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Hi All, I would like to know how to correctly add tolerance to manually added dimensions in Creo Drawing? Take below picture for example, the two red dimensions were added on the Creo drawing by "Show Model Annotations" ,and Creo can show it's tolerance correctly according to the tolerance table I specified.

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default scale is 1:1 and i want to change scale to some value e.g. 1/10 which will apply to all the drawing views available in drawing sheet. this is the purpose. Manually it is possible and easy but need to apply in macro base application. Thanks, Prashant Pandarkar

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1. Download Creo installation file, unzip it to the destination folder and navigate to the installation folder. Locate and right-click setup.exe and select Run as administrator .

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