

## Creo Training Manual

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This training guide is for the general purpose simulation software 'Creo Simulate 7.0' by Parametric Technology Corporation (PTC). This is a 'Hands-on', 'Step-by-Step' training guide with a series of example problems to cover the main fundamental concepts of simulation. This guide is intended for use by the Mechanical Design Engineer who wants to incorporate Mechanical Simulation into the design process. No prior simulation or analysis experience is required. This guide assumes the reader has a working knowledge of the basic Creo Parametric CAD modeling application and should have access to both Creo Parametric and Simulate to work through the examples.Topics Covered Include:Statics, Stress, Modal, Thermal, Vibration, Sensitivity StudiesAssemblies, Contact, Bolted Joints, Welded Joints, Thin-Walled Parts, Slender PartsAccuracy and Convergence issuesThe author, James Holst, is a registered Mechanical Engineer in the state of California and has been performing computer simulations of mechanical systems since the late 1970's mostly in the areas of stress, vibration, thermal and flow analysis. He has also provided simulation training classes to industry since the mid 1980's. He has used many different computer based simulation codes over the years from 'home grown' programs to commercially available applications integrated into CAD systems. He has extensive experience working in many different industries including. Much of his experience as an engineer, analyst and trainer has been incorporated into this guide.

This book starts with Creo Parametric 4.0 using step-by-step examples. It begins with creating sketches and parts, assembling them, and then creating print ready drawings. This book gives you an idea about how you can design and document various mechanical components, and helps you to learn some advanced tools and techniques. This book also follows some of the best practices in creating parts. In addition to this, there are some additional chapters covering sheet metal and surface design. Each topic in this book has a brief introduction and a step-by-step example. This will help you to learn Creo Parametric 4.0 quickly and easily. - Go through with the User Interface - A step-by-step practice to create sketches and 3D models - Teach you about advance Part Modeling tools - Learn the procedure to create Multiple-body parts - Learn to modify components at each step - Learn to create assemblies - Learn Top-down assembly design - Learn to create 2D drawings - Learn basic tools available in Sheet Metal and Surface Environment - Create sheet metal drawings - Create complex shapes using surface modeling tools

The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 7.0. The tutorial covers the major concepts and frequently used commands required to advance from a novice to an intermediate user level. Major topics include part and assembly creation, and creation of engineering drawings. Also illustrated are the major functions that make Creo Parametric a parametric solid modeler. Although the commands are presented in a click-by-click manner, an effort has been made, in addition to showing/illustrating the command usage, to explain why certain commands are being used and the relation of feature selection and construction to the overall part design philosophy. Simply knowing where commands can be found is only half the battle. As is pointed out numerous times in the text, creating useful and effective models of parts and assemblies requires advance planning and forethought. Moreover, since error recovery is an important skill, considerable time is spent exploring the created models. In fact, some errors are intentionally induced so that users will become comfortable with the [debugging] phase of model creation. At the end of each lesson is a short quiz reviewing the new topics covered in that chapter. Following the quiz are several simple "exercise" parts that can be created using new commands taught in that lesson. In addition to these an ongoing project throughout the book is also included. This project consists of several parts that are introduced with the early lessons and finally assembled at the end. Who this book is for This book has been written specifically with students in mind. Typically, students enter their first CAD course with a broad range of abilities both in spatial visualization and computer skills. The approach taken here is meant to allow accessibility to persons of all levels. These lessons, therefore, were written for new users with no previous experience with CAD, although some familiarity with computers is assumed.

The purpose of Creo Parametric 4.0 Advanced Tutorial is to introduce you to some of the more advanced features, commands, and functions in Creo Parametric. Each lesson concentrates on a few of the major topics and the text attempts to explain the [why] of the commands in addition to a concise step-by-step description of new command sequences. This book is suitable for a second course in Creo Parametric and for users who understand the features already covered in Roger Toogood's Creo Parametric Tutorial. The style and approach of the previous tutorial have been maintained from the previous book and the text picks up right where the last tutorial left off. The material covered in this tutorial represents an overview of what is felt to be the most commonly used and important functions. These include customization of the working environment, advanced feature creation (sweeps, round sets, draft and tweaks, UDF's, patterns and family tables), layers, Pro/PROGRAM, and advanced drawing and assembly functions. Creo Parametric 4.0 Advanced Tutorial consists of eight lessons. A continuing theme throughout the lessons is the creation of parts for a medium-sized modeling project. The project consists of a small three-wheeled utility cart. Project parts are given at the end of each lesson that utilize functions presented earlier in that lesson. Final assembly is performed in the last lesson.

The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 2.0. The tutorial covers the major concepts and frequently used commands required to advance from a novice to an intermediate user level. Major topics include part and assembly creation, and creation of engineering drawings. Also illustrated are the major functions that make Creo Parametric a parametric solid modeler. These topics are further demonstrated in the video files that come with every book. Although the commands are presented in a click-by-click manner, an effort has been made, in addition to showing/illustrating the command usage, to explain why certain commands are being used and the relation of feature selection and construction to the overall part design philosophy. Simply knowing where commands can be found is only half the battle. As is pointed out numerous times in the text, creating useful and effective models of parts and assemblies requires advance planning and forethought. Moreover, since error recovery is an important skill, considerable time is spent exploring the created models. In fact, some errors are intentionally induced so that users will become comfortable with the [debugging] phase of model creation. At the end of each lesson is a short quiz reviewing the new topics covered in that chapter. Following the quiz are several simple "exercise" parts that can be created using new commands taught in that lesson. In addition to these an ongoing project throughout the book is also included. This project consists of several parts that are introduced with the early lessons and finally assembled at the end.

The primary goal of Introduction to Finite Element Analysis Using Creo Simulate 7.0 is to introduce the aspects of finite element analysis (FEA) that are important to engineers and designers. Theoretical aspects of finite element analysis are also introduced as they are needed to help better understand the operations. The primary emphasis of the text is placed on the practical concepts and procedures of using Creo Simulate in performing Linear Statics Stress Analysis; but the basic modal analysis procedure is covered. This text is intended to be used as a training guide for both students and professionals. This text covers Creo Simulate 7.0 and the lessons proceed in a pedagogical fashion to guide you from constructing basic truss elements to generating three-dimensional solid elements from solid models. This text takes a hands-on-exercise intensive approach to all the important Finite Element Analysis techniques and concepts. This textbook contains a series of twelve tutorial style lessons designed to introduce beginning FEA users to Creo Simulate. The basic premise of this book is the more designs you create using Creo Simulate, the better you learn the software. With this in mind, each lesson introduces a new set of commands and concepts, building on previous lessons.

Michael A. Brattoli has over 35 years experience in new product development, quality engineering, project management and development, and engineering supervision in a variety of industries, from aerospace to faucets. As the Lead CAD Designer/PLM Administrator for Moen, Incorporated, he is responsible for all global aspects of CAD software/hardware installations as well as coordinating the activities of Moen's internal and external user communities, documenting and enforcing best practices, and providing mentoring and training as required. Mr. Brattoli currently holds multiple U.S. patents, both utility and design. He began using Pro/ENGINEER(R) with release 11, and has over 24 years experience using the software. He has been chosen as a presenter at numerous International PTC/User Conferences (1997, 2005, 2006, 2008, 2012, 2013, 2014, 2015, 2016, and 2017) focusing on areas relating to CAD training, Surfacing, Reverse Engineering, Rendering, Windchill, and Assembly functionality using Pro/ENGINEER(R) and Creo Parametric(R). Mr. Brattoli has been a Steering Group member of the PTC/USER Industrial Design Technical Committee (responsible for the surfacing, reverse engineering, and rendering modules) since 1996, and is the President of the Northern Ohio PTC/USER regional user Group (NOPUG). He also served on the PTC/USER board of directors in 2016 as the Director of Regional User Groups for the organization. As an adjunct professor he has been teaching Pro/ENGINEER(R) and Creo Parametric(R) at Lorain County Community College in Elyria, OH since the fall of 1996, beginning with release 15 of the software. Mr. Brattoli is the author of Presenting Creo Parametric 3.0, a training manual on the use of Creo Parametric(R) software. He has also authored Pro/ENGINEER(R) and Creo Parametric(R) training manuals covering releases Wildfire 5.0 through Creo 5.0 of the application. He has participated in numerous articles for Design News, Machine Design, Industry Week, and other magazines and industry periodicals on various subjects related to Creo Parametric(R) and Pro/ENGINEER(R)

▫ Uses step-by-step tutorials designed for novice users ▫ Explains not only how but also why commands are used ▫ Covers part and assembly creation, creating engineering drawings and parametric solid modeling The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 8.0. The tutorial covers the major concepts and frequently used commands required to advance from a novice to an intermediate user level. Major topics include part and assembly creation, and creation of engineering drawings. Also illustrated are the major functions that make Creo Parametric a parametric solid modeler. Although the commands are presented in a click-by-click manner, an effort has been made, in addition to showing/illustrating the command usage, to explain why certain commands are being used and the relation of feature selection and construction to the overall part design philosophy. Simply knowing where commands can be found is only half the battle. As is pointed out numerous times in the text, creating useful and effective models of parts and assemblies requires advance planning and forethought. Moreover, since error recovery is an important skill, considerable time is spent exploring the created models. In fact, some errors are intentionally induced so that users will become comfortable with the "debugging" phase of model creation. At the end of each lesson is a short quiz reviewing the new topics covered in that chapter. Following the quiz are several simple "exercise" parts that can be created using new commands taught in that lesson. In addition to these an ongoing project throughout the book is also included. This project consists of several parts that are introduced with the early lessons and finally assembled at the end. Who this book is for This book has been written specifically with students in mind. Typically, students enter their first CAD course with a broad range of abilities both in spatial visualization and computer skills. The approach taken here is meant to allow accessibility to persons of all levels. These lessons, therefore, were written for new users with no previous experience with CAD, although some familiarity with computers is assumed. The tutorials in this textbook cover the following topics: ▫ Introduction to the program and its operation ▫ The features used in part creation ▫ Modeling utilities ▫ Creating engineering drawings ▫ Creating assemblies and assembly drawings

Creo Simulate 7.0 Tutorial introduces new users to finite element analysis using Creo Simulate and how it can be used to analyze a variety of problems. The tutorial lessons cover the major concepts and frequently used commands required to progress from a novice to an intermediate user level. The commands are presented in a click-by-click manner using simple examples and exercises that illustrate a broad range of the analysis types that can be performed. In addition to showing the command usage, the text will explain why certain commands are being used and, where appropriate, the relation of commands to the overall Finite Element Analysis (FEA) philosophy are explained. Moreover, since error analysis is an important skill, considerable time is spent exploring the created models so that users will become comfortable with the "debugging" phase of modeling. This textbook is written for first-time FEA users in general and Creo Simulate users in particular. After a brief introduction to finite element modeling, the tutorial introduces the major concepts behind the use of Creo Simulate to perform Finite Element Analysis of parts. These include modes of operation, element types, design studies (analysis, sensitivity studies, organization), and the major steps for setting up a model (materials, loads, constraints, analysis type), studying convergence of the solution, and viewing the results. Both 2D and 3D problems are covered. This tutorial deals exclusively with operation in integrated mode with Creo Parametric. It is suitable for use with both Releases 7.0 of Creo Simulate.