

CREO

INTRODUCTION & UNDERSTANDING OF PARAMETRIC CONCEPT

- Creo Parametric Basic Modeling Process
 - Understanding of Parametric, Feature based and Associative Concepts
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SELECTING AND EDITING OF GEOMETRY, FEATURE AND MODELS

- Understanding Creo Parametric Basic Controls
 - Understanding of Creo Interface
 - Define Sketcher for Feature
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CREATING EXTRUDE, REVOLVE, RIB, BLEND, SWEEP AND DATUM FEATURE

- Extrude as Solid and Solid Cut
 - Extrude as Thicken Solid and Thicken Cut
 - Revolve as Solid & Solid Cut
 - Revolve as Thin Solid & Thin Solid Cut
 - Profile Rib
 - Trajectory Rib
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CREATING SWEEP BLEND, SWEEP AND DATUM FEATURE

- Create Sweep as a Solid and Solid Cut
 - Create Sweep as a Thin Solid and Thin Solid Cut
 - Creating Sweep Blend as a Solid Feature and Solid Cut
 - Creating Sweep Blend as a Thin Solid Feature and Thin Solid Cut
 - Create Datum Plane
 - Create Axis, Point and Co-ordinate
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CREATING ROUND, CHAMFER, HOLE, SHELL AND DRAFT

- Hole
- Shell
- Draft
- Round
- Chamfer

CREATING SHAPE BLEND AND HELICAL SWEEP

- Creating Parallel Shape Blend as a Solid Feature and Solid Cut
- Creating Parallel Shape Blend as a Thin Solid Feature and Thin Solid Cut
- Creating Revolve Shape Blend as a Solid Feature and Revolve Shape Blend Solid Cut
- Creating Thin Revolve Shape Blend as a Solid Feature and Thin Revolve Solid Cut
- Creating Helical Solid Feature and Cut
- Creating Thin Helical Solid Feature and Cut

CREATING PARTTEN, FAMILY, TABLE AND MIRROR

- Creating Pattern Feature by with Dimension, Along Direction, About Axis, Along Defined Curve, Fill
- Area, Table Pattern, and With Defined Point
- Family Table
- Mirror

MANAGING DESIGN INTENT & RESOLVING FEATURE OF MODEL, CPY, PASTE, MODELINSPECTION, LAYER MANAGEMENT AND UDF

- Managing Design Intent and Resolving Failure of Model
- Model Measurement & Inspection, Draft analysis
- UDF(user defined feature), Layer Management

ASSEMBLY WITH CONSTRAINTS

- Define Constrain on Planar and Circular Face
 - Define Constrain along Edge, Vertices and Axis
 - Calculate Mass Properties and CG of Assembly
 - Creating Component in Assembly
 - Adding and Subtracting Volume in Assembly
 - Set Explode View in Assembly
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CREATING NEW DRAWINGS, DRAWING VIEW, ADDING MODEL DETAILS, SYMBOL AND TOLERANCE, CREATE BILL OF MATERIAL FOR PRODUCT, SHEET FORMAT, CREATE SHEET TEMPLATE FOR DRAWING

- Creating Drawing View, Set Drawing Standard and Imposing Dimension
 - Create Drawing Views and Format Sheet, Editing of Drawing Views
 - Section Views
 - Dimension and Tolerances and Standard Symbol
 - Explode Views, Unexplode Views, Bill of Material, Table and Notes
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