

Applying Formats with Styles

This Tech Tip looks at styles in Solid Edge so that you can set standards within your organisation that everyone works to and obtain a uniform image.

If you want items of the same type to look alike, you should use styles, a collection of formats that can be applied to an item. You can apply styles to the following items:

- Drawing views
- Dimensions
- Annotations
- Fills
- Connectors
- Parts list and tables
- Views
- Faces
- Lines
- View annotations

Using styles ensures consistency. If you place one or more items, the formats in the style are applied directly to the items. You do not have to spend time formatting them by setting several options on a command bar or dialog box.

Solid Edge provides styles that conform to an engineering standard, such as ISO or ANSI.

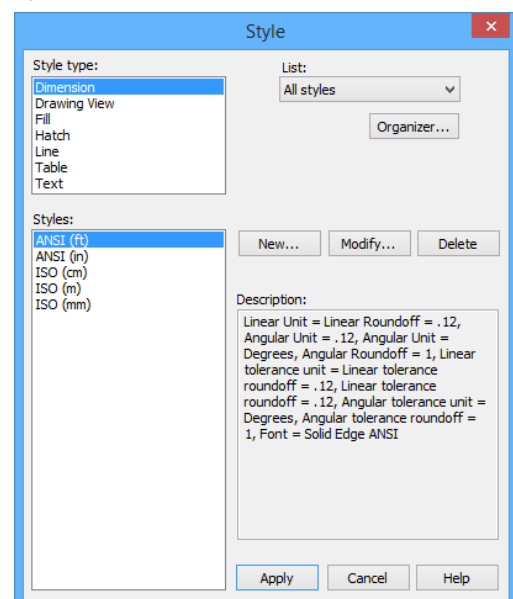
Managing styles

Using the Styles command, you can create several styles so that the items in a drawing appear the way you want. You can use or modify the styles delivered with the software, or you can set up new ones that conform to your unique requirements. One or more styles can be stored in a template so that you can use them in other drawings.

The software divides styles into types:

- Drawing view styles
- Line, fill and hatch styles
- Table styles
- Text styles (You can use text styles to change drawing text size and change PMI text size)
- Face styles
- View styles

A style type contains one or more styles. You can create styles for each style type.



Copying styles

The Organizer option on the Style dialog box allows you to copy a style from one document to another document. For example, you may have created a custom Faces style that you want to use in other documents. If you want to use a custom style you created in all the documents you create, you can copy the style to the templates you use to create new documents.

Applying styles

The style type determines what type of item you can apply a style to. Styles are applied automatically by the software but you can change styles. For example, in the Draft environment, if you click Smart Dimension, the ANSI or the ISO style appears on the command bar. Each dimension that you place receives the ANSI or ISO style. To apply a different style, you can change the style name on the command bar before or after you place an item.

Following are some examples of where and how you can use styles.

- In the Part environment
You can apply view styles to 3D views of a model. You can also apply dimension styles to PMI dimensions.
- In the Assembly environment
You can apply faces styles to faces; 3D view styles are also available.
- In the Material Table
You can create linear and radial hatch patterns for display in previews of materials in the Material Table dialog box. For example, the Wood (radial) pattern style uses concentric circles to mimic the circular growth-ring pattern of a crosscut tree trunk. The Wood (radial) pattern is available as a hatch style and as a fill style in the Part, Sheet Metal, Assembly and Draft environments.
- In the Draft environment
A text style can be applied to text within a text box or to text within a table. A fill style can be applied to a fill. A line style can be applied to centerlines and various lines displayed in the drawing, such as normal, dotted, hidden, visible, tangent and phantom lines.
- In drawing view
A drawing view style can be applied to drawing views and view annotations (Principal, 2D Model, Section (includes Cutting Plane Line), Auxiliary (includes Viewing Plane Line) and Detail (includes Detail Envelope).
- In dimensions and annotations

A dimension-type style can be applied to any of the following elements:

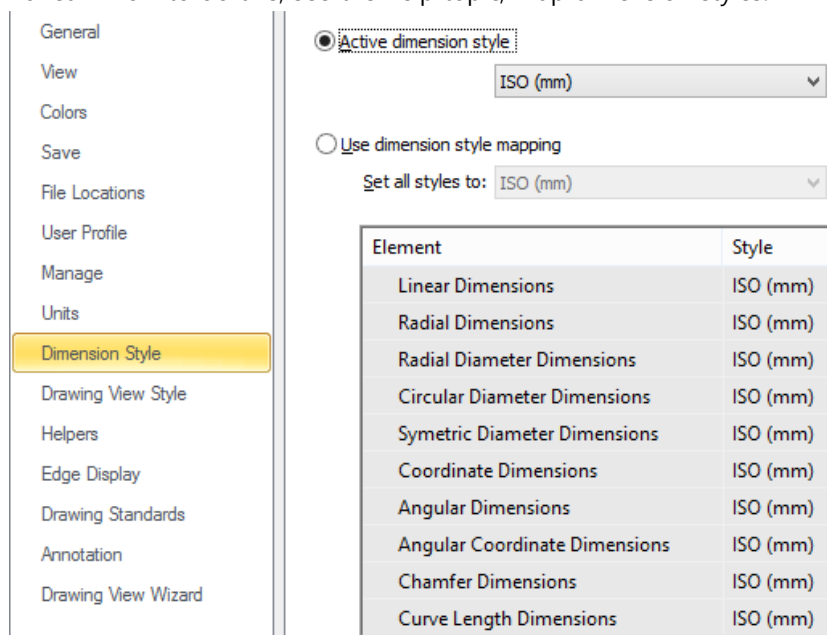
- Dimensions (Linear, Radial, Radial Dimeter, Circular Diameter, Symmetric Diameter, Coordinate, Angular, Angular Coordinate, Chamfer)
- Center Marks, Bolt Hole Circles
- Balloons
- Callouts
- Leaders
- Connectors
- Edge Conditions
- Feature Control Frames
- Datum Frames, Datum Points, Datum Targets
- Annotation Symbols (Surface Texture, Weld, Edge Condition)

Using style mapping to apply styles:

The element-to-style mapping tables in the Solid Edge Options dialog box ensure that the correct styles can be globally applied to all elements in the current document, without having to select the style from a list.

- The Dimension Style tab (Solid Edge Options dialog box) lists all of the Solid Edge element types that use the style named *Dimension*. These include dimensions, annotations and symbols. Here, you can select the style to apply to each element type, whether that style is an engineering standard, such as Solid Edge ANSI or Solid Edge ISO, or a custom style defined within Solid Edge.

To learn how to do this, see the help topic, Map dimension styles.



- The Drawing View Style tab (Solid Edge Options dialog box) lists all of the drawing view types and view annotation types that use the stlye named Drawing View.

To learn how to do this, see Map drawing view styles.

Once defined, you can apply style mapping by selecting this button on the command bar:

- Use Dimension Style Mapping (when placing a dimension or annotation).
- Use Drawing View Style Mapping (when creating a drawing view).

Style selection and mapping also can be set up in template files. If allowed, the designer can turn the Dimension Style Mapping button off so that the mapped style can be overridden in the local document by choosing a different style or by making modifications to style properties on the command bar.

Applying style overrides to more than one item

Selecting more than one item and then applying the same style to them removes any previous formats; the items now all have the formats from the style that you applied. To do this, you must select items that are the in the same style type, such as dimension or text. For example, in the Draft environment, you can not apply a text style to a dimension.

In the Part or Assembly environments, you can select several views and apply the same style with the Styles command on the View tab.