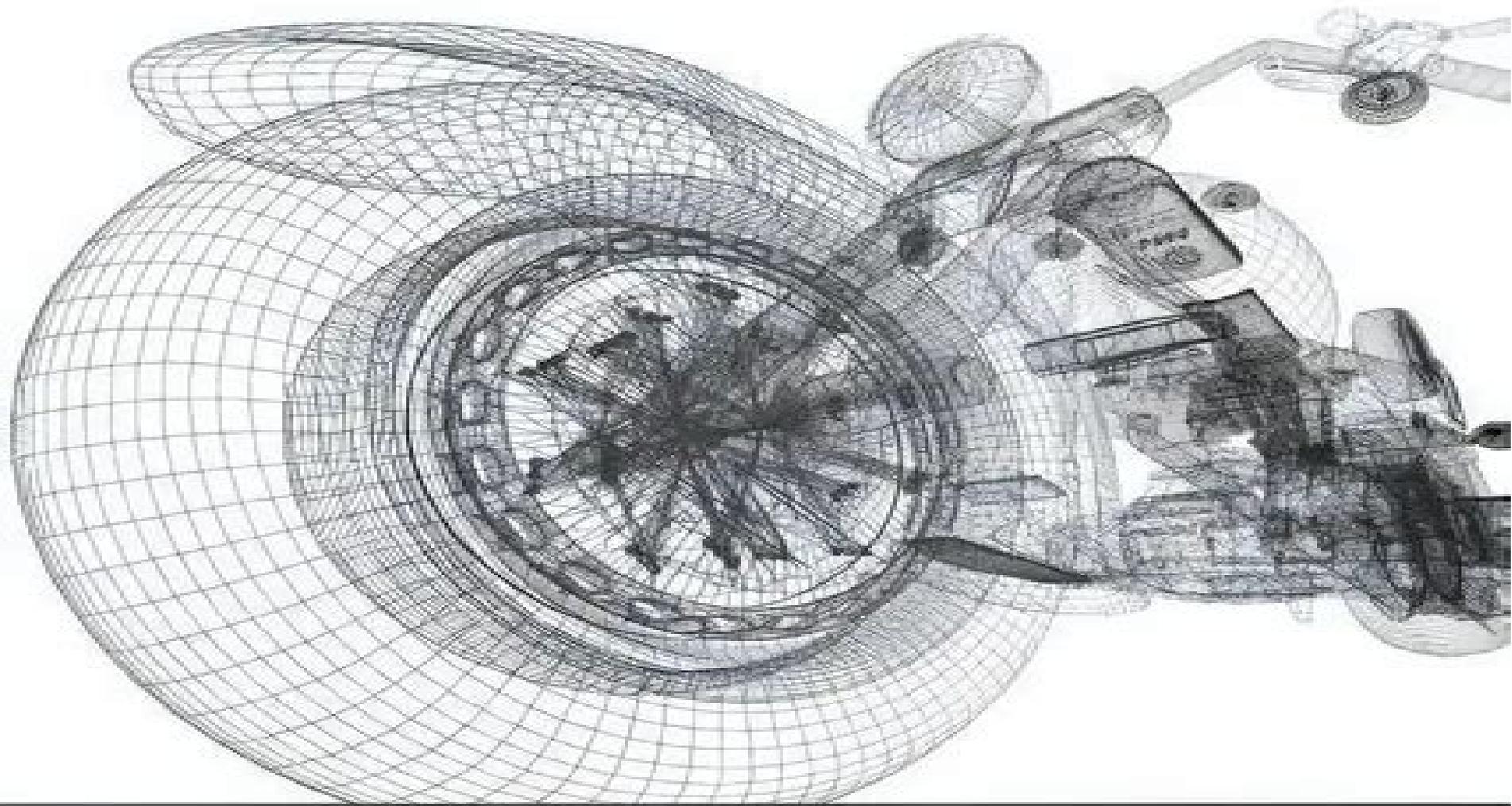




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ASCENT - Center for Technical Knowledge,2020-11-30 Note This learning guide is the first of a two part series with each guide sold separately The Creo Parametric 6 0 Introduction for Experienced 3D CAD Users learning guide is intended to provide accelerated introductory training in Creo Parametric 6 0 software This learning guide is designed for users that have 3D modeling design experience with other 3D CAD software packages e g CATIA TM Inventor TM NX TM SolidWorks R etc By leveraging the experience users gain in working with other 3D modeling software packages this hands on practice intensive guide is developed so that users who are new to Creo Parametric can benefit from a shorter introductory level learning guide You are taught how to find and use the modeling tools associated with familiar modeling strategies that are

used in other 3D CAD software You will acquire the knowledge necessary to complete the process of creating models from conceptual sketching through to solid modeling assembly design and drawing production Topics Covered Creo Parametric fundamentals and interface 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 Prerequisites Access to the Creo Parametric 6 0 software The practices and files included with this guide might not be compatible with prior versions Practice files included with this guide are compatible with the commercial version of the software but not the student edition Experience in mechanical design and drawing production using 3D CAD software This content was developed using Creo Parametric 6 0 Build 6 0 4 0 [Creo Parametric 3.0](#) ASCENT - Center for Technical Knowledge,2015-12-04 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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