

PRO-E

Curriculum

Introduction To Pro/E Wildfire.

- Introduction To Pro/E Interface Feature Based Nature
- File Menu Options
- Model Tree Toolbars And Navigators
- Appearance And Rendering

Creating Sketches In Sketch Mode

- The Sketching Environment
- Drawing A Sketch Using Line ,Rectangle ,Circle ,Arc
- Dimensioning The Sketched Entities
- Working With Constraints
- Creating Fillets

Creating Base Features

- Invoking The Part Mode
- The Default Datum Planes
- Extruding And Revolving The Sketch
- Parent Child Relationship

Understanding the Concept of Datums

- Needs For Datums IN Modeling
- Creating Datum Planes ,Datum Axis ,Datum Points
- Removing Material By Extruding And Revolving

Options Aiding Construction of Parts

- Creating Holes ,Rounds And Chamfers
- Understanding Ribs And Patterns
- Mirroring A Geometry
- Creating A Section Of Solid Model

Advanced Modeling Tools

- Sweep ,Shell And Blend Options
- Datum Curves
- Toroidal And Spinal Blend
- Warp And Wrap Buttons

Assembly Modeling

- Important Terms Related To Assembly Modeling
- Creating Top Down And Bottom Up Assembly

- Adding Simplified Representations
- Reordering Components

Generating, Modifying and Dimensioning the Drawing Views

- Generating The General Views
- Editing And Modifying The Drawing Views
- Adding Notes And Tolerances
- Sketching In The Drawing Mode

Surface Modeling

- Creating A Surface By Extruding
- Creating A Surface By Revolving
- Creating Blend And Sweep Surfaces

Working With Sheetmetal Components

- Invoking The Sheetmetal Mode
- Introduction To Sheetmetal Walls
- Creating A Flat Wall



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