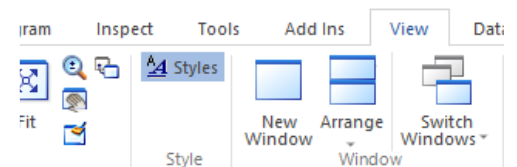# Settings

This Tech Tip looks at how to set up dimensions, lines styles, etc, so that they are how you want them on all new drawings.

One thing I commonly come across is new users asking me how to change a dimension style so that it is permanently set.
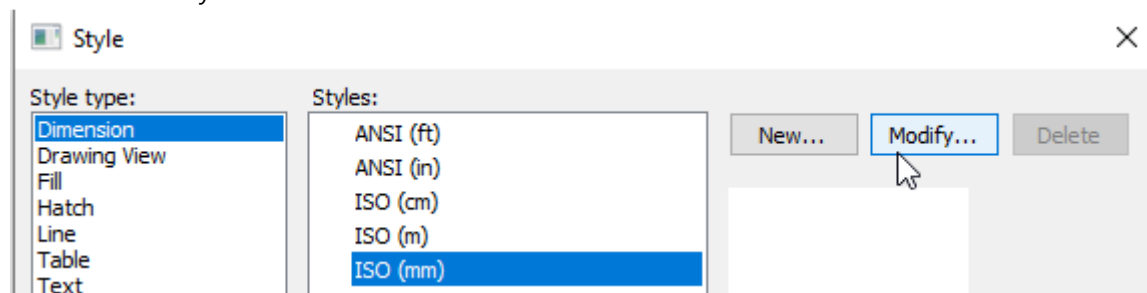
All new documents, whether they are draft, part or assembly files, are copied from the templates, so any setting you want to have permanently set, need to be applied to the template files you use, whether the delivered ones or your own one with a customised drawing border.

Open the draft template that you use when creating new draft documents.

From the View Tab, click on the Styles button.

Select the dimension style you normally use and click modify. If you want to create a new style, click a style similar to what you want and click New.

In this example, I will create a dimension style to display radial dimensions that have the text as horizontal. Ie.

In the Name tab, Give your style a relevant name.