



KLEIOS[®]
TECHNOLOGIES
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CREO TRAINING
COURSE SYLLABUS

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COURSE SYLLABUS

SL.NO	TOPICS
1	Introduction to Engineering Drawings
2	Creo Basic Parametric Concepts <ul style="list-style-type: none">• 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
3	Creating Sketch Geometry <ul style="list-style-type: none">• 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
- 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

	<ul style="list-style-type: none"> • 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 <p>CREATING BASE FEATURES</p> <ul style="list-style-type: none"> • 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
4	<p>Parts Creation Using Extrude, revolve And Ribs</p> <ul style="list-style-type: none"> • Creating a Protrusion • Extruding a Sketch • Revolving a Sketch • Creating Cuts • Removing Material by Using the Extrude Tool • Removing Material by Using the Revolve Tool • Understanding Ribs • Creating Trajectory Ribs • Creating Profile Ribs <p>OPTIONS AIDING CONSTRUCTION OF PARTS-II</p> <ul style="list-style-type: none"> • Introduction

	<ul style="list-style-type: none">• 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 <p>ADVANCED MODELING TOOLS-III</p> <ul style="list-style-type: none">• Advanced Modeling Tools• Toroidal Bend• Spinal Bend• Warp• Transform Tool• Warp Tool• Spine Tool• Stretch Tool• Bend Tool• Twist Tool• Sculpt Tool
5	Parts Creation Using Sweep and Blend Tool <ul style="list-style-type: none">• 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

	<ul style="list-style-type: none">• 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
6	Modifying Parts by Using Rounds, Chamfer and Drafts <ul style="list-style-type: none">• 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
7	Section Sweep, Helical Sweep & Swept Blends <ul style="list-style-type: none">• Advanced Feature Creation Tools• Variable Section Sweep Using the Sweep Option• Swept Blend• Helical Sweep• Blend Section to Surfaces

	<ul style="list-style-type: none">• Blend Between Surfaces
8	Creating Datum Features <ul style="list-style-type: none">• 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
9	Layers, Family Table and UDF
10	Assembling Using basic and Advance Constrains <ul style="list-style-type: none">• 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

	<ul style="list-style-type: none">• 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
11	Exploding Assemblies <ul style="list-style-type: none">• Creating the Exploded State• References Tab Offset Tab• Explode Line Tab• The Bill of Materials• Global Interference• Pairs Clearance
12	Surface Features <ul style="list-style-type: none">• Surface Modelling• 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

	<ul style="list-style-type: none">• 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
13	Creating Drawing views and Details <ul style="list-style-type: none">• 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

	<p>DIMENSIONING THE DRAWING VIEWS</p> <ul style="list-style-type: none"> • 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 <p>OTHER DRAWING OPTIONS</p> <ul style="list-style-type: none"> • 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
14	Component Interfaces
15	Flexible Components
16	Using Assembly Features and Shrink-wrap
17	Creating Skeletons
18	<p>Sheetmetal Design</p> <ul style="list-style-type: none"> • 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

	<ul style="list-style-type: none">• 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
19	Display Styles and Views
20	Basic Geometric Dimensioning and Tolerancing