GOVERNMENT OF TAMILNADU

DIRECTORATE OF TECHNICAL EDUCATION, CHENNAI-25

STATE PROJECT COORDINATION UNIT

(Established under Canada India Institutional Cooperation Project)

CURRICULUM

Course Name	**CIRCUIT SIMULATION USING MULTISIM**
Course Code	**EC/2020/009**
Course Duration	40 Hours
Minimum Eligibility Criteria	10th/+2 /ITI/Diploma/Graduate
Pre-requisites (if any)	-
Course Objectives	Training module has been designed for the participants to • Understandthe requirements of software simulation • Practice on Simulation of electroniccircuits such as rectifiers, filters and Amplifier in Multisim Environment • Practice on Designing of PCB Layout using Multisim
Course Outcomes	At the end of training, the trainees will be able to • Develop circuit in Multisim environment. • Simulate the electronic circuits and obtain the response of it. • Analyse the function of each component in the developed circuit • Design PCB layout.
Expected Job Roles	Electronic Circuit Designer

TEACHING AND SCHEME OF EXAMINATION

Course Code	Course Name	Hours		Assessment Marks		Duration of Examination
				Min	Max	
EC/2020/009	CIRCUIT SIMULATION USING MULTISIM	Theory	16	10	20	3 Hours
		Practical	24	40	80	
		Total	40	50	100	

EC/2020/009– CIRCUIT SIMULATION USING MULTISIM

DETAILED SYLLABUS

Unit No	Modules	No.of.Hours	
		Theory	Practical
I	**Introduction to Multisim**	**10 Hours**	
1.1	Introduction to software and installation	05	05
1.2	Start up screen, Toolbars, Pop-up Menus, Open/Crate		
1.3	Placing components, Components rotation and Making		
1.4	Changing components values		
1.5	Labelling - Grounding and Simulation		
1.6	Introduction to Multisim Instruments.		
II	**Circuit Simulation**	**20 Hours**	
2.1	Simulation of Series and Parallel circuits, Half wave and Full wave rectifier	07	13
2.2	Bridge rectifier, Power supply design with regulator		
2.3	Clippers and Clampers		
2.4	Types of logic gates		
2.5	Low pass and High pass filters		
2.6	Commutator and Attenuator		
2.7	Astable and Monostablemultivibrator		
2.8	Opamp applications – Instrumentation amplifier		
2.9	Frequency modulation and demodulation		
III	**PCB Design**	**10 Hours**	
3.1	Transferring the design for PCB layout	04	06
3.2	Board outline selection, Part placement		
3.3	Layer management		
3.4	PCB file generation		
3.5	Procedure for single sided PCB		
	Total theory / Practical Hours	**16**	**24**
	Total hours	**40**	

HARDWARE REQUIREMENT

S.NO	LIST OF TOOLS /EQUIPMENTS
1	PC/Laptop

SOFTWARE REQUIREMENT

S.NO	LIST OF SOFTWARE
1	Multisim Software

REFERENCE BOOKS

S.NO	NAME OF THE BOOK	AUTHOR	PUBLISHER
1	Multisim getting started	David J. Comer	Wiley, 2002
2	CIRCUITS	Fawmaz T. Ulaby, Michel M.Maharbiz& Cynthia M.Furse	National Technology & Science Press, 2009
3	Applications of NI Multisim in AC Circuit	Basel M. Anwari Korj	-
4	Circuit Analysis with Multisim	David Aez – Lpoez, Felix E.Guerrero - Castro	Morgan & Claypool Publishers

ASSESSMENT AND CERTIFICATION

S.No	Criteria for assessment
1.	A trainee will be assessed based on the performance in End Examination for Theory and Practical conducted internally in the CIICP Project Polytechnic College for a duration of 3 hours
2.	A trainee must have 75% of attendance to appear for End examination in Theory and Practical.
3.	The assessment for theory part will be based on the marks scored in the end examination on the knowledge bank of questions (1 word/objective type questions)
4.	The assessment for practical part will be based on the marks scored in the end examination conducted by the CIICP Project Polytechnic and assessed by the Examiners approved by Strategic Plan Implementation Committee (SPIC) of the project polytechnic.
5.	The passing criteria for successful completion of training is every trainee should score 50% of marks in theory and practical examination.
6.	On successful completion of training, certificate will be issued to the participants by the Directorate of Technical Education through the Project Polytechnics.

END EXAMINATION

ALLOCATION OF MARKS

S.NO	Description	Max. Marks
1.	Theory Examination	20
2.	Practical Examination	
	a)Objective and Simulation Circuit	20
	b)Procedure and Connections / Execution	20
	c)Result and Viva	20
	d)Record	20
	Total Marks	**100**

THEORY MODEL QUESTION PAPER

EC/2020/009 – CIRCUIT SIMULATION USING MULTISIM

(Maximum Marks: 20)

(N.B: Answer any Twenty questions) 20x1= 20 Marks

1. Write any two toolbars in Multisim.
2. Write any two pop-up menus in Multisim.
3. How to rotate the components in Multisim?
4. What is mean by grounding?
5. Which keyboard button is used for run a simulation?
6. What is the use of Multimeter?
7. What is the use of spectrum analyser?
8. Write an equivalent resistor expression of a DC series circuit.
9. How many diodes are used in full wave center tapped rectifier?
10. What is the output voltage of a voltage regulator circuit using LM7805CT?
11. Which are the basic components in a clipper circuit?
12. What is mean by logic gate?
13. Draw the symbol of NOT gate.
14. Sketch the truth table of OR gate.
15. What is low pass filter?
16. What is commutator?
17. What is an attenuator?
18. What is another name of an AstableMultivibrator?
19. What is another name of MonostableMultivibrator?
20. What is the name of PCB editor?
21. What is mean by board outline in PCB design?
22. Write any two versions of Gerber files.
23. Write any two types of single layer PCB.
24. Write any two applications of single layer PCB.
25. Write any two advantages of single layer PCB.