
Autodesk Products Tip & Tricks

Product: Inventor 2010
Topic: Sheet Metal Style Libraries
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Traditionally in Inventor the recommended method for storing all of your Sheet Metal Style information was to store it in the "sheet metal.ipt" template file, so that any sheet metal material styles that you want to use regularly would have to be created and saved in your template before beginning any new sheet metal part. That's great in theory, but a little less so in practice. Often we don't know the entire range of sheet metal sizes and materials that we may need. We may also create a new style in a part and forget to copy that to our template. So to help us out with this and to conform to the standard way of managing styles, Autodesk moved the Sheet Metal Styles into the Style Library. Now, as long as your project is set to use the style library, when you start a new part you have access to the sheet metal style you need. You will also be able to push new styles up to the library to make them available to other files and users if you choose to.

As you see in Figure 1, the Sheet Metal Rule and the Sheet Metal Unfold have been placed into the Style and Standard Editor along with Color, Lighting and Material Styles.

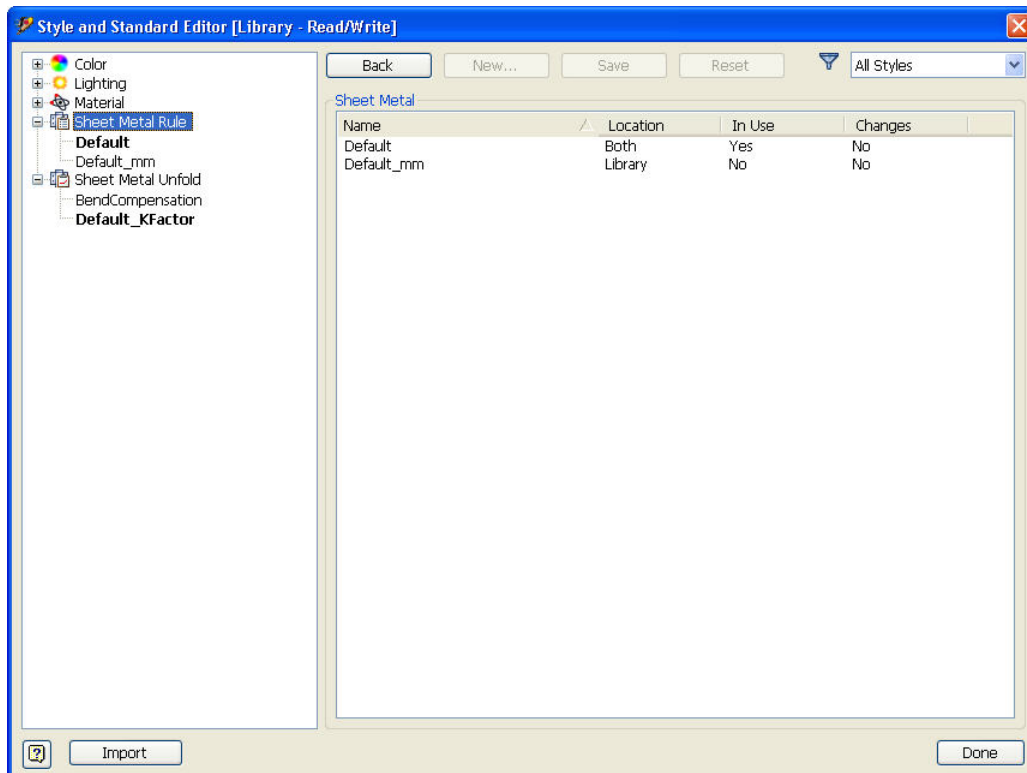


Figure 1

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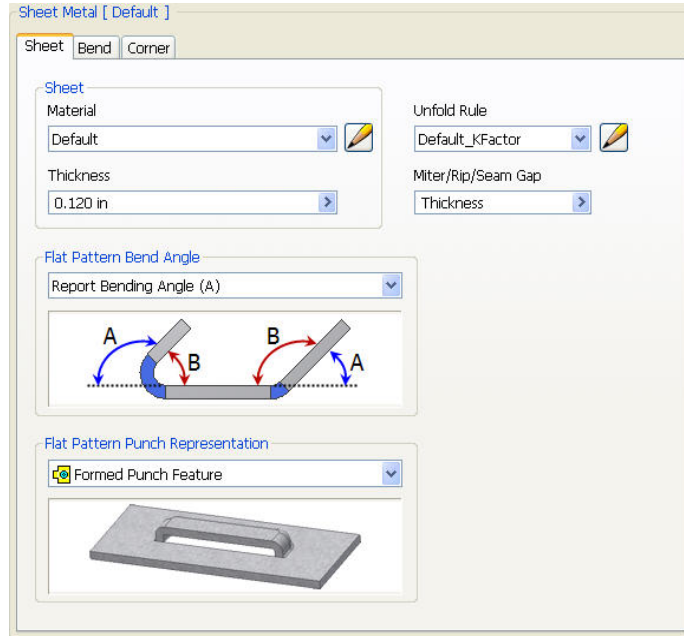


Figure 2

Here in Figure 2 we see an example of a default Sheet Metal Rule and the various options we have for controlling the style. We can set any number of these, maximizing our effort by copying Rules similar to what we want and tweaking the differences. The same can be done for the Sheet Metal Unfold, as seen in Figure 3.

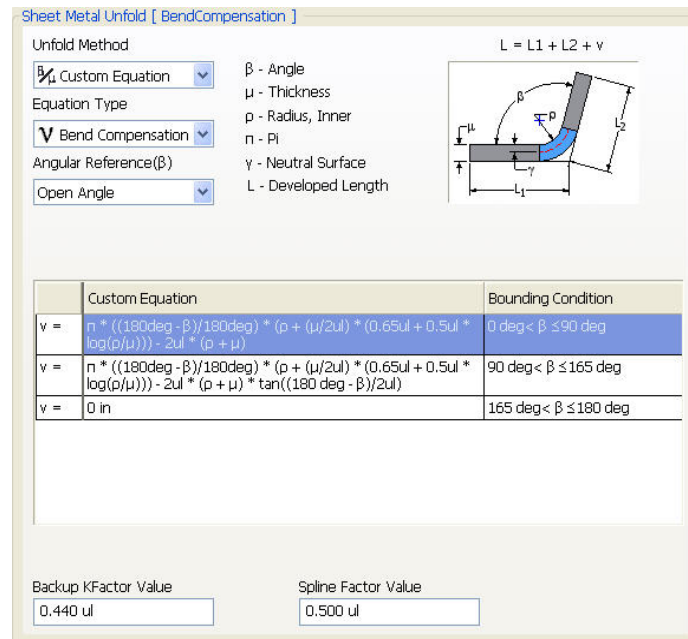


Figure 3

After creating a new sheet metal style via the style Library interface, remember that this style still only exists in the current part file. You will need to right-click on the new style and select “Save to Style Library” in order to make this new style available to all files.