*******Creo - (Pro/ENGINEER)*******

- Pro/ENGINEER concepts
- Using the Pro/ENGINEER interface
- Creating sketcher geometry
- Creating extrudes, revolves, and ribs
- Selecting and editing
- Creating datum features
- Utilizing internal sketches and embedded datums
- Creating sweeps and blends
- Creating holes and shells
- Creating rounds, chamfers and Drafts
- Variable section sweeps, Helical sweeps and Swept blends
- Creating patterns
- Group, copy, and mirror tools
- Measuring and inspecting models
- Advanced reference management
- Relations and parameters
- Layers, Family tables & UDF
- Assembling with constraints
- Exploding assemblies
- Creating surface features
- Editing surface features in Pro/ENGINEER
- Creating drawing views
- Creating drawing details
- Using advanced assembly constraints
- Creating and using component interfaces
- Creating and using flexible components
- Using assembly features and shrinkwrap

- Replacing components in an assembly
- Understanding simplified reps
- Creating cross-sections, display styles, and combined views
- Substituting components by rep, envelope, and model
- Creating and using assembly structure and skeletons
- Introduction to sheet metal design
- Primary walls, Secondary and unattached walls
- Unbend, bend back and cuts
- Notches and punches
- Sheet metal forms
- Bending & Unbending sheet metal geometry
- Converting solid parts
- Sheet metal drawings with flat states and bend order table
- Real time rendering