

# SolidWorks – Glossary of Terms

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## A

### absorbed

A feature, sketch, or annotation that is contained in another item (usually a feature) in the FeatureManager design tree. Examples are the profile sketch and profile path in a base-sweep, or a cosmetic thread annotation in a hole.

### align

Tools that assist in lining up annotations and dimensions (left, right, top, bottom, and so on). For aligning parts in an assembly, see *mate*.

### alternate position view

A drawing view in which one or more views are superimposed in phantom lines on the original view. Alternate position views are often used to show range of motion of an assembly.

### anchor point

(1) The end of a leader that attaches to the note, block, or other annotation. See *attachment point*. (2) Sheet formats contain anchor points for a bill of materials, a hole table, a revision table, and a weldment cut list.

### annotation

A text note or a symbol that adds specific design intent to a part, assembly, or drawing. Specific types of annotations include note, hole callout, surface finish symbol, datum feature symbol, datum target, geometric tolerance symbol, weld symbol, balloon, and stacked balloon. Annotations that apply only to drawings include center mark, annotation centerline, area hatch, and block.

### appearance callouts

Callouts that display the colors and textures of the face, feature, body, and part under the entity selected and are a shortcut to editing colors and textures.

### area hatch

A crosshatch pattern or fill applied to a selected face or to a closed sketch in a drawing. See *crosshatch*.

### assembly

A document in which parts, features, and other assemblies (subassemblies) are mated together. The parts and subassemblies exist in documents separate from the assembly. For example, in an assembly, a piston can be mated to other parts, such as a connecting rod or cylinder. This new assembly can then be used as a subassembly in an assembly of an engine. The extension for a SolidWorks assembly file name is .SLDASM. See *subassembly*, .

### attachment point

The end of a leader that attaches to the model (to an edge, vertex, or face, for example) or to a drawing sheet. See *anchor point*.

### axis

A straight line that can be used to create model geometry, features, or patterns. An axis can be made in a number of different ways, including using the intersection of two planes. See *temporary axis*, *reference geometry*.

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## B

### balloon

Labels parts in an assembly, typically including item numbers and quantity. In drawings, the item numbers are related to rows in a bill of materials (BOM). See **stacked balloon**.

### base

The first solid feature of a part.

### baseline dimensions

Sets of dimensions measured from the same edge or vertex in a drawing. See *ordinate dimensions*.

### bend

A feature in a sheet metal part. A bend generated from a filleted corner, cylindrical face, or conical face is a round bend; a bend generated from sketched straight lines is a sharp bend.

### bevel

See *chamfer*.

### bill of materials

A table inserted into a drawing to keep a record of the parts used in an assembly.

### blend

See *fillet*.

### block

A user-defined annotation that you can use in parts, assemblies, and drawings. A block can contain text, sketch entities (except points), and area hatch, and it can be saved in a file for later use as, for example, a custom callout or a company logo.

## BOM

See *bill of materials*.

### bottom-up design

An assembly modeling technique where you create parts and then insert them into an assembly. See *top-down design*.

### bounding box

An imaginary box created by SolidWorks that completely encloses a model, component, or drawing view.

### broken-out section

A drawing view that exposes inner details of a drawing view by removing material from a closed profile, usually a spline.

## C

### cavity

See *mold*.

### center mark

A cross that marks the center of a circle or arc.

### centerline

A centerline marks, in phantom font, an axis of symmetry in a sketch or drawing.

### chamfer

Bevels a selected edge or vertex. You can apply chamfers to both sketches and features.

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## **child**

A dependent feature related to a previously-built feature. For example, a chamfer on the edge of a hole is a child of the parent hole.

## **click-click**

As you sketch, if you click and then release the pointer, you are in click-click mode. Move the pointer and click again to define the next point in the sketch sequence.

## **click-drag**

As you sketch, if you click and drag the pointer, you are in click-drag mode. When you release the pointer, the sketch entity is complete.

## **closed profile**

Also called a closed contour, it is a sketch or sketch entity with no exposed endpoints; for example, a circle or polygon.

## **collapse**

The opposite of explode. The collapse action returns an exploded assembly's parts to their normal positions.

## **Collision Detection**

An assembly function that detects collisions between components when components move or rotate. A collision occurs when an entity on one component coincides with any entity on another component.

## **component**

Any part or subassembly within an assembly

## **configuration**

A variation of a part or assembly within a single document. Variations can include different dimensions, features, and properties.

## **coordinate system**

A system of planes used to assign Cartesian coordinates to features, parts, and assemblies. Part and assembly documents contain default coordinate systems; other coordinate systems can be defined with reference geometry. Coordinate systems can be used with measurement tools and for exporting documents to other file formats.

## **cosmetic thread**

An annotation that represents threads.

## **crosshatch**

A pattern (or fill) applied to drawing views such as section views and broken-out sections.

## **curvature**

Curvature is equal to the inverse of the radius of the curve. The curvature can be displayed in different colors according to the local radius (usually of a surface).

## **cut**

A feature that removes material from a part by such actions as extrude, revolve, loft, sweep, thicken, cavity, and so on.

# SolidWorks – Glossary of Terms

## D

### dangling

A dimension, relation, or drawing section view that is unresolved. For example, if a piece of geometry is dimensioned, and that geometry is later deleted, the dimension becomes dangling.

### Defeature

With the Defeature tool, you can remove details from a part or assembly and save the results to a new file in which the details are replaced by dumb solids (that is, solids without feature definition or history). You can then share the new file without revealing all the design details of the model.

### degrees of freedom

Geometry that is not defined by dimensions or relations is free to move. In 2D sketches, there are three degrees of freedom: movement along the X and Y axes, and rotation about the Z axis (the axis normal to the sketch plane). In 3D sketches and in assemblies, there are six degrees of freedom: movement along the X, Y, and Z axes, and rotation about the X, Y, and Z axes. See *under defined*.

### derived part

A derived part is a new base, mirror, or component part created directly from an existing part and linked to the original part such that changes to the original part are reflected in the derived part.

### derived sketch

A copy of a sketch, in either the same part or the same assembly, that is connected to the original sketch. Changes in the original sketch are reflected in the derived sketch.

### Design Clipart

Using SolidWorks Search, Design Clipart searches specific folders, finds and dissects files, and extracts data that you can reuse in SolidWorks.

### Design Library

Located in the Task Pane, the Design Library provides a central location for reusable elements such as parts, assemblies, and so on.

### design table

An Excel spreadsheet that is used to create multiple configurations in a part or assembly document. See *configuration*.

### detached drawing

A drawing format that allows opening and working in a drawing without loading the corresponding models into memory. The models are loaded on an as-needed basis.

### detail view

A portion of a larger view, usually at a larger scale than the original view.

### dimension line

A linear dimension line references the dimension text to extension lines indicating the entity being measured. An angular dimension line references the dimension text directly to the measured object.

### DimXpertManager

Located on the left side of the SolidWorks window, it is a means to manage dimensions and tolerances created using DimXpert for parts.

# SolidWorks – Glossary of Terms

## **DimXpert for parts**

A set of tools that applies dimensions and tolerances to parts according to the requirements of the ASME Y.14.41-2003 standard.

## **DisplayManager**

The DisplayManager lists the appearances, decals, lights, scene, and cameras applied to the current model. From the DisplayManager, you can view applied content, and add, edit, or delete items. When PhotoView 360 is added in, the DisplayManager also provides access to PhotoView options.

## **dock point**

A point on an annotation, shown by a dashed red square, where you can attach a multi-jog leader.

## **document**

A file containing a part, assembly, or drawing.

## **draft**

The degree of taper or angle of a face, usually applied to molds or castings.

## **drawing**

A 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is .SLDDRW.

## **drawing sheet**

A page in a drawing document.

## **driven dimension**

See *reference dimension*.

## **driving dimension**

Also referred to as a model dimension, it sets the value for a sketch entity. It can also control distance, thickness, and feature parameters.

## **Dynamic Clearance**

An assembly function that detects the clearance between components when the components move or rotate. The clearance is the minimum distance between any entity on one component to any entity on another component.

## **E**

### **edge**

A single outside boundary of a feature.

### **edge flange**

A sheet metal feature that combines a bend and a tab in a single operation.

### **envelope**

A reference component that you use to select components based on their positions relative to the envelope volume. Envelopes are ignored in assembly operations such as bill of materials and mass properties.

### **equation**

Creates a mathematical relation between sketch dimensions, using dimension names as variables, or between feature parameters, such as the depth of an extruded feature or the instance count in a pattern.

### **exploded view**

Shows an assembly with its components separated from one another, usually to show how to assemble the mechanism.

# SolidWorks – Glossary of Terms

## export

Save a SolidWorks document in another format for use in other CAD/CAM, rapid prototyping, web, or graphics software applications.

## extension line

The line extending from the model indicating the point from which a dimension is measured.

## extrude

A feature that linearly projects a sketch to either add material to a part (in a base or boss) or remove material from a part (in a cut or hole).

## F

### face

A selectable area (planar or otherwise) of a model or surface with boundaries that help define the shape of the model or surface. For example, a rectangular solid has six faces. See *surface*.

### fasteners

See *Smart Fasteners*.

### feature

An individual shape that, combined with other features, makes up a part or assembly. Some features, such as bosses and cuts, originate as sketches. Other features, such as shells and fillets, modify a feature's geometry. However, not all features have associated geometry. Features are always listed in the FeatureManager design tree. See *surface, out-of-context feature*.

## feature lines

In ScanTo3D, feature lines form the boundaries between regions. You can edit feature lines in the Automatic Surface Creation PropertyManager.

## FeatureManager design tree

Located on the left side of the SolidWorks window, it provides an outline view of the active part, assembly, or drawing.

## fill

A solid area hatch or crosshatch. Fill also applies to patches on surfaces.

## fillet

An internal rounding of a corner or edge in a sketch, or an edge on a surface or solid.

## fillet corner

A corner where exactly three filleted edges meet at one vertex.

## Fit tolerance

The tolerance between a hole and a shaft.

## forming tool

Dies that bend, stretch, or otherwise form sheet metal to create such form features as louvers, lances, flanges, and ribs.

## fully defined

A sketch where all lines and curves in the sketch, and their positions, are described by dimensions or relations, or both, and cannot be moved. Fully defined sketch entities are shown in black.

# SolidWorks – Glossary of Terms

## G

### geometric tolerance

A set of standard symbols that specify the geometric characteristics and dimensional requirements of a feature.

### global variable

A variable that you define, for use in equations, custom properties, and so on.

### graphics area

The area in the SolidWorks window where the part, assembly, or drawing appears.

### Grid Systems

You can use Grid Systems to guide placement of structural members, locate equipment, or provide visual reference to the overall design.

### guide curve

A 2D or 3D curve used to guide a sweep or loft.

## H

### handle

An arrow, square, or circle that you can drag to adjust the size or position of an entity (a feature, dimension, or sketch entity, for example).

### helix

A curve defined by pitch, revolutions, and height. A helix can be used, for example, as a path for a swept feature cutting threads in a bolt.

### hem

A sheet metal feature that folds back at the edge of a part. A hem can be open, closed, double, or tear-drop.

### HLR

(hidden lines removed) A view mode in which all edges of the model that are not visible from the current view angle are removed from the display.

### HLV

(hidden lines visible) A view mode in which all edges of the model that are not visible from the current view angle are shown gray or dashed.

### hole table

A table that lists the size and location (from a specified origin datum) of specified holes in a drawing view.

### hollow

See *shell*.

## I

### import

Open files from other CAD software applications into a SolidWorks document.

### in-context feature

A feature with an external reference to the geometry of another component; the in-context feature changes automatically if the geometry of the referenced model or feature changes.

# SolidWorks – Glossary of Terms

## inferencing

The system automatically creates (infers) relations between dragged entities (sketched entities, annotations, and components) and other entities and geometry. This is useful when positioning entities relative to one another.

## instance

An item in a pattern or a component in an assembly that occurs more than once. Blocks are inserted into drawings as instances of block definitions.

## Instant3D

Functionality that lets you quickly create and modify model geometry using drag handles and rulers.

## interference detection

A tool that displays any interference between selected components in an assembly.

## iso-parametric curves

Curves that follow constant UV directions.

## J

## jog

(1) A sheet metal feature that adds material to a part by creating two bends from a sketched line. (2) A sketch tool that adds jogs to sketches.

K

## knit

A tool that combines two or more faces or surfaces into one. The edges of the surfaces must be adjacent and not overlapping, but they cannot ever be planar. There is no difference in the appearance of the face or the surface after knitting.

## L

## layer

A layer in a drawing can contain dimensions, annotations, geometry, and components. You can toggle the visibility of individual layers to simplify a drawing or assign properties to all entities in a given layer.

## layout sketch

A sketch that contains important sketch entities, dimensions, and relations. You reference the entities in the layout sketch when creating new sketches, building new geometry, or positioning components in an assembly. This allows for easier updating of your model because changes you make to the layout sketch propagate to the entire model.

## leader

A solid line from an annotation (note, dimension, and so on) to the referenced feature.

## library feature

A frequently used feature, or combination of features, that is created once and then saved for future use.

## lightweight

A part in an assembly or a drawing has only a subset of its model data loaded into memory. The remaining model data is loaded on an as-needed basis. This improves performance of large and complex assemblies. See *resolved*.

## line

A straight sketch entity with two endpoints. A line can be created by projecting an external entity such as an edge, plane, axis, or sketch curve into the sketch.

# SolidWorks – Glossary of Terms

## linked dimensions

See *shared values*.

## loft

A base, boss, cut, or surface feature created by transitions between profiles.

## lofted bend

A sheet metal feature that produces a roll form or a transitional shape from two open profile sketches. Lofted bends often create funnels and chutes.

## M

### mass properties

A tool that evaluates the characteristics of a part or an assembly such as volume, surface area, centroid, and so on.

### mate

A geometric relationship, such as coincident, perpendicular, tangent, and so on, between parts in an assembly. See *SmartMates*.

### mate reference

Specifies one or more entities of a component to use for automatic mating. When you drag a component with a mate reference into an assembly, the software tries to find other combinations of the same mate reference name and mate type.

### Mates folder

A collection of mates that are solved together. The order in which the mates appear within the Mates folder does not matter.

## mirror

(1) A mirror feature is a copy of a selected feature, mirrored about a plane or planar face. (2) A mirror sketch entity is a copy of a selected sketch entity that is mirrored about a centerline. If the original feature or sketch is modified, the mirrored copy is updated to reflect the change.

## miter flange

A sheet metal feature that joins multiple edge flanges together and miters the corner.

## model

3D solid geometry in a part or assembly document. If a part or assembly document contains multiple configurations, each configuration is a separate model.

## model dimension

A dimension specified in a sketch or a feature in a part or assembly document that defines some entity in a 3D model.

## model item

A characteristic or dimension of feature geometry that can be used in detailing drawings.

## model view

A drawing view of a part or assembly.

## mold

A set of manufacturing tooling used to shape molten plastic or other material into a designed part. You design the mold using a sequence of integrated tools that result in cavity and core blocks that are derived parts of the part to be molded.

# SolidWorks – Glossary of Terms

## Motion Studies

Motion Studies are graphical simulations of motion and visual properties with assembly models. Analogous to a configuration, they do not actually change the original assembly model or its properties. They display the model as it changes based on simulation elements you add.

## multibody part

A part with separate solid bodies within the same part document. Unlike the components in an assembly, multibody parts are not dynamic.

## N

### native format

DXF and DWG files remain in their original format (are not converted into SolidWorks format) when viewed in SolidWorks drawing sheets (view only).

### non-intersection contour

Also called a profile, it is a sketch in which entities do not cross each other. For example, a rectangle is a non-intersecting contour, whereas a cross intersects itself.

## O

### OLE object

(Object Linking and Embedding) A Windows file format. You can embed OLE objects in SolidWorks documents.

### open profile

Also called an open contour, it is a sketch or sketch entity with endpoints exposed. For example, a U-shaped profile is open.

## ordinate dimensions

A chain of dimensions measured from a zero ordinate in a drawing or sketch.

## origin

The model origin appears as three gray arrows and represents the (0,0,0) coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the (0,0,0) coordinate of the sketch. Dimensions and relations can be added to the model origin, but not to a sketch origin.

## out-of-context feature

A feature with an external reference to the geometry of another component that is not open. See *feature*.

## over defined

A sketch is over defined when dimensions or relations are either in conflict or redundant.

## overlay

See *alternate position view*.

## P

### parameter

A value used to define a sketch or feature (often a dimension).

### parent

An existing feature upon which other features depend. For example, in a block with a hole, the block is the parent to the child hole feature.

# SolidWorks – Glossary of Terms

## **part**

A single 3D object made up of features. A part can become a component in an assembly, and it can be represented in 2D in a drawing. Examples of parts are bolt, pin, plate, and so on. The extension for a SolidWorks part file name is .SLDPRT. See *multibody part*.

## **path**

A sketch, edge, or curve used in creating a sweep or loft.

## **pattern**

A pattern repeats selected sketch entities, features, or components in an array, which can be linear, circular, or sketch-driven. If the seed entity is changed, the other instances in the pattern update.

## **Physical Dynamics**

An assembly tool that displays the motion of assembly components in a realistic way. When you drag a component, the component applies a force to other components it touches. Components move only within their degrees of freedom.

## **pierce relation**

Makes a sketch point coincident to the location at which an axis, edge, line, or spline pierces the sketch plane.

## **planar**

Entities that can lie on one plane. For example, a circle is planar, but a helix is not.

## **plane**

Flat construction geometry. Planes can be used for a 2D sketch, section view of a model, a neutral plane in a draft feature, and others.

## **point**

A singular location in a sketch, or a projection into a sketch at a single location of an external entity (origin, vertex, axis, or point in an external sketch). See *vertex*.

## **predefined view**

A drawing view in which the view position, orientation, and so on can be specified before a model is inserted. You can save drawing documents with predefined views as templates.

## **profile**

A sketch entity used to create a feature (such as a loft) or a drawing view (such as a detail view). A profile can be open (such as a U shape or open spline) or closed (such as a circle or closed spline).

## **projected dimension**

If you dimension entities in an isometric view, projected dimensions are the flat dimensions in 2D. See *true dimension*.

## **projected view**

A drawing view projected orthogonally from an existing view.

## **PropertyManager**

Located on the left side of the SolidWorks window, it is used for dynamic editing of sketch entities and most features.

# SolidWorks – Glossary of Terms

## R

### RealView

A hardware (graphics card) support of advanced shading in real time; the rendering applies to the model and is retained as you move or rotate a part.

### rebuild

Tool that updates (or regenerates) the document with any changes made since the last time the model was rebuilt. Rebuild is typically used after changing a model dimension.

### reference dimension

A dimension in a drawing that shows the measurement of an item, but cannot drive the model and its value cannot be modified. When model dimensions change, reference dimensions update.

### reference geometry

Includes planes, axes, coordinate systems, and 3D curves. Reference geometry is used to assist in creating features such lofts, sweeps, drafts, chamfers, and patterns. See *construction geometry*.

### referenced document

Any document that is referenced by another, typically part documents associated with an assembly or drawing documents associated with part or assembly documents.

### relation

A geometric constraint between sketch entities or between a sketch entity and a plane, axis, edge, or vertex. Relations can be added automatically or manually.

### relative view

A relative (or relative to model) drawing view is created relative to planar surfaces in a part or assembly.

### reload

Refreshes shared documents. For example, if you open a part file for read-only access while another user makes changes to the same part, you can reload the new version, including the changes.

### reorder

Reordering (changing the order of) items is possible in the FeatureManager design tree. In parts, you can change the order in which features are solved. In assemblies, you can control the order in which components appear in a bill of materials.

### replace

Substitutes one or more open instances of a component in an assembly with a different component.

### resolved

A state of an assembly component (in an assembly or drawing document) in which it is fully loaded in memory. All the component's model data is available, so its entities can be selected, referenced, edited, used in mates, and so on. See *lightweight*.

### revision table

A table that lists the revisions of a drawing.

### revolve

A feature that creates a base or boss, a revolved cut, or revolved surface by revolving one or more sketched profiles around a centerline.

# SolidWorks – Glossary of Terms

## **rip**

A sheet metal feature that removes material at an edge to allow a bend.

## **rollback**

Suppresses all items below the rollback bar.

## **round**

See *fillet*.

## **round bend**

See *bend*.

## **Routing Library Manager**

The Routing Library Manager can be opened independently of the SolidWorks application, and groups several functions together.

## **S**

### **section**

Another term for profile in sweeps.

### **section line**

A line or centerline sketched in a drawing view to create a section view.

### **section scope**

Specifies the components to be left uncut when you create an assembly drawing section view.

### **section view**

A section view (or section cut) is (1) a part or assembly view cut by a plane, or (2) a drawing view created by cutting another drawing view with a section line.

## **seed**

A sketch or an entity (a feature, face, or body) that is the basis for a pattern. If you edit the seed, the other entities in the pattern are updated.

## **shaded**

Displays a model as a colored solid. See *HLR*, *HLV*, and *wireframe*.

## **shared values**

Also called linked dimensions, these are named variables that you assign to set the value of two or more dimensions to be equal.

## **sharp**

A hard corner of a profile; any two contiguous sketch entities that do not have a tangent or equal curvature relation with each other.

## **sharp bend**

See *bend*.

## **sheet**

See drawing sheet.

## **sheet format**

Includes page size and orientation, standard text, borders, title blocks, and so on. Sheet formats can be customized and saved for future use. Each sheet of a drawing document can have a different format.

## **shell**

A feature that hollows out a part, leaving open the selected faces and thin walls on the remaining faces. A hollow part is created when no faces are selected to be open.

# SolidWorks – Glossary of Terms

## **silhouette edge**

The curve representing the extent of a cylindrical or curved face when viewed from the side.

## **sketch**

A collection of lines and other 2D objects on a plane or face that forms the basis for a feature such as a base or a boss. A 3D sketch is non-planar and can be used to guide a sweep or loft, for example.

## **Smart Fasteners**

Automatically adds fasteners (bolts and screws) to an assembly using the SolidWorks Toolbox library of fasteners.

## **SmartMates**

An assembly mating relation that is created automatically. See *mate*.

## **solid sweep**

A cut sweep created by moving a tool body along a path to cut out 3D material from a model. See *sweep*.

## **spiral**

A flat or 2D helix, defined by a circle, pitch, and number of revolutions.

## **spline**

A sketched 2D or 3D curve defined by a set of control points.

## **split line**

Projects a sketched curve onto a selected model face, dividing the face into multiple faces so that each can be selected individually. A split line can be used to create draft features, to create face blend fillets, and to radiate surfaces to cut molds.

## **stacked balloon**

A set of balloons with only one leader. The balloons can be stacked vertically (up or down) or horizontally (left or right).

## **standard 3 views**

The three orthographic views (front, right, and top) that are often the basis of a drawing.

## **stereolithography**

The process of creating rapid prototype parts using a faceted mesh representation in STL files.

## **subassembly**

An assembly document that is part of a larger assembly. For example, the steering mechanism of a car is a subassembly of the car.

## **suppress**

Removes an entity from the display and from any calculations in which it is involved. You can suppress features, assembly components, and so on. Suppressing an entity does not delete the entity; you can unsuppress the entity to restore it.

## **surface**

A zero-thickness planar or 3D entity with edge boundaries. Surfaces are often used to create solid features. Reference surfaces can be used to modify solid features. See *face*.

## **sweep**

Creates a base, boss, cut, or surface feature by moving a profile (section) along a path. For cut-sweeps, you can create solid sweeps by moving a tool body along a path.

# SolidWorks – Glossary of Terms

## T

### tangent arc

An arc that is tangent to another entity, such as a line.

### tangent edge

The transition edge between rounded or filleted faces in hidden lines visible or hidden lines removed modes in drawings.

### Task Pane

Located on the right-side of the SolidWorks window, the Task Pane contains SolidWorks Resources, the Design Library, and the File Explorer.

### template

A document (part, assembly, or drawing) that forms the basis of a new document. It can include user-defined parameters, annotations, predefined views, geometry, and so on.

### temporary axis

An axis created implicitly for every conical or cylindrical face in a model.

### thin feature

An extruded or revolved feature with constant wall thickness. Sheet metal parts are typically created from thin features.

### TolAnalyst

A tolerance analysis application that determines the effects that dimensions and tolerances have on parts and assemblies.

### top-down design

An assembly modeling technique where you create parts in the context of an assembly by referencing the geometry of other components. Changes to the referenced components propagate to the parts that you create in context. See *bottom-up design.translator*

Software that converts a file from one format to another.

### triad

Three axes with arrows defining the X, Y, and Z directions. A reference triad appears in part and assembly documents to assist in orienting the viewing of models. Triads also assist when moving or rotating components in assemblies.

### true dimension

If you dimension entities in an isometric view, true dimensions give you accurate model values. See *projected dimension*.

## U

### unabsorbed

A sketch or annotation that is not contained in a feature in the FeatureManager design tree. An example of an unabsorbed sketch is a layout sketch in an assembly. See *absorbed*.

### under defined

A sketch is under defined when there are not enough dimensions and relations to prevent entities from moving or changing size. See *degrees of freedom*.

### UV

Horizontal and vertical lines of the underlying parameterization of a curve.

# SolidWorks – Glossary of Terms

## V

### vertex

A point at which two or more lines or edges intersect. Vertices can be selected for sketching, dimensioning, and many other operations.

### viewports

Windows that display views of models. You can specify one, two, or four viewports. Viewports with orthogonal views can be linked, which links orientation and rotation.

### virtual sharp

A sketch point at the intersection of two entities after the intersection itself has been removed by a feature such as a fillet or chamfer. Dimensions and relations to the virtual sharp are retained even though the actual intersection no longer exists.

## W

### weldment

A multibody part with structural members.

### weldment cut list

A table that tabulates the bodies in a weldment along with descriptions and lengths.

### wireframe

A view mode in which all edges of the part or assembly are displayed. See *HLR*, *HLV*, *shaded*.

### witness line

See *extension line*.

## X

### X display

The size of the X in a chamfer dimension with two numbers, such as 1 X 45° (Length X Angle), 45° X 1 (Angle X Length) or 1 X 1 (Length X Length).

## Z

### zebra stripes

Simulate the reflection of long strips of light on a very shiny surface. They allow you to see small changes in a surface that may be hard to see with a standard display.