Tools/Options Settings

SW2005

There are two levels of settings: System Options and Document Properties

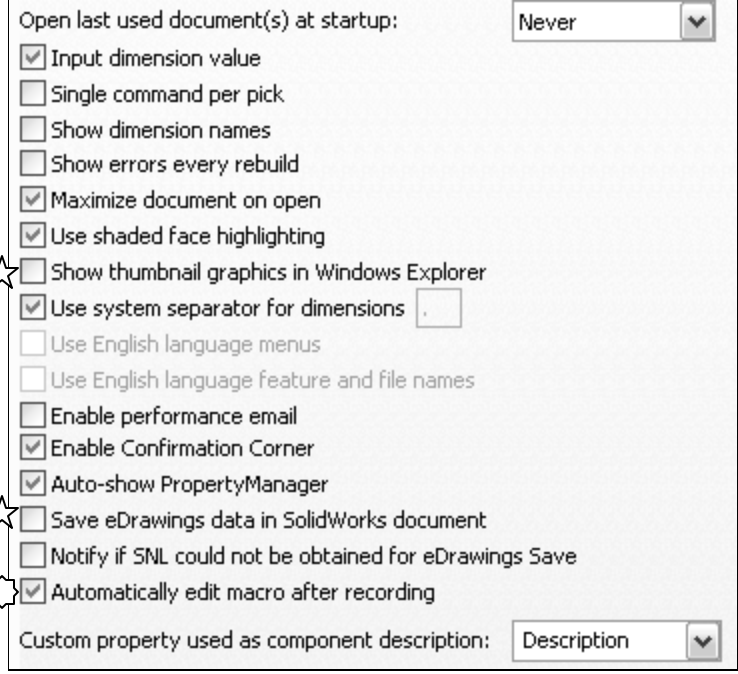
System Options apply to the local computer regardless of the settings the file was created with. System Options are stored in the registry of the OS/user.

Document Properties travel with the file from one system or user to another. Document Properties are set in the Document Template.

A red star (☆) is put next to settings that can affect speed or file size.

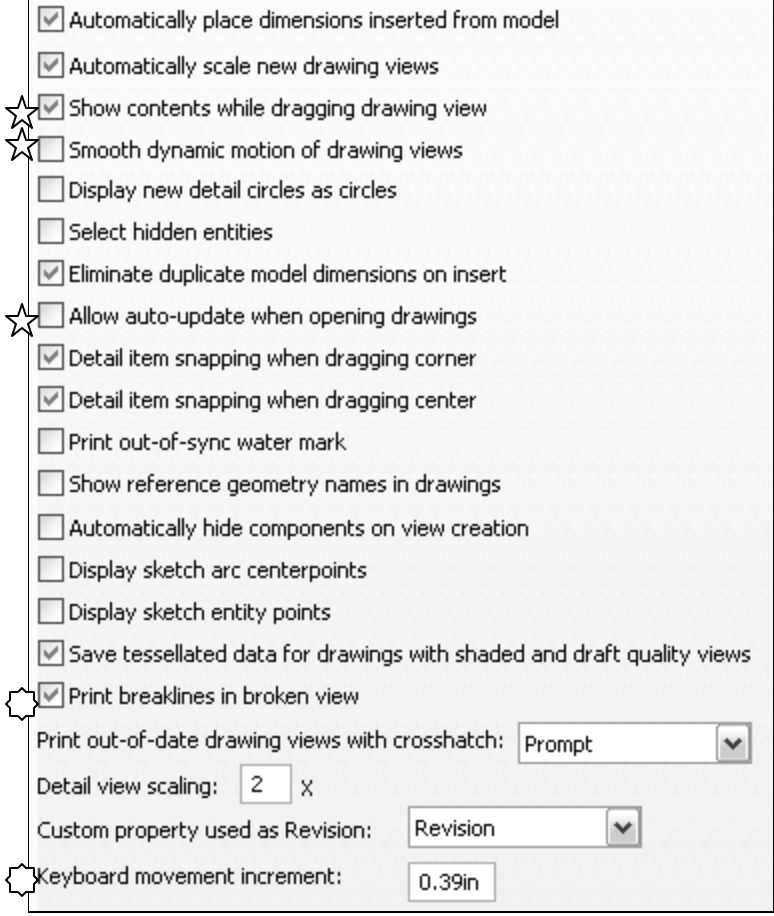
A green star (⯎) is put next to settings that are new in SW05.

System Options: General



- **Input dimension value** - Edit Dimension box opens when you create a new dimension
- **Single command per pick** - tools turn off after one use. To get tools to stay on until turned explicitly off, double click icon (override does not work on RMB)
- **Show dimension names** - puts dim name in parentheses next to dimension
- **Show errors every rebuild** - Error dialogs pop up every time model with errors is rebuilt
- **Maximize document on open** - new or opened document windows are maximized within the SW application window
- **Use shaded face highlighting** - when model is shaded and a face is selected, the whole face turns green, if this is off, only the edges turn green
- ☆ **Show thumbnail graphics in Windows Explorer** - replaces the SW part and assembly icons with previews of the doc. This is a big Windows performance hog when turned on, also previews are hard to see unless icons are set to large
- **Use system separator for dimensions** - for European style numbers (1.000,00 v 1,000.00)
- **Use English language** - to use SW in English when installed on a foreign language OS
- **Use English language feature and file names** – Forces non-english systems to use English default feature and file names
- **Enable performance email** - automatically sends emails to SW which contain information about hardware and crashes
- **Enable confirmation corner** - toggle for the triangle in the upper right of the graphics window with the check or "X"
- **Auto show PropertyManager** - shows prop mgr when things like sketch entities are selected
- ☆ **Save eDrawings data in SolidWorks document** - eDw will be able to view SW parts if they are saved with this setting turned on. Adds to file size.
- **Notify if SNL could not be obtained for eDrawings Save** – gives a warning if eDw network license cannot be obtained
- **Always show full shortcut menus** – Has been removed from SW04. Menus now controlled via Customize Menu. There is also a new section of the Tools, Customize menu – a tab called Options, which allows Show All or Default Settings for Pulldown and RMB menus
- ⬡ **Automatically edit macro after recording -** after recording a macro, SolidWorks automatically kicks you into VBA editor
- **Custom property used as component description** – controls BOM Description field with the selected custom property

System Options : Drawings

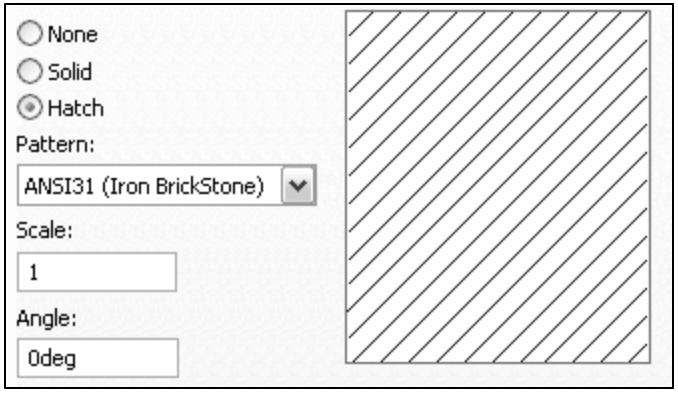


- **Automatic placement of inserted dimensions from model** - organizes dimensions for improved visibility when Insert Model Items is used
- **Display drawing view borders** – removed in 2005, borders no longer shown
- **Automatic scaling of new drawing views** – used to be “of 3view drawings” – sets the sheet scale so that new drawing views fit proportionally
- **Show contents while dragging drawing view** - toggles on/off the display of view geometry while view is dragged. Nice for aligning things, but big performance hit
- **Smooth dynamic motion of drawing view** – during pan and zoom
- **Dynamic drawing view activation** – removed in 2005, new activation functions
- **Select hidden entities** - allows selection of edges you have hidden
- **Display new detail circles as circles** - detail views may be created with non-circular sketches. If this is on, the shape of the sketch will be disregarded and the detail view will be bounded by a circle rather than an ellipse or rectangle, for example.
- **Eliminate duplicate model dimensions on insert** - if two dimensions call out the same geometry, only one is shown.
- **Allow auto-update**– automatically rebuilds drawing views on open
- **Open existing drawings with automatic view update off** - overrides Automatic View Update setting for pre-existing docs.
- **Detail item snapping when dragging corner/center** – alignment aid.
- **Print out-of-sync water mark in RapidDraft** - when a RapidDraft drawing is not up to date, a water mark will print.
- **Show reference geometry names in drawing** - when planes or axes are shown, names will also show
- **Automatic hiding of components on view creation** - parts hidden in the assy are listed in the view hidden component list
- **Display sketch points** – controls sketch display for drawings separately from model sketches
- **Save tessellated data for drawings with shaded and draft quality views –** file size for drawings with shaded and draft quality views will go up if this is checked, down if it is cleared. For a more detailed explanation, see SW Help.
- **Print breaklines in broken views –** in early versions of 2005, this is off by default, prevents break lines from printing.
- **Print out of date drawing views with crosshatch** - indicates drawing may not match recent changes to the model
- **Detail view scaling** - detail views are automatically created at this scale relative to the parent view.
- **Custom Property used as Revision** – for use with PDMWorks
- **Keyboard movement increment** – you can nudge views with the arrow keys by this amount

System Options : Drawings Display Style

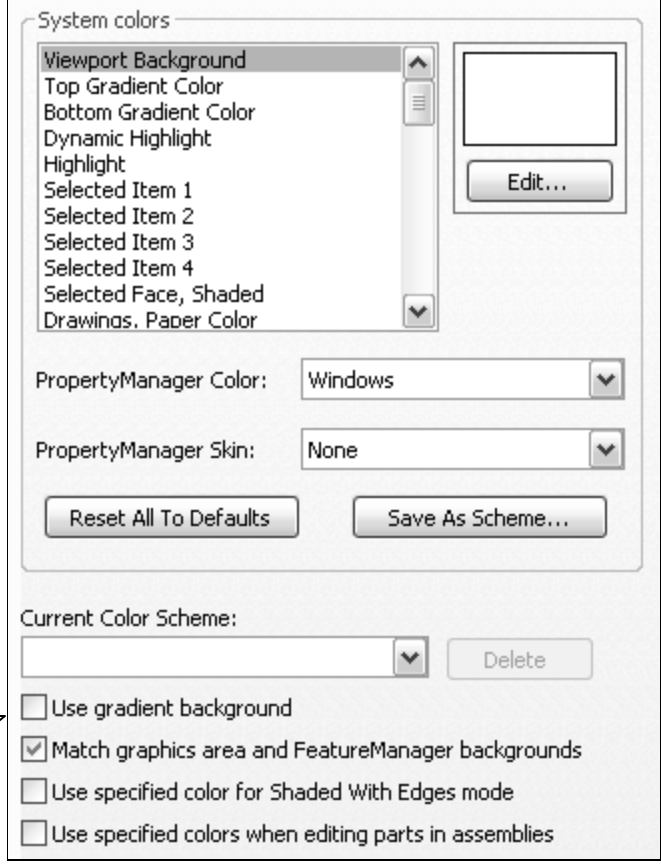
- **Display Style for new views** – "Shaded with edges" is a combination of Shaded and Hidden Lines Removed. Fastest is Shaded.
- **Display quality for new views** – see Help for limitations of Draft quality views.
- **Tangent edges in new views** – the "Removed" setting is not recommended, parts with lots of fillets tend to only show the part silhouette

System Options : Drawings Area Hatch



- **Pattern** - allows you to specify the default pattern, scale and angle for crosshatching and area hatch.
- **Solid** – new hatch option in SW03

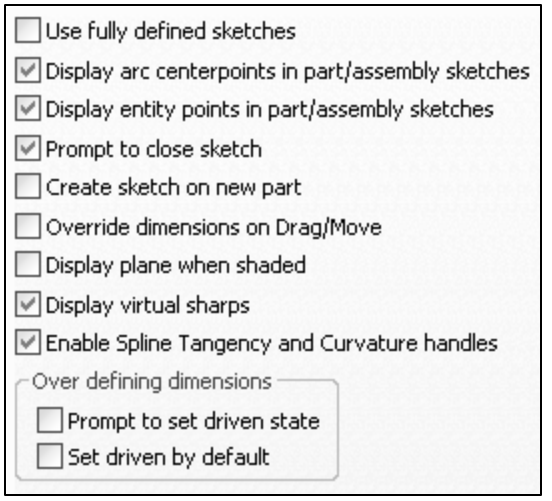
System Options : Color



- **System colors** - Set colors to suit you
  - **Issues to be aware of**
    - Some items that are black by default will not be visible on a black background
    - You can save and name a color scheme
    - ☆ A gradient background may slow your computer down if you have a poor graphics card
    - A "skin" is an image that can be applied to the feature/property manager. This can include special images for the buttons as well.
    - "Shaded with edges" display can have 3 states:
      - Off
      - Edges shown in color specified in System Colors
      - Edges shown as some variant (darker or lighter) of the part color specified in Document Properties

- **Use gradient background** – overrides Viewport Background color using Top and Bottom Gradient colors
- **Match graphics area and FeatureManager backgrounds –** matches colors including gradient
- **Use specified color for Shaded With Edges mode –** black edges sometimes look kind of "cartoony", I prefer to use the setting on Doc Properties, Colors
- **Use specified colors when editing parts in assemblies** – controls colors while editing parts in context
- **Go to Document Properties** – has been removed in 2005.

System Options : Sketch



•**Use fully defined sketches** - forces good discipline when sketching, but at the price of flexibility when rigor is not needed. This setting will not allow you to exit a sketch unless it is fully defined.

•**Display arc centerpoints** - puts a little "+" at the centers of sketch arcs, circles and ellipses.

•**Display entity points** - puts boxes at the endpoints of sketch entities, and displays asterisks (sketch points) at control points for splines, ellipses and parabolas (entity end points are smaller, almost invisible, on Win95, 98 and ME than on NT).

•**Infer from model** – removed in 2005, available in Sketch Settings menu

•**Prompt to close sketch** - will prompt you to close open profiles that can be closed automatically using model edges (for extrudes) or any open sketch for revolves by connecting open endpoints.

•**Create sketch on new part** - when you create a new part, SW will automatically open a sketch for you on the first plane. In the past, this has always created problems with Toolbox, and possibly other add ins.

•**Override dims on drag/move** - dimensioned entities can be dragged and the dimensions will change to the dragged positions.

•**Automatic relations** – removed in 2005, available in Sketch Settings menu, and Relations/Snaps page.

•**Display plane when shaded** - sketch plane appears shaded/translucent when editing sketch. Display properties follow doc props for transparency. SW help incorrectly ties shaded display to the grid being shown.

•**Display virtual sharps** - this toggles the virtual sharp display according to the settings in Doc Props\Detailing\Virtual Sharps.

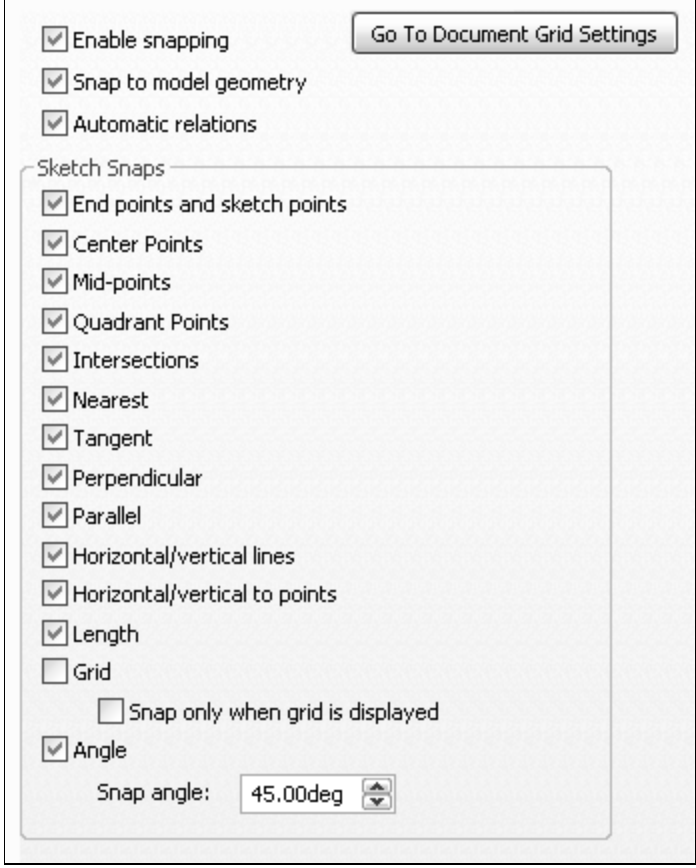
•**Enable spline tangency and curvature handles –** toggles the sometimes confusing or overwhelming display of spline control entities

•**Prompt to set driven state** - pops up a dialog when sketch is overdefined to ask if you want overdefining dimension to be driven or driving. If this is off, dim will be left overdefining and driving.

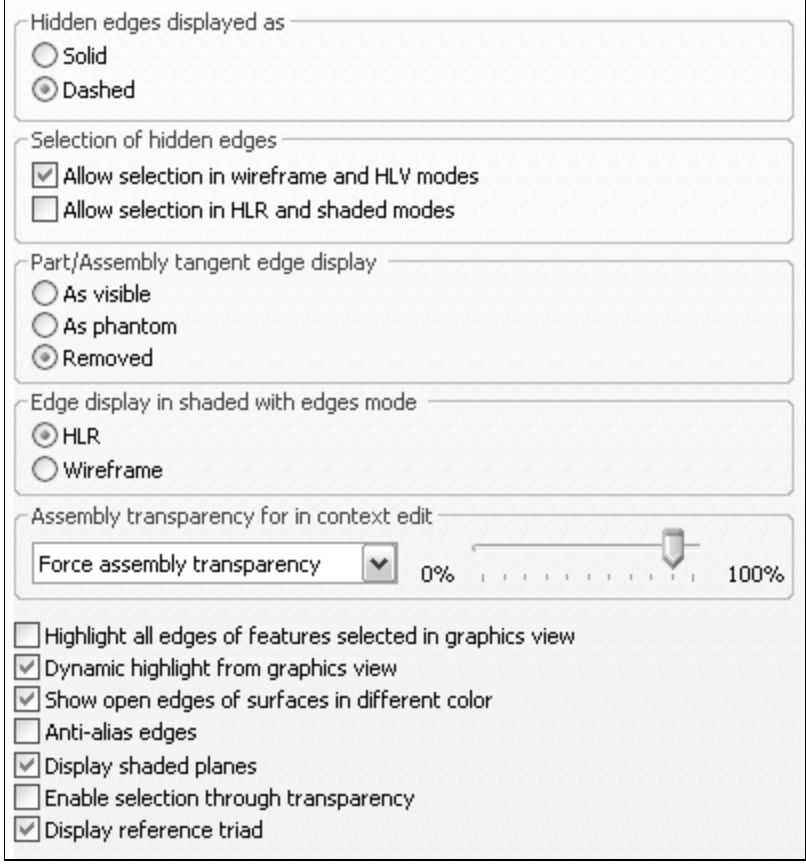
•**Set driven by default** - automatically sets overdefining dimensions to driven (reference dim).

System Options : Sketch, Relations/Snaps

- **This entire page is new for SolidWorks 2005**
- **Enable snapping –** toggles the snap function to the entities checked in the box below in sketch mode
- **Snap to model geometry** – allows sketch creation to snap to the snaps on edges instead of just snaps on other sketch geometry.
- **Automatic relations –** enables relations such as horizontal, vertical, tangent, etc. to be automatically applied
- **Sketch Snaps** – these are the types of entities the sketcher will allow to snap to. Some entities may be “woken up” by getting the cursor close to them.
- **Quick Snaps** – can be used on the fly to temporarily override the Tools, Options settings. Quick Snaps is available from the Quick Snaps toolbar or the Tools, Relations menu



System Options : Display\Selection



•**Hidden edges displayed as** - sets display for hidden edges in parts and assemblies.

•**Allow selection in wireframe and HLV modes** - hidden edges can be selected when they are visible.

•**Allow selection in HLR and shaded modes** - hidden edges can be selected when they are not visible.

•**Part/Assembly tangent edge display** - this is the equivalent setting of the previous one for drawings.

☆ •**Edges displayed in shaded mode** - this mixes shaded and wireframe display modes. It often makes small features more visible. There is a performance hit to use HLR and shaded together.

☆ •**Assembly transparency for in context edit** - The options are Opaque, Force transparency and Maintain transparency. The intent seems to be to give the user control of display options when editing in context.

•**Highlight all edges of features selected in graphics view** - in contrast to only highlighting the selected face of a feature.

•**Dynamic highlight from graphics view -** faces, edges and vertices highlight by running the cursor over them.

☆ •**High quality display of interfering bodies in HLR/HLG** – removed from SW03 – now this is turned on by default

•**Show open edges of surfaces in different color** - nice troubleshooting tool when trying to knit surfaces into a solid.

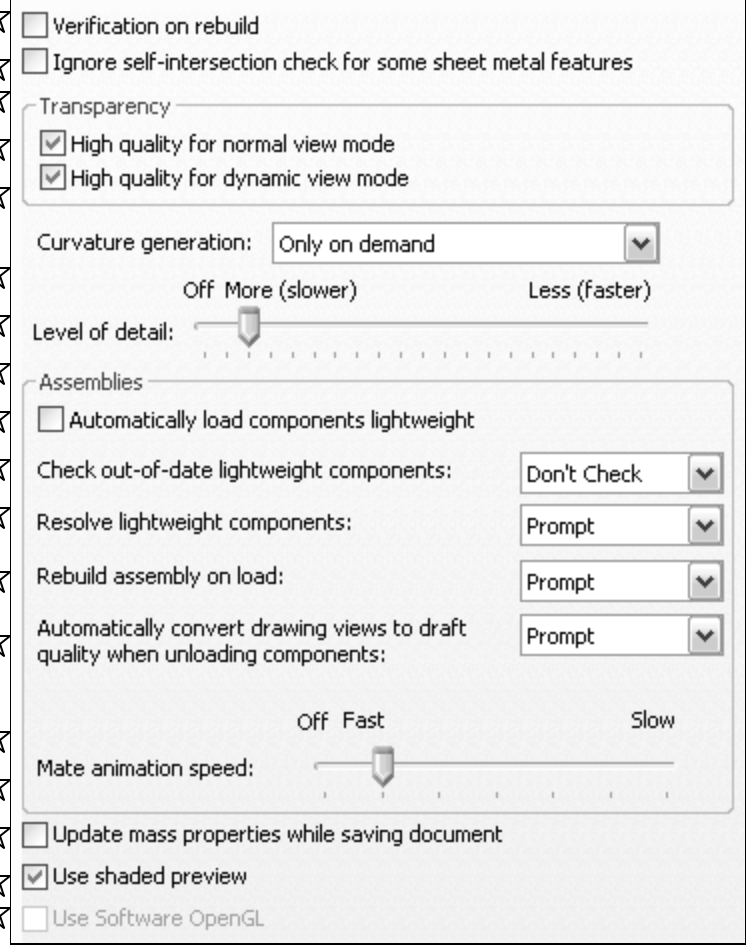
☆ •**Anti-alias HLR edges in shaded and fast HLR/HLG modes** - smoothes jagged display of edges when not rotating.

☆ •**Display shaded planes** - new for SW01+. Planes can be shown shaded, with different colored sides (like Pro/E).

•**Enable selection through transparency** – transparent parts are invisible to the cursor, use SHIFT select to over ride

•**Display reference triad** – a coordinate system indicator shows in the lower left of the screen

System Options : Performance



- **Verification on rebuild** - This is a more thorough error checking routine for problem parts. Use it sparingly, it is a big performance hog.
- **Ignore self-intersection …for sheet metal** – when checked will not return an error when sheet metal flat pattern flanges share an edge.
- **High quality for normal view mode** - low quality is called “screen door”. This is very video card dependent.
- **High quality for dynamic view mode** - this makes transparency show high quality when rotating the model. Very big performance hit.
- **Curvature generation** - this is a specialized tool that colors faces of the model according to the curvature. Useful to see smoothness of a face.
- **Level of detail** – controls the size of features which are left out or simplified during dynamic view rotation. Replaces Remove Detail…
- **Automatically load parts lightweight** - display data only is loaded for parts in an assembly. See Help for limitations.
- **Remove detail during zoom/pan/rotate** – replaced by Level of detail slider
- **Check out of date lightweight parts** - because only display data is loaded, lightweight part display can be out of date with the actual model.
- **Resolve lightweight parts** - when more than just display data needs to be loaded, this sets prompt or no prompt.
- **Rebuild assembly on load** - important when there have been changes to mated parts or incontext features.
- **Automatically convert drawing views to draft quality when unloading components** – decreases quality setting when views are set lightweight
- **Mate animation speed** - how fast should the part move from unmated to mated positions.
- **Enable clipping … for Win 95/98/ME** – removed in 2005, no longer compatible with those Os’s.
- **Update mass properties while saving document** - this takes extra time, but if you are using the mass properties in a BOM, this is a useful setting. Ordinarily, SW does not calculate the mass properties unless asked.
- **Use shaded preview** - this refers to the blue preview before performing extrude, revolve, etc. If not set, you still get a brown wireframe preview.
- **Use software OpenGL** - disables video card hardware acceleration. Depending on your video card, it may be grayed out and checked by default. If you have the option, this should be unchecked, except for video trouble shooting.

System Options : Assemblies

•**Move components by dragging** – allows moving components in assembly without using the Move tool

System Options : Large Assembly Mode

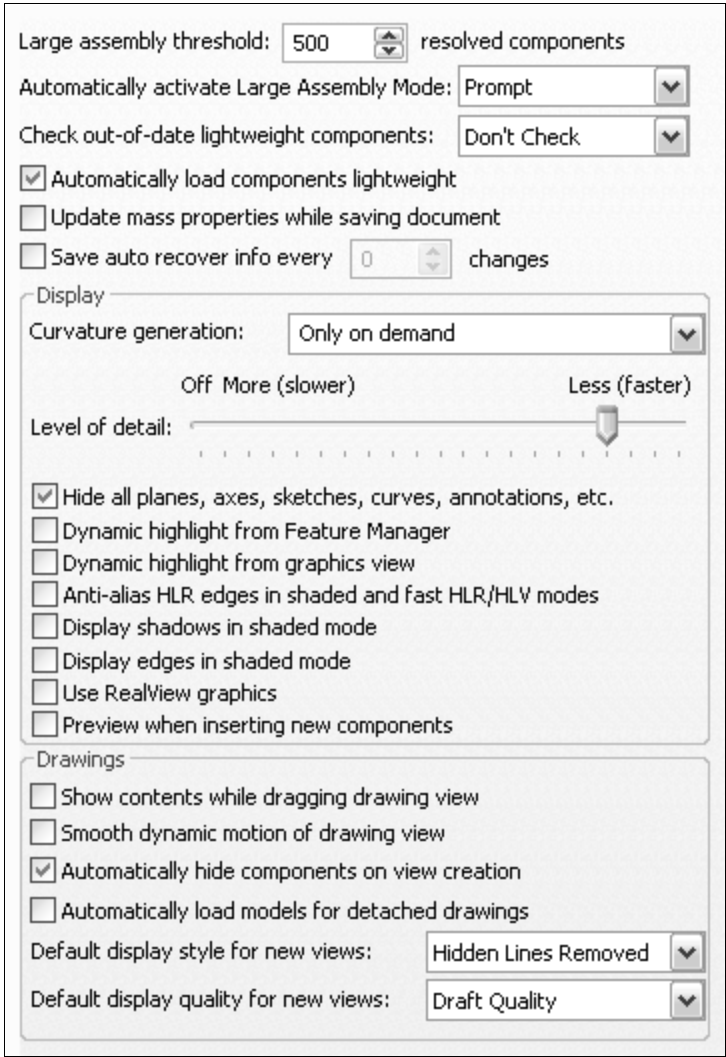
•**Large assembly threshold** - different settings take affect after this number of parts is reached in the assembly.

•**Automatically activate Large Assy Mode** - this switch disables Large Assembly Mode if you don't want to use it.

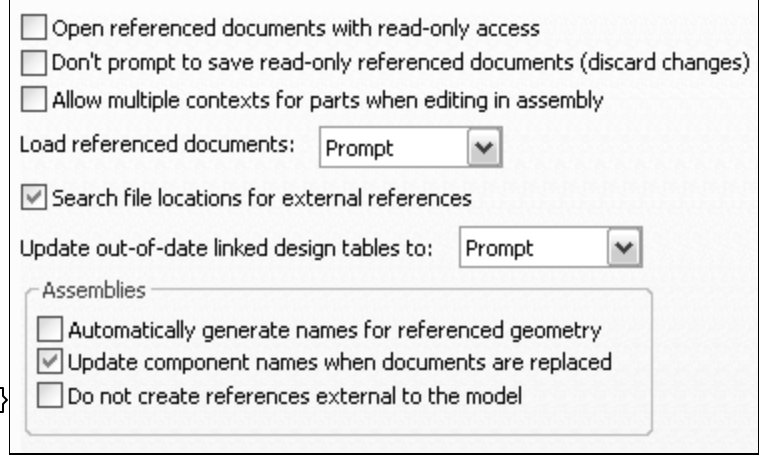
•The rest of the settings are the same as those on the Performance and Drawings pages except for…

•**RapidDraft** has been renamed "detached drawings" in SW 2004

•**Detached drawings** - a way to disassociate a drawing from the model for some operations. You can make a drawing a detached drawing by using "Save as type, Detached drawing" in the Save dialog. To change back to a regular drawing, do another Save As.



System Options : External References

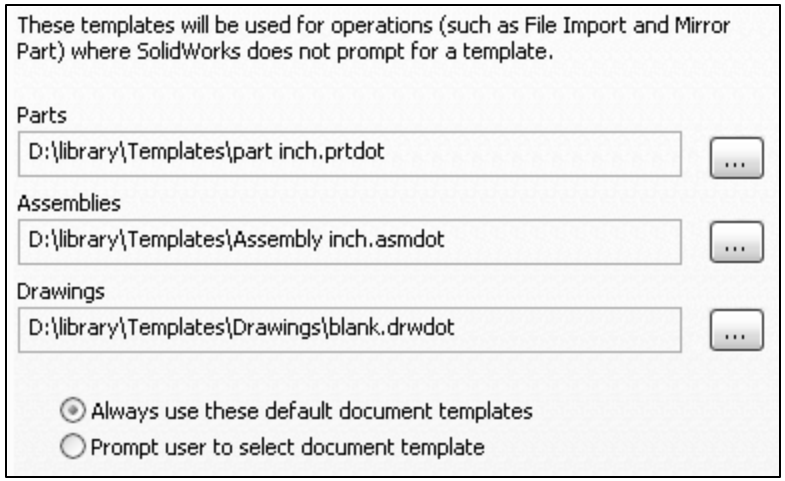


- **Open referenced documents with read-only access** - this may be an important setting in an environment where many people are sharing the same files. If one user opens an assembly, he gets write access to the assembly, and read only on all the parts. If someone else wants to work on a part in the assembly, they can get write access to it by opening it through the "Open" dialog.
- **Don't prompt to save read-only referenced documents** - if you save a large assembly with all/some of the parts loaded read only, with this checked, you will not have to cancel the "Save As" dialog that comes up when you try to save read-only parts.
- **Allow multiple contexts for parts when editing in assembly** - this is off by default. This switch toggles the ability for a part to have in context references from more than one assembly. This means that all assemblies where the in context relationships were created must be open to fully rebuild the part.
- **Load referenced documents** - referenced documents might include base parts, mirror parts, etc. If the child is opened, the parent will also be opened automatically.
- **Search file locations for external references** - when an assembly is opened, SW has to find the parts somewhere. It does this by a set of rules listed in Help. You can help it along by using this switch in conjunction with Referenced Documents setting on the File Locations page.
- **Update out-of-date linked design tables** – when there is a difference between the part and the DT, the DT is updated
- **Automatically generate names for referenced geometry** - this should only be switched on if you used named faces for part replacement. If it is on, you need write access to a part to mate it in an assembly.
- **Update component names when documents are replaced** - when this option is on, the Feature Manager name will change if a component is replaced. When it is off, you can use Component Properties to rename the part in the feature manager to be something other than the file name.
- **Do not create references external to the model** – prevents in-context type relations

System Options : Default Templates

•**Default templates** - you can point these defaults to a central network location

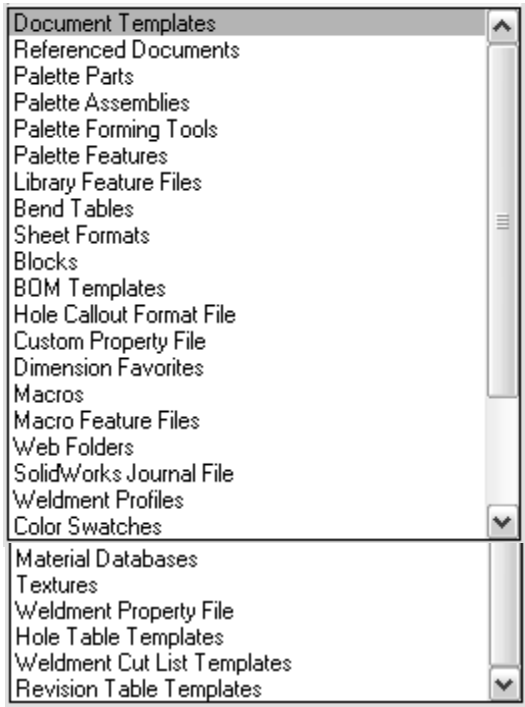
•**Always use these default document templates/Prompt user to select document template** - this applies to base parts, mirrored parts, incontext, translations, etc. with the exception of dwg/dxf.



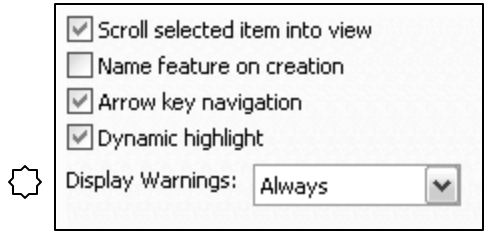
System Options : File Locations

•Here you specify folders for each of the listed data types. These can be network locations which are shared by multiple users, and can have more than one entry per data type. For example, Document Templates could be directed to both a network drive and a local drive.

•**Move Up/Down** - allows you to specify the relative priority if there are multiple drive locations listed for a given data type. For example, it there are two locations listed for Palette Parts, the first path on the list will be shown first in the Feature Palette window.



System Options : Feature Manager



•**Scroll selected item into view** - when a feature is selected from the graphics window, the Feature Manager will scroll to that feature.

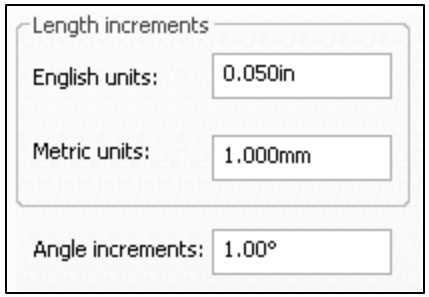
•**Name feature on creation** - automatically puts you into name edit mode for a feature as soon as it is created.

•**Arrow key navigation** - after the rollback bar has been dragged, the arrow keys switch to moving the bar up and down until you click in the graphics window again.

•**Dynamic highlight** - when on, features in the graphics window will highlight when the mouse is over the feature in the Feature Manager.

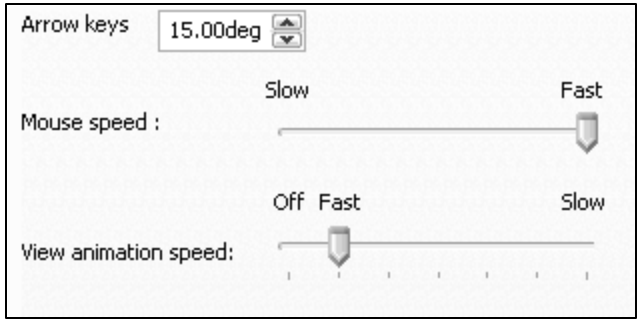
•**Display warnings** – this prevents the display of feature manager warnings (or), but does not apply to errors, which are always displayed.

System Options : Spin Box Increments



•Here you set the default values for the Modify spin box. Alternate values can be saved in the document template.

System Options : View Rotation

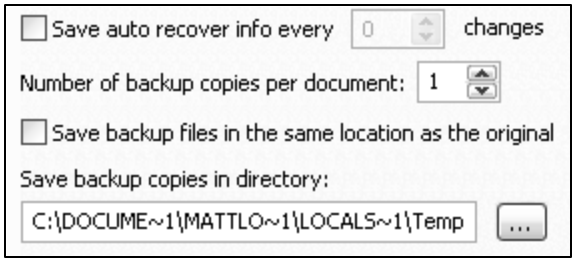


•**Arrow keys** - for each arrow keystroke, the view will rotate by this amount

•**Mouse speed** - speed at which mouse rotates the view

•**View animation speed** - when switching between standard views, “normal to” or shift-arrow, this controls the transition speed, or turns it off altogether

System Options : Backups



•**Save autorecover info every XXX changes** - every so many changes (defined as changes requiring rebuilds), SW saves data to your system temp directory (C:\Temp\swxauto). If SW terminates abnormally (crash or task manger), SW will automatically open these autorecovery files. Sometimes they open with no geometry showing, particularly when part of an assembly. You must open the individual files and ctrl-Q them to rebuild. This option is not compatible with all plug ins (such as Linius). ***This function is not the same as “back up”.***

•**Number of backup copies per document** - A “backup” is different from an “autorecovery” file. Backups are created when you actually hit “Save”, it moves the old version of the file to your backup directory (C:\Windows\TempSWBackup- Directory for example). This is not “autosave”.

•**Save backup files in the same location as the original -** saves backup parts with the original, with the prefix “Backup of…” on the file.

•**Save backup copies in directory -** specifies where to put the backups if the above switch is not used

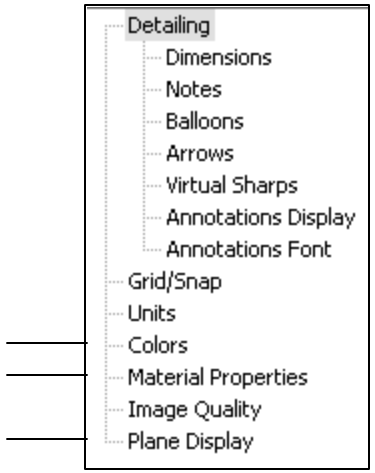
System Options : Data Options

•**Edit Standards Data –** The Toolbox editable standards have moved to this location

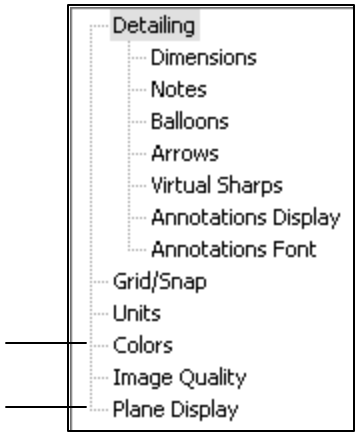
Document Properties : Type-specific properties

•Properties indicated by the red arrow are not universal to all types of documents. For example, you will have to have a drawing active when you start the Tools, Options dialog to see the Tables properties.

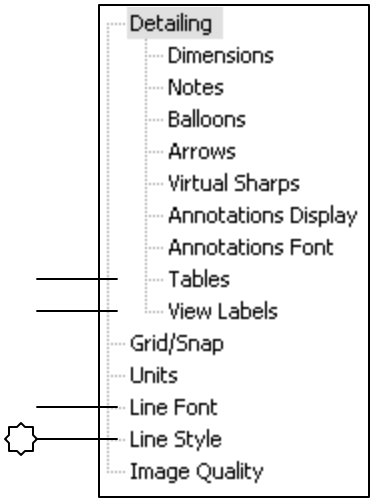
•PART



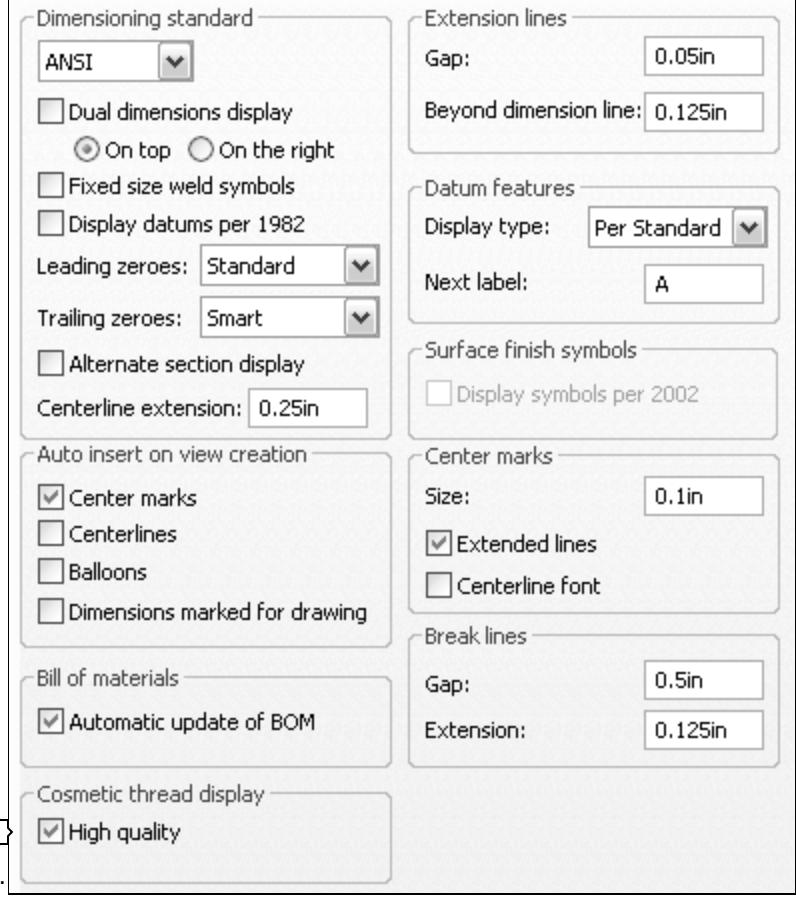
•ASSEMBLY



•DRAWING



Document Properties : Detailing



•**Dimensioning Standard** - changes display of dimensions per standard. Also affects datum display, weld symbols, etc.

•**Dual dimensions display** - the alternate dimension is always shown in square brackets and can only be placed above or to the right of the standard unit dimension.

•**Fixed size weld symbols** - toggles between control of the size of the symbol being taken from the drawing standard or the document font.

•**Display datums per 1982** - datums in box vs triangular attachment point.

•**Trailing zeroes** - Smart - trailing zeroes are trimmed for whole metric values (Conforms to ANSI and ISO standards) (SW Help) Remove and Show seem self explanatory

•**Alternate section display** - instead of a section line showing across the entire parent view, only the arrows and corners are shown.

•**Centerline extension** - when section plane is coincident with a temporary axis, the axis is shown extended beyond the model edges by this amount. Only applies to new section views (does not change centerlines shown in existing views).

•**Center marks** - controls size of centermarks. Applies to new and existing marks.

•**Show lines** - lines extend past the circular edge radially by 2 times the center mark size.

•**Witness line Gap** - gap between the witness line and the model. **Extension** - extension of witness line past dimension line.

•**Next datum feature label** - allows you to reset the internal counter keeping track of how many datums you have used.

•**Datum display type** - choices are Per Standard, Square and Round.

•**Break line gap** - gap of break in a broken view.

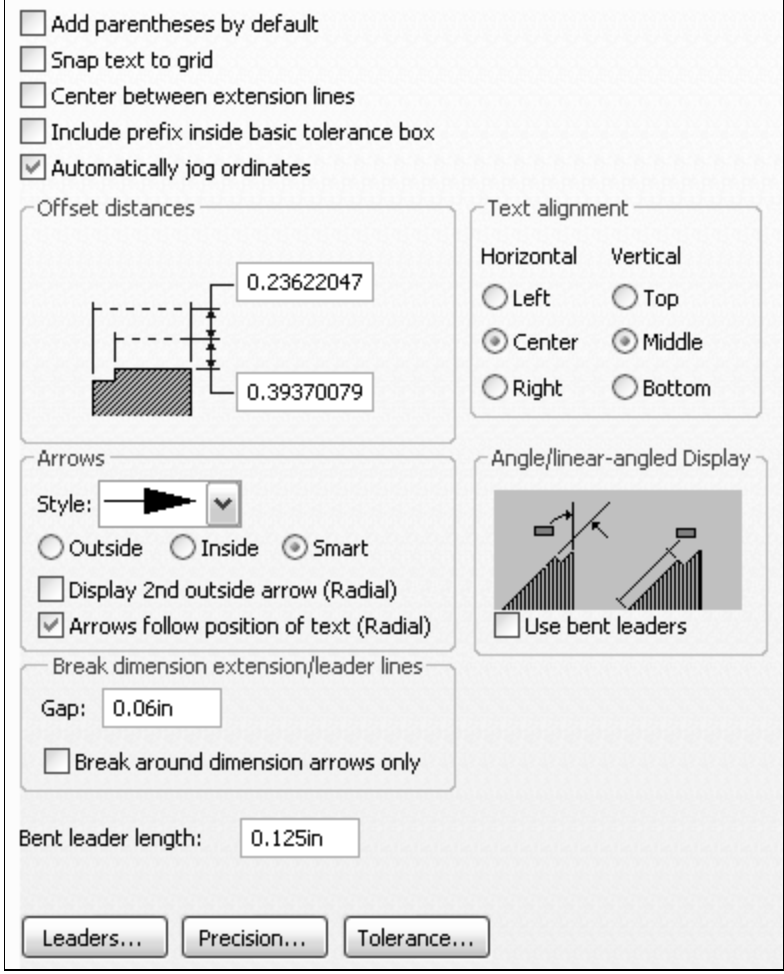
•**Detail/Section/View Arrow Fonts** – New in SW03, removed to new Annotations Font page

•**Automatic update of BOM** – New in SW03, moved from System Options, automatically updates the BOM on rebuild/open

•**Surface finish symbols** – Display symbols per 2002 activates for the following dimensioning standards: ISO, DIN, BSI, GOST, GB, to reflect recent revisions to those dimensioning standards

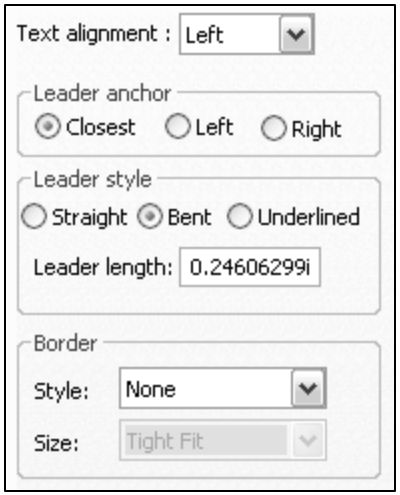
•**Cosmetic thread display** -

Document Properties : Detailing : Dimensions



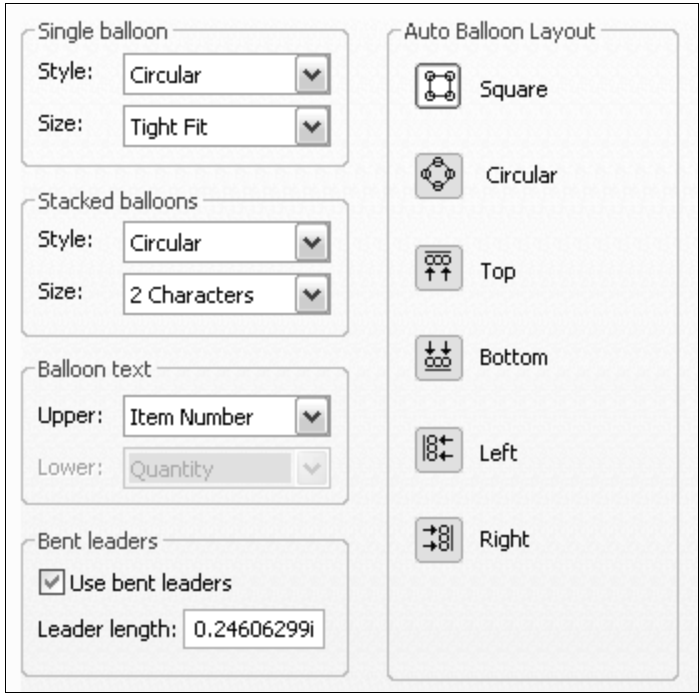
- **Add parentheses by default** - applies to reference dimensions only.
- **Snap text to grid** - use grid spacing settings to help align text on drawing.
- **Center between witness lines** - centers dimension text between witness lines.
- **Offset distances** - depicted graphically.
- **Arrow style** - select from various arrow styles
- **Outside/Inside/Smart** - where do the arrows go relative to witness lines
- **Display 2nd outside arrow (Radial)** - displays 2 arrows on diameter dims. SW Help incorrectly states “Radial Dimensions”.
- **Arrows follow position of text (Radial)** - arrows display on the same side of the arc as the text (inside, outside)
- **Break Dimension Witness/Leader Lines** - New for SW01+. When witness and dimension lines cross, this setting allows a break in one of the lines.
- **Break around dimension arrows only** - only break around arrows
- **Bent leader length** - this controls the bent leader length for dimensions. Bent leader length for notes is controlled elsewhere, as is the toggle that allows bent leaders in the first place.
- **Font** – New for SW03, removed to Annotations Font page
- **Leaders** - override the drawing standard display settings for dimension leaders.
- **Precision** - control the number of decimal places shown for Primary, Alternate and Angular Unit dimensions and tolerances.
- **Tolerance** - control the default tolerance value.
- **Text Alignment** – Help calls this “justification” rather than alignment. These are the default justification settings for multiline notes or dimensions with additional text.

Document Properties : Detailing : Notes



- **Font** – Moved to Annotations Font page
- **Text alignment** - text justification within the note
- **Leader anchor** - “closest” setting automatically switches leader to right or left depending on which is closest to the arrow head.
- **Bent leaders** - enables bent leaders and controls their length for notes only.
- **Border** - applies a balloon of different shapes/sizes around the note

Document Properties : Detailing : Balloons



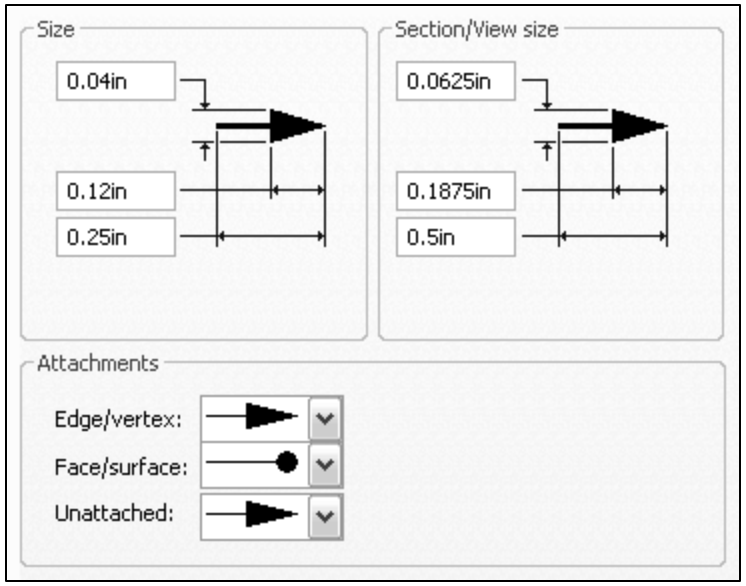
- **Single/Stacked ballons** - controls the default balloon style and size for balloons.
- **Balloon text** - For circular split line balloons, there are upper and lower values, for all others, only the upper applies. The choices are “Custom” which really means “manual”, “Item Number” which refers to the BOM item number, and “Quantity”, which again refers to the BOM.
- **Bent leaders** - enables bent leaders and controls their length for balloons only.
- **Auto Balloon Layout** – establishes the default layout for auto ballooning

Document Properties : Detailing : Arrows

•**Size** – the significance of the dimensions has changed since SW03

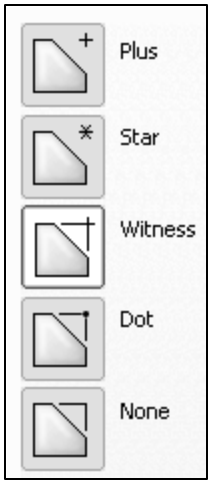
•**Section/View size** - size of view arrows.

•**Attachments** - assign different types of arrows to edges, verteces, faces and unattached arrows.

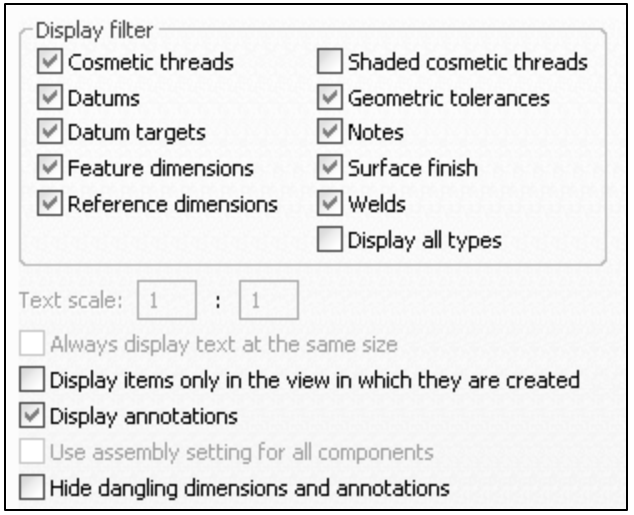


Document Properties : Detailing : Virtual Sharps

•Set how you would like virtual sharps to be displayed. Whether they are displayed or not is a System Option.



Document Properties : Detailing : Annotations Display



•This page will display differently depending on if you are editing the properties of a part, assembly or a drawing.

•**Display filter** - turn on the entity types that you would like to display on the drawing.

•**Text scale** - for drawings and if **Always display text at the same size** is checked, this box will be grayed out, other wise, this controls the size of dimension and note text for parts and assemblies.

•**Always display text at the same size** - text/dims remain the same size regardless of zoom state (not available in drawings)

•**Display items only in the view in which they are created** - this is dangerous if not used in a disciplined way. If you create an annotation after rotating the model with the mouse, you may never see the note again.

•**Display annotations** - displays all the annotation types that are selected in the Display filter box.

•**Use assembly's setting for all components** - in an assembly, this allows parts to display their annotations based on either the part settings or the assembly settings

•**Hide dangling dimensions and annotations** – if drawing dimensions or annotations go dangling, SW will automatically hide them.

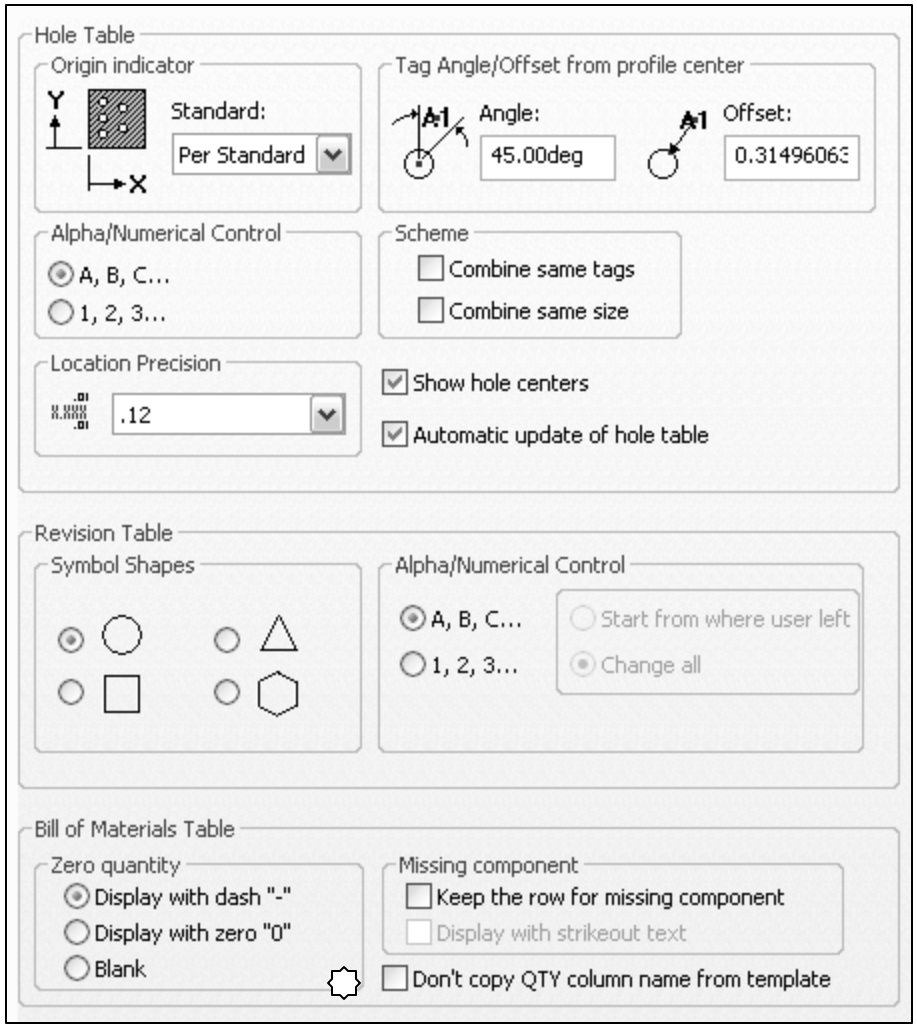
Document Properties : Detailing : Annotations Font

•This page consolidates all the various font settings.

•**Tables** and **Balloons** have been added to this page. Balloons were formerly combined with Notes.

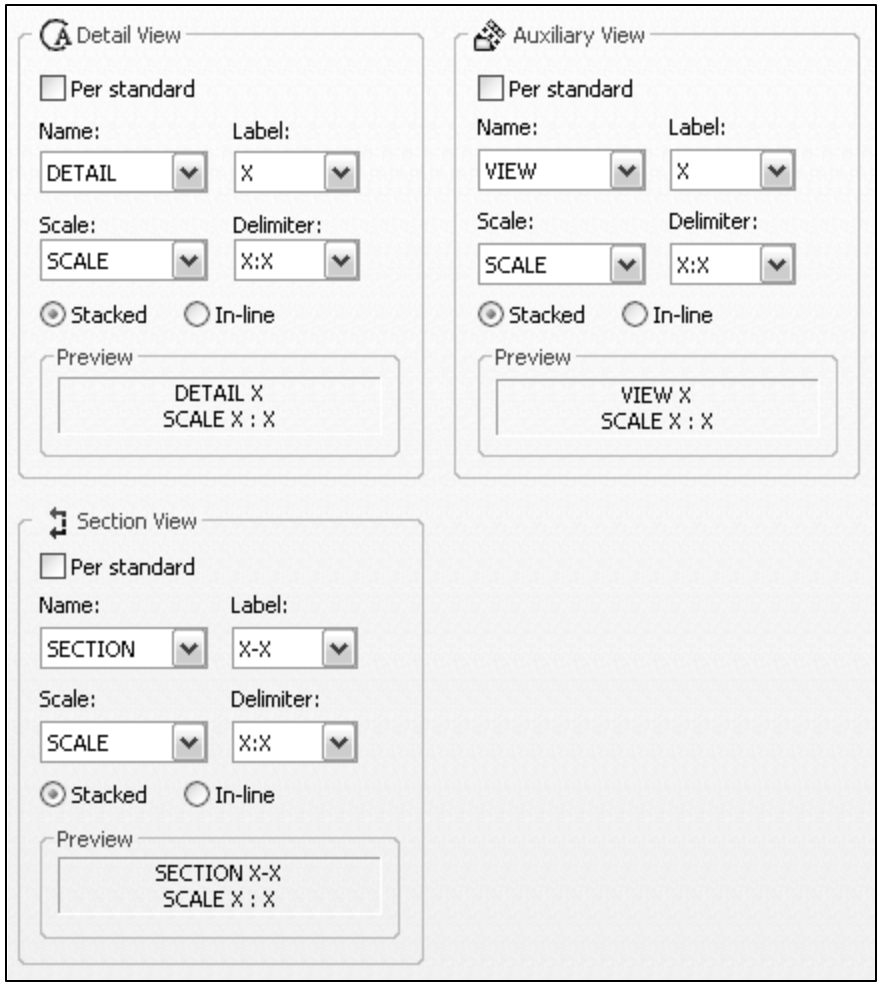
Document Properties : Tables

- **Origin indicator** – with or without zeroes
- **Tag angle/offset from profile center** – distance and angle from the vertical at which tag will be positioned. The diagrams are somewhat misleading. The "A1" shown would probably result from a 0 deg setting. The position of the angle is for the lower left of the tag, not the center as shown. The offset distance seems to be measured from the center, however.
- **Alpha/Numeric Control** – either letters or numbers can tag the holes.
- **Scheme** – identical tags are grouped together in the chart, and no X, Y data is given
- **Revision Table Symbol Shapes** – notes mark the section of the drawing where the revision note applies, and the notes are put inside a shape as specified here. Other shapes available in property manager
- **Alpha/Numerical control** – rev level can be letters or numbers, and will either restart the revisions or combine schemes. Starting in 2005, this works nicely with PDMWorks
- **BOM Zero quantity** – zero qty may happen by deleting or suppressing components
- **Don't copy QTY column name from template** – puts config name in BOM QTY column header

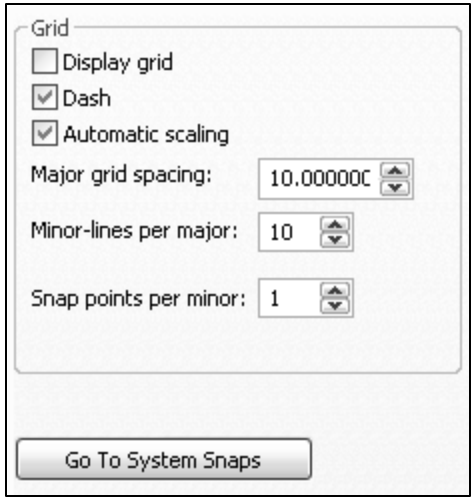


Document Properties : View Labels

•**Detail, Auxiliary and Section Views** – follows dimensioning standard set in Document Properties, or specify your own text to automatically accompany these types of views



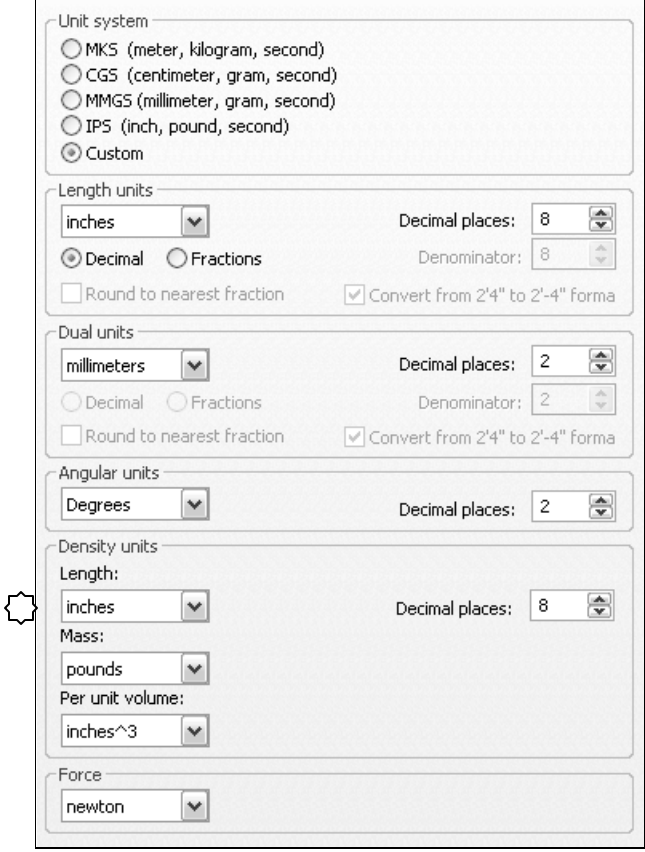
Document Properties : Grid



- **Display grid** - turns the grid on in sketch mode for parts and assemblies and all the time on drawings
- **Dash** - minor grid lines are dashed lines
- **Automatic scaling** - when on, number of minor grid lines decreases as you zoom out. If off, the grid disappears at some zoom factor.
- **Major grid spacing** - distance between major divisions of grid
- **Minor lines per major** - how many divisions between major grid lines
- **Snap points per minor** - snap positions per minor grid spacing
- **Snap –** removed in 2005. Most of these settings are now in the Tools, Options, Sketch, Relations/Snap area.

Document Properties : Units

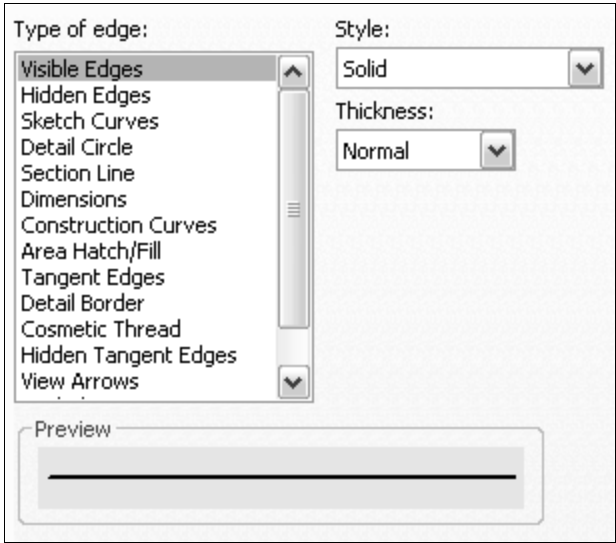
•**Unit system** – select a standard unit system or Custom to mix and match units



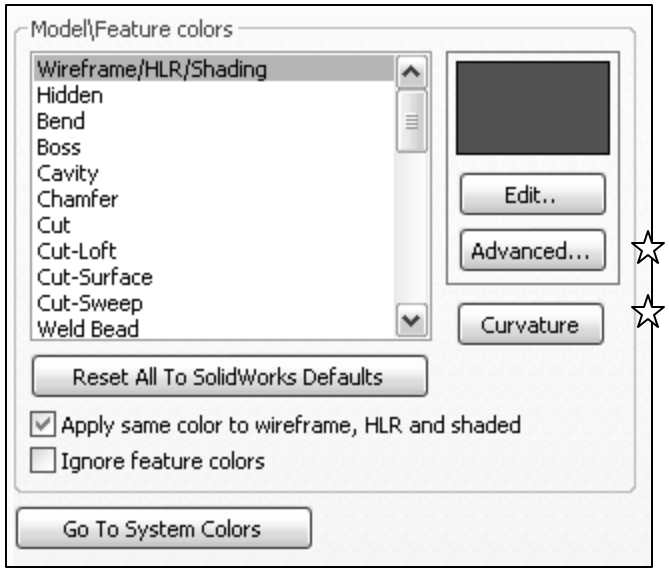
Document Properties : Line Font

•For drawings only

•**Type of edge** – set line font and thickness for selected type of edge



Document Properties : Colors



•Most of these options are only available for part documents.

•**Model/Feature colors** - control overall colors for the model, and for specific types of features

☆ •**Advanced** - allows you to assign transparency and other visual effects

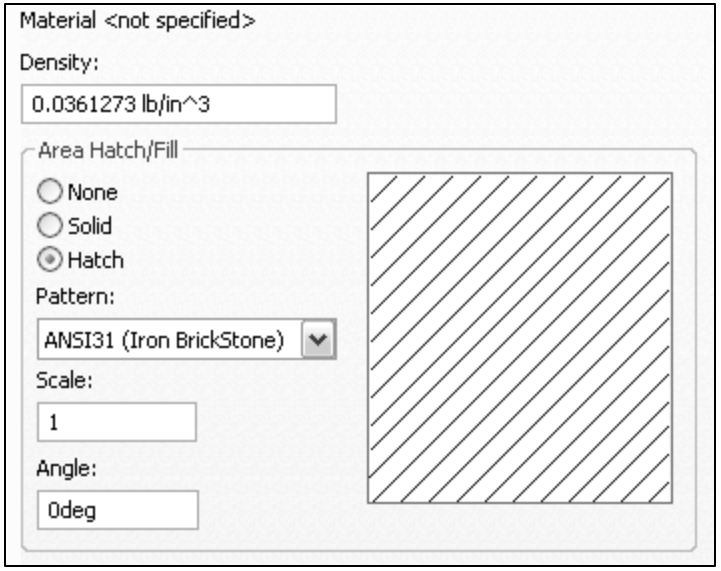
☆ •**Curvature** - invokes shading the model by curvature

•**Reset all to solidworks defaults** - gets you out of trouble if things get messy

•**Apply same color to wireframe, HLE and shaded** - this setting makes the wireframe and shaded colors the same. This is most important in wireframe display of assemblies. If your parts do not have this setting, all the parts will be black in wire frame, and indistinguishable from one another in the assembly.

•**Ignore feature colors** - override feature colors with part color

Document Properties : Material Properties



•This property is only available for part documents.

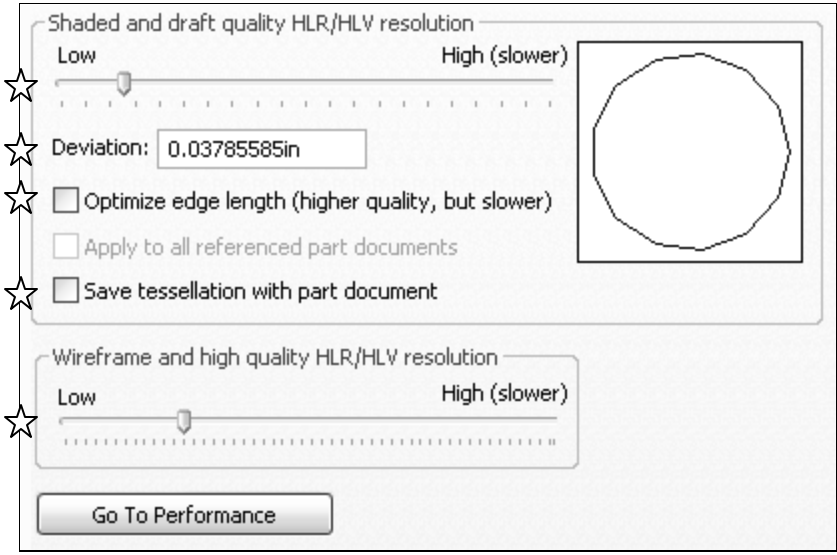
•Material (along with advanced graphics and Real View properties) can be specified in the Feature Manager in SW 2004, which overrides these settings

•**Density** - create different template files with different densities for different materials

•**Crosshatch** - assign different crosshatch patterns for different parts. New is the solid fill choice

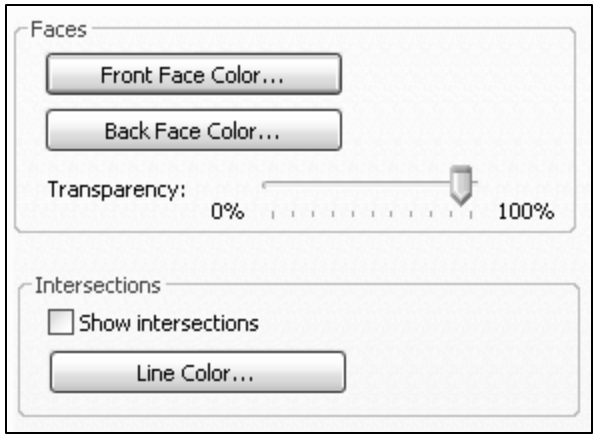
•**Scale / Angle** - control the density of the hatch and the angle.

Document Properties : Image Quality



- •**Deviation** - how far will you allow the facets to deviate from the actual geometry
- •**Apply to all referenced part documents** - available only in assembly documents. Assembly shaded image quality setting controls display for all parts displayed in the assembly.
- •**Save tessellation with part document** – turning this off saves space in the part document by not saving the display list. The eDrawing viewer is no longer able to view the file because the display list is not there. 2D thumbnails (preview bitmaps) are still stored, but not the 3D display data. This would seem to override the system property "Save eDrawing data in SolidWorks document". Saving file size means that the display data needs to be recalculated when the file is opened, so this setting should not be used to increase performance, but only for true space saving concerns or file transfer.
- •**Wireframe** - same settings as for shaded, but apply to wireframe. Use this if lines in wireframe display don't seem to touch. Wireframe display uses the CPU, and shaded mode leans more heavily on graphics card.

Document Properties : Plane Display



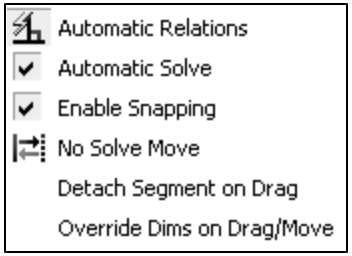
•These settings assume that you have Shaded plane display turned on in Tools/Options/Display Selection.

•**Front/Back face color** - The front and back of planes can have different colors to help you identify if you are looking at the front or back.

•**Transparency** - control transparency of shaded planes

•**Intersections** - intersections of planes can be shown as a dashed line.

Template Properties : Tools, Sketch Settings



•**Automatic Relations -** automatically adds relations like horizontal, vertical, parallel, etc. Also set on the Tools, Options, Sketch, Relations/Snap page

•**Automatic Solve -** sketches will not drag or update when dimensions are changed or relations are added if this is off. Mainly used when making a number of changes all at once without worrying about errors at intermediate stages.

•**Enable Snapping** – toggles automatic relations and snaps

•**No Solve Move** – enables moving of sketch geometry without solving relations. May give the option to keep or delete existing relations. Also seems to function like Detach Segment on Drag when enabled.

•**Detach Segment on Drag** – when lines have merged endpoints, they cannot be separated. Detach Segment on Drag allows you to “unmerge” endpoints by dragging a sketch entity away from other entities

•**Override Dims on Drag/Move** – when this is enabled, sketch geometry which is fully defined by dimensions can be dragged and the dimensions will update. This is kind of a frightening setting, since most of us put dimensions on specifically so the geometry will stay where it is put. This can be used in conjunction with the Move/Size Features toolbar button which allows you to drag sketch entities without being in sketch edit mode, most useful in testing concepts or tug-and-pull sketches.

Template Properties : View Menu

•These are global settings for the current document. For example, when Planes are turned on, individual planes may be either shown or hidden.

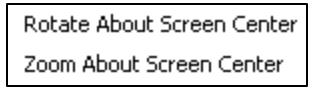
•**Hide All Types** – can be disabled by Large Assembly Mode. If you are in LAM, Hide all types is grayed out and turned on.

•**Hint** – make hotkeys to the various reference geometry and annotation types that you frequently want to show or hide. Notice the P, A, T, G hotkeys for toggling Planes, Axes, Temp Axes and Origins.

•**Sketch relations** – the sketch relations icons can be shown or hidden from here

•**Task Pane** – permanently hide the task pane from this menu

Template Properties : View, Modify Menu



•**Rotate about screen center** – on large parts where the center of the part is far off the screen, it is often clumsy to use the default SolidWorks view rotation scheme. You may want to try this setting which rotates about the center of the screen rather than the center of the part, although it may take some getting used to.

•**Zoom about screen center** – if you zoom with the scroll wheel, your part may walk off the screen if the cursor is not at the center of the screen. Some people prefer that the center of the zoom is at the center of the screen rather than at the cursor.

Template Properties : Not found in Tools/Options

- **Some settings that can be saved in document templates are not found in Tools Options:**
  - Link Values
  - "Thickness" link value for sheetmetal templates
  - Additional spin box increments
  - Default linetype/thickness/layers/color for drawings
  - Tools, Sketch Settings menu settings
  - Lights
  - Some (not all) of the View, Display menu settings
    - Excludes:
      - Perspective
      - Zebra Stripes
      - Curvature
      - Section View