INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal - 500 043, Hyderabad, Telangana

COURSE CONTENT

AUTOCAD FREE ALTERNATIVES								
III Semester: CE								
Course Code	**Category**	**Hours / Week**			**Credits**	**Maximum Marks**		
		L	**T**	**P**	**C**	**CIA**	**SEE**	**Total**
ACEE09	**Skill**	0	0	2	1	40	60	100
Contact Classes: NIL	**Tutorial Classes: NIL**	**Practical Classes: 30**				**Total Classes: 30**		
Prerequisite: Fluid Mechanics								

I. COURSE OVERVIEW:

This course introduces students to free and open-source Computer-Aided Design (CAD) software as practical alternatives to commercial CAD applications. Students gain hands-on experience in 2D drafting, parametric 3D modelling, and engineering design using tools such as Libre CAD, Free CAD, and Opens CAD. The course also covers cloud-based collaborative design platforms, file interoperability, and engineering drawing standards to ensure industry relevance. Through practical exercises and mini-projects, students develop the ability to create accurate technical drawings and 3D models for engineering applications. The course equips students with cost-effective CAD skills that support higher studies, professional practice, and careers in design, manufacturing, construction, and product development.

II. COURSE OBJECTIVES:

The students will try to learn

I. The landscape of free and open-source CAD tools, their features, licensing, and suitability for professional and academic engineering drawing tasks.
II. 2D drafting techniques in Libre CAD including layers, object snapping, dimensioning, hatching, and block management for technical drawings.
III. Parametric 3D modelling using Free Cad′s Part Design and Part workbenches with sketch-based features, constraints, and assembly management.
IV. Programmatic/script-based 3D geometry creation using Opens Cad′s CSG modelling language for automated and version-controlled design.

III. COURSE OUTCOMES:

At the end of the course students should be able to:

CO1	Identify free and open-source CAD alternatives to AutoCAD and navigate their interfaces to set up a working 2D drafting environment.
CO2	Apply 2D drawing and editing commands in Libre CAD to produce accurate technical drawings with proper layers, dimensions, and annotations.
CO3	Develop parametric 3D solid models using Free Cad's Part Design workbench with constraints, sketches, and feature-based modelling.
CO4	Construct script-based 3D models using Opens Cad's declarative language for automated, version-controllable design workflows.
CO5	Design collaborative architectural and mechanical drawings using cloud-based CAD tools such as On shape and DraftSight Free.
CO6	Demonstrate export workflows, DXF/DWG interoperability, and print/publish layouts from free CAD tools for real-world deliverables.

II. COURSE SYLLABUS:

MODULE - I: INTRODUCTION TO FREE CAD ALTERNATIVES

Overview of AutoCAD and its cost constraints – Free and open-source CAD ecosystem – LibreCAD, FreeCAD, OpenSCAD, Onshape (free tier), – Installation and system requirements – Comparison of features and file format support – Introduction to DXF and DWG formats – Interface tour of LibreCAD: menu bar, toolbars, command line, layer list, and drawing canvas – Setting up drawing units, grid, and snaps.

MODULE - II: 2D DRAFTING WITH LIBRECAD

Drawing tools: Line, Circle, Arc, Ellipse, Polyline, Rectangle, Polygon, Spline – Editing tools: Move, Copy, Rotate, Scale, Mirror, Offset, Trim, Extend, Chamfer, Fillet – Layers: creating, assigning colours/line types, freezing and locking layers – Object Snapping and Ortho mode for precision – Dimensioning: linear, angular, radial, diameter, and ordinate – Text and hatching – Block definition and insertion – Printing and exporting drawings as PDF or DXF.

MODULE - III: 3D PARAMETRIC MODELLING WITH FREECAD

FreeCAD interface and workbenches – Part Design workbench: creating a new body, Sketcher: constraints (geometric and dimensional), Pad, Pocket, Revolution, Groove, Chamfer, Fillet, Draft – Part workbench: Boolean operations (Union, Cut, Intersection) – Assembly workbench basics: importing parts, applying coincident and axial constraints – Generating 2D technical drawing sheets from 3D models using the TechDraw workbench – Exporting models as STEP, STL, and DXF.

MODULE - IV: SCRIPT-BASED DESIGN WITH OPENSCAD

OpenSCAD environment: editor, preview, and render windows – Primitive 3D shapes: cube, sphere, cylinder, polyhedron – Transformations: translate, rotate, scale, mirror – CSG operations: union(), difference(), intersection() – Variables, modules, and loops for parametric design – Customizer for parameter-driven designs – Exporting STL for 3D printing – Practical examples: nuts, bolts, brackets, enclosures.

MODULE - V: CLOUD CAD AND INTEROPERABILITY

Introduction to Onshape: account setup, Part Studios, Assemblies, and Drawings – Collaborative design features: branching, version history, real-time multi-user editing – DXF/DWG import and export in LibreCAD, FreeCAD, and Onshape – Converting between CAD formats: STL, STEP, IGES, SVG – Print layout and title block design – Comparing free tools for specific use-cases: architectural, mechanical, electrical.

III. TEXT BOOKS:

1. Krister Trangell, ″LibreCAD User Manual″, LibreCAD Community, 1st edition, 2023.
2. Brad Collette, ″FreeCAD How-To″, Packt Publishing, 1st edition, 2012.

IV. REFERENCE BOOKS:

1. Jochen Hanselmann, ″OpenSCAD for 3D Printing″, Maker Media, 1st edition, 2015.
2. Sandeep Dogra, ″FreeCAD 0.19 Black Book″, CADCAM Works, 2021.
3. Tim Rustige, ″FreeCAD for Beginners″, Independently Published, 2020.
4. Rick Ellis, ″CAD Manager′s Guide to Understanding Free CAD Alternatives″, Cadapult Press, 2019.

V. ELECTRONICS RESOURCES:

1. https://librecad.org/
2. https://www.freecad.org/
3. https://openscad.org/
4. https://www.onshape.com/

AUTOCAD FREE ALTERNATIVES

1. Introduction to Free CAD Alternatives

1.1 Overview of AutoCAD and Free CAD Ecosystem

AutoCAD is a commercial Computer-Aided Design (CAD) software developed by Autodesk. It is widely used in architecture, engineering, and construction for 2D drafting and 3D modelling. However, its high licensing cost makes it inaccessible for many students and small organizations. Free and open-source CAD alternatives provide similar functionality without the financial burden.

Popular Free CAD Tools

LibreCAD – A free, open-source 2D CAD application that reads DWG and DXF formats. Ideal for technical drawings, schematics, and laser cutting designs.

FreeCAD – A parametric 3D modeller with support for 2D drafting, finite element analysis, and CAM. Suitable for mechanical engineering and product design.

OpenSCAD – A script-based 3D CAD modeller using a programming language to create solid 3D objects. Ideal for parametric designs and 3D printing.

Onshape – A cloud-based CAD platform with real-time collaboration features. Offers a free tier for public projects.

Installation Requirements

Each tool has specific system requirements. LibreCAD runs on Windows, macOS, and Linux with minimal resources. FreeCAD requires more RAM (4GB+) for complex assemblies. OpenSCAD is lightweight and cross-platform. Onshape requires only a modern web browser.

File Format Support

DXF (Drawing Exchange Format) is the standard interchange format for 2D CAD. DWG is AutoCAD's native format. Most free tools support DXF import/export. STEP and STL formats are used for 3D model exchange, especially for 3D printing.

2. 2D Drafting with LibreCAD

2.1 LibreCAD Interface and Drawing Tools

LibreCAD provides a clean interface with menu bar, toolbars, command line, and drawing canvas. The command line accepts direct input for precise drawing. Key drawing tools include Line, Circle, Arc, Ellipse, Polyline, Rectangle, Polygon, and Spline.

Editing Tools

Move, Copy, Rotate, Scale, Mirror, Offset, Trim, Extend, Chamfer, and Fillet commands allow modification of drawn entities. Object Snap ensures precision by snapping to endpoints, midpoints, centers, and intersections.

Layers and Organization

Layers organize drawing elements by type (walls, dimensions, text, etc.). Each layer can have specific color, linetype, and lineweight. Layers can be frozen (hidden) or locked (non-editable) for better workflow management.

Dimensioning and Annotation
LibreCAD supports linear, angular, radial, diameter, and ordinate dimensions. Text annotations can be added with customizable fonts and sizes. Hatching patterns indicate materials or cross-sections.

3. 3D Parametric Modelling with FreeCAD

3.1 FreeCAD Workbenches and Part Design

FreeCAD uses a workbench-based interface where each workbench provides tools for specific tasks. The Part Design workbench is used for creating parametric solid models through sketching and feature-based operations.

Sketcher and Constraints
The Sketcher workbench allows creation of 2D profiles with geometric constraints (horizontal, vertical, parallel, perpendicular, tangent) and dimensional constraints (distances, angles). Fully constrained sketches are required for parametric modelling.

Feature-Based Modelling
Pad extrudes a sketch to create a solid. Pocket removes material. Revolution creates rotational features. Groove cuts rotational profiles. Chamfer and Fillet add beveled or rounded edges. All features are parametric and can be edited later.

Assembly and Technical Drawings
The Assembly workbench allows combining multiple parts with constraints (coincident, axial, planar). The TechDraw workbench generates 2D technical drawings from 3D models with orthographic projections, sections, and dimensions.

4. Script-Based Design with OpenSCAD

4.1 OpenSCAD Programming Environment

OpenSCAD uses a script-based approach where 3D models are defined using a programming language. The interface consists of an editor, preview window, and render window. Scripts can be saved, version-controlled, and shared.

Primitive Shapes and Transformations
Basic 3D primitives include cube(), sphere(), cylinder(), and polyhedron(). Transformations such as translate(), rotate(), scale(), and mirror() position and orient these shapes in 3D space.

CSG Operations
Constructive Solid Geometry (CSG) operations combine primitives: union() merges shapes, difference() subtracts one shape from another, and intersection() keeps only overlapping regions. These operations create complex geometry from simple building blocks.

Parametric Design with Variables and Modules
Variables store dimensions and parameters. Modules encapsulate reusable geometry. Loops (for, each) create patterns. The Customizer interface allows parameter adjustment without editing code. Models export as STL for 3D printing.

5. Cloud CAD and Interoperability

5.1 Onshape Cloud Platform

Onshape is a cloud-based CAD platform that runs in web browsers. It provides Part Studios for modelling, Assemblies for combining parts, and Drawings for 2D documentation. All data is stored in the cloud with automatic version control.

Collaborative Design Features

Onshape supports real-time multi-user editing where team members can work simultaneously. Branching allows parallel development. Version history tracks all changes. The free tier requires projects to be public.

File Format Conversion

DXF and DWG formats enable 2D data exchange between CAD systems. STL is the standard for 3D printing. STEP and IGES are neutral formats for 3D model exchange. SVG exports 2D vector graphics. Each tool supports different formats.

Print Layouts and Publishing

Technical drawings require title blocks with project information, scale, date, and designer name. Print layouts arrange multiple views on paper sizes (A4, A3, etc.). PDF export creates portable documentation. Each free CAD tool provides publishing workflows.

Tool Selection for Specific Use Cases

LibreCAD suits 2D architectural plans and laser cutting. FreeCAD excels at mechanical engineering and product design. OpenSCAD is ideal for parametric objects and 3D printing. Onshape enables team collaboration on complex assemblies. Choose based on project requirements.