

CHAPTER 1

Introduction to Parametric Design

1.1 Introduction

Computer-aided design (CAD) is the development of new part models and modification of already existing parts' designs for new prototypes of complex assemblies such as cars, planes, computers, etc. The development and modification of parts include analysis of design data from customers and other stakeholders. Additionally, the CAD provides communication tools such as drawings and simulations to accurately estimate and manufacture customer's specifications.

This chapter introduces the basic concepts behind parametric design and feature-based modeling. Parametric design is a process used to manipulate, regenerate, and design objects based on a set of rules or parameters. There are many concepts that appear regularly in parametric design projects. One important concept is repetition, which works best when it is used in parallel with rotation, scaling, and movement. This simply means to copy an object multiple times while introducing a gradual change in its scale for example. Within this chapter, engineering graphics and three-dimensional modeling concepts will be explored. Additionally, parametric modeling principles will be covered. Upon finishing this chapter, you will be able to:

- Describe the utilization of computer-aided design within engineering graphics.
- Compare three-dimensional modeling techniques.
- Describe concepts associated with parametric modeling and design.
- Describe the use of parametric design within a concurrent manufacturing.

CHAPTER OBJECTIVES

After completing this chapter, you should:

- Understand what Creo Parametric is.
 - Understand associativity, design intent, feature based modeling, parametric modeling, and solid modeling concepts.
 - Learn how design process occurs and how teams from various disciplines work together to accomplish their common goal
 - Learn how to create a free account, download and install Creo Parametric software
 - Understand various file types within Creo Parametric and how to use the Augmented Reality application that accompanies this book.
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1.1.1 What is Creo Parametric?

Creo Parametric (formerly known as Pro/Engineer) is a suite of programs that are used in the design, analysis, and manufacture of a virtually unlimited range of products. In a nutshell, Creo Parametric is a parametric, feature-based solid modeling software.

1.1.2 Definitions

Associativity	The sharing of a component’s database between its application modes.
Design intent	The intellectual arrangement of assemblies, parts, features, and dimensions to meet a design requirement.
Feature-based modeling	The building of a part model with intuitive and realistic features, such as holes, pads, pockets, fillets, and chamfers and leaving low-level geometry like lines, arcs, and circles for the Creo Parametric to figure out.
Parametric design	“Parametric” means that the physical shape of your part or assembly is driven by the attributes (dimensions) of its features. You can define or modify the attributes at any time and the model is regenerated throughout your model and assembly and drawings. Creo Parametric has bidirectional associativity – this means changing a dimension on a drawing changes the shape of the model and vice versa.
Solid modeling	The developed computer model contains all the necessary information that a real solid object would have.

This book discusses solid modeling of parts, developing technical drawings or blueprints for those parts with geometric dimensions and tolerances (GD&T), using these part components within a larger assembly file, developing assembly drawings with balloons and bill of materials, developing simulation reports for modeled parts with finite element analysis (FEA), and modeling sheet metal products. It is worth to recognize what file types Creo Parametric will generate for the type of required functions (Table 1.1):

TABLE 1.1 Naming Conventions Used in Creo Parametric in this Book

File Extension	File Type	Description
.sec	Section file	Saves sketches in .sec extension
.prt	Part file	Saves 3D modeled parts
.drw	Drawing file	Saves drawings of parts and assemblies
.asm	Assembly file	Saves assemblies of part components
.rwd	Simulate result definition file	Saves simulation results
.pbk	Mechanism playback file	Saves mechanism animation results
.mpeg	Moving picture experts group file	Saves captured video of mechanism animation

1.2 Introduction to Computer-Aided Design

Engineering design graphics has made significant changes since the early 1980s. For the most part, these changes are a result of the evolution of CAD. Before CAD, design was accomplished by traditional board drafting utilizing paper, pencil, and various other manual drafting devices.

1.2.1 Design Process

New design concepts or ideas should exist in the mind of the designer before they can be modeled. Working in a team continuous improvement process or a group of students in the classroom may generate a need to consult each other before starting the Creo Parametric. This will require clear expression of the ideas verbally, symbolically, and graphically. The design process is your ability to combine scientific principles, resources, and existing products into a problem solution. Suppose a customer called your company and said the tractor they purchased has stopped in the middle of a cornfield due to the broken part. A group of engineers will go to run the test and collect the data on the tractor. Once the data is collected, a team is formed with people who have specific skills. Each member will express their opinion regarding the problem. Set of quality control tools (such as why-why diagram or fishbone diagram) might be developed in order to conduct the root cause analysis. Notice that the identification of the problem and the brainstorming sessions are accompanied with evaluating the ideas and selecting the appropriate ones until the final possible solution is identified. Developing three-dimensional part and the drawing would be the next “implement” stage of the problem-solving phase. This stage needs to be checked and approved in order to manufacture a product. The design engineer is responsible for checking and approving the production drawings. Dimensions, tolerances, and manufacturing notes (such as material) must be present on the drawing and discussed with the supplier before sending for production.

1.2.2 Free Creo Download and Installation (Student Version)

Power to create (PTC) provides a free license for Creo Parametric for educational purposes. Although some of the functions are limited in the student version, all of the tutorials discussed in this book can be achieved with the free version. Because Creo Parametric is a Windows-based software, you can install Windows operating system on your Mac via Parallels software or through the Bootcamp.

Instructions to Download Creo Student Edition (Note: File size is approximately 4 GB for 64 bit)

1. Use your web browser to navigate to:

<https://www.ptc.com/en/academic-program/academic-products/free-software/creo-college-download>

2. Enter the following information to start the process to acquire the software:

First (given) name: (your name)

Last (family) name: (your name)

Email address: (your SHSU email address)

City: Where your school is located, i.e. Huntsville

State: Where your school city is located, i.e. Texas

Country: Where your school city is located, i.e. United States

School/Institution: Name of your school, i.e. Sam Houston State University

Grade level: Your school level, i.e. Undergraduate

How will this software be used? In Classroom

How many computers will you install the software on? 1

How did you find out about this software? Professor

3. Installation instructions

Follow the link to download CREO version your professor recommended. This link will lead you to a PDF with the instructions on how to proceed. *This PDF is important since it is your PRODUCT CODE, keep the PDF open (or save it if you need to repair or reinstall your software).*

Follow the link on the PDF to download PTC CREO version, which also lets you CREATE A NEW ACCOUNT. To create an ACADEMIC ACCOUNT, enter the following information:

First name: (your name)

Last name: (your name)

Email: (your SHSU email address)

Academic program: Other

Title: College/University Student

School: Your School name, i.e. Sam Houston State University

State/Province: Your Province or State, i.e. TX

Country: Your country of residency, i.e. United States

Create a password. DON'T FORGET IT. Then enter the CAPCHA into the box that are underlined and to the right of the *PTC Security Challenge* box.

- 4. With your account created, go back to the *Quick Installation Guide* PDF that PTC created for you and follow the instructions to download CREO version.

1.2.3 Creo Installation

- Once the download complete, extract (unzip) the download folder → Click “Setup” (Figure 1.1)

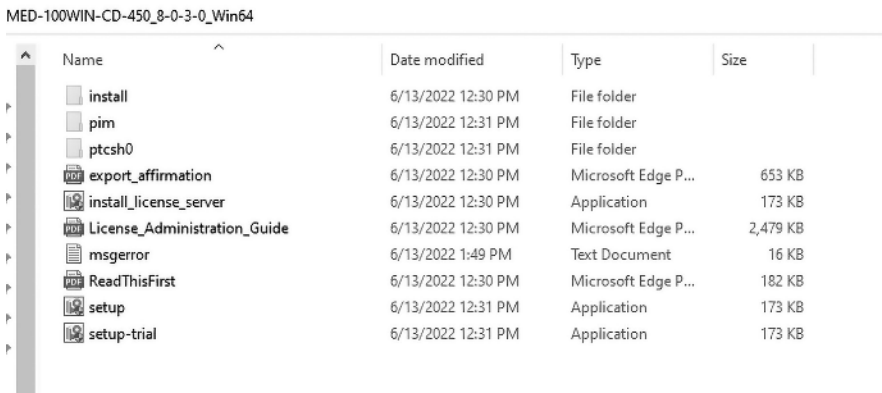


FIGURE 1.1 Creo Installation Folder

- Select “Install new software” (unless you are adding packages) → Click “Next” on the Creo Installation Assistant Window (Figure 1.2)



FIGURE 1.2 Creo Installation Assistant

6 CHAPTER 1 Introduction to Parametric Design

- Read through the Software License Agreement → Accept the license agreement → Click Next (Figure 1.3)

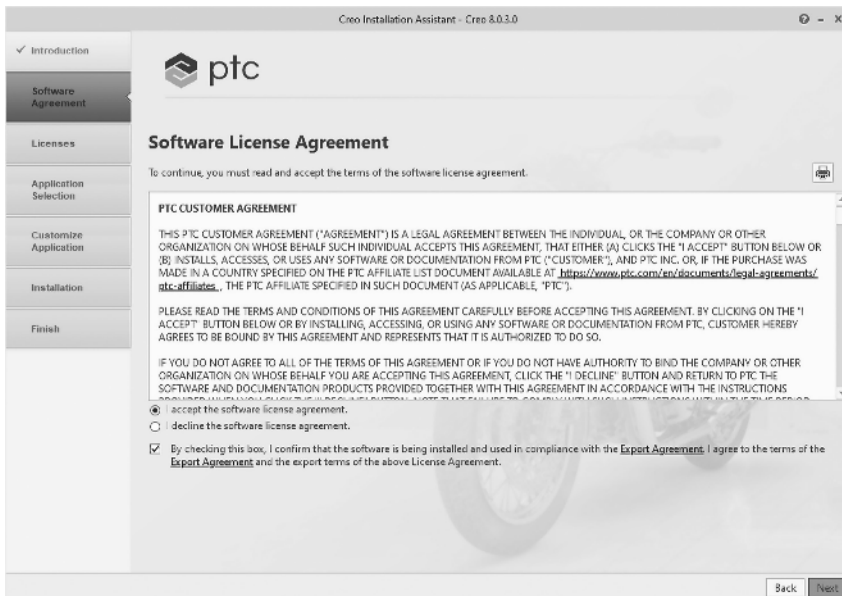


FIGURE 1.3 Creo License Agreement

- On the License Identification window (Figure 1.4), enter the serial number on your PDF document (Quick Installation Guide) you opened/saved earlier (the serial number will look like BK600908EDSTUDENTEDUNI)

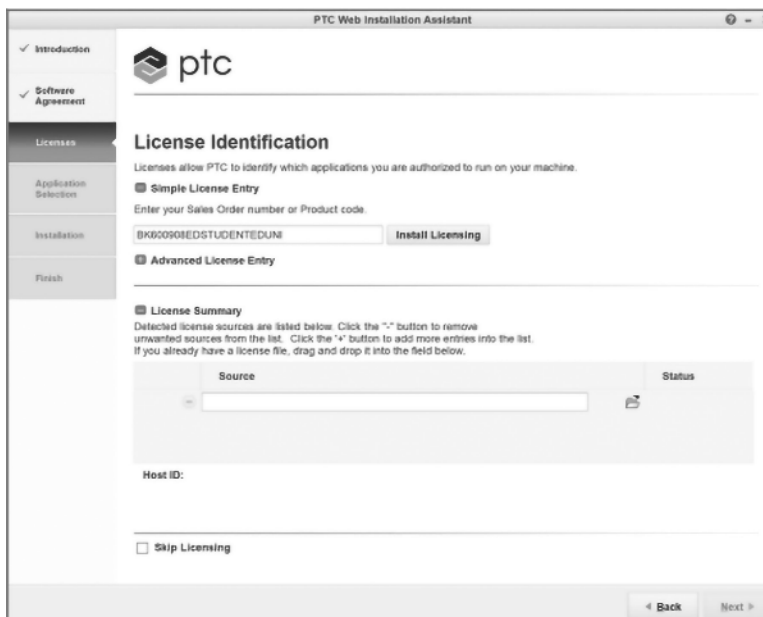


FIGURE 1.4 Creo Installation Assistant

- Enter your PTC username (the one you created earlier) and password → Click Login
- Once you see “Success,” click Next to start Creo Parametric and packages

1.2.4 Starting Creo Parametric

Previously known as ProEngineer Wildfire, Creo Parametric received its name in 2011 to include additional applications. The user interface of Creo Parametric is set up similar to all windows applications that comprises of the ribbon with most frequently used tools, menu tabs for additional tools, and the workspace, where all the modeling and sketching takes place. Initial start of Creo Parametric launches a welcome screen (Figure 1.5) with basic functions.

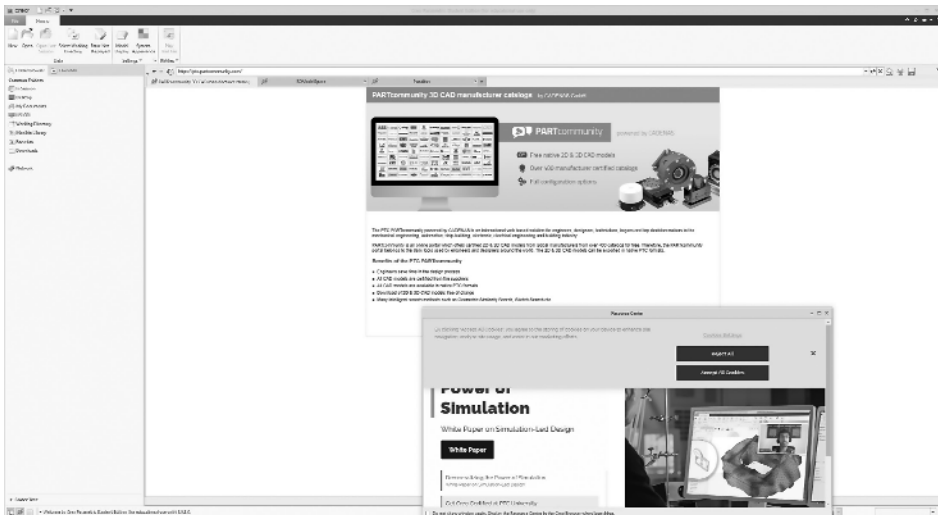


FIGURE 1.5 Creo Parametric Welcome Screen

Because Creo Parametric uses context-sensitive application menus, available menus can be used right away, while grayed out menus or options will become available as applicable. For instance, the “Open Last Session” tool is grayed out and is not available on Figure 1.5. This means we just started Creo Parametric and do not have an active last session. On the other hand, the “New, Open, Select Working Directory, etc.” tools are available for use. Later in Chapter 2, we will start our first model with clicking the “New” button and myriad of additional tools will be available for use. The left browser panel lets the user navigate through local files and folders, additionally, expanding the “Folder Tree” provides more navigation options. Once the part modeling starts, this left panel will become a “Model Tree,” where each modeling step is presented. Initially, the welcome screen contains built-in internet browser that you can even watch on YouTube videos. The purpose of this browser is, of course, not for YouTube videos but navigate through the database called PDMLink within the organization to check out and check in parts. For instance, you as a new design engineering in a manufacturing company revised one of the parts for future model car and save it. But where did you save? On your computer? How will other people have access to the part when you are

away? Yes, you could send them with an email, but that could take up email storage space. Therefore, the PDMlink database lets organizations check in (upload) their parts onto the server, which will instantly become available for other team members regardless their location in the world. This enables organizations to stay operational 24/7 as people from China or India can pick up your last modified work and continue as you go home after your shift in the States. Similarly, you will pick up the updated iteration of the model next morning with significant amount of progress on the part.

Although it is possible to use laptop's touchpads to model some basic 3D models in Creo Parametric, it is highly recommended and crucial to use a three-button mouse (left mouse button, right mouse button, and the scroll wheel, as referred to middle mouse button) as the middle mouse button can end a command or use for panning and zooming functions. Below (Figure 1.6) is the explanation of what mouse button functions are in Creo Parametric:

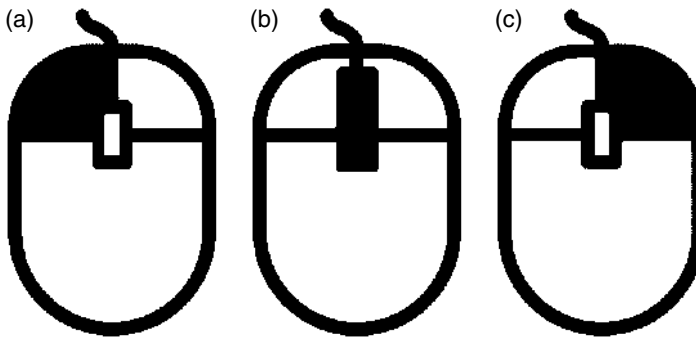


FIGURE 1.6 (a) Left Mouse Button (LMB), (b) Middle Mouse Button (RMB), (c) Right Mouse Button (RMB)

LMB (left mouse button): Mainly used to select tools, commands, and operations.

MMB (middle mouse button): A very handy tool to accept or end currently selected tool/operation or zooming in/out and rotating/spinning/panning the geometry.

RMB (right mouse button): Majority of times the right mouse button is used with clicking and holding the button, which brings quick access to most frequently used tools and commands.

It is a good practice to organize the folder before starting work with Creo Parametric. It is highly recommended to set up a working directory, so the saved files are stored in one place. Although setting up a working directory is discussed in Chapter 2, it is good to understand why we need to set it up. Consider a larger assembly file that comprises of 5 components. This assembly could be your pen and the components are as follows: 1, lead; 2, pen housing; 3, cap; 4, spring; and 5, button to retrieve the lead. Let us assume that all these components are saved in different locations of the computer, such as my documents, miscellaneous on drive d, or on a usb thumb stick. You can start a new assembly and locate all of these components from their stored locations and your assembly will be complete. However, the moment you

send the assembly to someone else or you want to open it on a different computer, the assembly will fail to open its components. This is because the folders that contain the components are inaccessible by the new computer, and it does not know where they are. Therefore, saving all those components in a single working directory will automatically retrieve all assembly components on a different computer. If you are using learning management system such as Blackboard, uploading a folder with all parts, drawings, and assembly files needs the folder compressed. To compress a folder, Right click → Send to → Zipped/compressed file (windows), on Mac, right click and compress.

1.2.5 Augmented Reality Companion Application and Image Targets

This book is accompanied with an Augmented Reality (AR) application that you can download and install on your Android (in development for iOS) devices. The application enables to visualize and understand the three-dimensional representation of the blueprint or the outcome of the part, assembly, or animations. The AR application provides interactive buttons to rotate, scale, assemble, and watch a short, animated clip of a tool. Simply start the application, point your Android device's camera on the figures within the textbook to experience the AR.

For effective use of the AR application, install the application on your Android device through the following QR code (Figure 1.7), open the application and point your device's camera to the image target.



FIGURE 1.7 QR Link to Download and Install the AR Companion App

Each chapter has an opening figure that can be augmented with the application (Figure 1.8). All image targets are compiled separately in the Appendix for reference. You can rotate each 3D model from the AR application to visualize and inspect various features of parts from different orientations. This version of



FIGURE 1.8 AR Companion App Projecting 3D Models

the AR companion application augments the completed part before starting the chapter. Animated tutorial clips are excluded from this version to reduce stress on the mobile device. Either all animated tutorials will be delivered as an update to the companion application or the user will be able to install it separately through the same QR code.