3D-ANNOTATION

Config.pro additional settings

1. Model_notes_as_labels = no
2. Create_drawing_dims_only = no – saves driven dims created in drawing mode on solid geometry in the solid.
3. Hole_diameter_override = yes
4. Parenthesize_ref_dim = yes
5. Web_browser_homepage = https://windchill.aps.nl.gov/Windchill
6. Windows_browser_type = ie_in_process, or Mozilla_based_browser
7. Save the changes

Drawing Options

1. Activate a drawing
2. File => Drawing Options
3. Sort Alphabetical
4. Allow_3d_dimensions = yes
5. Dual_dimensioning = primary[secondary]
6. Lead_trail_zeros = std_english[std_metric]
7. Gtol_datums = std_asme
8. Save the changes

Add Icons to Modeling toolbar. RMB in toolbar area=>commands=>View. To add a command drag it to a Toolbar/Menu pane.

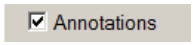
1.  Annotation Feature
2.  Annotation Element Display
3.  Annotation Orientation
4.  Activate Annotation Orientation
5.  3D Notes Display
6.  Full 3D Notes Display
7.  One, expand selected model only. Finds parts in the model tree.

Assembly Move Hotkeys

1. To move a component depress CTRL & ALT – RMB drag the part.
2. To roll or spin a component depress CTRL & ALT – MMB (WHEEL).

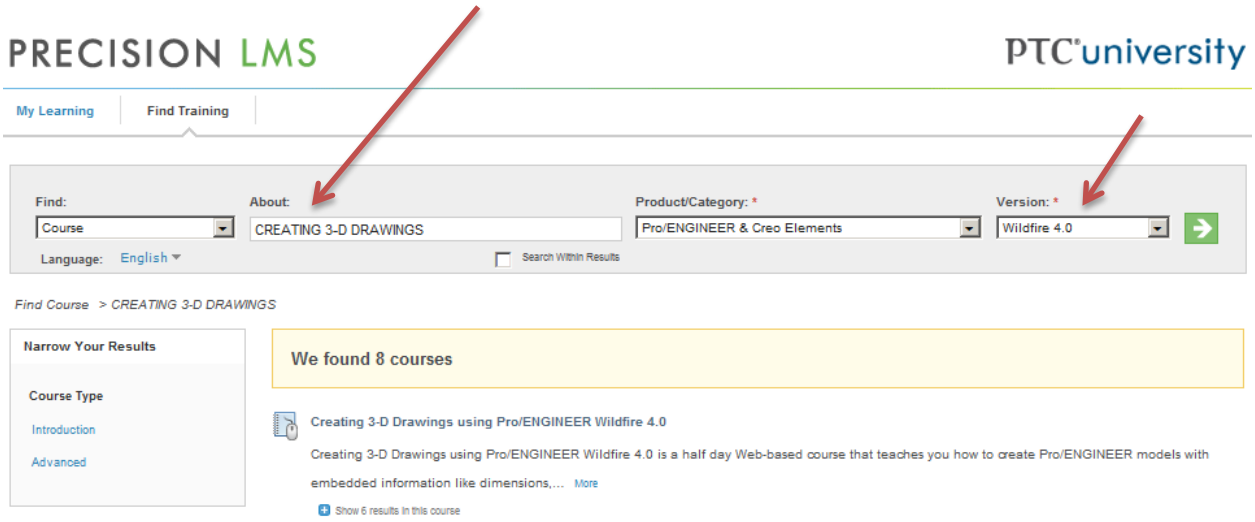
Model tree filters

1.  Settings => 

Tree Filters => Display => on 

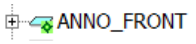
Annotations => OK
2. 

Save Settings File



1. Login
2. Find Training
3. Find: Course
4. About: Creating 3-D Drawings
5. Product/Category: Pro/ENGINEER & Creo Elements
6. Version: Wildfire 4.0
7. Green arrow
8. Creating 3-D Drawings using Pro/ENGINEER Wildfire 4.0
9. Download and install the model files
10. Complete the tutorials

My notes for 3d annotation

1. Create layers, Anno_Front, Anno_Top, Anno_Right, Anno_Iso
2. 

Activate the layer of the view you wish to dimension on.

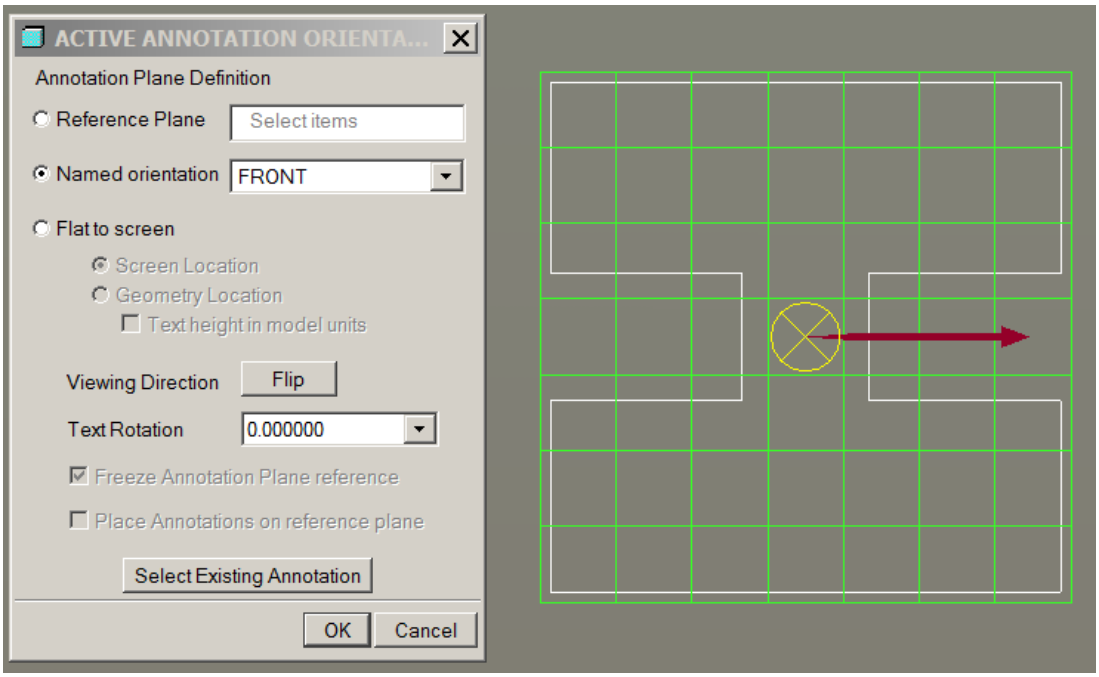
If you are going to be creating a group of annotation elements specific to a feature either manually or by using “Create Driving Dimension AE” command, you can define and activate the layer and have all of that information automatically populated to that layer.

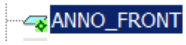
3.  Check the Named View List to make sure the view are there.
4. Create a new Sketch or Edit an existing one.
5. Place and arrange sketch dimensions like on a 2d drawing.
6. Extrude –Z direction (into the screen). Or +Z then select dimension and RMB ‘Move to Plane’, select the upper geometry surface, but the leader lines remain connected to the original plane.
7. For Revolved constructed parts, select the Right (side) plane and Top for the sketching planes so the Z-axis will be the centerline (beam direction).
8. Difference between Driven and Driving dimensions. Driving dims are from the Sketch (model), Driven Dims are Annotation Elements
9.  Toggle on ICON Annotation Element Display
10. 

Set the Selection Filter TAB to Annotation. To be able to select the dimensions from the screen.
11. For Holes to display the dimensions, addition layers must be unhidden. Dimensions, threads, notes, datum_axis, axis.
12. You can save layer display status: With the model and with combined view states.

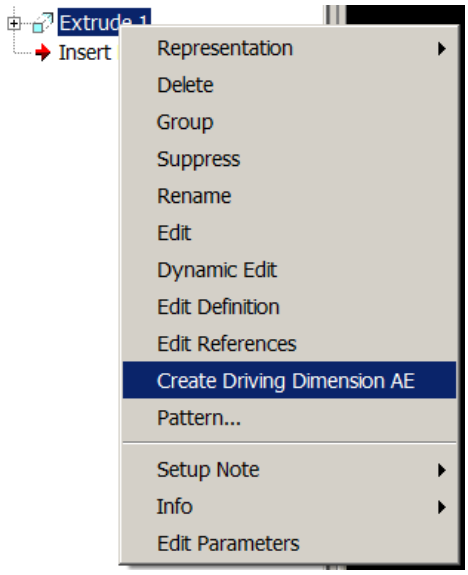
Procedure:

1. View => Annotation Orientation => Named orientation => the view you are going to dimension on (FRONT). Red and Yellow direction arrows for the dimension or text.

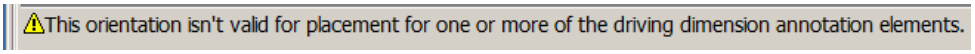


2. 

Activate the corresponding layer (FRONT).
3. In the Model Tree, select the main extrusion, RMB.
4. Select: Create Driving Dimension AE.



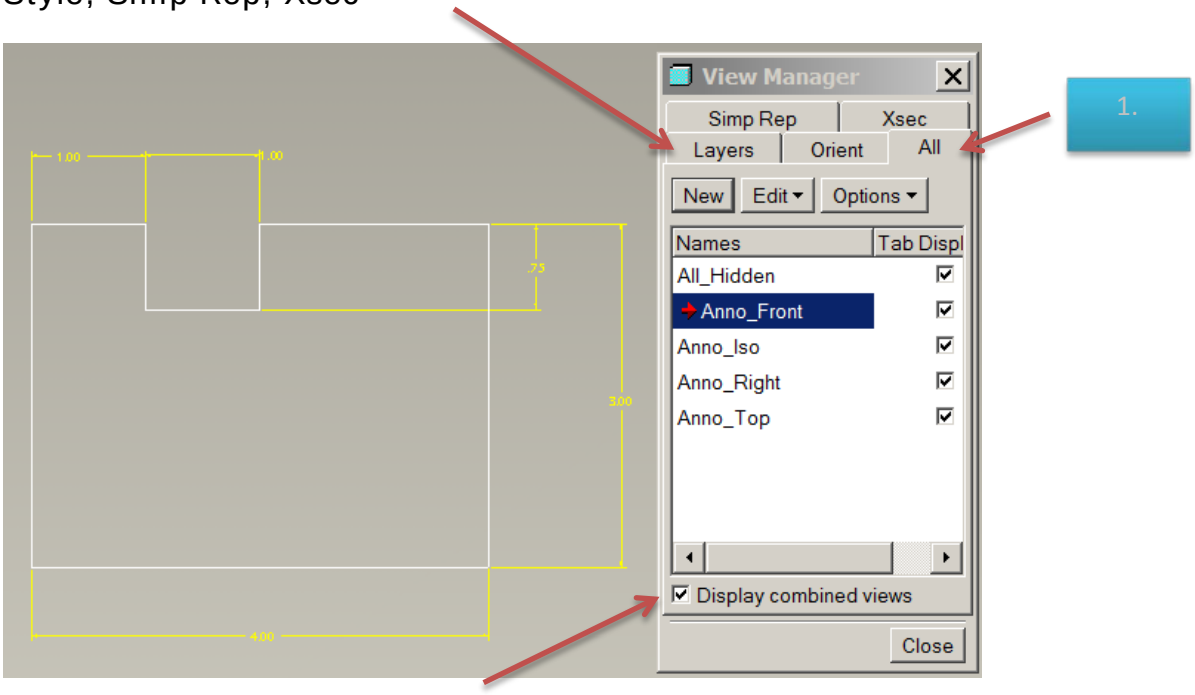
5. Read the message: One or more dimensions were not converted to driving dimension annotation element, because the extruded distance is along the z-axis. Add it in when dimensioning the top or right view.



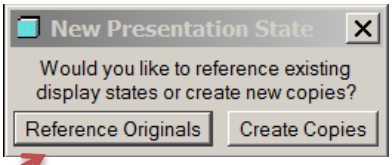
6. 

Set the Selection Filter TAB to Annotation. To be able to select the dimensions from the screen.
7. Select the Annotation Element (dimension) to move or modify. To move a dimension select it then RMB => Move, move the cursor to the location you want the dimension to be and click. This behavior is changed and available in Creo 2.0. To flip arrows select the AE then RMB Properties => Display => Flip Arrows.
8. Continue to create the other view.

 View Manager Combined Views: Combines view with Orient, layers, Style, Simp Rep, Xsec



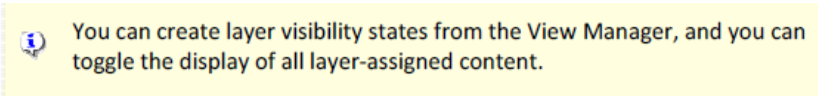
1. All tab => New => Anno_Front. Select Reference Originals



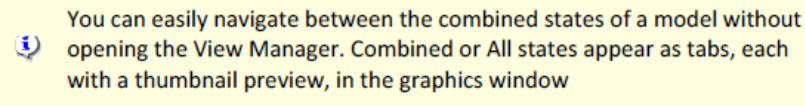
Because combined view states reference existing view states for their definition, the desired view states such as orientations and cross sections should be created before starting to define the combined view. When you begin the creation of a new combined view state, you are given two options for how to use the existing view states that are selected for the combined view definition:

- Reference Originals — References the original view states that were selected to define the combined view. If one of the original view states is modified, the combined view will update to reflect that change.
- Create Copies — Creates copies of the view states to be used in the combined view. The combined view is independent of the other original view states and will not reflect any changes that are made to them.

2. Orient tab => Front
3. Goto the Layer Tree and hide all but the Anno_Front and Dimension. If hole are present unhide additional layers (axis, datum axis, notes).
4. Go back to the ALL tab, RMB the Anno_Front select Redefine.
5. From the Orientation: select the view you are dimensioning from the drop down menu (Front).
6. From the Layers: select the corresponding layer, from the drop down. RMB to Save layer changes.



7. Green check
8. Check Tab Display and Display combined views
9. May have to move dimensions from the layer Dimension to the Anno_view.
10. Select TABs at the bottom of the screen to change views. Preview of the display is shown.



11. Remember to save the layer changes, hide and unhide.
12. Continue on with the remaining views.

Combined View States

Combined view states enable you to define the 3-D equivalent of a 2-D drawing view. You can use a combined view state to create a 3-D drawing view by specifying:

- The orientation of the view.
- A simplified rep to be used for the view.
- A cross section to be applied in the view.
- The layer display status to be used for the view.

By setting up combined view states you can control exactly how you want the annotations to display on the model. Other users can then toggle through the combined views to see the annotated view that you set up. However, unlike a 2-D drawing, the users still has the flexibility to zoom

and rotate the model however they wish if they cannot easily interpret an annotation in any of the combined view states.

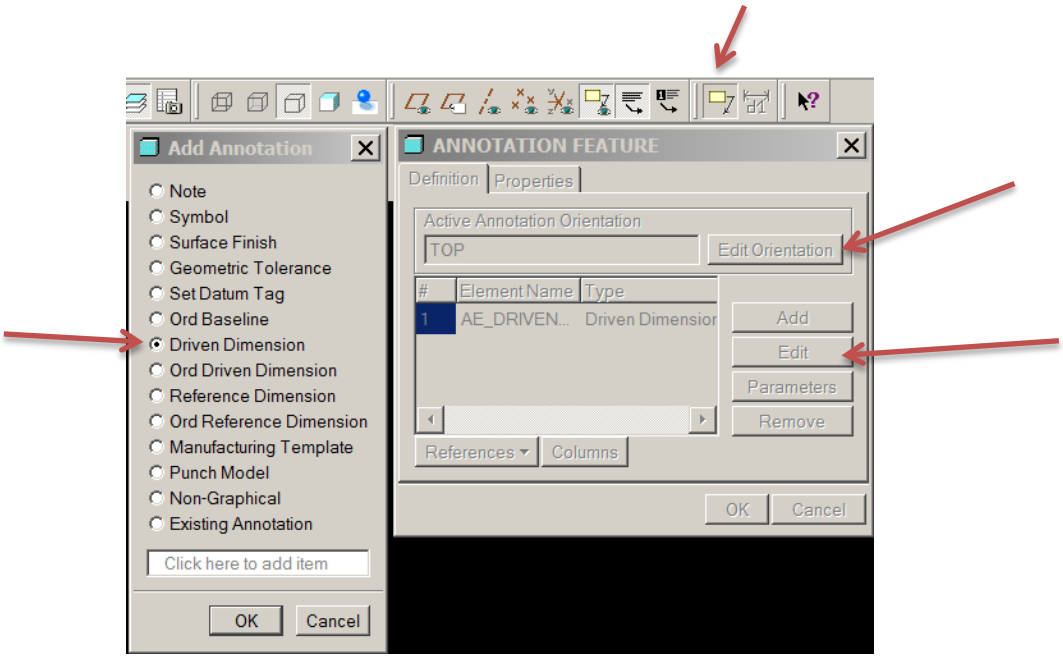
If a 2-D drawing is still necessary, the combined view states can be referenced when placing a view on the 2-D drawing, eliminating the need to define the same views twice.

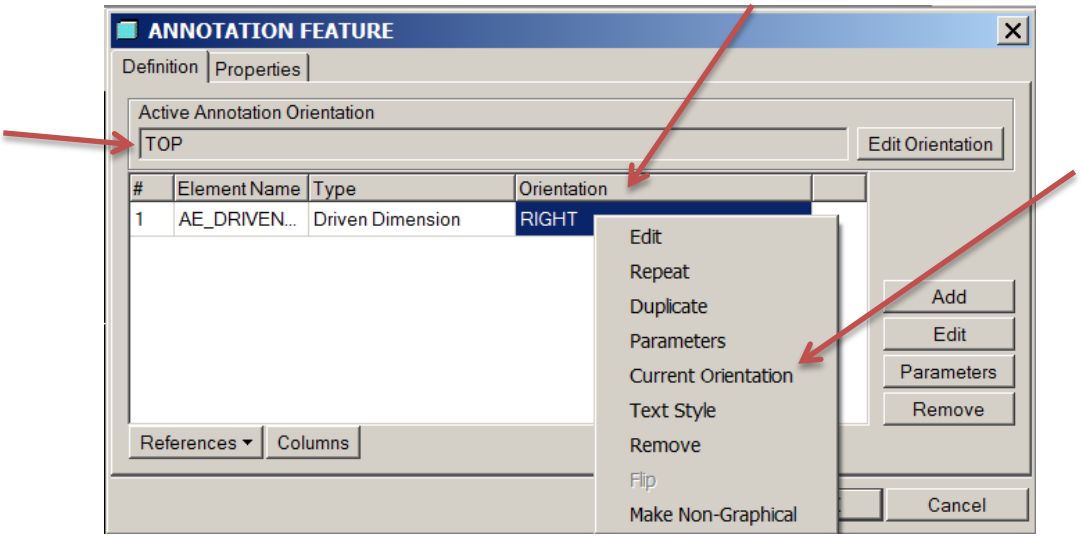
Drawing

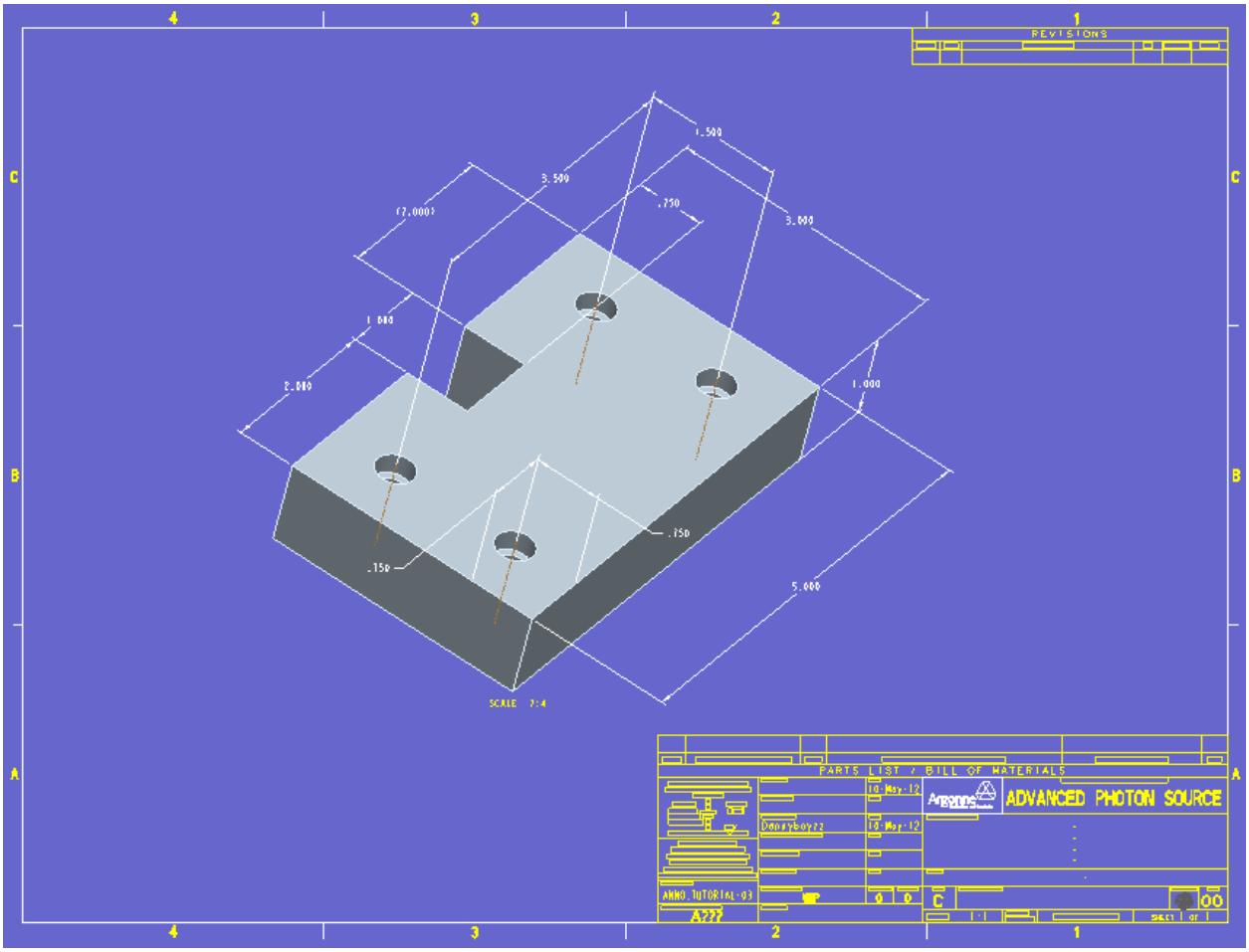
1. Create new drawing.
2. Place views
3. Create ISO view.
4. Annotation, show dimensions.

Assembly and part 3D dimensions, GD&T, note, symbol, surface finish, ect.

1. Activate layer, annotation orientation.
2. Use Annotation Features to Add Annotation.
3. Driven Dimensions => OK, select the geometry to dimension.
4. Attach type




ANNOTATION FEATURE
Definition
Properties
Active Annotation Orientation
TOP
Edit Orientation
| Element Name | Type | Orientation
1 | AE_DRIVEN... | Driven Dimension | RIGHT
Edit
Repeat
Duplicate
Parameters
Current Orientation
Text Style
Remove
Flip
Make Non-Graphical
Add
Edit
Parameters
Remove
References
Columns
Cancel


REVISIONS
PARTS LIST / BILL OF MATERIALS
ADVANCED PHOTON SOURCE

Definitions:

Driven dimension is created by the user. This type of dimension reports a value based upon the references selected when the dimension is created. That is, the dimension value is driven by the geometry selected, and therefore it is not possible to modify the value of a driven dimension. A driven dimension does not pass back to the model

Erasing and Deleting Annotations Theory

If, at any point during drawing creation you decide that you no longer want certain shown items, you can erase or delete them. The differences between these two options are as follows:

- Erase — Temporarily removes the items from the display. The items are shown grayed out in the drawing tree.
    - Erased items can be returned to the display by right-clicking and selecting Unerase.
- Delete — Removes the items from the drawing.
    - Any item originating in the model is retained in the model, and can be shown again.
    - Any item created in the drawing, such as dimensions or notes are deleted and will need to be re-created.

To erase/delete items, you select them in the drawing, then right-click and select Erase or Delete. You can select items to erase or delete using the following methods:

- Select an individual item.
- Press CTRL and select multiple items.
- Use a selection filter to quickly select desired items.
- Select items from the drawing tree.

Config options:

Display_annotations = Yes

Visible_annotations_scope = all

Questions

1. How to activate Annotation Orientation and View Manager All with the combined view?
2. How to activate the corresponding Layer when a view is selected?
3. How to have Annotation Orientation follow to the selected view?

Whats new in Creo 2 for 3d Annotation

Head-to-head comparisons for eight of the most common CAD activities show that design productivity is doubled with the Creo 2.0 interface compared to Pro/ENGINEER Wildfire 5.0. Specifically, 3D annotation is 50 percent faster and measuring time is reduced by 80 percent, along with many other time efficiency improvements. Capture detailed model information faster and easier in a new dedicated 3D annotation environment. Create annotations and combined states with ease.

SHORT DESCRIPTION	Pro/ENGINEER Wildfire 5.0	Creo 1.0	Creo 2.0
SHEET METAL (continued)			
Form feature creation (with optional rounds and taper) or a partial piercing from a sketch.			●
Sheet metal walls from two intersecting walls with a bend between them.			●
Automatic creation of a flat pattern family table instance from a sheet metal model.			●
Automatic highlighting of overlapping geometry in the flat pattern preview window.			●
DETAILING			
Improved user interface and workflows for key stages of detailing drawings.	●	●	●
Direct selection of annotation items from the new Drawing Tree.	●	●	●
Precise print preview, including pen mappings and line weights.	●	●	●
Streamlined and familiar ribbon interface in Drawing mode for faster access to commands.		●	●
Optimized drawing tables workflow focused on most common user interactions.		●	●
Improved workflow for creating and inserting tables.		●	●
Tables Gallery for previews of predefined tables.		●	●
Properties dialog for tables and BOM balloon regions.		●	●
Text wrapping in table cells.		●	●
Extended controls and setting for BOM balloons, including type and reference text.		●	●
Dynamic repositioning of dimensions by freely dragging the dimension text.			●
Dynamic repositioning of dimensions and snap to the center of witness lines or adjacent dimensions.			●
Dynamic repositioning of dimensions by locking dimension lines, while still freely moving dimension text.			●
3D ANNOTATIONS			
Streamlined and familiar ribbon interface in 3D Annotation mode for faster access to commands.	●	●	●
Tools to set and manage annotation plane orientations.	●	●	●
Streamlined tools for creating and manipulating annotation content.		●	●
Tools to create and manage combination states.		●	●
Tools to create 3D annotations based on existing model dimensions.		●	●
3D Detail options to control cosmetic appearance of annotations on models.		●	●
3D Detail options to control appearance of annotations per 3D standards (ASME Y14.41-2003, ISO 16792, etc.).		●	●
Options to dynamically reposition dimensions in 3D drawings just like dynamic dimension repositioning in 2D drawings.			●
Dynamic dragging of dimensions in 3D with intelligent snapping to center of witness lines and adjacent dimensions.			●
Selection of endpoints of edges as references for dimension annotations in 3D.			●
Selection of edges as edge references for dimensions if they are normal to the annotation plane.			●