

# INTRODUCTION TO CREO PARAMETRIC, PTC'S 3D CAD SOFTWARE

## Argomenti

### Introduction:

- Introduction to parametric modeling
- Creo Parametric interface

### Part Modeling:

- Sketch environment (basic)
- Reference features (basic)
- Creating extrusions, revolutions, and ribs
- Creating Sweeps and Blends (basic)
- Creating holes, shells, and drafts (basic)
- Creating rounds and chamfers (basic)
- Multibody modeling
- Family table (basic)

### Assembly Modeling:

- Assembly with constraints
- Assembly with connections (basic)
- Exploding assemblies

### Modification Features:

- Group, copy, mirror (basic)
- Patterns (basic)

### Drawing environment (basic):

- Drawing views and layouts
- Creating annotations

### General Model Management:

- Measurement and inspection of models
- Examination of parent/child relationships
- Error resolution
- Definition and management of design intents

## Descrizione

This course introduces the essential features of Creo Parametric.

It covers the main commands related to the creation of parts, assemblies, and drawings, as well as the general management of models.

## Durata

For users with significant experience in using other parametric CAD software: 3 days.

These data are indicative; for an accurate estimate, a technical assessment of the needs and background of the involved users is necessary.