

Introduction to Model Based Definition with Creo Parametric

Overview

Course Length

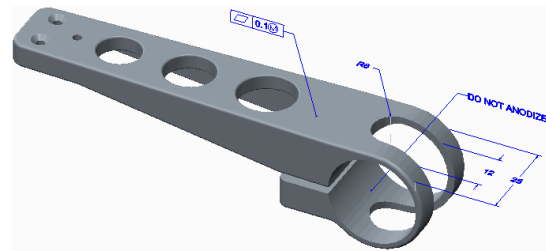
8 Hours

In this course, you will learn how to create an MBD model using a start part. You will discover how annotate mode provides the tools to create an MBD model using the MBD process within Creo Parametric. You will also learn about creating 3-D annotations and how to utilize them in combined states. You will manage the visibility of annotations within combination states. You will also create and modify various annotation elements, including dimensions, ordinate dimensions, datum reference features, geometric tolerances, surface finishes, symbols, notes, and datum targets.

After completing this course, you will have a better understanding of the MBD process and how to create MBD models using the MBD process.

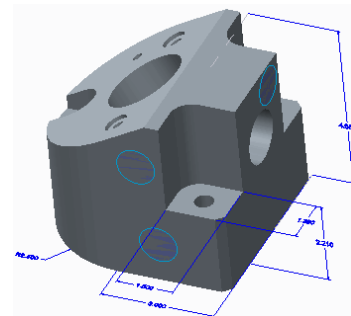
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric.



Course Objectives

- Understand the introduction to Model Based Definition
- Prepare models for annotation
- Create annotation elements
- Modify annotation elements
- Complete combination states
- Publish for technical data packages





Prerequisites

- Core Creo knowledge in parts and drawings
- Experience in Creo modeling
- Experience in Creo drawing creation

Audience

- This course is intended for any CAD designer involved in creating Model Based Definition models at his or her company. People in related roles will also benefit from taking this course.
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Agenda

Day 1

Module	1	Introduction to Model Based Definition
Module	2	Preparing Models for Annotation
Module	3	Creating Annotation Elements
Module	4	Modifying Annotation Elements
Module	5	Completing Combination States
Module	6	Publishing for Technical Data Packages
