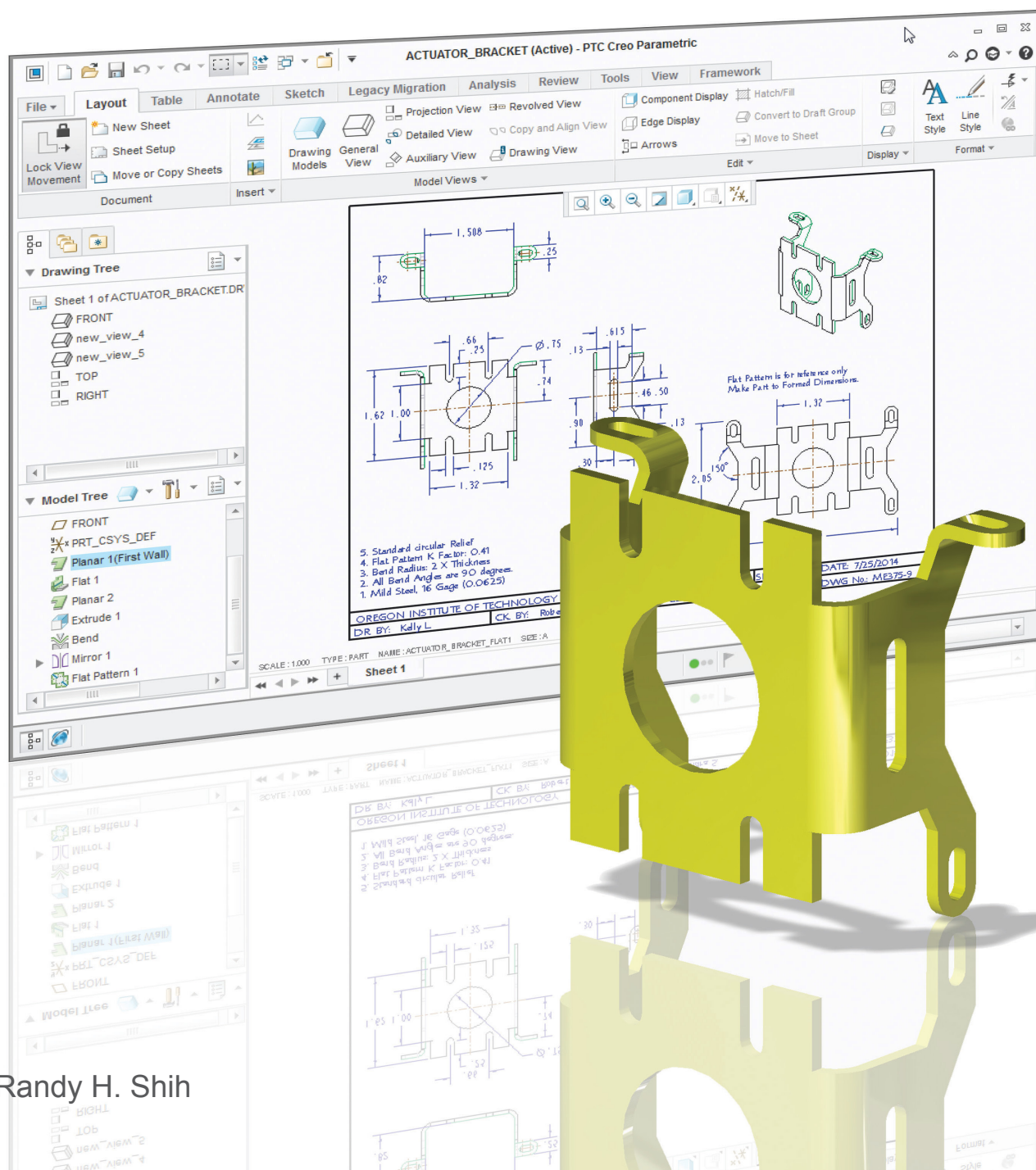


# Parametric Modeling with Creo® Parametric 3.0

**NEW**  
Sheet Metal Design

## An Introduction to Creo® Parametric 3.0



Randy H. Shih

Visit the following websites to learn more about this book:



[amazon.com](https://www.amazon.com)

[Google books](https://books.google.com)

[BARNES & NOBLE](https://www.barnesandnoble.com)

# Table of Contents

## Preface

## Acknowledgments

## Introduction

Introduction	Intro-2
Development of Computer Geometric Modeling	Intro-2
Feature-Based Parametric Modeling	Intro-6
Getting Started with Creo Parametric	Intro-7
Creo Parametric Screen Layout	Intro-8
Ribbon Toolbar	Intro-9
Quick Access Toolbar	Intro-9
Message area	Intro-9
Graphics display area	Intro-9
Navigator	Intro-10
Web Browser	Intro-10
Navigator Display Controls	Intro-10
Basic Functions of Mouse Buttons	Intro-11
Model Tree window and Feature toolbars	Intro-12
On-Line Help	Intro-13
Leaving Creo Parametric	Intro-14
Creating a CAD files folder	Intro-14

## Chapter 1

## Parametric Modeling Fundamentals

Introduction	1-2
The Adjuster design	1-3
Starting Creo Parametric	1-3
Step 1: Units and Basic Datum Geometry Setups	1-5
Adding the First Part Features — Datum Planes	1-7
Switching on the Plane Tag display	1-8
Step 2: Determine/Setup the Base Solid Feature	1-9
Sketching plane – It is an XY CRT, but an XYZ World	1-10
Defining the Sketching Plane	1-11
Defining the Orientation of the Sketching Plane	1-12
Step 3: Creating 2D Rough Sketches	1-14
The Creo Parametric SKETCHER and INTENT MANAGER	1-15
Graphics Cursors	1-16
Geometric Constraint Symbols	1-17
Dynamic Viewing Functions	1-18
Step 4: Apply/Modify Constraints and Dimensions	1-19
Modifying the dimensions in a sketch	1-22

Repositioning Dimensions	1-24
Step 5: Completing the Base Solid Feature	1-25
The Third Dynamic Viewing Function	1-26
Display Modes: Wireframe, Shaded, Hidden Edge, No Hidden	1-27
Step 6: Adding additional features	1-28
Creating a CUT Feature	1-33
Creating the 2D Section of the CUT Feature	1-34
Creating another CUT Feature	1-37
Delete/Select the Sketching References	1-38
Create the 2D Section	1-40
Save the Part	1-42
Review Questions	1-43
Exercises	1-44

## **Chapter 2**

### **Constructive Solid Geometry Concepts**

Introduction	2-2
Binary Tree	2-3
The <i>Locator</i> Design	2-4
Modeling Strategy - CSG Binary Tree	2-5
Starting Creo Parametric	2-6
Units Setup	2-7
Adding the First Part Features — Datum Planes	2-9
Base Feature	2-9
Create a 2D Parametric Section	2-11
Create the Second Solid Feature	2-14
Create the 2D Sketch	2-15
Create a CUT Feature	2-18
Create a 2-D Section	2-19
Redefine a Feature Element	2-22
Create a Placed HOLE Feature	2-24
Create the Final Feature	2-27
Create the 2-D Section	2-28
Review Questions	2-31
Exercises	2-32

## **Chapter 3**

### **Model History Tree**

Introduction	3-2
The <i>L-Support</i> Design	3-3
Starting Creo Parametric	3-3
Modeling Strategy	3-6

Create the Base Feature	3-7
2D Sketch of the Base Feature	3-7
Set up the Default View and Sketcher View	3-9
Add the Second Solid Feature	3-10
Rename the Part Features	3-14
Expand the Model Tree Listing	3-15
Create a Placed HOLE Feature	3-16
Create a Rectangular Cutout	3-18
Examine the Model History	3-22
History-based Part Modifications	3-23
Modify Dimensional Values	3-24
Modify the 2D Section of the Base Feature	3-26
Use the Measure Tools	3-28
Calculate the Associated Physical Properties	3-29
Review Questions	3-34
Exercises	3-35

## **Chapter 4**

### **Parametric Relations and Constraints**

Geometric Construction and Constraints	4-2
The <i>Gasket</i> Design	4-2
Modeling Strategy	4-3
Starting Creo Parametric and Using the Metric Template	4-4
Adjusting the Default Model Orientation and Sketcher Startup	4-5
Create the Base Feature	4-6
Implicit Geometric Relationships	4-7
Lock/Disable/Enable Constraints	4-8
Create and Divide a Circle	4-11
Create a Closed Region Sketch	4-12
Modify Geometry by Drag-and-Drop	4-13
Controlling Geometric Relationships	4-14
Add Rounded Corners	4-16
Adjusting the Constructed Geometry	4-17
Deleting Geometric Constraints	4-18
Weak Dimensions and Constraints	4-19
Complete the Base Feature	4-20
Using the Offset Loop Option to Create a CUT feature	4-21
Parametric Relations and Geometric Constraints	4-24
Create a Simple Plate Design	4-24
Using a Sketch File	4-25
Dimensional Values and Dimensional Variables	4-26
Save the Sketch File and Exit the Sketcher Mode	4-28
Sketch Files vs. Part Files	4-29
Starting a new Creo Parametric Part	4-29

Create the Base Feature	4-30
Placing the 2D Sketch	4-31
Display the Parametric Relations	4-33
Complete the Extrusion Feature	4-34
Parametric Relations at Different Levels	4-35
Review Questions	4-39
Exercises	4-40

## **Chapter 5**

### **Parent/Child Relationships**

Introduction	5-2
The <i>U-Bracket</i> Design	5-3
Preliminary Modeling Considerations	5-3
Starting Creo Parametric and Using the Metric Template	5-4
Create the Base Feature	5-5
Create a and Transform a Rough Sketch	5-6
Create the Second Solid Feature	5-14
Create a CUT Feature	5-16
Create the CENTER_DRILL Feature	5-17
Examine the Parent/Child Relationships	5-19
Display Parent/Child Info	5-20
Modify a Parent Dimension	5-21
A Design Change	5-22
Feature Suppression	5-23
Create the Circular Cut Feature	5-24
Listing Suppressed Features	5-25
Reactivate the CENTER_DRILL Feature	5-26
Reroute the CENTER_DRILL Feature	5-27
Suppress the RECT_CUT Feature	5-28
A Flexible Design Approach	5-29
Review Questions	5-30
Exercises	5-31

## **Chapter 6**

### **Datum Features, 3D Annotation and Part Drawings**

Datum Features	6-2
3D Annotation	6-2
Drawings from Solid Models	6-2
The <i>Rod-Guide</i> Design	6-3
Modeling Strategy	6-4
Starting Creo Parametric	6-5
Create the Base Feature	6-5

Create a Datum Axis	6-8
Create a Datum Plane at an Angle	6-9
Create the next Solid Feature	6-11
Create a Datum Plane Using the Filter Option	6-13
Create a Placed Feature	6-17
Add four holes to the Base Feature	6-18
3D Annotation	6-20
Create a Multi-View Drawing Using a Drawing Template	6-32
Create another Multi-View Drawing	6-34
Adding a new Sheet in the Existing Drawing	6-35
Adding a Primary Main View	6-36
Adding a Projected Auxiliary View	6-37
Adjust the Overall Drawing Display	6-38
Dimension Appearances	6-40
Displaying Center Axes	6-42
Annotation Appearances	6-43
Adding/Deleting Dimensions – Feature/Driven Dimensions	6-44
Add an Isometric View	6-46
Review Questions	6-49
Exercises	6-50

## **Chapter 7**

### **Symmetrical Features in Designs**

Symmetrical Features in Designs	7-2
A Revolved Design: <i>Wheel</i>	7-2
Modeling Strategy - A Revolved Design	7-3
Starting Creo Parametric	7-4
Create the Base Revolved Feature	7-4
Create Diameter Dimensions	7-6
Modify the Values of Dimensions	7-7
Complete the Revolved Feature	7-8
Using the Mirror Option	7-9
Add a Radial Hole Feature	7-10
Create a Circular Pattern	7-12
Parametric Relations	7-13
Create a Multi-View Part Drawing	7-15
Add the Primary Main View	7-16
Add a Projected Sectional View	7-17
Modify the Overall Drawing Display	7-20
Display Feature Dimensions	7-21
Adding Additional Driven Dimensions	7-23
Adjust Dimension Appearances	7-24
Display Center Axes	7-25
Adjusting the Length of Centerlines	7-26

Create Additional Centerlines	7-26
Overriding a Dimension Value	7-29
Associative Functionality – Modifying Feature Dimensions	7-30
Bi-Directional Associative Functionality	7-32
Create a Custom Creo Title Block	7-34
Create a Drawing Template with 2D Views and Title Block	7-39
Review Questions	7-46
Exercises	7-47

## **Chapter 8**

### **Three Dimensional Construction Tools**

Introduction	8-2
A Thin-Walled Part: <i>Dryer Housing</i>	8-2
Modeling Strategy	8-3
Starting Creo Parametric	8-4
Create the Base Feature	8-4
Create a Revolved Feature	8-6
Create the second Feature	8-7
Create a Blend Feature	8-9
Create 2D Blend Sections	8-11
Create 3D Rounds and Fillets	8-16
Create a Shell Feature	8-17
Create a Pattern Leader	8-19
Create a Rectangular Pattern	8-21
Create a Sweep Feature	8-23
➤ Define the Sweep Trajectory	8-23
➤ Define the Sweep Section	8-25
Review Questions	8-27
Exercises	8-28

## **Chapter 9**

### **Basic Sheet Metal Designs**

Sheet Metal Processes	9-2
Sheet Metal Modeling	9-4
K-Factor and Y-Factor	9-5
The Actuator Bracket Design – 16 Gage Sheet Metal	9-6
Examine and Set Sheet Metal Defaults	9-8
Create the Base Face Feature of the Design	9-10
Using the Flat Command	9-12
Add another Planar Feature	9-14



Add a Cut Feature	9-16
Create a Bend Feature	9-18
Create a Mirrored Feature	9-21
Confirm the Model Dimensions	9-22
Create the Associated Flat Pattern	9-23
Confirm the Flattened Length	9-24
Create an Instance of the Flat Pattern	9-25
Create a 2D Sheet Metal Drawing	9-26
Include a 2D View of the Flat Pattern	9-29
Display Center Axes and Dimensions	9-31
Review Questions	9-34
Exercises	9-35

## **Chapter 10**

### **Advanced Modeling Tools**

Introduction	10-2
Summary of Modeling Considerations	10-2
The <i>BRACKET</i> Design	10-2
Modeling Strategy	10-3
Starting Creo Parametric	10-4
Create the Base Feature	10-4
Using the CUT Option to Create a <i>Boolean INTERSECT</i>	10-6
Create a Reference Plane	10-10
Create Another Extrude Feature	10-11
Using the Draft Tool	10-14
Create the Top Cylindrical Feature	10-16
Create a Hole through the Cylinder	10-18
Create a Sketched Hole on the Base Feature	10-18
Create a Mirror Feature	10-21
Create Chamfers	10-22
Adding <i>SIMPLE</i> Rounds and Fillets - Edges	10-23
Adding <i>Advanced</i> Rounds and Fillets - Surfaces	10-24
Add the Last Feature	10-26
Changing Model Colors	10-27
Assign Material Properties and Save the Part	10-28
Review Questions	10-30
Exercises	10-31

## **Chapter 11**

### **Assembly Modeling – Putting It All Together**

Introduction	11-2
The <i>BRACKET</i> Assembly	11-2

Assembly Modeling Methodology	11-3
Additional Parts	11-4
(1) <i>Base Plate</i>	11-4
(2) <i>Bushing</i>	11-4
(3) <i>Cap Screw</i>	11-5
Create a Subassembly	11-6
Retrieve the <i>Bracket</i> Component	11-7
Retrieve the <i>Bushing</i> Component	11-7
Placement Constraints	11-8
Placing the <i>Bushing</i>	11-9
Create the Second Placement Constraint	11-11
Create the <i>Bracket</i> Assembly Model	11-13
Base Component	11-14
Fully Constraining the <i>Base-Plate</i> to the Assembly	11-15
Retrieve the <i>Bracket-Bushing</i> Subassembly	11-16
Placing the <i>Bracket-Bushing</i> Subassembly	11-16
Placing the <i>Cap Screws</i>	11-22
Exploding the Assembly	11-24
Bi-directional Associative Functionality	11-27
Create a Drawing of the Assembly Model	11-29
Create a Parts List	11-32
Use the Repeat Region Command	11-34
Add the Balloon Callouts	11-38
Review Questions	11-40
Exercises	11-41

## Chapter 12

### Advanced Assembly Modeling and Animation

Introduction	12-2
Joint Connections	12-3
Basic Joint Connections	12-4
Pin	12-4
Slider	12-4
Cylinder	12-5
Planar	12-5
Servo Motor	12-6
Bodies	12-6
Ground Body	12-6
The <i>Crank-Slider</i> Assembly	12-7
Creating the Required Parts	12-7
(1) <i>End Cap</i>	12-7
(2) <i>Connecting Rod</i>	12-8
(3) <i>Base Block</i>	12-9
(4) <i>Crank Shaft</i>	12-10

---

(5) <i>Piston</i>	12-11
Create the Crank Slider Assembly Model	12-12
Base Component	12-13
Assembling the <i>Crank Shaft</i>	12-14
Assembling the <i>Connecting Rod</i>	12-16
Assembling the <i>Piston</i>	12-18
Assembling the <i>End Cap</i>	12-20
Starting the Creo Animation Module	12-21
Defining an Animation in Creo Animation	12-22
The Basic Animation Body Definitions and Display Controls	12-23
Creating the Animation Using the Key Frame Approach	12-24
Adjusting the Time Domain of the Animation	12-29
Creating another Animation by using a Servo Motor	12-30
Output the Animation as a Video file	12-32
Conclusion	12-33
Review Questions	12-34
Exercises	12-35

## **APPENDIX**

## **INDEX**