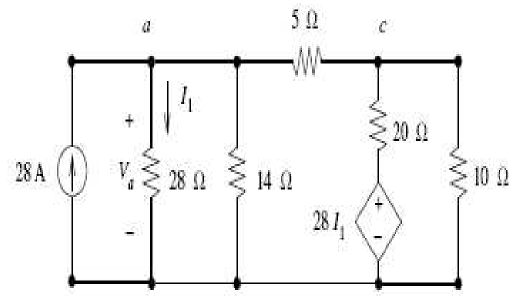
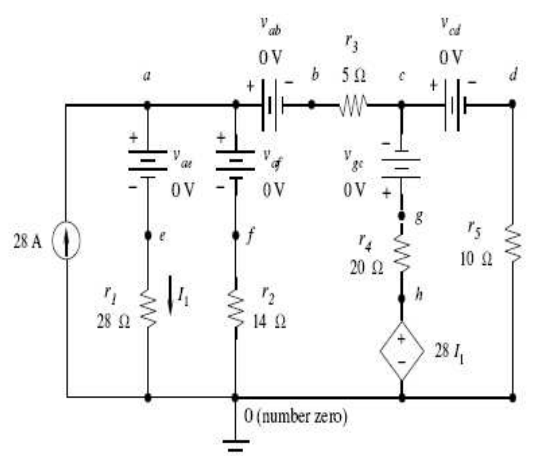
ANALOG SIMULATION EXERCISES USING SPICE OPUS

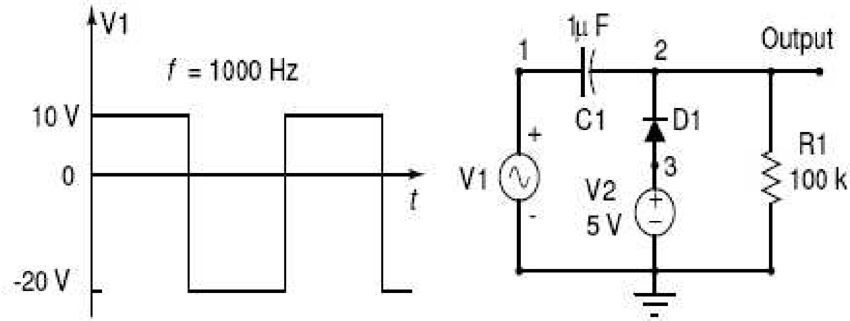
PROBLEM-1: In the network shown in next left Figure determine and print the voltage V_a and current delivered by the controlled source. (This is Problem is based on page no. 84, taken from, A. E. Fitzgerald, David E. Higginbotham, and Arvin Grabel, "Basic Electrical Engineering", ISBN 0- 07-021152-3, Fourth Edition, 1975, McGraw-Hill Kogakusha Ltd.)





In SPICE, we use dummy voltage source —a voltage source of zero value—as an ammeter to measure current. Spice Opus always calculates current entering the positive terminal of a voltage source. Dummy voltage source polarity is decided on this basis.

PROBLEM-2: Next Figure shows a diode clamper (Shift the dc voltage level of a signal) and its input voltage waveform. The diode is assumed to be 1N4007. The 1N4007 model is not present in parts.lib [1](library of components). Find the spice model from the manufacturer's website and create the library and proceed with the simulation according to next Figure. Plotting the Input (V1) and the Output signal together for the first 5ms



The model can be introduced as a model statement copied into parts.lib file[2], then, this can be recalled by .include parts.lib.

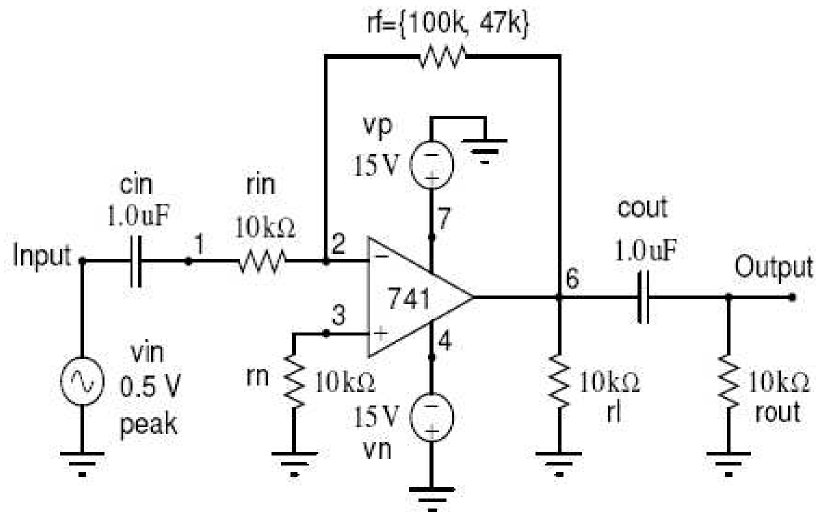
PROBLEM-3: Now we will simulate a non-inverting amplifier which uses op-amp μA741. This op-amp has a complex circuit in side. In SPICE we can define any circuit as a sub-circuit. Obtain the SPICE model from one of the manufacturer's web site and copy it in the library created at PROBLEM-2. At this stage, it is not required to understand how the model shown above is derived. Such models are available freely on the internet for most of the commonly used op-amps. Once it is copied to parts.lib, we can simply call this sub-circuit from our circuit. Plot the time (5ms) and frequency response (from 10 to

---

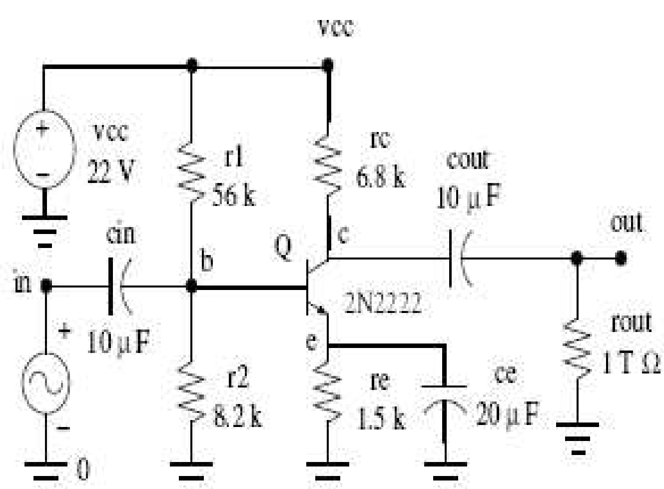
1Typing such a lengthy .model statement is cumbersome. So, all such .model statements for different devices can be collected together and stored in a library . You can give an appropriate name to that, say parts.lib. This can be stored in a directory which is accessed by Spice Opus (Usually /usr/lib/spice/lib/scripts in Linux machines. In Windows machines c:\SpiceOpus\lib\scripts). Then, the contents of this library can be included in the circuit by writing the statement *.include parts.lib* in the circuit. If it is not available, then, you have to create it by collecting the SPICE models from the Internet.

2 It is specified in the circuit by two statements having the format given below:
D??????? collector_node base_node emitter_node MODELNAME
.model MODELNAME D(Parameter1=x Parameter2=x ...)
Here ??????? is the label given to a particular transistor in the circuit, and MODELNAME should be same in both the statements. For each type number, its .model statement should appear only once in the circuit. This statement contains parameters of the transistor which depend upon the construction. And remember it is unnecessary to go in to the meaning of each of the parameters given in the statement above.

10^6Hz in log scale) of non-inverting amplifier using op-amp μA741.



PROBLEM-4: In the following circuit we will simulate an R-C coupled amplifier. We obtain the frequency response and time response of the amplifier when a sinusoidal signal is applied at the input. Next Figure shows the amplifier circuit to be simulated. To obtain the time response, the command is tran.



An NPN BJT q2N2222 is specified in the circuit by the model statement as a Bipolar transistor. Plot the time response of BJT amplifier, the phase angle of gain of BJT amplifier plotted versus frequency and the decibel gain of BJT amplifier plotted versus frequency (up to 10^7Hz).