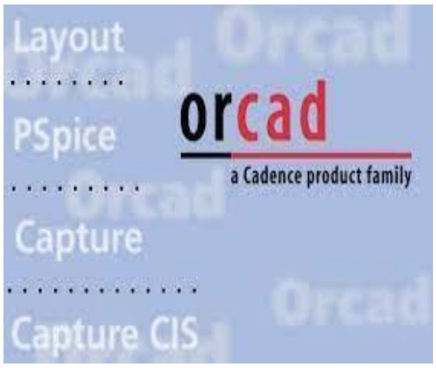
VASAVI COLLEGE OF ENGINEERING

(AUTONOMOUS)

IBRAHIMBAGH, HYDERABAD-31

Department of Electrical and Electronics Engineering



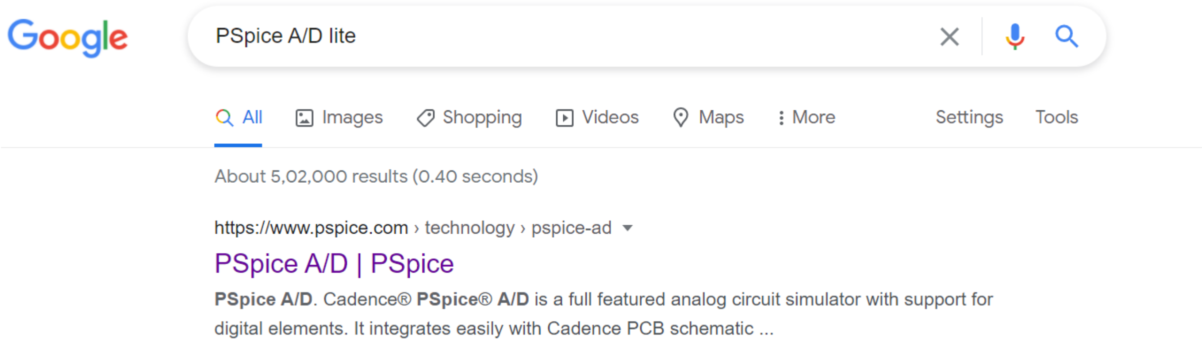
Simulation of Electric Circuits using PSpice

Contents

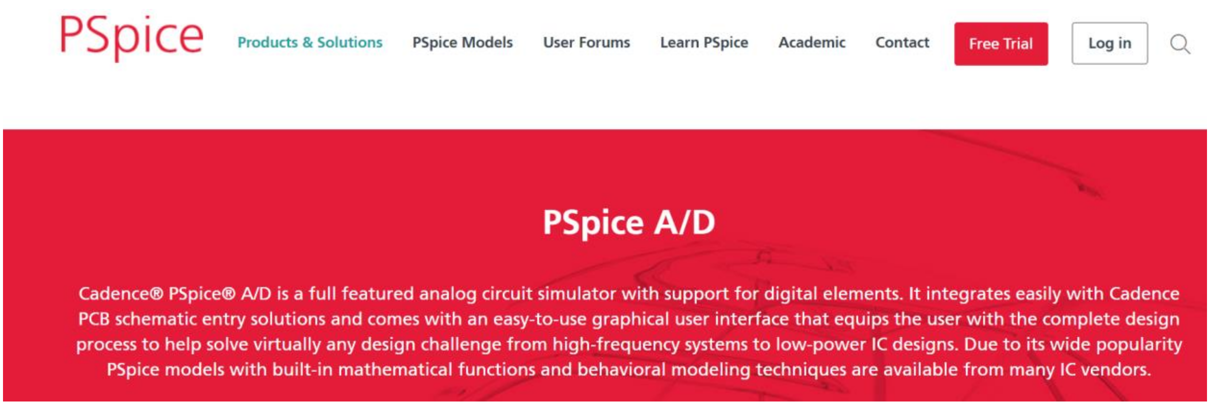
1.GETTING STARTED WITH PSPICE

Let us know how to download and install PSpice in your personal computer/laptop. Follow the below instruction to install PSpice free trial.

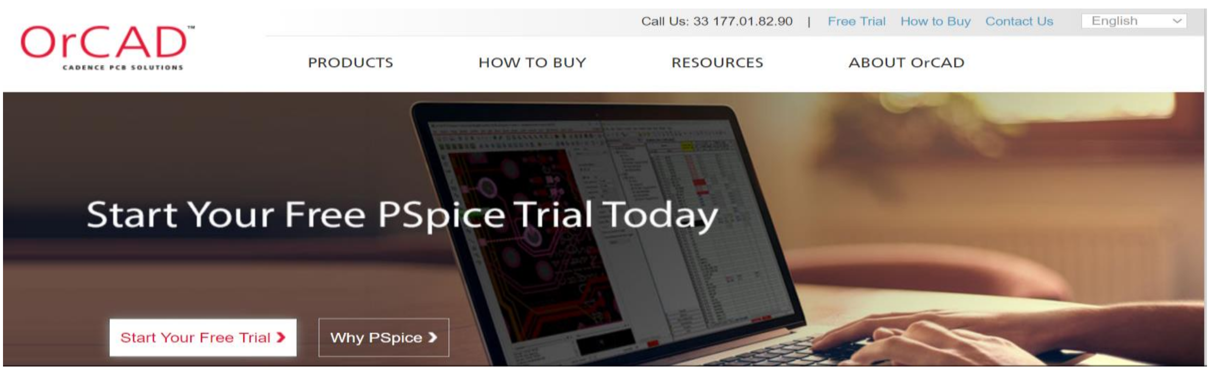
Step 1: Type “PSpice A/D lite” in the Google browser. You can see the below result.



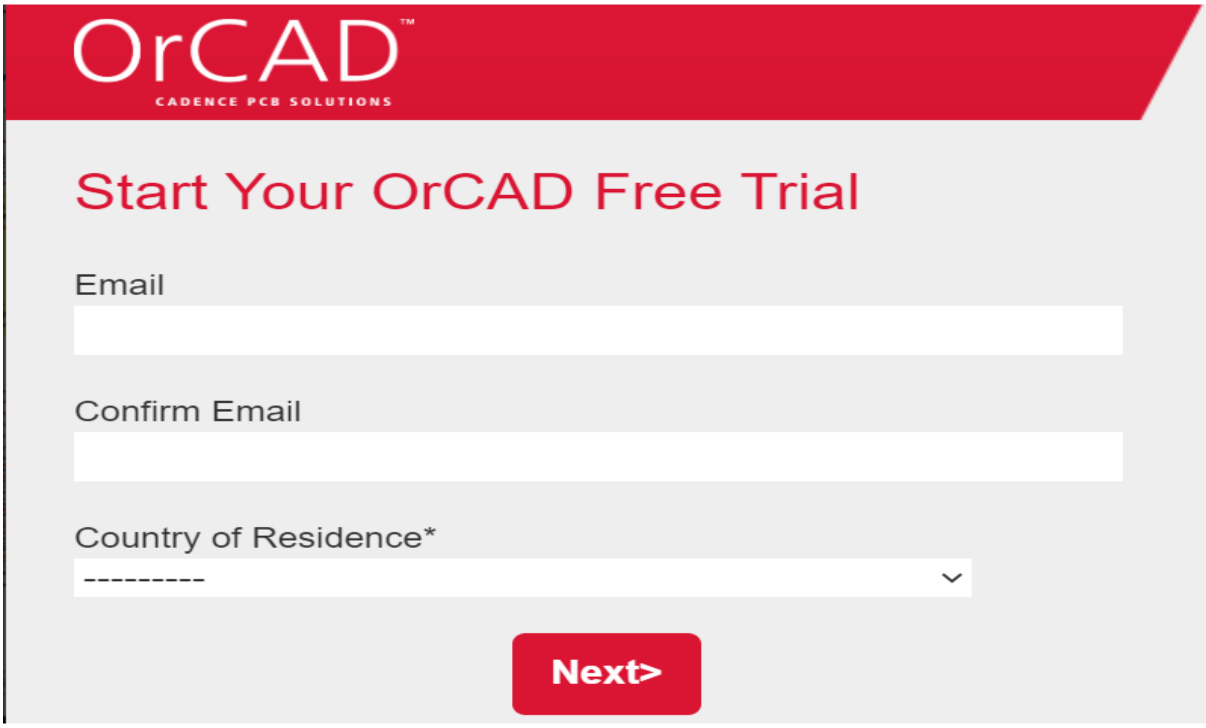
Step 2: Click on the link “https://www.pspice.com › technology › pspice-ad” , you can shifted to the PSpice technology (See the image below).



Step 3: Click on “Free Trial”, you will directed to orcad.com



Step 4: Click on "Start your Free Trial"



Step 5: Fill your details, you will get mail communication from orcad team. Start download the file and then install in your personal computer/laptop.

Step 6: After successful installation, goto Start>> Programs>> Orcad Release >> PSpice

(Courtesy: websites:: http://google.com, http://pspice.com and http://orcad.com)

2. INTRODUCTION TO PSPICE

SPICE stands for Simulation Program with Integrated Circuit Emphasis. PSPICE stands for PC version of SPICE. For computer aided analysis of circuits we can use PSPICE as a tool. There are two ways of circuit simulation in PSpice, namely, PSpice Schematics and PSpice A/D. In PSpice schematics we should draw the circuit in simulator using components and then go for analysis. Where as in PSpice A/D, we do analysis using coding rules. Here the analysis of circuits (throughout the text) is done with PSpice A/D simulation tool.

Some rules and facts about PSpice are listed under.

1. PSpice is not case sensitive. This means that names such as Vbus, VBUS, vbus and even vBuS are equivalent in the program.
2. All element names must be unique. Therefore, you can't have two resistors that are both named "Rbias," for example.
3. The first line in the data file is used as a title. It is printed at the top of each page of output. You should use this line to store your name, the assignment, the class and any other information appropriate for a title page. PSpice will ignore this line as circuit data. Do not place any actual circuit information in the first line.
4. There must be a node designated "0." (Zero) This is the reference node against which all voltages are calculated.
5. Each node must have at least two elements attached to it.
6. The last line in any data file must be ".END" (a period followed by the word "end.")
7. All lines that are not blank (except for the title line) must have a character in column 1, the leftmost position on the line.
    - Use "*" (an asterisk) in column 1 in order to create a comment line.
    - Use "+" (plus sign) in column 1 in order to continue the previous line (for better readability of very long lines).
    - Use "." (period) in column 1 followed by the rest of the "dot command" to pass special instructions to the program.
    - Use the designated letter for a part in column 1 followed by the rest of the name for that part (no spaces in the part name).
8. Use "whitespace" (spaces or tabs) to separate data fields on a line.

9. Use ";" (semicolon) to terminate data on a line if you wish to add commentary information on that same line.

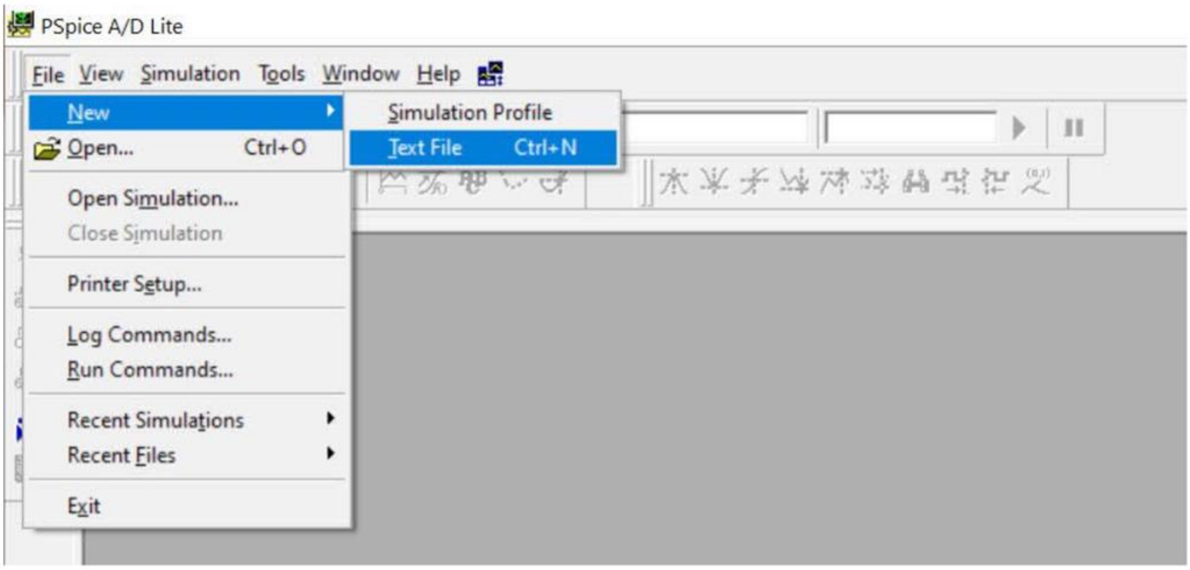
The above basic information is essential to use PSpice to write the code in simulator.

After learning the guidelines to write the program, we will start working on PSpice A/D.

Goto Start>> Programs>> Orcad Release >> PSpice (Click)

The workspace window for PSpice A/D will open as shown below

Then Select “File>> Text Window”



Later, write the sequence of instruction as per circuit node points and the values specified for the components.

After completion of coding, in order to save the file use extension “filename.cir”

Example: abc.cir

At the time of running the code, there are two extension files will create (as per the code instructions). Those are “filename.out” and “filename.prb”.

If circuit execution will success, filename.out will display code of instructions along with output specified parameters. Else it display the code along with errors.

With the filename.prb, we will obtain the simulation results in graphical manner.

3.SYNTAX FOR ELECTRICAL ELEMENTS

An electrical circuit is the interconnection of electrical elements. Those are voltage sources, current sources, resistors, inductors and capacitors. All the elements are having their own units. PSpice is a computer program used mostly by engineers and scientists. Accordingly, it was created with the ability to recognize the typical metric units for numbers. Unfortunately, PSpice cannot recognize Greek fonts or even upper vs. lower case. Thus our usual understanding and use of the standard metric prefixes has to be modified. The list of metric prefixes are mentioned under.

Number	**Prefix**	***Common Name***
10^{12}	"T" or "t"	*tera*
10^{9}	"G" or "g"	*giga*
10^{6}	"MEG" or "meg"	*mega*
10^{3}	"K" or "k"	*kilo*
10^{-3}	"M" or "m"	*milli*
10^{-6}	"U" or "u"	*micro*
10^{-9}	"N" or "n"	*nano*
10^{-12}	"P" or "p"	*pico*
10^{-15}	"F" or "f"	*femto*

These are the large and small numbers used by engineers, to specify voltage/current/resistance/capacitance/inductance.

An alternative to this type of notation, which is in fact, the default for PSpice output data, is "textual scientific notation." This notation is written by typing an "E" followed by a signed or unsigned integer indicating the power of ten. Some examples of this notation are shown below:

156,000 = 1.56E5

-0.0000335 = -3.35E-5

9460000 = 9.46E6

Now we come across the representation of large and small values of electrical elements. Let us look at the syntax of electrical elements in an order.

Ideal Independent Voltage Source and Ideal Independent Current Source:

To understand the syntax of ideal voltage source and ideal current source, let us consider a simple circuit as shown below.

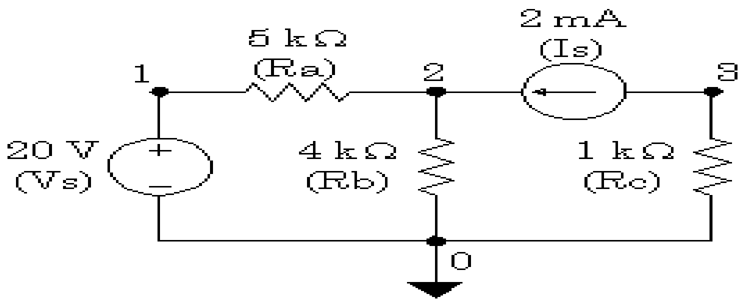


figure (1)

Ideal Independent Voltage Source:

The beginning letter for the representation of the ideal independent voltage source is "*V*." After the ‘V’ we can add any alphabet/numeric for suit of given circuit. From the sample circuit shown in figure (1), we can write the syntax for the voltage source as below.

Syntax:

```
*name  +node  -node   type   value   comment
  Vs      1      0      DC    20.0V;  "V" after "20.0"  is optional
```

Where **+node** means positive node; **-node** means negative node.

Ideal Independent Current Source:

The beginning letter for the representation of the ideal independent current source is "*I*." After the ‘I’ we can add any alphabet/numeric for suit of given circuit. From the sample circuit shown in figure (1), we can write the syntax for the voltage source as below.

Syntax:

*name	+node	-node	type	value	comment
Is	3	2	DC	2.0MA;	2mA flows from node 3 to 2

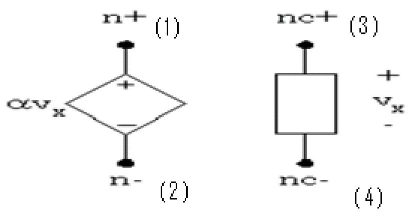
Resistor:

The first letter in the representation of a resistor must be "R"

Syntax:

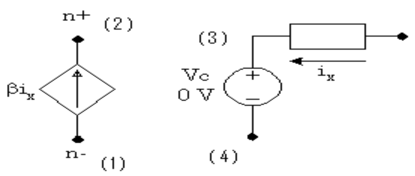
*name	+node	-node	value	comment
Ra	1	2	5k ;	reported current from 1 to 2
Rb	2	0	4k ;	reported current from 2 to 0
Rc	3	0	1k ;	reported current from 3 to 0

Voltage Controlled Voltage Source



Syntax:	***Name***	***n+***	***n-***	***nc+***	***nc-***	***gain(α)***
	Ebar	**1**	**2**	**3**	**4**	**24.0**

Current Controlled Current Source



Syntax:	****Name***	***n-***	***n+***	***Vmonitor***	***Gain***
	Fvc	**1**	**2**	**Vc**	**50.0**
	Vc	**3**	**4**	**DC**	**0V ; controls Fvc**

Linear Inductors in PSpice

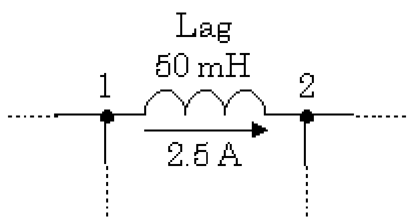
Name of linear inductor begins with the letter" L" in column 1 of the source listing.

Syntax:

Name	**nodelist**	**L_val**
Lag	**1 2**	**50m IC=2.5**
	(or)	
Lag	**1 2**	**50mH IC=2.5A;** IC is the initial condition as current

The inductance of the above element is 50mF. This can be represented as "50m" in PSpice. Note that the initial current is assumed to flow from the first node in the node list through the inductor towards the second node in the node list. If there is a need to change the direction of this initial current, either reverse the order of the nodes in the node list or place a minus sign in front of the value of the initial current.

Note: The "H" for henrys and the "A" for amps will be ignored by PSpice.



INDUCTOR

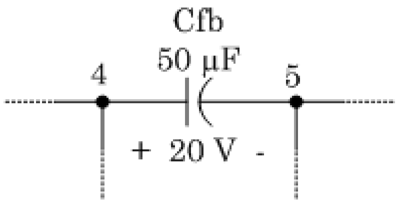
Linear Capacitors in PSpice

Name of linear capacitor begins with the letter" L" in column 1 of the source listing.

Syntax:

Name	**nodelist**	**C_val**
Cfb	**4 5**	**50uF IC=20**
	(or)	
Cfb	**4 5**	**50uF IC=20V;** IC is the initial condition as voltage

Note: PSpice would ignore the "F" for farads and the "V" for volts.

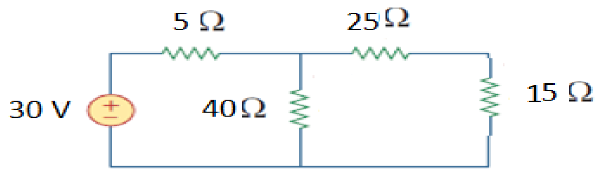


CAPACITOR

The capacitance of the above element is 50µF. This can be represented as "50u" in PSpice. Note that the polarity of the initial voltage (as shown) is such that the positive side is the first node in the list with the negative side on the second node in the list. To reverse the polarity of the initial voltage for the simulation, either reverse the order of the nodes in the node list or place a minus sign in front of the value in the "IC=" phrase.

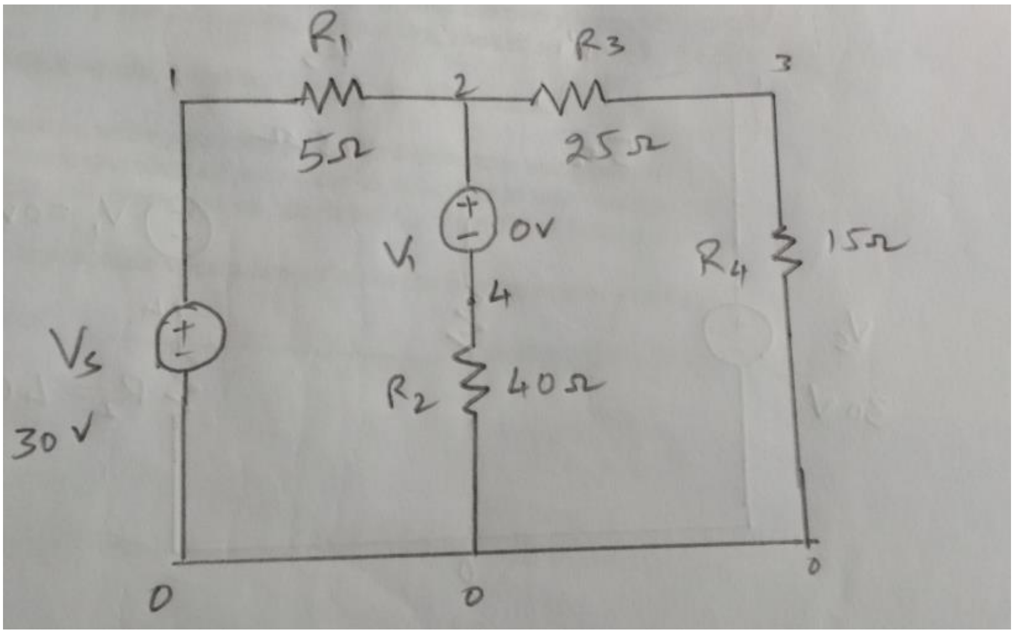
4.Develop a program for an electrical circuit

Let us consider a simple circuit as shown below to understand of writing a code in the PSpice A/D.



Problem statement: Determine the current flowing through the 40 Ω resistor.

Step 1: Assign the node numbers at all node points in the circuit. Out of them one node number should be '0' as reference.



Step 2: Write the circuit file in text simulator

CIRCUIT FILE :

** Determining the current flowing through 40 Ω resistor** -- First line is a Comment line (Left for headings like)

VS 1 0 DC 30V; Source voltage between node 0 and node 1

R1 1 2 5OHM; R1 kept between node 1 and node 2

R2 4 0 40OHM; R1 kept between node 1 and node 2

R3 2 3 25OHM; R1 kept between node 1 and node 2

R4 3 0 15OHM; R1 kept between node 1 and node 2

V1 2 4 DC 0V; DC voltage of 0V is required to know the current through the 40 Ω resistor

.END; To terminate the program

Now save the file with "name.cir"

After successful execution, it will provide the results at output file.

OUTPUT FILE :

**** 08/14/19 12:52:40 ************** PSpice Lite (Mar 2000) ****************

** Determining the current flowing through 40 Ω resistor** **** CIRCUIT DESCRIPTION

****

VS 1 0 DC 30V

R1 1 2 5OHM

R2 4 0 40OHM

R3 2 3 25OHM

R4 3 0 15OHM

V1 2 4 DC 0V

.END

**** 08/14/19 12:52:40 ************** PSpice Lite (Mar 2000) ****************

#node voltages#

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

****

NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE

(1) 30.0000 (2) 24.0000 (3) 9.0000 (4) 24.0000

VOLTAGE SOURCE CURRENTS

NAME	CURRENT
VS	-1.200E+00
V1	6.000E-01

TOTAL POWER DISSIPATION 3.60E+01 WATTS

JOB CONCLUDED

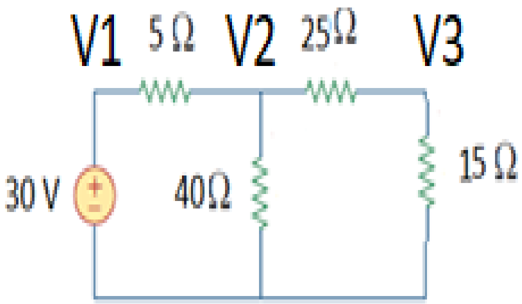
TOTAL JOB TIME .03

Here the current at V1 is the answer (i.e current flowing through the 40 Ω resistor).

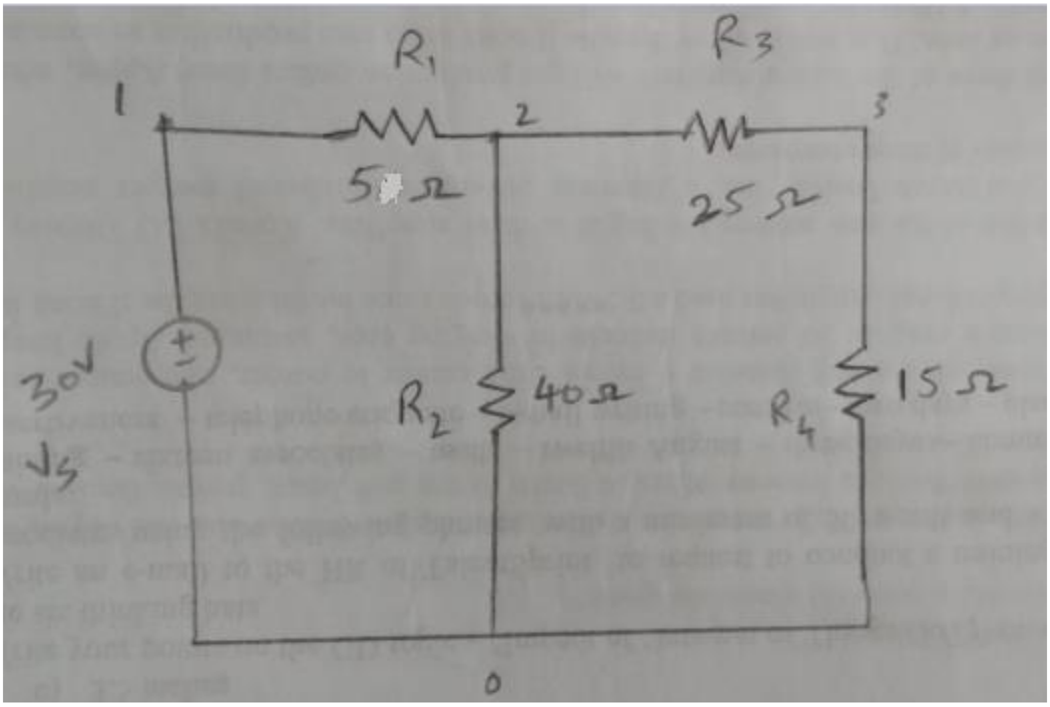
5. Analyse the response of DC circuits

Now we will see some problems on node voltage calculations and currents in any specific part of a circuit.

1. **Find V1, V2 and V3 by using pspice model ?**



Step 1: Assign nodes



Step 2: Create circuit file

CIRCUIT FILE :

```
#node voltages#
VS 1 0 DC 30V
R1 1 2 5OHM
```

R2 2 0 40OHM

R3 2 3 25OHM

R4 3 0 15OHM

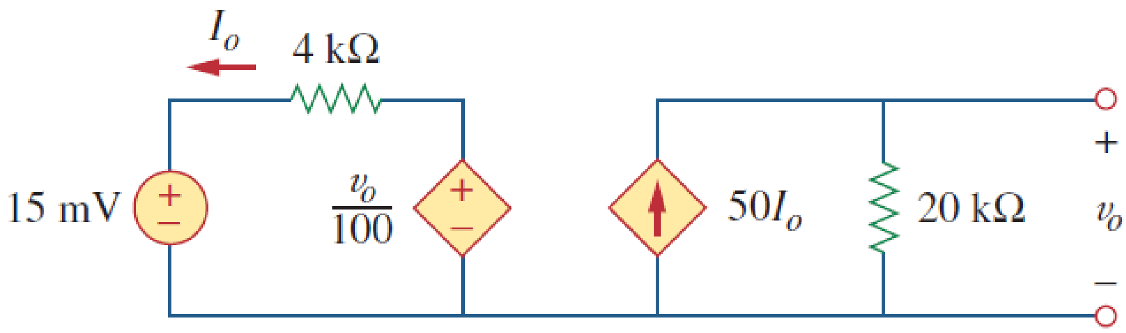
.END

Step 3: Results at output file

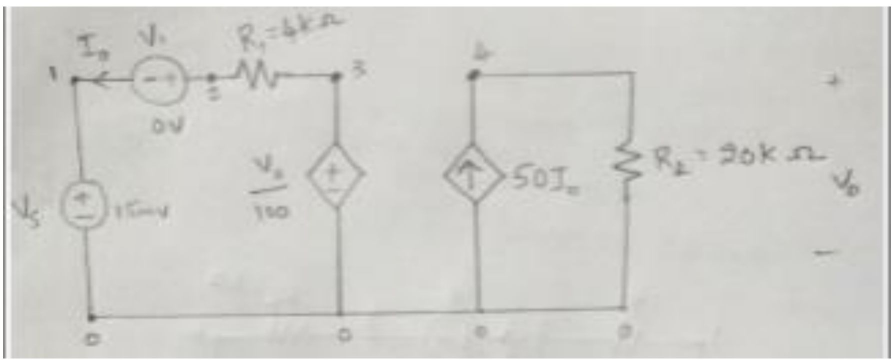
OUTPUT FILE:

```
**** 08/14/19 12:20:43 ************** PSpice Lite (Mar 2000) *****************
 #node voltages#
 ****     CIRCUIT DESCRIPTION
******************************************************************************
VS 1 0 DC 30V
R1 1 2 5OHM
R2 2 0 40OHM
R3 2 3 25OHM
R4 3 0 15OHM
.END
**** 08/14/19 12:20:43 ************** PSpice Lite (Mar 2000) *****************
#node voltages#
 ****     SMALL SIGNAL BIAS SOLUTION       TEMPERATURE =   27.000 DEG C
******************************************************************************
 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE
(1)   30.0000          (2)   24.0000          (3)    9.0000

   VOLTAGE SOURCE CURRENTS
   NAME        CURRENT
   VS        -1.200E+00
   TOTAL POWER DISSIPATION   3.60E+01  WATTS
        JOB CONCLUDED
        TOTAL JOB TIME            .02
```

2. For the circuit shown below, find Io and Vo by using pspice model ?



Step 1: Assign nodes



Step 2: Create circuit file

CIRCUIT FILE

VS 1 0 DC 3M

R1 2 3 4K

R2 4 0 20K

V1 2 1 DC 0V

E1 3 0 4 0 0.01

F1 4 0 V1 50

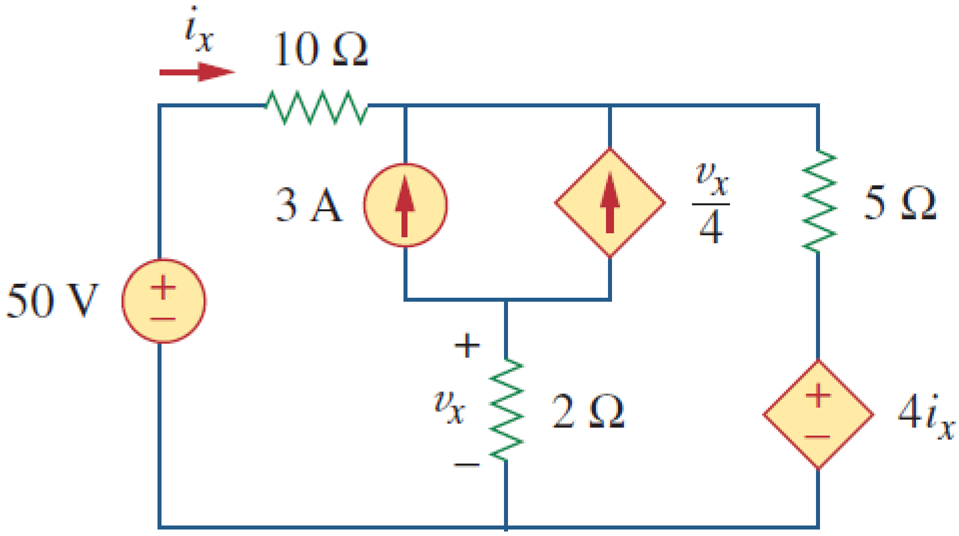
.END

Step 3: Results at output file

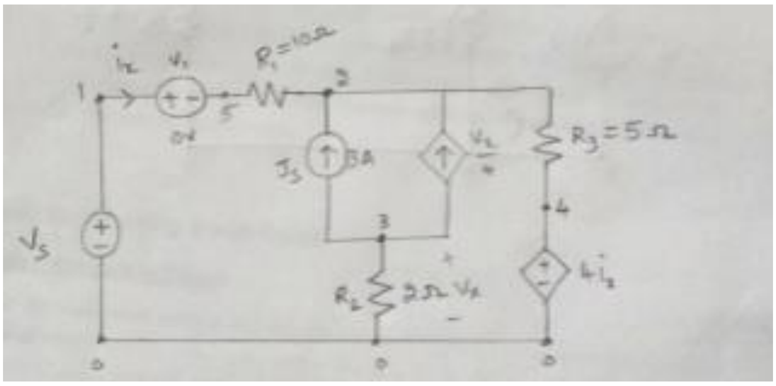
OUTPUT FILE

```
**** 08/16/19 11:20:30 ************** PSpice Lite (Mar 2000) *****************

 ****     CIRCUIT DESCRIPTION

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

VS 1 0 DC 3M

R1 2 3 4K

R2 4 0 20K

V1 2 1 DC 0V

E1 3 0 4 0 0.01

F1 4 0 V1 50

.END


**** 08/16/19 11:20:30 ************** PSpice Lite (Mar 2000) *****************

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

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

 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE

(    1)    .0030     (    2)    .0030     (    3)    .0021     (    4)    .2143


    VOLTAGE SOURCE CURRENTS

    NAME         CURRENT

    VS           -2.143E-07

    V1           -2.143E-07

    TOTAL POWER DISSIPATION   6.43E-10  WATTS

          JOB CONCLUDED

          TOTAL JOB TIME            .03
```

3. Find Ix and Vx by using pspice model.



Step 1: Assign nodes



Step 2: Create circuit file

CIRCUIT FILE

V dep I source, I dep v source

VS 1 0 DC 50V

R1 5 2 10OHM

R2 3 0 2OHM

R3 2 4 5OHM

IS 3 2 DC 3A

G1 3 2 3 0 0.25

V1 1 5 DC 0V

H1 4 0 V1 4

.OP

.DC VS 50 50 1

.PRINT DC I(R1) V(2,3)

.END

Step 3: Results at output file

OUTPUT FILE

#node voltages#

**** CIRCUIT DESCRIPTION

****

VS 1 0 DC 50V

R1 5 2 10OHM

R2 3 0 2OHM

R3 2 4 5OHM

IS 3 2 DC 3A

G1 3 2 3 0 0.25

V1 1 5 DC 0V

H1 4 0 V1 4

.OP

.DC VS 50 50 1

.PRINT DC I(R1) V(2,3)

.END

**** 08/14/19 15:45:42 ************** PSpice Lite (Mar 2000) *****************

#node voltages#

**** DC TRANSFER CURVES TEMPERATURE = 27.000 DEG C

****

VS I(R1) V(2,3)

5.000E+01 2.105E+00 3.295E+01

**** 08/14/19 15:45:42 ************** PSpice Lite (Mar 2000) *****************

#node voltages#

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

****

NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE

(1)   50.0000 (2)   28.9470 (3)   -4.0000 (4)   8.4211

(5)   50.0000

VOLTAGE SOURCE CURRENTS

NAME   CURRENT

VS   -2.105E+00

V1   2.105E+00

TOTAL POWER DISSIPATION   1.05E+02 WATTS

**** 08/14/19 15:45:42 ************** PSpice Lite (Mar 2000) *****************

#node voltages#

****   OPERATING POINT INFORMATION   TEMPERATURE =   27.000 DEG C

****

**** VOLTAGE-CONTROLLED CURRENT SOURCES

NAME   G1

I-SOURCE   -1.000E+00

**** CURRENT-CONTROLLED VOLTAGE SOURCES

NAME   H1

V-SOURCE   8.421E+00

I-SOURCE   4.105E+00

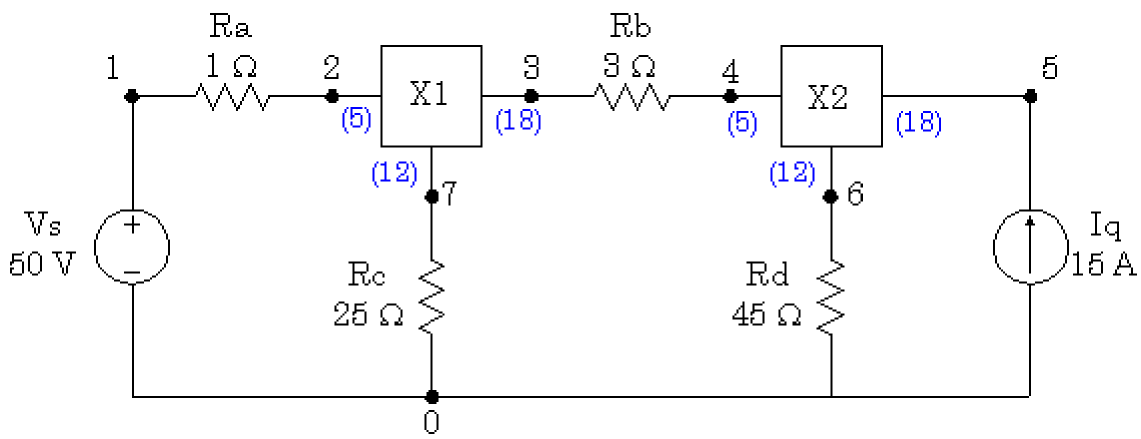
JOB CONCLUDED

TOTAL JOB TIME   0.00

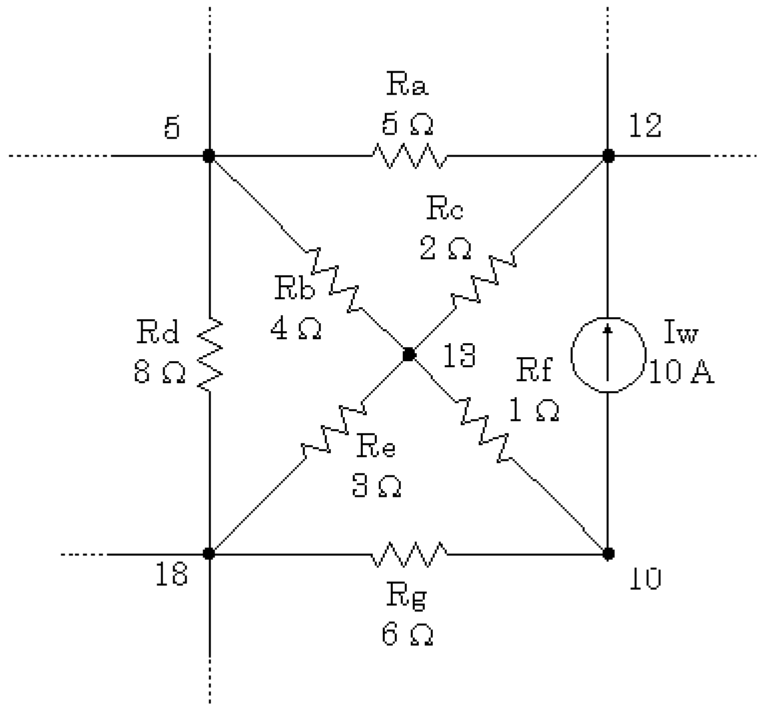
Sub-circuit model:

Here let us know how to create a sub-circuit, will helps in simplification of complex circuit models.

Find node voltages for below given circuit using pspice model ?



SUBCIRCUIT :



CIRCUIT FILE

```
Subcircuit
*       name        nodelist
.SUBCKT Example_1   5   12   18
Iw   10   12   DC   10A
Ra    5   12   2.0
Rb    5   13   5.0
Rc   12   13   2.0
Rd    5   18   8.0
Re   13   18   3.0
Rf   10   13   1.0
Rg   10   18   6.0
.ENDS; It indicates sub-circuit termination
Vs    1    0   DC   50V
Ra    1    2   1.0  ; different from Ra above
Rb    3    4   3.0  ; different from Rb above
Rc    7    0  25.0  ; different from Rc above
Rd    6    0  45.0  ; different from Rd above
*     nodelist      name
X1    2    7    3   Example_1
X2    4    6    5   Example_1
.END
```

OUTPUT FILE:

```
**** 08/19/19 00:47:37 *************** PSpice Lite (Mar 2000) *****************

 Subcircuit

 ****     CIRCUIT DESCRIPTION

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

*       name        nodelist

.SUBCKT Example_1   5   12   18

Iw   10   12   DC   10A

Ra    5   12   2.0

Rb    5   13   5.0

Rc   12   13   2.0

Rd    5   18   8.0

Re   13   18   3.0

Rf   10   13   1.0

Rg   10   18   6.0

.ENDS

Vs    1    0   DC   50V
```

```
Ra    1    2   1.0  ; different from Ra above
Rb    3    4   3.0  ; different from Rb above
Rc    7    0  25.0  ; different from Rc above
Rd    6    0  45.0  ; different from Rd above
*     nodelist      name
X1    2    7    3   Example_1
X2    4    6    5   Example_1
.END
**** 08/19/19 00:47:37 ************** PSpice Lite (Mar 2000) *****************
 Subcircuit Example No. 1
 ****     SMALL SIGNAL BIAS SOLUTION       TEMPERATURE =   27.000 DEG C
******************************************************************************
 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE

(    1)   50.0000        ( 2)   47.2110           (    3)   34.3490          (    4)   31.9190

(    5)   22.1730        (    6)   36.4570          (    7)   49.4640          (X1.10)   26.9140

(X1.13)   35.6740        (X2.10)   13.9800        (X2.13)   22.6150

    VOLTAGE SOURCE CURRENTS
    NAME         CURRENT
    Vs          -2.789E+00
    TOTAL POWER DISSIPATION   1.39E+02  WATTS
          JOB CONCLUDED
        TOTAL JOB TIME            .02
```

6.Analyse the response of AC circuits

Representation of ac sources and ac analysis:

AC Voltage and Current Sources:

Syntax:

```
Name  nodelist   type  value          phase(deg)
Vac    4  1      AC    120V            30    ;phase angle 30 degrees
Vba    2  5      AC    240                   ; phase angle 0 degrees
Ix     3  6      AC    10.0A          -45    ; phase angle -45 degrees
Isv   12  9      AC    25mA                  ; 25 milliamps @ 0 degrees
```

Use of the .PRINT AC Command:

Syntax:

```
Type#          points   start   stop
.AC    LIN       1      60Hz    60Hz;  <== what we want now.
.AC    LIN       11     100     200;   <== a linear range sweep
```

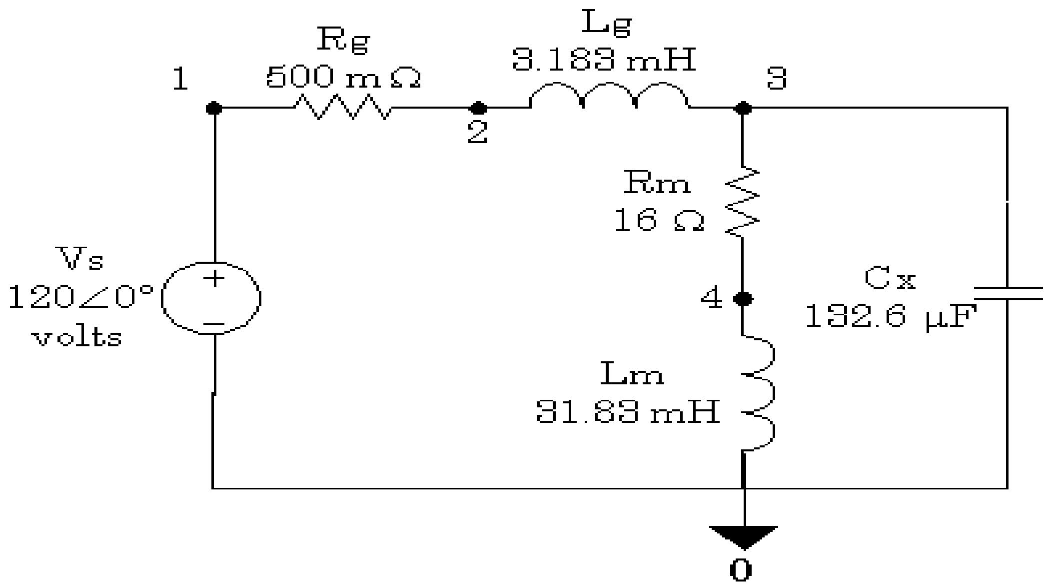
```
.PRINT   AC   VM(30,9)      VP(30,9)    ; magnitude & angle of voltage
.PRINT   AC   IR(Rx)        II(Rx)      ; real & imag. parts Rx current
.PRINT   AC    VM(17)     VP(17)    VR(17)    VI(17)  ; the whole works on
node 17
```

Example:

```
.AC   LIN      101  2k    4k;         101 points from 2 kHz to 4 kHz
.AC   LIN       11  800   1000;       11 pts from 800 Hz to 1 kHz
```

Example Circuit

We will analyze the following circuit at a frequency of 60 Hz.



CIRCUIT FILE:-

```
60 Hz AC Circuit
Vs  1  0  AC  120V  0
Rg  1  2  0.5
Lg  2  3  3.183mH
Rm  3  4  16.0
Lm  4  0  31.83mH
Cx  3  0  132.8uF
.AC LIN 1 60 60
.PRINT AC VM(3) VP(3) IM(Rm) IP(Rm) IM(Cx) IP(Cx)
.END
```

Points to remember in AC Phasor Circuit Analysis Using Pspice

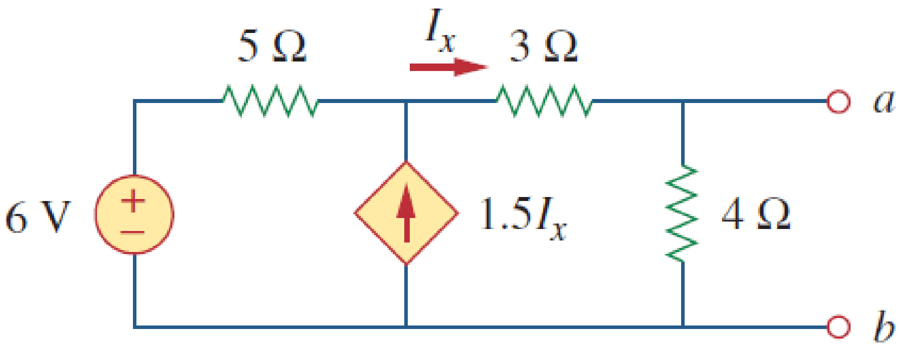
- Use AC as the type for all independent sources.
- Specify phase angle of sources if other than zero degrees.
- There must be a ".AC" command to specify the frequency to be used for all sources.
- Use a .PRINT AC command to specify which voltages and currents are to be listed in the output file.

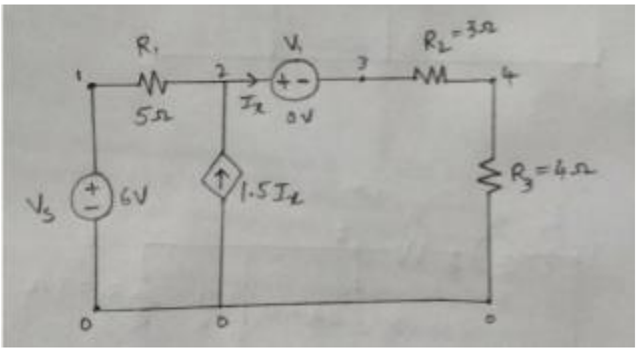
- M indicates magnitude, P indicates phase angle, R indicates real part and I indicates imaginary part, when these letters follow V (for voltage) or I (for current).

7.Perform network theorems

Problem on Thevenin's Theorem:

Find the Thevenin's equivalent circuit (Vth & Rth) for the below circuit to the left of the terminals by using pspice model ?





CIRCUIT FILE

Thevenin example

Vs 1 0 DC 6

R1 1 2 5

R2 3 4 3

R3 4 0 4

V1 2 3 DC 0

F1 0 2 V1 1.5

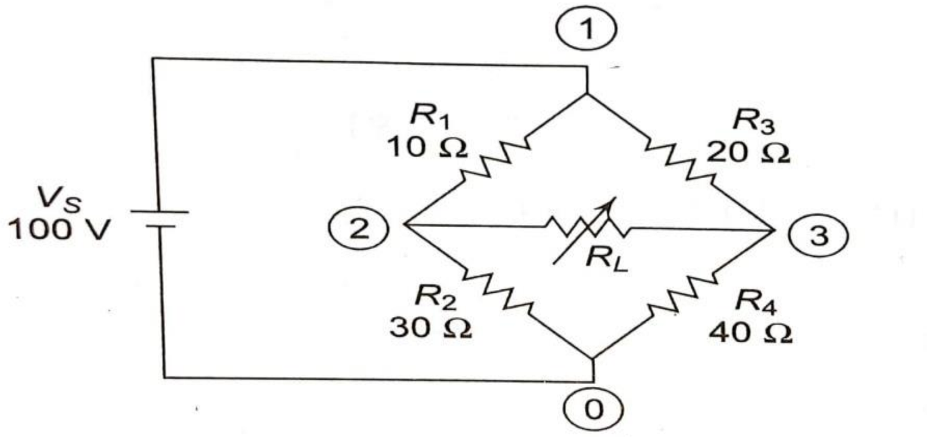
.TF V(4,0) Vs

OUTPUT FILE

```
**** 08/16/19 15:56:12 ************** PSpice Lite (Mar 2000) *****************

 Thevenin example

 ****     CIRCUIT DESCRIPTION

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

Vs 1 0 DC 6

R1 1 2 5

R2 3 4 3

R3 4 0 4

V1 2 3 DC 0

F1 0 2 V1 1.5

.TF V(4,0) Vs

**** 08/16/19 15:56:12 ************** PSpice Lite (Mar 2000) *****************

 thevinin example

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

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

 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE

   (    1)    6.0000        (    2)    9.3333         (    3)    9.3333        (    4)    5.3333


   VOLTAGE SOURCE CURRENTS

   NAME        CURRENT

   Vs          6.667E-01

   V1          1.333E+00


   TOTAL POWER DISSIPATION  -4.00E+00  WATTS

 ****     SMALL-SIGNAL CHARACTERISTICS

     V(4,0)/Vs =  8.889E-01

     INPUT RESISTANCE AT Vs = -9.000E+00

     OUTPUT RESISTANCE AT V(4,0) =  4.444E-01

        JOB CONCLUDED

        TOTAL JOB TIME            .05
```

Problem on Maximum Power Theorem:

Determine the load resistance to receive maximum power from the source; also find the maximum power delivered to the load in the circuit shown in below using PSpice.



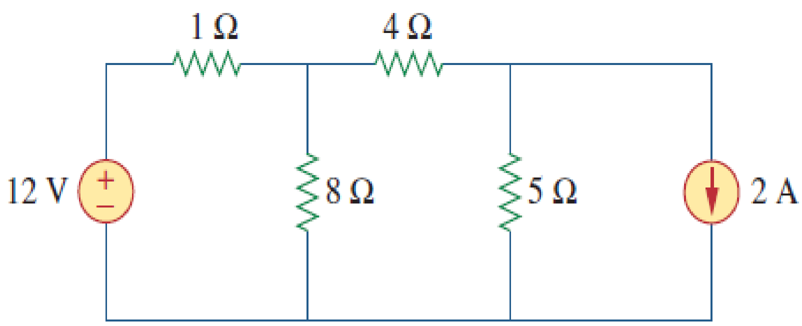
CIRCUIT FILE:

```
VS 1 0 DC 100
 R1 1 2 10
 R2 2 0 30
 R3 1 3 20
 R4 3 0 40
 RL 2 3 RLOAD 1
 .MODEL RLOAD RES(R=25)
 .DC RES RLOAD(R) .001 40 .001
 .TF V(2,3) VS
 .PROBE
 .END
```

8. Exercise Problems

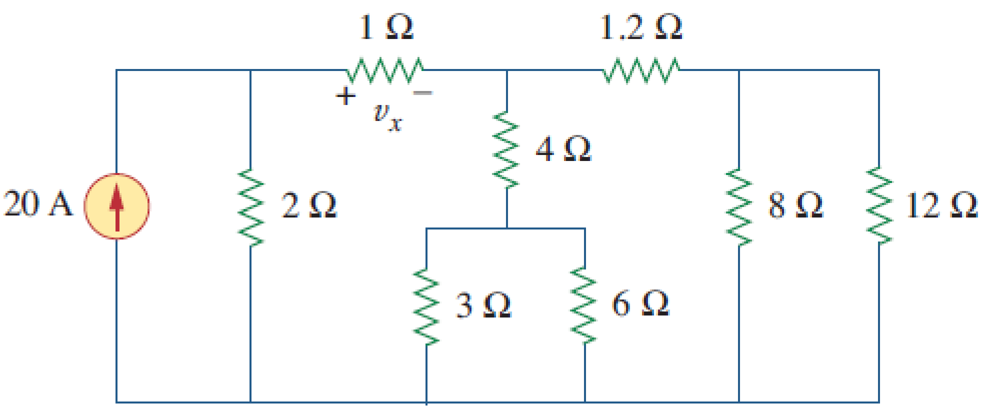
Problems on Node voltages and Specified branch currents:

1. **Determine node voltages for the given circuit using pspice model ?**



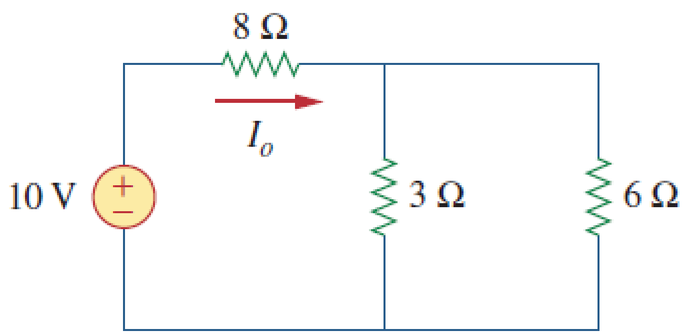
ANS:-V1=12V, V2=8.8089V, V3=0.449V.

2. **Determine Vx and power absorbed by 12 Ω Resistor using pspice model ?**



ANS:- 2V , 1.92 W.

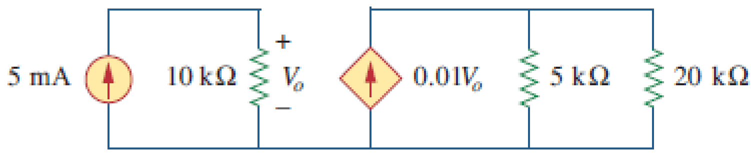
3. **Find node voltages and Io by using pspice model ?**



ANS:- V(1,0)=10V, V(2,0)=2V, and Io =1A.

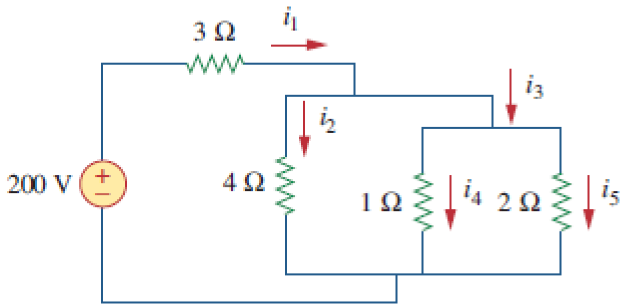
4. **Find node voltages for the given circuit and current in 5kΩ by using pspice**

model ?



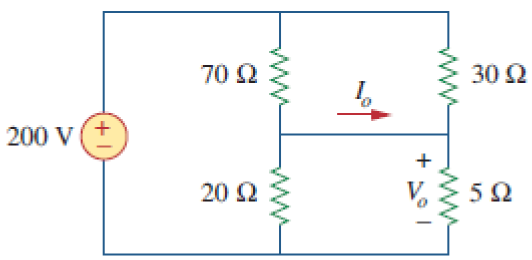
ANS:- Vo = 50V, V1 = 2000V, and I(5kΩ) = 0.4 A

5. **Determine i1 to i5 by using pspice model ?**



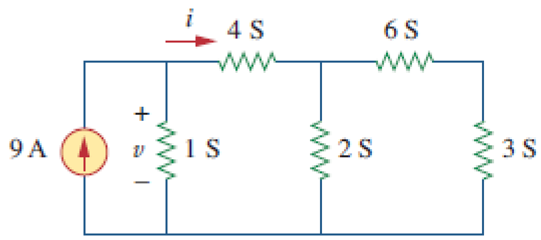
ANS:- i1=11.2A, i2=1.6A, i3=9.6A, i4=6.4A, i5=3.2A.

6. **Find Vo and Io by using pspice model ?**



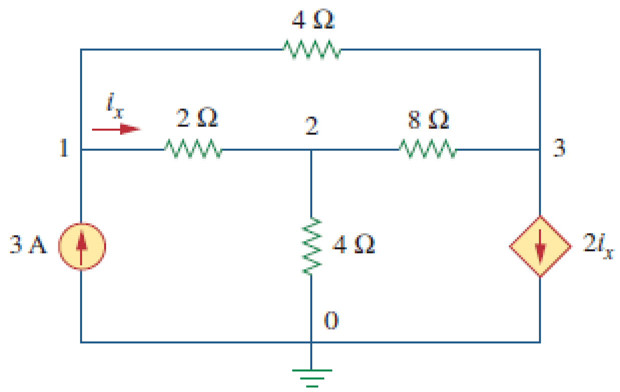
ANS:- Vo=32V, Io=0.8A.

7. **Find v and i by using pspice model ?**



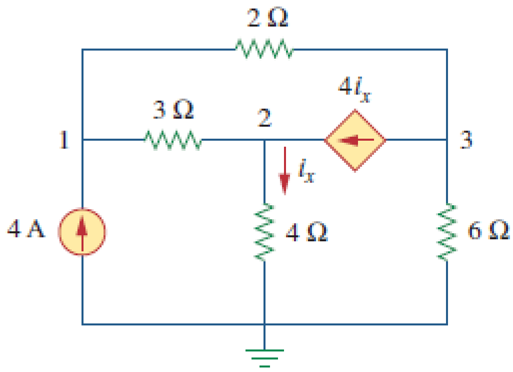
ANS:-v=3V, i=6A.

8. **Find the node voltages by using pspice model ?**



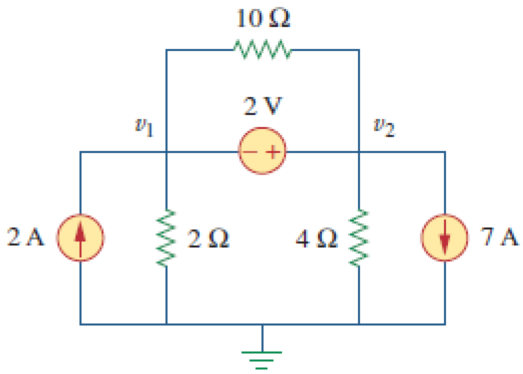
ANS:- V1=4.8V, V2=2.4V, V3=-2.4V.

9. **Find the node voltages by using pspice model ?**



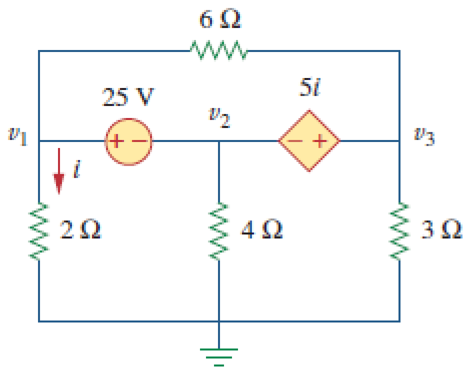
ANS:- V1=32V, V2=-25.6V, V3=62.4V.

10. **Find the node voltages by using pspice model ?**



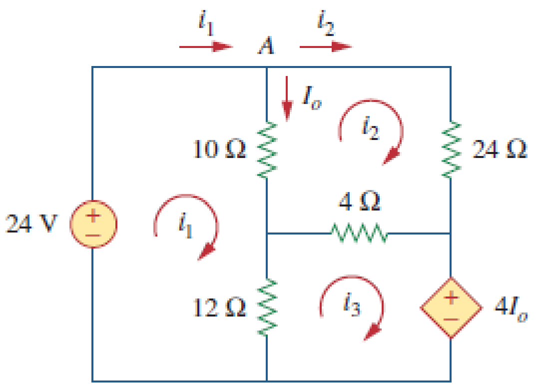
ANS:- V1=-7.33V, V2=-5.33V.

11. Find the node voltages by using pspice model ?



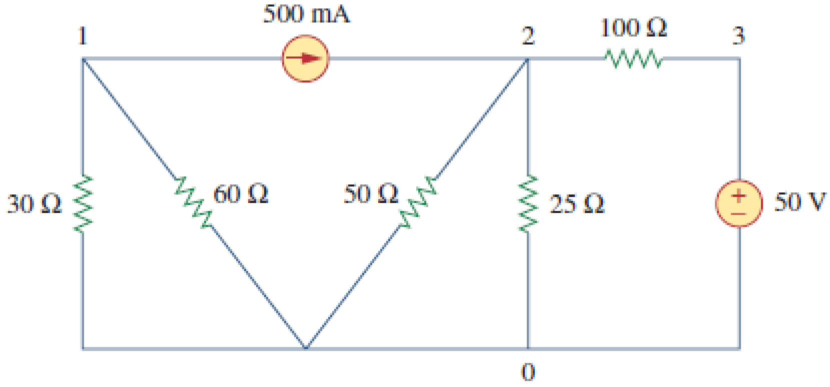
ANS:- V1=7.608V, V2=-17.39V, V3=1.6305V.

12.Calculate Io by using pspice model ?



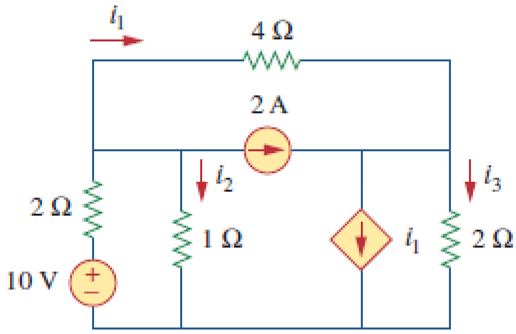
ANS:- Io=1.5A.

13. Find the node voltages by using pspice model ?



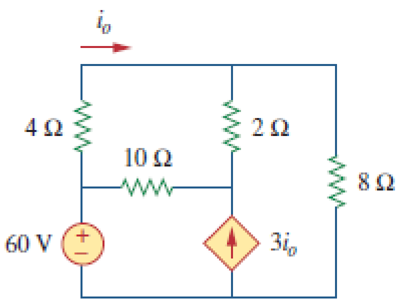
ANS:-V1=-10V, V2=14.286V, V3=50V.

14.Find currents i1, i2 and i3 by using pspice model ?



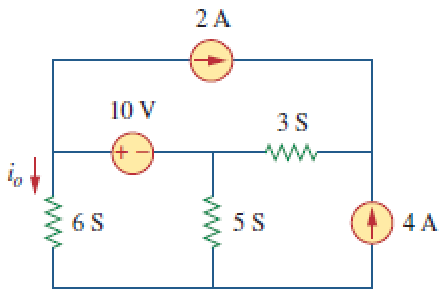
ANS:- i1=-0.4286A, i2=2.286A, i3=2A.

15.Find io by using pspice model ?



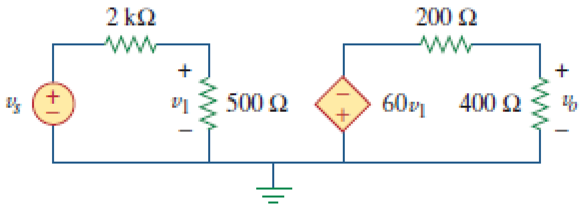
ANS:- io=1.73A.

16.Find io by using pspice model ?



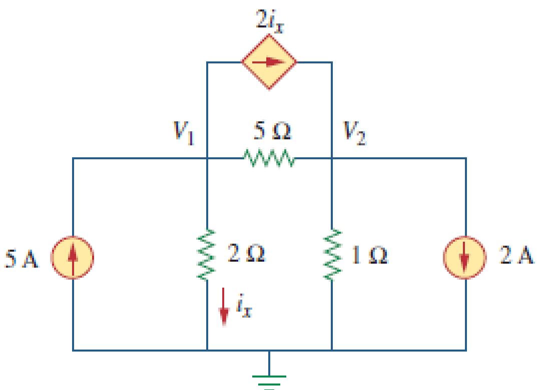
ANS:- io=29.45A.

17.Assume Vs = 100V, calculate V1 and Vo by using pspice model ?



ANS:- V1=20V, Vo=800V.

18.Find V1 and V2 by using pspice model ?

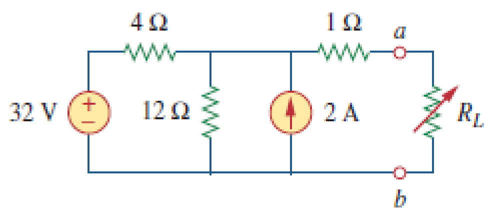


ANS:- V1=3.111V, V2=1.4444V.

Problems on Thevenin's Theorem

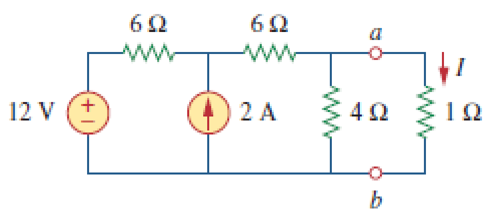
Q.FIND THEVENINS EQUIVALENT ACROSS a and b TERMINALS FOR FOLLOWING CIRCUITS USING PSPICE MODEL ?

1.



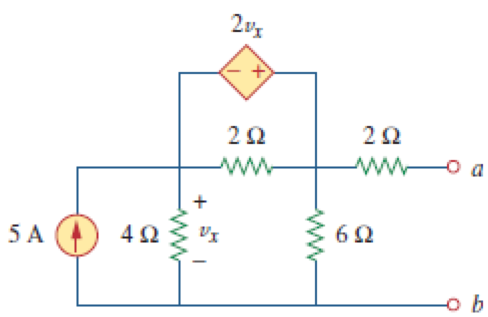
ANS:- Vth=30V, Rth=4Ω.

2.



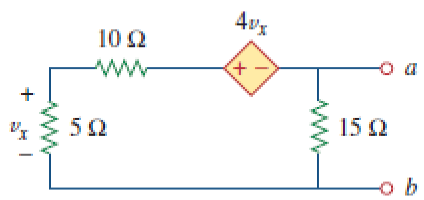
ANS:-Vth=6V, Rth=3Ω.

3.



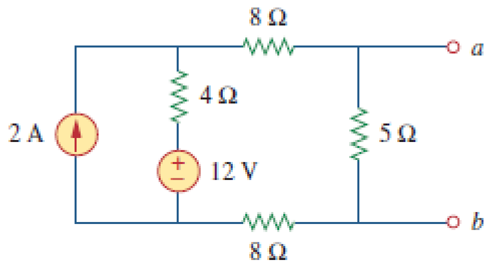
ANS:-Vth=20V, Rth=6Ω.

4.



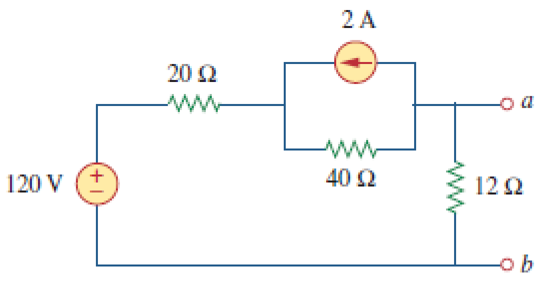
ANS:-Vth=0V, Rth=-7.5Ω.

5.



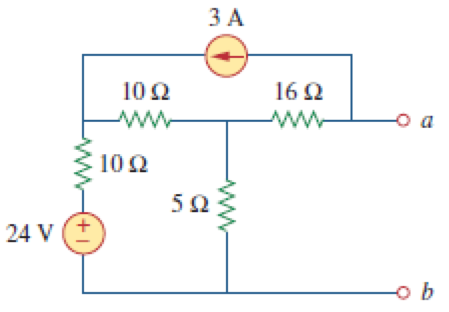
ANS:- Vth=4V, Rth=4Ω.

6.



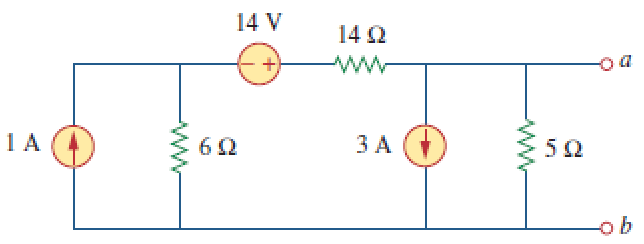
ANS:-Vth=6.667V, Rth=10Ω.

7.



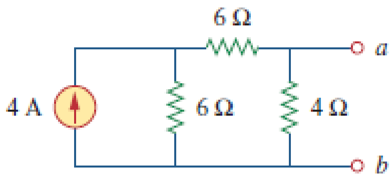
ANS:-Vth=-49.2V, Rth=20Ω.

8.



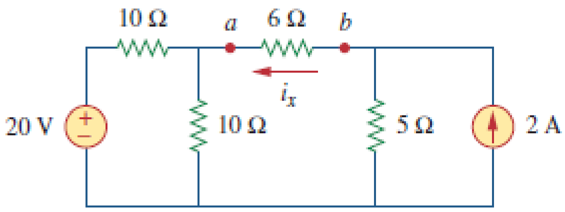
ANS:- Vth=-8V, Rth=4Ω.

9.



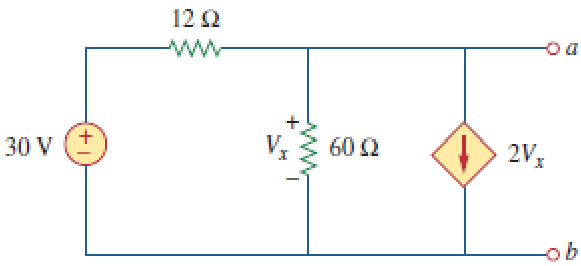
ANS:-Vth=6V, Rth=3Ω.

10.



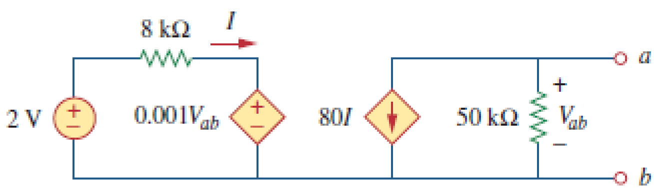
ANS:-Vth=0V, Rth=10Ω.

11.



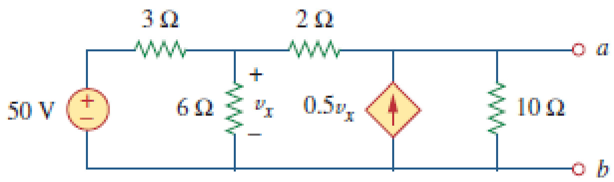
ANS:-Vth=1.19V, Rth=476.2mΩ.

12.



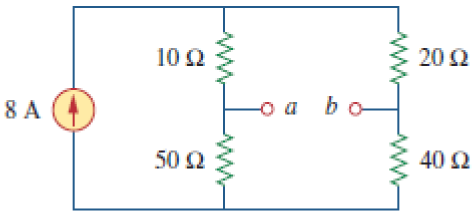
ANS:- Vth=-2000V, Rth=100kΩ.

13.



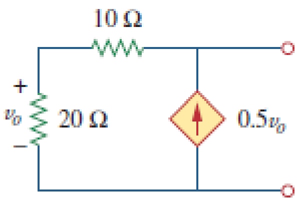
ANS:-Vth=166.67V, Rth=10Ω.

14.



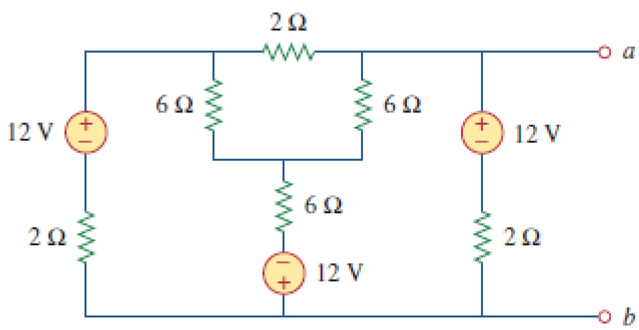
ANS:-Vth=40V, Rth=22.5Ω.

15.



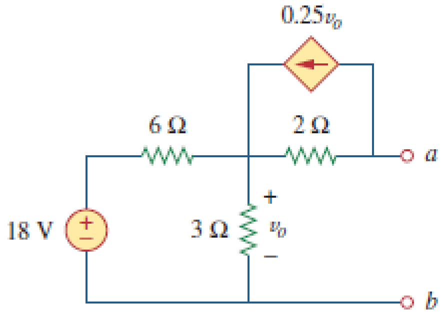
ANS:-Vth=0V, Rth=-3.333Ω.

16.



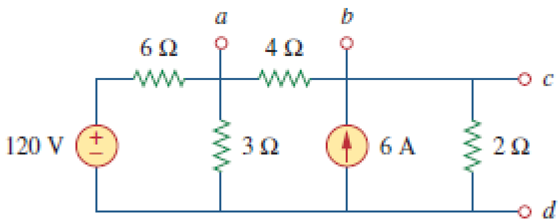
ANS:-Vth=9.6V, Rth=1.2Ω.

17.



ANS:-Vth=3V, Rth=3Ω.

18.



ANS:-Vth=14V, Rth=2Ω.

9.References

1. Spice for Circuits and Electronics using PSPICE by M H Rashid

2. Fundamentals of Electric Circuits by C K Alexander and M N O Sadiku

3. Network Analysis by Sudhakar and Shyammohan S Palli

For more details contact

Mr. P. Rajasekhara Reddy

Assistant Professor

Department of Electrical and Electronics Engineering

Vasavi College of Engineering (Autonomous)

Email: rajashekhar.p@staff.vce.ac.in

Mobile: +91-9177207976

&

Dr. M. Chakravarthy

Professor and Head of the Department

Department of Electrical and Electronics Engineering

Vasavi College of Engineering (Autonomous)

Email: hodeee@staff.vce.ac.in

Mobile: +91-9849979136