CATIAv5

General Course Outline

➢ 1.Introduction to the Interface Lecture on CATIA

2D Sketching

2D Modifications of sketch

Constraint tool bar

Dimensioning and Editing dimensions

Profile tool bar

Constraint tool bar

Modify tool bar

Sketch tool bar

Visualize tool bar

Measure tool bar

Filters

Standard tool bar

View tool bar

Inspection of Sketches or Profiles

2. Part Modeling

Use of Modeling Tree

Selection of planes

Datum plane creations

Basic steps for creating profiles in part Environment

What is modeling?

Sketch based features

Dress up features

Transformation features

Reference tool bar

Boolean operations

Advanced dress up features

Surface based features

Material application to the models

Rendering

View tool bar icons

Annotations

3. Assembly Design

Importing of models in assembly module

Place a base component

Constraint tool bar

Move tool bar

Snap tool use

BOM preparation

Balloon numbering

Reuse pattern tool application

Display of after material application

Update tool

Inspection of assembly using clashing tool

4. Drafting

Drawing fundamentals

Adding items to drawing

Creating and placing drawing views

Working with major view types
Projection views
Auxiliary views
Offset section views
Detailed views
Clipping views
Dimensioning
Controlling the model views
2D drafting
Creating dimensions
Drawing notes
Geometric Tolerances
Welding Symbol applications
Surface roughness symbols applications
Formate creating
BOM

➤ 5. Sheet Metal

Introduction to sheet metal
Sheet metal operations
Editing the Sheet and Tool Parameters
Modifying the Bend Extremities
Computing the Bend Allowance
Walls tool bar
Bends tool bar
Transformation features
Stamping/Cutting tool bar
Reference elements tool bar
Creating a Hooper
Folded/Unfolded views

➤ 6. Surface and wireframe Design

Select Toolbar
Wireframe Toolbar
Surfaces Toolbar
Operations Toolbar
Replication Toolbar
Generic Tools Toolbar
Developed Shapes Toolbar
Measure Toolbar