What's New Highlights for SolidWorks 2009

SolidWorks

Contents

Notices

Patents

U.S. Patents 5,815,154; 6,219,049; 6,219,055; 6,603,486; 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,184,044; and foreign patents (e.g., EP 1,116,190 and JP 3,517,643). U.S. and foreign patents pending.

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Contractor/Manufacturer:

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Introduction

Innovative SolidWorks® software is the standard in 3D mechanical design. SolidWorks offers best-in-class solid modeling, 2D drawing, and design analysis capabilities, plus more time-saving innovations than any other 3D CAD system.

About this Book

This book provides highlights of the new functionality in SolidWorks 2009. For a comprehensive description of changes to this SolidWorks release and to see a fuller treatment of the features presented here, see the companion documentation:

- **What's New in SolidWorks 2009** introduces concepts and provides step-by-step examples for many of the new functions. Click **Help > What's New** to view this document.
- **Interactive What's New**. Click the question mark next to new menu items and the titles of new and changed PropertyManagers to view the descriptions from the SolidWorks 2009 What's New document.

About the Company

Dassault Systèmes SolidWorks Corporation, a Dassault Systèmes S.A. company, develops and markets software for design, analysis, and product data management. It is the leading supplier of CAD software, helping hundreds of thousands of users speed next-generation products to market around the world. With SolidWorks software, designers excel at their jobs and make their companies more successful by bringing designs to life.

For the latest news, information, or an online demonstration, visit www.soldworks.com or call 1-800-693-9000 (North America) or 1-978-371-5000 (world-wide).

Product Name Changes

COSMOS and SolidWorks products have been renamed. The table below lists name changes for products represented in this book.

Previous Name	New Name
SolidWorks 3D MCAD Software	SolidWorks® Standard
SolidWorks Office Premium	SolidWorks® Premium
SolidWorks Office Professional	SolidWorks® Professional
PDMWorks® Enterprise	SolidWorks® Enterprise PDM
PDMWorks Workgroup	SolidWorks® Workgroup PDM
COSMOS	SolidWorks® Simulation
COSMOSFloWorks	SolidWorks® Flow Simulation
COSMOSMotion™	SolidWorks® Motion
COSMOSM	SolidWorks® Simulation Premium
COSMOSWorks Advanced Professional	SolidWorks® Simulation Premium
COSMOSWorks Designer	SolidWorks® Simulation
COSMOSWorks Professional	SolidWorks® Simulation Professional

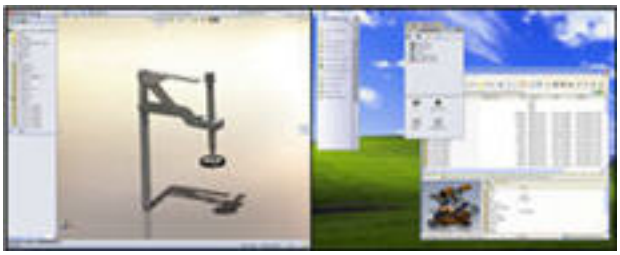
1 SolidWorks Standard

This chapter includes the following topics:

User Interface Enhancements

Multi-screen and Dock and Undock Support

Multiple and wide-screen monitors are now supported. You can also move the CommandManager to the top or side of the SolidWorks window and dock the PropertyManager on the top left or bottom right of the graphics area.

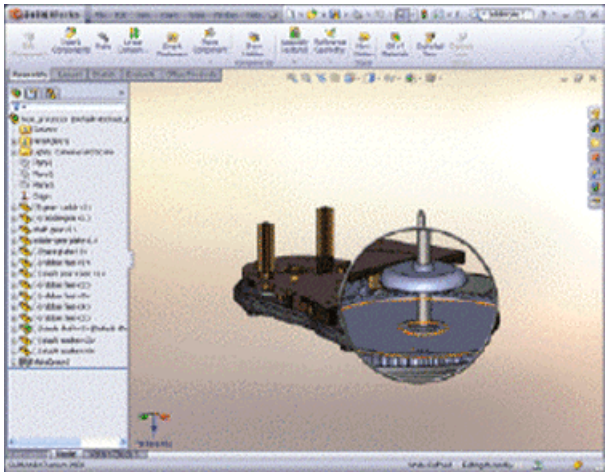


User Benefits

With multiple monitors and movable controls, you can have an unobstructed view of both the design and the commands.

Magnifying Glass

You can use the magnifying glass tool to inspect and select detailed areas of a model without changing its overall view.

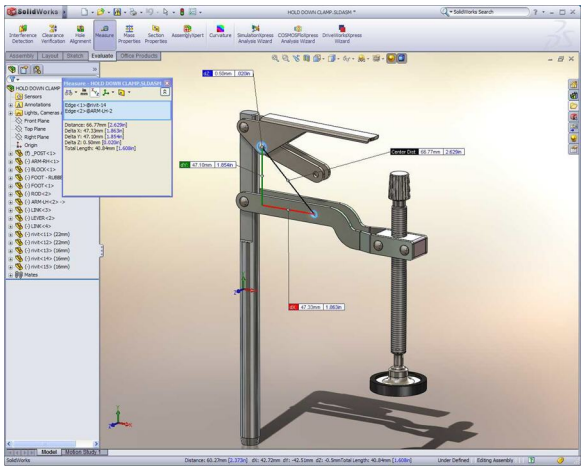


User Benefits

This tool lets you alternate between big-picture and highly detailed views without repetitive zooming.

Dual Units in the Measure Tool

You can now configure the Measure tool to display results in two different units of measure (for example, inches and millimeters).



User Benefits

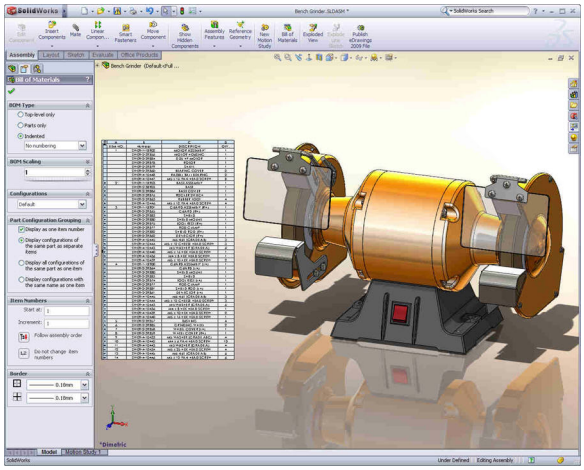
Working in dual units facilitates work in international environments. For example, designers from North America can work in inches while those from Europe or Japan can work in metric measurements.

Displaying dual units in the Measure tool can prevent unit conversion errors.

Assemblies

Assembly Bill of Materials

You can now create a bill of materials (BOM) directly in an assembly without first creating a drawing.

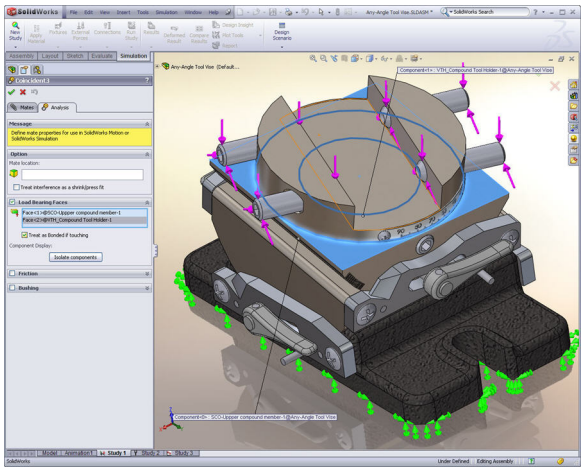


User Benefits

If you are not ready to create assembly drawings but need a BOM, creating one from the assembly saves time and lets managers project costs and margins earlier in the design process.

Mates for Simulation

You can now build design intent into SolidWorks mates to automate Simulation setup and part-to-part interactions across all studies in a model. You can define contact, shrink-fit, and bonding once, at the mate level.

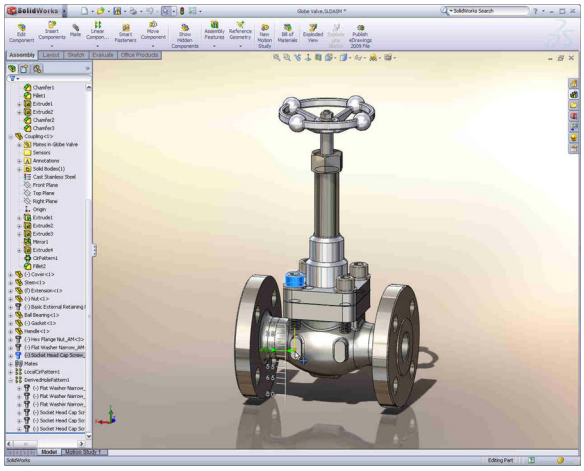


User Benefits

Capturing the design intent of interactions through mates lets you automate simulation tasks.

Dynamic Length Control of Toolbox Fasteners

SolidWorks now supports dynamic length control of fasteners supplied with Toolbox. The user interface is similar to that used in Instant3D.



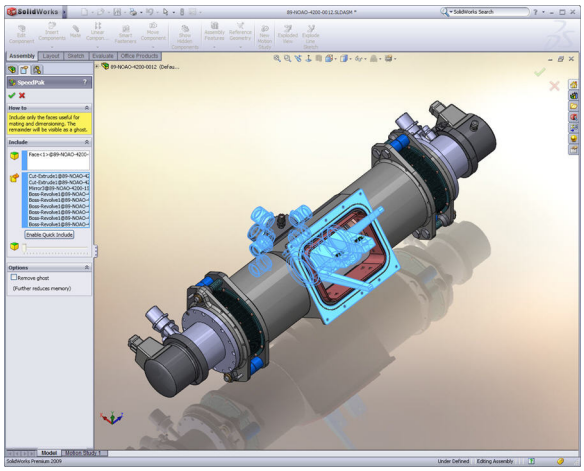
User Benefits

Dynamic length control enables adjustments to fastener length so that you can be sure to use the correct length for the application.

Large Assemblies

SpeedPak

The new SpeedPak technology creates a simplified version of a complex assembly without losing references. SpeekPak uses a subset of the parts or faces of an assembly, which reduces opening time, saves memory, and improves the performance of many operations.

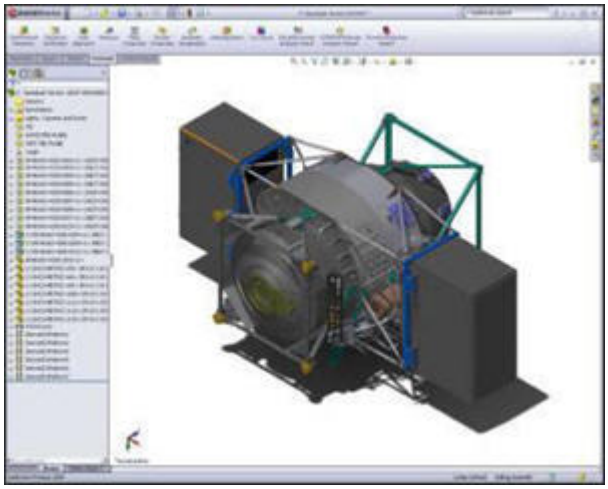


User Benefits

The larger and more complex the assembly, the greater the benefit SpeedPak provides.

Large Assembly Handling

Performance improvements affect window selection, manipulation of sub-assemblies, adding and editing mates, and working with drawings. You can now create assemblies larger than one kilometer.

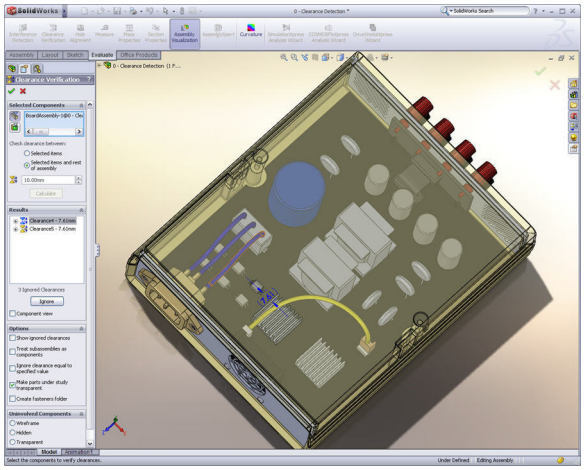


User Benefits

Refinements can be made more quickly to large assemblies and their corresponding drawings.

Clearance Verification

You can now use clearance verification to check the spacing between components in an assembly and report clearances that fail to meet minimum values. This capability complements Interference Detection.



Checking clearances between components, for example, between electronic controls, helps ensure proper performance.

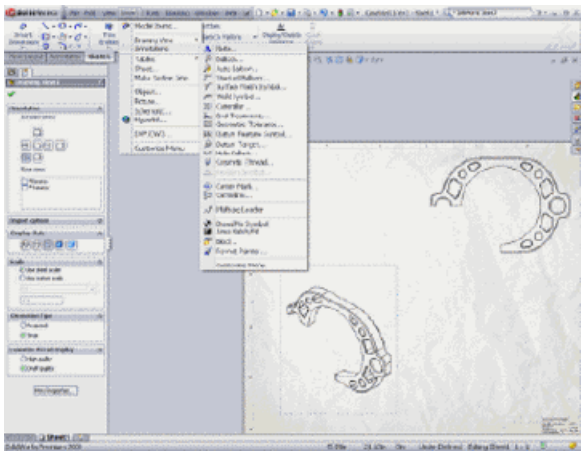
User Benefits

Checking clearances is particularly useful for producers of electronic packaging and energy conversion devices and designers in industries driven by agency approvals.

Drawings

Drawing - General Enhancements

Enhancements to drawings include support for customized detailing standards, extensive BOM editing, improved detail positioning control, new print options, custom line thicknesses and styles, and detailing for sketch slots.

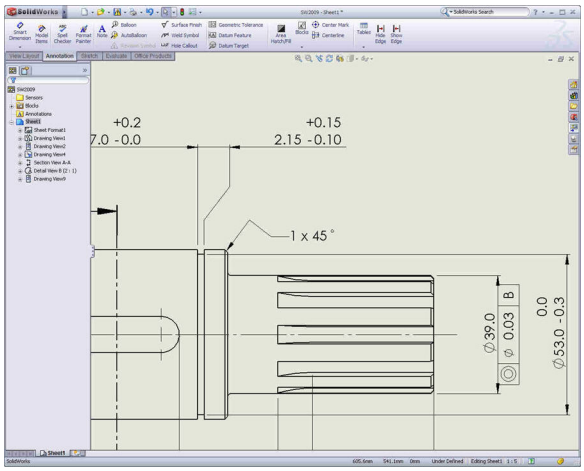


User Benefits

Drawing enhancements save time or improve drawing quality.

Dimension Jog

You can now jog an extension line.



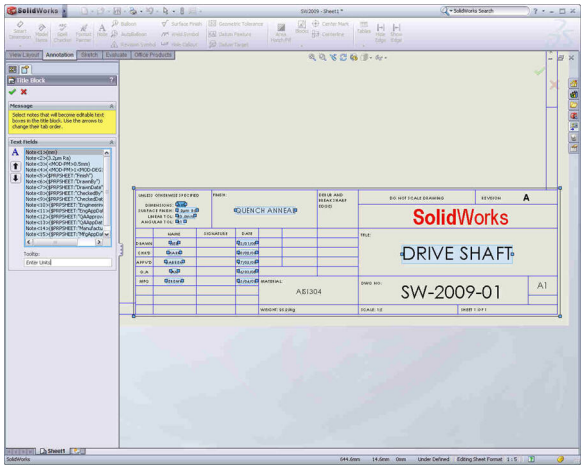
Jogging an extension line, such as the line identifying the snap ring groove, makes the drawing easier to understand.

User Benefits

Dimension jog makes details in drawings easier to read and interpret, reducing the chance of misinterpretation and corresponding quality problems in manufacturing.

Title Block

The **Title Block** PropertyManager streamlines the creation and editing of title block information.

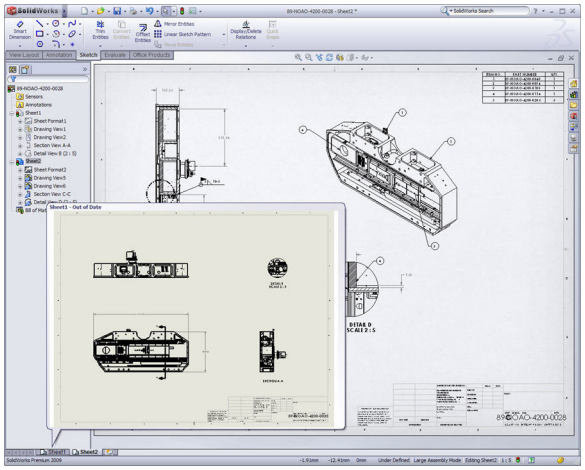


User Benefits

Standardization and consistent quality are easier to achieve.

Drawing Sheet Quick View

You can now get a preview of a drawing before you open it. Quick view is a read-only mode that provides a simplified drawing representation.

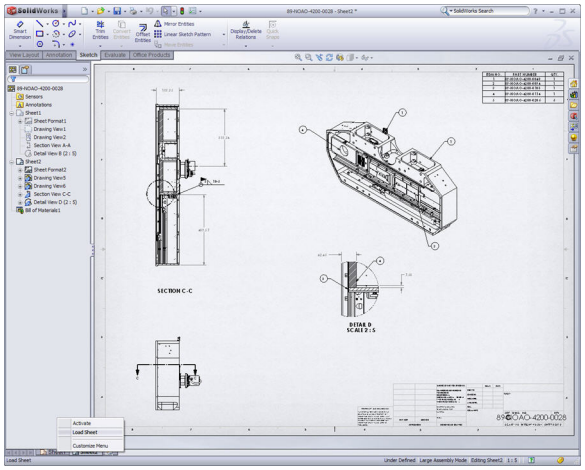


User Benefits

Designers can quickly check to see whether they are opening the right drawing. Managers can manage workflow and reviews more easily.

Load Sheet from Quick View

After reviewing a drawing using Quick view, you can select **Load Sheet** to edit it.

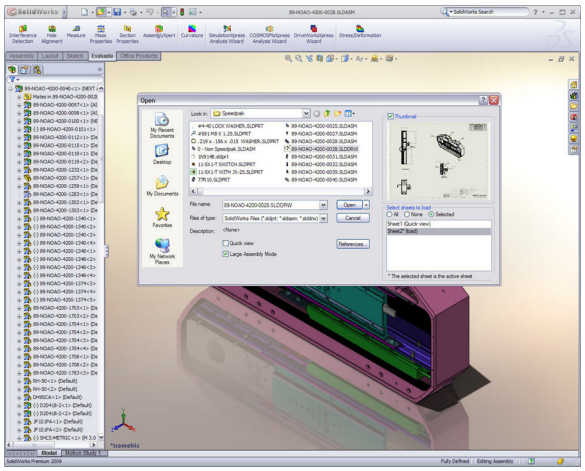


User Benefits

Loading only the sheet you need to work on is very fast.

Open Drawing to a Specific Sheet

Large, complex drawings typically have many sheets. You can now load a selected sheet while viewing all the others in Quick view.



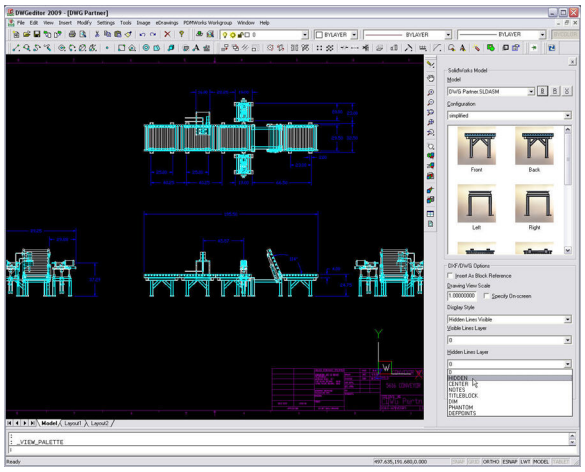
User Benefits

Checking a small detail on a large, multi-page drawing is now much faster.

DWGeditor

SolidWorks 2D View Palette

In 2D CAD systems such as DWGeditor®, you can use the View Palette to create drawings from SolidWorks parts and assemblies and add views to a drawing.



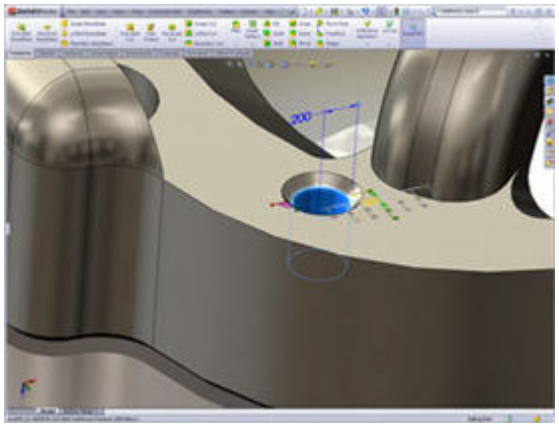
User Benefits

The View Palette lets 2D CAD users automatically create drawing views and link them to SolidWorks models. You can make changes in 2D CAD and link them back to SolidWorks models.

Instant3D

General Enhancements

The Instant3D editing tool now supports weldments and editing of internal sketch contours. Support for patterned and mirrored geometry is enhanced.

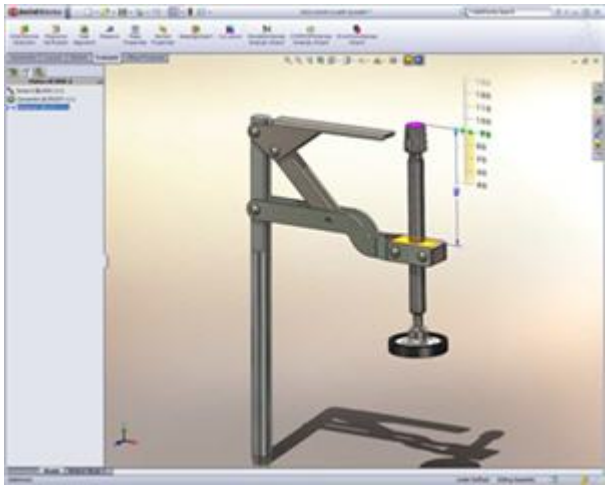


User Benefits

Instant3D supports click and drag and enables quick, intuitive edits with fewer mouse clicks.

Edit Parts in Assemblies

Instant3D can now be used to edit individual parts in an assembly and the relationships between them, such as mating criteria.

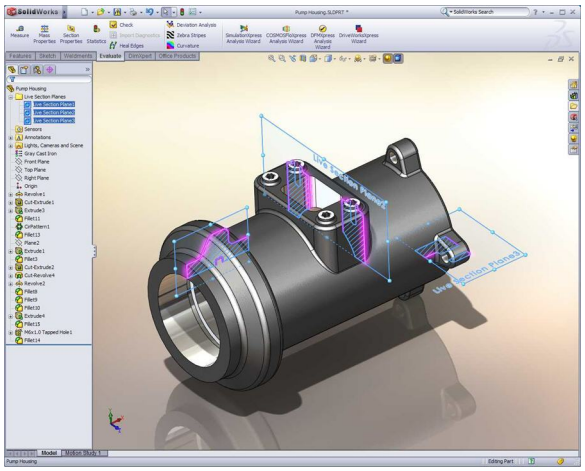


User Benefits

Using Instant3D for both parts and assemblies makes design editing easier.

Live Section Enhancements

Instant3D Live Section now supports the simultaneous use of multiple section planes, allowing you to edit different aspects of your design.



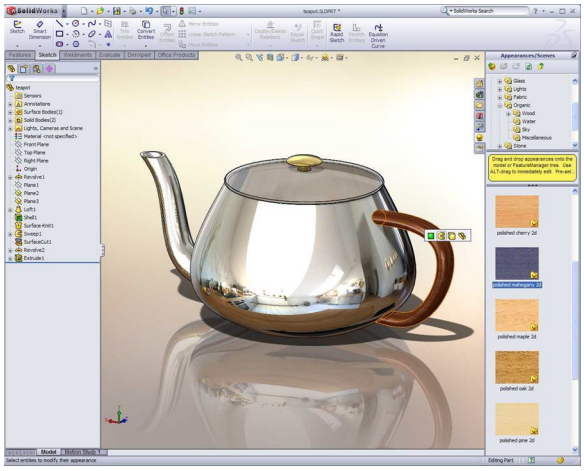
User Benefits

A designer can now use Instant3D on planes on more complex parts.

Materials and Appearances

Appearances Include Color and Textures

Colors and textures are now included in appearances. The visual attributes of a model are presented consistently in different modes, whether RealView is off or on or whether a model is rendered in PhotoWorks™.

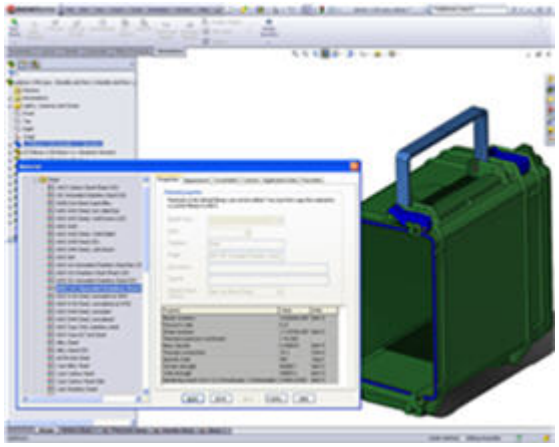


User Benefits

Appearances are now handled more consistently and are available in more places in the product.

Unified Materials Database and User Interface

SolidWorks and SolidWorks Simulation now share the same user interface for materials and share the same materials, including their physical properties, default crosshatching, and appearances. Custom materials you create are available for both design and simulation.



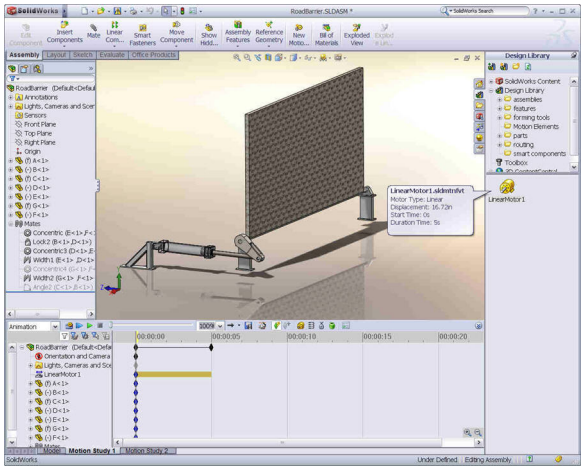
User Benefits

Having a common database for design and simulation saves time for users involved with both disciplines. It also helps users new to simulation get started with design testing.

Motion Studies

Design Library for Motion Elements

You can save motion elements such as springs, motors, or force specifications for reuse in other models.

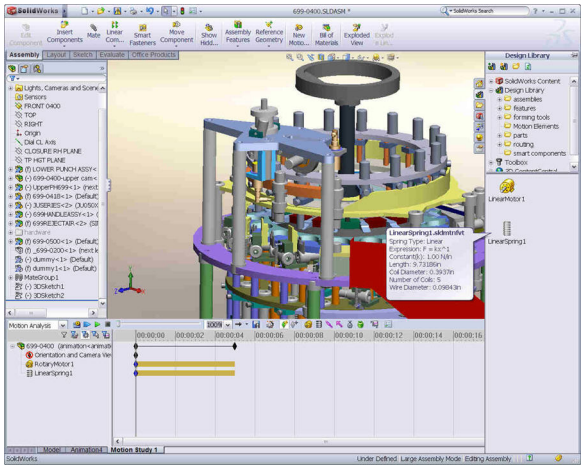


User Benefits

You can drag saved motion elements from the library to your model. Reusing design information reduces the risk that information will be lost.

Support for Lightweight Mode

You can run motion studies for an assembly in lightweight mode without first resolving it.



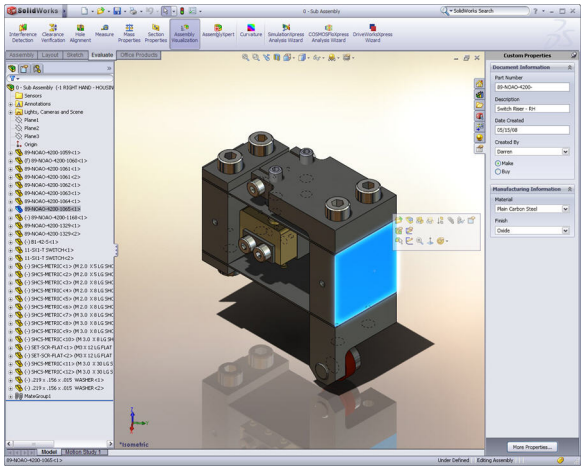
User Benefits

Productivity is increased when working with large assemblies in Motion simulation.

Parts

Custom Property Entry Form

You can now add custom properties information using a form that opens inside the Task Pane. You can add property data to a part itself or to the part while in assembly mode. Custom environments are available in part, assembly, and drawing environments.

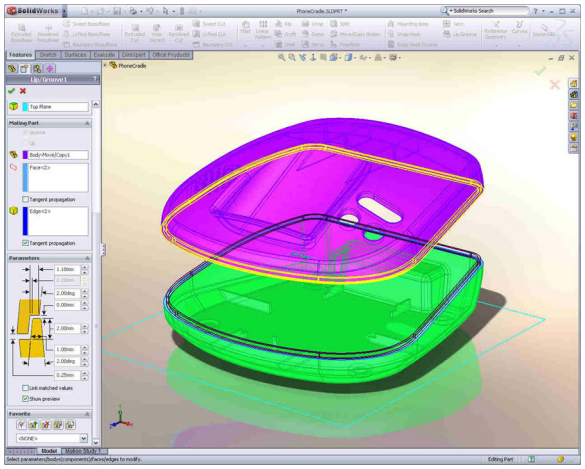


User Benefits

Adding custom property information to a part file enables searching, tracking, and passing data to a bill of materials.

Lip and Groove Feature

You can use lip and groove fastening features to align and assemble plastic parts automatically. When applying draft to a rib feature, you can specify the thickness of the rib at its intersection with the rest of the part.

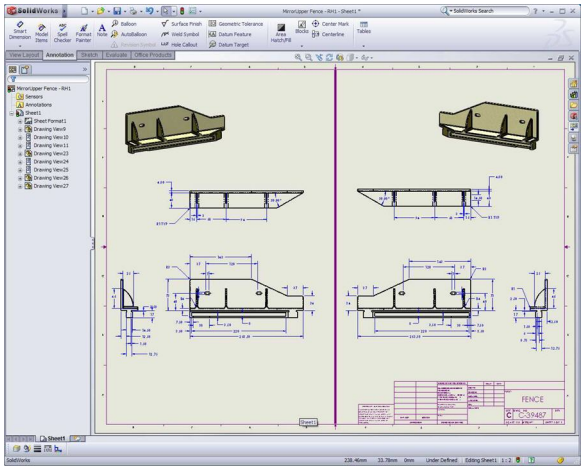


User Benefits

Using the lip and groove feature is faster than traditional methods. The new draft feature provides better control of the thickness of the rib, ensuring more uniform part thickness.

Mirrored Part with Dimensions

When creating a mirrored or derived part, you can now show sketch and feature dimensions of the original part in a drawing.



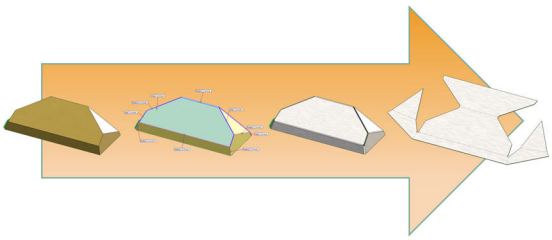
User Benefit

Users no longer need to manually recreate the dimensioning scheme on the mirrored part when creating a drawing.

Sheet Metal

Convert Solid to Sheet Metal

You can now generate a sheet metal part from a solid part.

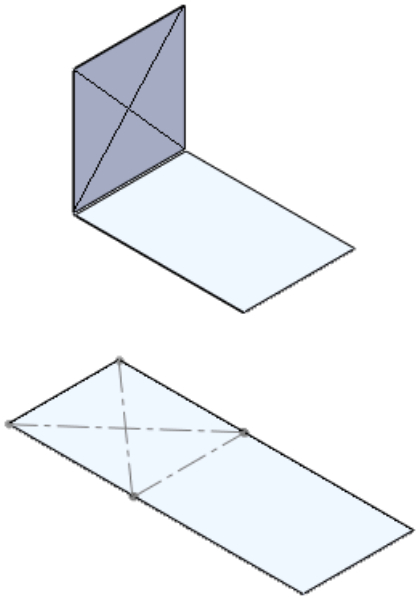


User Benefits

A solid part can be converted directly into a flattened sheet metal version. This is especially useful when a draft solid design is needed for a part that will later be manufactured as sheet meal.

Sheet Metal Cross Break

You can now add reinforcement cross breaks to sheet metal parts to stiffen the design, deflect water, and so on. The graphical representation of the cross break shows manufacturers where the cross break should be located.



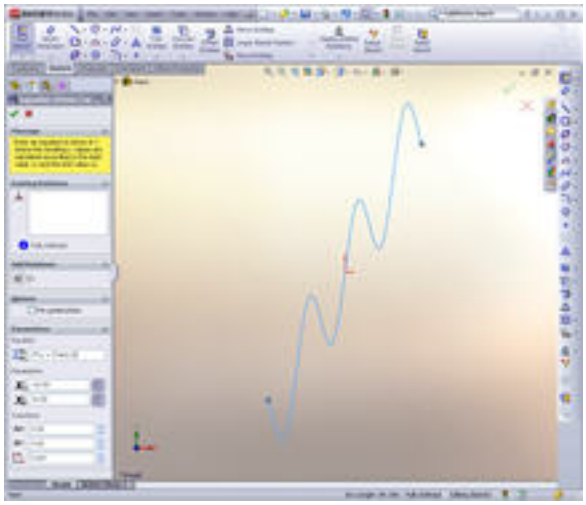
User Benefits

Graphical cross breaks are important design details for downstream manufacturing operations.

Sketch

Equation-Driven Curves

You can now use an equation to create a curve in a sketch.

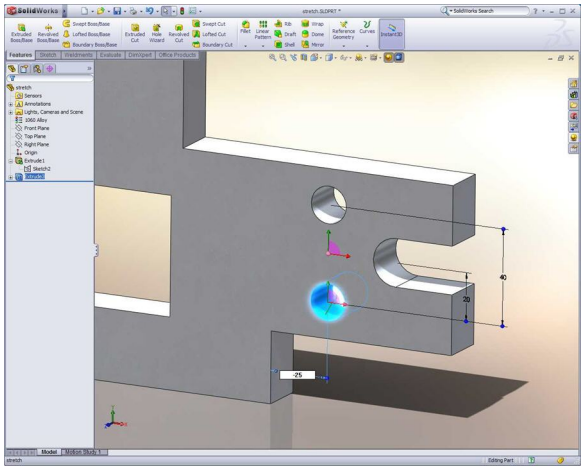


User Benefits

Using an equation to drive a curve adds flexibility and is faster than entering coordinates individually.

Zero and Negative Dimensions

You can now specify zero or negative values for sketch dimensions.

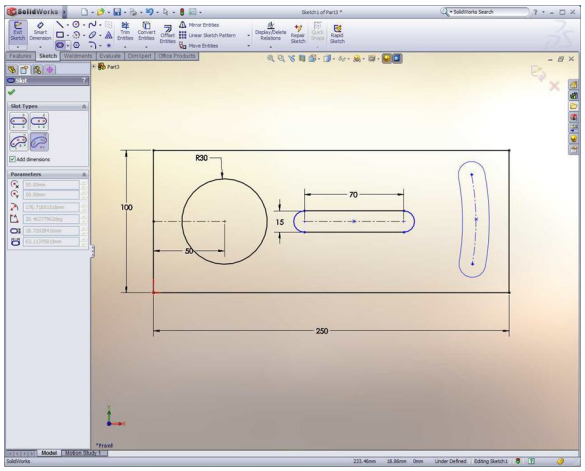


User Benefits

You do not need to use special techniques or change dimension references to reverse the direction of a dimension or set a dimension value to zero.

Slot Sketch Entity

You can use the new slot sketch entity to create four types of slots: straight, centerpoint straight, three-point arc, and centerpoint arc.

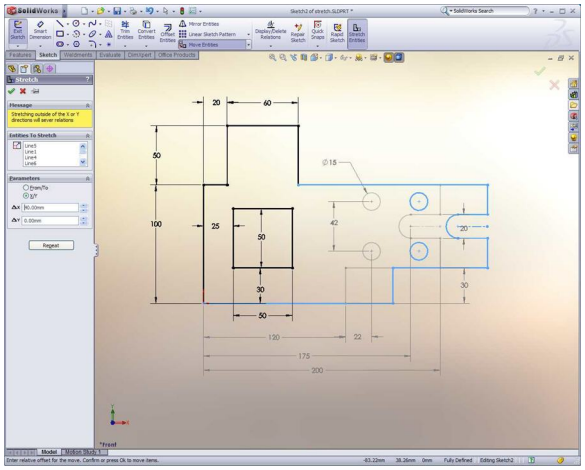


User Benefits

You do not need to sketch individual lines and arcs and assign sketch constraints.

Stretch Sketch Geometry

In a 2D sketch, you can now stretch multiple sketch entities as a single group rather than modifying each individually.



Select a group of entities to stretch together.

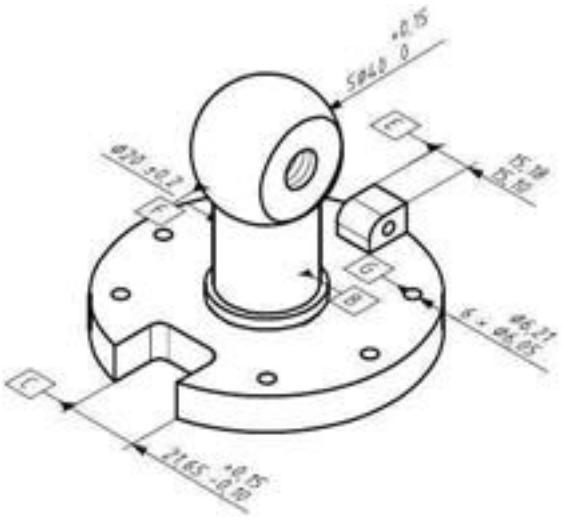
User Benefits

Treating multiple entities as a group saves time and mouse clicks.

Tolerancing

DimXpert ISO Support

DimXpert for parts now supports ISO standards 1101 and 16792, which specify the application of dimensions and tolerances and their 3D display.



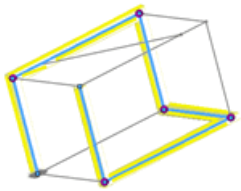
User Benefits

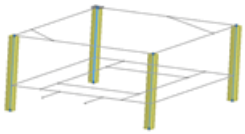
Designers who conform to the ISO standard can use DimXpert to automatically generate syntax and semantically correct dimensioning and tolerancing annotations directly on the 3D model.

Weldments

Weldment Groups

You can now define a weldment group, a collection of related segments in a structural member, and apply actions to the entire group.





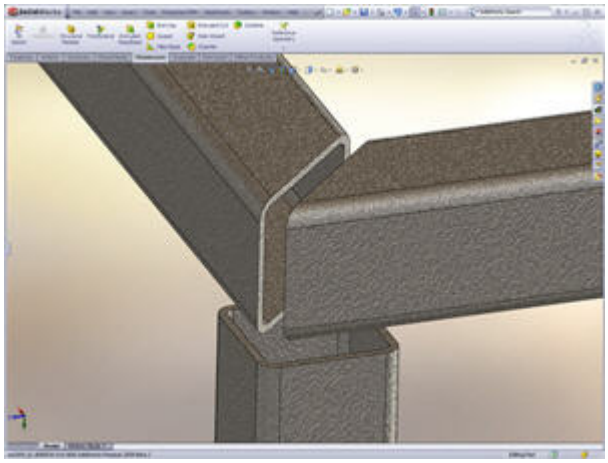
Weldment groups can contain contiguous segments or parallel segments.

Customer Benefits

You can perform operations on the group with one action, for example, changing corner treatments, creating weld gaps, or aligning all segments.

Weld Gaps

You can now add weld gaps between weldment segments to allow space for the weld bead.

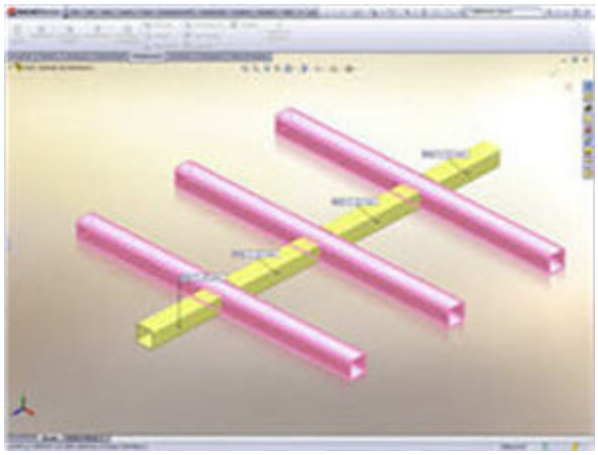


User Benefit

You can automatically extract cut length information that accounts for the gap for the weld beads, which saves design time, eliminates errors in calculating cut length, and ensures standard weld bead gaps.

Enhanced Trimming and Extending

The trim and extend tool lets you split intersecting segments, specify whether to keep or extend a split side, specify a weld gap between intersecting or abutting segments, or extend a segment.



User Benefits

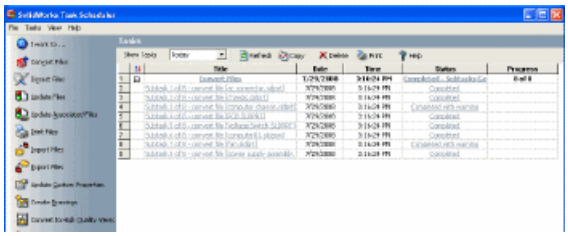
The functionality saves time over more manual methods and maintains the grouping of structural members.

2 SolidWorks Professional and Premium

This chapter includes the following topics:

File Conversions

File conversions are now performed by SolidWorks Task Scheduler rather than by the Conversion Wizard.



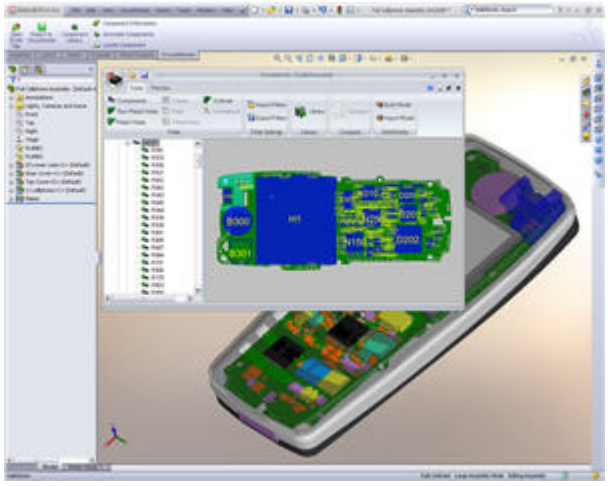
User Benefits

Administrators will see improved performance in Task Scheduler tasks. The increased speed of converting files when upgrading is especially significant.

CircuitWorks (Premium)

IDF and PADS ASCII Import

You can use the CircuitWorks™ add-in to automatically build SolidWorks assemblies from circuit boards and their components, showing outlines, keep-outs, regions, and annotations.



CircuitWorks displays a preview of the assembly and a tree view of the IDF data.

User Benefits

CircuitWorks supports the industry-standard Intermediate Data Format (IDF) used by a wide variety of Electrical CAD (ECAD) systems for printed circuit board design.

IDF Export

You can use CircuitWorks to export SolidWorks parts and assemblies in the IDF file format so they can be imported back into an ECAD system.

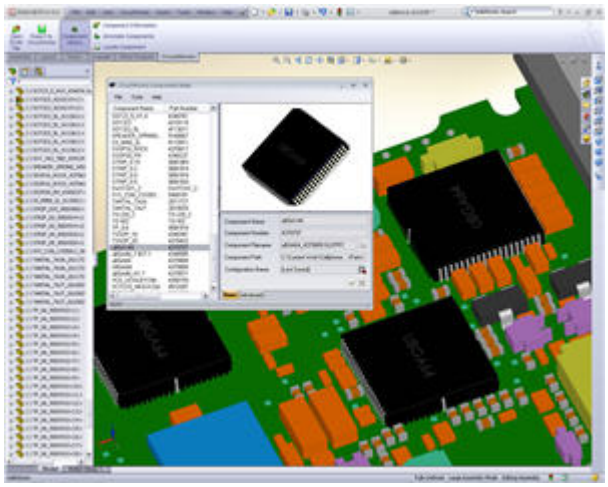
You can use the CircuitWorks compare tool to check for differences between two IDF files.

User Benefits

You can create complex PCB shapes in SolidWorks, transfer them accurately to the ECAD system, and then back into SolidWorks.

CircuitWorks Electrical Component Library

You can replace the components that CircuitWorks builds with more complex models, which CircuitWorks then uses in subsequent boards.



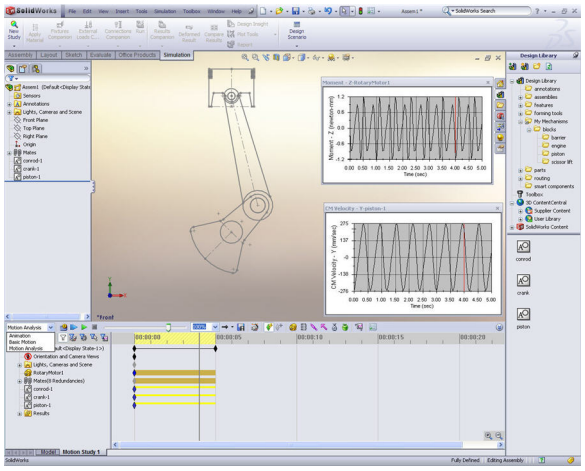
User Benefits

The library speeds the process of creating the final SolidWorks assembly. Each part you create is automatically added to the component library.

Motion Studies (Premium)

2D Mechanisms in Layout Sketches

You can now run animation and Motion Analysis studies for layout sketch mechanisms you create from sketch blocks. You can simulate the basic operation of 2D mechanisms in layout sketches before committing time to a detailed design.



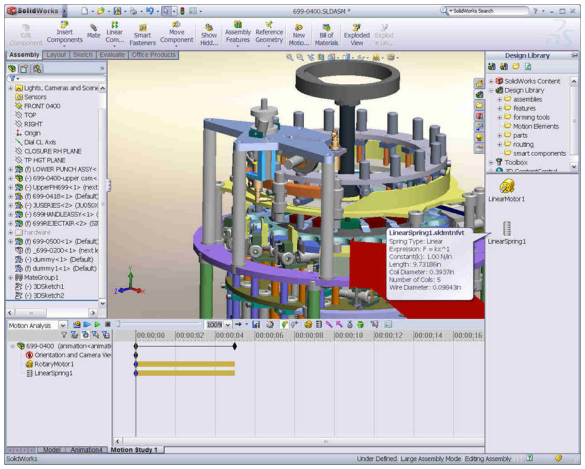
User Benefits

Experimenting in the sketching environment before creating 3D models can save significant time when implementing new design concepts.

You can simulate planar motion to improve and speed the design of 2D mechanisms.

Support for Lightweight Mode

You can run motion studies for an assembly in lightweight mode without first resolving it.



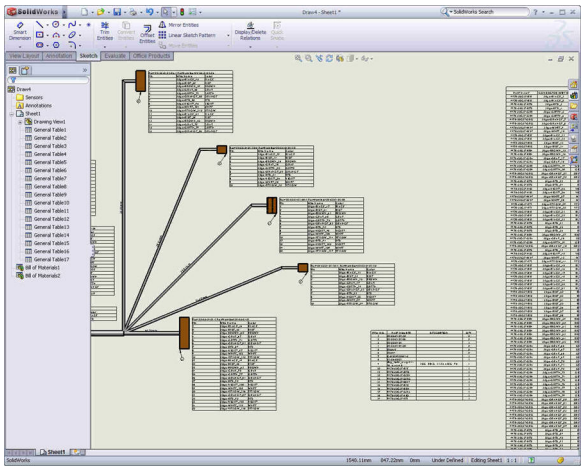
User Benefits

Productivity is increased when working with large assemblies in Motion simulation.

SolidWorks Routing (Premium)

Route Flattening and Drawing Clean-up

SolidWorks Routing can now create a flattened drawing of an electrical route such as a cable or harness with flattened views and documentation details such as circuit summaries and connector pin-out tables. Tables are automatically positioned to minimize the need for manual clean-up.



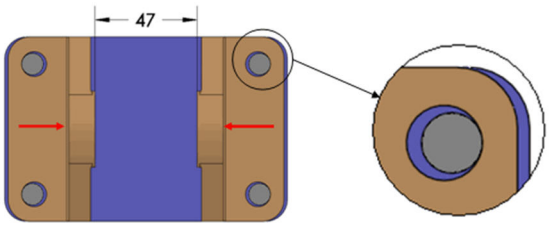
User Benefits

Manufacturing drawings for cables or harness routes are easer to create. The clean-up tasks typically required for harness drawings are also automated.

TolAnalyst (Premium)

Support for Fixed and Floating Fasteners

TolAnalyst™ now considers the clearances found in assemblies containing fixed and floating fasteners when calculating worst case tolerance conditions. When setting up TolAnalyst tolerance simulations, you can choose to float fasteners and pins.



User Benefits

You can more accurately simulate assembly conditions and test dimensioning and tolerancing schemes.

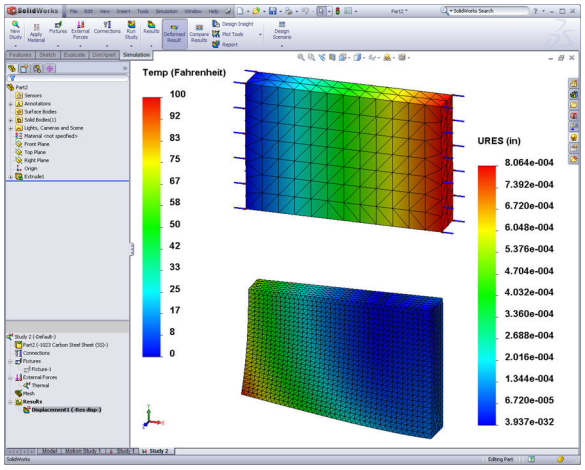
3
SolidWorks Simulation

This chapter includes the following topics:

SolidWorks Simulation Professional

Meshes for Thermal Temperature Studies

Interaction of thermal studies with static and nonlinear studies now accommodates dissimilar meshes. Temperatures from a thermal study with certain mesh properties are interpolated seamlessly to a static or nonlinear study with different mesh properties.



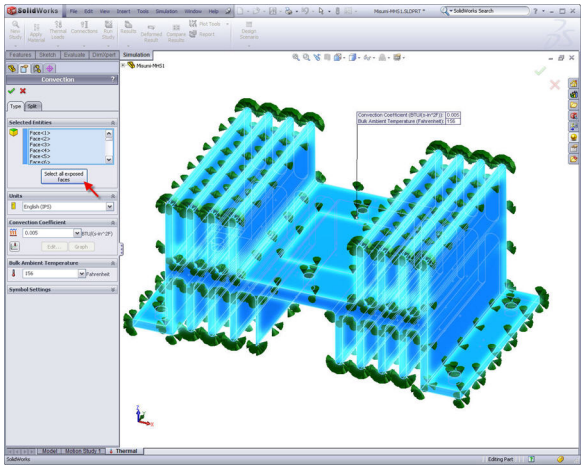
User Benefits

It is now easier and more productive to use temperature fields calculated by SolidWorks Simulation to determine thermal stress.

One-click Selection for Thermal Conditions

Click a single control to select all the exposed faces of a part or assembly to apply thermal loads. This new option is available in the **Temperature**, **Heat Flux**, **Heat Power**, **Radiation**,

and **Convection** PropertyManagers.

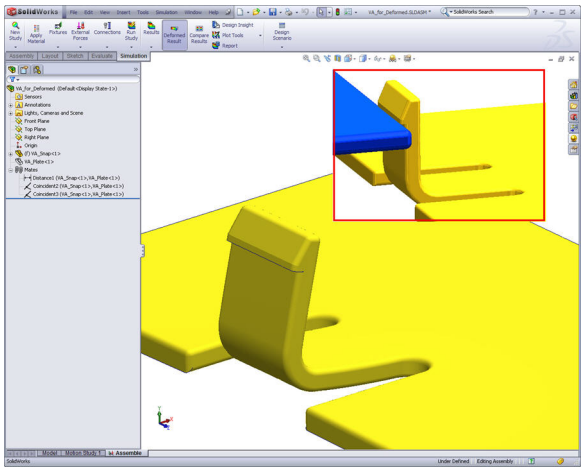


User Benefits

Design engineers get faster results by eliminating the time-consuming step of choosing all the faces required for a radiation or convection study.

Deformed SolidWorks Parts from Results

You can now save the deformed state of an assembly as a new multibody part document or as a new configuration.

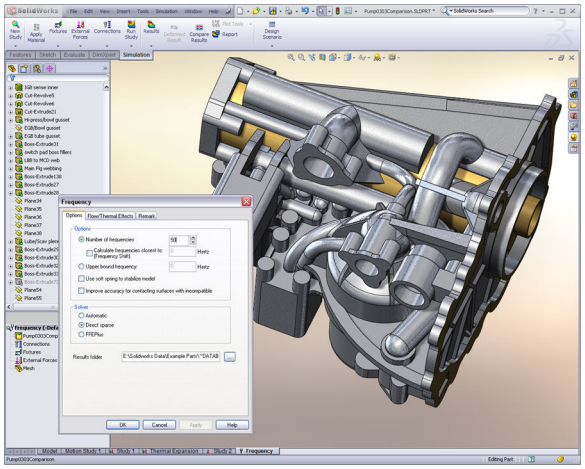


User Benefits

You can save a SolidWorks part or configuration that captures the deformed shape for more accurate assembly documentation or for use in a subsequent study.

64-bit Support for Frequency and Buckling Studies

Resource-intensive frequency and buckling studies can now take advantage of the expanded memory in 64-bit machines to improve performance for large problems.



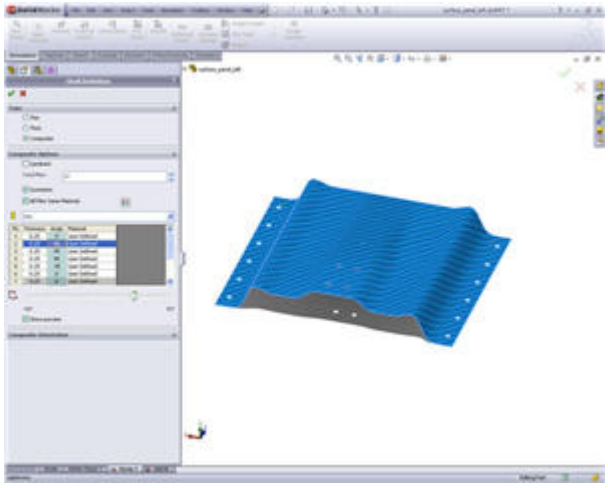
User Benefits

Design engineers no longer need to compromise on designs. The Sparse solver solves large models by optimizing memory allocation for frequency and buckling studies.

SolidWorks Simulation Premium

Composite Shells

You can now define composite shells with up to 50 layers. Ply orientation is displayed dynamically to provide visual as well as analytic feedback.

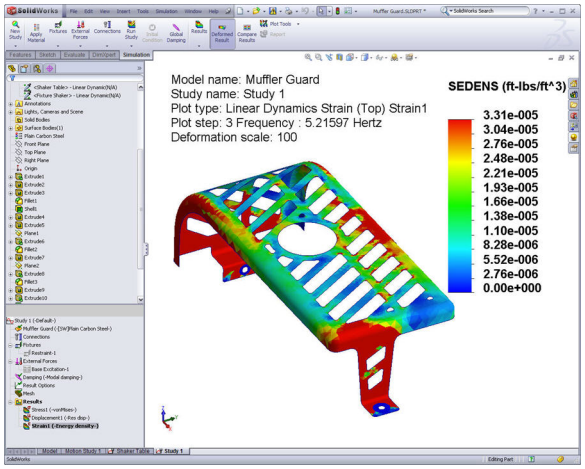


User Benefits

Design engineers can create composite materials easily. SolidWorks Simulation provides immediate visual feedback on composite layer orientation.

Energy Density in Linear Dynamics

Strain energy density plots have been added to the standard plot types in Linear Dynamics.

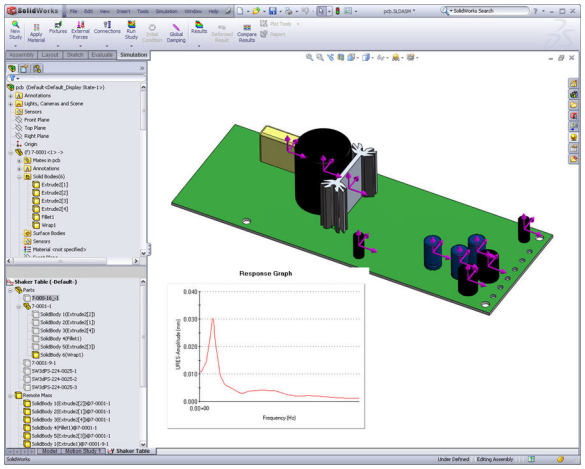


User Benefits

Strain energy density calculations help design engineers understand what portions of their design are most active in a dynamic situation. By reviewing these plots, designers can make better choices about improvements in vibratory environments.

Remote Mass in Linear Dynamics

For linear dynamic studies, designers now have the option to treat solid bodies as remote masses. Bodies treated as remote masses are excluded from meshing, but their mass properties and moments of inertia are considered in frequency and dynamic analyses.

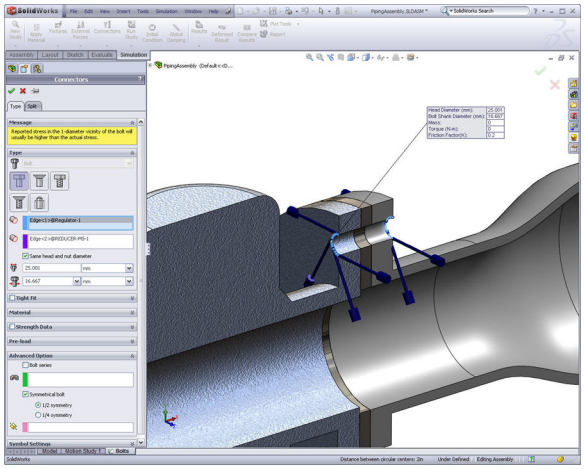


User Benefits

The application of remote mass reduces the mesh size and expedites the solution process, particularly for large assemblies in dynamic loading environments.

Symmetric and Multi-part Bolts in Nonlinear Studies

You can now define bolts on holes split by a symmetry plane or through more than two parts in a nonlinear study.



Multi-part and symmetric bolts can be defined in both static and nonlinear studies.

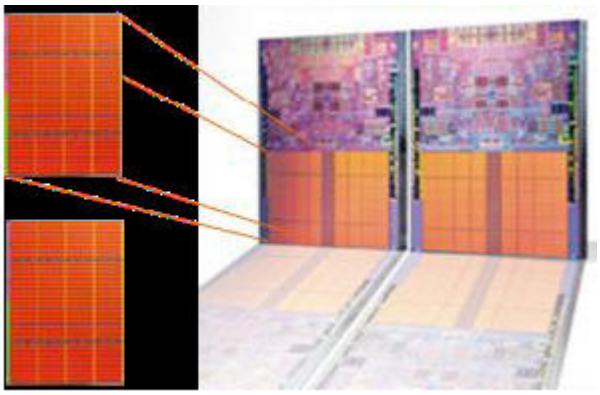
User Benefits

Design engineers will get faster results on their models by eliminating a time-consuming step.

SolidWorks Flow Simulation

Multi-processor Support

Flow solutions are iterative and can take a few hours to complete. Multi-core and multi-processor support enable Flow Simulation to run 1.3 to 1.5 times faster than in previous releases.

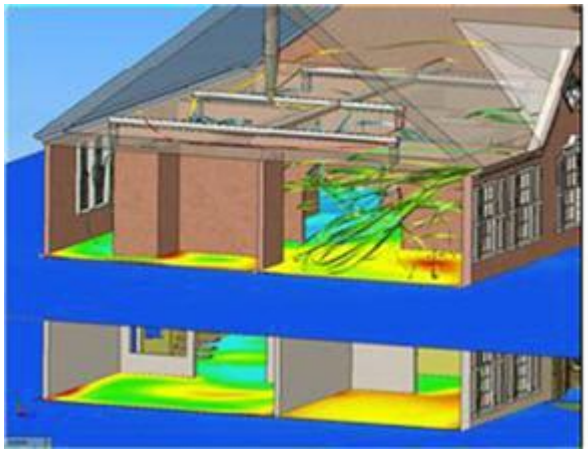


User Benefits

Multi-processor and multi-core support speeds meshing and solver activity.

Radiation and Non-Newtonian Models

You can now include bodies transparent to solar radiation. You can also test data on non-Newtonian fluids.

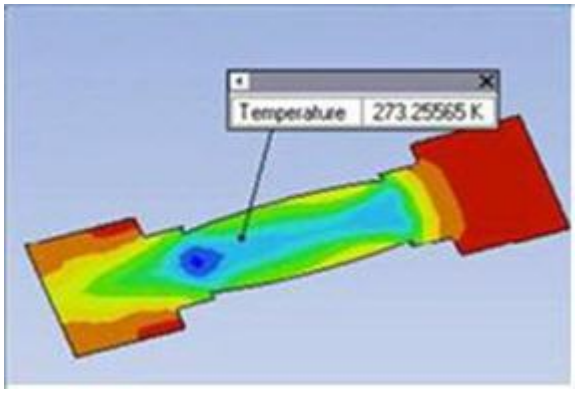


User Benefits

You can simulate greenhouse effects by including bodies transparent to solar radiation.

Improved Usability

The Flow Simulation user interface is now similar to the SolidWorks user interface. You can now edit callouts for loads and visualizing boundary conditions. You can also probe section plots to get feedback.



Users can probe section plots and get data to plot in a spreadsheet.

User Benefits

Flow Simulation is easier to use.

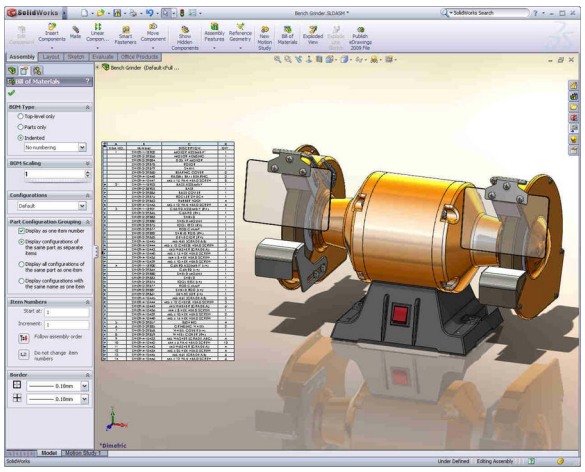
4
SolidWorks Enterprise PDM

This chapter includes the following topics:

Bill of Materials

SolidWorks Drawing and Assembly Bill of Materials

Bill of materials (BOM) tables created in SolidWorks drawing and assembly files are now displayed in Enterprise PDM. The BOMs are shown with the names assigned to them in the SolidWorks FeatureManager design tree.



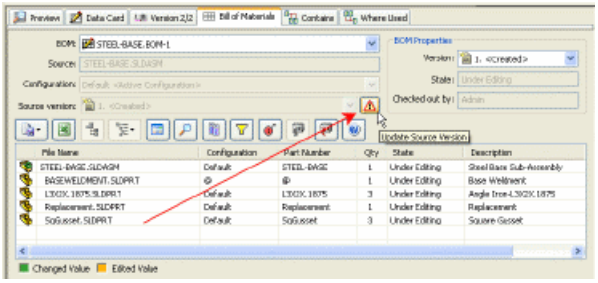
An assembly BOM in SolidWorks can now be displayed and edited in SolidWorks Enterprise PDM.

User Benefits

You can now check out, edit, and check in SolidWorks BOMs in Enterprise PDM.

Named BOMs

When you create a named BOM, the BOM is associated with its source - either a computed BOM or a SolidWorks drawing or assembly BOM. You can update the named BOM when you check in a new version of the source.



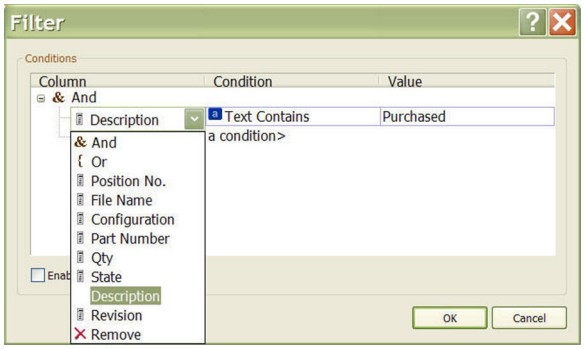
User Benefits

The display and editing of bills of materials in Enterprise PDM expands from a simple structure-based BOM to user-created Drawing and Assembly BOMs.

The BOM tab in the File Explorer view shows the Update Source Version, which allows associative Named BOMs.

Restructure BOMs and Apply Filters

You can now add rows, columns, and position numbers to named BOMs. You can rearrange and sort columns, and filter items based on the variables displayed in the table.



User Benefits

You can find and display information in BOMs more easily.

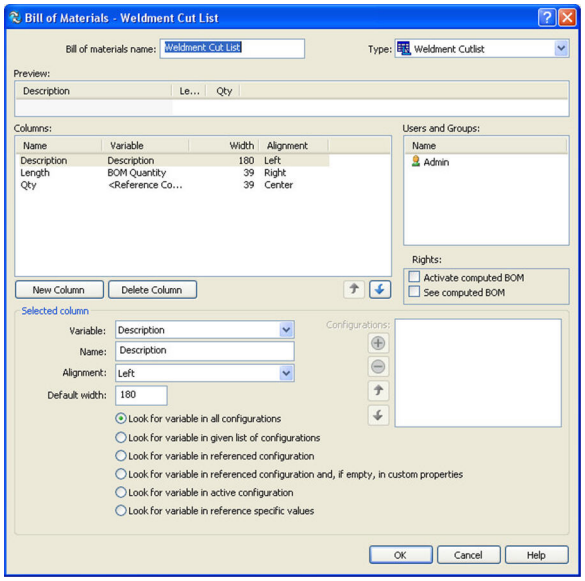
The **Filter** dialog box accessed from the BOM tab of the File Explorer lets you set up filters to apply to BOMs.

Cut Lists and Weldment BOMs

You can now display a cut list or a weldment BOM for a weldment part if these BOM types are configured for display by the administrator.

Customer Benefits

Users now have additional BOM viewing options within SolidWorks Enterprise PDM.



The **Bill of Materials List** dialog box in the Enterprise PDM Admin Tool lets you set parameters for the display of weldment cut lists.

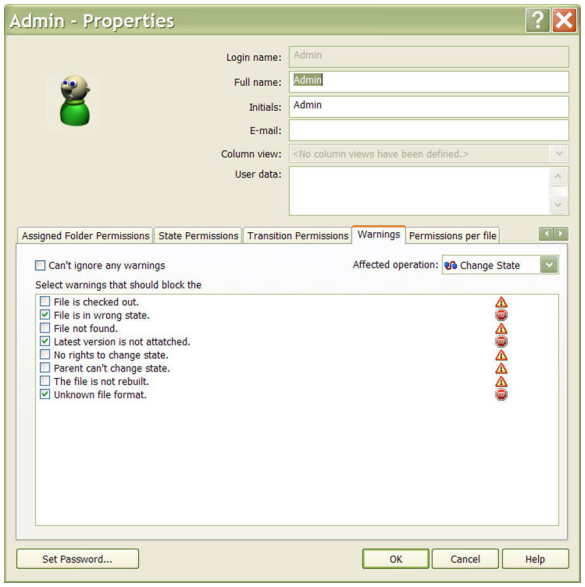
Security

Enhanced Security - Warnings Can Block Operations

If there are warnings for one or more files, users' ability to check in files can be limited. An administrator can select specific warnings to block check-ins or other operations.

User Benefits

Warnings give administrators tighter control over the approval process and ensure higher quality data and fewer manufacturing errors. For example, the use of warnings can ensure that a drawing reflect the latest changes to a part.



The Warnings tab of a user's **Properties** dialog box provides new block operation options.