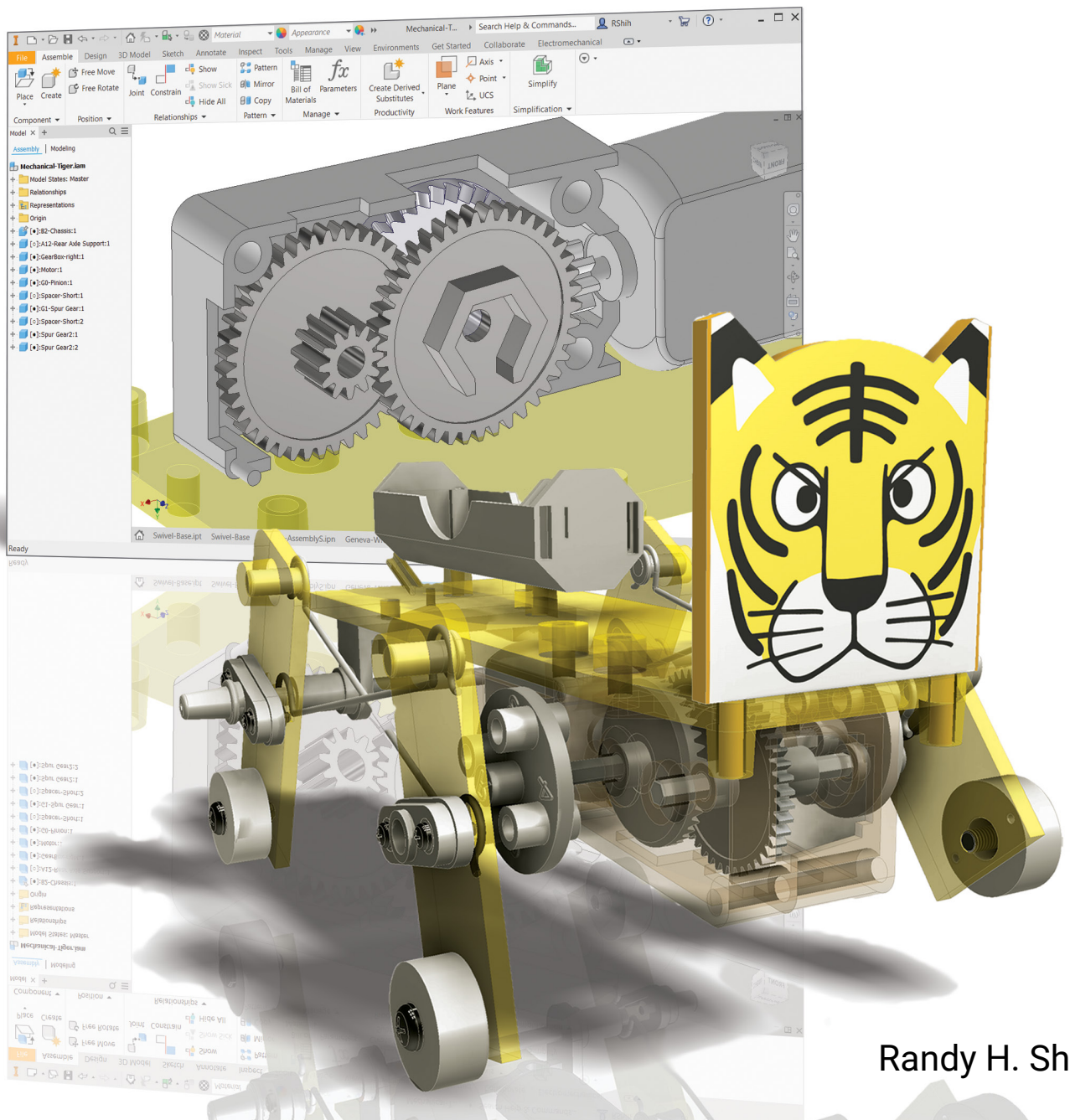


Learning Autodesk® Inventor® 2022

Modeling, Assembly and Analysis



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	i
Acknowledgments	ii
Table of Contents	iii
Chapter 1	
Getting Started	
Introduction	1-2
Development of Computer Geometric Modeling	1-2
Feature-Based Parametric Modeling	1-6
Getting Started with Autodesk Inventor	1-7
The Screen Layout and Getting Started Toolbar	1-8
The New File Dialog Box and Units Setup	1-9
The Default Autodesk Inventor Screen Layout	1-10
File Menu	1-11
Quick Access Toolbar	1-11
Ribbon Tabs and Tool Panels	1-11
Online Help Panel	1-11
3D Model Toolbar	1-12
Graphics Window	1-12
Message and Status Bar	1-12
Mouse Buttons	1-13
[Esc] - Canceling Commands	1-13
Autodesk Inventor Help System	1-14
Data Management Using Inventor Project Files	1-15
Setup of a New Inventor Project	1-16
The Content of the Inventor Project File	1-19
Leaving Autodesk Inventor	1-19
Chapter 2	
Parametric Modeling Fundamentals	
Introduction	2-2
The Tiger Head Design	2-3
Starting Autodesk Inventor	2-3
The Default Autodesk Inventor Screen Layout	2-5
Sketch Plane – It is an XY Monitor, but an XYZ World	2-6
Creating Rough Sketches	2-8
Step 1: Creating a Rough Sketch	2-9
Graphics Cursors	2-9
Geometric Constraint Symbols	2-10
Step 2: Apply/Modify Constraints and Dimensions	2-11
Dynamic Viewing Functions – Zoom and Pan	2-15
Modifying the Dimensions of the Sketch	2-15

Delete an Existing Geometry of the Sketch	2-16
Using the 3-Point Arc Command	2-17
Step 3: Completing the Base Solid Feature	2-19
Dynamic Rotation of the 3D Block - Free Orbit	2-20
Dynamic Viewing - Quick Keys	2-22
Viewing Tools – Standard Toolbar	2-23
Display Modes	2-27
Orthographic vs. Perspective	2-27
Disable the Heads-Up Display Option	2-28
Step 4-1: Adding an Extruded Feature	2-29
Step 4-2: Adding a Cut Feature	2-34
Step 5: Adding Additional Features	2-37
Using the Decal Command	2-40
Save the Model	2-43
Review Questions	2-44
Exercises	2-45

Chapter 3

CSG Concepts and Model History Tree

Introduction	3-2
Binary Tree	3-3
Model History Tree	3-4
The A6-Knee Part	3-5
Starting Autodesk Inventor	3-5
Modeling Strategy	3-6
The Autodesk Inventor Browser	3-7
Create the Base Feature	3-7
Add the Second Solid Feature	3-12
Create a 2D Sketch	3-13
Rename the Part Features	3-15
Adjusting the Dimensions of the Base Feature	3-15
History-Based Part Modifications	3-17
Add a Placed Feature	3-19
Set up the Auto-project options	3-21
Using the Offset Command to Create a Feature	3-22
Add another Hole Feature	3-25
Assigning and Calculating the Associated Physical Properties	3-27
Review Questions	3-30
Exercises	3-31

Chapter 4

Parametric Constraints Fundamentals

Constraints and Relations	4-2
Create a Simple Triangular Plate Design	4-2

Fully Constrained Geometry	4-3
Starting Autodesk Inventor	4-3
Displaying Existing Constraints	4-4
Applying Geometric/Dimensional Constraints	4-6
Over-Constraining and Driven Dimensions	4-10
Deleting Existing Constraints	4-11
Using the Auto Dimension Command	4-12
Constraint and Sketch Settings	4-17
The BORN Technique	4-18
Sketch Plane Settings	4-18
Apply the BORN Technique	4-20
Create the 2D Sketch for the Base Feature	4-22
Parametric Relations	4-23
Viewing the Established Parameters and Relations	4-25
Sketches vs. Profiles	4-28
Modify the Profile	4-31
Extrusion with the Taper Angle Option	4-32
A Profile Containing Multiple Closed Regions	4-35
Add a Feature Using Existing Geometry	4-37
Save the Model File	4-38
Using the Measure Tools	4-39
The Boot Part	4-43
Review Questions	4-46
Exercises	4-47

Chapter 5

Pictorials and Sketching

Engineering Drawings, Pictorials and Sketching	5-2
Isometric Sketching	5-7
Isometric Sketching Exercises	5-9
Oblique Sketching	5-19
Oblique Sketching Exercises	5-20
Perspective Sketching	5-26
Autodesk Inventor Orthographic vs. Perspective	5-27
One-point Perspective	5-28
Two-point Perspective	5-29
Perspective Sketching Exercises	5-30
Review Questions	5-36
Exercises	5-37

Chapter 6

Symmetrical Features and Part Drawings

Drawings from Parts and Associative Functionality	6-2
The A12- Rear Axle Support Design	6-3

Starting Autodesk Inventor	6-3
Modeling Strategy	6-4
Set Up the Display of the Sketch Plane	6-5
Creating the Base Feature	6-6
Creating a Symmetrical Cut Feature	6-9
Using the Projected Geometry Option	6-11
Creating a Revolved Feature	6-11
Create another Extrude Feature	6-14
Create a Cut Feature	6-17
Create a Mirrored Feature	6-19
Drawing Mode – 2D Paper Space	6-20
Drawing Sheet Format	6-21
Using the Pre-defined Drawing Sheet Formats	6-23
Activate, Delete and Edit Drawing Sheets	6-24
Add a Base View	6-25
Create Projected Views	6-26
Adjust the View Scale	6-27
Reposition Views	6-28
Display Feature Dimensions	6-29
Reposition and Hide Feature Dimensions	6-31
Add Additional Dimensions – Reference Dimensions	6-32
Add Center Marks and Center Lines	6-33
Complete the Drawing Sheet	6-35
Associative Functionality – Modify Feature Dimensions	6-36
Review Questions	6-39
Exercises	6-40

Chapter 7

Datum Features in Designs

Work Features	7-2
The B2-Chassis Part	7-2
Modeling Strategy	7-3
Starting Autodesk Inventor	7-4
Apply the BORN Technique	7-4
Create the Base Feature	7-6
Create the Second Extruded Feature	7-8
Create a Tapered Extruded Feature	7-10
Create an Offset Work Plane	7-11
Create a Revolved Feature	7-12
Create an Angled Work Plane	7-15
Create another Offset Work Plane	7-16
Create a 2D Sketch on Work Plane3	7-17
Complete the Solid Feature	7-19
Quick Change of the Appearance of the Solid Model	7-20
The Crank Right Part	7-21

The A10-Crank Left Part	7-23
The Motor	7-25
The A1-Axle End Cap Part	7-28
The Hex Shaft Collar Part	7-29
The A8-Rod Pin Part	7-33
Review Questions	7-34
Exercises	7-35

Chapter 8

Gear Generator and Content Center

Introduction to Gears	8-2
Spur Gear Nomenclatures	8-4
Basic Involute Tooth Profile	8-6
Gear Ratio	8-7
Starting Autodesk Inventor	8-9
The Inventor Spur Gear Generator	8-10
Modify the Generated Gears	8-14
Complete the Solid Feature	8-16
Create a Mirrored Feature	8-17
Import the Profile of the Pinion Gear	8-18
Create the G3-Spur Gear Part	8-23
Create another Spur Gear Set	8-26
Create the G0-Pinion Part	8-27
Start a New Part File	8-29
Export/Import the Generated Gear Profile	8-33
Inventor Content Center	8-41
Start another Autodesk Inventor Assembly Model	8-42
Using the Content Center	8-43
Review Questions	8-46
Exercises	8-47

Chapter 9

Advanced 3D Construction Tools

Introduction	9-2
A Thin-Walled Design: Battery Case	9-2
Modeling Strategy	9-3
Starting Autodesk Inventor	9-4
Set Up the Display of the Sketch Plane	9-4
Create the Base Feature	9-5
Create a Cut Feature	9-7
Create a Shell Feature	9-11
Create a Cut Feature	9-12
Create another Extruded Feature	9-14
Create another Cut Feature	9-18

Mirrored Features	9-20
Create another Cut Feature	9-22
Create the Last Feature	9-24
A Thin-Wire Design: Linkage Rod	9-27
Start another Model	9-27
Create a Swept Feature	9-28
Create a Mirrored Feature	9-32
The Gear Box Right Part	9-33
The Gear Box Left Part	9-38
Review Questions	9-42
Exercises	9-43

Chapter 10

Planar Linkage Analysis Using GeoGebra

Introduction to Four-Bar Linkage	10-2
Introduction to GeoGebra	10-5
Hide the Display of Objects	10-14
Add a Slider Control	10-16
Use the Animate Option	10-19
Tracking the Path of a Point on the Coupler	10-20
Exercises	10-25

Chapter 11

Design Makes the Difference

Engineering Analysis – How does this work?	11-2
Identify the Six-bar Linkage of the Mechanical Tiger	11-4
Starting GeoGebra	11-6
Add a Slider Control	11-14
Create the Second Four-bar Mechanism	11-16
Using the Animate Option	11-20
Tracking the Paths of the Feet	11-21
Adjusting the Crank Length	11-23
The Jansen Mechanism	11-24
The Klann Mechanism	11-25
Exercises	11-27

Chapter 12

Assembly Modeling and Motion Analysis

Introduction	12-2
Assembly Modeling Methodology	12-3
The Mechanical Tiger Assembly	12-4
Additional Parts	12-4
Starting Autodesk Inventor	12-6

Create the Leg Subassembly	12-7
Place the Second Component	12-8
Degrees of Freedom and Constraints	12-9
Assembly Constraints	12-10
Apply the First Assembly Constraint	12-13
Apply a Second MATE Constraint	12-14
Constrained Move	12-15
Placing the Third Component	12-16
Apply an Insert Constraint	12-17
Apply a Flush Constraint	12-19
Edit Parts in the Assembly Mode	12-21
Assemble the Boot Part	12-25
Use the Content Center and Assemble Two Screws	12-27
Start the Main Assembly	12-29
Assemble the Gear Box Right Part	12-31
Assemble the Motor and the Pinion Gear	12-34
Assemble the G1 Gear	12-38
Animation with the Inventor Drive Constraint Tool	12-40
Assemble the G2 Gear and the G3 Gear	12-41
Assemble the Crank Parts	12-47
Assemble the Rear Shaft and Legs	12-51
Assemble the Linkage-Rods	12-53
Complete the Assembly Model	12-55
Record an Animation Movie	12-56
Conclusion	12-58
Summary of Modeling Considerations	12-58
Review Questions	12-59
Exercises	12-60

Index