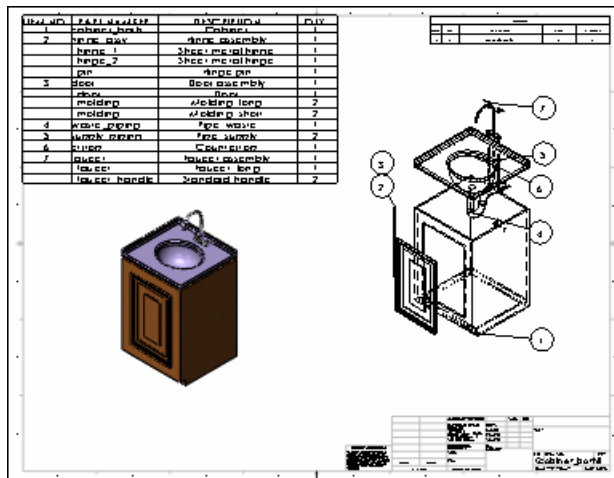# Lecture-09
Drawings

Drawings are 2D documents that convey a design to manufacturing.
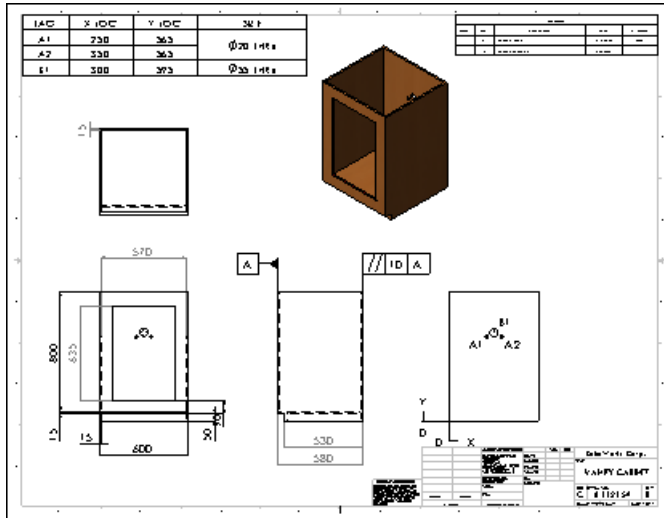

This chapter includes the following topics:

- **Drawing Documents**
- **Vanity Cabinet Drawing Sheet**
- **Faucet Assembly Drawing Sheet**
- **Vanity Assembly Drawing Sheet**

Drawing Documents

You create drawings from drawing templates. Within a drawing document are drawing sheets that contain drawing views. The drawing sheets have underlying formats.

Drawing templates and sheet formats are two distinct entities. The software comes with one drawing template and a set of sheet formats (in English and metric). When you begin a new drawing using the default drawing template, the size of the drawing is undefined. The software prompts you to select a sheet format. The sheet format controls:

- Size of the drawing sheet
- Drawing borders
- Title block
- Sheet scale

Drawing Templates

To start the drawing document, you open a drawing template. Drawing templates contain basic document information. You choose from templates supplied with the SOLIDWORKS software containing default drawing sheets or templates you customize. You can create custom drawing templates with any of the following characteristics:

- Drawing sheet size (for example, A, B, and C)
- Drawing standard (for example, ISO and ANSI)
- Units (for example, millimeters and inches)
- Company name and logo, author's name, and other information

Drawing Sheets

For the vanity drawings, a drawing template with a C-size drawing sheet in landscape orientation is appropriate. The standard drawing sheet formats contain borders and title blocks for the C-size landscape format:

The drawing document for the vanity contains three sheets. You can have any number of drawing sheets in a drawing document, like a set of drawings. You can add sheets at any time using any format, regardless of the format of other sheets in the document. Tabs with sheet names appear at the bottom of the graphics area.

Sheet Formats

The bottom right corner of the default sheet format contains a title block.

After changing the sheet scale, adding two sheets, and editing and adding notes, the title block appears as shown. The scale and page numbers are linked to system variables and updated automatically.

The sheet format underlies the drawing sheet and is separate from the drawing sheet. You edit the sheet format separately from the drawing sheet. Sheet formats can contain items such as lines, note text, bitmaps, and the bill of materials anchor point. You can link the notes to system properties and custom properties.

Drawing Views

Drawing views are placed on drawing sheets and contain the images of the models, plus dimensions and annotations.

Drawings begin with standard views. From those views, you can derive other types of views, such as projected, section, and detail.
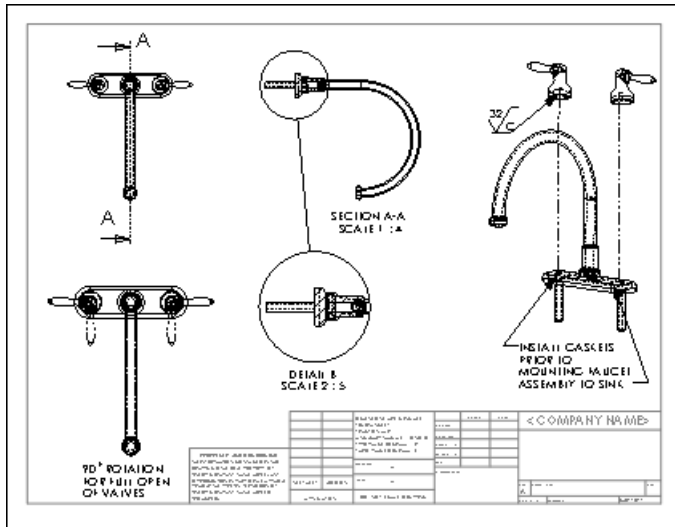
🎓 For a lesson on drawing documents, inserting standard views, and adding dimensions to drawings, see the *Lesson 3 - Drawings* tutorial.

💡 For more information on document templates, drawing sheets, and drawing views, see the Help.

Vanity Cabinet Drawing Sheet

The vanity cabinet sheet contains standard 3 views and named views that are generated from the part. The views are displayed in different modes and contain dimensions and annotations.

Standard Views

Drawings generally start with a standard 3 view or some type of named view, for example, front, top, isometric, or exploded. You can insert these views from an open part or assembly document, from a file, or from other views in the same drawing document.
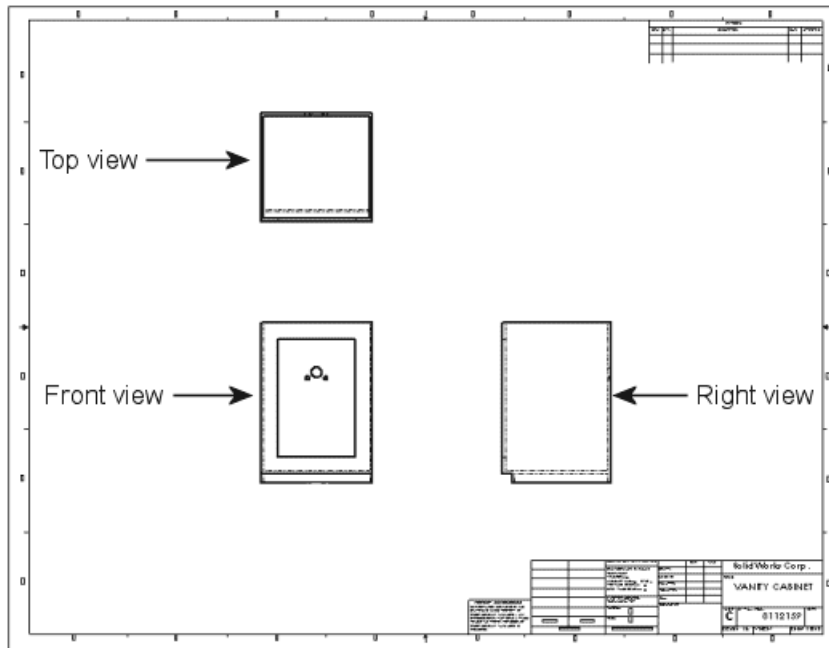
Standard 3 Views

Standard 3 views, as the name implies, comprise three views: front, top, and right (third angle projection) or front, top, and left (first angle projection). In third angle projection, the default front view is displayed at the lower left. In first angle projection, the front view is displayed at the upper left. First angle projection is typically used in Europe. Third

angle projection is typically used in the United States. The example in this section uses third angle projection.

> 💡 For more information on first and third angle projection, see *First Angle and Third Angle Projection* in the Help.

The standard 3 views of the vanity cabinet are the first views that are placed on this sheet.
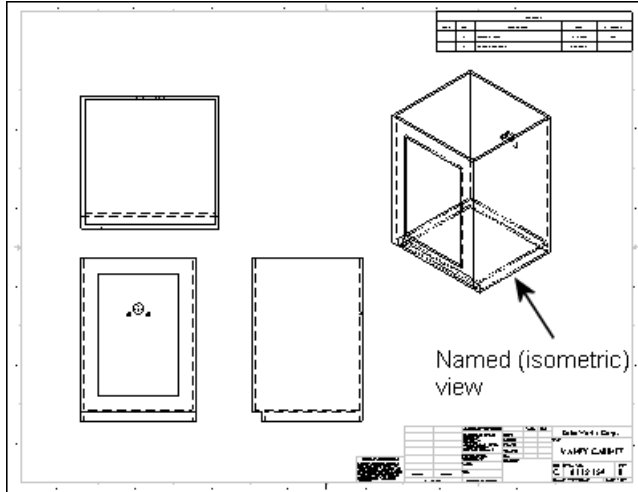

Named Views

Views are named in the model documents. Named views include:

- Standard orientations, for example, front, top, and isometric
- Current model view
- Custom named views

Next, you add an isometric view of the cabinet (a named view) to the drawing sheet.
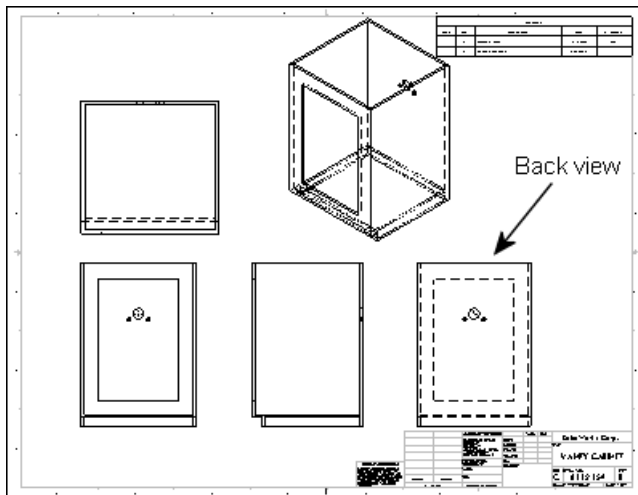
You select the view orientation as you bring the view into the drawing.

Projected Views

Projected views are orthographic projections of existing views.

The vanity has details in the back that are important to show. To create a back view, project the right view and place it to the right.

View Display and Alignment

You can choose various display modes for drawing views. On the Vanity Cabinet sheet, the back view is displayed with **Hidden Lines Removed**. The standard 3 views are displayed with **Hidden Lines Visible**. (Hidden lines appear gray on screen, but as dashed lines when printed.) The isometric view is displayed with **Shaded With Edges**.
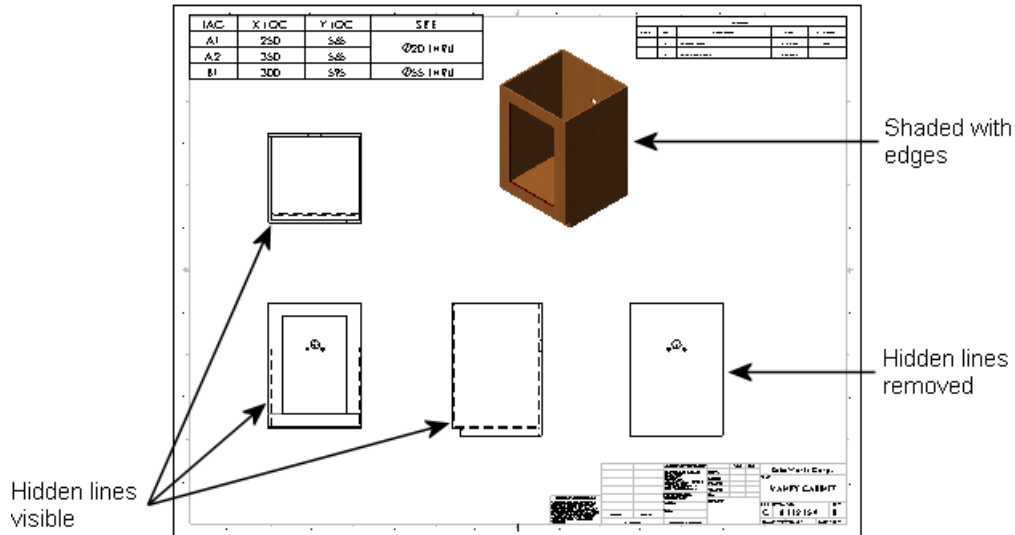
Some views are aligned automatically, but you can break the alignments. Standard 3 views are aligned so that if you drag the front view, both the top and right views move with it. The right view moves independently in the horizontal direction, but not vertically. The top view moves independently in the vertical direction, but not horizontally.

Section views, projected views, and auxiliary views automatically align in the direction of the view arrows. Detail views are not aligned by default.

You can align views that are not automatically aligned. For example, the back view of the cabinet is aligned horizontally with the right view, which is aligned with the front view by default.

> 💡 For more information on displaying, hiding, and aligning views, see *Drawing View Alignment and Display* in the Help.

Dimensions

Dimensions in a SOLIDWORKS drawing are associated with the model. Changes in the model are reflected in the drawing, and vice versa.

Typically, you create dimensions as you create each feature in a part, then insert those dimensions into the drawing views. Changing a dimension in the model updates the drawing, and changing a model dimension in a drawing changes the model.

You can also add dimensions in the drawing document, but these are *reference* dimensions, and are driven; you cannot edit the value of reference dimensions to change the model. The values of reference dimensions change when the model dimensions change.

You can set the units (for example, millimeters or inches) and the drawing standard (for example, ISO or ANSI) in the detailing options. The vanity is in millimeters in the ISO standard.

> 💡 For more information about dimensions in drawings, see *Dimensions Overview* in the Help.

Insert Model Items

You use the **Insert Model Items** tool as a convenient way to insert existing model dimensions into the cabinet drawing. Insert items for a selected feature, an assembly component, a drawing view, or all views.

When inserted into all views (as in the example), dimensions and annotations appear in the most appropriate view. Features that appear in partial views, such as as detail or section views, are dimensioned in those views first.

Once you insert the dimensions, you can manipulate them. For example, you can drag them into position, drag them to other views, hide them, or edit properties.

If the model contains annotations, you can also insert the annotations into drawings by the same procedure.
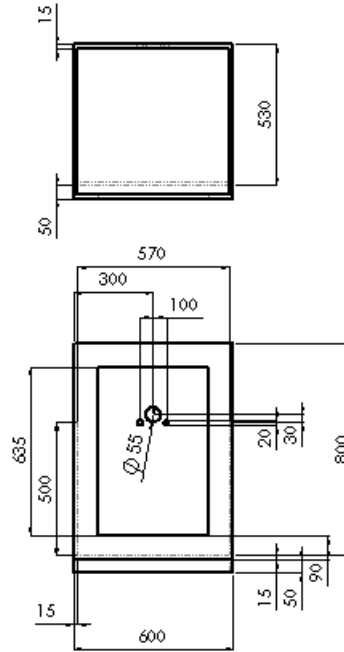
Reference Dimensions

The back view on the Vanity Cabinet sheet is included to show the dimensions of the holes in the cabinet for the supply and waste pipes.

Reference dimensions help you to locate the holes. You can choose whether to enclose reference dimensions in parentheses automatically.
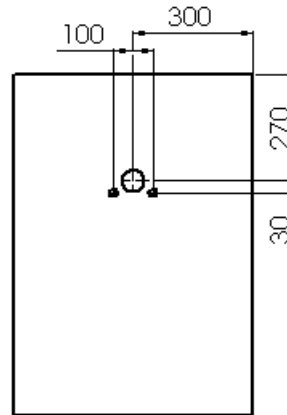
Other types of reference dimensions include baseline dimensions and ordinate dimensions. For example, you might add ordinate dimensions to the front view of the cabinet as shown. You can dimension to edges, vertices, and arcs. The dimensions jog automatically to avoid overlapping. You can display ordinate dimensions without the chain (the arrows between the dimension extension lines).
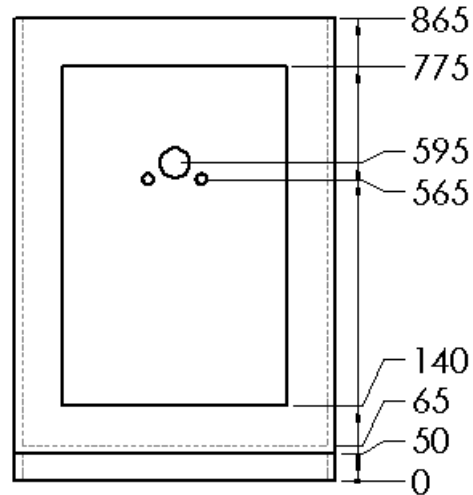
Hole Callouts

You can specify hole callouts when creating holes in models with the Hole Wizard. The Hole Wizard creates and positions holes you define for fasteners such as counterbore and countersunk screws and tap holes. Hole Wizard design data, such as diameter, depth, and counterbore, becomes part of the hole callout automatically.

Hole callouts help you to specify the size and depth of the holes in the cabinet. Hole callouts are annotations that are also dimensions. These hole callouts are in the back view.
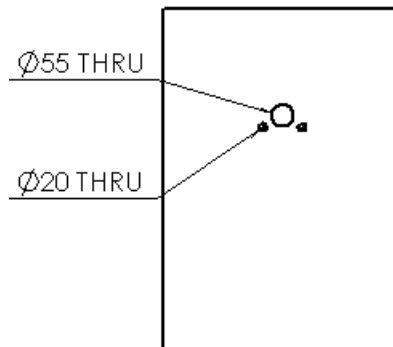
Annotations

In addition to dimensions, you can add other types of annotations to your models and drawings to convey manufacturing information:

- Notes
- Geometric tolerance symbols
- Datum feature symbol
- Center marks
- Surface finish symbols
- Datum target symbols
- Weld symbols
- Balloons and stacked balloons
- Blocks

- Multi-jog leaders
- Area hatches
- Dowel pin symbols

Most annotations can be added in part and assembly documents and inserted automatically into drawings in the same way that dimensions are inserted into drawings. Some annotations (center marks, multi-jog leaders, hole callouts, area hatch, and dowel pin symbols) are available in drawings only.

> 💡 For more information on annotations, see *Annotations Overview* in the Help.

Geometric Tolerance and Datum Feature Symbols

Geometric tolerance symbols display various manufacturing specifications, often in conjunction with datum feature symbols as shown in the example. You can insert these symbols in sketches and in part, assembly, and drawing documents.

In the right view of the cabinet, the back edge is specified with a geometric tolerance symbol as parallel to the front edge within 10mm.
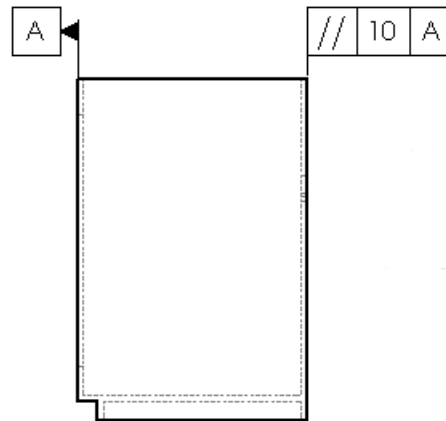
Center Marks

Center marks are annotations that mark circle or arc centers and describe the geometry size on the drawing.

In this example, center marks are added to the holes in the back view of the cabinet. You can place center marks on circles or arcs. Center marks can be used as reference points for dimensioning.

You can rotate center marks, specify their sizes, and choose whether or not to display extended axis lines.
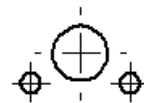
> 🎓 For a lesson on adding derived views, annotations, and exploded views to drawings, see the *Advanced Drawings* tutorial.

Here is the completed vanity cabinet drawing sheet.
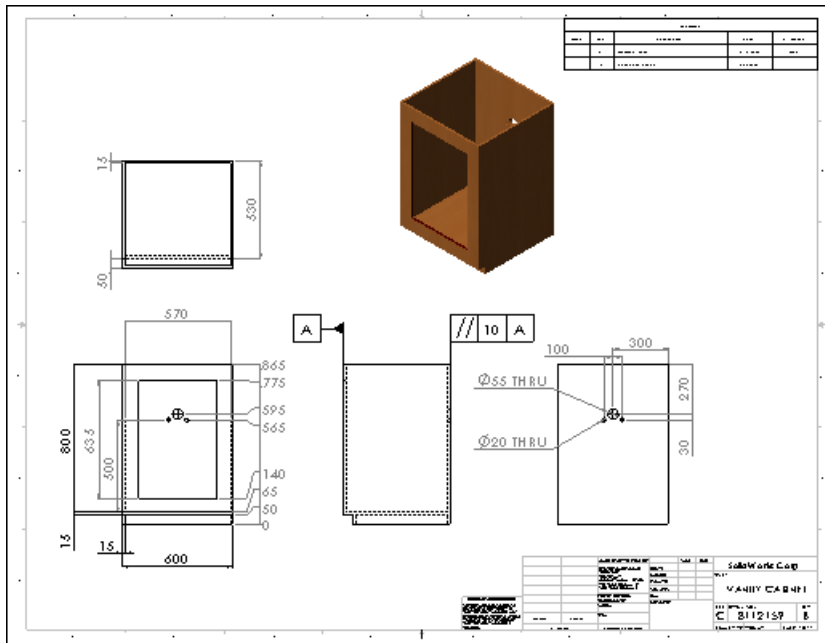
Faucet Assembly Drawing Sheet

The faucet assembly drawing sheet displays several derived views and annotations.

Explode Lines

The faucet assembly is shown in an isometric named view in its exploded configuration. Explode lines show the relationships between assembly components.

You add the explode lines to the assembly document in an explode line sketch. You can also jog the lines as needed. The lines are displayed in phantom line font.
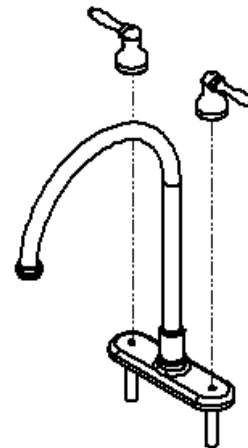

Derived Views

Derived views are created from the standard views. With standard 3 views or a named view in a drawing, you can create other views without going back to the model.

Section Views

You can create a section view in a drawing by cutting the parent view with a section line.

A section view of the faucet in the faucet assembly drawing shows the faucet pipe walls and connections. In this example, you insert a top view of the faucet assembly as the basis for the section view.


A-A (1 : 2)
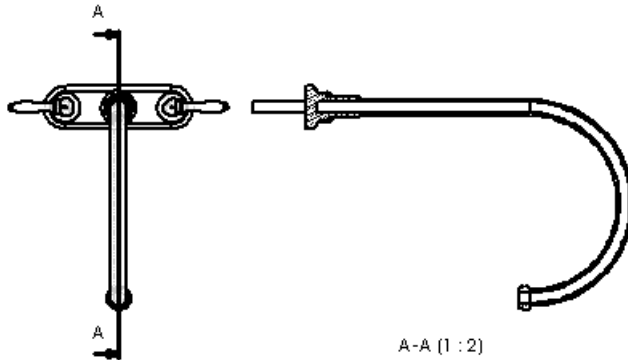
> There are other types of section views, such as aligned section views and broken-out section views.

The sectioned components automatically display crosshatching. You can edit the properties of the crosshatching (pattern, scale, and angle).

Detail Views

Detail views show a portion of an orthographic, 3D, or section view, usually at an enlarged scale.

The connection of the faucet is shown in a detail view. The parent view is the section view.

A-A (1 : 2)

B (1 : 1)
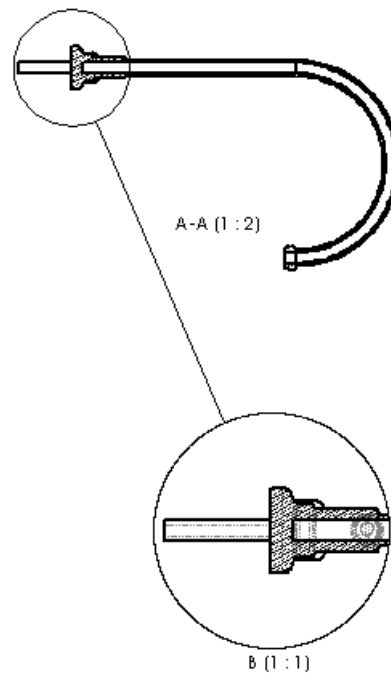
Additional Drawing Views

Alternate position views display overlays of two or more positions on the same view, often to show range of motion of an assembly component. The overlay views are displayed in the drawing in phantom lines.

The faucet handles are shown on the faucet assembly sheet in an alternate position view to display the range of motion of the handles.
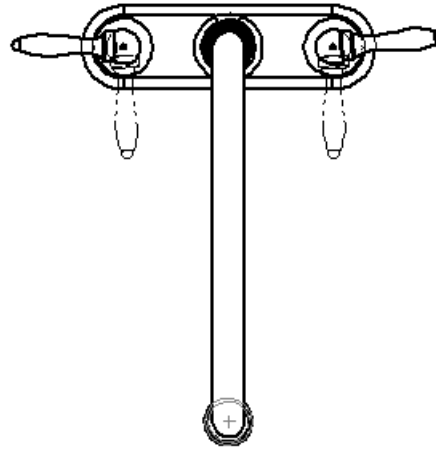
Other drawing views include:

|	
Auxiliary view	A projection normal to a reference edge
Crop view	Everything outside a sketched profile removed
Broken-out section	Material inside a profile removed to expose inner details
Broken view	Portion of a long part with a uniform cross-section removed

💡 For more information about drawing views, see *Derived Drawing Views* in the Help.
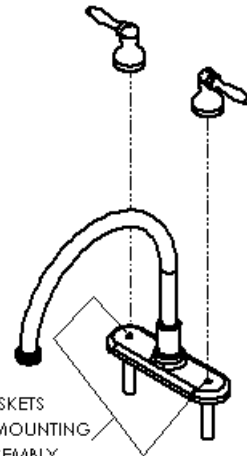
Notes and Other Annotations

Notes and Multi-Jog Leaders

The alternate position view has a note with a degree symbol. In the exploded faucet view, the note uses a multi-jog leader.

A note can be free floating, as in the first example, or pointing to an item (face, edge, or vertex) in the document, as in the second example.
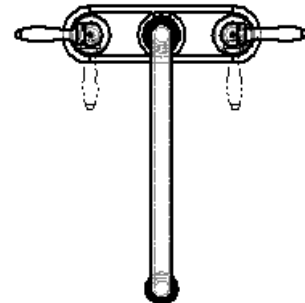
90° ROTATION
FOR FULL OPEN
OF VALVES

INSTALL GASKETS
PRIOR TO MOUNTING
FAUCET ASSEMBLY

Surface Finish Symbols

You can add surface finish symbols to part, assembly, or drawing documents. You can insert multiple symbols and multiple copies of a symbol.

Some of the characteristics that you can specify for a surface finish symbol include type of symbol, direction of lay, roughness, production method, material removal, and rotation.

The surface finish symbol attached to the faucet handle specifies a circular finish and the maximum roughness of the surface.

32

C
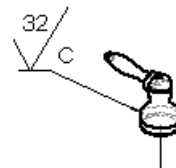
Here is the completed faucet assembly drawing sheet.
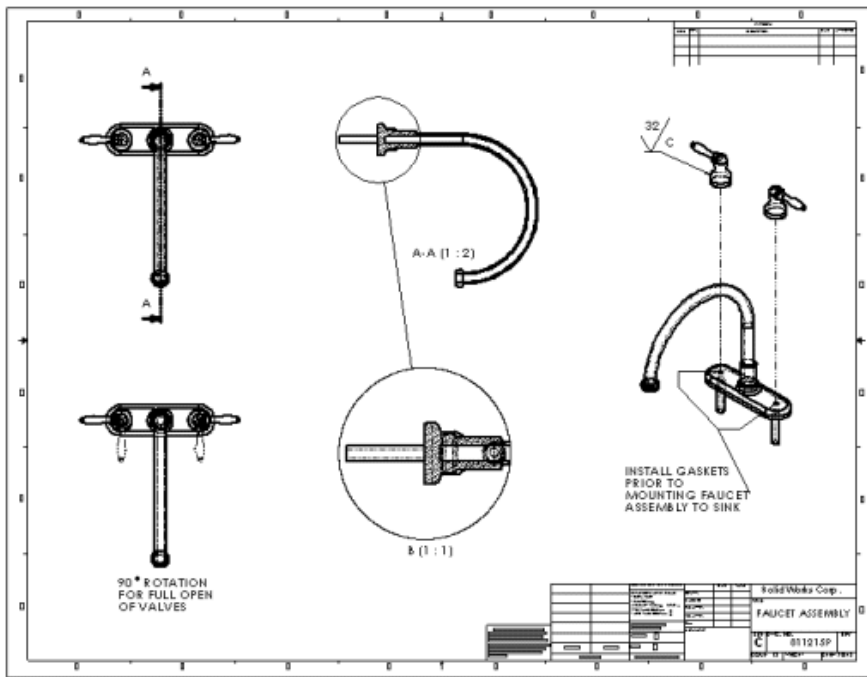
Vanity Assembly Drawing Sheet

This drawing sheet contains an exploded view, a bill of materials, and balloons.

Exploded Views

Exploded views are versions of named views defined in configurations in the assembly document. This drawing contains an exploded view of the vanity assembly.

The drawing also contains an isometric named view of the complete assembly, unexploded, at the lower left.
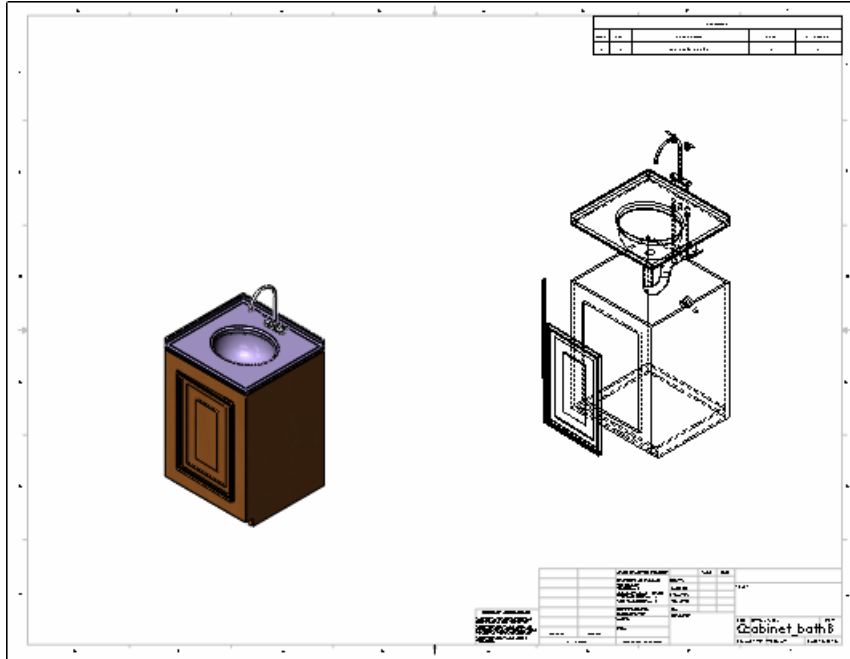
Bill of Materials

A bill of materials (BOM) is a table that lists the components of an assembly along with information needed in the manufacturing process. If the assembly or its components change, the BOM updates to reflect the changes.

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	cabinet_bath	Cabinet	1
2	hinge_assy	Hinge assembly	1
	hinge_1	Sheet metal hinge	1
	hinge_2	Sheet metal hinge	1
	pin	Hinge pin	1
3	door	Door assembly	1
	door	Door	1
	molding	Molding, long	2
	molding	Molding, short	2
4	waste_piping	Pipe, waste	1
5	supply_piping	Pipe, supply	2
6	ctrtop	Countertop	1
7	faucet	Faucet assembly	1
	faucet	Faucet, long	1
	faucet_handle	Standard handle	2

When you insert a BOM, you have a choice of BOM templates with various columns for data such as item number, quantity, part number, description, material, stock size, vendor number, and weight. You can also edit and save a custom BOM template.

The SOLIDWORKS software populates the item number, quantity, and part number columns automatically. The item numbers reflect the sequence in which the model was assembled.

You set the anchor point for the BOM in the drawing sheet format.

For more information, see *Bill of Materials - Overview* in the Help.

Balloons and Stacked Balloons

You can insert balloons in assembly and drawing documents. You can set the style, size, and type of information for balloons. In the example, the balloons display the item number corresponding to the BOM in a circle.

The exploded vanity assembly includes balloons and stacked balloons for each component. The item numbers appear in the balloons automatically.

Stacked balloons have one leader for a series of balloons. You can stack the balloons vertically or horizontally.

> For a lesson on bills of materials and balloons, see the *Advanced Drawings* tutorial.
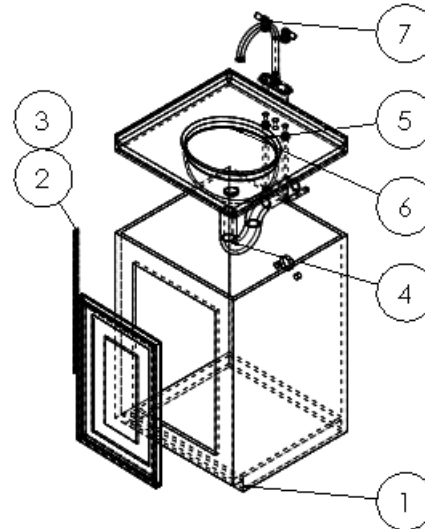
Here is the completed vanity assembly drawing sheet.