

SOLIDWORKS® 2024 Advanced Techniques

Mastering Parts, Surfaces, Sheet Metal,
SimulationXpress, Top Down Assemblies,
Core & Cavity Molds



Paul Tran CSWE, CSWI

Visit the following websites to learn more about this book:



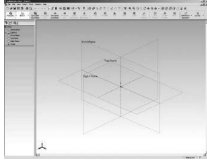
[amazon.com](https://www.amazon.com)

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

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

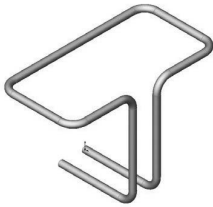
Table of Contents

Introduction SOLIDWORKS 2024 User Interface	21
The 3 references planes	22
The toolbars	22
The system feedback symbols	24
The status bar	24
2D sketch examples	25
3D feature examples	26



Advanced Modeling Topics

Chapter 1	Introduction to 3D Sketch	1-1
	Tools Needed	1-2
	Creating a 3D Sketch	1-3
	Completing the profile	1-4
	Adding dimensions	1-5
	Adding the sketch fillets	1-6
	Creating the swept feature	1-7
	Questions for review	1-8
	Exercise: Sweep with 3D Sketch	1-9
	Exercise: 3D Sketch & Planes	1-10
	Exercise: 3D Sketch & Composite Curve	1-17



Chapter 2	Plane Creation	2-1
	Advanced Topics	2-1
	Tools Needed	2-2
	Revolving the base	2-3
	Creating a tangent plane	2-4
	Creating a flat surface	2-5
	Extruding a cut	2-6
	Creating an at-angle plane	2-7
	Showing the sketches	2-8
	Creating a coincident plane	2-9
	Creating a parallel plane	2-10
	Creating a recess feature	2-11
	Creating an offset-distance plane	2-12
	Creating the bore holes	2-12
	Creating a perpendicular plane	2-13



Creating the side-grips	2-14
Creating a circular pattern of the grips	2-15
Creating a Mid-Plane	2-17
Adding fillets to all edges	2-19
Questions for Review	2-20
Exercise: Creating New Planes	2-22

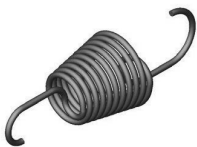
Chapter 3 Advanced Modeling 3-1

5/8” Spanner	3-1
Tools needed	3-2
Opening the spanner sketch document	3-3
Creating the transition sketch	3-4
Creating a new work plane	3-6
Creating the closed-end sketch	3-7
Extruding the closed-end feature	3-7
Adding a 12-sided polygon hole	3-8
Creating the recess profile	3-9
Mirroring the recessed feature	3-10
Adding fillets	3-11
Adding text	3-13
Extruding the text	3-14
Questions for Review	3-17
Exercise: Circular text wraps	3-19



Chapter 4 Sweep with Composite Curves 4-1

Helical Extension Spring	4-2
Tools needed	4-2
Converting to a helix	4-3
Creating a 2-degree plane	4-4
Sketching the large loop	4-5
Sketching the large hook	4-5
Creating a parallel plane	4-6
Creating a Composite Curve	4-8
Sweeping the profile along the path	4-11
Other spring examples	4-12
Questions for review	4-13
Exercise: Circular Spring – 180 deg.	4-14



Using Variable Pitch	4-17
Multi-Pitch Spring with Closed Ends	4-18
Tools Needed	4-18
Creating the base sketch	4-19
Creating a helix using variable pitch	4-19





Sweeping the profile along the path	4-21
Creating a trimmed sketch	4-22
Extruding a cut	4-22
Questions for Review	4-23
Exercise: Using Equation Driven Curve	4-24
Exercise: Projected Curve & Composite Curve	4-30

Chapter 5 Advanced Modeling with Sweep & Loft 5-1

Water Pump Housing 5-2



Tools Needed	5-2
Understanding the draft options	5-3
Extruding the base with draft	5-4
Sketching the upper inlet port	5-5
Revolving the upper inlet port	5-5
Adding fillets	5-6
Creating offset-distance planes	5-7
Creating a loft feature	5-10
Creating the mounting bosses	5-11
Sketching the rear inlet port	5-12
Revolving the rear inlet port	5-12
Adding face Fillets	5-13
Mirroring the rear inlet port	5-15
Shelling the part	5-16
Adding a rib	5-17
Mirroring the rib	5-18
Adding fillets	5-19
Exercise: Loft without guide curves	5-21
Exercise: Loft with guide curves	5-23

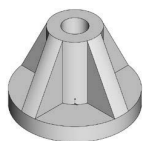


Chapter 6 Loft vs. Sweep 6-1

Water Meter Housing 6-2



Tools Needed	6-2
Sketching the loft profile	6-3
Constructing loft profiles / features	6-5
Creating the inlet feature	6-6
Constructing the centerline parameter	6-10
Creating the outlet loft feature	6-11
Shelling the part	6-13
Extruding the left / right brackets	6-14
Constructing the upper ring	6-15
Adding fillets	6-17
Adding chamfers	6-18
Questions for Review	6-19





Exercise: Loft	6-20
Exercise: Loft vs. boundary	6-23
Exercise: Mirror using two planes	6-26
Exercise: Tangent vs. curvature	6-27

Chapter 7

Loft with Guide Curves 7-1

Waved Washer 7-2

Tools Needed 7-2

Creating the construction profile 7-3

Creating an offset distance plane 7-4

Positioning the derived sketch 7-5

Creating a curve through reference points 7-5

Sketching the loft sections 7-7

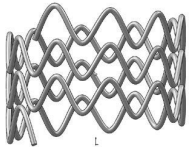
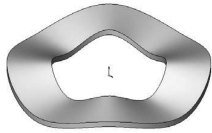
Creating the loft section using derived sketch 7-7

Creating a loft with guide curve 7-10

Hiding the construction sketches 7-11

Questions for review 7-12

Exercise: Using Curve Driven Pattern 7-13



Advanced Sweep 7-19

3D Sketch Sweep Path 7-20

Tools Needed 7-20

Creating the sweep path 7-21

Creating the sweep profile 7-22

Creating a sweep feature 7-23

Creating a circular Sketch pattern 7-25

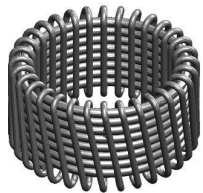
Converting to construction geometry 7-26

Creating a derived sketch 7-27

Creating a 3D sketch 7-28

Creating the sweep feature 7-32

Exercise: Using Curve Through Reference Points 7-33



Chapter 8

Using Surfaces 8-1

Advanced Modeling 8-2

Tools Needed 8-2

Constructing a new work plane 8-3

Sketching the 1st profile 8-3

Creating a surface-loft 8-6

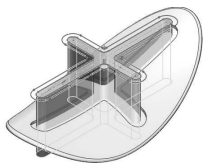
Splitting the surface 8-7

Deleting surfaces 8-8

Thickening the surface 8-9

Calculating the angle between the faces 8-10

Adding a full round fillet 8-12





Sketching the slot contours	8-13
Extruding cut the contour	8-15
Questions for Review	8-17



Lofted Surface	8-19
Creating new offset planes	8-19
Sketch the first profile	8-20
Selecting the loft profiles	8-21
Lofting between the profiles	8-22
Creating a revolved sketch	8-23
Copying the revolved surface	8-24
Trimming the base part	8-25
Hiding the surfaces	8-25
Filling the openings with surface-fill	8-26
Creating a Surface-Knit	8-29
Adding fillets	8-30
Creating a solid from the surface model	8-31
Removing the upper half	8-32
Questions for Review	8-36
Exercise: Loft & Delete Face	8-37



Chapter 9 Offset Surface & Ruled Surface 9-1







Tools Needed	9-2
Using offset & ruled surface options	9-3
Creating the base loft	9-4
Creating split lines	9-5
Creating offset surfaces	9-6
Creating a ruled surface	9-7
Knitting the 2 surfaces	9-8
Showing the solid body	9-10
Creating the surface cut	9-10
Exercise: Using Loft	9-13



Exercise: Advanced Surfacing Techniques	9-15
Exercise: Using Split	9-25
Creating a split line	9-25
Making an offset surface	9-26
Adding a ruled surface	9-26
Deleting a surface	9-27
Trimming surfaces	9-27
Knitting the surfaces	9-28
Adding fillets	9-29

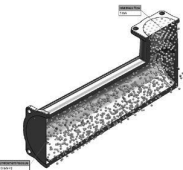


Chapter 10	Advanced Surfaces	10-1
	Computer Mouse	10-2
	Tools Needed	10-2
	Sketching the profiles	10-3
	Extruding a surface	10-4
	Trimming the surfaces	10-5
	Mirroring the surfaces	10-6
	Creating the lower sketches	10-7
	Creating a new plane	10-9
	Making a 3-Point Arc	10-9
	Creating a planar surface	10-12
	Creating a knit surface	10-13
	Using Filled Surfaces	10-15
	Patch with Curvature Controls	10-15
	Tools Needed	10-16
	Enabling the surfaces toolbar	10-17
	Creating a planar surface	10-18
	Creating a surface fill with tangent control	10-18
	Creating a surface fill with curvature control	10-20
	Knitting all surfaces	10-21
	Patch Types	10-22
	Questions for Review	10-24
	Using Ruled Surfaces	10-25
	Creating a split line	10-25
	Creating an offset surface	10-26
	Adding a ruled surface	10-26
	Knitting and cutting the surfaces	10-27
	Shelling the model	10-29
Chapter 11	Surfaces vs. Solid Modeling	11-1
	Safety Helmet	11-2
	Tools Needed	11-2
	Constructing the body of Helmet	11-3
	Creating a new work plane	11-4
	Sketching the sweep-profile	11-4
	Creating the sweep path	11-5
	Adding a planar surface	11-6
	Knitting the three surface bodies into one	11-6
	Creating a section view	11-7
	Adding an extruded cut feature	11-7
	Adding a revolve cut feature	11-9



Adding the side cut features	11-10
Creating the Cutout slot	11-11
Creating the sweep cut	11-13
Adding fillets	11-13
Exercise: Helmet_Surfaces	11-15
Exercise: Advanced Loft – Turbine Blades	11-25
Exercise: Advanced Sweep – Candle Holder	11-26
Advanced: Final Exam	11-33

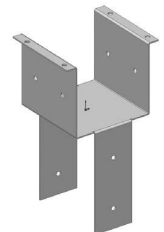
Chapter 12 SimulationXpress 12-1



Using the Analysis Wizard	12-2
Tools Needed	12-2
Starting SimulationXpress	12-3
Setting up the units	12-4
Adding a fixture	12-5
Applying a force	12-7
Selecting the material	12-8
Analyzing the model	12-9
Viewing the Results	12-10
Creating the report	12-12
Generating the eDrawings file	12-16
Questions for Review	12-19
Exercise 1: SimulationXpress: Force	12-20
Exercise 2: SimulationXpress: Pressure	12-21
Exercise: Using SOLIDWORKS FlowXpress	12-23

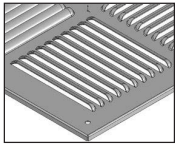
Sheet Metal Topics

Chapter 13 Sheet Metal Parts 13-1



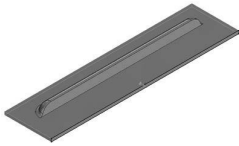
Post Cap	13-2
Tools Needed	13-2
Starting with the Base Profile	13-3
Extruding the Base Flange	13-3
Creating an Edge Flange	13-4
Editing the Edge Flange Profile	13-5
Viewing the Flat Pattern	13-6
Changing the Fixed Face	13-6
Creating a Sketch Bend	13-7
Adding Holes	13-9
Switching to the Flat Pattern	13-12

Using Sheet Metal Costing	13-13
Inputting the information	13-14
Setting the Baseline	13-15
Questions for Review	13-17



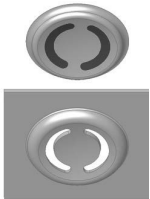
Vents **13-18**

Tools Needed	13-19
Sketching the first profile	13-20
Extruding the Base-Flange	13-21
Creating the Miter-Flanges	13-22
Flattening the Part	13-24
Creating a new Forming Tool	13-25
Other Rectangle Options	13-26
Extruding the Base	13-27
Building the louver body	13-27
Creating the Positioning Sketch	13-31
Saving the Form Tool	13-33
Applying the Form Tool	13-34
Positioning the Form Tool	13-35
Adding a mounting hole	13-36
Creating a Linear Pattern	13-37
Creating an Axis	13-38
Creating a Circular Pattern	13-39
Questions for Review	13-40



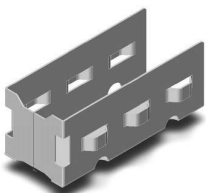
Chapter 14 Sheet Metal Forming Tools **14-1**

Button with Slots	14-2
Tools Needed	14-2
Creating the Base Block	14-3
Revolving the Body	14-4
Sketching Slot Profiles	14-5
Creating the Split Lines	14-7
Adding more fillets	14-9
Inserting the forming tool feature	14-9
Saving the forming tool	14-10
Questions for Review	14-12



Designing Sheet Metal Parts – Tool Holder **14-13**

Tools Needed	14-14
Starting with the Base Sketch	14-15
Creating an Edge Flange	14-16
Adding Cut features	14-17
Extruding a cut	14-17



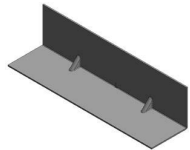


Using the Unfold Command	14-18
Using the Fold Command	14-20
Inserting a Sheet Metal Forming Tool	14-23
Locating the Bridge Lance	14-25
Creating the Linear Pattern of the Bridge Lance	14-26
Mirroring the Body	14-27
Adding Chamfers	14-30
Switching to the Flat Pattern	14-31
Questions for Review	14-32

Chapter 15 Sheet Metal Conversions 15-1



From IGES to SOLIDWORKS	15-2
Tools Needed	15-2
Opening an IGES Document	15-3
Creating the Rips	15-4
Inserting the Sheet Metal Parameters	15-5
Adding Fillets	15-6
Switching to a Flat Pattern	15-7
Questions for Review	15-8



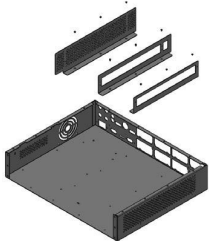
Sheet Metal Gussets	15-9
Opening a sheet metal document	15-9
Creating a gusset	15-9
Viewing the resulted gusset	15-11
Mirroring the gusset	15-12
Exercise: Hem & Vent Features	15-13



Flat Pattern Stent	15-16
Tools Needed	15-17
Converting to Sheet Metal	15-19
Unfolding the Part	15-19
Adding the Sketch Pattern	15-20
Folding the Part	15-22
Creating a new Configuration	15-23
Adding Fillets	15-23
Switching to Flatten Mode	15-24
Questions for Review	15-25
Exercise: Screen Mesh - Sheet Metal Approach	15-26

Chapter 16 Working with Sheet Metal STEP Files 16-1

Tools Needed	16-2
Opening an Assembly Step File	16-3
Mating the components	16-4



Adding the Sheet Metal tool tab	16-7
Inserting Sheet Metal parameters	16-8
Viewing the Flat Pattern	16-9
Converting other components	16-9
Using the Hole Series	16-11
Using the Hole Wizard	16-13
Adding the Smart Fasteners	16-15
Creating an Exploded View	16-17

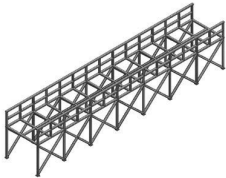


Adding Parts to the Toolbox Library	16-18
Starting the Toolbox Settings Utility	16-18
Activating Toolbox	16-21
Locating the new part	16-22
Viewing the new part	16-22
Adding a Part Number and Description	16-23
Sheet Metal Assembly	16-24

Chapter 17 Advanced Weldments 17-1

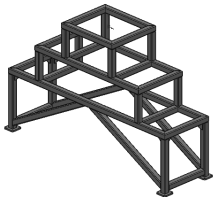
Weldments Platform 17-2

Tools Needed	17-2
Adding the weldments toolbar	17-3
Adding the structural members	17-4
Viewing the overlapped areas	17-8
Trimming the overlapped areas	17-9
Updating the cut list	17-12



Using Weldments – Structural Members 17-14

Enabling the Weldment Toolbar	17-14
Adding Structural Members	17-15
Setting the Corner Treatments	17-15
Adding Structural Members to Contiguous Groups	17-16
Adding Structural Members to the Parallel Groups	17-17
Trimming the Structural Members	17-19
Adding the foot pads	17-26
Adding the Gussets	17-27
Adding the Fillet Beads	17-29
Viewing the Weldment Cut List	17-31
Updating the Cut List	17-32
Creating a drawing	17-33



Mold Tools Design Topics

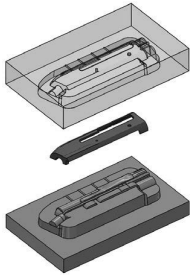
Chapter 18 **Creating a Core and Cavity** 18-1

Linear Parting Lines	18-2
Tools Needed	18-2
Opening an existing Parasolid document	18-3
Creating the Parting Lines	18-4
Creating the Shut-Off Surfaces	18-5
Creating the Parting Surfaces	18-6
Sketching the profile of the mold blocks	18-7
Creating the Tooling Split	18-8
Saving the bodies as part files	18-10
Separating the 2 blocks	18-11
Exercise: Linear Parting Lines	18-13
Questions for Review	18-19



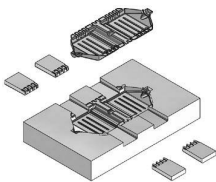
Chapter 19 **Non-Planar Parting Lines** 19-1

Mold-Tooling Design	19-2
Tools Needed	19-2
Enabling the Mold Tools toolbar	19-3
Creating the Parting Lines	19-4
Creating the Shut-Off Surfaces	19-5
Creating the Parting Surfaces	19-6
Creating a Ruled Surface	19-7
Creating the patches	19-9
Knitting the surfaces	19-13
Trimming the bottom of the ruled surface	19-15
Creating the Tooling Split sketch	19-16
Separating the solid bodies	19-18
Making the body transparent	19-19



Creating Slides and Cores 19-21

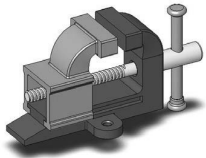
Tools Needed	19-22
Opening a part document	19-23
Analyzing the undercuts	19-23
Analysis parameters explained	19-24
Scaling the part	19-25
Creating the parting lines	19-26
Creating the parting surfaces	19-27
Creating a new sketch	19-28
Creating the tooling split	19-29
Finalizing the block sizes	19-29



Renaming the bodies and assigning materials	19-32
Creating the front slide core	19-32
Creating the back slide core	19-34
Creating an exploded view	19-35

Chapter 20 Top Down Assembly – Part 1 20-1

Miniature Vise	20-2
Tools Needed	20-2
Creating the Base part	20-3
Adding the side flanges	20-5
Creating an offset distance plane	20-7
Creating the 3D Guide Curves	20-9
Creating the fixed jaw clamp	20-11
Creating a new component: The slide jaw	20-14
Using the offset entities command	20-15
Sketching the Guide Curve	20-20
Creating the clamp block	20-22
Extruding the clamp block	20-23
Creating the Internal threads	20-26
Creating a Section View	20-29
Questions for Review	20-32



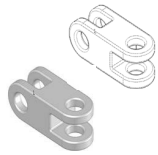
Chapter 21 Top-Down Assembly – Part 2 21-1

Water Control Valve	21-2
Tools Needed	21-2
Starting a New Assembly Template	21-3
Creating the 1st Component	21-4
Adding the Inlet Flange	21-5
Adding the Mounting Holes	21-6
Adding Chamfers and Fillets	21-8
Saving as Virtual Component	21-10
Creating the 2nd Component	21-10
Creating the Transition Body	21-12
Adding another mounting Flange	21-13
Adding an Offset-Distance plane	21-14
Exiting the Edit Component Mode	21-20
Applying dimension changes	21-20
Viewing the External Reference Symbols	21-22
Inserting other components	21-23
Questions for Review	21-25

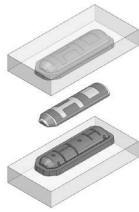


Chapter 22 External References & Repair Errors 22-1

External Reference Symbols	22-2
----------------------------	------



Understanding & Removing External References	22-3
Removing External References	22-3
Understanding the External Symbols	22-4
Repairing the Sketches	22-5
Rebuilding the model	22-7
Questions for Review	22-8
Understanding and Repairing Part Errors	22-9
Repair Errors & External References	22-16



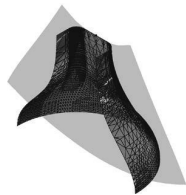
Final Exam_1 of 2_Handle Mold	22-24
Final Exam_2 of 2_Mouse Mold	22-28

Chapter 23 Using Appearances 23-1

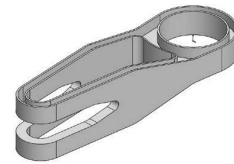
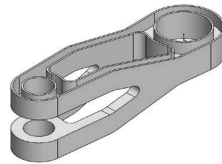


Modeling diamond knurls	23-1
Applying the knurl appearance	23-5
Applying wire mesh appearance	23-8
Applying the Car-Paint Appearance	23-11
Flatten Surfaces	23-16
Exercise: Flattening a shoe sole	23-20

Certification Practice for the CSWP Mechanical Design Exam 24-1



Challenge I: Part Modeling & Modifications	24-2
Challenge II: Part Modifications & Configurations	24-18
Challenge III: Bottom Up Assembly & Mates	24-39



Glossary, Index, and SOLIDWORKS 2024 Quick-Guide

Quick Reference Guide to SOLIDWORKS 2024 Command Icons and Toolbars.

