

Bangladesh University of Engineering and Technology
Electrical Circuits II Laboratory (EEE106)
Simulation Experiment no 4

Sub-Circuits using Net listing and Schematics and Three Phase Circuits

Objective:

The objective of this experiment is to learn how to write and invoke a sub-circuit and implement the concept in three phase circuits.

Simple Subcircuits in PSpice

One of the more useful concepts in PSpice is the use of *subcircuits* to group elements into clusters in order to replicate the clusters without having to re-enter all the elements each time. This is very useful for several reasons. First is the labor savings of replacing many lines of circuit data with a single subcircuit call. This also reduces the chance of making a typo. Second, the use of a subcircuit usually improves clarity by removing confusing clutter. The user can suppress printing unwanted details internal to a subcircuit, thus making the output easier to understand. If desired, the user can place often-used subcircuits into an *include* file so that the main source file for the problem is kept simple. Then the definition of the subcircuit is out of sight entirely.

Coding a Subcircuit

Each subcircuit used in a study must have a unique name. This is true of any other circuit element. Also, there must be a list of at least two nodes that can be connected to elements external to the subcircuit. A subcircuit can have many external node connections, if needed. Later, we will find that parameters can be passed to a subcircuit in order to allow unique behavior and responses from an instance of a subcircuit.

The initial line of a subcircuit section must begin with ".SUBCKT," followed by the name and then the external node list. After that, optional features (not to be discussed yet) can be added. The best method of understanding the use of a subcircuit is by example. Below, we find a cluster of components that can be combined into a subcircuit.

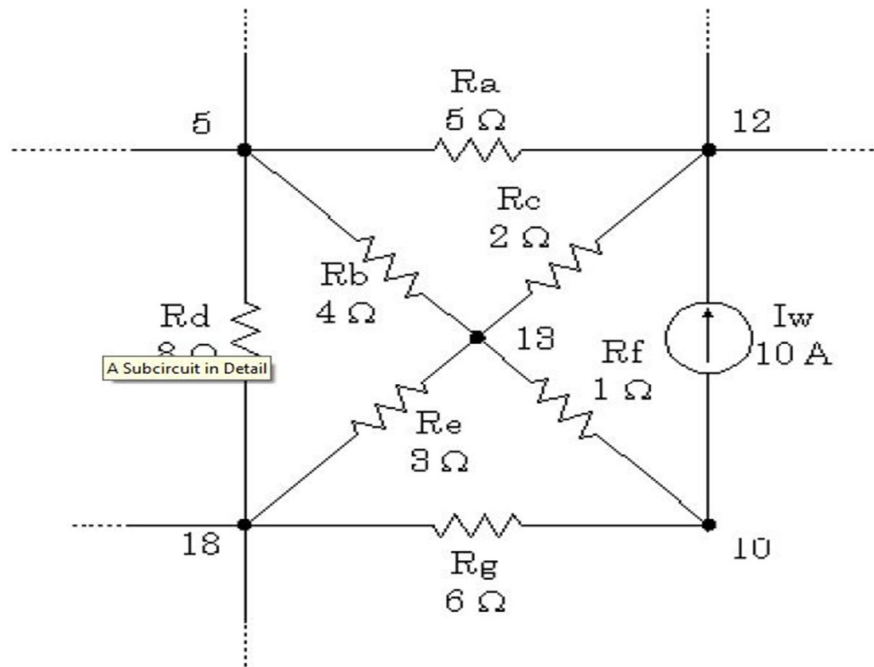


Fig. 1 : Subcircuit of Example 1

Note that nodes 5, 12 and 18 have external connections. Therefore, they must be included in the node list in the subcircuit definition. Nodes 10 and 13 do not have external connections and need not be (indeed *should* not be) included in this node list. They are internal nodes and will be used to help define the subcircuit. Now, we can code the above subcircuit as follows. Note that the code could be embedded into the rest of the code for the main circuit or could be placed in a separate *include* file.

```

*          name          nodelist
.SUBCKT Example_1      5    12    18
Iw      10    12    DC    10A
Ra       5     12     5.0
Rb       5     13     4.0
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

```

Note that the subcircuit section must be terminated with a ".ENDS" command.

Invoking a Subcircuit

All subcircuit calls are made by declaring a part with a unique name beginning with "X," followed by the node list and then the subcircuit name. The node list in the calling statement must have the same number of nodes as the node list in the subcircuit definition. To demonstrate the use of the calling statement, we present the following main circuit which contains two instances of the above subcircuit. X1 and X2 are the two instances of the subcircuit "Example_1." For added clarity, the subcircuit's defined external nodes are shown in parentheses. Note that these nodes are mapped into the main circuit by *different numbers*.

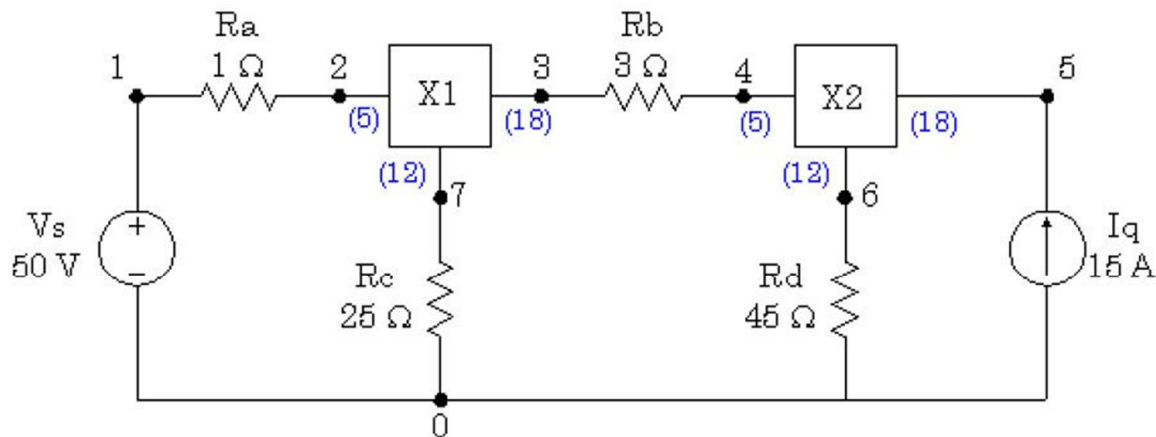


Fig. 2: Circuit using the subcircuit Example_1

```

Subcircuit Example No. 1
*
      name      nodelist
.SUBCKT Example_1  5  12  18
Iw      10      12      DC      10A
Ra       5       12      5.0
Rb       5       13      4.0
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

Iq      0      5      DC  15A
*       nodelist  name
X1      2      7      3  Example_1
X2      4      6      5  Example_1
.END

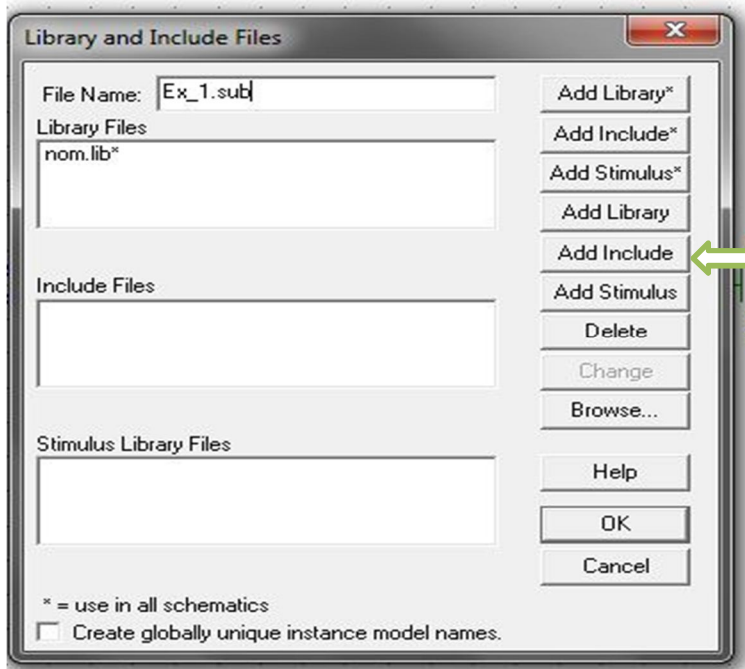
```

Subcircuits using Schematics:

DCLICKL=Double CLICK Left, CLICKL=CLICK Left

Step 1: Create the MODEL

1. Draw RLC series circuit with $R=100\Omega$, $L=0.31\text{H}$ and $C=31.83\mu\text{F}$.
2. Replace the input and output nodes with the *interface port symbols* IF_IN and IF_OUT (Search IF_IN and IF_OUT from the part list). To label each port symbol, **DCLICKL** on each symbol and fill in with any name you want.
3. Save the schematic to file *Ex_1.sch*.
4. Go to **Tools, Create Subcircuit** to generate the subcircuit MODEL definition. This model definition is automatically stored as *Ex_1.sub* in the same directory as *Ex_1.sch*.
5. To make the MODEL definition (*Ex_1.sub*) available locally (valid only to *Ex_1.sch*): Go to **Analysis, -> Library and Include Files**, enter *Ex_1.sub* in the File Name Field, **Add Library** (no asterisk), **OK**. (Add Library* would make the model global and available to all schematics).



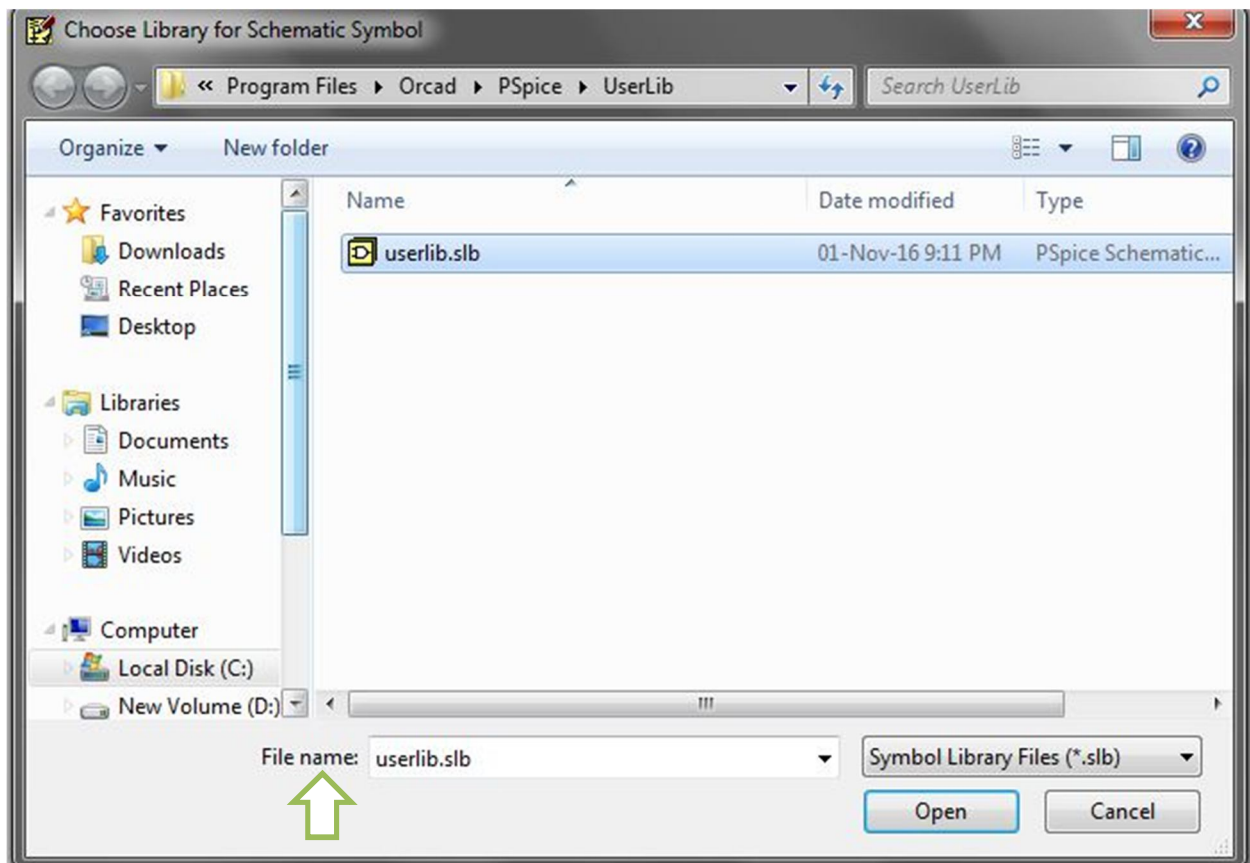
Step 2: Create the PART

6. To create a PART, we first create a symbol. Assign the symbol a PART name (such as Ex_1), and store the PART in a symbol library (such as *userlib.slb*). The procedure to do this is as follows:

File, Symbolize, type PART name *Ex_1* in *Save As* dialog box, **OK**



After clicking OK the *Choose Library for Schematic Symbol* dialog box will appear.



To save part *Ex_1* in library *userlib.slb* in a specific directory **CLICKL** to select *userlib*, Then click **OPEN**.

NOTE: If no .slb file appears you can write any name in the File name box i.e “uselib.slb”/ “anyname.slb”. Then click OPEN. A .slb file will be created automatically

7. The next step is to edit the new PART symbol by bringing up the Symbol Editor: Click **File**, Then **Edit Library**. A new Window will open. Then in the new window click **File, Open, DCLICKL** on *userlib.slb*,

Part, Get, DCLICKL on *Ex_1* to bring up the initial symbol.

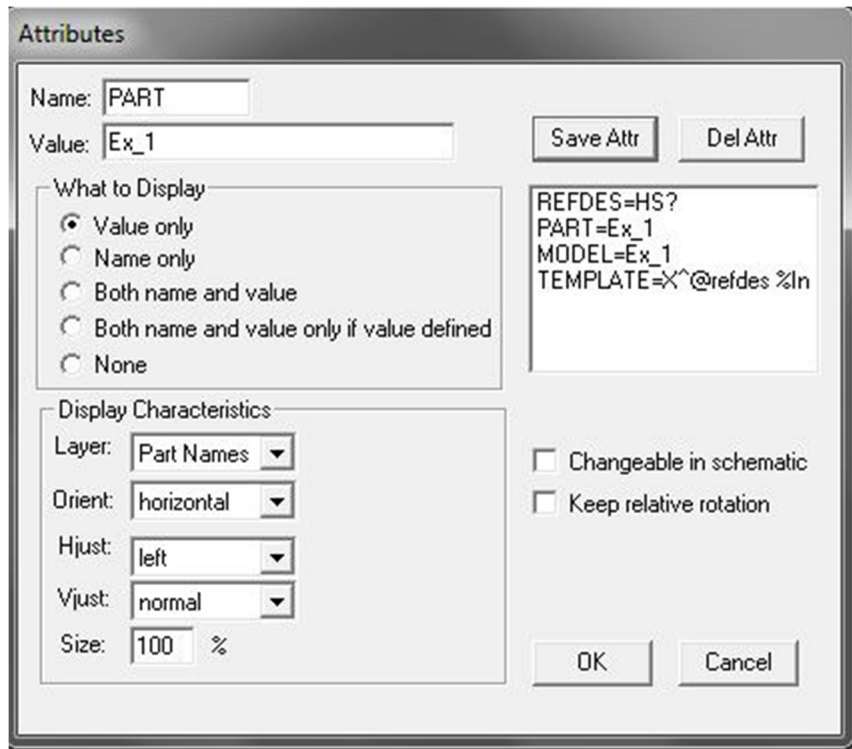


8. Modify the initial symbol by moving the pins and attributes by **select, DRAG**. Also, **DCLICKL** on any attribute or pin to place pin numbers and reorient pin names. You may wish to remain with the initial symbol as well.

9. To save the edited PART symbol again in the symbol library (*userlib.slb*): **File, Save** (and if necessary, **Yes** add to list of configured libraries).

Step 3: Associate the MODEL with the PART

10. To associate the MODEL definition (*Ex_1.sub*) with the PART symbol (in *userlib.slb*): **Part, Attributes, CLICKL** on PART=, enter *Ex_1* in Value field, **Save Attr, CLICKL** on MODEL=, enter *Ex_1* in Value field, **Save Attr, OK**. (**DRAG** part name *Ex_1* to desired position.)



11. To return to Schematics, **File, Close**, (and if necessary, **Yes** save changes to Part, **Yes** save changes to library).

12. **File, Close** to exit the *Ex_1.sch* window (and if necessary, **Yes** save all changes).

Circuit design using the subcircuit

13. Create a new schematic screen.

14. Now If you search in the Part_list you will find “Ex_1” as a part .

Exercise 1:

Use the subcircuit in three different phases of a balanced three phase supply with $f=50\text{Hz}$ and amplitude= 100V . Connect the sources and the loads in

- a) YY,
- b) $Y\Delta$,
- c) ΔY
- d) $\Delta\Delta$

Use Line impedances of $1 + j20 \Omega$. Use another subcircuit for the line impedance .

For each case Plot

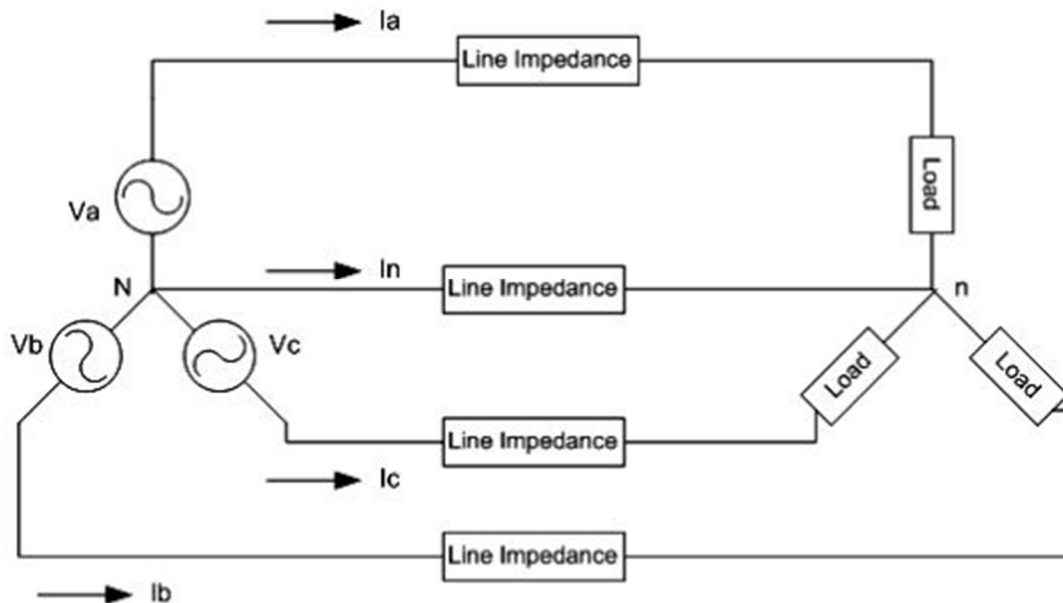
- a) Line voltage and phase voltage in the same window. What is the relationship between the two voltages?
- b) Line current and phase current in another window .What is the relationship between the two currents?
- c) Plot the voltage and current of the load in the same plot. Find the power factor of the load from the zero crossings of voltage and current waveform. (Use proper scaling if necessary)

Note: While working with delta-connected sources, you have follow the instructions otherwise Pspice will show errors.

1. insert a resistor of negligible resistance (say, $1\ \mu\Omega$ per phase) into each phase of the delta-connected source.
2. insert balanced wye-connected large resistors (say, $1\text{M}\Omega$ per phase) in the delta-connected source so that the neutral node of the wye-connected resistors serves as the ground node 0.

Post Lab:

1. Unbalanced Load



assume that the three-phase circuits are unbalanced and operating in the positive sequence with $V_a = 170 \angle 0^\circ$ V. Use the line impedance $(1 + j10)$ impedance, but the load is now $(20 + j20)\ \Omega$ for phase A, $(50 + j10)\ \Omega$ for phase B, and $(5 + j50)$ for phase c. Find

- a) the line currents I_a I_b I_c and the neutral current (I_n) in peak values
- b) the power loss in each line, including the neutral
- c) the power factor for each phase of the load.
- d) Complex, Real and reactive power absorbed by load

Does the answer match with your calculation?

2. Power Factor correction

Now suppose that the loads from the previous postlab task are balanced ,with impedance of $20+j20$ Find

- the power factor for each phase of the load.
- From the circuit provided below find the kVAR rating of the three capacitors delta connected in parallel with the load that will raise the power factor to 0.9 lagging.

3. Balanced and Unbalanced load together

When you are trying to find the magnitude and phase of any signal you can also use IPRINT and VPRINT parts from the part list. Using these parts from the part list complete the following task.

A 3-phase line has an impedance of $3 + j4 \Omega$. The line feeds two balanced three-phase loads that are connected in parallel. The first load is Y-connected and has an impedance of $30 + j40 \Omega/\text{phase}$. The second load is delta connected and has an impedance of $60 - j45 \Omega/\text{phase}$. The line is energized at the sending-end from a 3-phase balanced supply of

line to neutral voltage $V_{LN} = 200 \angle 0^\circ \text{ V}$ (rms), 60 Hz. Determine

- Current in the line for each phase.
- Current in each phase of the Y-connected loads.
- Current in each phase of the delta connected loads.