VEMU INSTITUTE OF TECHNOLOGY:: P.KOTHAKOTA

Chittoor-Tirupathi National Highway, P Kothakota, Near Pakala, Chittoor (Dt.), AP - 517112
(Approved by AICTE, New Delhi, Recognized Under 2(f) &12(B) of UGC Act Accredited by NAAC, NBA (EEE, ECE & CSE) & An ISO 9001:2015 Certified Institute)

Department of Electrical and Electronics Engineering

VEMU/EEE/2021-22

Date: 06-05-2022

Circular

It is hereby informed to all the EEE Lab Technicians that there is a one day Workshop on **"Simulation of Basic Circuits using PSPICE"** on **07/05/2022** by inviting Mr.S.Venkat Rao, Assistant Professor / EEE, Vemu institute of technology, P.Kothakota. Hence all the EEE Lab Technicians are instructed to attend the workshop without fail.

Venue : Simulation lab, Block-2, Vemu IT.

Time : 10.00 Am - 12.15 Pm

1.30 Pm - 4.00 Pm

HOD / EEE
HEAD
DEPARTMENT OF EEE
VEMU INSTITUTE OF TECHNOLOGY
P. KOTHAKOTA - 517 112

Copy To: The Principal Desk,

The Staff In-charge,

Circulate among all Lab Technicians.

P. yellaiah..
Sweetha
Durga
Sujatha
A. Vinayak

VEMU INSTITUTE OF TECHNOLOGY:: P.KOTHAKOTA

Chittoor-Tirupathi National Highway, P.Kothakota, Near Pakala, Chittoor (Dt.), AP - 517112

(Approved by AICTE, New Delhi, Recognized Under 2(f) &12(B) of UGC Act. Accredited by NAAC, NBA (EEE, ECE & CSE) & An ISO 9001:2015 Certified Institute)

Department of Electrical and Electronics Engineering

A WORKSHOP ON "SIMULATION OF BASIC CIRCUITS USING PSPICE"

Attendance

S.No.	Name of the Participant	07/05/2022	
		FN	AN
1.	Mr.P.Yerraiah	P. Yerraiah	P. Yerraiah
2.	Mrs.M.Sujatha	Sujatha	Sujatha
3.	Mr.A.Vinayakam	Avinayak	A. vinayak
4.	Mrs.B.Swetha	Swetha	Swetha
5	Miss.T.Durga	Durga	Durga

Co-ordinator

HOD/ELE

HEAD
DEPARTMENT OF EEE
VEMU INSTITUTE OF TECHNOLOGY
P. KOTHAKOTA

VEMU INSTITUTE OF TECHNOLOGY

NEAR PAKALA, CHITTOOR-517112
(Approved by AICTE, New Delhi & Affiliated to JNTUA, Anantapuramu)

PARTICIPATION CERTIFICATE

This is to certify that Mr/~~Ms/Miss~~ P. Yerraiah of EEE department has actively participated in One day Workshop on **"Simulation of Basic Circuits using PSPICE"** on **07/05/2022** organized by Department of Electrical & Electronics Engineering, Vemu Institute of Technology, P.Kothakota.

Dr.A.Hema Sekhar
HOD-EEE

Dr. Naveen Kilari
Principal

VEMU INSTITUTE OF TECHNOLOGY

NEAR PAKALA, CHITTOOR-517112
(Approved by AICTE, New Delhi & Affiliated to JNTUA, Anantapuramu)

PARTICIPATION CERTIFICATE

This is to certify that ~~Mr/Ms~~/Miss T. Durga of EEE department has actively participated in One day Workshop on **"Simulation of Basic Circuits using PSPICE"** on **07/05/2022** organized by Department of Electrical & Electronics Engineering, Vemu Institute of Technology, P.Kothakota.

Dr.A.Hema Sekhar
HOD-EEE

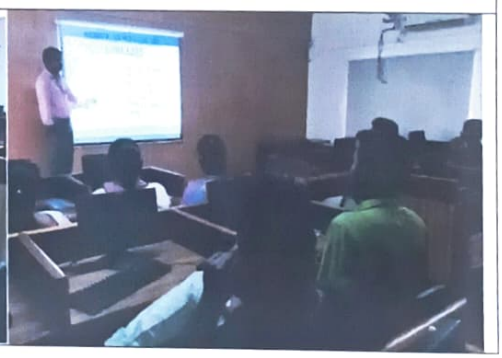
Dr. Naveen Kilari
Principal

VEMU INSTITUTE OF TECHNOLOGY:: P.KOTHAKOTA

Chittoor-Tirupathi National Highway, P Kothakota, Near Pakala, Chittoor (Dt), AP - 517112
(Approved by AICTE, New Delhi, Recognized Under 2(f) &12(B) of UGC Act Accredited by NAAC, NBA (EEE, ECE & CSE) & An ISO 9001 2015 Certified Institute)

Department of Electrical and Electronics Engineering

REPORT

Event Type	**Offline Staff Skill Development Program**
Title of the Event	**One day Workshop on "Simulation of Basic Circuits using PSPICE"**
Dates	**07/05/2022**
Keynote speaker	**Mr.S.Venkat Rao, Assistant Professor / EEE, Vemu institute of technology, P.Kothakota**
Faculty Coordinator	**Dr.S.Mallikarjunaiah**
Venue	**Simulation Lab**
Benefited Persons	**EEE Lab technicians (5 Participants)**
Description about the Event	The main objective of this Staff Skill Development Program is to provide the hands on training to EEE Lab technicians to simulate simple DC and AC circuits using PSPICE software. The full form of PSPICE is Personal Simulation Program with Integrated Circuit Emphasis. PSpice for TI is a design and simulation environment that helps evaluate functionality of analog circuits. This full-featured, design and simulation suite uses an analog analysis engine from Cadence. PSpice makes it easier to validate component yield and reliability, verify electrical performance, and optimize your designs. OrCAD is the general name for the software suite that helps you design PCBs. OrCAD capture is the tool within that OrCAD suite that lets you 'draw' schematics. PSpice is the name given to a basic circuit interface that simulates circuit behavior using SPICE. PSpice comes installed 'inside' OrCAD Capture. To use PSpice, start with the PSpice Schematics program. When you start up you will get a screen which looks like this. To put in a component, use the Draw drop-down menu, and select Get new part (or use the shortcut Ctrl-G). This will bring up a dialog box which will allow you to select pats from libraries.
Outcome	❖ Simulation of DC circuits, ❖ Simulation of AC circuits, ❖ Simulation of Series Resonance, ❖ Simulation of DC transient and ❖ Simulation of Parallel resonance.
Photos	 

Co-ordinator

HOD / EEE

HEAD
DEPARTMENT OF EEE
VEMU INSTITUTE OF TECHNOLOGY
P. KOTHAKOTA - 517 112