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Sheet Metal Thickness And Material in CUSTOMTOOLS

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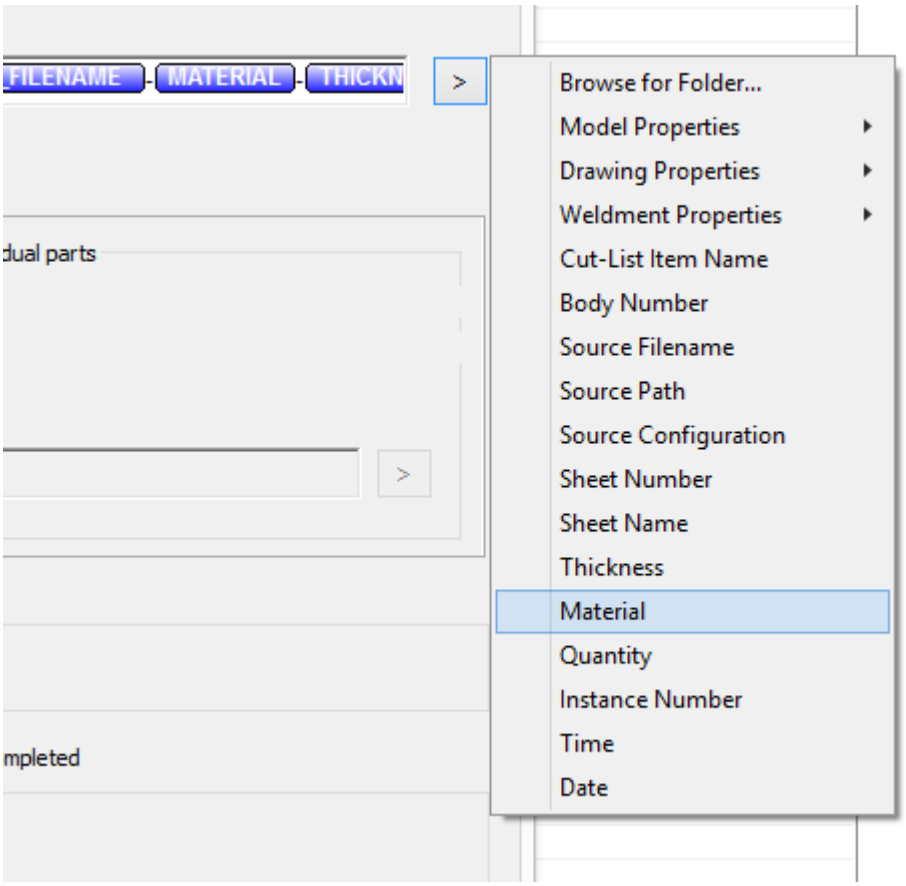
The purpose of this document is to clarify how CUSTOMTOOLS resolves material and thickness in file conversion e.g. when converting models to DXF files.

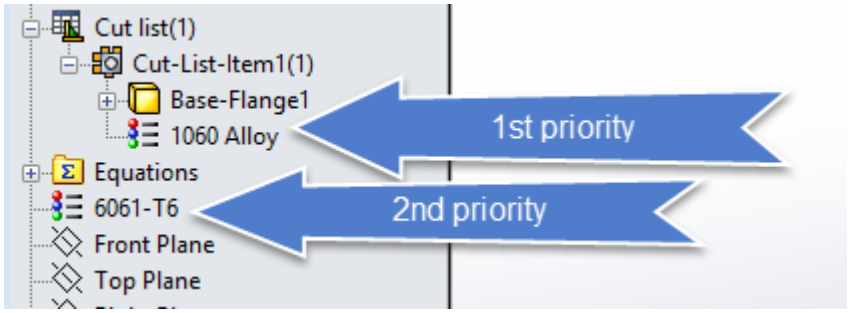
Material

It's safe to use material in output folder even when working with multi-body parts. CUSTOMTOOLS first reads material from body and if that's not defined then material assigned to model is used.



In the settings

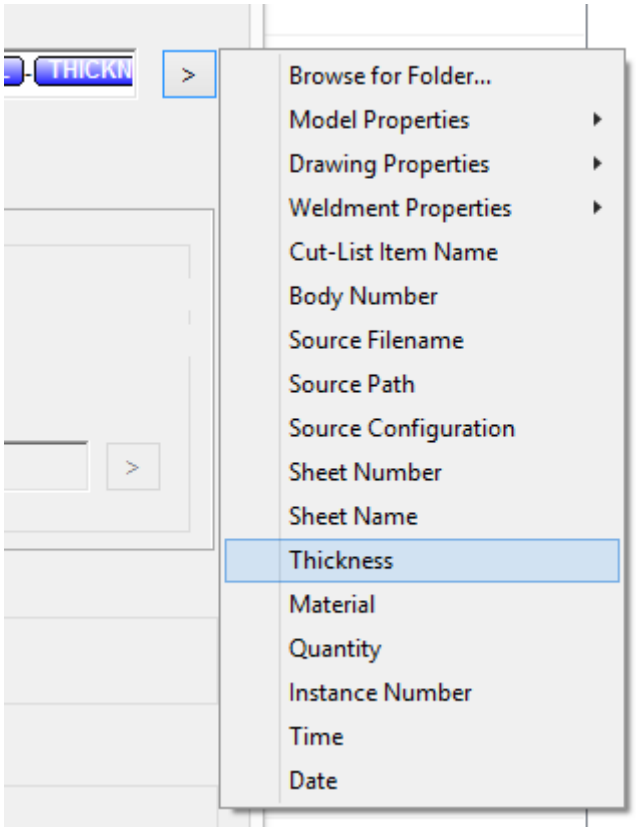
During conversion:



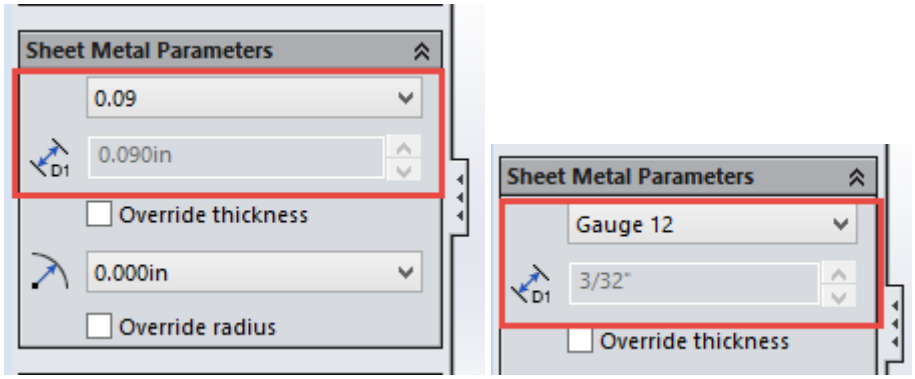
In this case 1060 Alloy would be used as value for material in the output file.

Thickness

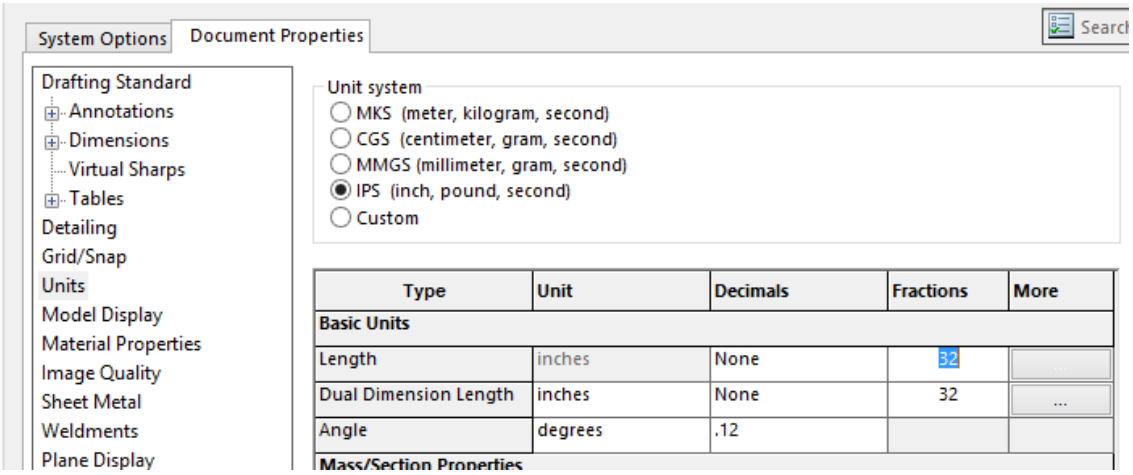
Thickness is read from Sheet Metal Parameters. In the settings thickness is defined as follows



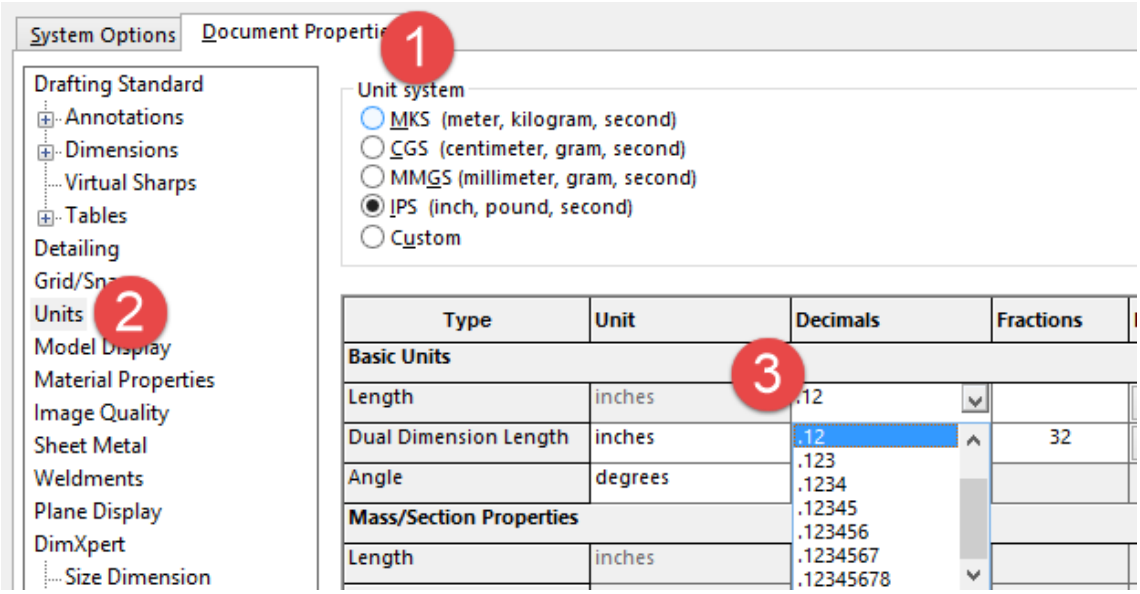
And during conversion it's read from **Sheet Metal Parameters**.



When thickness is used in output filename it's always presented as decimal number as slash in fractional number can't be used in filename. The unit in which value is presented is determined by source model document properties as well number of decimals. Thickness value is rounded (not truncated) per number of decimals. Trailing zeros are never displayed. Decimal separator is always a dot (.) and unit is not included.



When using fractions you won't see decimals before clicking *None* in **Decimals**.



In this case you would have prevision of two decimals. You can change the value and then after clicking **Fractions** cell again **Decimals** become *None* and fractions shows the current value but SOLIDWORKS will save **Decimals** value nevertheless.