

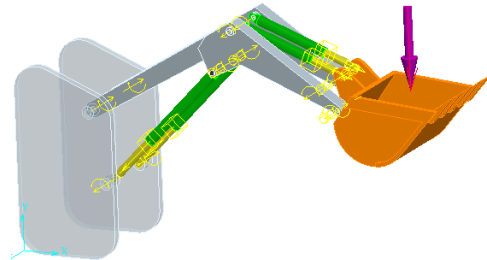
Mechanism Design using Creo Parametric

Overview

Course Code TRN-3427-T

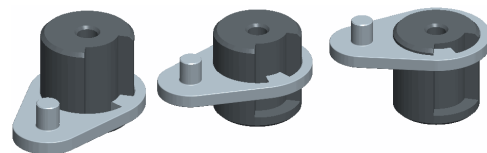
Course Length 1 Day

In this course, you will learn about creating mechanism connections, configuring the mechanism model, creating a kinematic analysis, and evaluating results. Mechanism Design using Creo Parametric is designed for experienced users who want to add motion to their models by creating mechanism connections and servo motors. In Creo Parametric you can add motion to your models using the standard mechanism functionality, often referred to as the Mechanism Design Extension (MDX). These topics will enable you to simulate the range of motion between components in your moving assemblies, create gear connections that simulate the gear ratios, create Cam connections that enable Creo Parametric parts to “push” other parts they come into contact with, and check for collisions between moving components. After completing this course, you will be prepared to work on mechanism designs using Creo Parametric Mechanism Design. At the end of each module, you will complete a skills assessment. The questions are used to help reinforce your understanding of the module topics and form the basis for review of any topics, if necessary.



Course Objectives

- Introduce the mechanism design process
- Create mechanism connections
- Configure motion and analysis
- Evaluate analysis results



Prerequisites

- Introduction to Creo Parametric

Audience

- Design engineers and mechanical designers who need to add and evaluate the motion of their assemblies.
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Agenda

Day 1

Module	1	Introduction to the Mechanism Design Process
Module	2	Creating Mechanism Connections
Module	3	Configuring Motion and Analysis
Module	4	Evaluating Analysis Results
