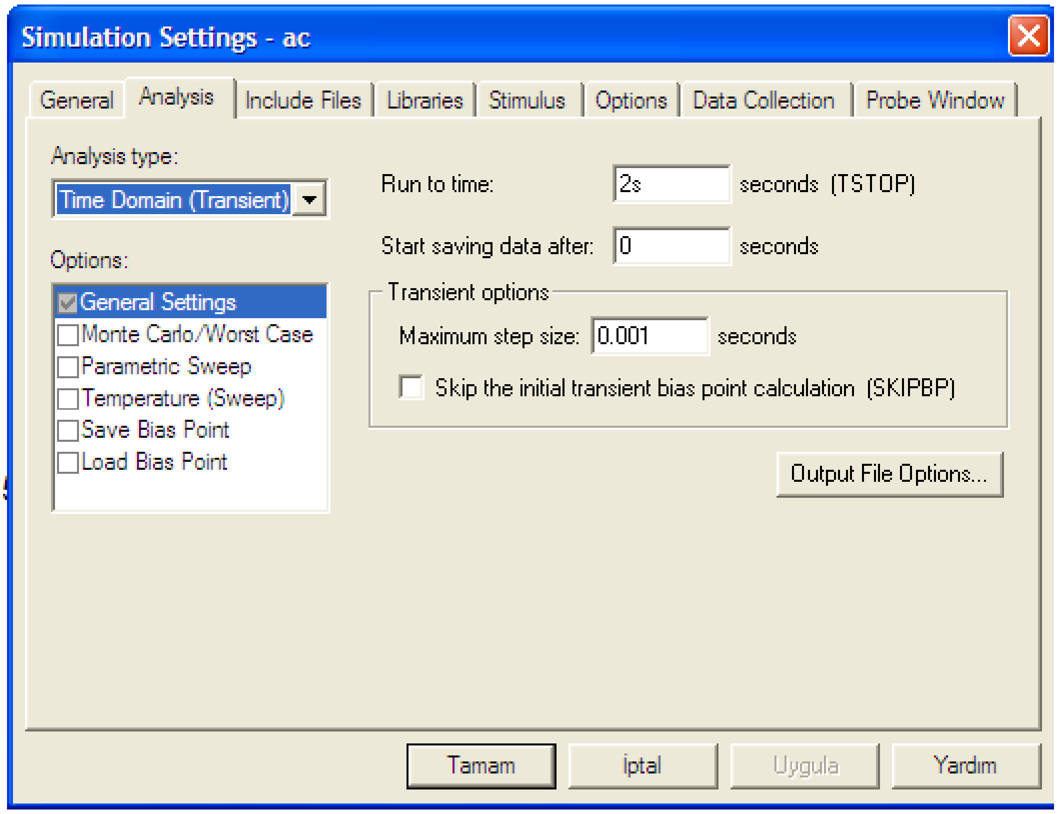
Full Wave Rectifier

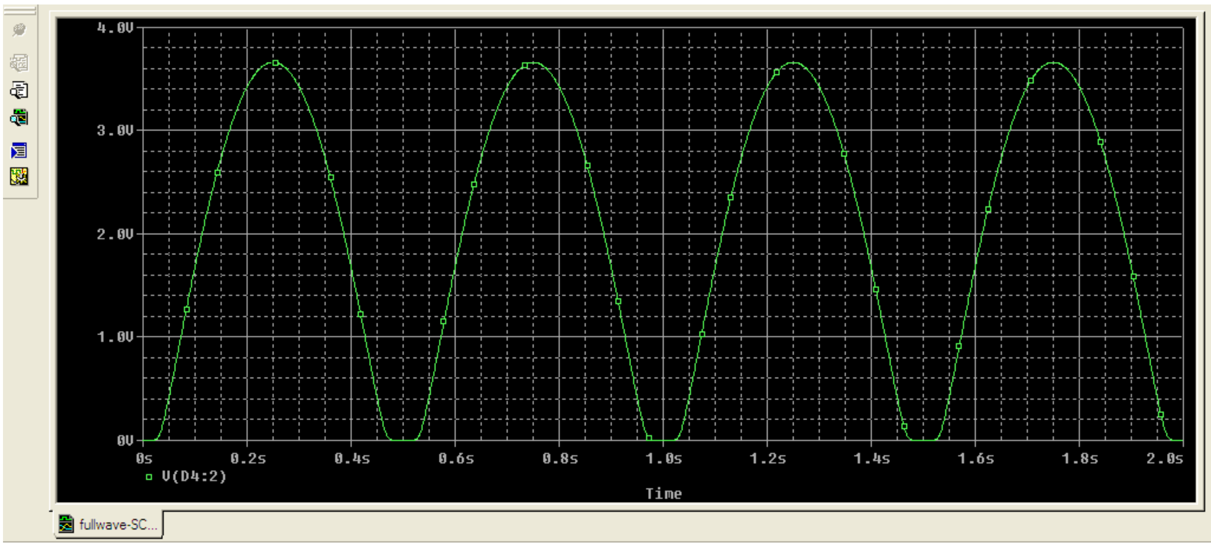
Set up the circuit given below. If you are using full version PSPICE, you can add diodes from DIODE library.



Simulation adjustments

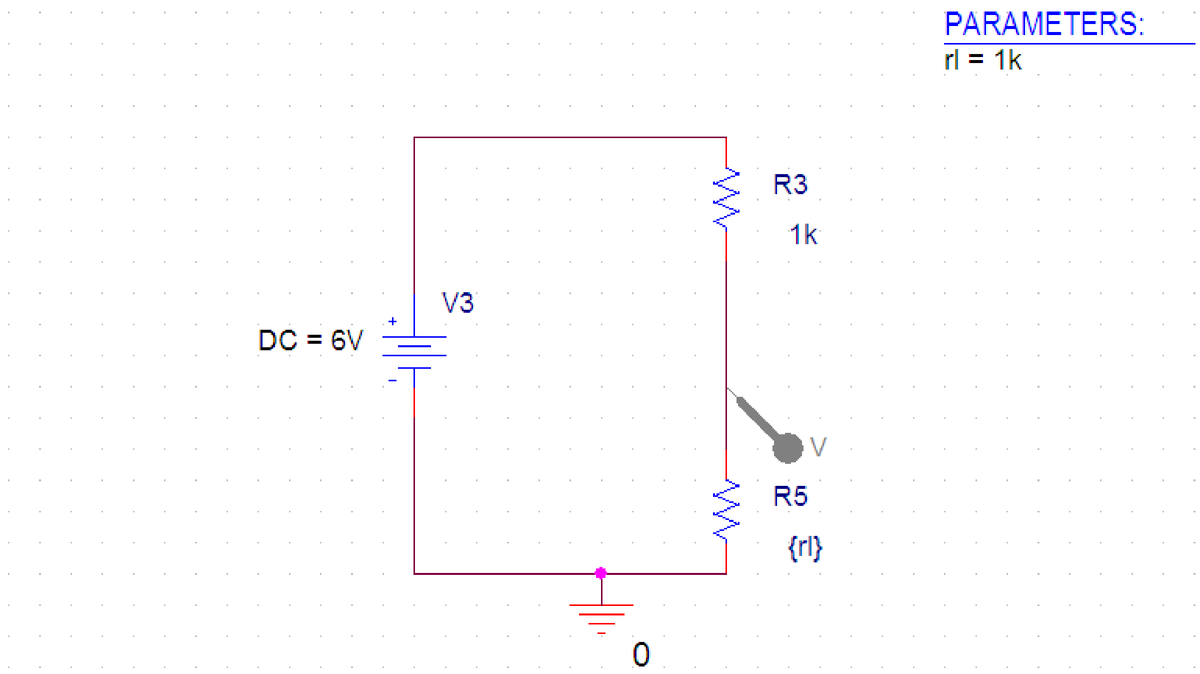
CUE: If you can not obtain smooth graphics, it means that you don't take enough sensitive value for "maximum step size". If you obtain irrelevant graphics although this, you may be taking the simulation duration too long or too short.





Parametric Circuit Element Usage

In this example, resistor value will change in the interval of desired values. Since the value of resistor changes, it is a potentiometer simulation.



1. Firstly, take the value for the circuit element which we want to change its value as {rl}, the important thing is writing it in the braces, you can use any name. (be sure that you didn't change the "value" instead of name)
2. Parça ekleme menüsünden (kısayol P) PARAM/SPECIAL'I çizim ekranına ekleyin . (SPECIAL kütüphanesi eklenmemişse , "add library " butonunu kullanarak ekle yebilirsiniz.)
3. Open the properties window by two times clicking on the PARAM part. Here, write "RL" on the name place and 1k on the value part by using "New Column" buton.
4. Choose Rl coloumn by clicking one time and then by right clicking open "display" menu. Enter the "Name and Value" choice.
5. Now, simulation adjustment will be done.

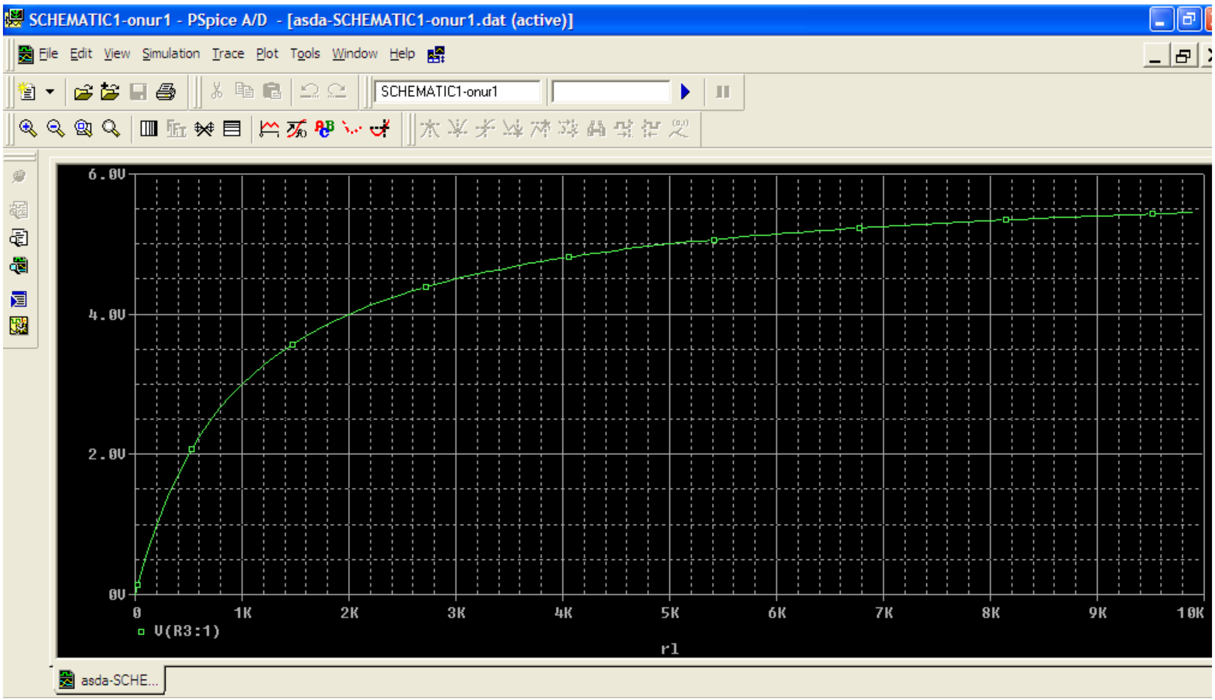
Adjustments

Analysis type : DC Sweep
Options : Primary Sweep (Not Parametric Sweep!)
Sweep variable : Global parameter
Parameter name: RL (name in the braces)
Sweep Type : Linear

Enter the values of Start, End, and Increment. These values determine the value interval that parametric variable will take and the increment of the value. 0,1,10k,100 are used respectively.

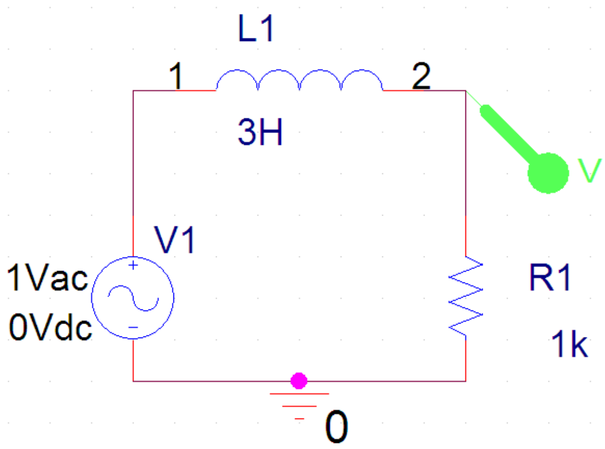
CUE: Be carefull about that resistor value doesn't start from 0.

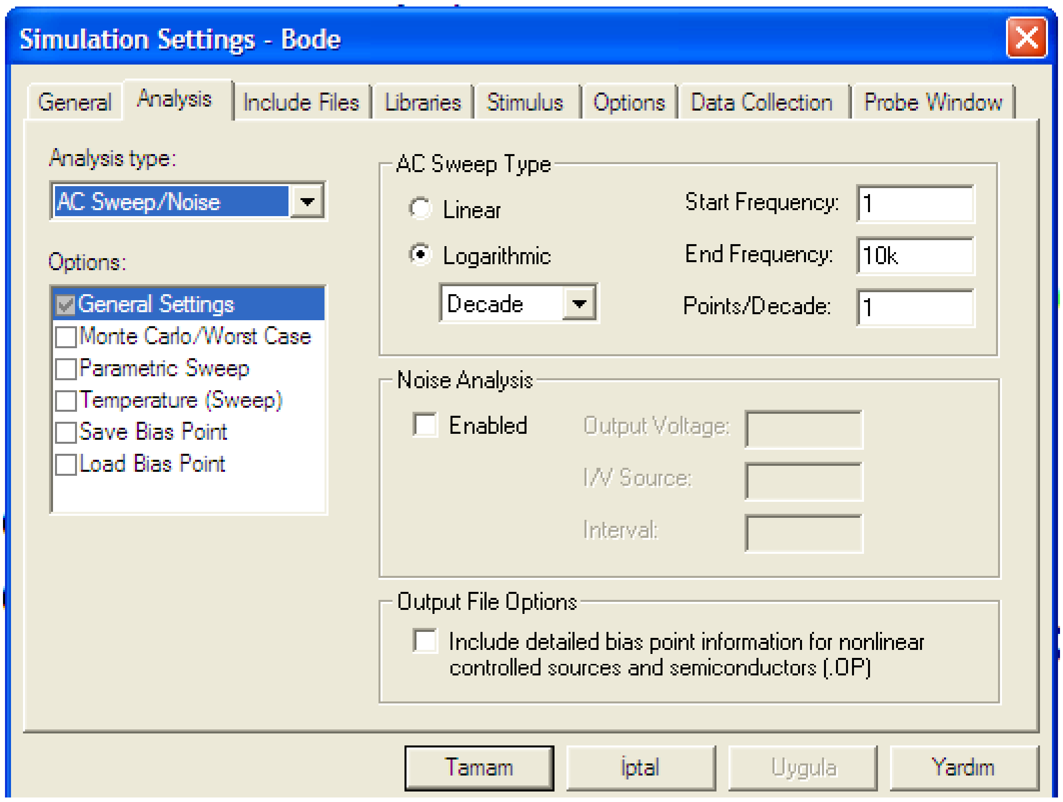
6. Finally, by placing a "probe" anywhere you want, you can observe the unstable value.



Bode diagram of Inductive Lowpass Filter

Lowpass filter transfers the signals under the cut off frequency and doesn't transfer above the cut off frequency ideally. Bode diagram is the drawing of gain versus frequency. For obtaining this graphic in PSPICE environment, "AC Sweep" simulation will be used. Firstly, set up the circuit given in the following.

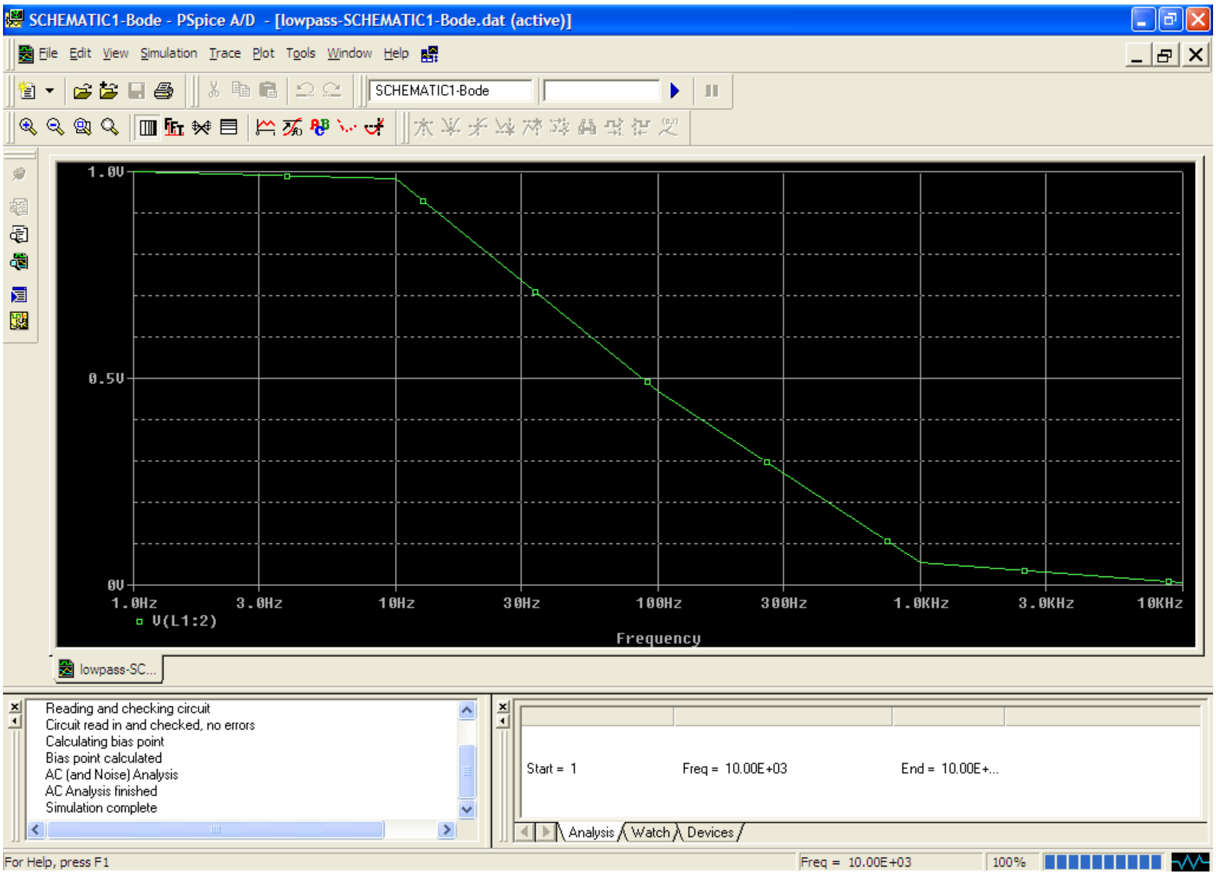




In Ac Sweep analysis, for obtaining bode diagram, you must choose logarithmic option and then you must enter the frequency values which will be scanned.

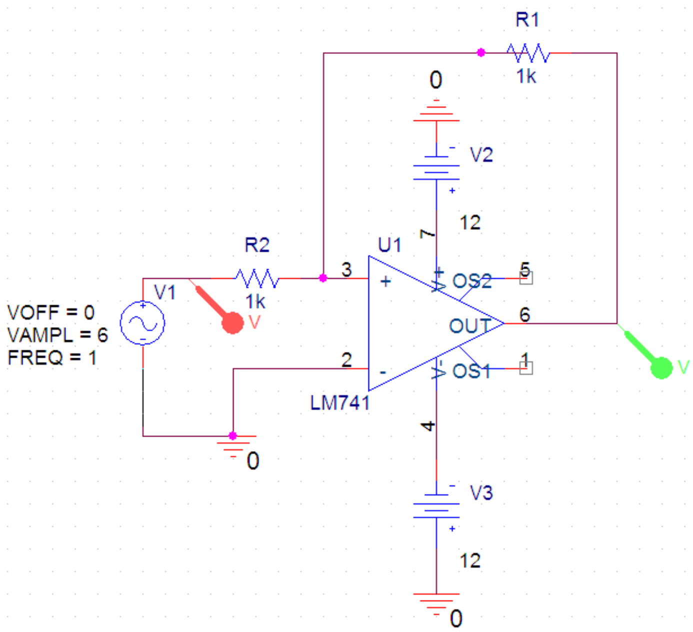
CUE: You must be carefull about "Start Frequency" value shouldn't be değerinin 0 !

The obtained graphic is as given below.



OPAMP Usage

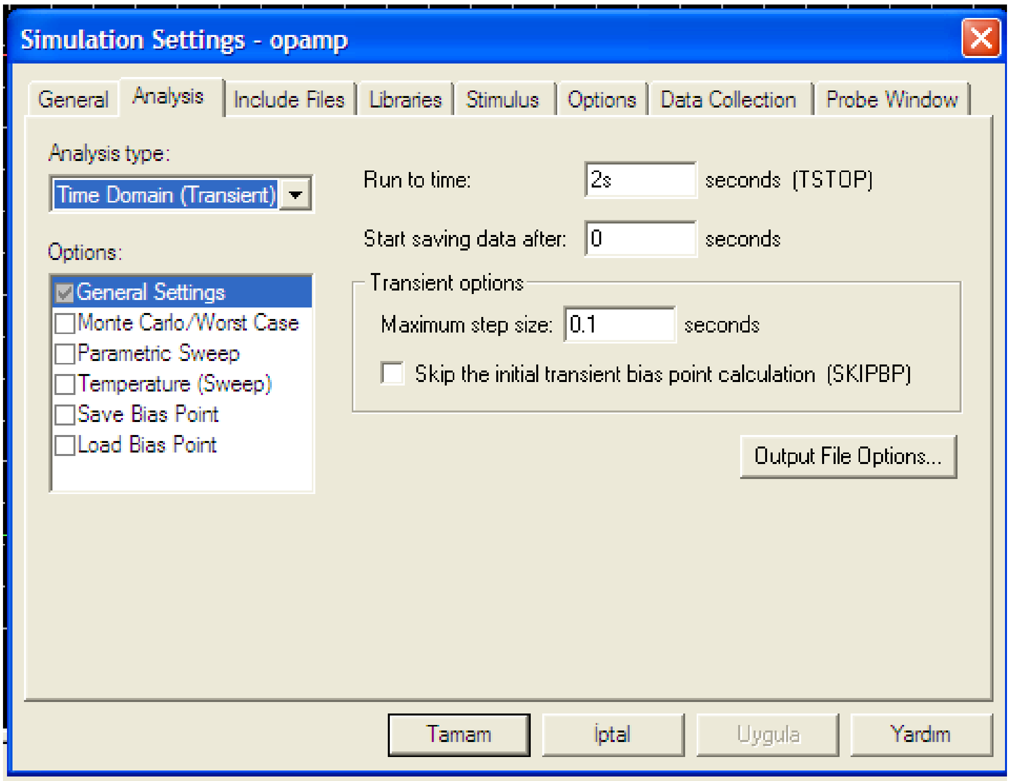
For this example, it will be designed a complementer OPAMP system which has a gain about 1. This means that the input signal will be multiplied by -1 in the output. Set up the circuit given in the following. You can find LM741 in the OPAMP library for full version PSPICE.



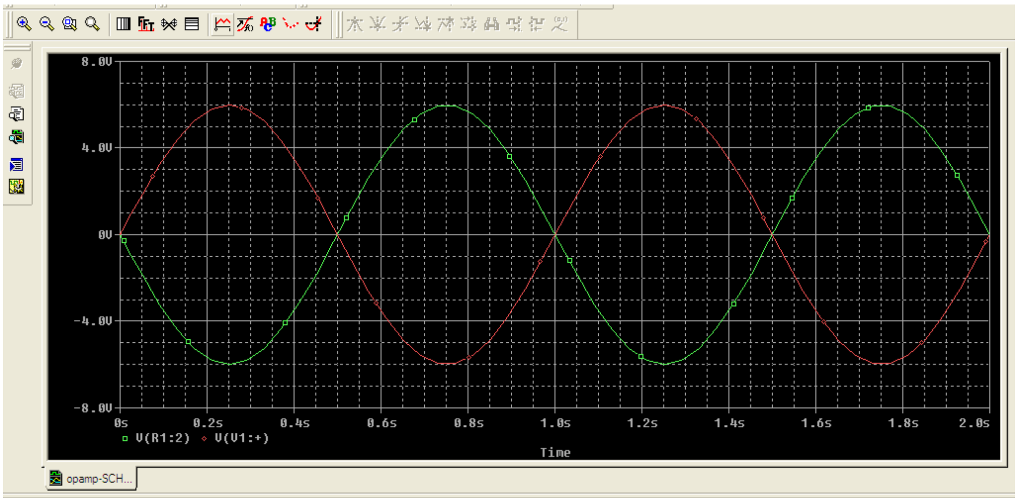
CUE: Here, supply voltage is given for 7 and 4 numbered jacks of LM741 OPAMP. Be carefull that the 12V source is negative in the 4 numbered input. Also, two "voltage probes" are used in order to see input and output simultaneously.

Simulation settings

As similar to other examples, "Transient Analysis" will be used.



In the graphic, V(V1) is input signal and V(R1) is output signal.



INVESTIGATING TRANSIENT ANALYSIS FOR RL, RC AND RLC CIRCUITS

If you want to investigate an electrical circuit in time domain, you must write either entegrated differential equations or state equations of that circuit. As it is known, the solution for differential equations has two parts: The first part of the solution is determined by the initial conditions in the circuit and the second part of the solution is determined by the sources in the circuit. These parts of the solution are named as Natural and Forced Solution respectively. For an asymptotically stable circuit in other words for a circuit which as $t \rightarrow \infty$ state transition matrix f(t) goes to zero, natural solution arrives to zero and forced solution arrives to natural solution. In a clear manner, for an asymptotically stable circuit after a while for investigating the circuit, exact solution is nearly equal to the particular solution. For an asymptotically stable circuit, exact solution is composed of transient solution and steady solution. For an asymptotically stable circuit, the homogen solution of differential equation system is named as transient, particular solution is named as steady solution. Although the transient solution is very large at first, after a while for circuit running, it gets smaller and approaches to zero. Steady solution is the solution which will continue so far as sources present in the circuit. Although its effect is too short, transient solution is important for determining the circuit's elements. It is important investigating circuits in the time domain in terms of understanding their principles and functions.

After that, we will consider simple *RC*, *RL* and *RLC* circuits and investigate the solutions of these circuits in case of they are stimulated by step, pulse and square wave sources.

RC Circuit:

Let's consider *RC* circuit given in Figure 1. State equations of this circuit are,

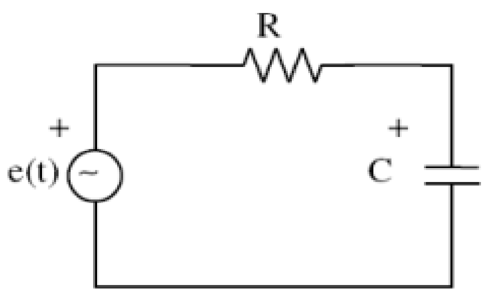


Figure 1

$$\frac{dv_c}{dt} = -\frac{1}{RC} v_c + \frac{1}{RC} e(t) \qquad (1)$$

If $e(t)=Eu(t)$ is step function in the equation (1), the solution of the circuit is;

$$v_C(t) = e^{-t/RC} v_C(0) + E(1-e^{-t/RC}) \qquad (2)$$

If $v_C(0) = 0$, the variance of voltages at the C and R are shown in Figure 2.

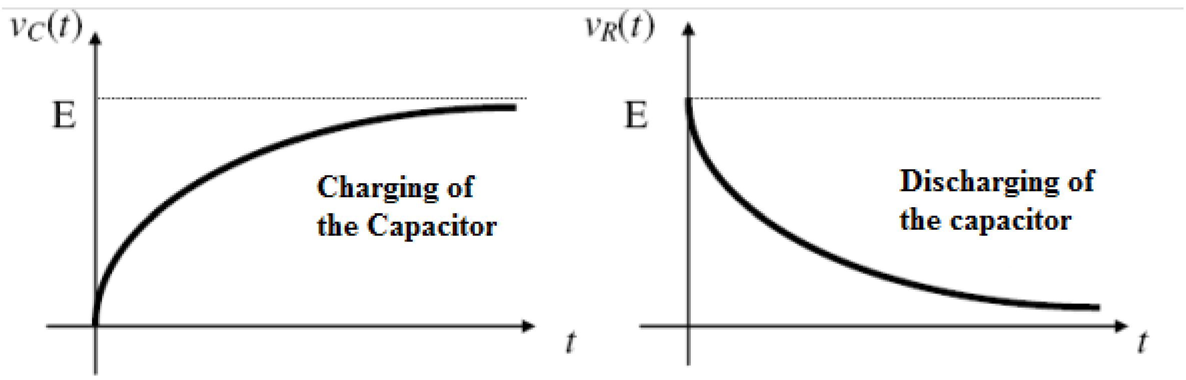


Figure 2

If you remove $e(t)$ source and its place is short circuited, ($e(t) = 0$ in equation (1)), the solution of equation (1),

$$v_C(t) = e^{-t/RC} v_C(0) \qquad (3)$$

Variance of this voltage according to time is given in the Figure 3. RC is the time constant of the circuit in equation (2) and (3); if you put R as ohm and C as farad, its unit is seconds.

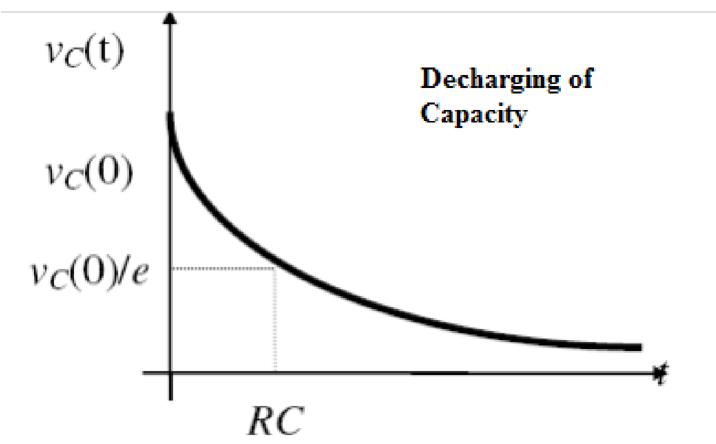


Figure 3

e(t) source is a pulse source shown in the Figure 4a , ($e(t) = E[u(t) - u(t$-D)]), $vC(0) = 0$ and voltage of the capacity;

$$v_C(t) = E(1- e^{-t/RC})\mathrm{u}(t) - E(1- e^{-(t-\Delta)/RC})\mathrm{u}(t - \Delta) \qquad (4)$$

The voltages of C capacity and R resistor are shown as Figure 4b and Figure 4c.

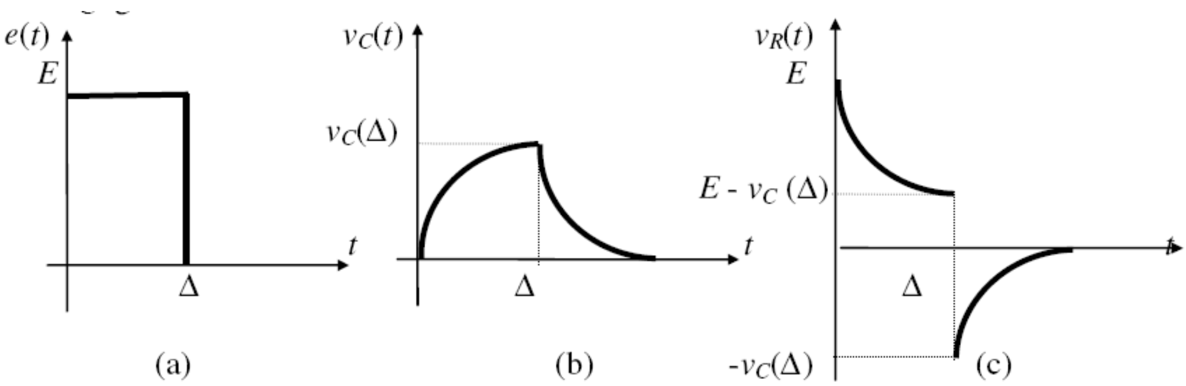


Figure 4

If e(t) source is a pulse source as shown in Figure 4a, one can think circuit in Figure 1 as Figure 5.

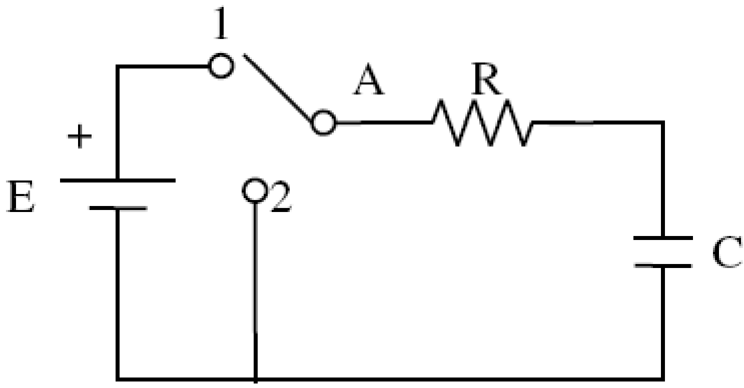


Figure 5

For this case, it is necessary to indicate that A key in Figure 5 is kept on 1 position for D time, then it is taken to position 2. When A key is on position 1, in the interval $0 \leq t < \Delta$, the voltage expression of C capacity is defined by curve from 0 to Δ shown in Figure 4b. If $t = \Delta$, the voltage of the capacity is $v_C\Delta = v_C(\Delta)$. When t equals to Δ, the capacity will start to decharge since A key is taken to position 2. The curve given in Figure 3 can be used for t ³ Δ but it is necessary to shift this curve by Δ and replace $v_C(0)$ with $v_C(\Delta)$. The operation done on the Figure 3 is equal to replacing t with (t - Δ) and $v_C(0)$ with $v_C(\Delta)$ in equation (3). These explanations can be summarized by the equation given below;

$$v_C(t) = \begin{cases} E(1 - e^{-t/RC}) & ; 0 \leq t < \Delta \\ v_C(0)\, e^{-(t-\Delta)/RC} & ; t \geq \Delta \end{cases} \qquad (5)$$

In equation (5), $v_C(0) = E(1 - e^{-(t-\Delta)/RC})$. If E, R, C and t, Δ is given as numerically, $v_C(0)$ can be computed as numerically, also. Similarly, $v_R(t)$ can be found as given below;

$$v_R(t) = \begin{cases} E\, e^{-t/RC} & ; 0 \leq t < \Delta \\ v_R(t) = () & \\ v_c(0)\, e^{-(t-\Delta)/RC} & ; t \geq \Delta \end{cases}$$

Let's assume that e(t) source in Figure 1 is a rectangular wave source as in Figure 6. It is necessary to compare the R.C time constant with the period of rectangular pulse when it is started to investigate the voltage changing on the C capacity.

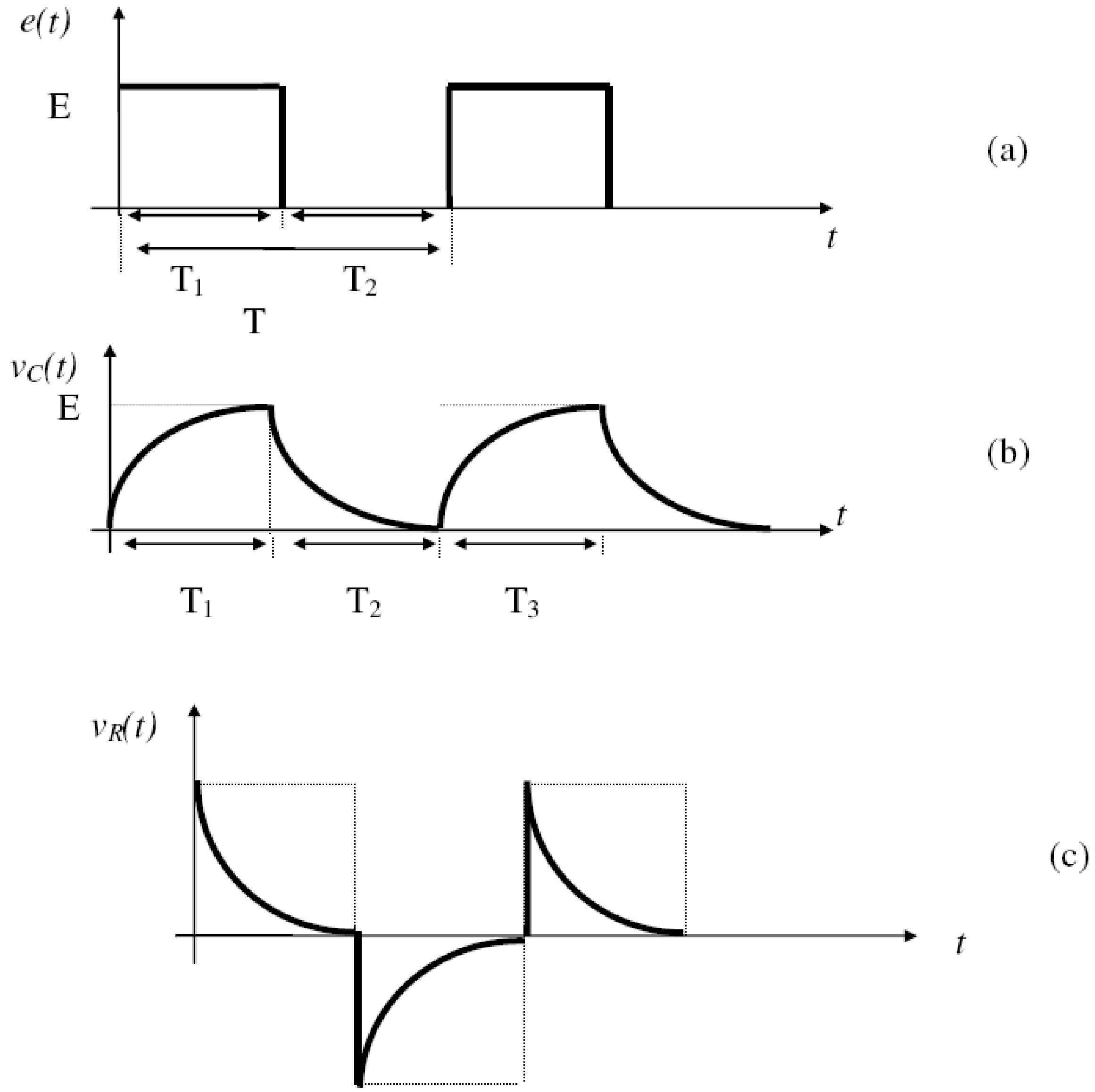


Figure 6

a) If *R.C* << *T*, capacity is charged during *T*1 time with primary pulse and it decharges until the second pulse in the *T*2 time interval. Since the time constant is small, it can be assumed that voltage in the capacity edges reaches to zero in the T2 time. The phenomena is repeated for second and afterward pulses as the same manner; voltage of the capacity changes as in Figure 6b periodically. By remarking that this voltage is periodic and replacing Δ with T1, it can be defined with equation (5). The variance of voltage on the R resistors edges is as shown in Figure 6c.

b) If *RC* time constant is enough large to compare with the period, the variance of $v_C(t)$ will be as Figure 7. Capacity will charge with first pulse, during T2 time, second pulse will come before the capacity discharge exactly. This situation will continue so for the pulses in the beginning. After a certain number of pulses, variance of the voltage on the capacity edges will be periodic.

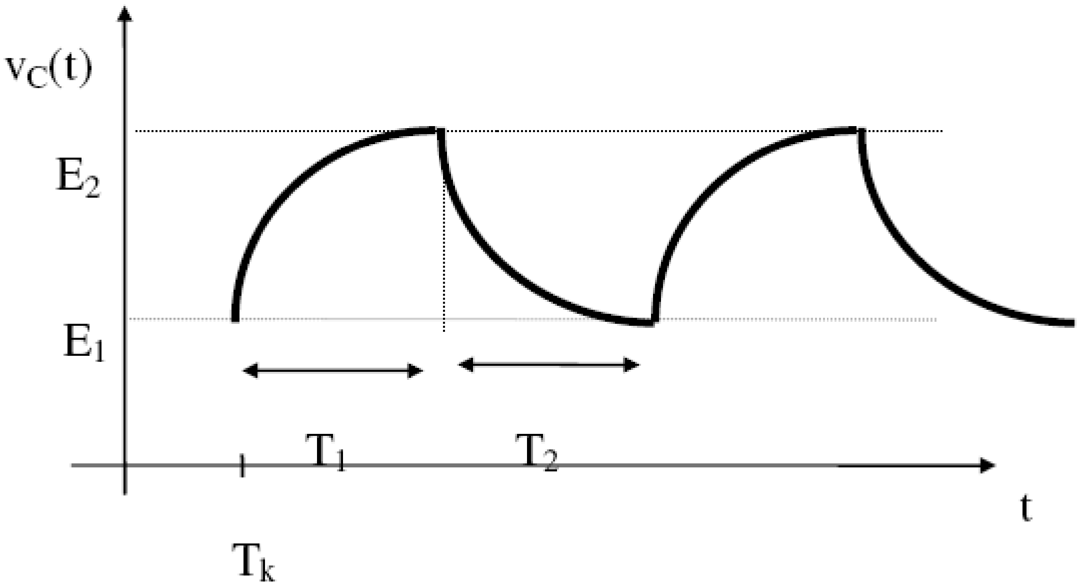


Figure 7

The time T_k when the phenomena became periodic can be considered as beginning. Because of this, for explain the phenomena equations (2) and (3) can be used;

$$v_C(t) = \begin{cases} e^{-t/RC} E_1 + E(1 - e^{-t/RC}) & ; 0 \leq t < T_1 \\ E_2 \, e^{-(t - T1)/RC} & ; t \geq T_2 \end{cases} \qquad (6)$$

If it is assumed that $v_C(T1) = E2$ at $t = T1$ time and $v_C(T) = E1$ at $t = T1+T2$ time, by using equation (6);

$$\left. \begin{aligned} E_2 &= E_1 e^{-T1/RC} + E(1 - e^{-T1/RC}) \\ E_1 &= E_2 \, e^{-T2/RC} \end{aligned} \right\} \qquad (7)$$

From this;

$$E_2 = E \frac{1 - e^{-T1/RC}}{1 - e^{-T/RC}} \qquad E_1 = E \frac{e^{-T2/RC} - e^{-T/RC}}{1 - e^{-T/RC}} \qquad (8)$$

is found. If the pulse duration of rectangular wave is T1, period is T, amplitude is E and numerical values of R and C are given by using equations in (8), E1 and E2 are calculated numerically and also (6) equations which give the variation of $v_C(t)$ with time can be found numerically. If $v_C(t)$ is clear, $v_R(t)$ can be found as given in Figure 8.

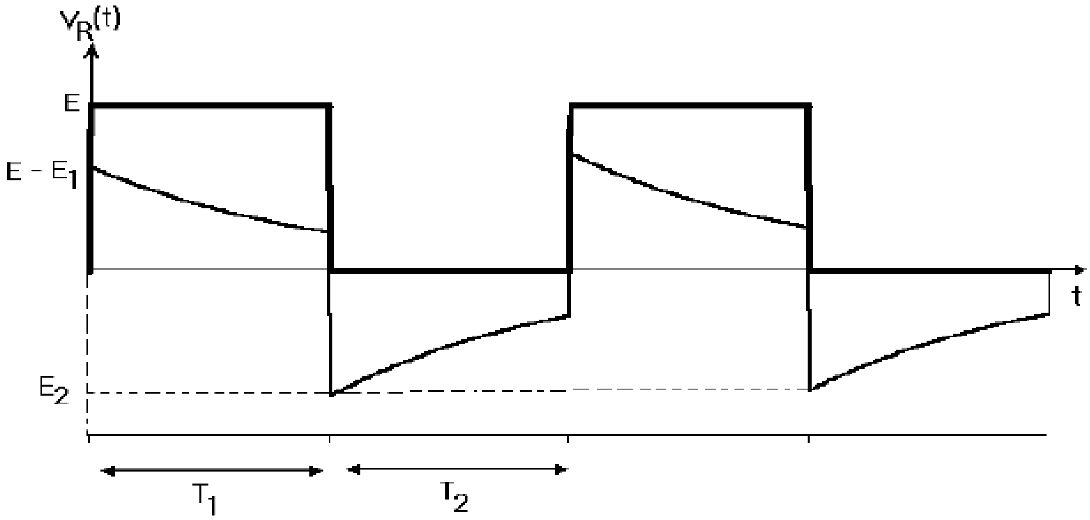


Figure 8

c) If RC >> T, (6) and (8) equations are valid again. In other words, if RC >> T,

$$e^{-t/RC} \cong 1 - t/RC \qquad 0 \leq t < T_1$$

$$e^{-(t-T1)/RC} \cong 1 - (t - T_1)/RC \qquad T_1 \leq t < T_2$$

can be written. So, the variation of $v_C(t)$ will be linear as shown in Figure 9.

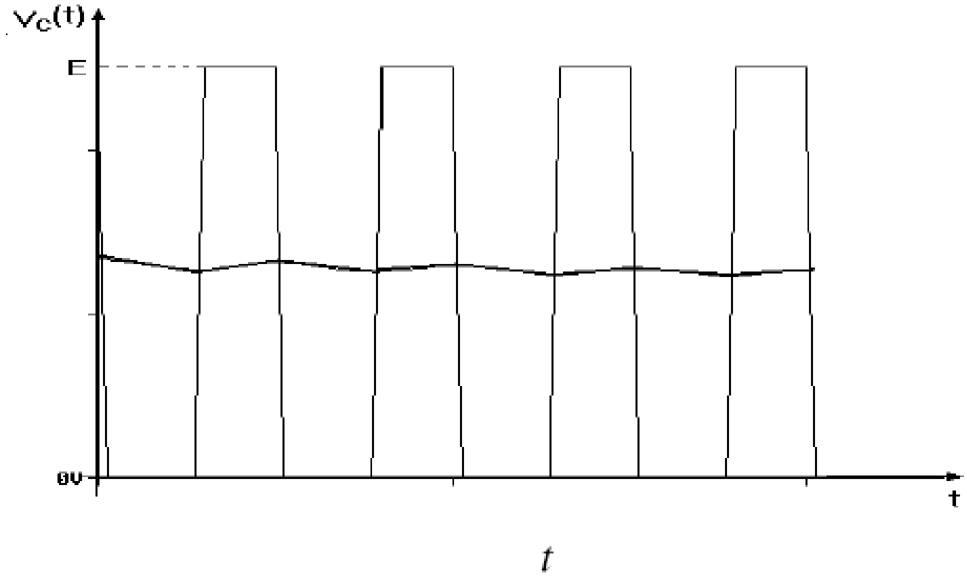
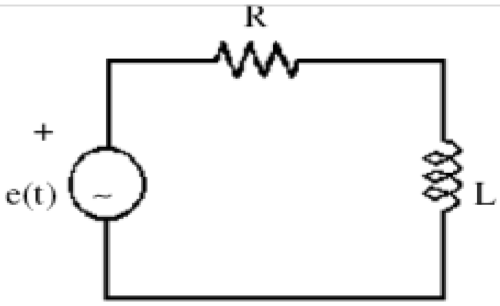


Figure 9

RL Circuit :

Let’s consider RL circuit in Figure 10.



$$\frac{di_L}{dt} = -(R/L)i_L(t) + \frac{1}{L}e(t) \tag{9}$$

Figure 10

The structure of equation (9) is same as equation (1). You can apply all steps for RL circuit like as RC circuit. The circuit shown in Figure 11 can be used for modelling an energy distribution system simply. Here, e(t) is a source with 50Hz frequency.

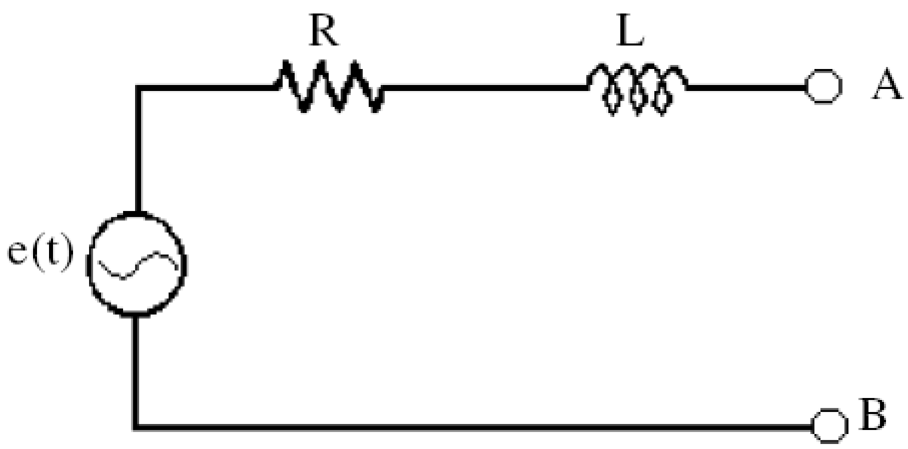


Figure 11

$$e(t)=\sqrt{2}E\sin(\omega t) \tag{10}$$

In this case, particular solution of equation (9) is;

$$i_L(t) = \frac{\sqrt{2}E}{Z}\sin(\omega t - \psi) \tag{11}$$

This equation is;

$$\left.\begin{aligned} Z &= \sqrt{R^2 + \omega^2 L^2} \\ \psi &= \mathrm{arctg}\left(\frac{\omega L}{R}\right) \end{aligned}\right\} \tag{12}$$

The solution of the homogen part of the (9) equation is,

$$i_L(t) = I_h e^{-(R/L)t} \tag{13}$$

Exact solution is;

$$i_L(t) = I_h e^{-(R/L)t} + \frac{\sqrt{2}E}{Z}\sin(\omega\, t\text{-}\psi\,) \qquad (14)$$

Before it is short circuited, there is no current transition on the inductance, $i_L(0)$ is 0. This first condition is yield in equation (4),

$$I_h(\text{t}) = \frac{\sqrt{2}E}{Z}\sin\psi \qquad (15)$$

is obtained. So, equation (14) can be written as;

$$i_L(t) = (\frac{\sqrt{2}E}{Z}\sin\psi\,)e^{-(R/L)t} + \frac{\sqrt{2}E}{Z}\sin(\omega\, t\text{-}\psi) \qquad (16)$$

First term is transient solution and second term is steady solution in equation (16). After a time as five times of (L/R), transient solution can be ignored. After a time stated from the short circuit current is appeared, short circuit current pass whose effective value of the circuit equals to (E/Z). $i_L(t)$ gets its maximum value for $\omega t = \pi/2 + \psi$. This value of current is named as short circuit pulse current and defined as equations 17a and 17b;

$$I_s = \frac{\sqrt{2}E}{Z}\left(1 + \frac{\omega\, L}{Z} e^{-\frac{R}{\omega\, L}\left(\frac{\pi}{2}+\psi\right)}\right) \qquad (17a)$$

$$\text{I}_\text{s} = \sqrt{2}\ \text{I}''_\text{k}\ x \qquad (17b)$$

Here, I_k'' is the value of steady short circuit current and equals to E/Z.

$$x = 1 + \left(1/\sqrt{\left(\frac{R}{\omega\, L}\right)^2 + 1}\right)\text{e}^{-\left(\frac{R}{\omega\, L}\left(\frac{\pi}{2}+\psi\right)\right)} \qquad (18)$$

As seen from equation (18), coefficient x is a constant dependent to R/(wL). If R=0, x value will be 2. So, the current passes in the circuit because of the transient solution is the twice of steady short circuit current. Consider that here, generator is modelled with a voltage source and an internal resistance and an internal inductance connected series to it.

RLC Circuit:

Let’s consider RLC circuit given in Figure 12.

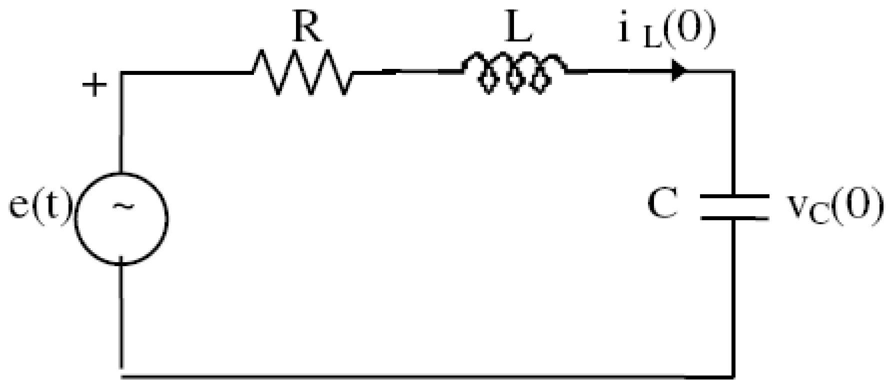


Figure 12

State equations of this circuit are,

$$\frac{d}{dt}\begin{bmatrix} v_C(t) \\ i_L(t) \end{bmatrix} = \begin{bmatrix} 0 & (1/C) \\ (-1/L) & (-R/L) \end{bmatrix}\begin{bmatrix} v_C(t) \\ i_L(t) \end{bmatrix} + \begin{bmatrix} 0 \\ (1/L) \end{bmatrix} e(t) \qquad (19)$$

Characteristic equation of this equation system is,

$$p^2 + 2\zeta\,\omega_o p + \omega_o^{\,2} = 0 \qquad (20)$$

$$\omega_o = \frac{1}{LC} \quad , \quad \zeta = \frac{R}{2}\sqrt{\frac{C}{L}} \qquad (21)$$

By taking $v_c(0) = 0$, $i_L(0) = 0$, for three situation the variance of $v_c(t)$ with time is drawn in Figure 13 rudely.

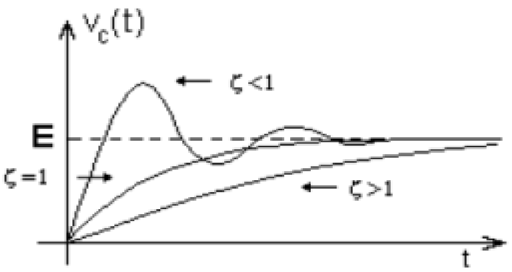


Figure 13

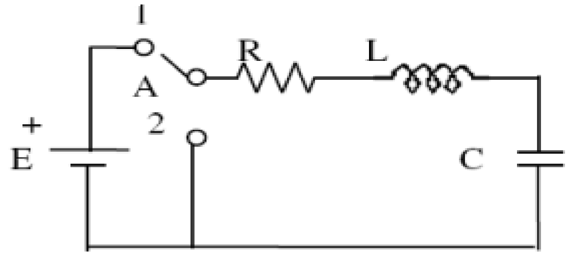


Figure 14

As seen in Figure 13, the voltage on the capacity edges goes to E for both of the three condition. Let's consider e(t) is a source in the form of pulse as E(u(t)-u(t-D)). In this case, it will be easy to interpret the circuit as Figure 14. After A key is kept on (1) position for D time, it will be taken to position (2). Assume that the initial conditions are $v_C(0)$, $i_L(0)$. During the key is kept on position (1), in the interval of $0 \leq t < \Delta$, by considering value of z, one can use either equation (23) or (24) and (25). At the time $t = \Delta$, we can find $i_L(\Delta)$ and $v_C(\Delta)$ by using these equations. If key is on the position (2), since there is no source in the circuit, only natural solution is obtained. For $t > \Delta$, in order to find $v_C(t)$ and $i_L(t)$, it can be used one of the equations of (23), (24), (25) in terms of z values. For $t > \Delta$, in order to determine $v_C(t)$ and $i_L(t)$, it is enough to write $(t - \Delta)$ instead of t; $v_C(0)$ and $i_L(0)$ instead of $v_C(\Delta)$ and $i_L(\Delta)$; zero instead of E. If we assume that Δ is big enough, the change of $v_C(t)$ in terms of ζ will be as shown in Figure 15.

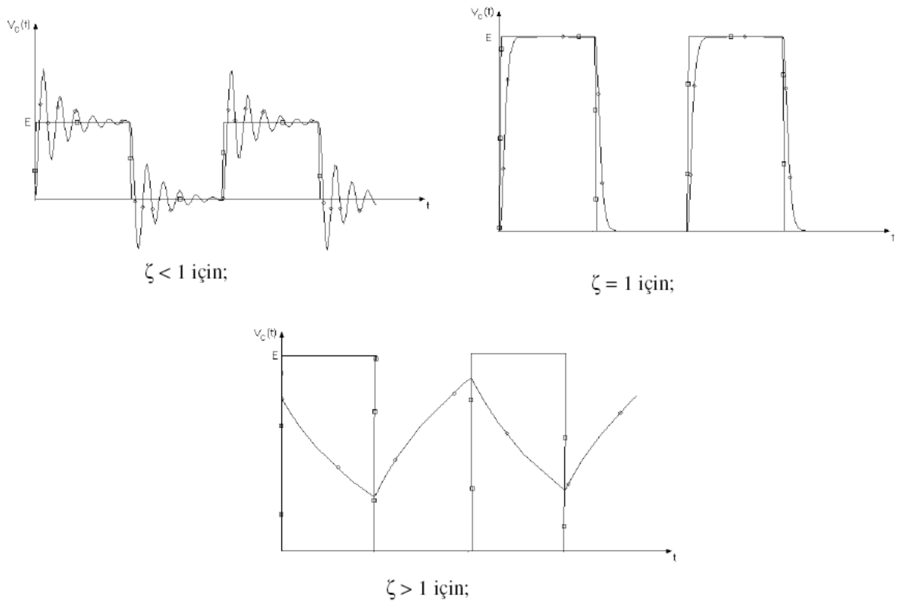


Figure 15

LABORATORY EXPERIMENTS

EXPERIMENT-I : Frequency, Amplitude and Phase Measurement for Basic AC /RC Circuits

EXPERIMENT-II : Basic AC RC and RL Circuits

EXPERIMENT-III : Serial AC RLC Circuits

EXPERIMENT-IV : Parallel RLC Circuits

EXPERIMENT-V : Power in AC Circuits

EXPERIMENT-VI : Application of Transformer

EXPERIMENT-VII : Balanced Three-Phase Y-Y Connected Circuits

EXPERIMENT-VIII: Balanced Three-Phase Δ-Δ Connected Circuits

EXPERIMENT-IX : Passive Filters

I. EXPERIMENT: Frequency, Amplitude and Phase Measurement for Basic AC /RC Circuits

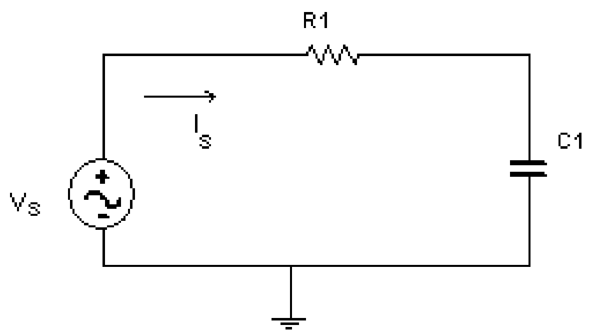
PURPOSE: The purpose of this experiment is to investigate amplitude and phase for serial AC RL and RC circuits. Theorical knowledge about this subject is given in EEM 204 lecture.

PRELIMINARY STUDY: All formulas will be derivatized and each values will be calculated for the circuit given below. So you can compare the theorical values calculated and experimental values measured during the experiment and write conclusion at the end of the report about the differences between these values you obtained.

EXPERIMENT :

1. Generate $v_s(t)=10\cos(2000\pi t)$ signal by using function generator and show it on the oscilloscope screen (channel 1).
2. Design the circuit given in the figure: Show Is, V_{C1} and V_{R1} on Channel 2.
3. Draw I_s, Vs, V_{C1} ve V_{R1}. Determine the frequency, period, amplitude and phase values by measuring from oscilloscope.
4. Measure the RMS values of I_s, Vs, V_{C1} and V_{R1} by using multimeter.

CIRCUIT DIAGRAM :



$R_1=330\ \Omega$ $C_1=1,5\ \mu F$

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

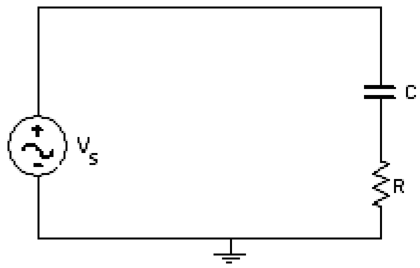
2. Compare the theorical and measurement values of the circuit.

II. EXPERIMENT: Basic AC RC and RL Circuits

PURPOSE: The purpose is to learn about basic RC and RL AC circuits analysis in this experiment.

PRELIMINARY STUDY:

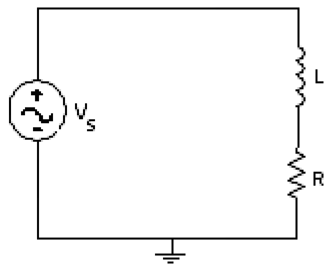
1. Find the V_R and V_C voltages of basic AC-RC circuit given in Figure 1 by using phasor representation. Input AC voltage' frequency and values of elements used in the circuit are given in the table. V_S=10Cos(2πft)



f (frequency)	100 Hz, 1kHz, 5kHz
C	1μF
R_1	1 kΩ, 10 kΩ

Figure 1. RC Circuit

2. Find the V_R and V_L voltages of basic AC-RC circuit given in Figure 1 by using phasor representation. Input AC voltage' frequency and values of elements used in the circuit are given in the table. V_S=10Cos(2πft)



f (frequency)	100 Hz, 1kHz, 2kHz, 5kHz
L	100 mH
R	330 kΩ

Figure 2. RL Circuit

<u>EXPERIMENT</u> :

1. Design the circuit given in Figure 1 on the breadboard. Show the voltage V_S with V_L,V_R voltages on the oscilloscope screen by changing input AC voltage frequencies for 100 Hz,1kHz, 2kHz, 5kHz. Determine how measures differ if R and C values are changed.

2. Set up the circuit given in Figure 2 on the breadboard. Show the voltages V_L,V_R with V_S on the oscilloscope screen by changing frequency values of input AC voltage for 100 Hz,1kHz, 2kHz, 5kHz. Draw the graphics on the squared paper. Determine how the measurements differ by changing R and C values.

CONCLUSION and COMMENTS

Compare and interpret the results you obtained from experiments. Express the results clearly.

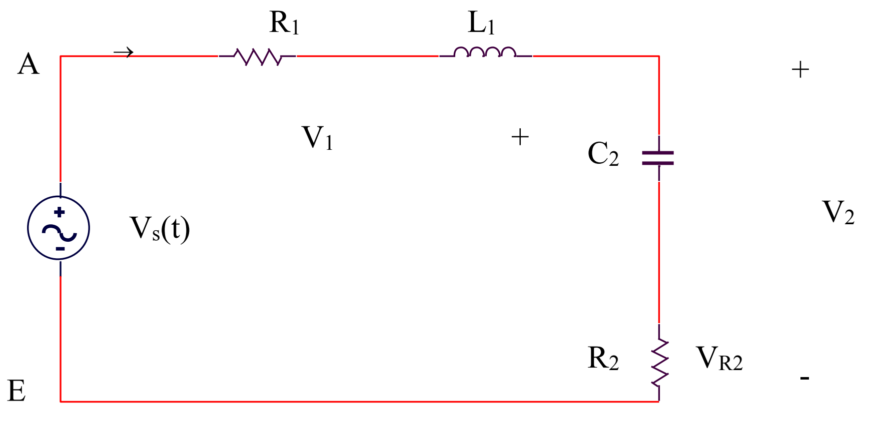
III. EXPERIMENT: Serial AC RLC Circuits

PURPOSE: The purpose of this experiment is to apply voltage divider rule for serial AC circuits. The theorical knowledge about this subject is given in the content of EEM 204 lesson. Voltage divider circuit will be analysed first mathematically, then experimentally. Lastly, the mathematically and experimental results will be compared.

PRELIMINARY STUDY:

In the circuit below $V_s(t)$ is defined as $V_s(t)=5\cos(1000\pi t)$.

1) Calculate the amplitude and phase angles for V_1, V_2, V_{R2} by using the values given in the table.
2) Calculate the amplitude and phase angles for current I_s.



Resistor	Value
R_1	1 kohm
R_2	0.82 kohm
L_1	100 mH,1mH
C_2	0.2 µF

<u>EXPERIMENT</u>:

1. Set up the circuit in the figure and obtain Vs voltage on the oscilloscope.
2. Measure the amplitude or phase angles of V_2 and V_{R2} according to Vs (Channel 1). Note: Connect the ground node of oscilloscope to E node.
3. Measure the amplitude or phase angles of V_1 according to Vs (Channel 1). Note: Connect the ground node of oscilloscope to A node.

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

IV. EXPERIMENT: Parallel RLC Circuits

PURPOSE: The purpose of this experiment is to apply voltage divider rule for parallel AC circuits. The theorical knowledge about this subject is given in the content of EEM 204 lesson. Voltage divider circuit will be analysed first mathematically, then experimentally. Lastly, the mathematically and experimental results will be compared.

PRELIMINARY STUDY:

In the circuit below $V_s(t)$ is defined as $V_s(t)=5\cos(1000\pi t)$.

1. Calculate the amplitude and phase angles for V_1, V_2, V_{R2} by using the values given in the table.
2. Calculate the amplitude and phase angles for current I_s.

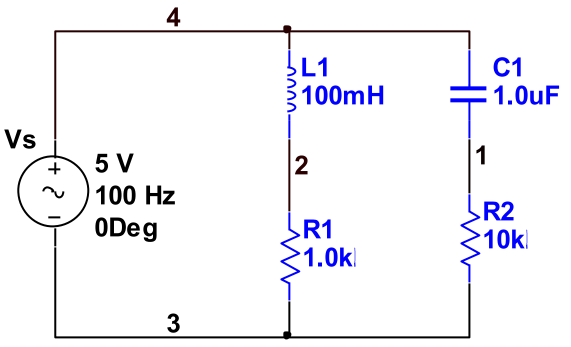


Figure 1

Resistor	Value
R_1	1 kohm,10kohm
R_2	10 kohm
L_1	1mH,100mH
C_2	1 μF,10μF

EXPERIMENT :

1. Set up the circuit in the figure and obtain Vs voltage on the oscilloscope.
2. Measure the amplitude or phase angles of V_2 and V_{R2} according to Vs (Channel 1).

3. Measure the amplitude or phase angles of V_1 according to Vs (Channel 1).

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

V. EXPERIMENT: Power in AC Circuits

PURPOSE: The purpose of this circuit is investigating the power analysis in AC circuits.

EXPERIMENT :

1. Set up the circuit given in Figure 1 except capacitor. Use values in the table. $V_s(t)=10\cos(2000\pi t)$.
2. Calculate voltage Vs and current I_s. Calculate phase difference between V_{R1} and Vs.
3. Connect capacitor as seen in the figure and repeat step 2.
4. Interpret the results.

R_1	100 ohm
R_2	1 kohm
L_1	100 mH
C_1	1 μF

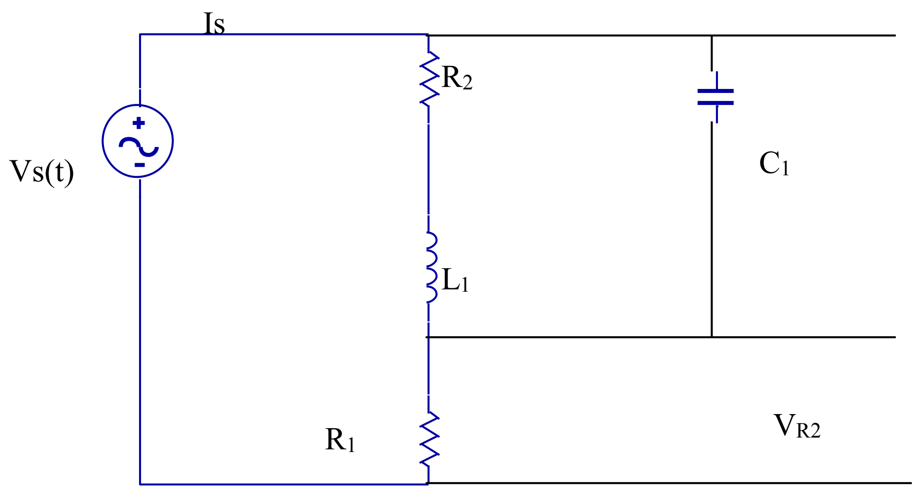


Figure 1

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

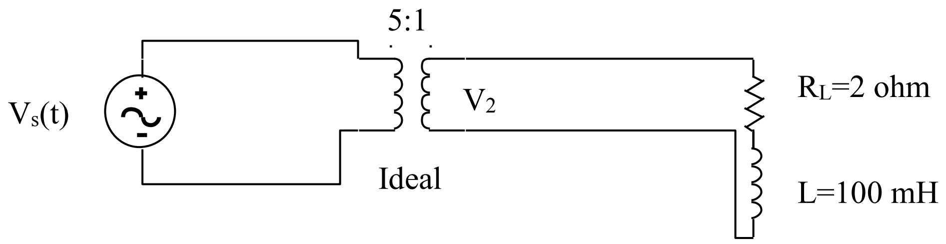
2. Compare the theorical and measurement values of the circuit.

VI. EXPERIMENT: Application of Transformer

PURPOSE: In this experiment, voltage, current and power analysis of basic transformer circuit will be done theorically and experimentally.

PRELIMINARY STUDY:

1. In the figure below Vs(t) is defined as $V_s(t)=5\cos(2000\pi t)$. Find V_2 and I_2.
2. Determine the average power and reactive power which source generated.
3. Determine the average power and reactive power consumed on the load.



<u>EXPERIMENT</u> :

1. Set up the circuit given in the figure. Measure the amplitude, phase and frequency of V_2, I_2.
2. Measure the average power and reactive power that source generated.
3. Measure the average power and reactive power consumed on the load.

CONCLUSION and COMMENTS

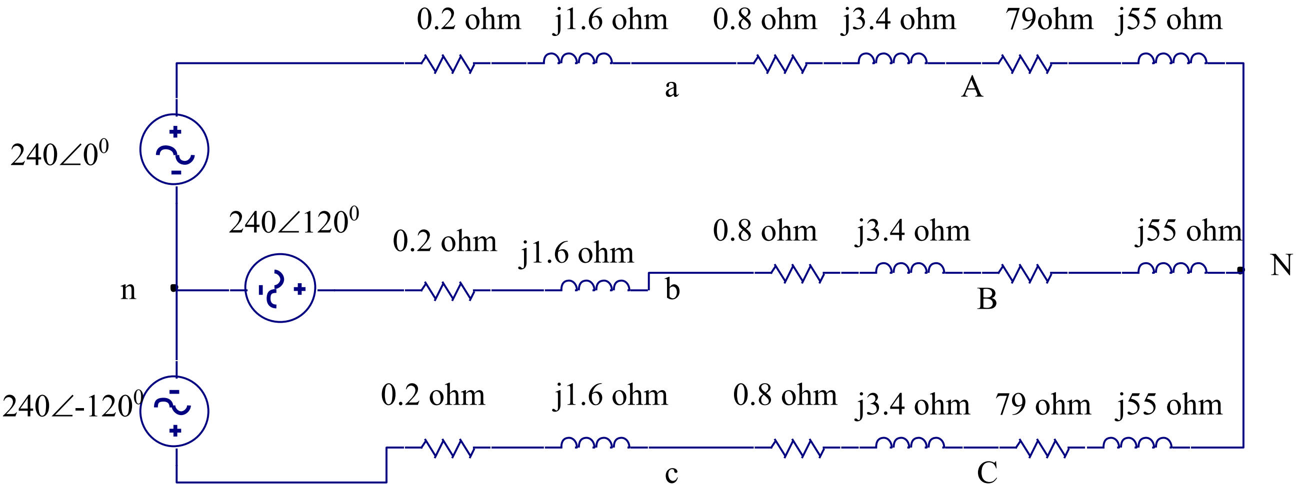
1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

VII. EXPERIMENT: Balanced 3-Phase Y-Y Connected Circuits

PURPOSE: The purpose of this experiment is to investigate voltage, current and power analysis for 3-phase Y-Y connected circuits experimentally and theoretically.

PRELIMINARY STUDY:



1. F is 2000 Hz for the 3-phase circuit given in the figure. Determine the line and phase voltage and current for the load (as amplitude and phase).
2. Find average power that each load used up. Find the average power that load used up.
3. Find the total average power that load used up by using 2-wattmeter method. Compare the result with you found in step 2.
4. Find the percent ratio of average power that got lost in the transmitting line (single phase) to the power generated by source (single phase).

EXPERIMENT:

1. F is 2000 Hz for the 3-phase circuit given in the figure. Determine the line and phase voltage and current for the load (as amplitude and phase).
2. Measure the power that each phase load used up (According to the ground). Measure the total average power that load used up.
3. Find the total power that load used up by using 2-wattmeter method. Compare the result with you found in step 2.

4. Measure the average power lost in the transmitting line (single phase) and power generated by source (single phase). Find the loss ratio.

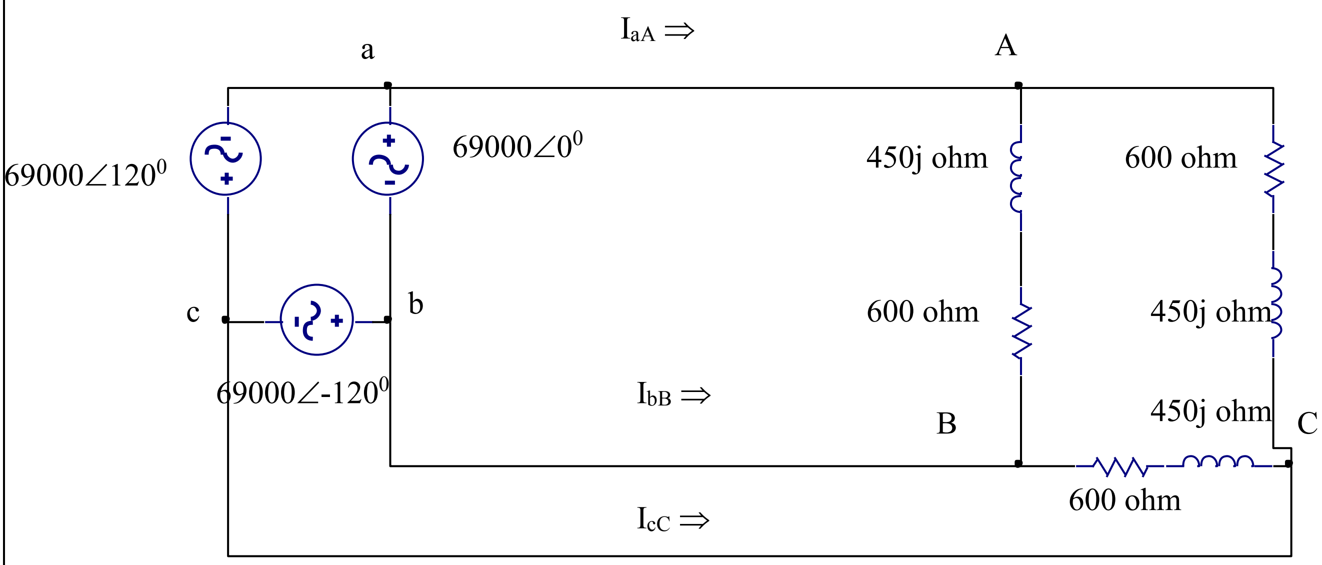
CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

VIII. EXPERIMENT: Balanced 3-Phase Δ-Δ Connected Circuits

PURPOSE: The purpose of this experiment is to investigate voltage, current and power analysis for 3-phase Δ-Δ connected circuits experimentally and theoretically.



PRELIMINARY STUDY:

For the 3-phase circuit in the figure, f is 2000 Hz.

1. Determine the currents I_{AB}, I_{BC}, I_{CA}, I_{aA}, I_{bB}, I_{cC}, I_{ba}, I_{cb}, I_{ac} .
2. Find the voltages V_{AB}, V_{BC}, V_{CA} .
3. Calculate the power in each phase of the load.

EXPERIMENT:

1. Measure the currents I_{AB}, I_{BC}, I_{CA}, I_{aA}, I_{bB}, I_{cC}, I_{ba}, I_{cb}, I_{ac} .
2. Measure voltages V_{AB}, V_{BC}, V_{CA} .
3. Measure the power in each phase of the load.

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

IX. EXPERIMENT: Passive Filters

PURPOSE: The purpose of this experiment is to investigate the responses of passive filters for different amplitudes and frequencies.

PRELIMINARY STUDY:
Determine the transfer function for the circuits 1, 2, 3 and 4 and calculate the cut-off frequencies.

EXPERIMENT:

A) Low-Pass Filter

1) Design a low-pass filter circuit by using 10 kohm and 1μF for R and C, respectively. Adjust the amplitude of signal source for 4 V (p-p), measure the output voltages for 1, 2, 4, 6, 8, 10, 12, 14, 16, 18, 20, 40, 60, 80 and 100 (kHz) input frequencies.
2) Convert the voltage values you measured to decibel by using the formula given below,

$$dB=20 \log (V_0/V_i) \quad (V_i \text{ is constant at } 4 \text{ V (p-p)})$$

3) Draw the decibel values according to frequency by using semilog graphic paper.
4) Determine the actual cut off frequency by using drawing.

B) High-Pass Filter

Design a high-pass filter circuit with C=1 μF and R=4,7 kohm and realize the process steps 2-4 given above.

C) Band-Pass Filter

Design a band-pass filter circuit with C_1=1 μF, C_2=1 μF and R_1=10 kohm, R_2=10 kohm and realize the process steps 2-4 given above.

D) Band Stop Filter

Design a band-stop filter circuit C_1=1000 pF, C_2=1000 pF, C_3=2000 pF and R_1=47 kohm, R_2=47 kohm, R_3=47 kohm and realize the process steps 2-4 given above.

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.