Plastic Part Design using Creo Parametric

Target Audience

Mechanical engineers, product design engineers, and CAD designers involved in plastic product development who want to learn industry-standard plastic design practices using Creo Parametric.

Course Objectives

- Understand plastic product design principles.
- Learn Creo Parametric tools used in plastic part modeling.
- Apply plastic design rules such as draft, ribs, bosses, and snap fits.
- Design manufacturable plastic components.
- Validate plastic designs using draft and thickness analysis.

Course Outcomes

By the end of the training, participants will be able to:

- Design plastic components using Creo Parametric.
- Apply industry design rules for injection molded parts.
- Create ribs, bosses, snap fits, and mounting features.
- Perform draft analysis and thickness validation.
- Develop complete plastic enclosures ready for manufacturing review.

Course Outline (40 Hours)

The training consists of theory and hands-on labs divided into 10 chapters.

Chapter 1: Introduction to Plastic Product Design

- Overview of plastic products in industry
- Differences between plastic and metal design
- Plastic product development workflow
- Introduction to Creo Parametric interface
- Plastic component examples

Chapter 2: Plastic Materials and Design Considerations

- Thermoplastics vs thermosets
- Common engineering plastics (ABS, PP, PC, Nylon)
- Material properties affecting design
- Shrinkage considerations
- Material selection guidelines

Chapter 3: Plastic Design Rules

- Importance of uniform wall thickness
- Draft angle concept and guidelines
- Fillets and corner radii
- Sink marks and warpage causes
- Basic manufacturability considerations

Chapter 4: Creo Parametric Fundamentals for Plastic Modeling

- Sketching tools and constraints
- Feature creation (Extrude, Revolve, Hole)
- Reference geometry
- Feature editing and modification
- Model tree and parametric design concepts

Chapter 5: Draft and Shell Features

- Understanding mold opening direction
- Applying draft features
- Using draft analysis tools
- Shell feature for hollow parts
- Design validation using draft analysis

Chapter 6: Rib Design

- Purpose of ribs in plastic parts
- Rib thickness and height rules
- Rib spacing guidelines
- Creating ribs in Creo

- Strengthening structures with ribs

Chapter 7: Boss Design

- Screw bosses and mounting bosses
- Boss diameter and thickness rules
- Reinforcing bosses with ribs
- Avoiding sink marks near bosses
- Creating bosses in Creo

Chapter 8: Snap Fits and Assembly Features

- Types of snap fits
- Cantilever snap fit design
- Annular snap fits
- Clip features
- Designing snap fits in Creo

Chapter 9: Plastic Product Case Study

- Designing a plastic enclosure
- Split housing design
- Mounting features
- Assembly considerations
- Interference checking

Chapter 10: Validation and Final Project

- Draft analysis
- Thickness analysis
- Design review and optimization
- Preparing models for manufacturing
- Final plastic product design project