Department of Electronic Science

ELSEC 351: Paper X:SEC 1: Electronic Design Automation Tools

Unit 1: Introduction to EDA Tools

Sr.No.	Questions
1	What is EDA? Ans: Electronic Design Automation.
2	Which Component are used in EDA? Ans: Resistor, Capacitor, Transistor, Inductor, wire, Transformer, IC.
3	What is Simulation? Ans: Simulations are usually computer-based, using a software-generated model to provide support for the decisions of managers and engineers as well as for training purposes. Simulation techniques aid understanding and experimentation, as the models are both visual and interactive
4	Write the step of Transient analysis using LTSPICE/SPICE? Ans: - Choose Simulate →Edit Simulation Cmd - Indicate Stop Time of 0.1 s - Click OK and place SPICEdirective somewhere on your circuit - Ready to go
5	Write the step of AC analysis using LTSPICE/SPICE? Ans: - Choose Simulate →Edit Simulation Cmd - On AC analysis tab specify type of sweep (decade), number of points per decade, initial and final frequencies - Click OK, plant SPICE directive somewhere - Ready to go
6	What is EDA? Ans: Electronic Design Automation or EDA is a market segment consisting of software, hardware, and services with the collective goal of assisting in the definition, planning, design, implementation, verification, and subsequent manufacturing of semiconductor devices, or chips.
7	Write the step LTSPICE/SPICE analysis? Ans: "Draw" your circuit, then specify all of the components, then select Simulate →Edit Simulation Cmd

		- Transient analysis - AC small signal analysis - DC sweep - Noise analysis - DC transfer function - DC operating point
8		**Write the Simulation Tools?** **Ans:** Simulation tools *not* considered in this paper are: pure Computer Aided Robotics tools such as Rotsy (Motoman) - Business Process Re-engineering tools, IDEF-like modeling tools, Virtual Reality tools - Numerical simulation tools (Matlab, etc.) - Pure simulation languages such as SLAM, Siman, or SimScript.
9		**Why Simulation is used.** Simulation is used to evaluate the effect of process changes, new procedures and capital investment in equipment. Engineers can use simulation to assess the performance of an existing system or predict the performance of a planned system, comparing alternative solutions and designs. Simulation is used as an alternative to testing theories and changes in the real world, which can be costly. Simulation can measure factors including system cycle times - throughput under different loads, - resource utilization - bottlenecks and choke points - storage needs - staffing requirements - effectiveness of scheduling and control systems.
10		**Write the types of Simulation.** Ans: Simulation can be broken down into three overarching types, as follows: **1)Discrete Event Simulation**

Modelling a system as it progresses through time, for example;

- factory operations (stamping, turning, milling)
- traffic analysis (roads, networks, queues)

2) Dynamic Simulation

Modelling a system as it progresses through space, for example;

- machine kinematics
- human ergonomics
- aerodynamic testing
- virtual prototyping

3)Process Simulation

Modeling physical interactions between two or more systems, for example;

- in-service product modeling
- in-manufacture product modeling
- weather forecasting

11	Why do we need Simulation. • Simulation in logistics is needed to respond to requirement changes in industry, to improve operations and to quantify the potential impact of such improvements. • It is useful when experimentation with the real system is expensive • dangerous or likely to cause significant disruption (e.g. transport systems, nuclear reactor and airline systems). • It might also be an option when mathematical modeling of a system is impossible • Although mathematical analysis methods are available some are so complex that simulation may provide a simpler solution • Oil exploration, meteorology and computer networks are all examples when simulation might be favoured over mathematical modeling. • Logistics operations management requires more sophisticated technologies, such as simulation that can handle the inherent uncertainty of real-world logistics systems.

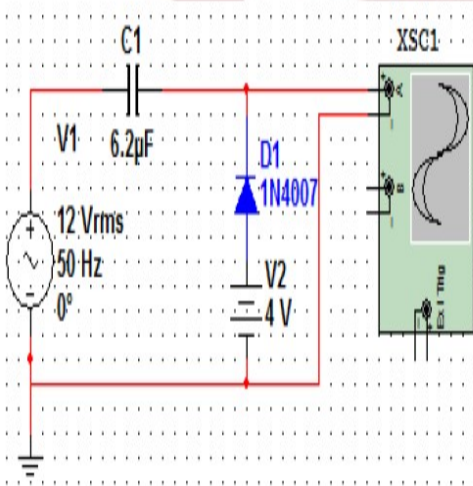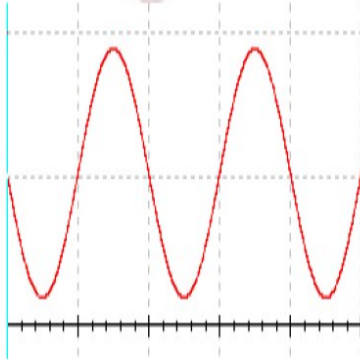|	
12	What is the Difference between schematic and wiring diagram? Ans: **Schematic diagram** - A **schematic diagram** is a circuit which shows the connections in a clear and standardized way - A schematic diagram shows the components and their values and connections in an understandable manner - It is usually used to communicate or intended to convey the connections and working of the circuit to other engineers. **Wiring diagram:** - A **wiring diagram** is mainly intended to convey the wiring or connection between the components in a proper way without any confusion - one can create a prototype easily using that.

Department of Electronic Science

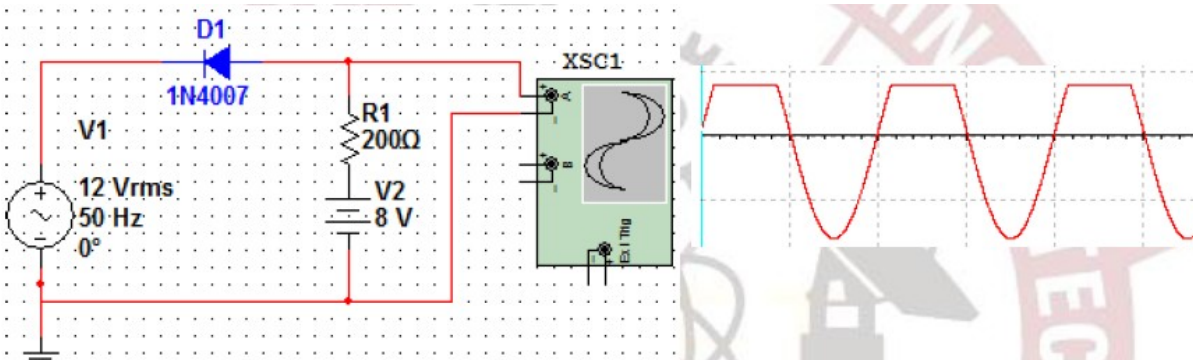## ELSEC 351: Paper X:SEC 1: Electronic Design Automation Tools

Unit 2: Electronics Designing using LTSPICE/PSPICE

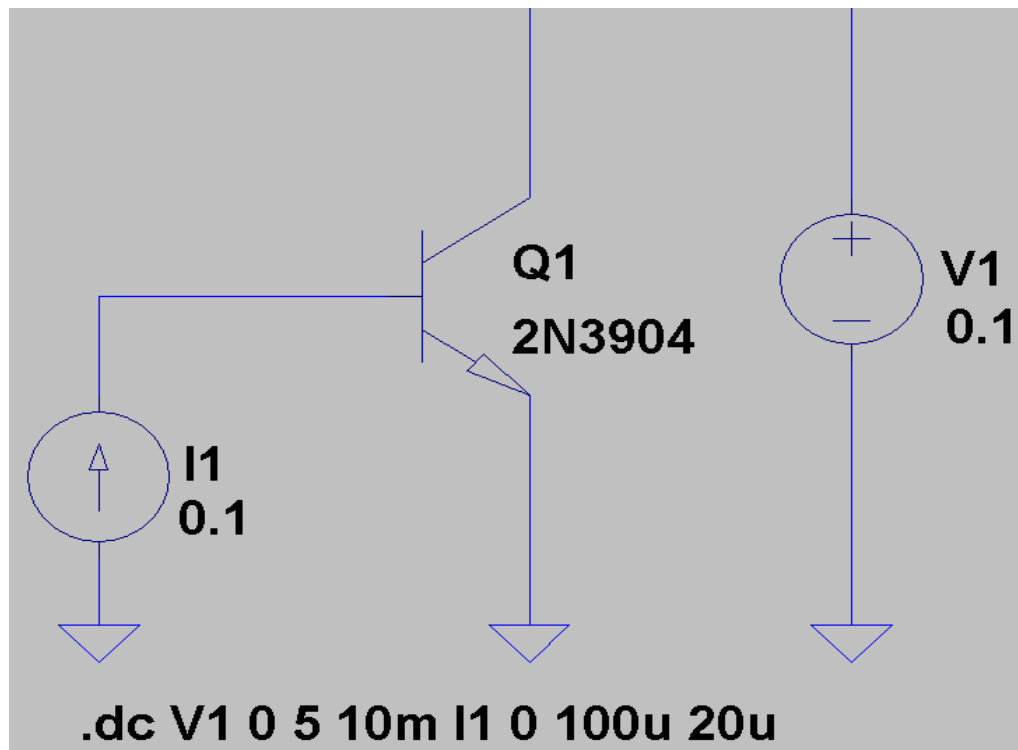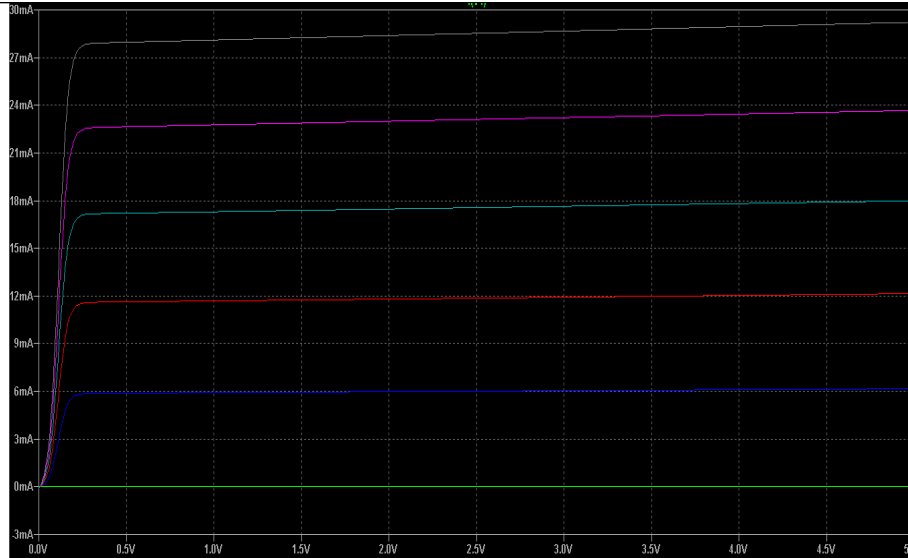| Sr.No.	Questions

1	Write Full form LTSPICE? Ans: Linear Technology Simulation Program with Integrated Circuit Emphasis.
2	What is PSpice stand for? Ans: PSpice is an acronym for Personal Simulation Program with Integrated Circuit Emphasis.
3	What is LTSPICE? Ans: - Ans: LTSPICE has schematic capture and is much easier to use than traditional text-based SPICE. - The user can enter a circuit to be simulated via a graphical user interface.
4	What component are used in LTSPICE? Ans: Resistor, Capacitor, Transistor, Inductor, wire, Transformer, IC.
5	What are the advantages of PSpice? Ans: compared to any other IC vendor solution - Seamless product and library update PSpice is considered as one of the most accurate mixed signal simulators in the market - It is widely used in Automotive, Power Supply and Internet of Things domain.
6	Which simulation are used in LTSPICE/SPICE? Ans: There are a number of powerful simulation tools available - Mathematical input - Netlist input - Graphical input
7	What is LTspice software used for? Ans: LTspice is Analog Devices' high performance circuit simulation program - It allows you to draft, probe, and analyze - the performance of your circuit design LTspice contains an integrated schematic editor, waveform viewer, and advanced features they are easy to use once you learn some basic commands.
8	Why is PSpice used? Ans: PSpice lets you simulate and analyze your analog and mixed-signal circuits within OrCAD - PSpice calculates complex node voltages and branch currents at each frequency across your design - to place probes and generate waveform plots for further analysis

Prepared by Ms. Jayashree B Khilari (8624053490)_EDA Tools

9	What are the features of PSpice? Ans: PSpice analog and digital circuit simulator helps you with: • Model development. • High performance. • Accuracy and better reliability. • Mixed-Signal simulations. • Flexibility.
10	What is the simulation Command? Ans: i) To run a simulation specify the type of analysis to be performed ii) There are six different types of analysis • Transient analysis • Small signal AC • DC sweep • Noise • Dc transfer function • DC operating point iii) Simulation commands are placed on the schematic as text • Dot commands
11	Why use LTspice? Ans: i) Stable SPICE circuit simulation • Unlimited number of nodes • Schematic/symbol editor • Waveform viewer • Library of passive devices. ii) Fast simulation of switch mode power • Steady state condition • Turn on transient • Step response • Efficiency iii) Advanced analysis and simulation options iv) outperforms or as powerful as pay for tools. v) Automatically builds syntax for common tasks.
12	What is circuit design and circuit analysis Ans: **Circuit analysis**: • Circuit analysis answers the question "given this circuit and these known voltages and currents what are these other voltages and currents" • There are a number of very good tools for answering this question. **Circuit design**: • Circuit design asks more-or-less the opposite question: "given these

	known voltages and currents (e.g. inputs) • these desired voltages and currents (e.g. outputs) what circuit will accomplish this
13	How To Make a Schematic Diagram. Ans: • Create the First Symbol. From (File > New) under Name: type Schematic. • Use the Pick tool to select both lines, and press Ctrl + D to duplicate them. • Right-click on the selected lines and select Symbol > New Symbol. • Add More Symbols. • Lay out the Symbols. • Connect the Circuit. • Add Text.
14	Difference between LTSPICE and SPICE Ans: LTSPICE: • LTspice is a high performance SPICE simulation software • schematic capture and waveform viewer with enhancements • models for easing the simulation of analog circuits. **PSPICE:** • PSpice technology provides the best, high-performance circuit simulation to analyze • refine your circuits, component • parameters before committing to layout and fabrication
15	Draw the circuit diagram of positive clamper circuit?What is the output on simulation using LTSPICE/SPICE? Ans:

16	Which Components are displayed on Schematic Toolbar in LTSPICE/SPICE? Ans: • Arbitrary behavioral source • Voltage dependent voltage • Current dependent • Lossy transmission line • Bipolar transistor current • Voltage controlled switch • Voltage dependent current • Current dependent voltage • Independent current g • Lossless transmission line • Uniform RC-line • source • Independent voltage • JFET transistor • Mutual inductance source • Current controlled switch • Sub circuit • MOSFET transistor • MESFET transistor
17	Draw the circuit diagram of positive clipper circuit?What is the output on simulation using LTSPICE/SPICE? Ans:
18	Draw the circuit of transitor biasing circuit? What is the output on simulation using LTSPICE?SPICE? Ans:

.dc V1 0 5 10m I1 0 100u 20u

Department of Electronic Science

ELSEC 351: Paper X:SEC 1: Electronic Design Automation Tools

Unit 3: Introduction of Multisim/CircuitMod

Sr.No	Questions
1	Write the flow of Schematic Ans: A blank schematic Circuit 1 is automatically created. To create a new schematic click on File – New – Schematic Capture. • To save the schematic click on **File /Save As**. • To open an existing file click on **File/ Open** in the toolbar.
2	What is place components? Ans: • To Place Components click on **Place/Components.** • On the Select Component Window click on **Group** to select the components needed for the circuit**.** • Click OK to place the component on the schematic.
3	What is wiring components? Ans: • To connect resistors click on **Place/Wire** drag and place the wire. • Components can also be connected by clicking the mouse over the terminal edge of one component and dragging to the edge of another component.
4	What is Multisim?

|	
	Ans: Multisim software provides SPICE simulation, analysis, and printed circuit board (PCB) tools to help you quickly iterate through designs and improve prototype performance
5	What are the features of Multisim software? Ans: - Expanding the Multisim Experience. - Create Circuit Schematics on any platform. - Interactive Circuit Simulation With No Installation. - Same Industry-Standard SPICE Simulation. - Share Your Circuit Brilliance. - Complete Circuits Teaching Solution. - New Features All the Time.
6	What are three types of circuit analysis are available in Multisim? Ans: There are three types of analysis that can be performed on the circuit while the component is manipulated: - DC Operating Point - Transient Analysis - AC Analysis
7	Write the Application of Multisim/CircuitMod? Ans: - Semiconductor Analysis Applications - Aerospace and National Research Applications - Low-Frequency Circuit Design for RF Applications - Automotive Electronics Applications
8	Write the steps for AC Voltage using Multisim/CircuitMod. Ans: - To place an AC signal source, go to the Sources group and highlight the SIGNAL_VOLTAGE_SOURCES family. - Select the AC_VOLTAGE component - Place and wire the AC voltage source - Place and wire a Ground component to the AC source
9	How to use Multisim/CircuitMod software. Ans: - Open Multisim/CircuitMod software. - Building your first circuit. - Simulating your circuit. Saveing, editing and deleting circuits.
10	What is Multisim/CircuitMod? Ans: Multisim is an industry-standard, best-in-class SPICE

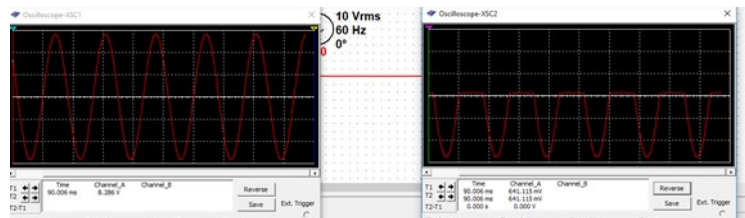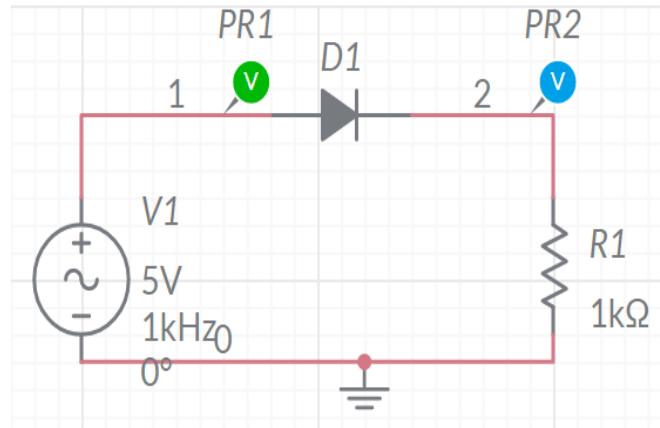|	
	simulation environment. • It is the cornerstone of the NI circuits teaching solution to build expertise through practical application in designing, prototyping, and testing electrical circuits. • It is industry standard SPICE simulation and circuit design software for analog, digital, and power electronics in education and research. • Multisim software integrates industry-standard SPICE simulation with an interactive schematic environment to instantly visualize and analyze electronic circuit behavior.
11	Draw the circuit diagram of Positive clipper circuit? What is output on Multisim/CircuitMod? Ans:
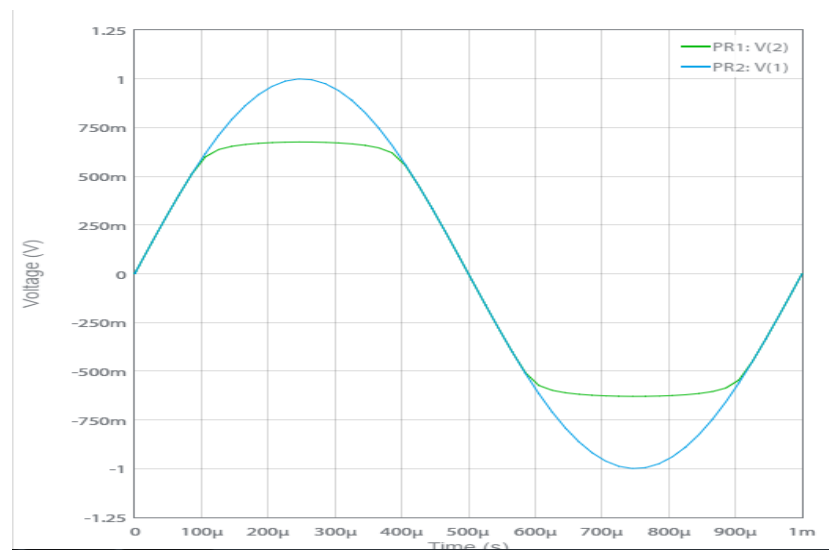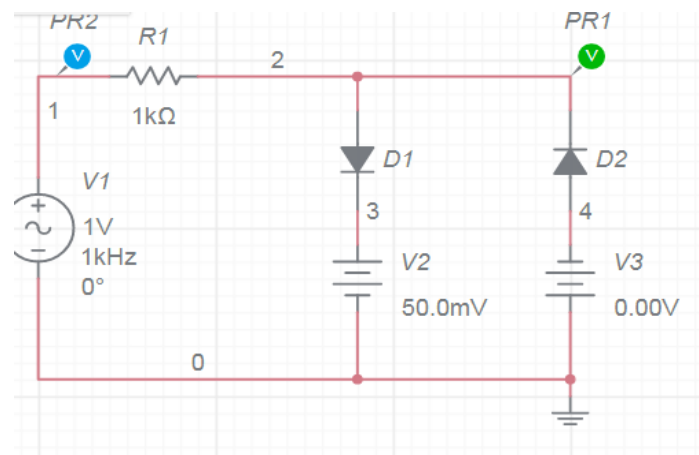
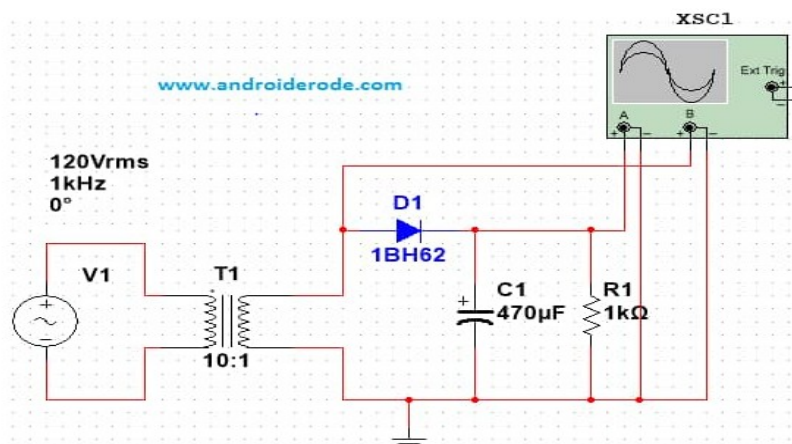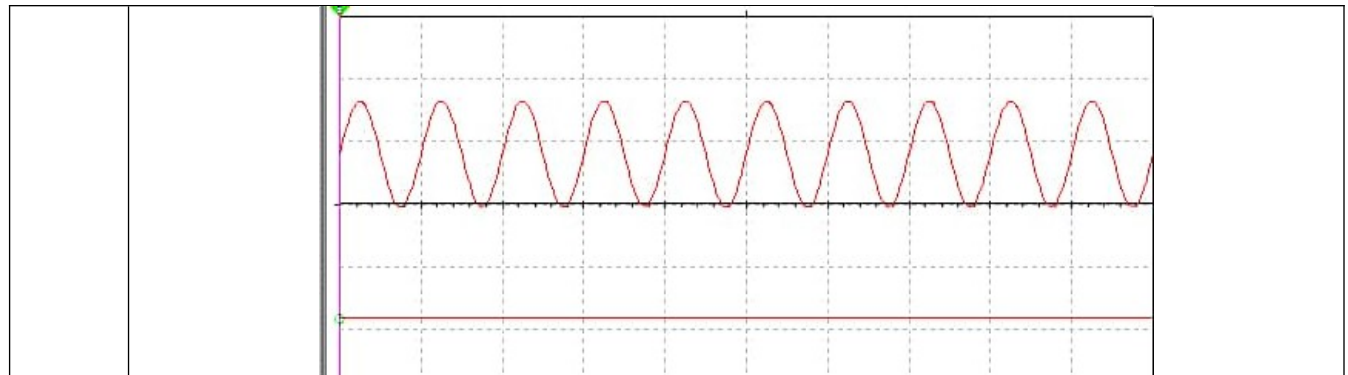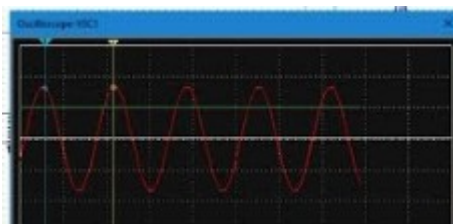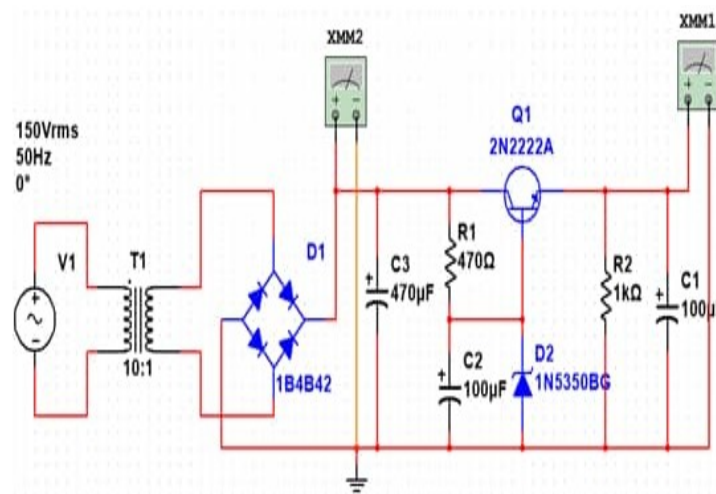| 12	Draw the circuit diagram of negative clipper circuit?What is output on Multisim/CircuitMod? Ans:
13	Draw the circuit diagram of half wave rectifier?What is output on Multisim/CircuitMod? Ans:

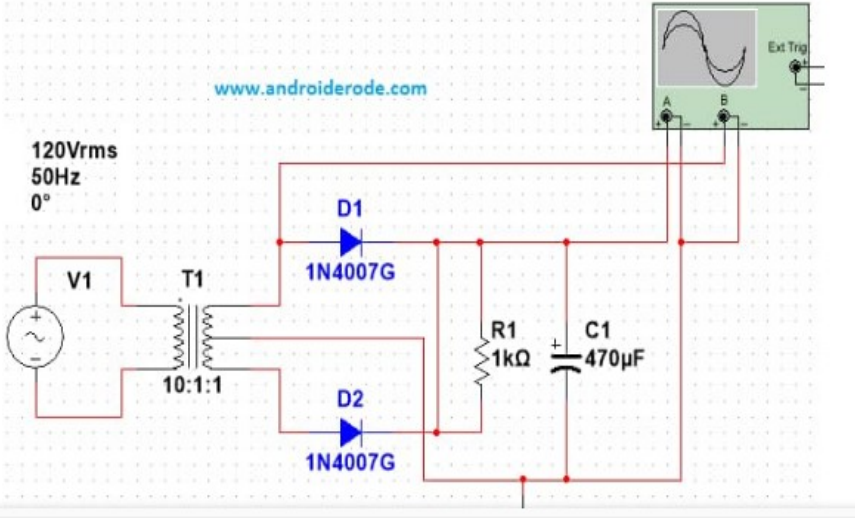| 14	Draw the circuit diagram of voltage regulator?What is output on Multisim/CircuitMod? Ans:
15	Write the Steps Placing Resistors, Inductors or Capacitors? Ans: The procedure used to place resistors, inductors or capacitors is quite similar to the procedure used to place other types of components. However, there are some differences. Complete the following steps to place an R, L or C component: o Click on the Basic group in the Components toolbar. The Select a Component dialog box appears with the Basic component group displayed. Alternatively, you can display the Select a Component dialog box by choosing Place» Component and selecting the Basic group from the Group drop-down list. Or Right-click in the workspace and select Place Component from the pop-up that appears.

		o If it is not already selected, select the Master Database in the Database drop-down list. o Click on the desired component family in the Family list, for example Resistor. o Type the value of the component that you want to place in the field at the top of the Component list. The value does not need to appear in the list to be placed on the schematic. • Optionally, enable the Save unique component on placement checkbox. When this is enabled, any component with a unique combination of the values in the various fields of the Select a Component dialog box is saved.
16		Write the steps for Schematic using Multisim/CircuitMod? Ans: Multisim is a modeless wiring environment. This means that Multisim determines the functionality of the mouse pointer by the position of the mouse. You do not have to return to the menu to select between the placement, wiring, and editing tools. • Begin wiring by moving the mouse pointer close to a pin of a component. The mouse appears as a crosshair rather than the default mouse pointer. • Place an initial wire junction by clicking on the pin/terminal of the part (in this case, the output pin of the opamp). • Complete the wire by moving the mouse to another terminal or just double-click to anchor the termination point of the wire to a floating location somewhere in the schematic window. • Create a copy of the ground symbol using Copy <Ctrl+C> and Paste <Ctrl+V>. • Complete the wiring as shown in Figure 5. Do not worry about the labeled numbers on the wires (also called nets). The last key step is to connect the power supply terminals to the positive and negative power rails of the opamp via a virtual connection using On-page connectors. • Select **Place»Connectors»On-page** connector and connect it to the positive terminal of the **V1** power supply. The **On-page Connector** window will open. • Enter **+V** in the **Connector name** field and click **OK**. • Select another On-page connector and connect it to terminal 8 of the opamp. The **On-page Connector** window will open again. • Select the **+V** connector in the **Available connectors** list and click **OK**. The positive terminal of the **V1** DC power supply is now connected to pin 8 of the opamp via a virtual connection. • Repeat steps 6 to 9 to connect the negative terminal of **V2** to pin 4 of the opamp

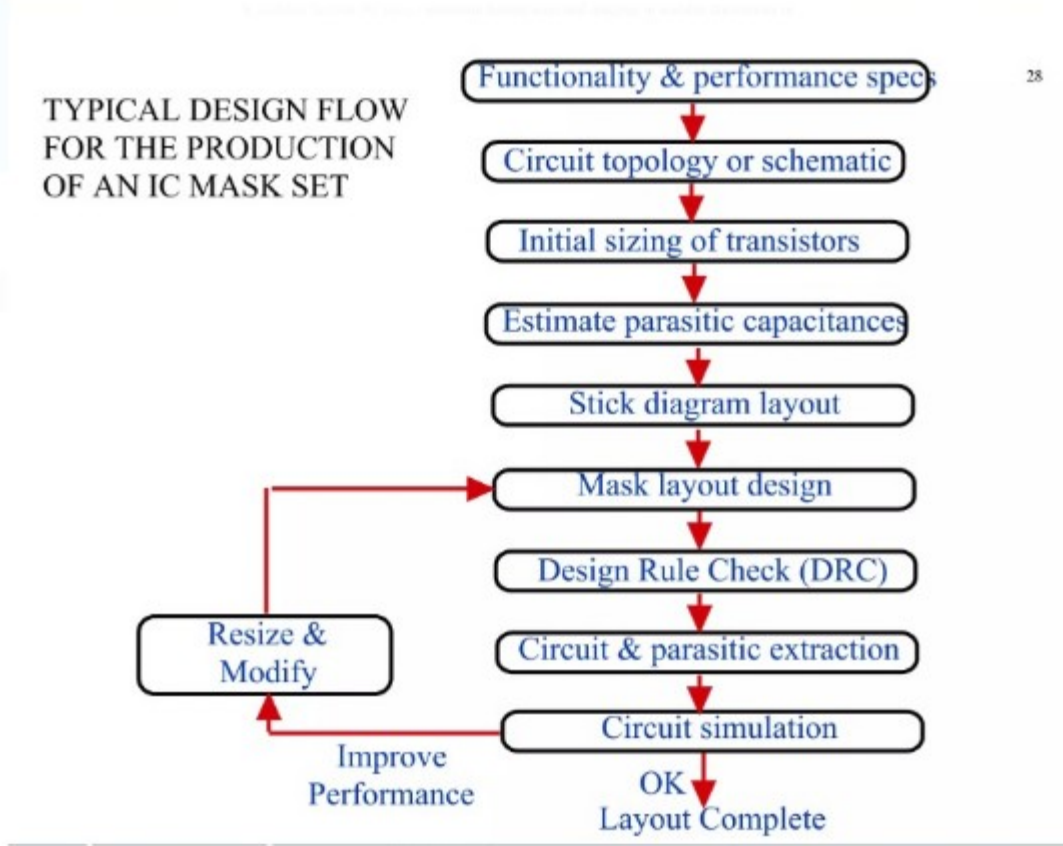|	
17	Write Steps of Simulation using Multisim/CircuitMod? Ans: • Select the **Oscilloscope** from the menu and place this onto the schematic. • Wire the Channel A and Channel B terminals of the Oscilloscope to both the input and output of the amplifier circuit. • Place a ground component and connect it to the negative terminals of the Oscilloscope. • Right-click the wire connected to Channel B and select **Segment color**. • Select a shade of blue and click the **OK** button. • Select **SimulateRun** to start the simulation. • Double-click on the **Oscilloscope** to open its Front Panel and observe the simulation results (see Figure 9). As expected, the input signal is being amplified by a factor of 2. • Stop the simulation by pressing the red stop button in the simulation toolbar.
18	Draw the circuit diagram of Full wave rectifier?What is output on Multisim/CircuitMod? Ans:

Oscilloscope-XSC1   www.androidedode.com   FWR

Department of Electronic Science

ELSEC 351: Paper X:SEC 1: Electronic Design Automation Tools
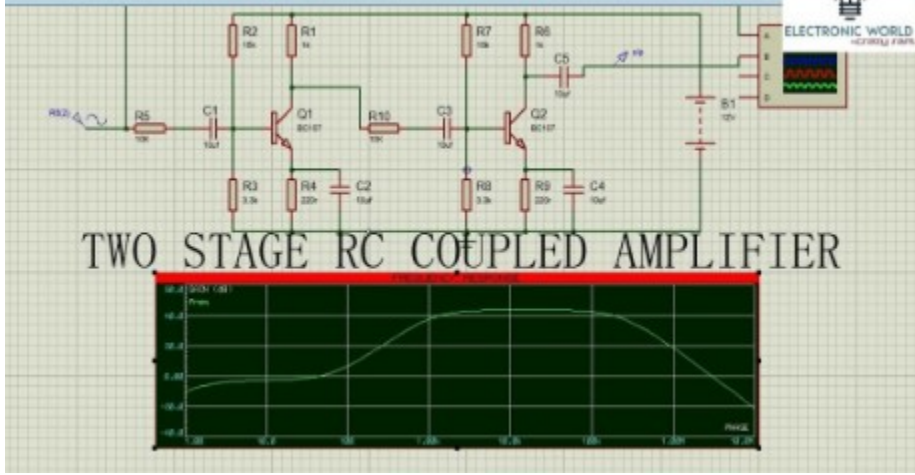
Unit 4: Introduction of Proteus/OrCAD

Sr.No	Questions
1	What is Proteus/OrCAD? Ans: Proteus is used to simulate, design and drawing of electronic circuits proteus you can make two-dimensional circuits designs as well
2	What is Schematic in Proteus/OrCAD? Ans: The schematic should provide this additional information to ensure that appropriate components are selected.
3	What is netlist in OrCAD? Ans: If a net is a connection between two components, a netlist is simply a list of the electrical connections that describe a circuit. Netlists can vary widely in terms of formats and the amount of information they convey.
4	What are the 3 basic types of layout? Ans: There are mainly three types of layout: • Product or Line Layout • Process Layout • Combination of Product and Line Layouts.
5	What is DRC? Ans: • A Design Rule Check (DRC) checks your design for violations and identifies problem-areas with schematic error markers. • The DRC report lists all objects checked and the rule violations detected.
6	Which software used in PCB layout? Ans: Proteus, OrCAD.
7	What is DRC What kind of DRC errors you find in PCB design? Ans: • Image result for what are different types of design rule check (drc) in proteus and orcad • A design rule check is a set of rules used by a designer to ensure the schematic matches all manufacturing considerations and dimensional tolerances set for a board. • Production processes have a margin of error for the variation in the products that are manufactured.

8	Write Features of Proteus/OrCAD? Ans: • There are 2 main parts of proteus first is used to design and draw different circuits and the second is for designing of PCB layout. • First is ISIS that used to design and simulate circuits. And second is ARES that used for designing of a printed circuit board. • It also provides features related to the three-dimensional view of design in PCB
9	What are the four types of design rules? Ans: There is several levels of design rules: • well rules • transistor rules • contact rules • metal rules • via rules • other rules
10	How to select the components in proteus /OrCAD software ?Explain? Ans: • Open the Part Picker (S12 or M14) • After the "Pick Devices" window has appeared, begin writing keywords Click "OK" to add the part to your list of component • Repeat step 2 for all other components while making sure that all components have "Simulator Model(s)". If "No Simulator Model" is not written, then it is assumed that there is a simulator model for the selected component. • Now we add other required components such as: Switches, LEDs, Clock, etc.
11	What are the rules to be followed in layout? Ans: • The layout design rules provide a set of guidelines for constructing the various masks needed in the fabrication of integrated circuits. • Design rules are consisting of the minimum width and minimum spacing requirements between objects on the different layers.
12	Expand the Online DRC window. Ans: • Online DRC is new version in 17.4 reports real-time warning violations. Enabling you to correct errors while you design. • the other windows in the Capture workspace, the Online DRC window can be either floating or docked. • Cross probe between the violation and the schematic by double clicking a line item. • In the project hierarchy, select the design file. • lect PCB > Design Rule Check from the menu

	• In the Options tab, for DRC Action set Run on Design • For Use Properties (Mode) set Instances (Preferred) • For Warning set Create DRC Markers. • For Show DRC Output set DRC Window. • Browse to the location to save the report..
13	Write the design flow of PCB? Ans: TYPICAL DESIGN FLOW FOR THE PRODUCTION OF AN IC MASK SET Functionality & performance specs → Circuit topology or schematic → Initial sizing of transistors → Estimate parasitic capacitances → Stick diagram layout → Mask layout design → Design Rule Check (DRC) → Circuit & parasitic extraction → Circuit simulation → OK → Layout Complete Resize & Modify — Improve Performance
14	How place a components in Proteus/OrCAD? Ans: • Proteus works with a live netlist so the PCB Layout module already has much of the information we need to start the layout process. • In particular, we have specified which footprints are associated with each schematic symbol and Layout module can therefore pre-select these for us ready for placement. • This brings us to an important distinction in the software; the difference between a component and a package. • Selecting component mode will access footprints which have been specified as relating to parts in Proteus and carry connectivity information whereas selecting package mode will access 'unbound' physical instances of a footprint. • When working with a layout which is driven from a schematic we

	therefore exclusively use component mode.
	- The Component Mode icon is second from the top directly underneath the Selection icon. - Clicking on this will display a list of items in the Object Selector which correspond directly to the parts in Proteus that we used to create the schematic. The Package Mode icon is directly underneath the Component mode icon and clicking on this will show us the physical footprints corresponding to the components in the layout
15	Write the steps for Proteus/OrCAD? Ans: - Click on "New Project" under the "Start" header in the home page: - Choose a name and file-path for your project - Select the Schematic Design: - Select the PCB Layout - Select Firmware - Editing source code is beyond the scope of this guide, select "No Firmware Project" and click "Next" - Summary page - You should now see a summary of all your previous selections. If any mistakes were made, you can click "Back" and correct them. - Click "Finish" to open the Schematic Capture module and begin working on your project
16	How to Make Circuit in Proteus Ans: - Step 1 First of all click on proteus Icone in your computer and click on a new file option - step 2: After that, you will see the drawing sheet - Step 3: After a move to the component option - Step 4: After clicking on components mode you will see two buttons P and L. If you move to P button you will see Pic from Libraries. - Step 5: When you will click on the P button - Step 6: When you will select components for your project you will see them in a box - Step 7: When you connect all components in the circuit like run button in left bottom - Step 8: When you will observe the simulation of your circuit than click on stop button on the left bottom to stop the working of the circuit.
17	Draw the circuit diagram of RC coupled amplifier?What is output on Proteus/OrCAD? Ans:

TWO STAGE RC COUPLED AMPLIFIER

18	Difference between OrCAD and Proteus? Ans: Proteus – - Proteus is a great electrical suite for circuit simulation purposes. - You can simulate a single processor or multiple ones at the same time. - The application is a great alternative for Virtual System Modelling. OrCAD: - OrCAD PSpice technology provides the best, high-performance circuit simulation to analyze and refine your circuits, components, and parameters before committing to layout and fabrication - OrCAD is the general name for the software suite that helps you design PCBs. - OrCAD EE typically runs simulations for circuits defined in OrCAD Capture - it can optionally integrate with MATLAB/Simulink, using the Simulink to PSpice Interface