

Introduction to Creo Direct 4.0

Overview

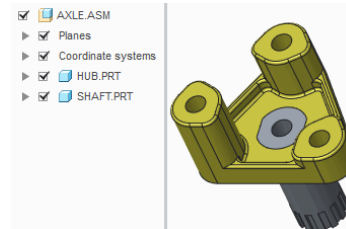
Course Code **TRN-5110-T**

Course Length **8 Hours**

In this course, you will learn direct modeling using Creo Direct 4.0. You will become familiar with Creo Direct's interface and approach to direct modeling, including how to quickly create sketches with precision, transform sketches into 3-D shapes, and directly manipulate existing geometry with ease. You will also learn how to assemble and reposition components in an assembly, as well as use existing geometry in an assembly to create part geometry.

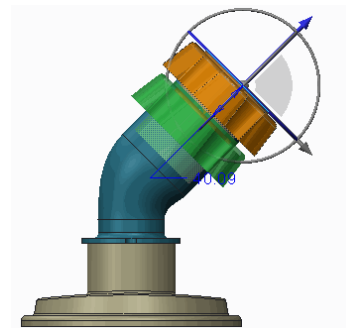
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Direct 4.0.



Course Objectives

- Understand the Direct Modeling approach to creating geometry
- Sketch precise 2-D geometry without constraints
- Create 3-D shapes from the 2-D geometry by extruding, revolving, and sweeping sketches and sketch regions
- Create datum axes and datum planes
- Create engineering geometry, including holes, rounds, chamfers, draft, and shells
- Pattern geometry in one and two directions
- Use measure tools and create part and assembly cross-sections
- Rapidly select and directly modify 3-D shapes in your models
- Understand the direct approach to creating assemblies
- Edit component position in an assembly
- Create new components and design components in the context of an assembly



Prerequisites

- Introduction to Creo Parametric or equivalent experience

Audience

- This course is intended for design engineers, mechanical designers, and industrial designers. People in related roles will also benefit from taking this course.
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Agenda

Day 1

Module	1	Introduction to Creo Direct
Module	2	Creating Sketches in 2-D Mode
Module	3	Creating Geometry Shapes
Module	4	Creating Datums
Module	5	Creating Engineering Geometry
Module	6	Measurements and Sections
Module	7	Editing 3-D Geometry
Module	8	Working with Assemblies

Course Content

Module 1. Introduction to Creo Direct

- i. Understanding Direct Modeling
- ii. Understanding the User Interface
- iii. Understanding the Mini Toolbar
- iv. Opening and Creating a Creo Direct Model
- v. Orienting and Positioning the Model
- vi. Understanding Selection Behavior
- vii. Using Marquee Selection
- viii. Understanding Datum and Sketch Display

Module 2. Creating Sketches in 2-D Mode

- i. Understanding 2-D Mode
- ii. Configuring the 2-D Grid
- iii. Sketching Using Guides
- iv. Sketching Lines
- v. Sketching Arcs
- vi. Sketching Rectangles and Parallelograms
- vii. Sketching Circles
- viii. Sketching Fillets
- ix. Sketching Chamfers
- x. Sketching Construction Geometry
- xi. Sketching Text
- xii. Sketching Ellipses
- xiii. Sketching Splines
- xiv. Using Sketched Geometry Tools

Module 3. Creating Geometry Shapes

- i. Understanding Sketches and Sketch Regions
- ii. Creating Extrude Geometry
- iii. Creating Revolve Geometry
- iv. Creating Snapped Geometry Depth
- v. Removing Material and Specifying the Side
- vi. Projecting 3-D Geometry into a Sketch
- vii. Creating Sweep Geometry

Module 4. Creating Datums

- i. Creating Datums Theory
 - ii. Creating Datum Axes
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- iii. Creating Datum Planes

Module 5. Creating Engineering Geometry

- i. Creating Holes
- ii. Creating and Editing Circular Rounds
- iii. Creating and Editing Chamfers
- iv. Creating Drafts
- v. Shelling Solid Geometry
- vi. Patterning Geometry in One Direction
- vii. Patterning Geometry in Two Directions

Module 6. Measurements and Sections

- i. Using the Measure Tools
- ii. Creating Part Cross-Sections
- iii. Creating Assembly Cross-Sections

Module 7. Editing 3-D Geometry

- i. Using Shape Selection
- ii. Understanding Shape Selection Types
- iii. Leveraging Geometry Selection Rules
- iv. Understanding the Dragger
- v. Moving and Rotating Geometry
- vi. Moving Geometry by Dimension
- vii. Analyzing Side Surface Options
- viii. Offsetting Geometry
- ix. Managing Tangency
- x. Modifying Analytic Surfaces
- xi. Removing Geometry
- xii. Mirroring Selected Geometry
- xiii. Using the Geometry Search Tool

Module 8. Working with Assemblies

- i. Analyzing Component Display
- ii. Editing Component Position
- iii. Creating Assemblies and Inserting Components
- iv. Creating Components
- v. Designing Components in Assembly
- vi. Modifying Part Geometry Using Boolean Operations

