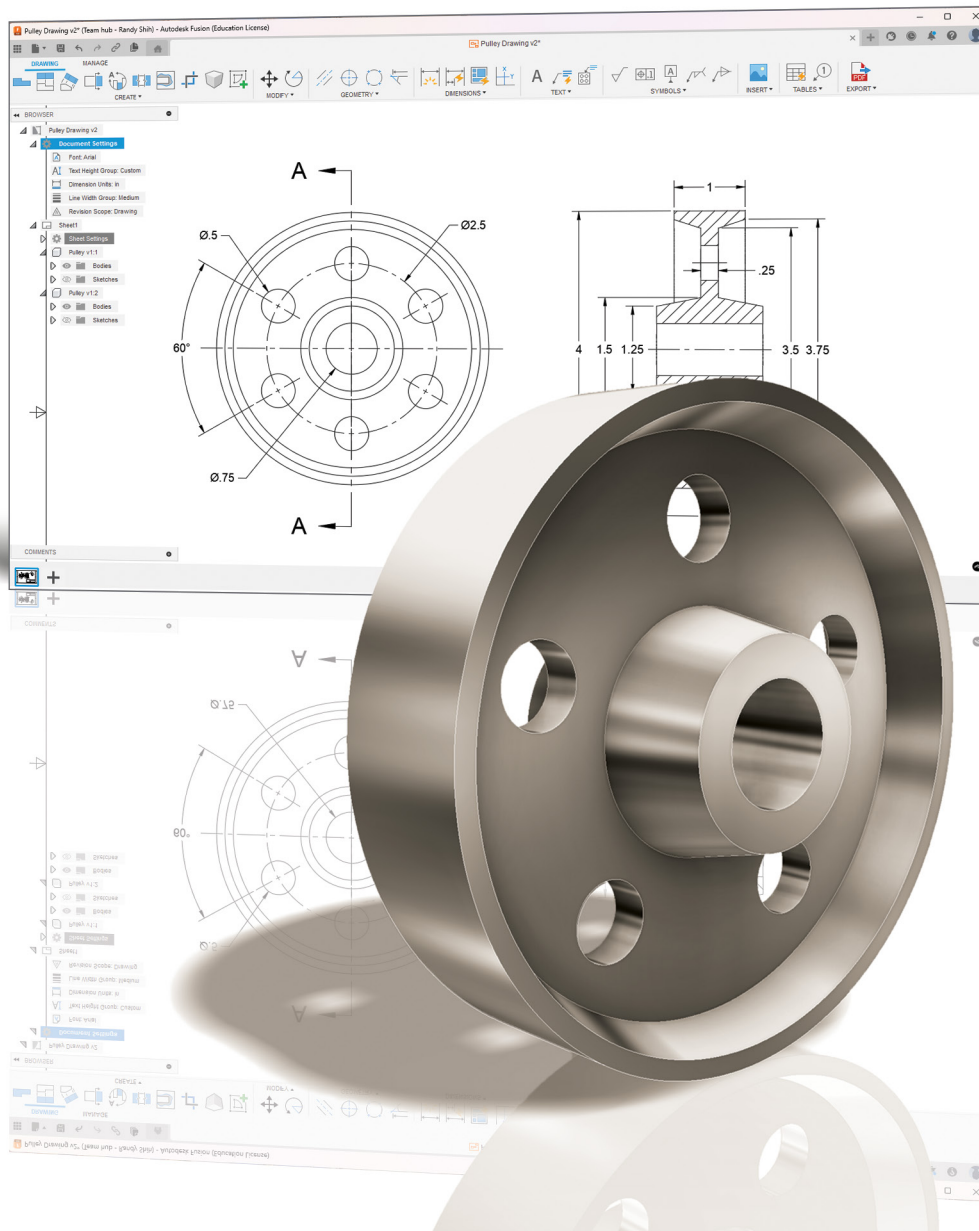


# Parametric Modeling with Autodesk® Fusion™

Spring 2025 Edition



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

<b>Preface</b>	i
<b>Acknowledgments</b>	ii
<b>Table of Contents</b>	iii

## **Chapter 1**

### **Introduction - Getting Started**

Introduction	1-2
Development of Computer Geometric Modeling	1-2
Feature-Based Parametric Modeling	1-6
Getting Started with Autodesk Fusion	1-7
The Autodesk Fusion Quick Setup	1-8
The Autodesk Fusion Screen Layout	1-9
Quick Access Toolbar	1-10
Ribbon Tool Panels	1-10
Display Control Panel	1-10
View Cube	1-10
Comments Panel	1-10
Mouse Buttons	1-11
[Esc] - Canceling Commands	1-11
Autodesk Fusion Help System	1-12
Autodesk Fusion Data Management	1-13
Browser Access to Fusion	1-16

## **Chapter 2**

### **Parametric Modeling Fundamentals**

Introduction	2-2
The Adjuster Design	2-3
Starting Autodesk Fusion	2-3
The Autodesk Fusion Screen Layout	2-4
Units Setup	2-5
Y-Up Orientation Vs Z-Up Orientation	2-6
Sketch Plane – It is an XY Monitor, but an XYZ World	2-7
Creating Rough Sketches	2-9
Step 1: Creating a Rough Sketch	2-10
Graphics Cursors	2-10
Geometric Constraint Symbols	2-11
Step 2: Apply/Modify Constraints and Dimensions	2-12
Dynamic Viewing Functions – Zoom and Pan	2-15
Modifying the Dimensions of the Sketch	2-15
Step 3: Completing the Base Solid Feature	2-16
Dynamic Rotation of the 3D Object - Free Orbit	2-18
Dynamic Viewing - Quick Keys	2-20

Viewing Tools – Display Control Bar and View Cube	2-21
Display Modes	2-23
Orthographic vs. Perspective	2-23
Step 4-1: Adding an Extruded Feature	2-24
Step 4-2: Adding a Cut Feature	2-28
Step 4-3: Adding another Cut Feature	2-31
Save the Model	2-35
Review Questions	2-36
Exercises	2-37

## **Chapter 3**

### **Constructive Solid Geometry Concepts**

Introduction	3-2
Binary Tree	3-3
The Locator Design	3-4
Modeling Strategy - CSG Binary Tree	3-5
Starting Autodesk Fusion	3-6
Model Dimensions Format	3-7
Base Feature	3-8
Modify existing Dimensions	3-11
Reposition Dimensions	3-11
Using the Measure Tool	3-12
Completing the Base Solid Feature	3-14
Creating the Next Solid Feature	3-15
Creating a CUT Feature	3-19
Creating a Placed Feature	3-22
Creating a Rectangular Cut Feature	3-24
Save the Model	3-26
Review Questions	3-27
Exercises	3-28

## **Chapter 4**

### **Model History Tree**

Introduction	4-2
The Saddle Bracket Design	4-3
Starting Autodesk Fusion	4-3
Modeling Strategy	4-4
The Autodesk Fusion Browser and Timeline Control	4-5
Create the Base Feature	4-5
Create the Second Solid Feature	4-8
Create a 2D Sketch	4-9
Renaming the Part Features	4-11
History Based Modifications to the Model	4-12
Adding a Placed Feature	4-13
Create a Rectangular Cut Feature	4-14

A Design Change - Edit the 2D sketches	4-16
Edit the Sketch through the Timeline Control	4-18
Direct Part Modifications	4-19
Assigning and Calculating the Associated Physical Properties	4-21
Review Questions	4-23
Exercises	4-24

## **Chapter 5**

### **Parametric Constraints Fundamentals**

CONSTRAINTS and RELATIONS	5-2
Create a Simple Triangular Plate Design	5-2
Fully Constrained Geometry	5-3
Starting Autodesk Fusion	5-3
Display and Hide Existing Constraints	5-4
Autodesk Fusion Geometric Constraints	5-5
Over-Constraining and Driven Dimensions	5-9
Delete Existing Constraints	5-10
Add the Location Dimensions	5-11
Parametric Relations	5-16
Dimensional Values and Dimensional Variables	5-19
Parametric Equations	5-20
Viewing the Established Parameters and Relations	5-21
Use the Measure Tools	5-23
Review Questions	5-26
Exercises	5-27

## **Chapter 6**

### **Geometric Construction Tools**

Introduction	6-2
The Gasket Design	6-2
Modeling Strategy	6-3
Starting Autodesk Fusion	6-4
Create the first 2D Sketch	6-5
Edit the Sketch by Dragging the Sketched Entities	6-8
Adding Dimensions and Constraints	6-9
Use the Trim and Extend Commands	6-10
Create Fillets and Completing the Sketch	6-12
Fully Constrained Geometry	6-13
Profile Sketch	6-14
Redefine the Sketch and Profile	6-15
Create an Offset Cut Feature	6-18
Review Questions	6-22
Exercises	6-23

## Chapter 7

### Parent/Child Relationships and the BORN Technique

Introduction	7-2
The BORN Technique	7-2
The U-Bracket Design	7-3
Apply the BORN Technique	7-4
Create the 2D Sketch for the Base Feature	7-6
Create the First Extrude Feature	7-9
The Implied Parent/Child Relationships	7-10
Create the Second Solid Feature	7-10
Create the Rectangular Cut Feature	7-14
The Second Cut Feature	7-15
Examine the Parent/Child Relationships	7-17
Modify a Parent Dimension	7-18
A Design Change	7-19
Feature Suppression	7-20
Redefine the sketch plane of the Center_Drill Feature	7-21
Suppress the Rect_Cut Feature	7-22
Create a Circular Cut Feature	7-23
A Flexible Design Approach	7-25
View and Edit Material Properties	7-26
Predefined keyboard and mouse shortcuts in Fusion	7-29
Review Questions	7-30
Exercises	7-31

## Chapter 8

### Part Drawings and Associative Functionality

Drawings from Parts and Associative Functionality	8-2
Starting Autodesk Fusion	8-3
Drawing Mode - 2D Paper Space	8-3
Add a Base View	8-5
Create Projected Views	8-6
Adjust the View Scale	8-7
Repositioning Views	8-7
Add and adjust Center Marks	8-8
Add and adjust Centerline Bisectors	8-10
Dimensioning the Drawing	8-11
Repositioning Dimensions and 2D views	8-13
Drawing Dimensions Format	8-14
Complete the Drawing Sheet	8-15
Associative Functionality – Modifying Feature Dimensions	8-16
Review Questions	8-20
Exercises	8-21

## **Chapter 9**

### **Datum Features and Auxiliary Views**

Work Features	9-2
Auxiliary Views in 2D Drawings	9-2
The Rod-Guide Design	9-2
Modeling Strategy	9-3
Starting Autodesk Fusion	9-4
Creating the Base Feature	9-4
Create an Angled Work Plane	9-7
Create a 2D Sketch on the Work Plane	9-8
Use the Projected Geometry Option	9-8
Complete the Solid Feature	9-12
Create an Offset Work Plane	9-13
Create another Cut Feature Using the Work Plane	9-14
Start a New 2D Drawing	9-16
Add a Base View	9-17
Create an Auxiliary View and an ISO View	9-18
Adjust the View Scale	9-20
Add and Adjust Center Marks	9-21
Add Dimensions	9-23
Complete the Drawing Sheet	9-24
Review Questions	9-26
Exercises	9-27

## **Chapter 10**

### **Introduction to 3D Printing**

What is 3D Printing?	10-2
Development of 3D Printing Technologies	10-3
Primary Types of 3D Printing Processes	10-6
Stereolithography	10-6
Fused Deposition Modeling & Fused Filament Fabrication	10-7
Laser Sintering / Laser Melting	10-8
Primary 3D Printing Materials for FDM and FFF	10-9
From 3D Model to 3D Printed Part	10-11
Starting Autodesk Fusion	10-12
Export the Design as an STL file	10-13
Using the 3D Printing software to create the 3D Print	10-16
Review Questions	10-24

## **Chapter 11**

### **Symmetrical Features in Designs**

Introduction	11-2
A Revolved Design: PULLEY	11-2
Modeling Strategy - A Revolved Design	11-3

Starting Autodesk Fusion	11-4
Creating the Base Feature	11-4
Create the Revolved Feature	11-8
Mirroring Features	11-9
Create a Pattern Leader Using Construction Geometry	11-12
Circular Pattern	11-18
Start a New 2D Drawing	11-20
Add a Base View	11-21
Create a Section View	11-22
Create an Isometric View	11-24
Add Center Mark on Patterned feature	11-25
Complete the Drawing Sheet	11-26
Review Questions	11-28
Exercises	11-29

## **Chapter 12**

### **Advanced 3D Construction Tools**

Introduction	12-2
A Thin-Walled Design: Dryer Housing	12-2
Modeling Strategy	12-3
Starting Autodesk Fusion	12-4
Creating the Base Feature	12-4
Create a Revolved Feature	12-7
Create Offset Work Planes	12-8
Start 2D Sketches on the Work Planes	12-9
Create a Lofted Feature	12-12
Create an Extruded Feature	12-14
Complete the Extruded Feature	12-16
Create 3D Rounds and Fillets	12-17
Create a Shell Feature	12-18
Create a Pattern Leader	12-19
Create a Rectangular Pattern	12-22
Create a Swept Feature	12-24
Define a Sweep Path	12-24
Define the Sweep Section	12-26
Complete the Swept Feature	12-27
Review Questions	12-29
Exercises	12-30

## **Chapter 13**

### **Sheet Metal Designs**

Sheet Metal Processes	13-2
Sheet Metal Modeling	13-4
K-Factor	13-5
The Actuator Bracket Design	13-6



Starting Autodesk Fusion	13-7
Sheet Metal Rules	13-8
Create the Base Feature of the Design	13-10
Create another Flat Face with the Flange command	13-13
Create a Bend Feature	13-16
Mirroring the Sheet Metal Part	13-19
Create another Flange Feature	13-20
Create a Cut Feature	13-21
Create the Associated Flat Pattern	13-23
Confirm the Flattened Length	13-25
Export the Flat Pattern as a DXF file	13-26
Review Questions	13-27
Exercise	13-28

## **Chapter 14**

### **Assembly Modeling - Joint & Animation**

Introduction	14-2
The Crank and Slider Assembly	14-2
Assembly Modeling Methodology	14-3
Autodesk Fusion Bodies and Components	14-4
The Assembly Joint Command	14-4
The Autodesk Fusion Joint Origin	14-8
Joint Creation order – “Put This, There.”	14-8
Starting Autodesk Fusion	14-9
Creating the Base Feature	14-9
Convert the first Body to the CS-Base Component	14-13
Create the next Component	14-14
Create the CS-Rod Component	14-16
Create the CS-Slider Component	14-17
Assemble the First Component	14-19
Assemble the Second Component	14-21
Constrained Move	14-24
Assemble the CS-Rod Component	14-24
Assemble the CS-Slider Component	14-27
Apply another Joint Connection	14-30
Animate the Assembly Model	14-32
Review Questions	14-33
Exercises	14-34

## **Appendix**

## **Index**