











## Document - CS31562

### Document Information

Type	Article
Date Created	04-Oct-2011
Date Last Updated	27-Feb-2017
Other Versions Available	 <a href="#">Simplified Chinese</a>

### Document Details

Title	Options "update_drawing all" versus "enable_auto_drawing_update" and "auto_drawing_update" options in Creo Elements/Pro 5.0
Description	<ul style="list-style-type: none"> <li>• Is it possible and does it make sense to replace all the auto_drawing_update options in a config.sup with one line update_drawing all ?</li> <li>• Does it make sense to activate update_drawing all and the above listed options at the same time?</li> <li>• What exactly will the option update_drawing and antique_drawing option do?</li> <li>• General benefit/damage of update_drawing all option.</li> <li>• Does it make sense to activate update_drawing all option permanently?</li> <li>• Why do old drawings in new releases behave and look different from newly created drawings?</li> <li>• A software correction for drawings is not working for existing drawings</li> <li>• Some drawings display incorrectly, but it cannot be reproduced when recreating the drawing</li> <li>• How can I make new drawings behave and look like old drawings</li> <li>• How to apply drawing updates on retrieval</li> <li>• Unable to retrieve drawing and add drawing updates when opened</li> <li>• How to automatically apply a couple of update drawing values and prevent the users from setting additional values for the session</li> </ul>
Applies To	<ul style="list-style-type: none"> <li>• Pro/ENGINEER and Creo Elements/Pro Wildfire 2.0 M140 to Wildfire 5.0</li> <li>• Creo Parametric Creo 1.0 to Creo 3.0</li> </ul>
Cause	<ul style="list-style-type: none"> <li>• When a change to functionality or a bug fix is applied, that potentially modifies the display of a drawing, it will be applied only to newer drawing <ul style="list-style-type: none"> <li>• The idea is that older drawings of customers may have been approved with the old display, maybe with manual corrections added</li> <li>• If the display changes with a new release or datecode, it would invalidate the approval</li> <li>• Hence preserving the existing display of a drawing between releases is an important asset</li> </ul> </li> <li>• However, if the old behavior is undesirable for a customer and recreating the drawing is no option, the drawing needs to be updated</li> </ul>
Resolution	<ul style="list-style-type: none"> <li>• Extreme caution should be used when enabling the DTL and config.pro options below, as they may cause unexpected side-effects <ul style="list-style-type: none"> <li>• The DTL option update_drawing allows to update a drawing by simply entering it into the Detail options</li> <li>• The DTL option antique_drawing allows to reverse the update</li> </ul> </li> <li>• The hidden detail setup options update_drawing and antique_drawing were introduced in Pro/ENGINEER 20.0 to resolve issues with old drawings where bug fixes already exist, by changing the display for the drawings they are applied to (one by one manual process): <ul style="list-style-type: none"> <li>• The options can be set to the SPR number it resolves, some identifying string defined by Development or to ALL. <ul style="list-style-type: none"> <li>• Setting the option to an SPR number instructs Pro/ENGINEER to apply or remove the bug fix for this SPR (You can stack any number of update instructions)</li> <li>• There are a few of string values as well, that apply or remove certain changes to the drawing</li> <li>• Setting the option to ALL instructs Pro/ENGINEER to update/downgrade the drawing to all the modern settings for versioning attributes <ul style="list-style-type: none"> <li>• Since this will apply hundreds of individual fixes cumulated over the years, with unpredictable reciprocity, it should only be used for testing, whether any fix exists</li> </ul> </li> </ul> </li> </ul> </li> </ul>

	<ul style="list-style-type: none"> <li>• If the test shows positive, use database query or involve PTC Technical Support to figure out the appropriate number(s) to set</li> <li>• The configuration options <code>enable_auto_drawing_update</code> and <code>auto_drawing_update</code> were introduced in Pro/ENGINEER Wildfire 2.0 M140 to automate the drawing update process</li> <li>• Enabling this functionality will result in drawings to update automatically upon retrieval and increase the danger of unexpected display changes: <ul style="list-style-type: none"> <li>• The option <code>enable_auto_drawing_update</code> can be set to either YES or NO to enable or disable the automatic update functionality. <ul style="list-style-type: none"> <li>• It can only be set in the <code>config.sup</code> file, not the <code>config.pro</code> (<code>config.sup</code> must be placed in the text directory)</li> </ul> </li> <li>• The option <code>auto_drawing_update</code> can be set in <code>config.sup</code> or <code>config.pro</code>, but is only enabled if <code>enable_auto_drawing_update</code> is set to YES <ul style="list-style-type: none"> <li>• It behaves similarly to the detail setup option <code>update_drawing</code> and can be set to NONE, ALL or a specific value. <ul style="list-style-type: none"> <li>• Setting the option to NONE results in no drawing updating.</li> <li>• Setting the option to a specific value (i.e. 1234567) forces drawings to update to that bug fix and like the detail setup option the configuration option can be stacked (set multiple times).</li> <li>• Setting the option to ALL instructs Pro/ENGINEER to update all drawings to all the modern settings.</li> </ul> </li> </ul> </li> <li>• A special feature of the options <code>enable_auto_drawing_update</code> and <code>auto_drawing_update</code> is that they both can be used multiple times in the <code>config.sup</code> (usually options can be used only once in the <code>config.sup</code> and further entries of the same option are ignored)</li> <li>• This is useful to specify a couple of drawing updates and then prevent the users from setting additional ones</li> <li>• For example to set the two updates 1234567 and 7654321 and then disallow setting more values, add to <code>config.sup</code>: <pre>enable_auto_drawing_update yes auto_drawing_update 1234567 auto_drawing_update 7654321 enable_auto_drawing_update no</pre> <ul style="list-style-type: none"> <li>• If the fourth line <code>enable_auto_drawing_update no</code> is removed, additional update values can be set in the <code>config.pro</code> of system and user</li> </ul> </li> </ul> </li> </ul>
Related	<ul style="list-style-type: none"> <li> <a href="#">Unable to create BOM balloons using the "Create Balloons - By Component" and view option in Creo Parametric 2.0</a></li> <li> <a href="#">Re: How to update all sheets automatically?</a></li> <li> <a href="#">Re: Creo 3 Drafting</a></li> <li> <a href="#">Re: Delete drawing sheet without selecting it?</a></li> <li> <a href="#">Restore drawing Show/Erase functionality in order to search all non-displayed dimensions (erased or deleted)</a></li> <li> <a href="#">Sketched entity fill in a drawing disappears after saving, closing, and reopening drawing in Creo Parametric 3.0</a></li> </ul>
Others also liked..	<ul style="list-style-type: none"> <li> <a href="#">"Which data exchange versions of: AutoCAD, IGES, Parasolid CADD5, CATIA V4, CATIA V5, Creo Elements/Direct (CoCreate), Creo View (ProductView), DXF &amp; DWG, Inventor, JT, NX / Unigraphics, Rhinoceros, Solidworks, STEP, are supported in Pro/ENGINEER and Creo"</a></li> <li> <a href="#">"On Windchill PDMLink with Creo View Adapters, all Creo Parametric workers fail to publish with the error \"Failed to register Windchill server - -15\""</a></li> </ul>

### SPR Information

### Related Documents

Type	Number	Status	Description
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