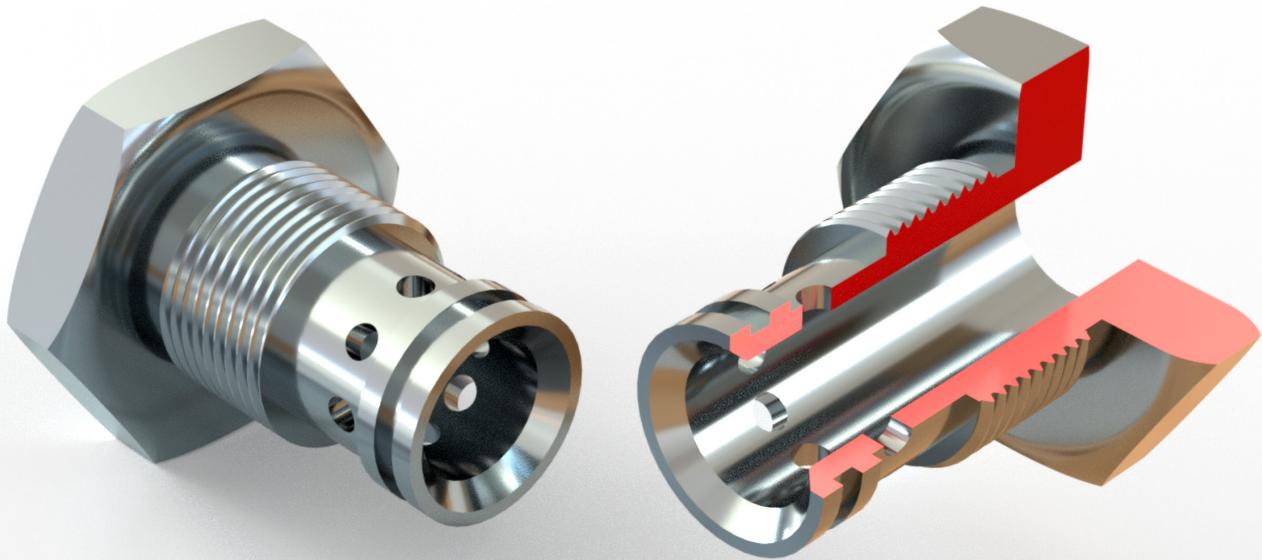


New
Chapter Covering the
CSWP-DT Exam

Mastering Modern CAD Drawings with SOLIDWORKS® 2026

Applying ASME Standards to Engineering Drawings



Lani Tran, CSWE

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

Lesson 1:	INTRODUCTION TO CAD DRAWINGS	1-1
	Manual vs. Computer Aided Drawings	1-1
	Technical drawings	1-1
	Sketching	1-1
	Manual drawings	1-2
	Computer Aided Design	1-3
	Limitations of 2D CAD	1-5
	Benefits of 3D CAD	1-5
	SOLIDWORKS – The industry standard	1-6
	Which industries use SOLIDWORKS	1-7
	Why companies use SOLIDWORKS	1-8
	SOLIDWORKS drawings	1-8
	System Options & Document Properties	1-11
Lesson 2:	DETAILING A MACHINED PART	2-1
	Stainless Steel Flange	2-1
	Opening a part document	2-1
	Setting the drawing properties	2-2
	Creating an engineering drawing	2-3
	Changing the scale of the drawing Views	2-5
	Rearranging the drawing Views	2-6
	Modifying the sheet scale	2-7
	Creating a section view	2-8
	Modifying the crosshatch properties	2-9
	Adding the centerlines	2-10
	Changing the Font Size	2-10
	Changing the drawing paper size	2-12
	Inserting the model dimensions	2-14
	Moving dimensions	2-15
	Removing dimension overlaps with Break-Lines	2-16
	Inserting other dimensions	2-17
	Adding callouts	2-18
	Adding additional callouts	2-19
	Adding datums	2-21
	Adding the General Notes	2-23
	Adding the Revision Table	2-24
	Filling out the Title Block	2-26
	Switching back to the Sheet	2-27
	Exercise: Detailing a Machined Part	2-29

Lesson 3:	UNDERSTANDING TOLERANCE, PRECISION, & GD&T	3-1
	Using GD&T in a Drawing	3-1
	What is GD&T?	3-2
	Rule #1 and #2	3-3
	Terminology used in GD&T	3-4
	Maximum material condition	3-6
	Least material condition	3-7
	Size, location, orientation, Form	3-8
	Types of GD&T Tolerances	3-9
	Material modifier	3-10
	Straightness	3-11
	Flatness – Surface	3-12
	Flatness – FOS/DMP	3-13
	Circularity	3-14
	Cylindricity	3-15
	Line profile	3-16
	Surface profile	3-17
	Parallelism – FOS	3-18
	Parallelism – Surface	3-19
	Perpendicularity – Surface	3-20
	Perpendicularity – FOS	3-21
	Angularity	3-22
	Runout	3-23
	Total runout	3-24
	Position	3-25
	Concentricity	3-26
	Symmetry	3-27
	GD&T in a nut shell	3-28
	What are the advantages of GD&T	3-28
Lesson 4:	USING TOLERANCE, PRECISION, and GD&T	4-1
	Stainless steel flange	4-1
	Opening a drawing document	4-1
	Adding a symmetric tolerance	4-2
	Adding a bilateral tolerance	4-3
	Adding a limit tolerance	4-4
	Adding a basic dimension	4-5
	Adding a unilateral tolerance	4-7
	Adding a true position tolerance	4-9
	Bonus tolerance formular	4-10
	Adding a concentricity control	4-11

	Exercise: Detailing a machined part	4-13
	Creating the drawing views	4-14
	Adding dimensions	4-15
	Adding tolerance and precision	4-16
	Adding the general notes	4-17
	Adding a revision block	4-18
	Filling out the title block	4-19
	Exercise: Detailing a machined part	4-21
	Transferring to a drawing	4-21
	Creating the drawing views	4-22
	Adding the model dimensions	4-23
	Adding the General Notes	4-24
	Filling out the Title Block	4-24
Lesson 5:	DETAILING PLASTIC PARTS	5-1
	Plastic Gear	5-1
	Opening a part document	5-1
	Transferring to drawing	5-2
	Creating the drawing views	5-3
	Creating a section view	5-4
	Changing the hatch pattern	5-5
	Creating a detail view	5-6
	Adding centerlines automatically	5-7
	Adding centerlines manually	5-7
	Inserting the model dimensions	5-8
	Moving the dimensions	5-9
	Moving and Hiding dimensions	5-10
	Adding the missing dimensions	5-11
	Adding datums	5-14
	Adding Geometric tolerance	5-14
	Adding the general notes	5-16
	Adding the revision block	5-17
	Filling out the title block	5-18
	Exercise 1: Detailing a Plastic Part 1	5-20
	Creating the drawing views	5-21
	Making a section view	5-22
	Modifying the hatch pattern	5-23
	Adding the detail views	5-24
	Creating a crop view	5-25
	Breaking the view alignment	5-26
	Adding dimensions	5-26

	Adding the general notes	5-28
	Filling out the title block information	5-29
	Exercise 2: Detailing a Plastic Part 2	5-31
	Exercise 3: Detailing a Plastic Part 3	5-35
Lesson 6:	DETAILING SHEET METAL PARTS	6-1
	Detailing a Sheet Metal Part	6-1
	Opening a part document	6-1
	Setting up the drawing sheet	6-2
	Creating the drawing views	6-3
	Creating an auxiliary view	6-4
	Creating a crop view	6-5
	Inserting the model dimensions	6-6
	Rearranging and adding the dimensions	6-7
	Adding the general notes	6-8
	Filling out the title block	6-9
	Adding the isometric flat pattern	6-10
	Exercise: Detailing a Sheet Metal Part	6-12
	Transferring to a drawing	6-12
	Creating the drawing views	6-13
	Adding centerlines and center marks	6-14
	Inserting the model dimensions	6-14
	Rearranging and modifying Dimensions	6-15
	Adding the general notes	6-16
	Filling out the title block	6-17
	Exercise: Detailing a Sheet Metal Part	6-19
	Exercise: Adding Dimensions to a Spline	6-25
Lesson 7:	DETAILING CASTINGS AND FORGINGS	7-1
	Detailing a Casted Part	7-1
	Opening a part document	7-1
	Drawings overview	7-2
	Adding a 2nd sheet	7-3
	Creating the casted drawing	7-3
	Inserting the model dimensions	7-4
	Creating the general notes	7-6
	Filling out the title block information	7-7
	Creating the machined drawing	7-8

	Inserting the model dimensions	7-9
	Adding datums	7-12
	Adding geometric tolerances	7-13
	Filling out the title block	7-14
	Exercise: Detailing a casted part 1	7-16
	Exercise: Detailing a casted part 2	7-35
	Exercise: Detailing a casted part 3	7-39
Lesson 8:	ASSEMBLY DRAWINGS	8-1
	Detailing an assembly drawing	8-1
	Opening an assembly document	8-1
	Making a drawing from assembly	8-2
	Creating the drawing views	8-3
	Adding balloons	8-4
	Adding the number of instances	8-6
	Inserting a bill of materials	8-7
	Adjusting the BOM	8-9
	Filling out the BOM manually	8-10
	Filling out the title block information	8-11
	Exercise: Detailing an assembly drawing	8-13
	Transferring to a drawing	8-13
	Creating the drawing views	8-14
	Adding balloons and instance counts	8-15
	Inserting the bill of materials	8-16
	Changing the custom properties	8-17
	Filling out the title block information	8-18
	Exercise: Detailing an assembly drawing	8-21
Lesson 9:	CUSTOM PROPERTIES	9-1
	Creating the custom properties – Part 1	9-1
	Opening an assembly document	9-1
	Opening a part from an assembly	9-2
	Creating the custom properties	9-2
	Entering the custom property information	9-3
	Making a drawing from assembly	9-5
	Creating the drawing views	9-6
	Adding balloons	9-7
	Inserting the bill of materials	9-8

	Switching the custom properties	9-9
	Filling out the title block information	9-10
	Part 2: Using the Property Tab Builder	9-12
	Starting a new drawing	9-12
	Accessing the custom properties tab	9-13
	Switching the Template type	9-14
	Modifying the Groupbox	9-14
	Adding a textbox field	9-15
	Adding a list field	9-15
	Adding another Textbox field	9-16
	Saving the Property Tab template	9-17
	Testing the DRW Properties Template	9-17
	Adding a Textbox field	9-18
	Saving the Property Tab Template	9-19
	Testing out the Part Properties Template	9-20
	Creating a sample part	9-20
	Making drawing from part	9-21
Lesson 10:	CONFIGURATIONS AND DESIGN	10-1
	Creating a Design Table using MS-Excel	10-1
	Opening a part document	10-1
	Renaming the dimensions	10-2
	Inserting an Excel-based design table	10-3
	Adding new configurations in Excel	10-4
	Adding new features to the design table	10-5
	Assigning the feature's states	10-6
	Switching back to SOLIDWORKS application	10-7
	Viewing the new configurations	10-8
	Adjusting the rows and columns	10-9
	Making a drawing from part	10-10
	Inserting the design table	10-11
	Adjusting the table header in Excel	10-12
	Switching back to the drawing	10-13
	Filling out the title block information	10-14
	Creating a tabulated table	10-15
	Exercise: Design Table - Creating a tabulated table	10-18

Lesson 11:	ADDITIONAL DRAWING TOOLS	11-1
	Detailing the base mount block	11-1
	Opening a part document	11-1
	Adding the standard drawing views	11-2
	Creating an auxiliary view	11-3
	Breaking the view alignment	11-4
	Creating a broken-out section view	11-5
	Modifying the hatch pattern	11-6
	Creating a detail view	11-7
	Adding centerlines and center marks	11-8
	Inserting the model dimensions	11-9
	Rearranging dimensions	11-10
	Moving dimensions	11-11
	Reviewing the drawing	11-12
	Changing the color of a dimension	11-13
	Showing a sketch from the model	11-14
	Inserting the pattern instances	11-15
	Creating a parametric note	11-16
	Changing the number of pattern instances	11-17
	SOLIDWORKS DrawCompare	11-19
	Comparing the drawings	11-20
	Viewing the differences	11-21
	SOLIDWORKS Pack and Go	11-23
	Opening an assembly document	11-23
	Launching the Pack and Go utility	11-24
	Exercise: Detailing the aluminum block	11-25
	Opening a part document	11-25
	Creating the additional drawing views	11-26
	Inserting the model dimensions	11-27
	Adding the instance callouts	11-28
	Filling out the title block	11-29
	Exercise: Other options in assembly drawings	11-31
	Adding the explode lines	11-32
	Collapsing the assembly	11-35
	Making drawing from assembly	11-36
	Inserting the drawing views	11-37
	Adding balloons	11-38
	Inserting a bill of materials	11-39
	Formatting the BOM	11-40

	Changing the custom properties	11-41
	Hiding components	11-43
	Other display options	11-46
Lesson 12:	SOLIDWORKS MBD - Model Based Definition	12-1
	Part 1: Model Based Definition – Prismatic parts	12-1
	Opening a part document	12-1
	Understanding the types of parts	12-2
	Understanding the DimXpert tool	12-2
	Setting up the DimXpert options	12-3
	Adding datums	12-5
	Showing tolerance status	12-8
	Opening another part document	12-9
	Adding datums	12-9
	Adding the size dimensions	12-10
	Checking the constraint status	12-11
	Showing all annotations	12-12
	Part 2: Model Based Definition – Turned parts	12-13
	Opening a part document	12-13
	Adding datums	12-14
	Rearranging the annotations	12-16
	Recreating the basic dimensions	12-17
	Opening another part document	12-19
	Using the plus and minus tolerances	12-19
	Using polar	12-20
	Part 3: Model Based Definition – Capturing 3D Views	12-22
	Switching to 3D View tab	12-23
	Capturing the Front Views	12-24
	Capturing the orthographic views	12-25
	Creating a zonal section view	12-26
	Capturing the zonal section view	12-27
	Testing the 3D views	12-28
	Publishing to 3D-PDF document	12-29
	Saving the 3D-PDF	12-31
	Publishing to eDrawing	12-32
	Viewing the animation	12-33

Lesson 13:	CSWP-Drawing Tools Exam Preparation Material	13-1
	Challenge 1 – Part Drawings:	
	View Creation & Projected Dimensions	13-3
	Opening an existing drawing	13-3
	Adding the Front drawing view	13-3
	Adding dimensions	13-4
	Creating the 1st section view	13-5
	Measuring the perimeter	13-6
	Creating the 2nd section view	13-7
	Measuring the perimeter	13-8
	Challenge 2 – Assembly Drawings:	
	View Creation & Measurements	13-9
	Opening an assembly document	13-9
	Changing the orientation	13-9
	Finding the angle	13-10
	Entering the angle for Arrow Key rotation	13-11
	Rotating the model	13-12
	Updating the standard views	13-13
	Making a drawing	13-15
	Adding a drawing view	13-17
	Moving the drawing view	13-18
	Lock View Focus vs. Lock Sheet Focus	13-19
	Lock Sheet Focus	13-20
	Finding the angle	13-21
	Sketching a reference line	13-22
	Adding another reference line	13-23
	Finding the distance	13-24
	Creating an Auxiliary View	13-25
	Creating an Alternate Position View	13-26
	Finding the angle	13-27
	Challenge 3 – Assembly Drawings:	
	Drawing & BOM Creation	13-29
	Opening an assembly document	13-29
	Making a drawing	13-30
	Creating a Bill of Materials	13-31
	Finding the quantity of a component	13-32
	Switching the Custom Properties	13-33
	Finding the Item Number of a component	13-34
	Switching the BOM Type	13-35
	Finding the Total Cost	13-36
	Adding a Note	13-41

Challenge 4 – Assembly Drawings:	
Hide and Show Components	13-43
Opening an assembly document	13-43
Finding the hidden components	13-44
Measuring the angle	13-45
Saving the drawing	13-46
Challenge 5 – Part Drawings:	
Broken Out Section & Insert Model Items	13-47
Opening a drawing document	13-47
Creating a Broken Out Section view	13-48
Measuring the perimeter	13-50
Inserting the Model Dimensions	13-51
Saving the drawing	13-52
Challenge 6 – Part Drawings:	
View Creation & Feature Measurement	13-53
Opening a drawing document	13-53
Creating a custom dimensioning view	13-54
Saving the custom view	13-56
Inserting the custom view	13-57
Creating a section view	13-58
Saving your drawing	13-59

Index and Glossary

Model library

The models and drawings used in this textbook were created by the author using SOLIDWORKS. All rights reserved.

