

CREO Syllabus

1: INTRODUCTION TO CREO PARAMETRIC

- Introduction to Creo Parametric
- Feature-Based Nature
- Bidirectional Associative Property
- Parametric Nature
- System Requirements
- Getting Started with Creo Parametric
- Important Terms and Definitions
- File Menu Options
- Managing Files
- Menu Manager
- Model Tree
- Understanding the Functions of the Mouse Buttons
- Ribbon
- Toolbars
- Navigator
- Creo Parametric Browser
- Appearance Gallery
- Rendering in Creo Parametric
- Colour Scheme Used in this Book

2: CREATING SKETCHES IN THE SKETCH MODE-I

- The Sketch Mode
 - Working with the Sketch Mode
 - Invoking the Sketch Mode
- The Sketcher Environment
- Working with a Sketch in the Sketch Mode
- Drawing a Sketch Using tools available in the Sketch Tab
 - Placing a Point
 - Drawing a Line
 - Drawing a Centreline
 - Drawing a Geometry Centreline
 - Drawing a Rectangle
 - Drawing a Circle
 - Drawing an Ellipse
 - Drawing an Arc
- Dimensioning the Sketch
 - Converting a Weak Dimension into a Strong Dimension
 - Dimensioning a Sketch Using the Normal Tool
- Dimensioning the Basic Sketched Entities
 - Linear Dimensioning of a Line

- Angular Dimensioning of an Arc
- Diameter Dimensioning Radial
- Dimensioning Dimensioning
- Revolved Sections
- Working with Constraints
 - Types of Constraints
 - Disabling Constraints
 - Modifying the Dimensions of a Sketch
 - Using the Modify Button
 - Modifying a Dimension by Double-Clicking on it
 - Modifying Dimensions Dynamically
- Resolve Sketch Dialog Box
- Deleting the Sketched Entities
- Trimming the Sketched Entities
- Mirroring the Sketched Entities
- Inserting Standard/User-Defined Sketches
- Drawing Display Options

3: CREATING SKETCHES IN THE SKETCH MODE-II

- Dimensioning the Sketch
 - Dimensioning a Sketch Using the Baseline Tool
 - Replacing the Dimensions of a Sketch Using the Replace Tool
- Creating Fillets
 - Creating Circular Fillets
 - Creating Elliptical Fillets
- Creating a Reference Coordinate System
- Working with Splines
 - Creating a Spline
 - Dimensioning of Splines
 - Modifying a Spline
- Writing Text in the Sketcher Environment
- Rotating and Resizing Entities
- Importing 2D Drawings in the Sketch Mode

4: CREATING BASE FEATURES

- Creating Base Features
- Invoking the Part Mode
- The Default Datum Planes
- Creating a Protrusion
 - Extruding a Sketch
 - Revolving a Sketch
- Understanding the Orientation of Datum Planes
- Parent-Child Relationship
 - Implicit Relationship

Explicit Relationship
Nesting of Sketches

5: DATUMS

Datums

Default Datum Planes Need for
Datums in Modeling Selection
Method in Creo Parametric Datum
Options

Datum Planes

Creating Datum Planes

Datum Planes Created On-The-Fly

Datum Axes

Datum Points

Creating Cuts

Removing Material by Using the Extrude Tool

Removing Material by Using the Revolve Tool

6: OPTIONS AIDING CONSTRUCTION OF PARTS-I

Options Aiding Construction of Parts

Creating Holes

The Hole Dashboard

Important Points to Remember While Creating a Hole

Creating Rounds

Creating Basic Rounds

Creating a Variable Radius Round

Points to Remember While Creating Rounds

Creating Chamfers

Corner Chamfer

Edge Chamfer

Understanding Ribs

Creating Trajectory Ribs

Creating Profile Ribs

Editing Features of a Model

Editing Definition or Redefining Features

Reordering Features

Rerouting Features

Suppressing Features

Deleting Features

Modifying Features

7: OPTIONS AIDING CONSTRUCTION OF PARTS-II

Introduction

Creating Feature Patterns

Uses of patterns

Creating Patterns

Deleting a Pattern

Copying Features

New Refs

Same Refs

Mirror

Move

Select

Mirroring a Geometry

Creating a Section of a Solid Model

Work Region Method

8: ADVANCED MODELING TOOLS-I

Other Protrusion Options

Sweep Features

Creating Sweep Protrusions

Aligning a Sketched Trajectory to an Existing Geometry

Creating a Thin Sweep Protrusion

Creating a Sweep Cut

Blend Features

Parallel Blend

Rotational Blend

General Blend

Using Blend Vertex

Shell Feature

Creating a Constant Thickness Shell

Creating a Variable Thickness Shell 8

Datum Curves

Creating a Datum Curve by Using the Curve Button

Creating a Datum Curve by Sketching

Creating a Curve by Using the Intersect Option

Creating a Curve by Using the Project Option

Creating a Curve by Using the Wrap Option

Creating Draft Features

9: ADVANCED MODELING TOOLS-II

Advanced Feature Creation Tools

Variable Section Sweep Using the Sweep Option

Swept Blend

Helical Sweep

Blend Section to Surfaces

Blend Between Surfaces

10: ADVANCED MODELING TOOLS-III

Advanced Modeling Tools

Toroidal Bend

Spinal Bend

Warp

Transform Tool

Warp Tool

Spine Tool

Stretch Tool

Bend Tool

Twist Tool

Sculpt Tool

11: ASSEMBLY MODELING

Assembly Modeling

Important Terms Related to the Assembly Mode

Top-down Approach

Bottom-up Approach

Placement Constraints

Package

Creating Top-down Assemblies

Creating Components in the Assembly Mode

Creating Bottom-up Assemblies

Inserting Components in an Assembly

Assembling Components

Displaying Components in a Separate Window

Displaying Components in the Same Window

3D Dragger

Applying Constraints

Status Area

Placement Tab

Move Tab

Packaging Components

Creating Simplified Representations

Redefining the Components of an Assembly

Reordering Components

Suppressing/Resuming Components

Replacing

Assembling Repeated Copies of a Component

Modifying the Components of an Assembly

Modifying Dimensions of a Feature of a Component

- Redefining a Feature of a Component
- Creating the Exploded State
 - References Tab Offset Tab
 - Explode Line Tab
- The Bill of Materials
- Global Interference
- Pairs Clearance

12: GENERATING, EDITING, AND MODIFYING THE DRAWING VIEWS

- The Drawing Mode
- Generating Drawing Views
 - Generating the General View
 - Generating the Projection View
 - Generating the Detailed View
 - Generating the Auxiliary View
 - Generating the Revolved Section View
 - Generating the Copy and Align View
 - Generating the 3D Cross-Section View
- Editing the Drawing Views
 - Moving the Drawing View
 - Erasing the Drawing View
 - Deleting the Drawing View
 - Adding New Parts or Assemblies to the Current Drawing
- Modifying the Drawing Views
 - Changing the View Type
 - Changing the View Scale
 - Reorienting the Views
 - Modifying the Cross-sections
 - Modifying Boundaries of Views
 - Adding or Removing the Cross-section Arrows
 - Modifying the Perspective Views
- Modifying Other Parameters
 - Editing the Cross-section Hatching

13: DIMENSIONING THE DRAWING VIEWS

- Dimensioning the Drawing Views
 - Show Model Annotations Dialog Box
- Adding Notes to the Drawing
- Adding Tolerances in the Drawing Views
 - Dimensional Tolerances
 - Geometric Tolerances
- Editing the Geometric Tolerances
- Adding Balloons to the Assembly Views
- Adding Reference Datums to the Drawing Views

Modifying and Editing Dimensions

Modifying the Dimensions Using the Dimension Properties Dialog Box

Modifying the Drawing Items Using the Shortcut Menu

Cleaning Up the Dimensions

14: OTHER DRAWING OPTIONS

Sketching in the Drawing Mode

Modifying the Sketched Entities

User-Defined Drawing Formats

Retrieving the User-Defined Formats in the Drawings

Adding and Removing Sheets in the Drawing

Creating Tables in the Drawing Mode

Generating the BOM and Balloons in Drawings

15: SURFACE MODELING

Surface Modeling

Creating Surfaces in Creo Parametric

Creating an Extruded Surface

Creating a Revolved Surface

Creating a Sweep Surface

Creating a Blended Surface

Creating a Swept Blend Surface

Creating a Helical Sweep Surface

Creating a Surface by Blending the Boundaries

Creating a Variable Section Sweep Surface Using the Sweep Tool

Creating Surfaces the Using the Style Environment of Creo Parametric
Style Dashboard

Surface Editing Tools Mirroring

the Surfaces Merging the

Surfaces Trimming the

Surfaces Creating the Fill

Surfaces Creating the Intersect

Curves Creating the Offset

Surfaces Adding Thickness to
a Surface

Converting a Surface into a Solid

Creating a Round at the Vertex of a Surface

Freestyle modelling environment

Freestyle Dashboard

16: WORKING WITH SHEET METAL COMPONENTS

Introduction to Sheet metal

Invoking the Sheet metal Mode

Introduction to Sheet metal Walls

- Creating the Planar Wall

- Creating the Unattached Revolve Wall

- Creating the Unattached Blend Wall

- Creating the Unattached Offset Wall

- Creating Reliefs in Sheet metal Components

- Creating a Flat Wall

- Creating a Twist Wall

- Creating an Extend Wall

- Creating a Flange Wall

- Creating the Bend Feature

- Creating the Unbend Feature

- Creating the Bend Back

- Conversion to Sheet metal Part

- Creating Cuts in the Sheet metal Components