

Learning SolidWorks® 2012

Modeling, Assembly and Analysis

Randy H. Shih

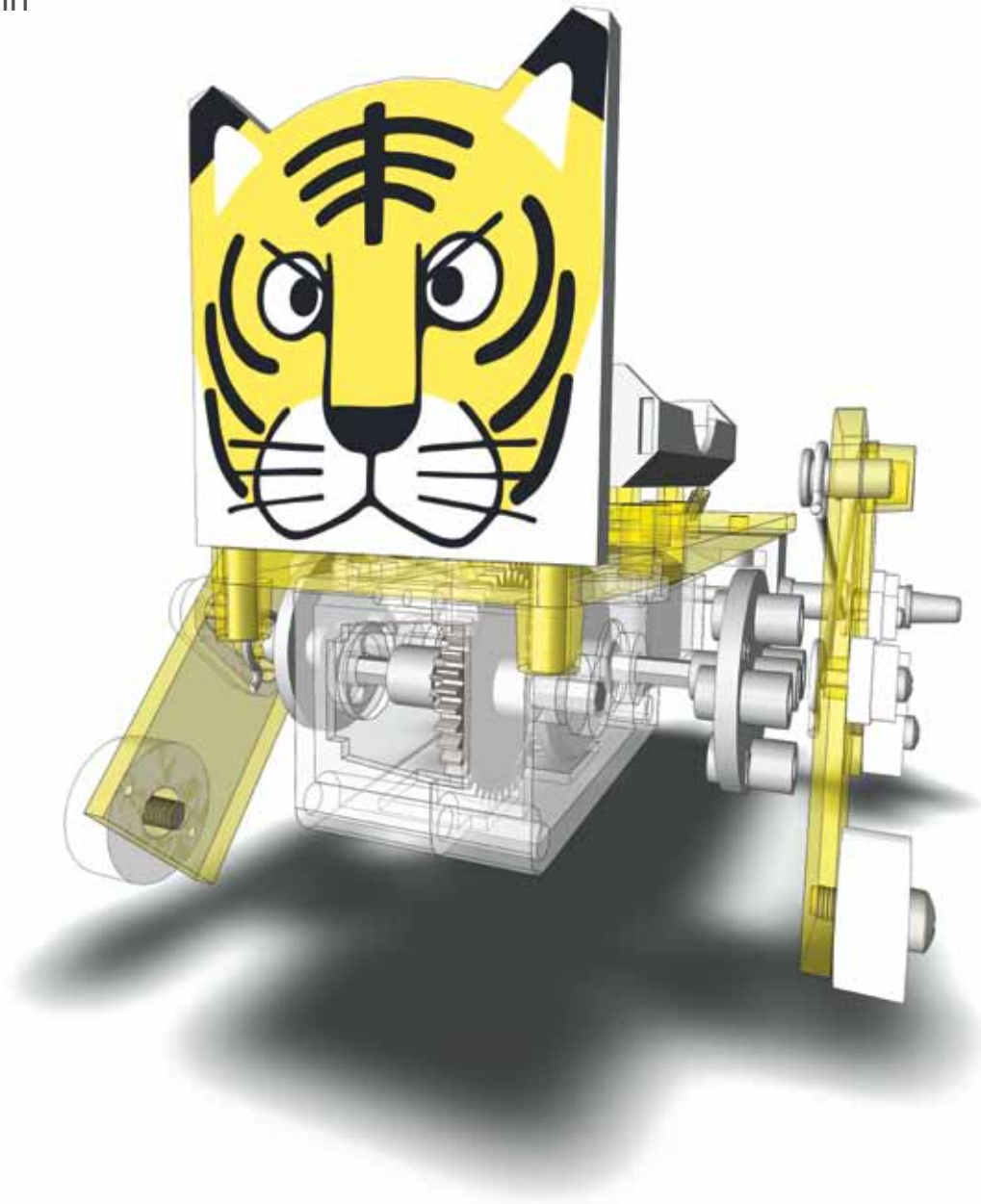


Table of Contents

Preface	i
Acknowledgments	ii
Table of Contents	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 SolidWorks	1-7
The Screen Layout and Getting Started Toolbar	1-8
Units Setup	1-9
SolidWorks Screen Layout	1-11
Menu Bar	1-11
Menu Bar Pull-down menus	1-12
Heads-up View Toolbar	1-12
Features Toolbar	1-12
Sketch Toolbar	1-12
Property Managers	1-13
Graphics area	1-14
Reference Triad	1-14
Origin	1-14
Confirmation Corner	1-14
Graphics Cursor or Crosshairs	1-14
Reference Triad	1-14
Message and Status Bar	1-14
Using the SolidWorks Command Manager	1-15
Mouse Buttons	1-16
[Esc] - Canceling Commands	1-16
SolidWorks Help System	1-17
Leaving SolidWorks	1-17
Creating a CAD files folder	1-18
Chapter 2	
Parametric Modeling Fundamentals	
Introduction	2-2
The Tiger Head Design	2-3
Starting SolidWorks	2-3
SolidWorks Screen Layout	2-4

Step 1: Determine/Setup the Base Solid	2-6
Sketching plane – It is an XY CRT, but an XYZ World	2-6
Creating a Rough Sketch	2-8
Step 2: Creating a Rough Sketch	2-9
Graphics Cursors	2-9
Geometric Constraint Symbols	2-10
Step 3: Apply/Modify Constraints and Dimensions	2-11
Dynamic Viewing Functions – <i>Zoom</i> and <i>Pan</i>	2-16
Delete an Existing Geometry of the Sketch	2-17
Using the 3-Point Arc Command	2-17
Step 4: Completing the Base Solid Feature	2-19
Isometric View	2-20
Rotation of the 3-D model – Rotate View	2-20
Rotation and Panning –Arrow Keys	2-22
Dynamic Viewing - Quick Keys	2-23
Viewing Tools – Heads-up View Toolbar	2-25
View Orientation	2-26
Display Style	2-27
Orthographic vs. Perspective	2-27
Customizing the Heads-Up View toolbar	2-27
Step 5-1: Adding an Extruded Feature	2-28
Step 5-2: Adding a Cut Feature	2-32
Step 6: Adding Additional Feature	2-34
Adding a Decal	2-36
Save the Model	2-39
Questions	2-40
Exercises	2-41

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 SolidWorks	3-5
Modeling Strategy - CSG Binary Tree	3-6
The <i>SolidWorks Feature Manager</i>	3-7
Base Feature	3-7
<i>Units</i> Setup	3-8
Create the Base Feature	3-9
Adding the Second Solid Feature	3-13
Renaming the Part Features	3-15
Adjust the Dimensions of the Base Feature	3-15
History-Based Part Modifications	3-18
Add a Placed Feature	3-19

Create an Offset Feature	3-22
Adding another Hole Feature	3-25
Assigning and Calculating the Associated Physical Properties	3-27
Questions	3-30
Exercises	3-31

Chapter 4

Parametric Constraints Fundamentals

DIMENSIONS and RELATIONS	4-2
Create a Simple Triangular Plate Design	4-2
Fully Constrained Geometry	4-3
Starting SolidWorks	4-3
Create a User-Defined Part Template	4-4
Start a New Model using the New Template	4-6
Display/Hide Applied Geometric relations	4-7
Applying Geometric/Dimensional Constraints	4-8
Geometric Editing with Drag and drop	4-12
Create Fully Constrained Geometry	4-14
Over-defining and Driven Dimensions	4-15
Delete the Fix Constraint	4-17
Use the Fully Define Sketch Tool	4-18
Constraint and Sketch Settings	4-16
Add additional Geometry	4-20
Relations Settings	4-24
Model the B3-Leg Part	4-25
Create the 2D Sketch for the Base Feature	4-26
Parametric Relations	4-27
Use the Equations Command	4-29
Complete the Base Feature	4-30
Sketches vs. Profiles	4-32
Redefine the Profile with Contour Selection	4-35
Extrusion with the Taper Angle option	4-36
A Profile containing Multiple Closed Regions	4-39
The Convert Entities Option	4-41
Add a Feature using Existing Geometry	4-41
Saving the Model File	4-43
Using the Measure Tools	4-43
Create a Metric Part Template	4-46
The <i>Boot</i> Part	4-48
Questions	4-51
Exercises	4-52

Chapter 5

Pictorials and Sketching

Engineering Drawings, Pictorials and Sketching	5-2
Isometric Sketching	5-7
Isometric Sketching Exercises	5-9
Oblique Sketching	5-10
Oblique Sketching Exercises	5-11
Perspective Sketching	5-12
SolidWorks Orthographic vs. Perspective	5-13
One-point Perspective	5-14
Two-point Perspective	5-15
Perspective Sketching Exercises	5-16
Questions	5-17
Exercises	5-18

Chapter 6

Symmetrical Features and Part Drawings

Drawings from Parts and Associative Functionality	6-2
The <i>A12- Rear Axle Support</i> Design	6-3
Starting SolidWorks	6-3
Modeling Strategy	6-4
Create the Base Feature	6-5
Create a Symmetric Cut Feature	6-8
Using the Projected Geometry Option	6-11
Create a Revolved Feature	6-11
Create another Extrude Feature	6-15
Create a Cut Feature	6-18
Create a Mirrored Feature	6-20
Drawing Mode – 2D Paper Space	6-21
The Drawing Sheet Properties	6-22
Adding a Base View	6-24
Adding Center Lines	6-26
Displaying Feature Dimensions	6-29
Adding Additional Dimensions – Reference Dimensions	6-31
Completing the Drawing Sheet	6-32
Associative Functionality – Modifying Feature Dimensions	6-34
Questions	6-37
Exercises	6-38

Chapter 7

Datum Features in Designs

Reference Features	7-2
The <i>B2-Chassis</i> Part	7-2
Modeling Strategy	7-3
Starting SolidWorks	7-4
Applying the BORN Technique	7-5
Create the Base Feature	7-6
Create the second Extrude Feature	7-8
Create a Tapered Extruded Feature	7-10
Create an Offset Reference Plane	7-11
Create a Revolved Feature	7-12
Create an Angled Reference Plane	7-15
Create another Offset Reference Plane	7-16
Create a 2D Sketch on Reference Plane ³	7-18
Change the Appearance of the Solid Model	7-22
Create another Metric Template	7-24
The <i>Crank Right</i> Part	7-25
The <i>A10-Crank Left</i> Part	7-29
The <i>Motor</i>	7-31
The <i>A1-Axle End Cap</i> Part	7-34
The <i>Hex Shaft with Collar</i> Part	7-35
The <i>A8-Rod Pin</i> Part	7-39
Questions	7-40
Exercises	7-41

Chapter 8

Gears and SolidWorks Design Library

Introduction to Gears	8-2
Spur Gear Nomenclatures	8-4
Basic Involute Tooth Profile	8-6
Gear Ratio	8-7
The SolidWorks Gear Toolbox	8-9
Starting SolidWorks	8-10
Open the SolidWorks Design Library	8-10
The SolidWorks Spur Gear Toolbox	8-12
Create a 42 Teeth Spur Gear	8-13
Modify the Generated Gears	8-14
Create a Mirrored Feature	8-18
Importing the Profile of the Pinion Gear	8-19
Complete the <i>G2-Spur Gear</i> Part	8-25
Create the <i>G3-Spur Gear</i> Part	8-27
Create the <i>G0-Pinion</i> Part	8-31
Start a New Part File	8-33

Export/Import the Generated Gear Profile	8-36
Create a Circular pattern	8-39
Questions	8-46
Exercises	8-47

Chapter 9

Advanced 3D Construction Tools

Introduction	9-2
A Thin-Walled Design: <i>Battery Case</i>	9-2
Modeling Strategy	9-3
Starting SolidWorks	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	9-16
Mirror the Last Feature	9-18
Create another Cut Feature	9-19
Complete the Model	9-20
A Thin-Wire Design: <i>Linkage Rod</i>	9-22
The Sweep Operation	9-22
Start a New Model	9-22
Completing the Swept Feature	9-27
Create a Mirrored Feature	9-28
The <i>Gear Box Right</i> Part	9-29
The <i>Gear Box Left</i> Part	9-34
Questions	9-38
Exercises	9-39

Chapter 10

Planar Linkage Analysis using GeoGebra

Introduction to Four-Bar Linkages	10-2
Introduction to GeoGebra	10-5
Turn off the Display of Objects	10-14
Adding a Slider Control	10-16
Using 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 <i>Mechanical Tiger</i>	11-4
Starting GeoGebra	11-6
Adding 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	11-33

Chapter 12

Assembly Modeling and Motion Analysis

Introduction	12-2
Assembly Modeling Methodology	12-3
The <i>Mechanical Tiger</i> Assembly	12-4
Additional Parts	12-4
Starting SolidWorks	12-6
Document Properties	12-7
Place the Second Component	12-8
Place the Second Component	12-8
Degrees of Freedom and Assembly Relations	12-10
Assembly Mates	12-11
Apply the First Assembly Mate	12-12
Apply a Second MATE Mate	12-14
Constrained Move	12-15
Place the Third Component	12-16
Applying a Coincident Mate	12-17
Apply an Anti-Aligned Mate	12-19
Edit Parts in the Assembly Mode	12-21
Assemble the <i>Boot</i> Part	12-25
Use the Design Library and Assemble Two Screws	12-27
Start the <i>Main Assembly</i>	12-29
Assemble the <i>Gear Box Right</i> Part	12-32
Assemble the <i>Motor</i> and the <i>Pinion Gear</i>	12-35
Assemble the <i>G1 Gear</i>	12-38
<i>SolidWorks</i> Motion Study	12-41
Assemble the <i>G2 Gear</i>	12-43
Assemble the <i>G3 Gear</i>	12-46
Assemble the Hex Shafts	12-48

Assemble the <i>Crank</i> Parts	12-51
Assemble the <i>Rear Shaft</i> and <i>Legs</i>	12-55
Assemble the <i>Linkage-Rods</i>	12-57
Complete the Assembly Model	12-62
Record an Animation Movie	12-63
Conclusion	12-64
Questions	12-65
Exercises	12-66

Index