

Artikel - CS331165

Dragging the dimension of the drawing view of an inherited model forces to set the dimension as a varied item in Creo Parametric 4.0 M130 and onward

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Betrifft

- Creo Parametric 7.0.0.0 to 7.0.2.0

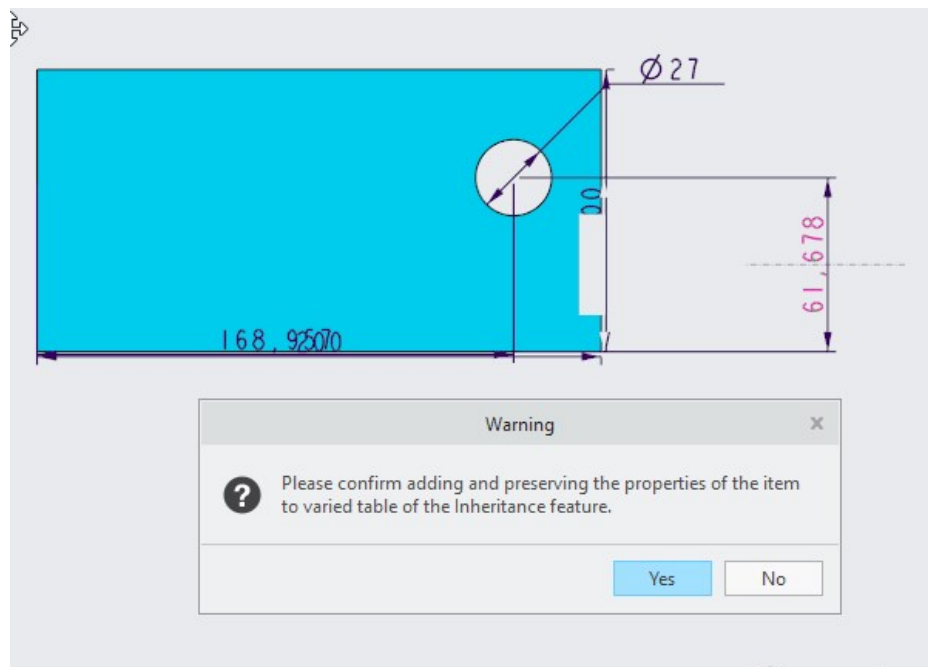
- ~~Creo Parametric 6.0.5.1~~

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- ~~Creo Parametric 4.0 M130 to M140~~

Beschreibung

- Dragging the dimension of the drawing view of an inherited model forces to set the dimension as a varied item
- Getting the warning: *Please confirm adding and preserving the properties of the item to varied table of the Inheritance feature.* when moving the placement of a drawing shown dimension of an inherited model.
- Getting the warning (German): *Hinzufügen und Erhaltung der Eigenschaften des Elements zu der variierten Tabelle im Vererbungs-KE bestätigen.*



- The placement of **Shown Dimensions** in a drawing should:
 - not be added to the varied items table
 - not prompt varied items warning

Ursache

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- Change of behavior introduced intentionally with SPR 8022182 as per fixes submitted starting from:
 - Creo Parametric 4.0 M130
 - Creo Parametric 6.0.5.0
 - Creo Parametric 7.0.0.0
 - The targeted entities were:
 - **Annotation Feature's Annotation Elements**
 - **Stand Alone Annotations**

Auflösung

- Reported to R&D as SPR 10854172
 - Corrected in Creo Parametric 4.0 M150, 7.0.3.0, 8.0.0.0
- Workaround:
 1. Hit **Yes** in the dialog, which should not appear, to complete dragging
 2. Remove the just added varied items manually from varied items table

Rechtliche Hinweise

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