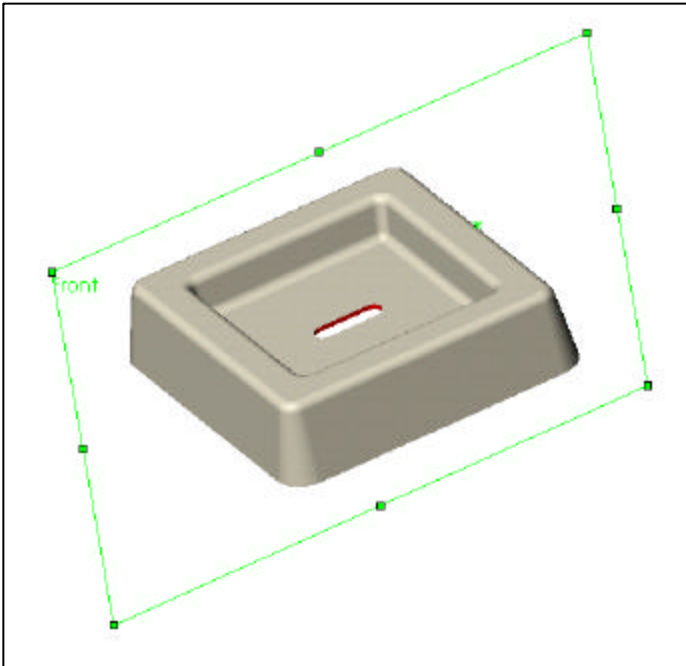


# SW Tips/Tricks

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If you would be interested in reading about a specific SolidWorks topic or if you have questions you would like to see addressed in future issues of SW Tips/Tricks please contact me by email at [sluder@triaxialdesign.com](mailto:sluder@triaxialdesign.com).

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## Dimensioning to Virtual Sharps

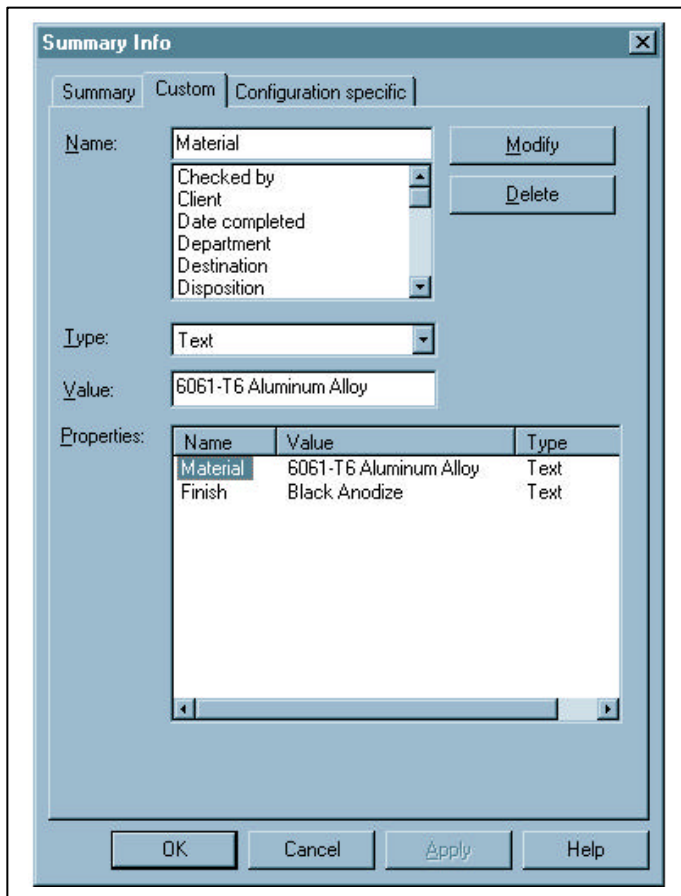
Virtual sharps can be used when both dimensioning the model and dimensioning the drawing. In the model, open a sketch on a plane or surface that you would like the virtual sharp point to be placed. Select two planar surfaces (the two selected surfaces do not have to be adjacent to each other) and then pick the Point entity in the Sketch tools toolbar. The intersection of the two selected surfaces is a line, and at the location where that line pierces the sketch plane SolidWorks will place a virtual sharp point. This point can be used to dimension or relate other entities of features. To use this tool in a drawing, select two edge entities in a drawing view, and pick the Point entity in the Sketch tools toolbar. SolidWorks will place a virtual sharp point. The style of this entity can be changed from Tools, Options, Detailing Tab, and then the Virtual sharp button. You can select several different options.

## Projecting Broken Views

Often when you create a broken view you need to another view of the same part or assembly that it broken in the same location. If you want the vertical or horizontal break to line up exactly with each other, just create the first view, insert the break lines, adjust the break width, and break the view. Then use the Projected View tool to create the other view and it will already be broken in the same location as the parent view.

## Custom File Properties on Drawing Template

Each part or assembly file has individual file properties. If you are familiar with database terms, the file would be the "record" and the individual file properties would be the "fields". You can look at these file properties at any time by opening the file and picking File, Properties. A Summary Info dialog



will appear. This will have a tab for Custom, and Configuration specific. If you switch to the Custom tab, You will see a list of fields that has been pre-defined. This list cannot be changed, however you can add fields of your own. Open a part file and then go to File, Properties, go to the custom tab, and add a new field in the Name box. Let's add a field named "Material". Then go down to the Type box and specify Text, then go to the Value box and type the material specification. Let's use "6061-T6 Aluminum Alloy" for this example. After this be sure and pick the Apply button. You should see your custom property listed in the lower part of the dialog box. Repeat this for a field called "Finish" and a value of "Black Anodize". The part file now has a custom properties associated with it. You can add properties for anything you would like, i.e. weight, vendor, price, etc.

Now what can you do with these properties. You can access these properties form Windows Explorer. Just right select the filename and select properties. You can access and modify the same way you did inside of SolidWorks.

You can use these custom properties inside of SolidWorks documents. You add a custom column to your Bill of Materials on your drawings. Then whenever you use the individual part in an assembly, and create a Bill of Materials for that assembly, the custom column will contain the custom file property for the individual part.

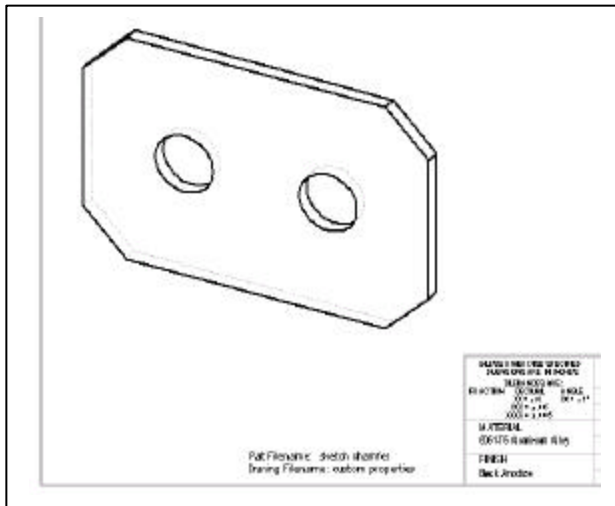
You can modify your drawing template so whenever you use the template, the custom file properties for the part or assembly is being documented will show in the template. As you customize the drawing template, that is, add your company logo, change the default tolerance block, etc., add some text that is actually linked to the custom properties of the drawing file or the model that is being documented in the drawing.

For this example we are going to add five different items in the template that will link to the custom file properties of the model and drawing file.

1. Current sheet and total number of sheets in the drawing
2. Drawing filename
3. Model filename
4. Model material
5. Model finish

To do this, first open a new drawing using the template you want to modify. We need to right select inside the template, and select Edit Template.

Zoom to the portion of the template you would like to place the first notation, Current sheet and total number of sheets in the drawing. Pick the location, and Insert, Annotation, Note, to get the dialog box. In the box type, SHEET and then a space. Locate the icon on the right of the dialog with the folder, magnifying glass, chain link, and a sheet of paper (below the web link icon). Pick the icon and you see the Link to Property dialog. Picking the drop down list arrow reveals several predefined SW properties. We are interested in the "SW-Current Sheet" and "SW-Total Sheets" right now. Pick "SW-Current Sheet" form the list, make sure External model reference is not checked, and pick OK. This adds some special syntax to the note that will be replaced in the final drawing with the value for the current sheet number. Then type another space, the word "OF", another



space, then once again pick the link icon. Now we want to go to the drop down list and pick “SW-Total Sheets”. Again make sure External model reference is not checked, and pick OK. This adds some more syntax to the note that will be replaced in the final drawing with the value for the total number of sheets. Now pick OK on the Properties dialog for the note, and the annotation you have just added should appear in blue. This is telling you the note is linked (just like a link on a website is blue). The next two items, Drawing file name and model filename are similar to what we just did. You can pick a location, add a note annotation, and go to the link icon. Now, pick the “SW-Filename” from the dropdown list. If you leave the External model reference box unchecked, the “SW-Filename” will link to the drawing filename. If you check the External model reference box, “SW-Filename” will link to the first model you insert a view of into your final drawing. You may want to add text such as “Drawing Filename:” before picking “SW-Filename” so the final annotation will be easy to interpret which file name you are linking to, the drawing or the model. For the last two items, you need to add some custom fields. They will not be listed in the drop down list. Insert your annotation, then type the name of the custom field you want to display in the final drawing. Do this for both the “Material” and the “Finish”. Be sure you check the External model reference box, because you want to link to the first model inserted into the drawing, not the “Material” of the drawing file itself.

As a final step you want to go to the file pull down menu and pick File, Save Template. This will save the template you just customized so you can use it on future drawing documents. Now close the drawing

file without saving changes.

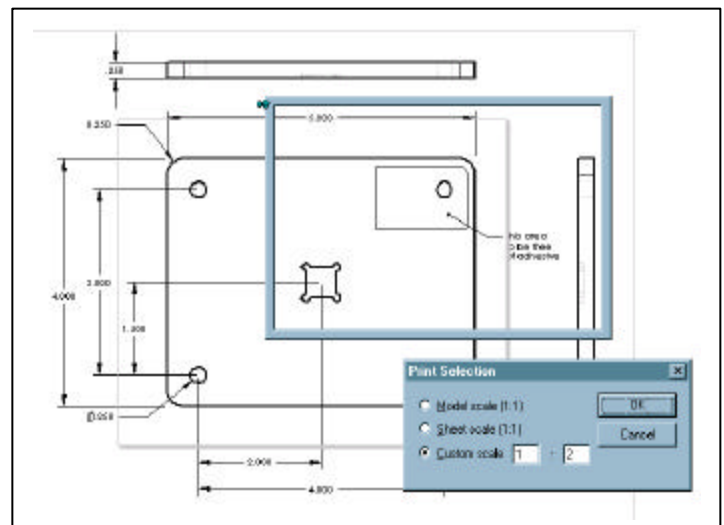
Now create a new drawing of the part you added the custom file properties to at the beginning of this tip. Use the template that you have modified to link to those same custom properties. The template will automatically link the information in the drawing and model files to the annotations in the title block. After you learn these linking techniques, your imagination is the only limit.

### Rotate View about a Point, Edge, or Surface

You may at one time picked the Rotate view icon, then went to the graphics area to rotate your model, and found it rotated funny. Once you realize what you have done, you can use these tools to rotate your model more precisely. To perform these simply pick the Rotate view icon as usual, then pick either a vertex, linear edge, or a planar face. Next drag the mouse and you will see the model rotate around a single point, around the linear edge, or around an axis normal to the point you picked on the surface respectively.

### Print Selection

There is a way to print just a portion of your model or drawing. Under File, Print dialog box, there are three selections under the Print range. These are All, Pages, and Selection. If you choose Selection when you pick OK, another Print Selection dialog box appears along with a frame enclosing a portion of your drawing. This frame represents the portion of your model or drawing that will print. You can change the size of this frame by adjusting the print selection dialog box. If you pick the custom scale, it allows you to enter any scale you desire.



If you would like to receive issues of SW Tips/Tricks please provide us the following information by:  
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## Calendar of Events

**San Diego SolidWorks User Group  
Digital Dimensions, Inc.**

**3934 Murphy Canyon Road Suite B-100  
2<sup>nd</sup> Wednesday of the Month at 7:00pm**

Group discussions, tips, and ideas. Various beginning and advanced topics presented each month. Arrive early for pizza/soda. For info call Phil Sluder at (619) 460-0216

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