

Ist dies das von Ihnen gesuchte Dokument ?    Nein, keineswegs      Ja, definitiv

Beantwortet dieses Dokument Ihre Frage ?    Nein, keineswegs      Ja, vollständig

**Abschicken**

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Dieses Dokument wurde noch nicht übersetzt.

<b>Title</b>	Suggested Technique For Having a Drawing's Versioned, and File-Based Attributes Show The Same Value As The Model In Pro/INTRALINK.						
<b>Product</b>	Pro/INTRALINK	<b>Module</b>	PRO/WORKSPACE	<b>TPI ID</b>	105936	<b>Created</b>	29-DEC-00
<b>Workstation</b>	All			<b>Reported In Release</b>	2.0	<b>Reported In Datecode</b>	
<b>SPR</b>	None			<b>Resolved In Release</b>		<b>Resolved In Datecode</b>	

Description

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 Frequently when working in Pro/INTRALINK, it is desirable to see drawing attribute information that contains specific information from the model. Yet via Pro/INTRALINK there are no means to copy attribute information directly from one object to another. The following procedure can be used to have a drawing's attributes get populated with its model's values and could possibly be expanded to further uses.

Resolution

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1. Create an attribute in Pro/INTRALINK that matches the name and type of the common parameter in Pro/ENGINEER.
  2. Perform a #tools #sync with CS from a workspace to ensure that the attribute information is fully available to the client.
  3. Assuming the parameter has been defined in the model and the drawing, designate both parameters in Pro/ENGINEER so that the value propagates into the Workspace when the objects are saved.
  4. Then in the drawing, use the following steps:
    - 4.1. In the drawing menu, #advanced #program #edit program #edit line
    - 4.2. In the edit line box enter the following : '<parameter name>:d=<parameter name>:0'
    - 4.3. Continue hitting #Done or #Done/Return until returning back to the top drawing menu.
  5. Upon examing the parameter value as seen via #advanced #parameters #drawing #info it will read for the drawing the same value as the part.
  6. Save the drawing to the workspace, and then whenever the part attribute is changed in Pro/INTRALINK, the drawing is regenerated in Pro/ENGINEER, and then saved to the workspace, the drawing attribute will read the same value as the part attribute.