Glossary

animate View a model or eDrawing in a dynamic manner. Animation simulates motion or displays different views.

assembly An assembly is 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.

axis An axis is 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.

boss/base A base is the first solid feature of a part, created by a boss. A boss is a feature that creates the base of a part, or adds material to a part, by extruding, revolving, sweeping, or lofting a sketch, or by thickening a surface.

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 A closed profile (or closed contour) is a sketch or sketch entity with no exposed endpoints; for example, a circle.

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

component A component is any part or subassembly within an assembly.

Configuration Manager The Configuration Manager on the left side of the SolidWorks window is a means to create, select, and view the configurations of parts and assemblies.

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.

document A SolidWorks document is a file containing a part, assembly, or drawing.

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

double-click Click two times with the left mouse button.

drawing sheet A drawing sheet is a page in a drawing document.

eDrawings file Compact representation of a part, assembly, or drawing. eDrawings files are compact enough to email and can be created for a number of CAD file types including SolidWorks.

face A face is 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.

feature A feature is 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.

FeatureManager design tree The FeatureManager design tree on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing.

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

line A line is 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.

mate A mate is a geometric relationship, such as coincident, perpendicular, tangent, and so on, between parts in an assembly.

mategroup A mategroup is a collection of mates that are solved together. The order in which the mates appear within the mategroup does not matter.

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

named view A named view is a specific view of a part or assembly (isometric, top, and so on) or a user-defined name for a specific view. Named views from the view orientation list can be inserted into drawings.

open profile An open profile (or open contour) is a sketch or sketch entity with endpoints exposed. For example, a U-shaped profile is open.

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

part A part is 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.

planar An entity is planar if it can lie on one plane. For example, a circle is planar, but a helix is not.

plane Planes are 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 point is 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).

profile A profile is 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).

Property Manager The PropertyManager is on the left side of the SolidWorks window for dynamic editing of sketch entities and most features.

rebuild The rebuild tool 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.

relation A relation is 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.

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

section A section is another term for profile in sweeps.

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.

sheet format A sheet format typically 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 Shell is a feature tool 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.

sketch A 2D sketch is 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 nonplanar and can be used to guide a sweep or loft, for example.

SmartMates A SmartMate is an assembly mating relation that is created automatically.

subassembly A subassembly is an assembly document that is part of a larger assembly. For example, the steering mechanism of a car is a sub-assembly of the car.

surface A surface is 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.

underdefined A sketch is underdefined when there are not enough dimensions and relations to prevent entities from moving or changing size.