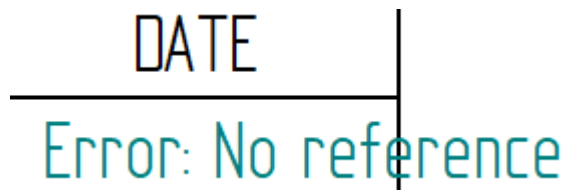


Managing Property Text in the Title Block

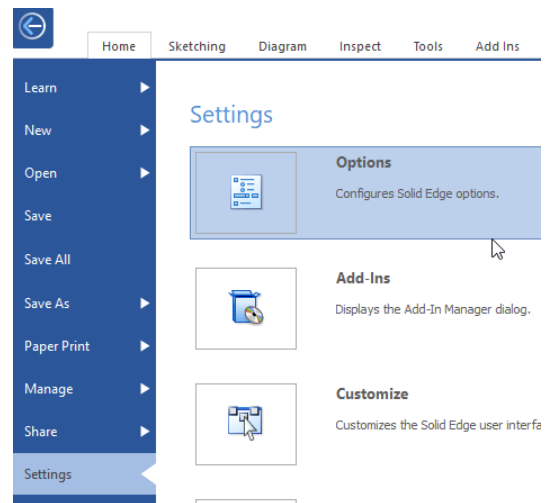
This Tech Tip looks at how to change the settings in a draft file so that if a piece of property text is not present, then the text does not display.



When you set up your drawing border, it is quite common to place text holders that pull property text from the model to display in the title block. This means that when you drag your model into the drawing, those values held in the part or assembly property fields (such as title) are used to automatically populate the drawing title block. This technique falls down when the data hasn't been entered, or is not relevant.

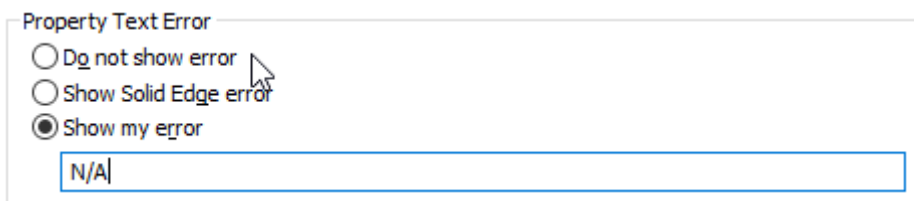
For Example, you may want to display the bend radius of a sheet metal part (%{Bend Radius|R1}), but when you place a part in the file, you end up with an error (as shown above).

To overcome this, you have 2 options – show no error or to substitute your own piece of text in it's place. While in the draft file (or draft template if you want to set it up as a standard), open the Solid Edge Options (found under Settings on the Application button).



In the Options Dialog box, click on the "General" option on the left and scroll down to the bottom of the list of general options.

Here, you can set that you do not want to display any error message or click on the "Show my error" option and enter the text that you would like displayed, for example "N/A".



	NAME	DATE
DRAWN	User	☒

Figure 1: No error

	NAME	DATE
DRAWN	User	N/A

Figure 2: My error