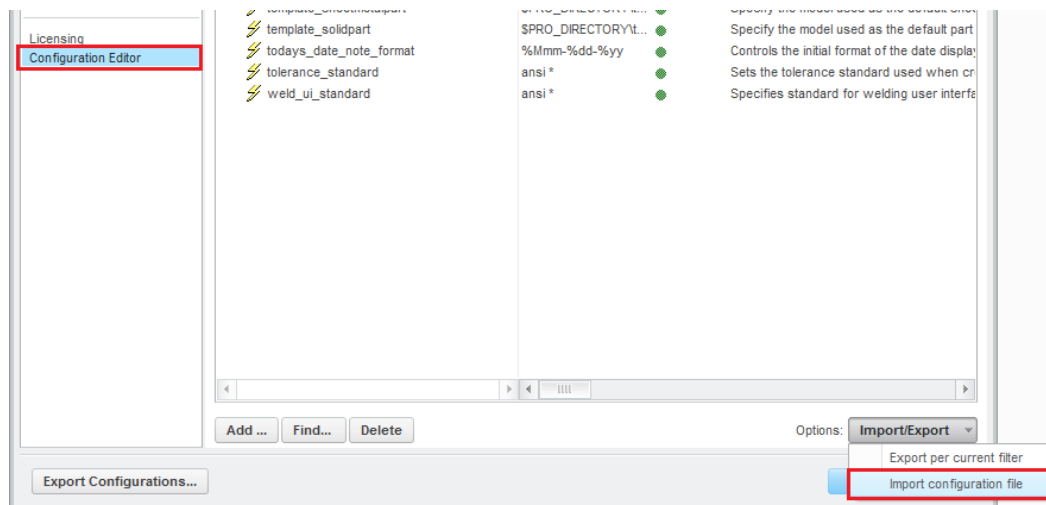


Instructor: Mike Philpott (email: mphilpot@illinois.edu)

Configuration of Creo

Prepare Creo for drawing creation.

- Open Creo Parametric 3.0 from the Start Menu.
- Set your working directory.
- Select "File" → "Options."
- Click "Configuration Editor" at the bottom of the column on the left.
- Click "Import/Export" → "Import configuration file" at the bottom left corner of the "Options" window.
- Select the "config.pro" file in your working directory.
- Click "Open". Click OK. Click "No".
- Numerous settings have just been applied to your Creo session. Import this file each time you start working in Creo to ensure that your future parts and drawings have the correct settings. Consult the "Parameter Changes" files for a complete list of changes.



Drawing Tips

The following tips and notes should assist you with detail drawing creation.

Modifying the Sheet Scale

- Double click the scale in the bottom left-hand corner of the screen (image below). Enter a new scale value in the prompt emitting the colon (eg. enter '2' for a 2:1 scale) and click the green checkmark to confirm the scale change.
- The "Scale" section of the title block will update automatically.

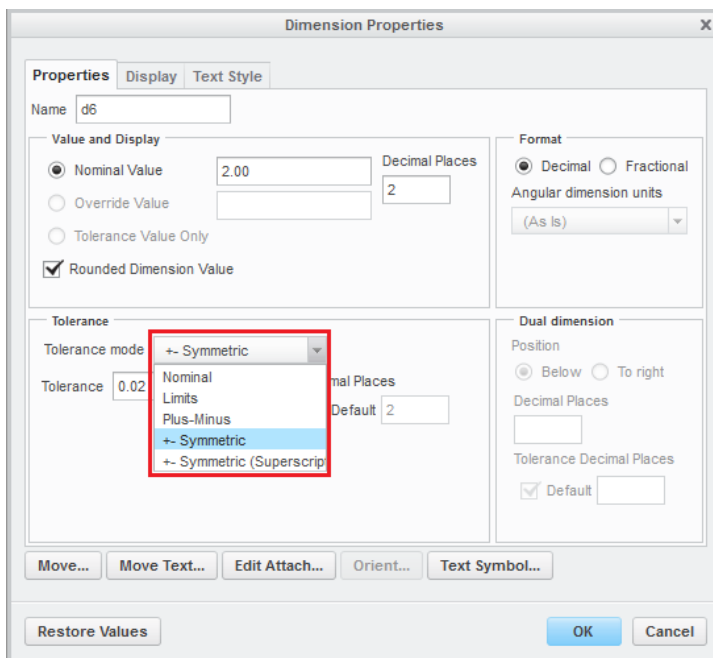


Flipping Dimension Text and Dimension Arrows

- It is often necessary for aesthetic or layout considerations to flip the arrows or text of a dimension. To do this, select the dimension you would like to edit. In the RMB menu, select "Flip Text" or "Flip Arrows" as needed.

Tolerance Specification

- Select the dimension to tolerance. Click "Properties" in the RMB popup menu. In the "Dimension Properties" window, select the appropriate tolerance scheme from the "Tolerance Mode" dropdown menu (image below). Enter tolerance values. Click OK.



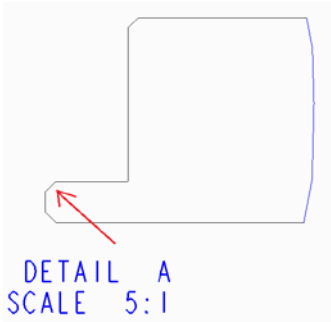
Creating the Correct Detail View in 4455-002

- Delete the detail view created in Exercise 16.
- Delete the chamfer dimension created in Exercise 16.
- Create a new detail view for the opposite side of the part.

- Show the chamfer dimension for that side of the part. Select "Chamfer 2" in the model tree. Select "Show Model Annotations" in the RMB menu. Select the dimension to display. Click OK.
- Edit the attachment and the text of the chamfer dimension (instructions below).

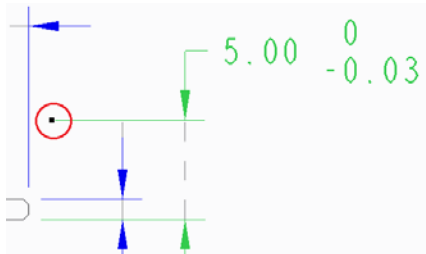
Editing Dimension Attachment

- Select the dimension to move. Click "Edit Attachment" in the RMB menu.
- Select a new edge/surface for attachment (image below).



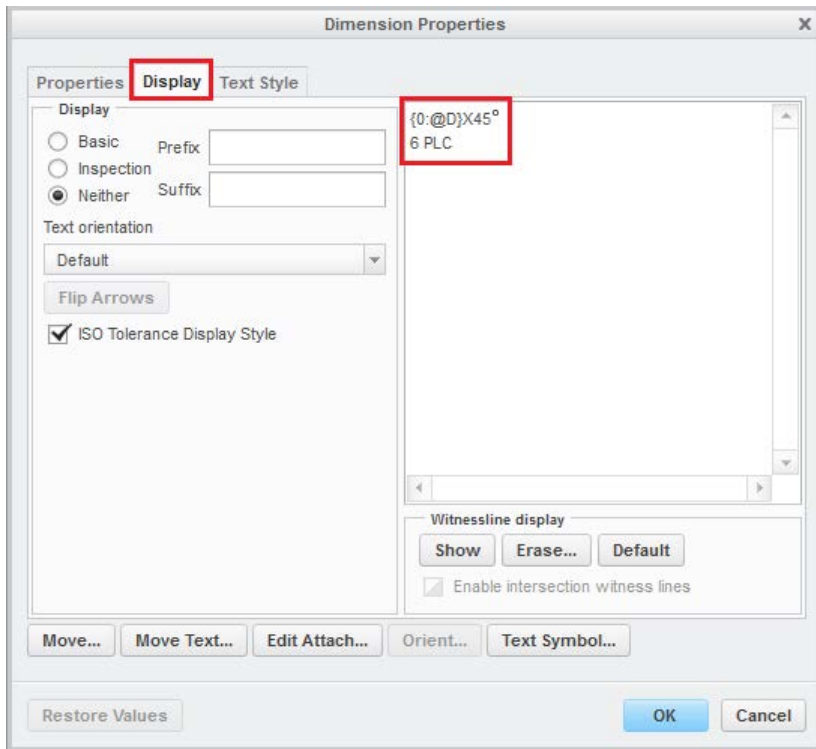
Editing Witness Lines

- Select the dimension to edit. Place your mouse over the end of the dimension's witness line. A drag handle should appear (image below). Drag the handle to a new location. Click open space to deselect the dimension. It is OK if the witness line overlaps a visible line. Creo will automatically clip the witness lines when printing.



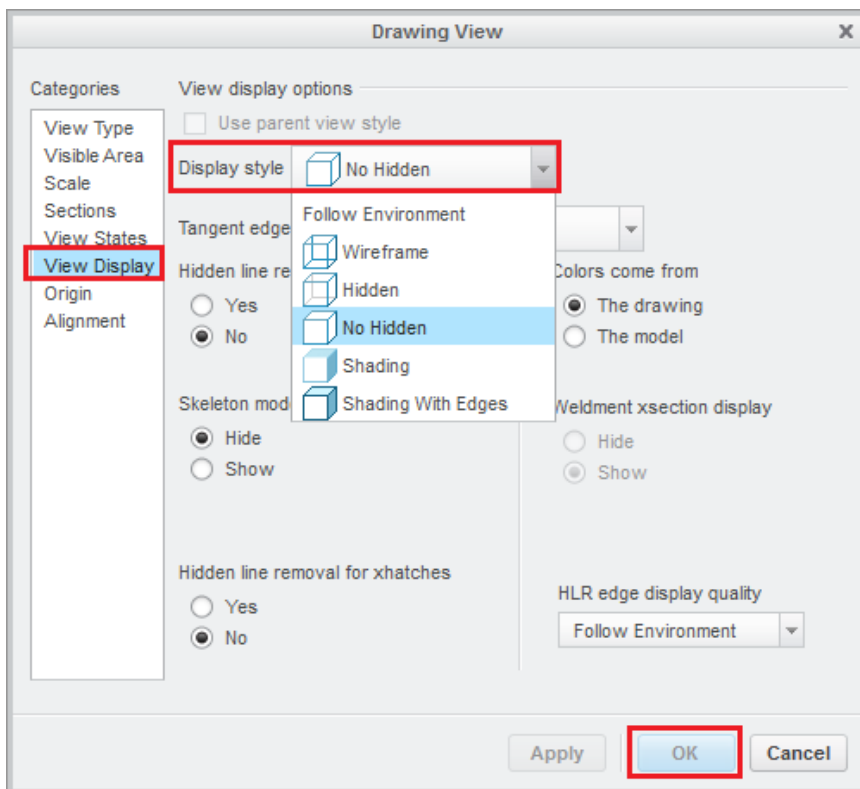
Adding Text or a Note to a Dimension

- Select the dimension to edit. Click "Properties" in the RMB popup menu. In the "Dimension Properties" window, navigate to the "Display" tab. Enter any additional text in the text box on the right side of the "Dimension Properties" window (image below). Be careful not to delete the existing smart text (eg. @D, {0:@D})! Click OK.



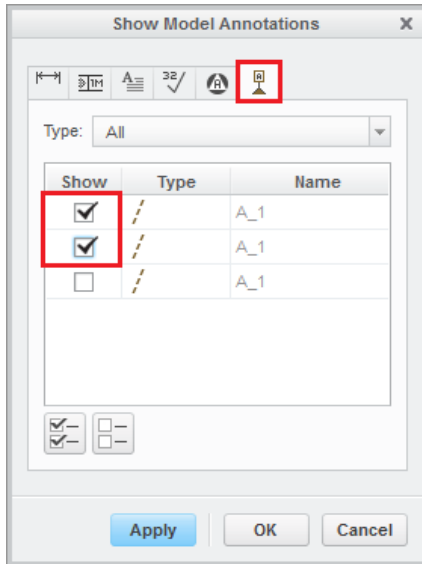
Changing a View's Display Style (eg. from "Hidden" to "No Hidden")

- All isometric and assembly views should NOT show hidden lines. All other part views should show hidden lines. The drawing setup file sets "hidden" as the default display style for new views.
- Left click and select the view on the drawing sheet. The view can also be selected from the "Drawing Tree" (the upper part of the column on the left side of the screen).
- Select "Properties" from the right mouse button menu.
- Click the "View Display" category in the "Drawing View" window.
- Select the appropriate display style from the "Display Style" drop-down menu.
- Click OK.



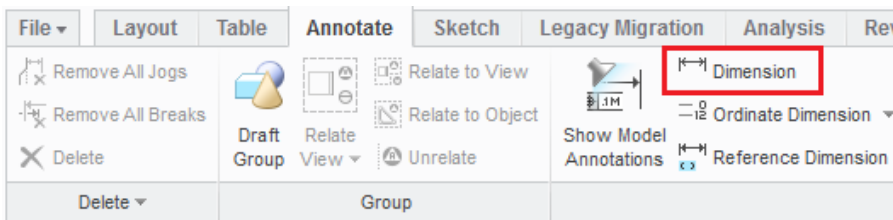
Displaying Centerlines

- Select the feature(s) requiring centerlines (often holes and revolves).
- Select "Show Model Annotations in the RMB menu.
- Navigate to the "Show model datums" tab in the "Show Model Annotations" window (image below).
- Select the appropriate axes/centerlines to display. Note that we do not display axes or centerlines in the isometric part view. Click OK.



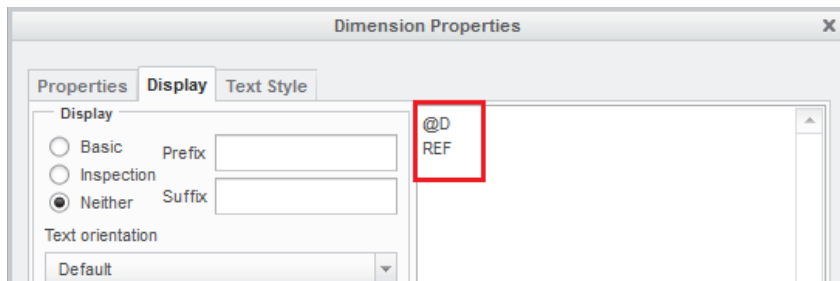
Creating Reference Dimensions in the Drawing

- Navigate to the "Annotate" tab. Click the "Dimension" button (image below).



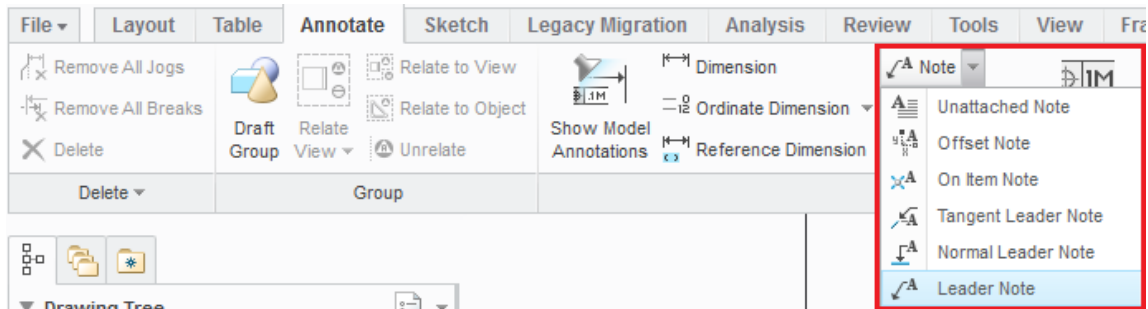
- Select the appropriate attachment reference in the "Select Reference" box.
 - o Note that the default "Select an entity" button is good for most dimensioning.
 - o The "Select the tangent" button option will be necessary if dimensioning to the outer edge of a circle or arc.
 - o Note that the selection option can be changed between dimension reference selections. In other words, you can select one edge for a dimension, click a new button in the "Select Reference" box, and then select the second edge for the dimension - all while holding down the CTRL key.

- Select the first edge reference for the dimension.
- While holding CTRL, select the second edge reference for the dimension.
- Press the middle mouse button to place the dimension.
- Press the middle mouse button again to close the dimensioning tool.
- Select the newly created dimension. Click "Properties" in the RMB menu.
- Navigate to the "Display" tab.
- Enter the text "REF" below (or on the same line as) the dimension smart text in the dimension text box as shown below. Click OK.



Creating Notes with Leaders

- Navigate to the "Annotate" tab. Click the down arrow next to the "Note" button and select the "Leader Note" tool (image below).
- Left click to select a line for leader attachment. Press the middle mouse button to place the note.



Editing the Title Block

- The title block was created using tables. To edit the text of these tables, first navigate to the "Table" ribbon at the top of the screen. Double click the text in the title block to edit it. Note that you cannot edit tables unless the "Table" ribbon is displayed in Creo's drawing environment.

Adding a Section View

- First, add the appropriate projection view or general view for the section. The 'appropriate' view is the one where the sectioned plane is parallel to the plane of the paper.
- Select the view where you would like to show the section. In the RMB popup menu, select "Properties".
- Select the "Sections" category in the left column of the "Drawing View" window.
- Select the "2-D Cross Section" radio button. Click the "+" button.
- Select an available section and click OK. Available sections have a green checkmark displayed next to the section name. The cross-section should display in the drawing.
- You will have to open the assembly and define a new cross section ' B' in order to create the second cross-section.

Adding Section View Arrows

- Deselect everything.
- On the "Layout" ribbon, click the "Arrows" button.
- Click the view for which you would like to display the arrows (the section view).
- Click the view on which you would like to display the arrows (the top view).

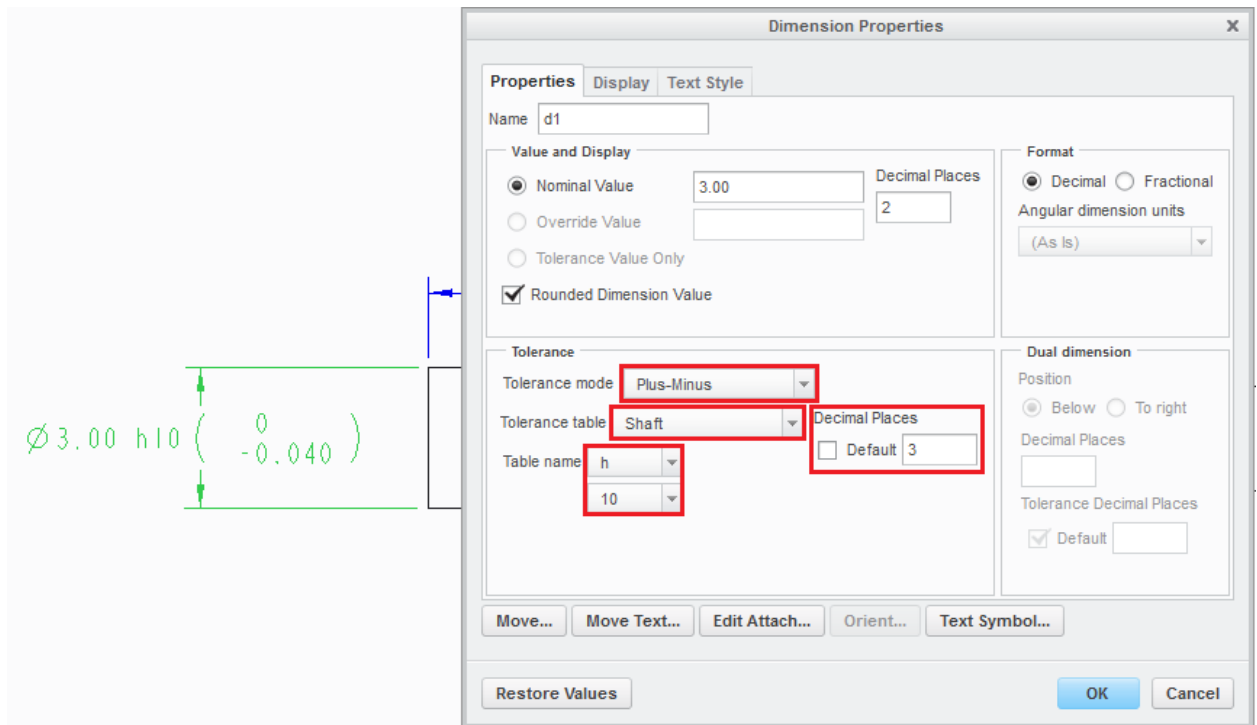
Aligning Views

- Select the view to be aligned. In the RMB popup menu, select "Properties."
- Select the "Alignment" category in the left column of the "Drawing View" window.
- Check the "Align this view to other view" box. Select a view for alignment.

Adding an ISO Fit Tolerance to a Hole or Shaft

- Set the part tolerance mode to ISO/DIN & load a tolerance table.
 - o Open the part file, not the drawing.
 - o Select "File" → "Prepare" → "Model Properties."
 - o On the "Tolerance" line, click "change".
 - o In the menu manager in the upper right corner of the screen, click "Standard". Click "ISO/DIN". Click "Yes" to regenerate.
 - o Click "Tol Tables" in the menu manager. Click "Retrieve".

- Select the appropriate table from the file browser and click "Open".
 - Click "Yes" to regenerate. Click "Done/Return". Click "Close".
 - Save your part file.
- Open the drawing of your part.
 - Edit your dimension to show an ISO tolerance (image below).
 - Select the hole or shaft dimension. In the RMB popup menu, select "Properties."
 - Set the tolerance mode to "Plus-Minus".
 - Select the appropriate tolerance table and table name.
 - Uncheck the "Default" box under "Decimal Places". Specify and appropriate value for the number of decimal places.
 - Click OK and save your drawing.
 - If you need a different tolerance table or additional tolerance tables, be sure to load them in the part/design environment, NOT the drawing environment.



Printing Drawings

- Before printing, double check the drawing.
 - o Ensure that all text is capitalized.
 - o "Cleanup" all dimensions, where possible.
 - o Check that no lines or text overlap.
 - o Look for clutter and overall aesthetics. Ensure that there is adequate space between lines, text, borders, and views.
 - o Ensure that all datum feature visibility is turned off! With the exception of datum features specifically added to the drawing, all planes, axis, datum points, and coordinate systems should be hidden! Visibility of these features is controlled under the "View" tab.
- Print your drawing to a PDF.
 - o Select "File" → "Print." Click the "Print" button in the "Print Preview" ribbon.
 - o Select the "Adobe PDF" printer. Click OK.
 - o Save the PDF to your working directory with the same filename as your part and drawing (eg. <netid>_<partnumber>.pdf).
- Send your drawing to the physical printer.
 - o Open the PDF. Select "File" → "Print".
 - o Select the appropriate printer.
 - o Ensure that there is no page scaling. (Page scaling is set to "None".)
 - o Click OK to print your drawing.

Printing Multiple Drawing Sheets at Once

- After selecting "File" → "Print". Click the "Settings" button on the "Print Preview" tab. Select "All" under "Sheets". Click OK. Print the drawing as per the instructions above.