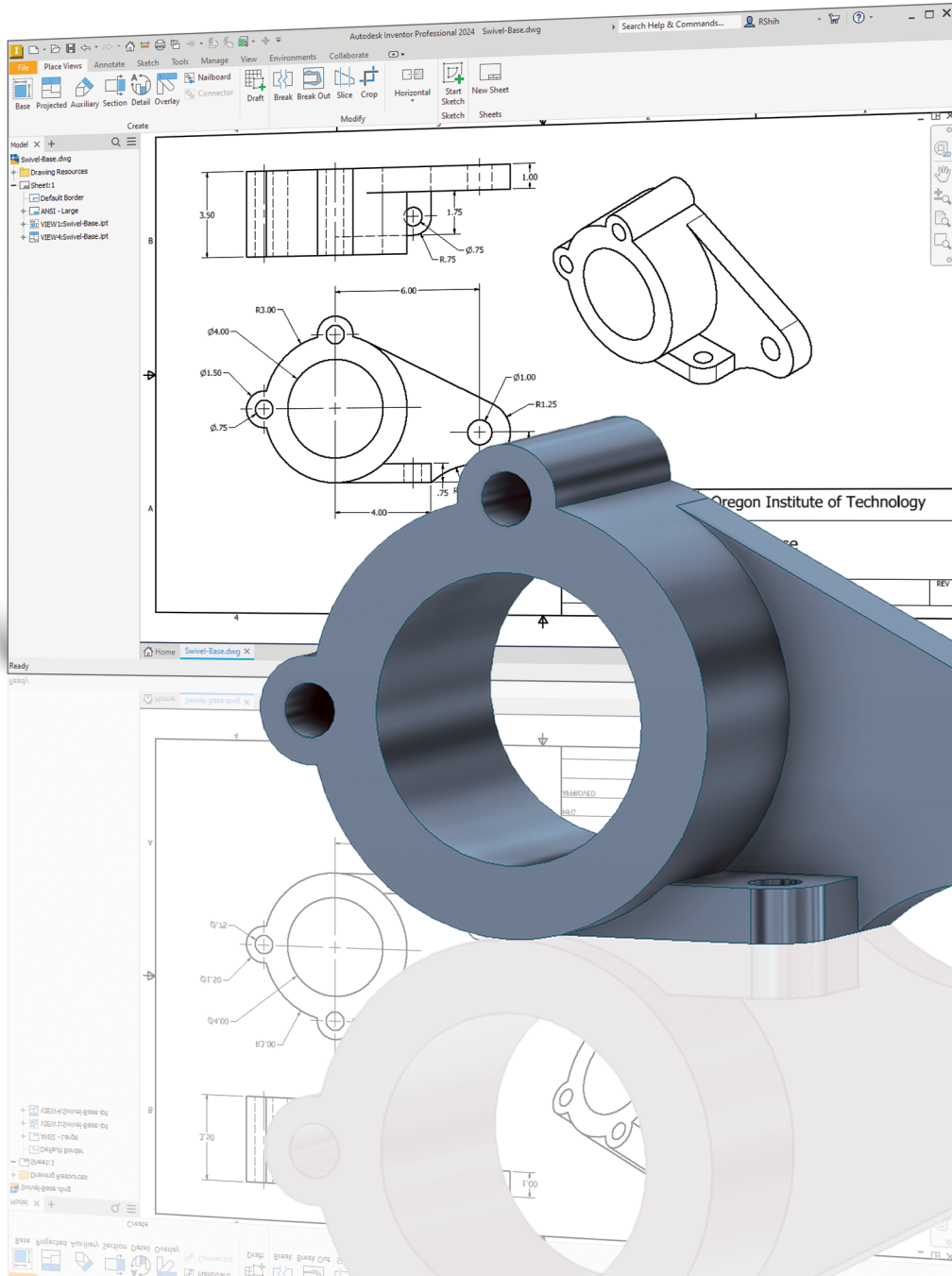


VIDEOS
Includes Extensive
Video Instruction

Parametric Modeling with Autodesk® Inventor® 2024



Randy H. Shih
Luke Jumper



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	i
Acknowledgments	ii
Table of Contents	iii
Autodesk Inventor Certified User Examination Overview	xiii

Chapter 1

Getting Started

Introduction	1-3
Development of Computer Geometric Modeling	1-3
Feature-Based Parametric Modeling	1-7
Getting Started with Autodesk Inventor	1-8
The Screen Layout and Getting Started	1-9
The New File Dialog Box and Units Setup	1-10
The Default Autodesk Inventor Screen Layout	1-11
File Menu	1-12
Quick Access Toolbar	1-12
Ribbon Tabs and Tool Panels	1-12
Online Help Panel	1-12
3D Model Toolbar	1-13
Graphics Window	1-13
Message and Status Bar	1-13
Mouse Buttons	1-14
[Esc] - Canceling Commands	1-14
Autodesk Inventor Help System	1-15
Data Management Using Inventor Project files	1-16
Set up of a New Inventor Project	1-17
The Content of the Inventor Project File	1-20
Leaving Autodesk Inventor	1-20

Chapter 2

Parametric Modeling Fundamentals

Introduction	2-3
The Adjuster Design	2-4
Starting Autodesk Inventor	2-4
The Default Autodesk Inventor Screen Layout	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
Isometric View	2-17
Dynamic Rotation of the 3D Block - Free Orbit	2-18
Dynamic Viewing - Quick Keys	2-20
Viewing Tools – Standard Toolbar	2-21
Display Modes	2-25
Orthographic vs. Perspective	2-25
Disable the Heads-Up Display Option	2-26
Step 4-1: Adding an Extruded Feature	2-27
Step 4-2: Adding a Cut Feature	2-31
Step 4-3: Adding another Cut Feature	2-34
Save the Model	2-36
Review Questions	2-38
Exercises	2-39

Chapter 3

Constructive Solid Geometry Concepts

Introduction	3-3
Binary Tree	3-4
The Locator Design	3-5
Modeling Strategy - CSG Binary Tree	3-6
Starting Autodesk Inventor	3-7
Base Feature	3-8
GRID Display Setup	3-9
Model Dimensions Format	3-12
Modifying the Dimensions of the Sketch	3-12
Repositioning Dimensions	3-13
Using the Measure Tools	3-14
Completing the Base Solid Feature	3-17
Creating the Next Solid Feature	3-18
Creating a Cut Feature	3-22
Creating a Placed Feature	3-25
Creating a Rectangular Cut Feature	3-27
Save the Model	3-29
Review Questions	3-30
Exercises	3-31

Chapter 4

Model History Tree

Introduction	4-3
--------------	-----

The Saddle Bracket Design	4-4
Starting Autodesk Inventor	4-4
Modeling Strategy	4-5
The Autodesk Inventor Browser	4-6
Creating the Base Feature	4-6
Adding the Second Solid Feature	4-9
Creating a 2D Sketch	4-10
Renaming the Part Features	4-12
Adjusting the Width of the Base Feature	4-13
Adding a Placed Feature	4-14
Creating a Rectangular Cut Feature	4-16
History-Based Part Modifications	4-17
A Design Change	4-18
Assigning and Calculating the Associated Physical Properties	4-21
Review Questions	4-23
Exercises	4-24

Chapter 5

Parametric Constraints Fundamentals

Constraints and Relations	5-3
Create a Simple Triangular Plate Design	5-3
Fully Constrained Geometry	5-4
Starting Autodesk Inventor	5-4
Displaying Existing Constraints	5-5
Applying Geometric/Dimensional Constraints	5-7
Over-Constraining and Driven Dimensions	5-11
Deleting Existing Constraints	5-12
Using the Auto Dimension Command	5-13
Constraint and Sketch Settings	5-18
Parametric Relations	5-19
Dimensional Values and Dimensional Variables	5-21
Parametric Equations	5-22
Viewing the Established Parameters and Relations	5-24
Saving the Model File	5-25
Using the Measure Tools	5-26
Review Questions	5-30
Exercises	5-31

Chapter 6

Geometric Construction Tools

Introduction	6-3
The Gasket Design	6-3
Modeling Strategy	6-4

Starting Autodesk Inventor	6-5
Create a 2D Sketch	6-6
Edit the Sketch by Dragging the Sketched Entities	6-8
Add Additional Constraints	6-10
Use the Trim and Extend Commands	6-11
The Auto Dimension Command	6-13
Create Fillets and Completing the Sketch	6-15
Fully Constrained Geometry	6-16
Profile Sketch	6-18
Redefine the Sketch and Profile	6-19
Create an Offset Cut Feature	6-23
Review Questions	6-26
Exercises	6-27

Chapter 7

Parent/Child Relationships and the BORN Technique

Introduction	7-3
The BORN Technique	7-3
The U-Bracket Design	7-4
Sketch Plane Settings	7-5
Apply the BORN Technique	7-6
Create the 2D Sketch for the Base Feature	7-8
Create the First Extrude Feature	7-11
The Implied Parent/Child Relationships	7-12
Create the Second Solid Feature	7-12
Create a Cut Feature	7-16
The Second Cut Feature	7-17
Examine the Parent/Child Relationships	7-19
Modify a Parent Dimension	7-20
A Design Change	7-21
Feature Suppression	7-22
A Different Approach to the Center_Drill Feature	7-23
Suppress the Rect_Cut Feature	7-25
Create a Circular Cut Feature	7-26
A Flexible Design Approach	7-28
View and Edit Material Properties	7-29
Review Questions	7-31
Exercises	7-32

Chapter 8

Part Drawings and 3D Model-Based Definition

Drawings from Parts and Associative Functionality	8-3
3D Model-Based Definition	8-4

Starting Autodesk Inventor	8-5
Drawing Mode - 2D Paper Space	8-5
Drawing Sheet Format	8-7
Using the Pre-Defined Drawing Sheet Formats	8-9
Activate, Delete, and Edit Drawing Sheets	8-10
Add a Base View	8-11
Create Projected Views	8-12
Adjust the View Scale	8-13
Repositioning Views	8-14
Display Feature Dimensions	8-15
Repositioning and Hiding Feature Dimensions	8-17
Add Additional Dimensions – Reference Dimensions	8-19
Add Center Marks and Center Lines	8-20
Complete the Drawing Sheet	8-23
Associative Functionality – Modifying Feature Dimensions	8-24
3D Model-Based Definition	8-27
Review Questions	8-35
Exercises	8-36

Chapter 9

Datum Features and Auxiliary Views

Work Features	9-3
Auxiliary Views in 2D Drawings	9-3
The Rod-Guide Design	9-3
Modeling Strategy	9-4
Starting Autodesk Inventor	9-5
Apply the BORN Technique	9-5
Creating the Base Feature	9-7
Create an Angled Work Plane	9-9
Create a 2D Sketch on the Work Plane	9-10
Use the Projected Geometry Option	9-10
Complete the Solid Feature	9-14
Create an Offset Work Plane	9-15
Create another Cut Feature Using the Work Plane	9-16
Start a New 2D Drawing	9-18
Add a Base View	9-19
Create an Auxiliary View	9-20
Display Feature Dimensions	9-22
Adjust the View Scale	9-24
Retrieving Dimensions in the Auxiliary View	9-25
Add Center Marks and Center Lines	9-27
Complete the Title Block with iProperties	9-30
Edit the Isometric view	9-32
Review Questions	9-33
Exercises	9-34

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
Primary 3D Printing Materials for FDM and FFF	10-9
From 3D Model to 3D Printed Part	10-11
Starting Autodesk Inventor	10-12
Export the Design as an STL File	10-13
Using the 3D Printing Software to Create the 3D Print	10-16
Questions	10-23

Chapter 11

Symmetrical Features in Designs

Introduction	11-3
A Revolved Design: Pulley	11-3
Modeling Strategy - A Revolved Design	11-4
Starting Autodesk Inventor	11-5
Set Up the Display of the Sketch Plane	11-5
Creating the 2D Sketch for the Base Feature	11-6
Create the Revolved Feature	11-10
Mirroring Features	11-11
Create a Pattern Leader Using Construction Geometry	11-13
Circular Pattern	11-18
Examine the Design Parameters	11-20
Drawing Mode – Defining a New Border and Title Block	11-20
Create a Drawing Template	11-24
Create the Necessary Views	11-25
Retrieve Model Annotations – Features Option	11-28
Associative Functionality – A Design Change	11-30
Add Centerlines to the Pattern Feature	11-32
Complete the Drawing	11-33
Additional Title Blocks	11-36
Review Questions	11-38
Exercises	11-39

Chapter 12

Advanced 3D Construction Tools

Introduction	12-3
A Thin-Walled Design: Dryer Housing	12-3
Modeling Strategy	12-4
Starting Autodesk Inventor	12-5

Set Up the Display of the Sketch Plane	12-5
Create the 2D Sketch for the Base Feature	12-6
Create a Revolved Feature	12-9
Create Offset Work Planes	12-10
Start 2D Sketches on the Work Planes	12-11
Create a Lofted Feature	12-14
Create an Extruded Feature	12-16
Complete the Extruded Feature	12-18
Create 3D Rounds and Fillets	12-19
Create a Shell Feature	12-20
Create a Pattern Leader	12-21
Create a Rectangular Pattern	12-24
Create a Swept Feature	12-26
Define a Sweep Path	12-26
Define the Sweep Section	12-28
Complete the Swept Feature	12-30
Review Questions	12-32
Exercises	12-33

Chapter 13

Sheet Metal Designs

Sheet Metal Processes	13-3
Sheet Metal Modeling	13-5
K-Factor	13-6
The Actuator Bracket Design	13-7
Starting Autodesk Inventor	13-8
Sheet Metal Defaults	13-9
Create the Base Face Feature of the Design	13-12
Using the Flange Command	13-15
Mirroring Features	13-19
Create a Cut Feature	13-20
Create a Fold Feature	13-22
Create the Associated Flat Pattern	13-25
Confirm the Flattened Length	13-26
Create a 2D Sheet Metal Drawing	13-27
Review Questions	13-34
Exercises	13-35

Chapter 14

Assembly Modeling - Putting It All Together

Introduction	14-3
Assembly Modeling Methodology	14-4
The Shaft Support Assembly	14-5

Additional Parts	14-5
(1) Collar	14-5
(2) Bearing	14-6
(3) Base-Plate	14-6
(4) Cap-Screw	14-7
Starting Autodesk Inventor	14-8
Placing the First Component	14-9
Placing the Second Component	14-10
Degrees of Freedom and Constraints	14-11
Assembly Constraints	14-12
Apply the First Assembly Constraint	14-15
Apply a Second Mate Constraint	14-16
Constrained Move	14-17
Apply a Flush Constraint	14-18
Placing the Third Component	14-20
Applying an Insert Constraint	14-21
Assemble the Cap-Screws	14-22
Exploded View of the Assembly	14-23
Editing the Components	14-25
Adaptive Design Approach	14-26
Delete and Re-apply Assembly Constraints	14-30
Set up a Drawing of the Assembly Model	14-32
Creating a Parts List	14-34
Edit the Parts List	14-35
Change the Material Type	14-37
Add the Balloon Callouts	14-39
Completing the Title Block Using the iProperties option	14-39
Bill of Materials	14-41
(a) BOM from Parts List	14-41
(b) BOM from Assembly Model	14-42
Review Questions	14-43
Exercises	14-44

Chapter 15

Content Center and Basic Motion Analysis

Introduction	15-3
The Crank-Slider Assembly	15-4
Create the Required Parts	15-4
Starting Autodesk Inventor	15-6
Placing the First Component	15-7
Placing the Second Component	15-8
Apply the Assembly Constraints	15-9
Apply a Second Mate Constraint	15-10
Constrained Move	15-11
Place the Third Component	15-11

Assemble the CS-Rod Part	15-14
Make a Copy of the Pin	15-15
Assemble the CS-Slider Part	15-16
Add an Angle Constraint to Fully Constrain the Assembly	15-21
Interference Analysis	15-23
Basic Motion Analysis	15-24
3D Grip Editing the CS-Slider Part	15-27
Review Questions	15-31
Exercises	15-32

Chapter 16

2D Design Reuse, Collision and Contact

Introduction	16-3
The Geneva CAM Assembly	16-4
Download the Geneva-Wheel DWG File	16-4
Opening AutoCAD DWG File in Inventor	16-5
Switch to the AutoCAD DWG Layout	16-6
2D Design Reuse	16-8
Complete the Imported Sketch	16-12
Create the First Solid Feature	16-14
Create a Mirrored Feature	16-15
Circular Pattern	16-16
Complete the Geneva Wheel Design	16-17
Additional Parts	16-18
Start a New Assembly	16-20
Placing the Second Component	16-21
The Assembly Joint Command	16-22
Create a Joint Connection	16-23
Constrained Move	16-24
Placing a Copy of the Geneva-Driver Part	16-24
Create a Second Joint Connection	16-25
Assemble the Geneva-Pin Part	16-26
Repositioning the Pieces	16-28
Animation with Drive Tool	16-29
Use the Inventor Contact Solver	16-31
Constrained Move with Contact Solver	16-33
Review Questions	16-34
Exercises	16-35

Chapter 17

Introduction to Stress Analysis

Introduction	17-2
Problem Statement	17-4

Preliminary Analysis	17-4
• Maximum Normal Stress	17-4
• Maximum Displacement	17-5
Finite Element Analysis Procedure	17-6
Create the Autodesk Inventor Part	17-7
Create the 2D Sketch for the Plate	17-7
Assigning the Material Properties	17-10
Switch to the Stress Analysis Module	17-11
Apply Constraints and Loads	17-14
Create a Mesh and Run the Solver	17-16
Refinement of the FEA Mesh – Global Element Size	17-18
Refinement of the FEA Mesh – Local Element Size	17-20
Comparison of Results	17-23
Create an HTML Report	17-24
Geometric Considerations of Finite Elements	17-25
Conclusion	17-26
Summary of Modeling Considerations	17-26
Review Questions	17-27
Exercises	17-28

Appendix

Index