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I think (having not tried it) you could do the following (I'm using Creo 2.0): (1) Do the mass properties analysis. (2) Save the analysis as a feature. (3) Create a point with any X, Y, Z you like. (4) Using relations, set the X, Y, Z coordinates of the point to the analysis results.

Centre of Gravity (CG) for parts, assemblies and d ...

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(2) Axes of extruded surfaces, even edge rounds, can be created after the fact by selecting the Create Axis icon and the 'revolved' surface. (3) For extruded features, the axis can be created ahead of time (while sketching) by selecting Axis Point from the Sketch drop-down list, IF the sketch is an Internal Sketch started after the Protrusion is started.

how to show or draw center mark - PTC Community

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Creo Parametric Help Center - PTC

Bing: Creo 2 Design Centre 2 - 2 Creating a Simple Object (Part I) Figure 1 Part at the end of this lesson Figure 2 Creating a new part Start Creo Parametric as usual. If it is already up, close all windows (except the base window) and erase all objects in session using File ' Manage Session ' Erase Current and/or File ' Manage Session ' Erase Not Page 4/9. File Type PDF Creo 2 Design Centre ...

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