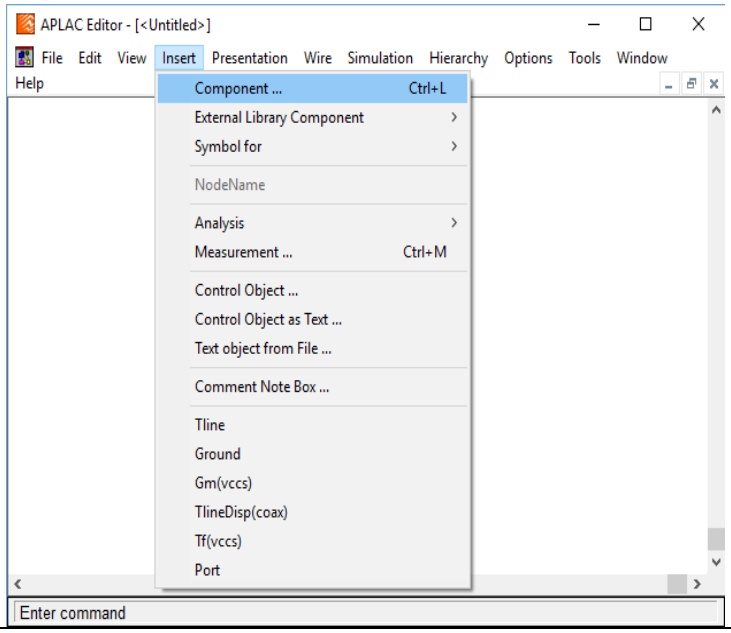
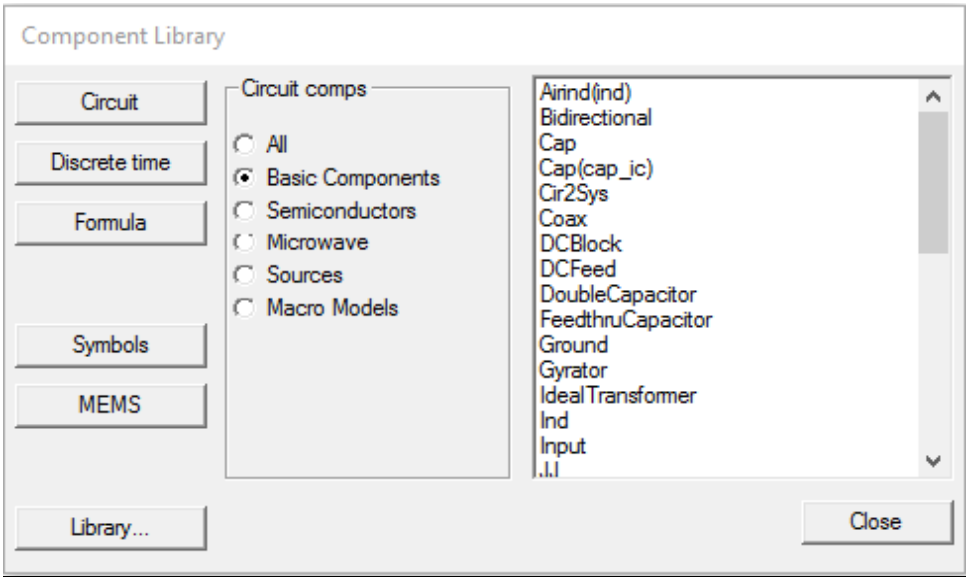
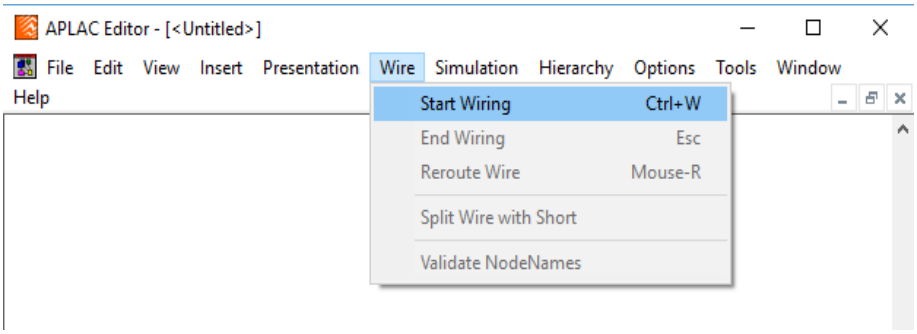
Experiment No:1	**Design Single stub matching network using APLAC software**
Aim:	To study the single stub matching and simulate it using APLAC software
Theory:	Transmission lines are used for the transmission of power and information. For radio frequency power transmission, it is highly desirable that maximum power is transmitted from the generator to the load and less power is lost on the line itself. This will require that the load be matched to the characteristic impedance of the line so that the standing-wave ratio on the line is as close to unity as possible. If it is not so then power is reflected from the source and due to this reflection, it may be possible that the source get damaged. Matching the source and load to the transmission line or waveguide in a general microwave network, we have used two types of impedance matching, single-stub and double-stub. In this experiment, we are going to study single-stub for impedance matching.
Problem Statement	Design single-stub (short circuit) shunt tuning networks to match a load impedance $Z_L = 60 - j80\ \Omega$, to a 50 Ω line. Assuming that the load is matched at 2 GHz.
	1. Design the circuit as per the specifications given. Open APLAC software. 2. Design the circuit as per the specifications given. Open APLAC software. 3. From File menu select New Circuit (Ctrl+N). 4. How to draw the circuit into the drawing space provided: Insert > Components. Now Select Circuit > Basic Components > Select the component of your choice and drop it in drawing space. 

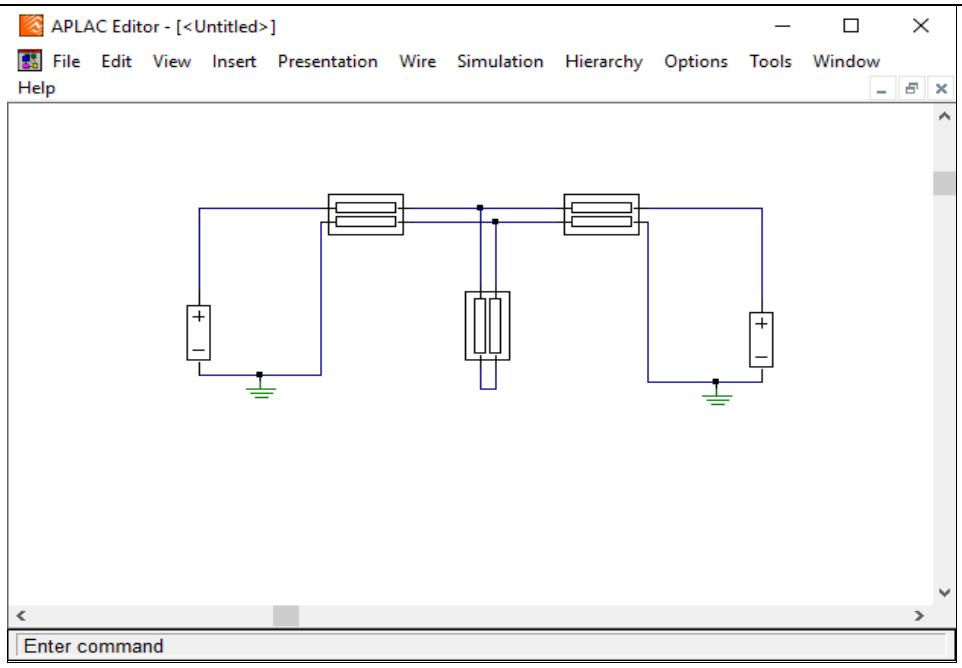
Procedure:



Or Right click anywhere in the open space > Select Basic > Select the component Ind and Cap. Or Right click anywhere > Select Basic > Select the component of your choice. First insert 'Tline' (for transmission line). Then insert two 'Port' (one for source and other for load).

5. To wire the components click on Wire menu > Start Wiring (Ctrl+W). Join the terminals of the components using wires accordingly. Press Esc to stop wiring. Add the Ground at the load resistance.

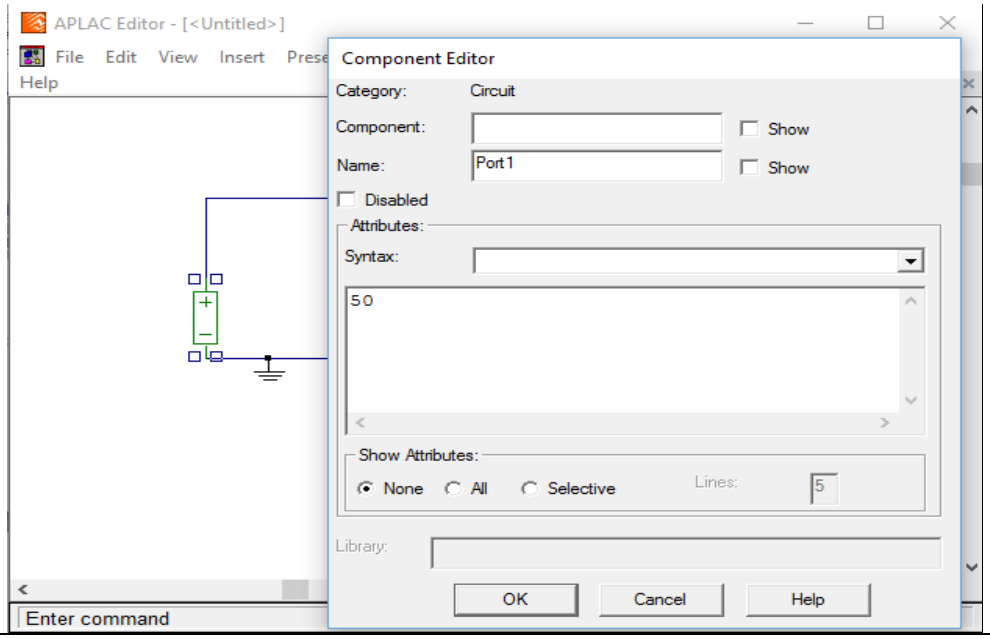




6. Assign specifications of components: Double click on the component to change its values.

   a. For Input/Output port:

   Input port impedance default value is 50 Ohms (No need to state the units). You can change this source impedance value.

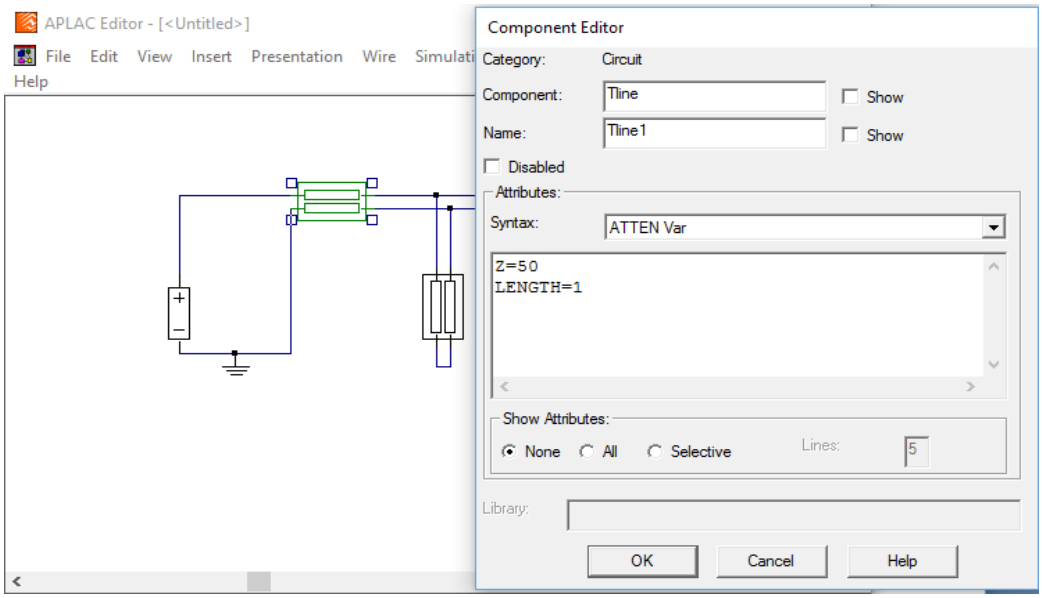


Similarly double click on output port and enter impedance value (eg .75+150*j).

b. For Transmission Lines:

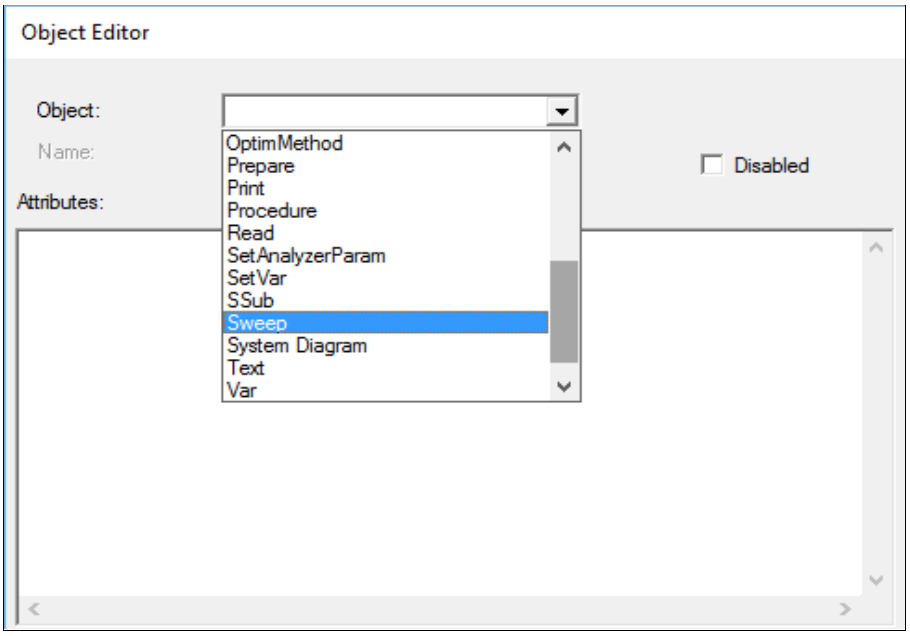
Here we need to specify the characteristic impedance and length of each transmission line.

For the transmission line joining the source and the circuit, enter arbitrary length. Usually 1 meter. Eg. Z=50
LENGTH=1 (The length is in meters)

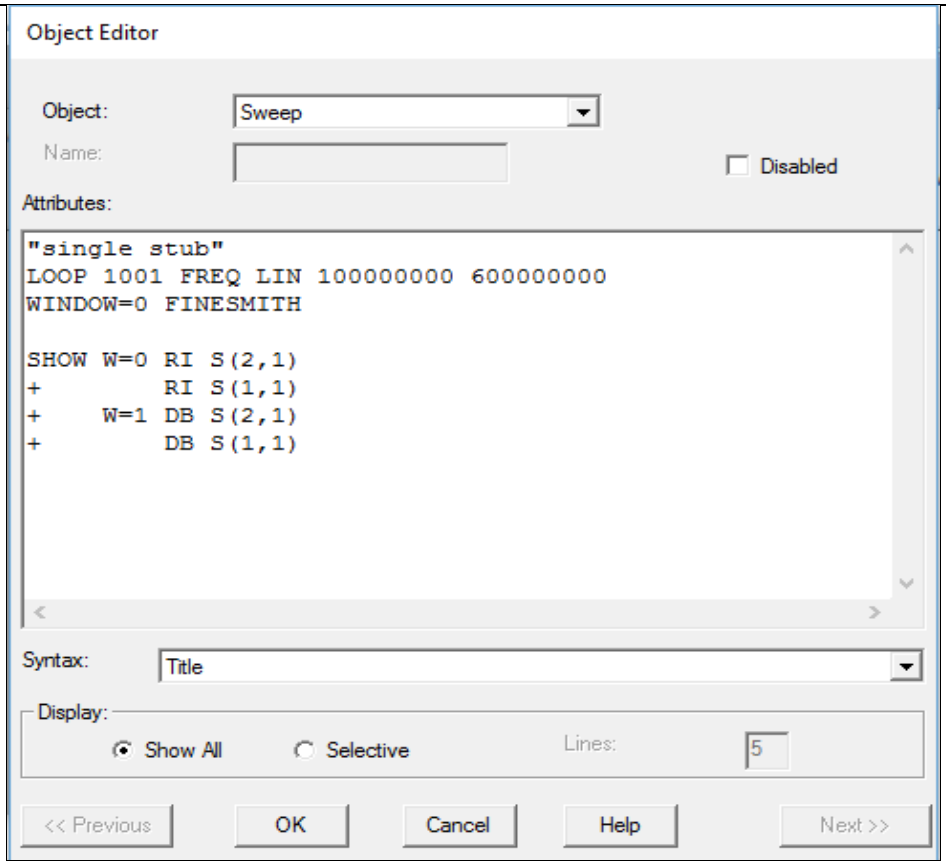


Similarly calculate and enter impedance and length of other transmission lines.

6. To assign the working conditions of the circuit:
   Select Insert menu > Control object
   In the Object Editor window select Sweep mode for the Object.



In the text space provided type the following code:



7. To simulate the file click on the Simulation menu > Simulate (Ctrl+S). Observe the S_{11}, S_{21} parameters and the Smith Chart graph.