

Creo Parametric 5.0 for SolidWorks Users

Agenda

Day 1

Module 1 Introduction to the Creo Parametric Basic Modeling Process

Module 2 Using the Creo Parametric Interface

Module 3 Editing Geometry, Features, and Models

Module 4 Sketcher Geometry and Tools

Day 2

Module 5 Creating Extrudes, Revolves, and Ribs

Module 6 Creating Sweeps and Blends

Module 7 Creating Holes, Shells, and Draft

Module 8 Creating Rounds and Chamfers

Day 3

Module 9 Project I

Module 10 Assembling with Constraints

Module 11 Introduction to Drawings

Module 12 Resolving Failures and Seeking Help