

CATIA V5- Fundamental & Structure Analysis	
Course Code: COEAMS - T09 Contact Hours Per Week: L-5 T-0 P- 10	Duration: 20 Hrs Course Category: Training

Introduction: The basic aim of this Training is to impart skills on CATIA V5 Design Software being used in Industry, so that students get the required knowledge and learn technique to implement design projects in time bound manner.

Course Objectives:

The course is designed for students, Working professionals and faculty members to achieve following objectives:

- To learn the interface of CATIA V5 cloud and/or site license software;
- To learn developing digital model of a hardware component and Assembly;
- To learn generating Engineering Drawings;
- To learn designing for static loading using structure analysis tools in the software.

Pre-Requisites:

Nil Course

Outcomes:

Having successfully completed this course, the student will be able to:

- Work on Sketcher Module and develop sketch sheet;
- Develop Part structures and make digital twin of a hardware component;
- Develop digital twin of Assemblies;
- Create Engineering Drawings for Shop-floor;
- Perform Stress Analysis and generate result sheets for static loading.
- Work on design projects and generate reports.

Pedagogy: The Lab sessions involve hybrid teaching by faculty using white board / PowerPoint slides, followed by demonstration on software. A cloud license of software is provided to students for the duration of the course and the teacher notes and tutorials are shared. Demonstration by teacher is followed by hands on practice by students on their system / lab computers under guidance of Teacher. The students attend the classes on Saturdays, practice the whole week and submit

weekly Assignment. At the end of the training students submit their project and a conference / journal paper for the same.

Contents:

Unit 1	8 hours
	<p>Introduction : Introduction to CAD, Introduction to CAM, Introduction to CAE , Hands-on on CATIA- V5, R21.</p> <p>Introduction to the CATIA V5: Modelling Process, About CATIA V5, CATIA V5 Basic Modelling Process, Starting CATIA V5, CATIA Interface, Windows Philosophy, Introduction to V5 Documents, The Workbench Concept, CATIA User Interface, Workbenches, Menus and Toolbars, Finding Tools, The Specification Tree, Manipulating the Specification Tree, Selecting Objects with the Mouse, The Object/Action and Action/Object Approaches, Using the CATIA Dialog Boxes, Using Dialog Boxes and Right-click, Moving Objects with the Mouse, Compass, Graphic Properties.</p> <p>Sketcher: Sketch Definition, Create a New Part , Part Design Workbench , Select Appropriate Sketch Support, Reference Planes, Sketch Support, Create Sketched Geometry, Basic Sketching, Positioned Sketching, Grid.</p> <p>Create Sketched Geometry: Geometry Creation, Lines, Polyline, Construction Geometry, Constraining the Sketch, Geometric and Dimensional Constraints, Create: Points, Circles, Ellipse, Parabola, Hyperbola, Spline, Conics, Pre-defined Profiles, User-defined Profiles, Operation tool bar. Constrain the Sketch: Geometric and Dimensional Constraints, Fully-Constrained Sketches, Create the Pad Feature, completing the Feature, Using a Pad to Create the First Feature, Save and Close the Document, Saving Documents- Saving a Document with a New Name, Closing a Document. Sketcher Transformation Tools: Mirror and Symmetry Options, Translation, Rotation, Scale, Offset, propagation Modes, Offset, Project 3D Elements, 3D Geometry Elements, Isolate Projected Elements, Sketch Analysis, Sketch Analysis Window, Performing a Quick Geometry Diagnosis. Exercises.</p>
Unit 2	10 hours
	<p>Part Design: Determine a Suitable Base Feature, Part Design Terminology, Creating a Base Feature, Selecting a Base Feature, Features that Add or Remove Material, Create Pad and Pocket Features Creating Pads, Creating a Simple Pocket, Pad and Pocket Limits, Restrictions for Pad/Pocket Profile , Sketches, Open Profiles.</p> <p>Shaft and Groove Features: Creating an Axis, Dimensioning to an Axis, Revolved Features, Axis of Revolution, Shafts, Creating Grooves, Restrictions for Revolved Features, Shell the Model. Create Holes: Create Fillets and Chamfers, Create Feature Profiles and Axis system, Additional Sketcher Tools, Sketcher Re-limitation Tools, Sketcher Re-limitation Tools, Re-limitations, Trim Options, Quick Trim Options.</p>

Unit 3	6 hours
	<p>Introduction to Generative Drafting: General Process, The Drawing Environment, Start a New Drawing, Setting the Drawing Sheet Format and Drafting Standards, Starting a Drawing with a Blank Sheet, Sheet Properties, Drawing Title Blocks, Introduction to 2D Catalogue, Inserting Catalogue Item.</p> <p>Create Views: Types of Views, Create a Front View, Using the Compass. Projection Plane - Add Projection Views, Add an Isometric View, Generating views using the View Wizard, Create Additional Views, Section Views and Section Cut, Add a Simple Section View on a Drawing, Create a Section View Using a 3D Profile Definition, Add a Detail View, Create a Clipping View, Creating a Broken View, Create a Breakout View, Add an Auxiliary View.</p> <p>Exercises.</p>

Books / Help / Notes	
1.	Faculty Notes
2.	Online Help

For Registration and further Details visit:

<https://sites.google.com/view/advanced-mechatronics/home>