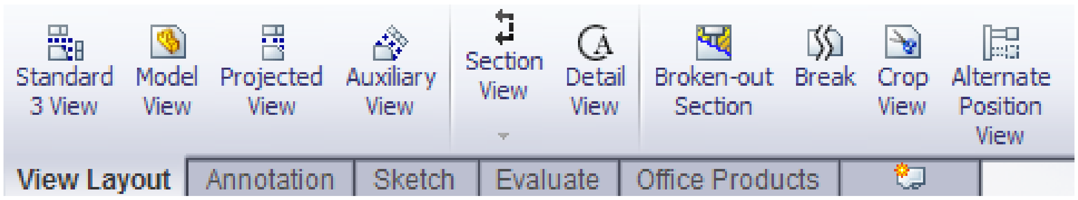
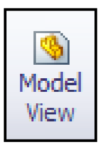
Drawing Tools

View Layout Toolbar At A Glance

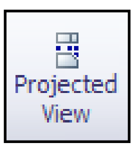


View Layout Tools

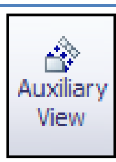
Standard 3 View:
Inserts and creates three default orthographic views that relate to one another. One view will act as a parent view to the other two. When the parent view moves the other two move and when the parent scale is changed the other two views scale will change with it.



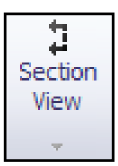
Model View:
Insert orthographic views into a drawing manually by selecting one, or multiple views. The user can specify if they want standard views, trimetric, diametric, or a current view of the model (only if the model is open). See model view property manager for more information.



Projected View:
Views created from orthogonal views and projected onto non defaulted plane. A dotted line will indicate where the view is being projected from.



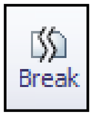
Auxiliary View:
A view which is unfolded normal to a reference *edge* from an existing orthographic view or projection.



Section View:
Create a view of a rotated section drawing view containing the details of an imaginary cut through a part. SolidWorks allows the user to change the hatch style, the display style, and flip the cut diraction. Also found under this tool is aligned section view which performs similarly to section view except that it has two cut lines rater than one.

Detail View:
Create an enlarged portion of a view to show more complex parts and limit dimension clutter. Detail views can be applied to an orthographic, section, cropped, exploded assembly, or another detail view.

Broken-out Section:
A closed profile, typically a spline, around a desired view at a specified depth that cuts away a part of the current view to expose inner details of the part. This does not create a separate view but remains apart of the selected view.



Break:
Interrupt a view in a drawing by cutting away a portion of an existing view. Allows the user to utilize a larger scale when working with a larger component. The referenc e and model dimensions on a broken view reflect actual component values.

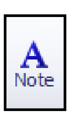
Crop View:
The user specifies a closed profile initially (either draws a spline, circle square, ect.) then selects crop view and the only portion of the view that will remain is the portion enclosed in the closed profile. To remove a crop view **right click** on the view and then select **crop view** and select **remove crop**.

Alternate Position View:
Superimpose one drawing view directly ontop of another. The addition on the other view is displayed with phanton lines. Typically used to show the range of motion within an assembly.

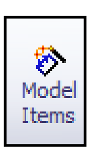
Annotation Toolbar At A Glance

Annotation Tools

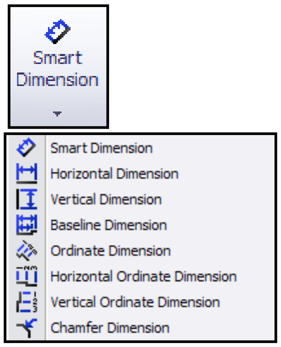
Spell Checker:
Cycles through all annotations and checks for spelling mistakes.



Note:
A free floating or fixed comment attatched to a veiw, drawing sheet, or drawing package. A note can contain simple text, sybols, parametric text, and hyperlinks.



Model Items:
Insert dimensions, annotations, and reference geometry from a part or assembly document into a drawing. In the property manager choose whether to add annotations to the entire model, a selected feature, a selected component, or only an assembly.



Smart Dimension:

Add key dimensions to a drawing document by specifying the type of dimension to add. The user can specify the type of dimension needed by changing the dimensioning tool. Smart dimension determines the type of dimension needed based on the user's selection. However, others, such as horizontal, vertical, baseline, and chamfer can be selected when the user desires to choose the type of dimension and then make selections on the part or assembly. Often, choosing a type before selecting can be more accurate on a difficult part drawing.

Baseline dimensioning references the dimension from a single edge or vertex. Ordinate dimensioning reference a chosen zero point (edge, point, vertices, or arc) and the selected starting point is denoted with a zero. A chamfer dimension accurately dimensions a chamfer.

Note: Most drawings employ baseline dimensioning since it makes fabrication easier.

Balloo/AutoBalloon:

The Balloon tool labels or calls out components within an assembly by *manually* adding a numbered balloon. AutoBalloon perfoms the exact way that the balloon tool does except that it will label all components in the sheet automatically rather than manually. The user does not need to add a bill of materials to insert balloons. The balloons are numbered in accordance to the default value SolidWorks assigns. If a bill of materials exists SolidWorks uses those values.

Revision Symbol:

Inserts a revision table into the drawing document to track document revisions and includes the revision symbols.

Surface Finish:

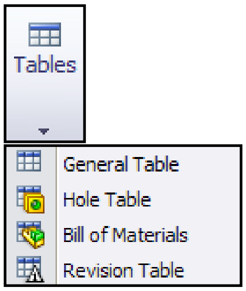
Allows the user to specify the surface texture of a components face.

Weld Symbol:

Specify the weld parameters on a welded structure.

Hole Callout:

This tool uses the information from hole wizard to dimension a hole in a drawing model. If the hole is changed in the part or assembly document, the drawing will update automatically. The number of instances will be included if the pattern was created using the hole wizard feature.



<u>Tables:</u>

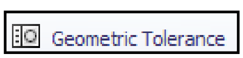
Note: all tables functions similarly to an excel spreadsheet.

<u>General Table</u>: Create a unique table to suit the users specific needs or purpose.

<u>Hole table</u>: creates a table that measures the positions of holes from a selected origin datum. SolidWorks labels each hole with a label and corresponds that with a row in the table.

<u>Bill of Materials</u>: creates a table that identifies components in an assembly. The table can be anchored moved, and edited. Typical headers in a BOM are item number (identified by ballons), part number (identified by user), description, and quantity. Other columns can be added to include such information as price, manufacturer, or stock size. To add or edit the table **left click** on the table, then **right** click on a column or row. All these properties can be linked in a part document so that the user does not need to manually type out this information. If linked the information will update in the drawing document if the part properties is editted and saved.

<u>Revision Table</u>: creates a table to aid in tracking document revisions and adds revision symbols. Helpful in large project management.

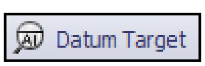


<u>Geometric Tolerance:</u>

Place geometric tolerance symbols with or without leaders in a drawing to specify machining standards for the part.

<u>Datum Feature:</u>

Insert an exact reference point, line or surface to specify where measurements are taken in the drawing and or in machining.

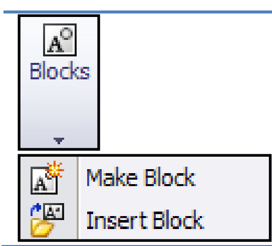


<u>Datum Target:</u>

Insert a specific refernce point, line, or surface to establish a datumn.

<u>Area Hatch/Fill:</u>

Apply a crosshatch or solid fill to a face on a component, a closed profile sketch, or a region bounded by model edges.



<u>Blocks:</u>

Make, save, edit, and insert a user defined annotation into a drawing. The user can add blocks such as text, sketch entities, balloons, imported entities, and an area hatch, and then store them in a specified file folder. Extremely helpful when adding duplicate entities in several sheets or drawing views.

	*Center Mark:* Place a centermark on circles or arcs on views within a drawing document. This tool can be helpful when dimensioning a part.
	*Centerline:* Insert manually or automatically centerlines, where appropriate, in drawing views.