



SolidWorks Refresher (2+ Days)

+ * **Ve-I Bonus!** * 2016 Update Training + AutoCAD to SOLIDWORKS

Drawings

Drawing Views

Model View	Explains how to use the model view command to create a drawing view
Projected View	Explains the projected view option in a drawing
Auxiliary View	Explains how to create an auxiliary view in a drawing
Section Views I	Explains how to create a section view in a drawing
Section Views II	Explains how to create a section view in a drawing
Detail View	Explains how to create a detail view in a drawing
Broken View	Explains how to break a view in a drawing
Model Break Views	Explains how to create Model Break Views in drawings
Assembly Section View	Explains how to create a section view in an Assembly drawing

Dimensions and Annotations

Assembly Explode	Explains how to add an exploded view in a drawing
Display Options	Explains how to control how a view is displayed in a drawing
View Alignment	Explains how to adjust the alignment of views in a drawing
Scale	Explains how to control the scale of views in a drawing
Balloons	Explains how to add balloons in a drawing
Magnetic Lines	Shows how to align balloons using magnetic lines
Dimensions	Explains the difference between driving and driven dimensions in a drawing
Driving Dimensions	Explains how to import and adjust driving dimensions in a drawing
Driven Dimensions	Explains how to add driven dimensions in a drawing
Formatting Dimensions	Shows methods for controlling the appearance and layout of dimensions
Ordinate Dimensions	Explains how to use ordinate dimensions in a drawing
Baseline Dimensions	Explains how to add baseline dimensions in a drawing
Center Marks	Explains how to add center marks in a drawing
Note Annotations	Explains how to add notes in a drawing
Flag Notes	Shows how to create Flag Notes

Drawing Templates and Sheet Formats

Create Custom Templates	Explains how to design and save custom drawing templates
Custom Sheet Formats	Explains how to create custom sheet formats
Custom Properties and Parametric Notes	Shows how to add smart notes and custom properties

Bill of Materials

Bill of Materials	Explains how to create a bill of materials in a drawing
Advanced BOM Options	Explains the options available for Bills of Material
Linking BOM Tables	Shows how to link a BOM from an assembly file in a drawing

2016 Core Concepts for Parts and Assemblies

Sketch Tools

Sketch Fillets	Explains how to add a fillet to sharp sketch segments
Offset Entities	Explains how to create an offset copy of another entity
Trim and Extend Entities	Explains how to trim and extend lines in a sketch
Mirroring Sketch Entities	Explains how to create a symmetrical copy of sketch entities
Sketch Patterns	Shows how to create both linear and circular patterns of sketch entities
Sketches on Faces	Introduces how to create a 2-D sketch on a face of the model
Rapid Sketch	Introduces the rapid sketch tool
Sketched Text	Shows how to add text to a sketch

Sketched Features

Sketched Features Introduction	Overview
Extruded Boss	Explains how to extrude a 2-D sketch into a 3-D part
Extruded Cut	Explains how to remove material from a part
Contours and Thin Features	Explains how to extrude a selected portion of a sketch
Revolved Boss	Explains how to revolve a 2-D profile to create 3-D geometry

Revolved Cut	Explains how to remove material using a revolved cut
Sweeps I	Explains how to create 3-D geometry by using a profile and a path
Sweeps II	Explains how to create a circular profile sweep
Loft	Explains how to loft two profiles together to create 3-D geometry
Ribs	Explains how to create Rib Features on a part

Applied Features

Applied Features Introduction	Applied Features Overview
Fillets	Explains how to use fillets to round off 3-D parts
Chamfers	Explains how to break sharp edges on 3-D parts
Shell	Explains how to take solid geometry and make it hollow
Hole Wizard	Explains how to create holes in parts used for fasteners
Draft	Shows how to add a draft angle to a face

Reference Geometry

Planes	Shows how to create a reference plane
Creating Axes	Explains how to manually create axes on a 3-D body
Reference Points and Coordinate Systems	Explains how to create reference points in a model and define new reference coordinate systems

Patterns and Mirroring

Linear Patterns	Shows how to create multiple instances of a feature in a linear fashion
Circular Patterns	Shows how to create multiple instances of a feature in a circular fashion
Vary Pattern Instances	Shows how to vary the dimensions of pattern instances
Mirroring Features	Shows how to symmetrically replicate features on a part
Mirroring Bodies	Shows how to replicate geometry about a plane

Editing Parts and Design Changes

Editing Parts	Explains how to edit a part
Rollback Bar	Shows how to use the rollback bar to figure out how a part was built
Parent Child Relations	Explains how some features are dependent upon one another
Selection Breadcrumbs	Learn how to use Selection Breadcrumbs to refine your selections

Adding Features	Shows how to add features to the design tree to change the design
Edit Sketch Plane	Explains how to change the plane a sketch was built on without deleting the original sketch geometry
Instant 3D	Explains how to modify a part dynamically using Instant 3D

Troubleshooting Parts

Troubleshooting Parts - Introduction	Introduces the procedure to troubleshoot parts, along with the "What's Wrong" dialog box
Troubleshooting Sketches	Shows how to troubleshoot the common sketch errors
Troubleshooting Features	Shows how to troubleshoot the common feature errors

Analyzing Part Properties

Measure	Shows how to use the measure tool
Materials	Shows how apply a material with properties to a part
Mass Properties	Shows how check the mass and other properties of a part
Custom Properties	Shows how add custom properties to a part

Configurations

Configurations - Introduction	Introduces how different configurations can be used on a part
Feature Suppression	Explains what Feature Suppression is and shows how to suppress features in a Part
Configurations - Dimension Changes	Explains how to create different configurations for a part
Configurations - Feature Suppression	Explains how to create different configurations for a part
Configurations - Table	Explains how to create different configurations for a part

Analyzing Assemblies

Analyzing Assemblies - Introduction	Overview
Assembly Mass Properties	Explains how to check the mass properties of an assembly
Interference Detection	Explains how to detect interferences between components
Dynamic Clearance	Explains how to detect the amount of clearance between components
Collision Detection	Explains how to detect collisions with moving components
Exploding Assemblies	Explains how to create an exploded view of an assembly

Assembly BOM

Explains how to create a BOM in an assembly file

2016 Advanced Parts

Advanced Sketching

Polygon	Explains how to sketch polygons
Ellipse	Explains how to sketch ellipses
Partial Ellipse	Explains how to sketch partial ellipses
Parabola	Explains how to sketch a parabola
Conic Curves	Shows how to sketch conic curves driven by endpoints and Rho value
Splitting Entities	Explains how to split a sketch entity
Sketch Segments	Explains how to use the Sketch Segment tool
Derived Sketches	Explains how to use the derived sketch command
Sketch Xpert	Shows how to resolve sketches using the Sketch Xpert
Sketch Picture	Shows how to insert an image into a sketch
AutoTrace	Shows how to use autotrace to automatically trace a sketch picture and convert it to sketch entities
3D Sketching I	Explains how to create a 3D sketch
3D Sketching II	Explains how to create a more complex 3D sketch
3D Sketching with Planes	Shows how to create 3D sketches on reference planes
3D Sketch Planes	Shows how to create and define 3D sketch planes in a 3D sketch

Curves

Helixes and Spirals	Explains how to sketch a helix or a spiral
Curve Through XYZ Points	Explains how to create a curve using data points
Composite Curve	Explains how to create composite curves
Curve Through Reference Points	Explains how to create a curve through the use of reference points
Projected Curves	Explains how to use the projected curve command
Intersection Curve	Explains how to sketch an intersection curve

Advanced Fillets

Fillet Options	Explains the different fillet options
Variable Size Fillets	Explains how to use the variable radius fillet command
Face Fillets	Explains how to use the face fillet command
Hold Line Fillets	Explains how to use the hold line fillet command
Curvature Continuous Fillets	Explains how to use the curvature continuous fillet command
Full Round Fillets	Explains how to use the full round fillet command
Conic Fillets	Explains how to use the Conic Rho and Conic Radius profile options
Fillet Xpert	Shows how to apply, change, and remove fillets using the Fillet Xpert
Corner Xpert	Shows how to use the Corner Xpert to control filleted corners

Sweeps

Introduction to Sweeps	Introduces the sweep feature
The Basic Sweep	Explains how to use the sweep feature
Circular Profile Sweeps	Explains how to create a circular profile sweep
Path Options	Explains the different path options for the sweep feature
Guide Curves	Explains how to add guide curves when using the sweep feature
Multiple Guide Curves	Explains how to add multiple guide curves when using the sweep feature
Profile Orientation	Explains how to use different orientations when creating a sweep feature
Twist Options	Explains how to twist a swept feature
Swept Cut	Explains how the sweep feature can be used to remove material

Lofts

Introduction to Lofts	Explains the difference between the loft and the sweep feature
The Basic Loft	Explains how to use the loft feature to create a part
Closing a Loft	Explains the close loft option when using the loft feature
Loft Profiles	Explains the different profiles that can be used with the loft feature
Start/End Constraints	Explains how to use constraints to alter the loft
Guide Curves	Explains how to use guide curves when creating a loft
Profile Geometry	Explains how to create lofts using profile geometry
Centerline	Explains how to control a loft using centerlines
Adding Loft Sections	Explains how to add loft sections to a completed loft feature
Cutting with a Loft	Explains how to remove material from a part using the loft feature

2016 Advanced Assemblies

Assembly Structure

Creating Sub-Assemblies	Shows several methods for Creating Sub-Assemblies
Flexible Sub-Assemblies	Explains how sub-assemblies will move when as either Rigid or Flexible
Empty Sub-Assemblies	Shows how to create a new sub-assembly from an empty sub-assembly
Folders vs. Sub-Assemblies	Explains the difference between using sub-assemblies and folders
Sub-Assemblies in Drawings	Explains how Sub-Assemblies will behave in the BOM of a Drawing

Component Patterns

Linear Patterns	Shows how to use a Linear Pattern to create more instances of a component in an assembly
Circular Patterns	Shows how to use a Circular Pattern to create more instances of a component in an assembly
Pattern Driven Patterns	Shows how to pattern components in an assembly based on a pattern feature from another component
Sketch Driven Pattern	Shows how to pattern components in an assembly based on a sketch
Curve Driven Pattern	Shows how to pattern components in an assembly based on a curve
Mirroring Components	Shows how to create more instances of a component using the Mirror Component command

Troubleshooting Mates

Assembly Mate Errors	Introduces Assembly Mate Errors and how they can occur
Repairing Mate Errors - Part A	Shows how to Identify and Repair Mate Errors
Repairing Mate Errors - Part B	Shows how to Identify and Repair Mate Errors
Mate Xpert	Shows how to use the Mate Diagnostics command to resolve mate issues
Replace Mate Entities	Shows how to reattach mate entities to replaced components

Assembly Configurations

Assembly Configurations - Part A	Shows how to create Assembly Configurations
Assembly Configurations - Part B	Shows how to create Assembly Configurations

Advanced Options	Shows how changes to one configuration will affect other configurations
Automatic Part Configurations	Shows how to create or specify configurations for a group of parts within an assembly
Assembly Design Tables - Part A	Shows how to utilize design tables at the assembly level
Assembly Design Tables - Part B	Shows how to utilize design tables at the assembly level
SpeedPak	Shows how to create simplified SpeedPak configurations

Top Down Assembly Design

Introduction to Top Down Assembly Design	Design components in the context of an assembly
Edit Part Mode	Shows the difference between Edit Part Mode and Assembly Mode
Component Visibility	Shows how to control the display of components in Edit Part mode.
In-Context Features - Part A	Shows how to add a feature to a component in the context of an assembly
In-Context Features - Part B	Shows how to add a feature to a component in the context of an assembly
Inserting a New Component	Shows how to create a new component in the context of an assembly
Remove External References - Part A	Shows how to remove or lock references from components/ features created within an assembly
Remove External References - Part B	Shows how to remove or lock references from components/ features created within an assembly
Remove External References - Part C	Shows how to remove or lock references from components/features created within an assembly
No External References	Shows how to design a component in the context of an assembly without external references.
Virtual Components	Shows how to create components that are contained within the assembly

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2016 Update Training

Fundamentals

Interface	Goes over the new Interface Enhancements, including dynamic highlight, expandable input boxes, and keyboard inputs
Selection Breadcrumbs	Selection breadcrumbs help you easily visualize the context of a selection and navigate to related elements
Confirmation Corner Options	Press "D" on the keyboard to move the confirmation corner to your cursor

Sketching

Convert and Offset Entities	Goes over the new Convert and Offset entities enhancements
Equal Length and Equal Radius Relations	You can add equal length and equal radius parametric relations between different types of sketch entities
Dimensions	Instant3D enhancements makes the dimensioning process more efficient
Segment Tool	Goes over the segment tool enhancements
Style Splines	Style Splines now support B-Splines of degree 3, 5, and 7
Unmerge Sketch Endpoints	You can unmerge a sketch entity at a single endpoint

Parts and Features

Curvature Continuous Edge Fillets	Shows how to create curvature continuous edge fillets
Helical Threads	The Helical Thread feature lets you model threads in a single step
Intersect Tool Enhancements	Goes over the intersect tool enhancements
Pattern Enhancements	New pattern enhancements include expanded functionality of linear patterns, removed pattern instance limitations, and excel data, integration for variable patterns
Reference Geometry	SOLIDWORKS added a few, small enhancements that make working with reference geometry easier than before
3D Curves	Reference curves are no longer absorbed by features, letting you reuse them for any feature.

Convert Surface into Solid	You can create solid bodies using the Boundary Surface and Trim Surface features
Flatten Surface Enhancements	Goes over the Flatten Surface enhancements
Sweeps	You can create bidirectional sweeps using the Sweep feature

Assemblies

Selecting and Copying Multiple Components	Goes over the new methods for copying and selecting multiple components at once
Mating Components	Goes over the new enhancements when mating components, like the Component Preview Window
Globally Replacing Failed Mate References	The "Replace Mate Entities" tool lets you globally replace failed mates at once
Mate Controller	The Mate Controller allows you to create different sets of mate values that you can easily recall to switch between different positions of your assembly
Mirroring Assembly Features	Assembly-level features can be mirrored in assemblies
Syncing Mirrored subassemblies	The movement of patterned or mirrored sub-assemblies can be synced together quickly and easily
Cleaning up Assemblies	"Purge Unused Features" lets you delete unused features in an assembly

Drawings & Detailing

Drawing Enhancements	You can quickly foreshorten linear dimensions and change the size of centermarks when working in drawings
Model Break Views	You can create Model Break Views in drawings
Part and Assembly Watermarks	You can add watermarks to parts and assemblies
Flag Notes	Flag notes are Balloon-style notes that are parametrically linked to notes that exist in a list elsewhere on the drawing
Custom Properties Enhancements	There are new options available when using custom properties that allow you to easily define properties in drawings containing subassemblies and multiple sheets
Sheet Enhancements and Automatic Borders	Goes over sheet enhancements, like editing the sheet scale from the status bar and editing the title block using the Title Block Fields icon

Sheet Metal

Swept Flanges with Cuts	Cuts you make in the bend region of a swept flange are now mapped correctly onto the flattened geometry
Edge Flanges	You can extend edge flange profiles beyond its adjoining edge
Flattened Mass	When working with Sheet Metal, it's now possible to show the Flattened-Mass of a part in the Cut-List Properties or Summary Information dialog boxes

Weldments

Weldment End Caps	When you create Weldment End Caps for both Linear and Curved Structural Members, SOLIDWORKS adds Reference Dimensions to mark the inset distance.
Structural Member Enhancements	When you create structural members, the size lists are ordered alphanumerically, making it easier to find profiles, and the structural member features are named based on the profile used
Cut List Properties for Derived Parts	SOLIDWORKS added a "Total Length" property for weldment derived parts
Transferring Material Properties from Library Profiles	When you're working with structural members, you have the option to transfer over the material properties from the library profiles you're using

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AutoCAD to SOLIDWORKS

Introduction

Introduction - Value of 2D Data	Introducing the transition between 2D to 3D
What is SOLIDWORKS?	Introduces SOLIDWORKS behavior and Design Intent
What is DraftSight?	Gives an overview of the DraftSight 2D program
Import Options	Explains how to import a .dwg or .dxf file and covers some of the options available

Importing to Drawings

Import to Drawing	Explains how to import a .dwg file into a SOLIDWORKS drawing
Embedded AutoCAD Drawing	Explains how to import to a SOLIDWORKS drawing as an embedded dwg format
Save as DXF or DWG	Shows how to save SOLIDWORKS drawings or drawing views back to DXF or DWG if needed

Importing to Part

Import to Part	Explains how to import a dwg file into a 3D model
Import to Part - Each Layer as Sketch	Explains how to import a dwg file into a 3D model, with each layer as its own sketch
Orienting and Aligning Sketches	Explains how to align and orient imported sketches
3D Features	Explains how to create 3D geometry from imported sketches
Import to Part - 3D	Explains how to import a dwg file that already has embedded 3D data into a 3D model
Blocks and Mechanisms	Explains how to import and work with sketch blocks in SOLIDWORKS

Drawing to Part Conversion

Drawing to Part Conversion	Introduces the section on converting 2D data from SOLIDWORKS Drawings to 3D parts
Convert Drawing View to Sketch	Shows how to convert a SOLIDWORKS Drawing view to a Sketch
Creating Parts from Drawings	Shows additional methods for creating parts from SOLIDWORKS drawings

Section Test
