
Insufficient rights to read or write to file location C:\Users\Raymond\Downloads\Solidworks. This article shows how to add an annotation on a wireframe or mesh object in a drawing. This article also shows you how to hide a text box from the annotation settings. it is owned by Microsoft. Check that all the files have been unzipped and extracted. To ensure a smooth transfer of files, check that the destination path you specify is writeable. SOLIDWORKS APPLICATION NOTE: To ensure a smooth transfer of files, check that the destination path you specify is writeable before you zip and transfer files. Grow pattern There is no design option available to hide texts from the annotation settings. SOLIDWORKS Installation Manager This procedure also applies to Open office. How to Back Up and Restore Files. SOLIDWORKS Installation Manager The white feature path is not shown in the grid. Dimensions that are between 2 mm and 50 mm may be considered hidden when annotated on a 2D drawing. Grow pattern Oct, 11, 2013 When you create a 3D model by combining sketches or components, you may need to modify some of the component dimensions. For example, you may need to modify the height of a solid feature to make it fit the shape of a component. When you modify a component dimension, such as a height or diameter, the Feature Manager opens a dialog box that allows you to specify a modified value. You can either enter a new, modified value or the value that already exists in the drawing. If you choose to enter a new value, you can use the following techniques to enter a value for a feature path of a component: Select the edit icon , or Select the Add button , and enter the modified value in the field that appears. If you modify a component feature path and update a dimension of the attached Solid feature path, the modified dimensions are automatically entered to the next highlighted dimension. If you modify the attribute of a Solid feature path, the attribute is modified for the component. The modified values are automatically entered to the next highlighted attribute. SOLIDWORKS displays in front of the next highlighted attribute. SOLIDWORKS displays in front of the next highlighted dimension. Return to the Component dialog box. Whether or not you use any other dimensioning technique, you must manually enter a value for the dimension to which you applied the attribute

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