CETPA INFOTECH PVT. LTD.

CURRICULUM OF CREO (PRO/E)

Creo (Pro/E) Foundation Training

INTRODUCTION OF DESIGN CONCEPT AND PROCEDURE

- Detailed Concept Of Cad
- Need & Importance Of Cad
- Overview About Actual Designing In Industries, Fundamentals Of Design And Its Implementation Methods
- All Characteristics Of Creo (Pro/E) To User Friendly Atmosphere
- Superiority Of Creo (Pro/E) With Its Use And Demand In Industries

INTERFACE WITH GUI

- Menu Manager
- View Toolbar
- Controlling The View
- Model Display
- Datum Display Toolbar
- Working With Document
- File Tools
- System Colors
- Selecting The Entities
- Working With Mouse Button
- Selecting The Working Directory For Saving The Document
- Model Tree
- Pro-E Help Option
- Document In Session
- Set The Parameter

SKETCHER

- Sketcher Diagnostics Tools
- Creating The Vector Shapes
- Working With Grids
- Snap Mode
- Creating The Coordinate
- Creating Spline And Its Geometry Control
- Display Dimension
- Work With Weak And Strong Dimension
- Edit Definition
- Creating Axis For Reference
- Work With Geometrical And Dimensional Constraint
- Insert Design From Palette
- Creating Text
- Import Data From Dxf. File
- Deleting And Trimming The Sketch Entities
- Analyze The Sketch For Opening Edges
- Modifying The Design

Creo (Pro/E) Foundation Training

BASE FEATUES OF PART DESIGNING

- Creating Solid Geometry
- Selecting The Part Environment
- Selecting Datum Planes (Top, Front, Right)
- Creation Of Sketch For Solid Modeling
- Converting An Area Into Volume Using Extrude Features
- Define The Limits Of Extrusion And Its Controls
- Remove The Cavity From The Solid Part
- Creating Revolute Design
- Cut Out Part By Revolute
- Sweep Feature
  - Protrusion
  - Thin Protrusion
  - Cut Sweep
  - Surfaces
- Blend Features
  - Parallel Blend
  - Rotational Blend
  - Blend Surfaces
- Create Solid Profile Using Swept Blend Features
- Creating Spring By Helical Sweep
- Boundary Blend
- Creating Axis And Points
- Generate New Datum Planes
- Creating Datum Curves

EDITING FEATURES OF PART DESIGNING

- Mirroring Features
- Moving Features
- Suppressing Features
- Copying Features
- Deleting Features
- Patterning Features

ENGINEERING FEATURES OF PART DESIGNING

- Create Draft
  - Constant Angle Draft
  - Variable Angle Draft
- Creating Drill Hole
- Rib Features
- Create A Shell Body
  Creating Round And Corners

SETUP & UTILITIES

- Select The Material
- Specify The Accuracy
- Mass Properties
- Setting The Pro/E Environment

ASSEMBLY MODELING

- Top Down Assembly
- Bottom Up Assembly
- Inserting The Component Into Assembly
- Create A New Component In Assembly
- Placing &Replacing The Component
- Defining Assembly Constraint
    - Define Mating Between Parts
    - Create Align Constraint
    - Define Tangency
    - Define Point On Line &Surface
    - Automatic Constraint
    - Fix The Part
- Create Multiple Copies Of Parts In Assembly
- Flexible Component
- Edit Constraint Definition
- Patterning
- Change The Order Of Component
- Deleting Component
- Explode The Assembly
  Creating Bill Of Material (BOM)

DRAFTING (DETAILING)

- Introduction Of Drafting
- Need & Importance Of Drafting
- Starting The Drafting Workbench
- Defining The Sheet & Sizes
- Adjusting Of Drawing Sheet According To Object/Assembly
- Types Of Projection
- Using Predefined Drafting Styles
- Scaling The Drafted View

GENERATIVE DRAFTING

- What Is Detailing?
- Setting Up And Configuring Drawings
- Using Drawing Tree, Layers, And Drawing Parameters
- Creating Drawings
- Adding Models To Drawings
- Creating The Drawing Layout
- Using Model Grids
- Inserting OLE Objects
- Importing Draft Data From External Applications
- Working With Drawing Tables
    - Creating Drawing Tables
    - Generate Report For Drawing
    - Using BOM Balloons
- Add Dimension In The Drawing
- Create A Detail View
- Working With Dimension Display

INTERACTIVE DRAFTING

- Sketching In Drawing Mode
- Drafting In Drawing Mode
- Working With Draft Cross Sections
- Relating Detail Objects
- Grouping Detail Objects
- Reviewing And Publishing Drawings
    - Reviewing Drawings
    - Comparing Drawings
    - Publishing Drawings
    - Exporting Drawings To A Pdf Format
    - Exporting Drawings To Other Formats
- Using Drawing Representations And Programs
    - Working With Drawing Representations
    - Creating Drawing Programs

DATA EXCHANGE

- Converting Files For Transferring
- Converting Into IGES, STEP,PARASOLID Etc.
- Convert Into Jpeg, Mpeg, Tiff, Pdf Files

SHEET-METAL DESIGNING

INTRODUCTION TO THE CREO SHEET-METAL DESIGN PROCESS

- Creo Sheet-Metal Design Process

SHEET-METAL MODEL FUNDAMENTALS

- Sheet-Metal Model Fundamentals
- Understanding Developed Length
- Creating a New Sheet-Metal Part in Assembly Mode
- Creating a New Sheet-Metal Model in Part Mode
- Converting a Solid Model to a Sheet-Metal Model

CREATING PRIMARY SHEET-METAL WALL FEATURES

- Understanding Sheet-Metal Wall Features
- Creating Flat Walls
- Extruded Sheet-Metal Wall Features
- Revolved Sheet-Metal Wall Features
- Sheet-Metal Wall Sketching Tools

CREATING SHEET-METAL SECONDARY WALL FEATURES

- Understanding Secondary Walls
- Creating Secondary Flat Walls
  Using Flange Walls
- Using Extruded Walls
- Wall Dashboard Options
- Understanding Relief

MODIFYING SHEET-METAL MODELS

- Bends
- Bend Options
- Unbend Features
- Bend Back Features
- Flat Pattern
- Sheet-Metal Cuts
- Die Form Features
- Punch Form Features
- Rip

SHEET-METAL BENDS AND SETTING UP THE SHEET-METAL ENVIRONMENT

- Order of Bend Features
- Fixed Geometry
- Flat States

SURFACE DESIGNING

SURFACE MODELING OVERVIEW

- Introduction to Surfacing
- Surface Modeling Uses
- Surface Modeling Paradigms
- Freeform Overview
- Blending Surface Modeling Paradigms
- Surfacing Terms

ADVANCED SELECTION

- Advanced Chain Selection
- Advanced Surface Selection

BASIC Surfacing Tools

- Creating Surface Extrude Features
- Creating Surface Revolve Features
- Creating Fill Surfaces
- Creating Sweep Surfaces with Open Trajectories
- Creating Parallel Blend Surfaces
- Creating General Blend Surfaces

HELICAL Sweeps

- Understanding Helical Sweeps Theory
- Utilizing Helical Sweeps for Surfaces

CREATING AND EDITING SOLIDS USING QUILTS

- Thickening Surface Quilts
- Solidifying Quilts to Add Material
- Solidifying Quilts to Remove Material
- Solidifying Quilts to Replace Material
- Offsetting Surfaces using the Replace Option

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