

Creo Parametric 3.0: Introduction to Solid Modeling - Part 2 (Volume 2)



Note: To complete this course, *Creo Parametric 3.0: Introduction to Solid Modeling - Part 1* is required. The *Creo Parametric 3.0: Introduction to Solid Modeling* training guide provides you with an understanding of the process of designing models with *Creo Parametric 3.0* through a hands-on, practice-intensive curriculum. You will learn the key skills and knowledge required to design models using *Creo Parametric 3.0*, starting with 2D sketching, through to solid part modeling, assembly creation, and drawing production.

Topics include: *Creo Parametric* fundamentals and interface Principles behind design intent Manipulating a model *Creo Parametric* file management Part creation and modification Sketching and creating geometry Sketcher mode functionality (sketching and dimensioning) Datum features Duplication techniques (patterns, mirroring) Creating relations to capture design intent *Creo Parametric* customization Design documentation and detailing Feature management Sweeps and blends Assembly creation and manipulation Parent/Child relationships in *Creo Parametric* models Model Analysis Feature failure resolution Effective modeling techniques Prerequisites: Experience in mechanical design and drawing production is recommended. *Creo Parametric 3.0: Introduction to Solid Modeling - Part 1*

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