GUJARATTECHNOLOGICALUNIVERSITY

Program Name: Diploma Engineering

Level: Diploma

Branch: Electrical Engineering / Renewable Energy

Subject Code: DI04000331

Course/Subject Name: Electrical AutoCAD & Simulation

w.e.f. Academic Year:	2025-26
Semester:	4th
Category of the Course:	PCC

Prerequisite:	A basic understanding of electrical and electronic circuit fundamentals and computer operations is required
Rationale:	This course is designed to learn students with fundamental skills in computer-aided drawing and circuit simulation essential for electrical and electronic system design. The Computer Aided Drawing (CAD) and simulation (SCILAB, MATLAB, PSpice, MULTISIM, PSIM, PSCAD, ETAB) software emphasizes the ability to interpret, construct and analyze various circuits. By integrating theoretical understanding with hands-on practice, the course enables learners to achieve accuracy, efficiency, and innovation in design and testing, preparing them for the technology-driven industrial environment of modern electrical engineering.

Course Outcomes: After Completion of the Course, Student will be able to:

No	Course Outcomes	RBT Level*
01	Understand the simulation software to model, solve and simulate basic mathematical and engineering problems.	U,A
02	Simulate the basic electrical and electronic circuits using appropriate computer-based simulation software to verify performance and analyze system behavior.	U,A
03	Draw electrical and electronics circuit using CAD software	A

**Revised Bloom's Taxonomy (RBT)*

Teaching and Examination Scheme:

Teaching scheme (in Hours)			Total Credits L+T+(PR/2)	Assessment Pattern and Marks				Total Marks
				Theory		Tutorial/ Practical		
L	T	PR	C	ESE (E)	PA(M)	PA(I)	ESE (V)	
0	1	4	3	00	00	20	30	50

Course Content: (Tutorial discussions emphasize understanding and applying open-source simulation tools.)

Unit 1& 2 : SCILAB / MATLAB / PSIM / PSCAD/FALSTAD any of the software, UNIT 3: Electrical Autocad)

Unit No	Content	No of Hours	% of Weightage
1	**Unit 1 – Simulation and analysis of Electrical Engineering** • Introduction to simulation concepts and importance in engineering design. • Features of typical simulation environments (menus, components, libraries). • Representation of passive elements (R, L, C) and source modeling. • Building and simulating simple DC and AC circuits. • Series, parallel, and mixed circuits (R-L, R-C, R-L-C). • Steady-state and transient analysis of first-order circuits. • Voltage, current, and power measurement during simulation. • Simulation and programming idea in electrical machines and transformer. • Simulation and programming concept in electrical power system engineering. • Exporting plots, graphs, and simulation reports.	14	20
2	**Unit 2 Simulation and analysis of electronic circuits** • Overview of electronic circuit simulation tools and environments. • Representation and modeling of electronic circuits like diodes, BJTs, FETs, and OPAMPs. • Diode characteristics and rectifier circuits (half-wave, full-wave, bridge). • Clipper, clamper, and voltage regulator circuit simulation. • BJT and FET amplifier simulation — biasing and gain study. • RC-coupled and two-stage amplifier circuits. • Oscillator circuits — RC, Hartley, Colpitt, Wien Bridge, Crystal.	10	20%

	• OPAMP-based applications: inverting, non-inverting, integrator, differentiator, comparator. • Timer IC 555 - Astable and Monostable simulations. • Logic gate simulation (AND, OR, NOT, NAND, NOR, XOR, XNOR) • Verification of De Morgan's Theorem and universal gate operation • Sequential circuits: half adder, full adder, subtractor, flip-flop basics • Transient and frequency response analysis for electronic circuits • Comparative study of simulation outputs with expected theoretical results.		
3	**Unit 3 : Electrical Autocad** • Ribbon, Menu browser, AutoCAD Classic workspace: Title bar, Menu bar, Standard tool bar, Properties tool bar, Draw tool bar, Modify tool bar, Draw area, UCS • Command prompt window: Cross hair, Layout tabs, Function tabs • Keyboard Function keys: F1 to F12 • Screen Menu & Layout • Dimensions in AutoCAD • Coordinate system: Absolute and Polar • Draw command: Line, Polyline, Spline, X Line, Ray, Polygon, Rectangle, Circle, Arc, Ellipse, Hatch • Modify command: Erase, Trim, Extend, Stretch, Break, Fillet, Chamfer, Offset, Explode, Join, Move, Copy, Rotate, Mirror, Array • Drafting settings: Snap and grid, Polar tracking, Object snap, 3D object snap • 3D Modelling: Box, Sphere, Cylinder, Cone, Wedge, Torus, Pyramid, Psolid • Boolean operation: Union, Subtract, Intersect, Extrude, Slice • Plot and Print preview		

Suggested Specification Table with Marks (Theory):

Distribution of Theory Marks (in%)					
R Level	U Level	A Level	N Level	E Level	C Level
10 %	40 %	50 %	00	00	00

Where R: Remember; U: Understanding; A: Application, N: Analyze and E: Evaluate C: Create (as per Revised Bloom's Taxonomy)

References/Suggested Learning Resources:

(a) Reference Books:

1 Sandeep Nagar, Introduction to Scilab: For Engineers and Scientists. Apress publisher, New York, USA, 2017
2 . S. Nair, SCILAB: A Free Software to MATLAB, S. Chand Publishing, New Delhi, India,)12
3 Ramesh S. Gaonkar, Circuit Simulation and Analysis, 1st Edition, Technical Publications, Pune, 2018, ISBN-13: 9388971444-978
4 S. K. Bhattacharya, Fundamentals of Simulation Techniques, 2nd Edition, New Age International Publishers, New Delhi, 2013, ISBN-13: 8122431695-978
5 Gordon Birrell, Modeling and Simulation in Engineering, 1st Edition, CRC Press (Taylor & Francis Group), Boca Raton, 2019, ISBN-13: 0367-978264198
6 R. M. Mittal, Electronic Circuits: Principles, Design, and Simulation, 1st Edition, Tata McGraw-Hill Education, New Delhi, 2012, ISBN-13: .0070700185-978
7 Muhammad H. Rashid, Introduction to PSpice Using OrCAD for Circuits and Electronics, 4th Edition, PHI Learning Pvt. Ltd., New Delhi, 2017, ISBN-13: -978 8120341569

(b) Open-Source/CommercialSoftwareandWebsite:

1. https://www.mathworks.com/products/matlab.html — Official website for MATLAB, a leading platform for numerical computation, data analysis, and system simulation.
2. https://www.scilab.org — Official website for Scilab, an open-source software for numerical computation and simulation
3. https://powersimtech.com/products/psim/ — Official website for PSIM, used for power electronics, motor drive, and renewable energy system simulation.

4. https://www.pscad.com — Official website for PSCAD, a professional tool for electromagnetic transient simulations in power systems.
5. https://etap.com — Official website for ETAP, software for electrical power system modeling, analysis, and operation.
6. https://www.digsilent.de — Official website for DIgSILENT PowerFactory, used for advanced analysis and design of electrical networks.
7. https://www.ni.com/multisim — Official website for Multisim, a SPICE-based circuit simulation software for electronics design and education.
8. https://www.pspice.com — Official website for PSpice, a widely used simulation program for analog and digital circuit design and analysis.
9. https://www.analog.com/ltspice — Official website for LTspice, a free SPICE simulation software for analog circuit analysis.
10. https://octave.org — Official website for GNU Octave, an open-source platform compatible with MATLAB for numerical computation.
11. https://openmodelica.org — Official website for OpenModelica, an open-source environment for modeling, simulation, and optimization.
12. https://www.circuitlab.com — CircuitLab, an online schematic editor and circuit simulator for electronics learning and design.
13. https://www.tinkercad.com/circuits — TinkerCAD Circuits, a browser-based electronics simulator for students and beginners.
14. https://students.autodesk.com — Autodesk Student Portal, offering free educational licenses for AutoCAD and other Autodesk tools.
15. https://www.autodesk.com/products/autocad/ — Official website for AutoCAD, a commercial computer-aided design (CAD) and drafting software for 2D and 3D design.
16. https://www.vlab.co.in — Virtual Labs (IITs) platform for conducting simulated experiments in electrical, electronics, and control systems.

(c) Suggested Course Practical List: Each week includes two lab sessions (2 hours each)

Sr. No.	Practical Outcome/ Title of experiment	Unit/ CO	Approx. Hours required
1	Familiarization with simulation interface and tool navigation.	1	2
2	Simulate DC circuits and verify Ohm's and Kirchhoff's Laws.	1	2
3	Simulate AC circuits and observe waveform, current, and phase.	1	2
4	Analyze transient and steady-state response of RL and RC circuits.	1	2
5	Simulate RLC circuit and determine resonance frequency.	1	2
6	Simulate DC shunt motor performance and plot torque–speed characteristics	1	2
7	Simulate open-circuit and short-circuit tests of a single-phase transformer to determine efficiency and voltage regulation.	1	2
8	Simulation of single-line and three-line power system networks to observe current and voltage flow	1	2
9	Simulate diode characteristics and different rectifier circuits.	2	2
10	Simulate clipper and clamper circuits.	2	2
11	Simulate amplifier circuits and observe gain.	2	2
12	Simulate oscillators and measure output frequency.	2	2
13	Simulate OPAMP circuits (comparator, adder, integrator).	2	2
14	Simulate logic gates and half/full adder circuits.	2	2
15	Conduct frequency response analysis for an amplifier.	2	2
16	Draw electrical & electronics symbols using CAD	3	4
17	Draw 2D object using line, polyline, regular polygon, circle, rectangular and arc command.	3	4
18	Draw 2D object and modify it by using break, erase, trim, fillet, chamber, array and rotate command.	3	2
19	Draw parts of DC machines.	3	4

20	Draw single line diagram of transformer.	3	2
21	Draw stator and rotor of three phase induction motor by using array command.	3	2
21	Draw control and power wiring of DOL Starter.	3	4
22	Draw single line diagram of substation with incoming line and outgoing line	3	2
23	Draw single phase half wave, full wave and bridge rectifier circuit.	3	2
24	Draw 3D rotor and stator of induction motor	3	2

(d) Suggested Activities for Students:

1) Practice drawing and simulating basic electrical and electronic circuits using suitable software.
2) Compare theoretical and simulated results for different circuit types.
3) Perform parameter variation to observe its effect on circuit performance.
4) Analyze waveforms and responses for electrical and electronic circuits.
5) Design and simulate small practical circuits such as rectifiers, amplifiers, or logic gates.
6) Prepare a brief report including circuit diagrams, simulation results, and observations.

****