

ZENUS INFOTECH INDIA PVT. LTD.
CURRICULUM OF CREO/ Pro-E

Course Duration: 6/8 Weeks*

Basic Solid Part Modeling

- Understanding Creo Concepts
- Using the Creo Interface
- Selecting and Editing
- Creating Sketcher Geometry
- Using Sketcher Tools
- Creating Sketches for Features
- Creating Datum Features:
Planes
- Creating Extrudes, Revolves,
and Ribs
- Creating Sweeps and Blends
- Creating Holes, Shells, and
Draft
- Creating Rounds and Chamfers
- Copy and Mirror Tools
- Creating Patterns
- Measuring and Inspecting
Models

BASIC assembly Creation

- Assembling with Constraints
- Exploding Assemblies
- Using Layers
- Managing Design Intent
- Resolving Failures and Seeking
Help

Adv solid part modeling

- Advanced Datum Features

- Relations and Parameters
- Advanced Blends
- Variable Section Sweeps
- Swept Blends
- Family Tables
- Reusing Features

Flexible modeling

- Introduction to Flexible
Modeling
- Editing, Transformations &
Recognition in Flexible
Modeling

Adv Assembly Design

- Using Advanced Assembly
Constraints
- Creating and Using Component
Interfaces
- Creating and Using Flexible
Components
- Replacing Components in an
Assembly
- Understanding the Basics of
Simplified Reps
- Creating Cross-Sections,
Display Styles and Combined
Views
- Creating and Using Assembly
Structure and Skeletons

Sheetmetal Design

- Introduction to the Creo Sheetmetal Design Process
- Sheetmetal Model Fundamentals
- Creating Primary Sheetmetal Wall Features
- Creating Sheetmetal Secondary Wall Features
- Modifying Sheetmetal Models
- Sheetmetal Bends and Setting Up the Sheetmetal Environment

Detailing (Drawing)

- Introduction to Drawings

- Creating New Drawings
- Creating Drawing Views
- Adding Model Details to Drawings
- Adding Notes to Drawings
- Adding Tolerance Information
- Using Report Information in Drawings
- Using Layers in Drawings
- Creating and Using Tables in Drawings
- Adding Draft Geometry and Symbols