

# SOLIDWORKS® 2021 Intermediate Skills

Expanding on Solids, Surfaces, Multibodies,  
Configurations, Drawings, Sheet Metal  
and Assemblies



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 2021 User Interface**

**XIX**

The 3 reference planes

**XX**

The toolbars

**XX**

The system feedback symbols

**XXII**

The status bar

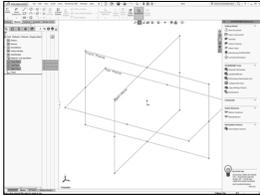
**XXII**

2D sketch examples

**XXIII**

3D feature examples

**XXIV**



**Chapter 1:**

**Document Properties**

**1-1**

**Setting up the Document Template**

**1-1**

Different standards in the world

**1-2**

Starting a new part document

**1-3**

Setting the ANSI drafting standards

**1-3**

    Setting the dimension options

**1-4**

    Setting the virtual sharp options

**1-5**

    Setting the units options

**1-5**

    Setting the image quality options

**1-6**

Saving the settings as a Part Template

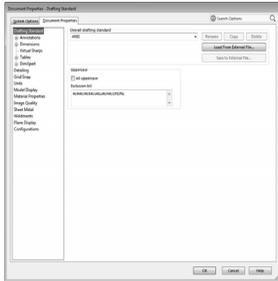
**1-6**

Customizing Keyboard Shortcuts

**1-7**

Customizing Tool Buttons

**1-12**



**Chapter 2:**

**Sketching**

**2-1**

**Handle**

**2-1**

Tools Needed

**2-2**

Starting a new part document

**2-3**

Changing the System Options

**2-3**

Creating the Parent Sketch

**2-4**

Revolving the parent sketch

**2-7**

Adding the tip detail

**2-7**

Revolving the sketch

**2-8**

Transitioning from line-to-arc

**2-8**

Mirroring in sketch mode

**2-9**

Adding dimensions

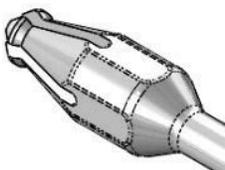
**2-10**

Extruding a cut

**2-10**

Creating a circular pattern

**2-11**





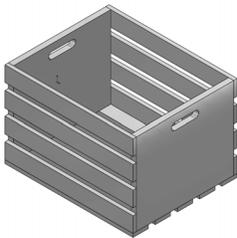
Adding other cut features	2-12
Creating another circular pattern	2-13
Adding a constant size fillet	2-14
Assigning material to the model	2-15
Calculating the mass of the model	2-16
Saving your work	2-16
Working with Sketch Pictures	2-17
Re-use geometry	2-37



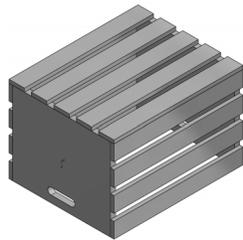
**Chapter 3:**

**Creating Multibody Parts 3-1**

**Wooden Crate 3-1**



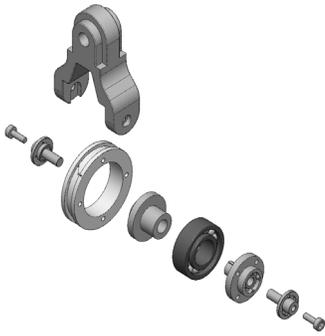
Tools needed	3-2
Starting a new part document	3-3
Sketching the first body profile	3-4
Extruding the first body	3-6
Copying the body	3-6
Creating the upper plank	3-7
Creating the first linear body	3-9
Mirroring the planks	3-10
Creating the lower plank	3-10
Creating the second linear pattern	3-12
Creating an exploded view	3-13
Collapsing the view	3-15
Saving your work	3-16
Exercise: Combining Multibodies	3-17



**Chapter 4:**

**Working with Multibody Parts 4-1**

**Creating Mates & Exploded Views 4-1**



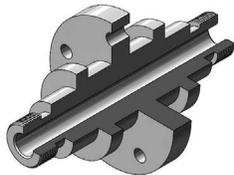
Tools needed	4-2
Opening a part document	4-3
Creating an exploded view	4-4
Collapsing the solid bodies	4-6
Inserting another part	4-7
Constraining the solid bodies	4-8
Adding a concentric mate	4-8
Adding a coincident mate	4-9
Creating a new mirror feature	4-10
Creating a mirror body	4-11
Insert another part	4-13
Adding a concentric mate	4-14
Adding a coincident mate	4-14

## SOLIDWORKS 2021 | Intermediate Skills | Table of Contents



Adding another instance of the bolt	4-15
Adding a concentric mate	4-15
Adding a coincident mate	4-16
Editing the exploded view	4-16
Saving your work	4-17
Working with Multibodies	4-19

### Chapter 5: Revolved and Thread Features



<b>Cylinder</b>	5-1
Tools needed	5-2
Opening a part document	5-3
Creating a revolved feature	5-3
Adding chamfers	5-4
Creating the mounting holes	5-4
Adding threads	5-7
Adding threads to the opposite end	5-9
Creating a zonal section view	5-11
Saving your work	5-12
Exercise: Revolved & Threads	5-13

### Chapter 6: Flex Bending



<b>Spanner</b>	6-1
Tools needed	6-2
Flex Bending - Part 1	6-3
Opening a part document	6-3
Creating the first flex bending feature	6-3
Creating the second flex bending feature	6-5
Flex examples	6-6
Flex twisting	6-6
Flex stretching	6-6
Flex tapering	6-7
Flex bending	6-7
Flex Bending - Part 2	6-8
Opening a part document	6-8
Creating a flex bending	6-8
Creating a linear pattern	6-10
Using the Deform feature	6-11
Opening a part document	6-11
Creating a Curve-to-Curve deform	6-11
Save and close	6-12
Opening a part document	6-13



Creating a Curve to Curve deform	6-13
Mirroring a surface body	6-14
Save and close all documents	6-14
Deform using a 3D sketch	6-15

**Chapter 7: Sweep with Guide Curves**

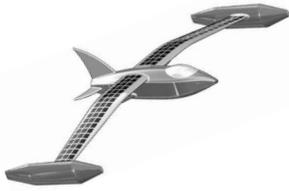


<b>Plastic Bottle</b>	7-1
Tools needed	7-1
Opening a part document	7-2
Creating the main body	7-3
Creating the first split line	7-3
Adding a face fillet	7-5
Creating the second split line	7-6
Creating the first offset surface	7-7
Creating a ruled surface	7-8
Creating the first knit surface	7-8
Adding the first fillet	7-9
Making the first cut with surface	7-9
Creating the first offset surface	7-10
Creating the second offset surface	7-11
Creating the second knit surface	7-11
Adding the second fillet	7-12
Making the second cut with surface	7-12
Adding the third fillet	7-13
Creating a shell	7-14
Adding the neck feature	7-14
Revolving the sketch	7-15
Creating a new plane	7-16
Creating a helix	7-16
Sketching the thread profile	7-17
Making the threads	7-18
Rounding off the ends of threads	7-19
Revolving the sketch	7-19
Rounding off the opposite side	7-20
Sweep with Solid Body	7-20
Exercise: Sweep vs. Loft	7-21

**Chapter 8: Lofts and Boundaries**



<b>Solar Boat</b>	8-1
Tools needed	8-1
Opening a part document	8-2
Creating a lofted feature	8-3



Creating a boundary feature	8-5
Creating the wing	8-6
Creating the wing support	8-7
Constructing the nose feature	8-8
Creating the horizontal tail fins	8-11
Creating the vertical tail fin	8-12
Creating a split line	8-13
Creating the canopy's first profile	8-14
Creating a new plane	8-15
Sketching the guide curve	8-15
Creating the canopy's second loft profile	8-16
Creating the canopy's third loft profile	8-16
Creating the canopy	8-17
Creating a mirror feature	8-17
Adding fillets to the main body	8-18
Adding fillets to the float and wing	8-18
Adding fillets to the ends of the floats	8-19
Adding fillets to the tail fins	8-19
Adding fillets to the ends of the fins	8-20
Creating a face fillet	8-20
Creating a split line for the solar panels	8-21
Changing color of the solar squares	8-22
Changing the edge display	8-23
Exercise: Surface & Solid Modeling	8-25
Exercise: Using Split Lines	8-33

**Chapter 9:**

<b>Surfaces and Patches</b>	<b>9-1</b>
<b>Welding Fixture</b>	<b>9-1</b>
Tools needed	9-2
Opening a part document	9-3
Creating a swept surface	9-3
Creating the first extruded surface	9-4
Creating the second extruded surface	9-4
Creating the third extruded surface	9-5
Creating the first trimmed surface	9-5
Creating the second trimmed surface	9-6
Creating the third trimmed surface	9-7
Constructing a lofted surface	9-9
Creating the first boundary surface	9-10
Creating the second boundary surface	9-11
Creating the first filled surface	9-12
Creating the second filled surface	9-13



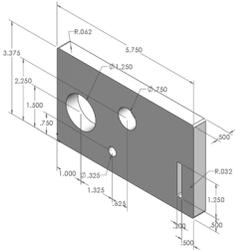


Creating a face fillet	9-14
Knitting all surfaces	9-15
Thickening the surface model	9-16
Creating a section view	9-17
Changing the edge display	9-18
Exercise: Surface Modifications	9-19

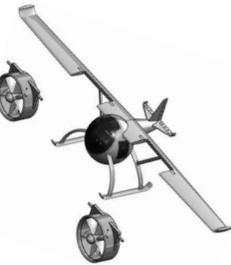
**Chapter 10:**



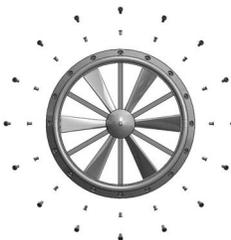
<b>Configure Features</b>	<b>10-1</b>
<b>Wooden Crate</b>	10-1
Tools needed	10-2
Opening a part document	10-3
Configuring features	10-3
Saving the table	10-5
Adding new configurations	10-5
Viewing the new configuration	10-6
Adding more configurations	10-7
Renaming a configuration	10-9
Inserting a design table	10-10
Adding a material column	10-11
Exercise: Design Tables & Tabulated Tables	10-15



**Chapter 11:**

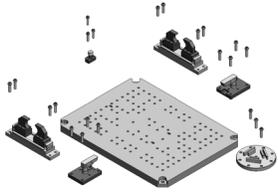


<b>Assembly Motions &amp; Mates</b>	<b>11-1</b>
<b>HeliDrone</b>	11-1
Tools needed	11-2
Opening an existing assembly document	11-3
Using the width mate	11-3
Adding a concentric mate	11-4
Adding another concentric mate	11-6
Testing the assembly motions	11-6
Adding a limit-angle mate	11-7
Adding a parallel mate	11-8
Adding a gear mate	11-9
Testing the assembly motions	11-10
Radial Explode	11-11
Opening an existing assembly	11-11
Creating the first pattern driven	11-11
Creating the second pattern driven	11-12
Creating the first radial exploded view	11-13
Creating the second radial exploded view	11-13
Changing to the configuration tree	11-14
Verifying the exploded view	11-14

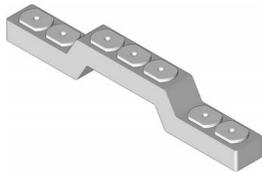


Exercise: Assembly Motions 11-15

**Chapter 12: Using Smart-Mates 12-1**



**Fixture Assembly 12-1**  
 Tools Needed 12-2  
 Opening an existing assembly document 12-3  
 Enabling the selection options 12-3  
 Exploring the Smart-Mate options 12-4  
 Using Smart-Mate Concentric 12-5  
 Creating a Smart-Mate Concentric & Coincident 12-6  
 Repeating the previous mate 12-7  
 Mating other components 12-8  
 Checking the status of the components 12-9  
 Switching configuration 12-9  
 Creating an instance of the bolt 12-10  
 Creating another instance of the bolt 12-11  
 Adding more bolts 12-12  
 Repeating 12-13  
 Mate Controller 12-15  
 Using copy with mates 12-25



**Chapter 13: Top Down Assembly 13-1**



**Car Remote Control Housing 13-1**  
 Tools Needed 13-2  
 Opening a part document 13-3  
 Making an assembly from part 13-3  
 Placing the first component 13-4  
 Creating a new part 13-4  
 Converting the entities 13-5  
 Extruding the sketch 13-6  
 Adding the fillet 13-6  
 Shelling the part 13-7  
 Hiding the component 13-7  
 Adding features to the part 13-8  
 Creating offset entities 13-9  
 Extruding the sketch 13-10  
 Adding the fillet 13-10  
 Showing a component 13-11  
 Creating a section view 13-11  
 Applying dimension changes 13-12  
 Creating an exploded view 13-14  
 Playing the animation 13-15



Changing the display mode	13-16
Creating a Flat Spring Assembly	13-17
Exercise: Spring Assembly	13-29

**Chapter 14: Using the Lip & Groove Options 14-1**

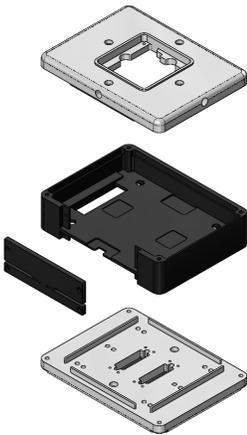


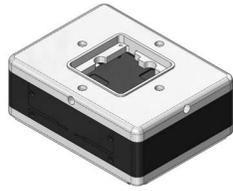
Lesson overview	14-1
Tools Needed	14-2
Top Down Assembly	14-3
Opening the existing assembly	14-4
Editing the housing	14-4
Creating a section view	14-7
Using the Mounting Boss Options	14-8
Opening the existing assembly	14-9
Editing part	14-9
Setting the parameters	14-9
Adding two more mounting bosses	14-11
Toggling between Explode and the collapse views	14-11
Creating the mating bosses	14-12
Creating a section view	14-14
Repeating	14-14



**Chapter 15: Assembly Drawings & BOMs 15-1**

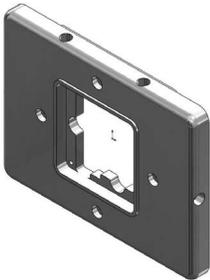
<b>Aluminum Enclosure</b>	15-1
Tools needed	15-2
Starting a new drawing document	15-3
Selecting the ANSI standard sheet size	15-3
Switching to ANSI standards	15-4
Using the view palette	15-5
Inserting a Bill of Materials	15-6
Modifying the BOM's Row Height	15-8
Adding a new column	15-8
Customizing the new column	15-9
Changing the color of the paper	15-10
Adding balloons	15-11
Adding an isometric view	15-12
Changing the display of a drawing view	15-13
Editing the sheet format	15-13
Switching back to the sheet	15-16
Reference reading: GD&T	15-18
Form examples	15-18
Orientation examples	15-19
Profile examples	15-19



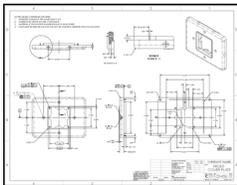


Runout examples	15-20
Location examples	15-20
ASME – Y14.5 Symbol descriptions	15-22
Feature control frames	15-25
MMC maximum material condition	15-26
LMC least material condition	15-26

**Chapter 16: Drawings & Detailing 16-1**



<b>Front Cover Plate</b>	16-1
Tools needed	16-2
Opening a part document	16-3
Creating the drawing views	16-4
Changing to phantom line style	16-6
Creating a partial section view	16-7
Flipping the section direction	16-8
Changing the hatch pattern	16-8
Creating a detail view	16-9
Inserting the model dimensions	16-10
Cleaning up the dimensions	16-11
Inserting dimensions to another view	16-12
Breaking the view alignment	16-13
Creating a projected view	16-14
Adding the centerline symbol	16-15
Adding datums	16-16
Adding a flatness tolerance	16-18
Copying the control frame	16-20
Modifying the geometric tolerance	16-21
Adding another position tolerance	16-22
Adding a symmetric tolerance	16-23
Adding a bilateral tolerance	16-23
Adding a limit tolerance	16-24
Adding a basic tolerance	16-24
Adding general notes	16-25
Filling out the title block	16-26
Exercise: Creating a drawing from a model	16-29
Exercise: Attaching a note or symbol to a dimension	16-30



**Chapter 17: SOLIDWORKS MBD 17-1**

<b>Introduction to Model Based Definition</b>	17-1
Tools needed	17-2
Prismatic Parts	17-3

## SOLIDWORKS 2021 | Intermediate Skills | Table of Contents



Opening a part document	17-3
Enabling SOLIDWORKS MBD	17-3
Setting the options	17-3
Adding datums	17-5
Using scope	17-6
Showing the constraint status	17-7
Adding the size dimensions	17-8
Showing the tolerance status	17-8
Exercise: Adding the Size and Location Dimensions	17-9
Turned Parts	17-13
Capturing 3D Views	17-19
Publishing PDF and eDrawing Files	17-23
3D PDF Template Editor	17-29

### Chapter 18:

### Sheet Metal Parts 18-1

#### Hard Drive Enclosure 18-1

Tools needed 18-2

Starting a new part document 18-3

Creating the Base flange 18-4

Creating a cut feature 18-5

Creating the first edge flange 18-6

Creating the second edge flange 18-9

Copying and Pasting a sketch 18-11

Creating more cuts 18-13

Creating a library feature cut 18-15

Mirroring a feature 18-17

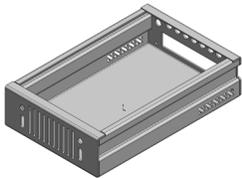
Toggling the flat display 18-18

Creating the flat pattern 18-19

Accessing the sheet properties 18-20

Multibody sheet metal parts 18-21

Using FeatureWorks – Feature Recognition 18-37



### Chapter 19:

### Plastic parts 19-1

#### Plastic Part\_Design1 19-1

Tools Needed 19-2

Making the base feature with draft 19-3

Creating a curved cut feature 19-4

Adding the raised features 19-5

Adding fillets 19-6

Shelling the model 19-6

Making the interlock feature 19-7



Creating a face fillet	19-8
Removing the sharp edges	19-9
Adding the side holes	19-10



<b>Plastic Part_Design2</b>	19-11
Making the base feature with draft	19-11
Creating a curved cut	19-12
Adding a new plane	19-14
Making the button holes	19-14
Shelling the model	19-16
Creating the interlock feature	19-18
Adding cuts	19-20
Creating the mounting bosses	19-22
Removing the sharp edges	19-24

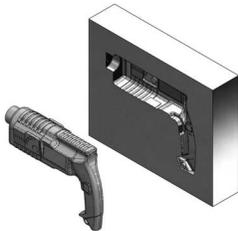
**Chapter 20:**

<b>Smart Component</b>	<b>20-1</b>
<b>Auto Size &amp; Configure Table</b>	20-1
Tools needed	20-2
Opening a part document	20-3
Showing the dimensions and their names	20-3
Splitting the FeatureManager tree	20-4
Adding a new configuration	20-4
Creating mate references	20-6
Testing the mate references	20-7
Making a smart component	20-8
Testing the smart component	20-9
Adding other instances of the cap	20-10



**Chapter 21:**

<b>Using Subtract &amp; Intersect</b>	<b>21-1</b>
<b>Simple Cavity</b>	21-1
Tools needed	21-2
Opening a part document	21-3
Scaling the part	21-3
Extruding the mold block	21-3
Assigning material	21-4
Copying a solid body	21-4
Creating the cavity	21-5
Separating the solid bodies	21-6
Using the intersect tool	21-7
Exercise: Creating a Cavity with Combine Subtract	21-10



	<b>Chapter 22: Using Magnetic Mates</b>	<b>22-1</b>
	<b>Deck Assembly</b>	22-1
	Tools needed	22-2
	Opening a part document	22-3
	Publishing an asset	22-4
	Opening another part document	22-6
	Defining another asset publisher	22-7
	Opening an assembly document	22-8
	Adding the section1 component to the assembly	22-9
	Adding another instance of section1	22-10
	Adding more instances	22-11

	<b>Chapter 23: Hybrid Modeling</b>	<b>23-1</b>
	<b>Remote Control</b>	23-1
	Tools needed	23-2
	Opening a part document	23-3
	Extruding the first surface	23-3
	Extruding the second surface	23-4
	Trimming the surfaces	23-4
	Thickening the surfaces	23-5
	Creating a split line feature	23-5
	Adding draft	23-6
	Adding fillets	23-7
	Shelling the solid body	23-8
	Cutting the solid body	23-8
	Saving the upper half	23-9
	Saving the lower half	23-9
	Flipping the cut direction	23-9
	Inserting the top half	23-10
Creating the cut for the keypad	23-11	
Separating the two halves	23-12	
Changing appearance	23-12	

**Glossary**

**Index**

**SOLIDWORKS 2021 Quick-Guides**

