

SolidWorks Options

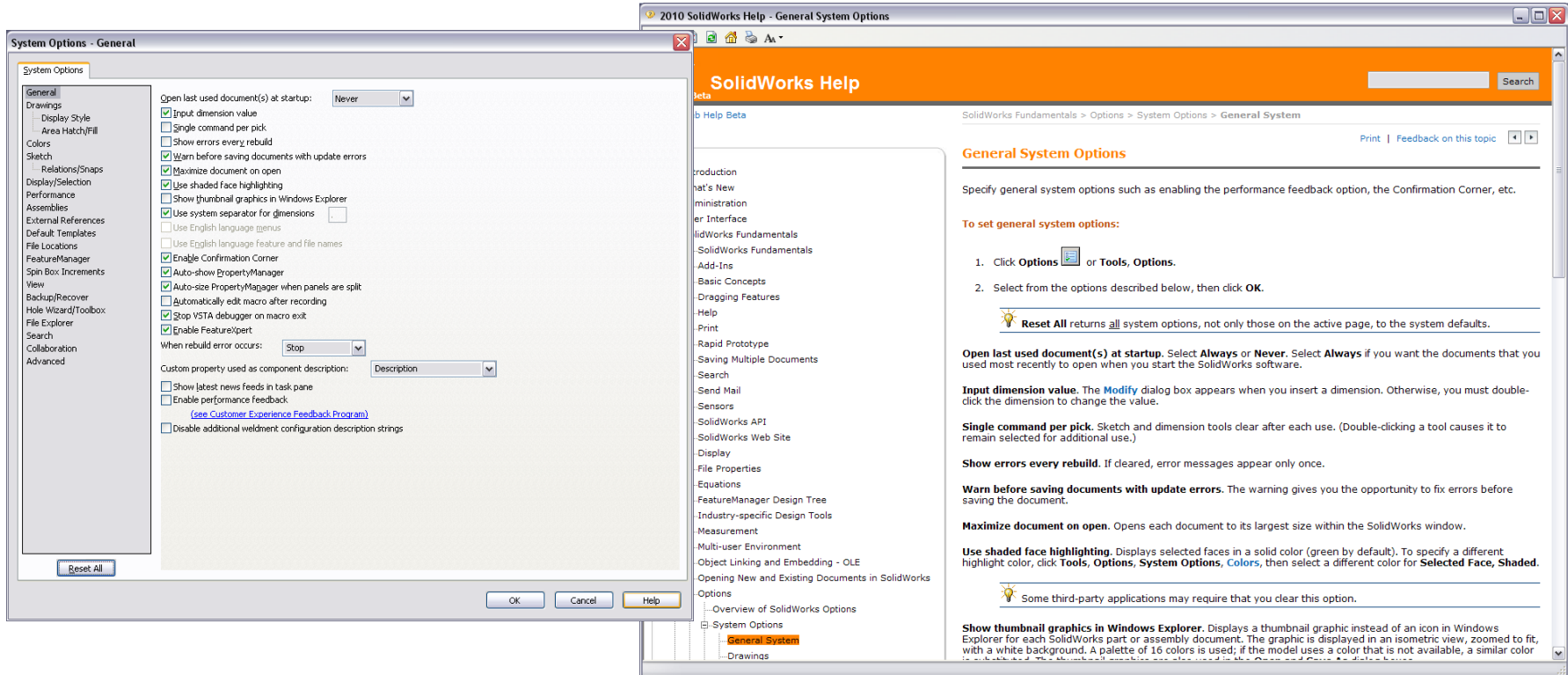
John Matrishon

Wmass-SWUG Leader



Special thanks to Kelly Frasier, Colorado Swug for some clips in her presentation from SWW10

- Help. It's really good to have help!



- What to do when everything is perfect?
 - Copy Settings wizard
 - Settings aren't saved until you have a successful exit

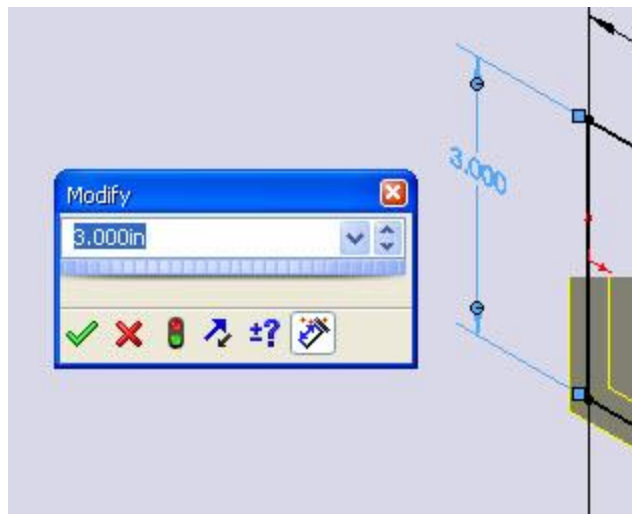
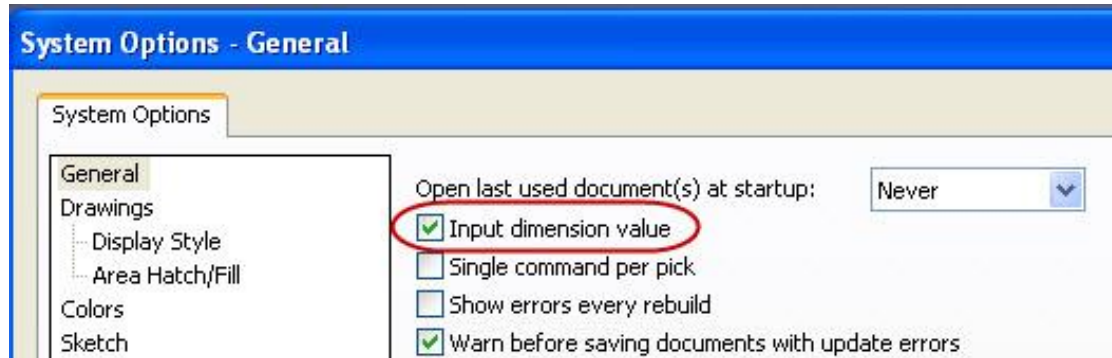
- **System options** apply to your environment inside SolidWorks. They are saved in the registry.
- **Document Properties** are particular to a file, and are saved with that file, and only that file.



- Settings, settings, settings.....why so many?
- What do they control?
 - These are YOUR settings that are applied on YOUR profile
 - They are stored in the registry
 - They are not part of the document
 - However, they do affect documents current and future
- Do I need to change them?
 - Not everyone works the same, it's up to you.



Reset All returns all system options, not only those on the active page, to the system defaults.

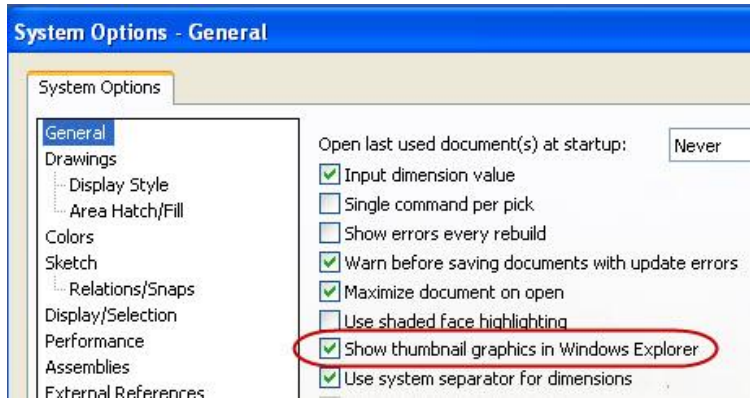


The Modify dialog box appears when a dimension is inserted.

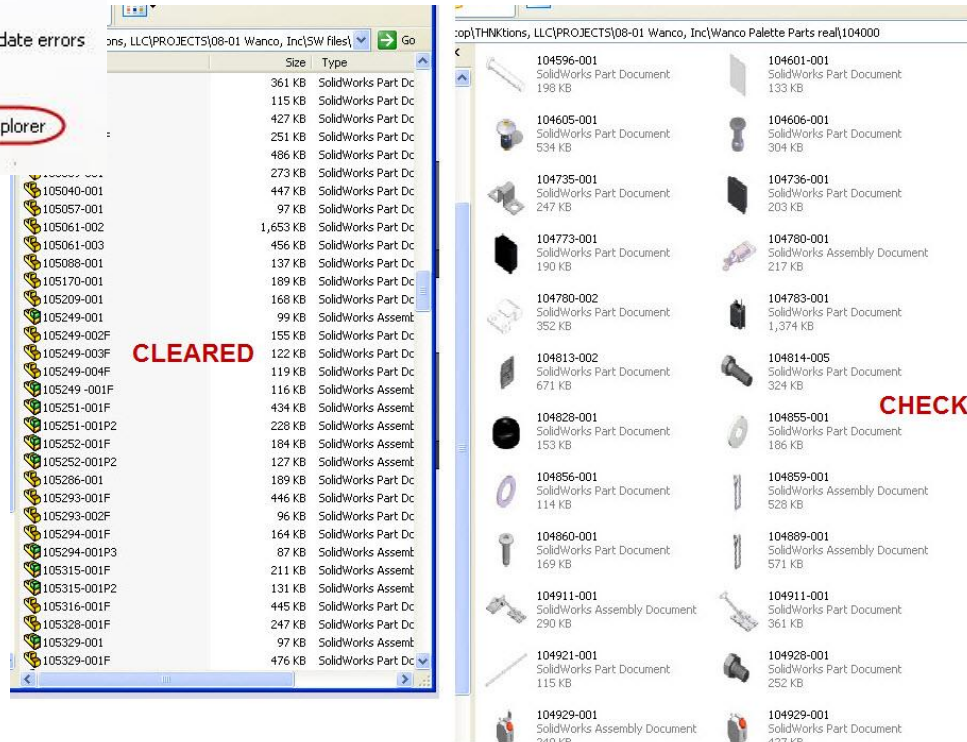
Otherwise, the dimension must be double-clicked to change the value.

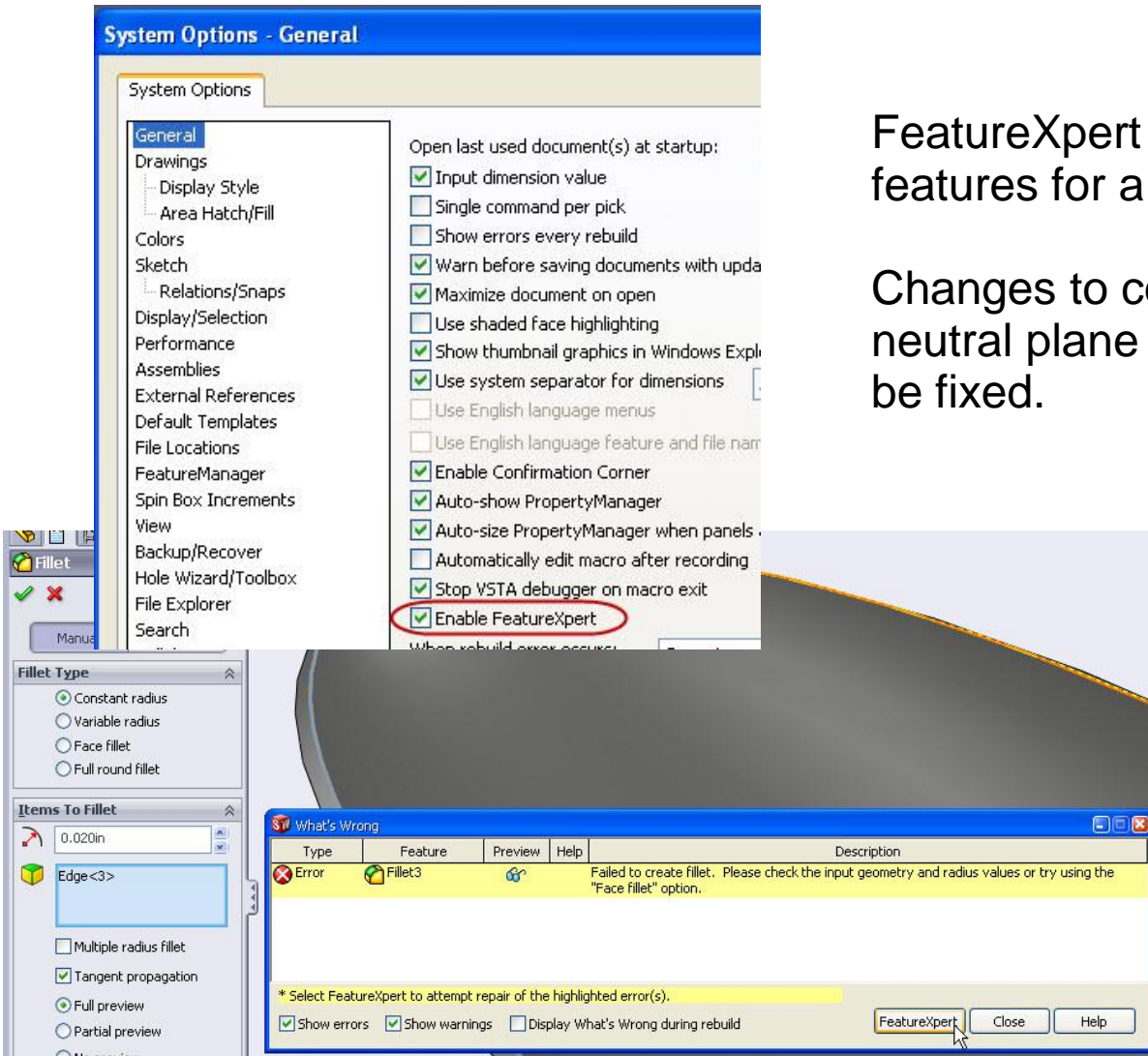
Note: some slow lag times can be attributed to video card settings

Displays a thumbnail graphic instead of an icon for each SolidWorks part or assembly document.



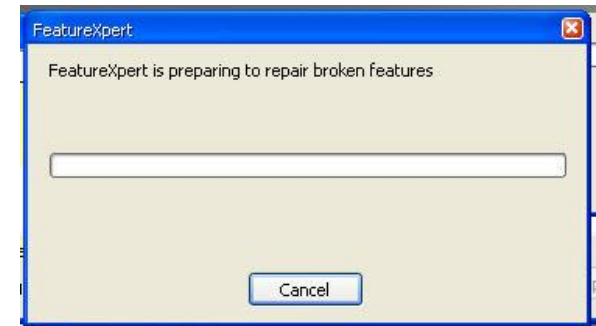
Note: I recommend you uncheck this box if you're running a slower machine, or high resolution settings

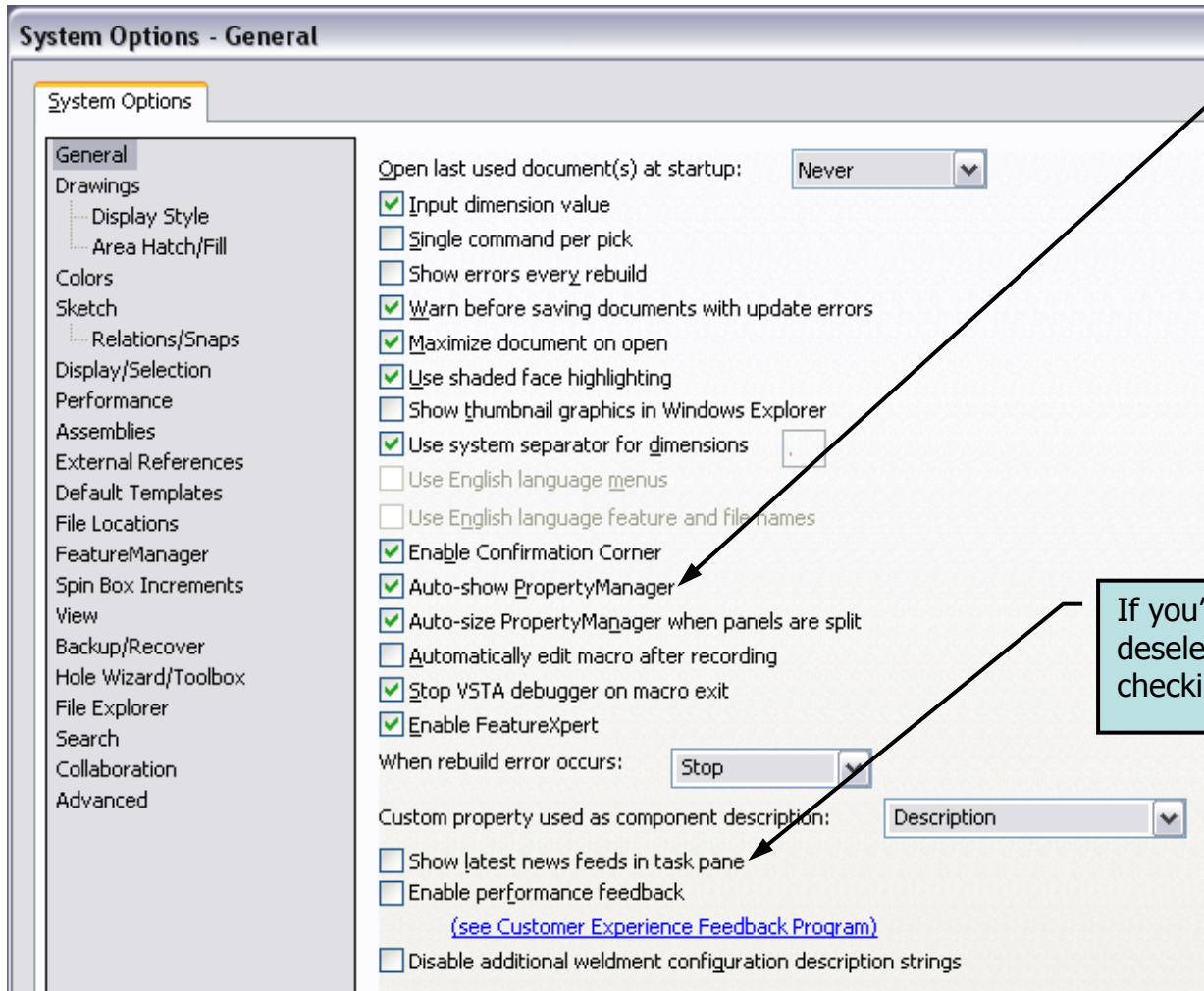




FeatureXpert automatically fixes some features for a successful rebuild.

Changes to constant radius fillets and neutral plane drafts can be attempted to be fixed.





The PropertyManager appears when an existing entity, dimension, or annotation is selected in the graphics area.

If the option is cleared, and an item is selected, the PropertyManager tab must be manually selected to be accessed

If you're struggling with system performance, deselect this box to stop your system from checking the internet

System Options - Drawings

System Options

- General
- Drawings**
 - Display Style
 - Area Hatch/Fill
- Colors
- Sketch
 - Relations/Snaps
- Display/Selection
- Performance
- Assemblies
- External References
- Default Templates
- File Locations
- FeatureManager
- Spin Box Increments
- View
- Backup/Recover
- Hole Wizard/Toolbox
- File Explorer
- Search
- Collaboration
- Advanced

Eliminate duplicate model dimensions on insert
 Mark all part/assembly dimension for import into drawings by default
 Automatically scale new drawing views
 Show contents while dragging drawing view
 Display new detail circles as circles
 Select hidden entities
 Allow auto-update when opening drawings
 Disable note/dimension inference
 Print out-of-sync water mark
 Show reference geometry names in drawings
 Automatically hide components on view creation
 Display sketch arc centerpoints
 Display sketch entity points
 Print breaklines in broken view
 Save tessellated data for drawings with shaded and draft quality views
 Automatically populate View Palette with views
 Show sheet format dialog on add new sheet
 Reduce spacing when dimensions are deleted or edited (add or change tolerance, text etc...)
 Override quantity column name in Bill Of Materials
 Name to use:
 Detail view scaling: x
 Custom property used as Revision:
 Keyboard movement increment:

Reset All

OK Cancel Help

When selected, new profiles for detail views appear as circles.

When cleared, the sketched profile appears.

Notes & dimensions snapping to other entities

If you like clean drawings without showing sketch entities like these, deselect them

System Options - Display Style

System Options

General

Drawings

Display Style

Area Hatch/Fill

Colors

Sketch

Relations/Snaps

Display/Selection

Performance

Assemblies

External References

Default Templates

File Locations

FeatureManager

Spin Box Increments

View

Backup/Recover

Hole Wizard/Toolbox

File Explorer

Search

Display style for new views

- Wireframe
- Hidden lines visible
- Hidden lines removed
- Shaded with edges
- Shaded

Tangent edges in new views

- Visible
- Use font
- Hide ends
- Removed

Display quality for new views

- High quality
- Draft quality

Sometimes this gets people and they don't know why new views keep coming up in Draft quality.

How many of you use Draft quality views?

System Options - Colors

System Options - Colors

System Options

- General
- Drawings
 - Display Style
 - Area Hatch/Fill
- Colors**
- Sketch
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- Display/Selection
- Performance
- Assemblies
- External References
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- FeatureManager
- Spin Box Increments
- View
- Backup/Recover
- Hole Wizard/Toolbox
- File Explorer
- Search
- Collaboration
- Advanced

Current color scheme: [Dropdown] [Delete]

Color scheme settings

- Drawings, Paper Color
- Drawings, Background
- Drawings, Visible Model Edges
- Drawings, Hidden Model Edges
- Drawings, Model Edges (SpeedPak)
- Drawings, Model Tangent Edges
- Dimensions, Imported (Driving)
- Dimensions, Non Imported (Driven)
- Dimensions, Dangling
- Dimensions, Not Marked for Drawing
- Dimensions, Controlled by Design Table

Background appearance:

- Use document scene background
- Plain (Viewport Background color above)
- Gradient (Top/Bottom Gradient colors above)
- Image file:
 - smoke.png

Color

Basic colors:

[Red]	[Yellow]	[Green]	[Cyan]	[Blue]	[Magenta]	[Pink]
[Orange]	[Light Green]	[Light Blue]	[Dark Blue]	[Purple]	[Dark Magenta]	[Dark Pink]
[Dark Red]	[Dark Orange]	[Dark Green]	[Dark Cyan]	[Dark Blue]	[Dark Purple]	[Dark Magenta]
[Black]	[Dark Olive Green]	[Dark Teal]	[Dark Blue-Gray]	[Dark Purple-Gray]	[Dark Magenta-Gray]	[Dark Pink-Gray]

Custom colors:

[White]	[Light Blue]	[Yellow]	[Green]	[Light Gray]	[Light Blue-Gray]	[Black]
[Black]	[Black]	[Black]	[Black]	[Black]	[Black]	[Black]

Define Custom Colors >>

ColorSolid: [Red] Hue: [Value] Sat: [Value] Lum: [Value]

Buttons: OK, Cancel, Add to C...

Buttons: Reset All To Defaults, Save As Scheme...

Checkboxes:

- Use specified color for drawings paper color (disable image in sheet background)
- Use specified color for Shaded with Edges mode
- Use specified colors when editing parts in assemblies

Dangling dimensions - Please change this to something else like red or orange

Use these settings to help with certain types of renderings or screen captures

System Options - Sketch

System Options

- General
- Drawings
 - Display Style
 - Area Hatch/Fill
- Colors
 - Sketch**
 - Relations/Snaps
- Display/Selection
- Performance
- Assemblies
- External References
- Default Templates
- File Locations
- FeatureManager
- Spin Box Increments
- View
- Backup/Recover
- Hole Wizard/Toolbox
- File Explorer
- Search
- Collaboration
- Advanced

- Use fully defined sketches
- Display arc centerpoints in part/assembly sketches
- Display entity points in part/assembly sketches
- Prompt to close sketch
- Create sketch on new part
- Override dimensions on drag/move
- Display plane when shaded
- Display virtual sharps
- Line length measured between virtual sharps in 3d
- Enable spline tangency and curvature handles
- Show spline control polygon by default
- Ghost image on drag
- Show curvature comb bounding curve
- Enable on screen numeric input on entity creation
- Over defining dimensions
 - Prompt to set driven state
 - Set driven by default

Starts a new model with a sketch on the front plane

Adds the dimension while you are sketching

Controls how over defined dimension react

The screenshot shows the 'System Options' dialog box with the 'Relations/Snaps' tab selected. The left sidebar lists various system options, with 'Relations/Snaps' highlighted. The main area contains the following settings:

- Enable snapping
- Snap to model geometry
- Automatic relations

Sketch snaps

- End points and sketch points
- Center Points
- Mid-points
- Quadrant Points
- Intersections
- Nearest
- Tangent
- Perpendicular
- Parallel
- Horizontal/vertical lines
- Horizontal/vertical to points
- Length
- Grid
 - Snap only when grid is displayed
- Angle
 - Snap angle: 45.00deg

More Global settings

Use Quick Snaps to override these settings, either a right click or F3

This one can be the reason if you see an hidden edge and you just can't select it

HLV = Hidden lines Visible
HLR= Hidden lines Removed

Controls the transparency when parts are edited in an assembly. This only affects the components that are **not** being edited.

Highlights a face, edge, or vertex when your cursor is over them without selecting

You will be able to select through transparent models to the ones behind. Hold "shift" to override.

To the screen or to the 3D annotation standards

Performance Options

System Options | Document Properties

General
Drawings
 Display Style
 Area Hatch/Fill
Colors
Sketch
 Relations/Snaps
Display/Selection
Performance
Assemblies
External References
Default Templates
File Locations
FeatureManager
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Advanced

Verification on rebuild (enable advanced body checking)

Ignore self-intersection check for some sheet metal features

Transparency

High quality for normal view mode

High quality for dynamic view mode

Curvature generation: Always (for every shaded model)

Off More Less (faster)

Level of detail:

Assemblies

Automatically load components lightweight

Always resolve sub-assemblies

Check out-of-date lightweight components: Don't Check

Resolve lightweight components: Prompt

Rebuild assembly on load: Always

Off Fast Slow

Mate animation speed:

Update mass properties while saving document

Use shaded preview

Use software OpenGL

No preview during open (faster).

Go To Image Quality

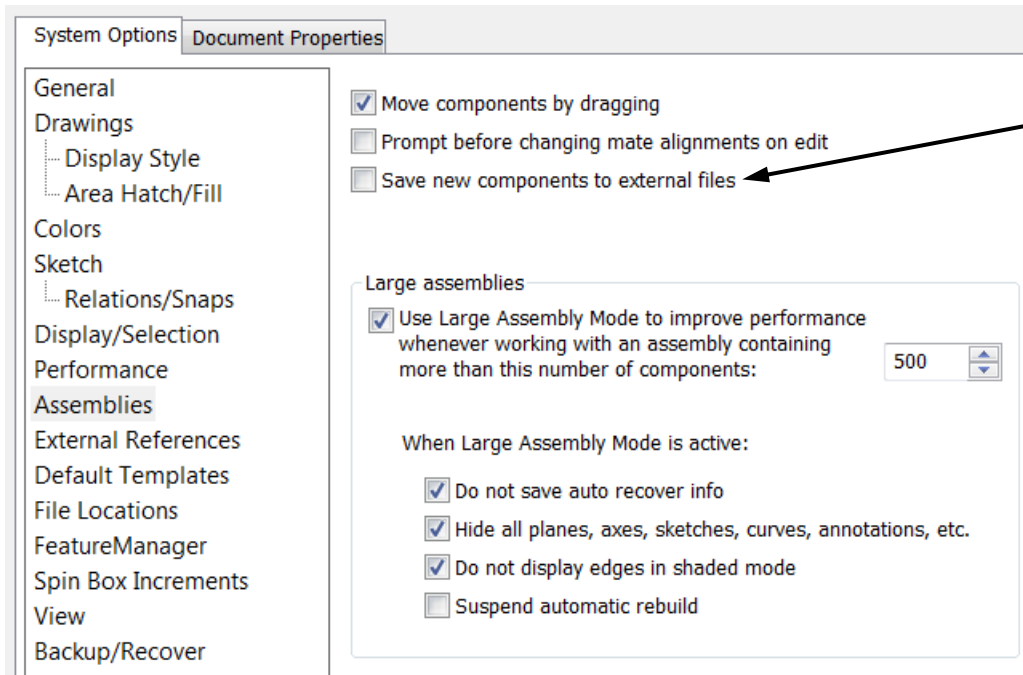
If you are working on complex surfaces and having issues, you may want to turn this on

This will be related to your video card and the amount of RAM that is used as you open files

When you have large assemblies, and performance issues or an older computer, use lightweight to help conserve RAM

With NO files open, this option is available if you are trying to trouble shoot video card problems. If the problem goes away, chances are it's your video card that is the issue.

Note: Changes done here will not affect documents that were already open, and many are not available when Large Assy Mode is "on"

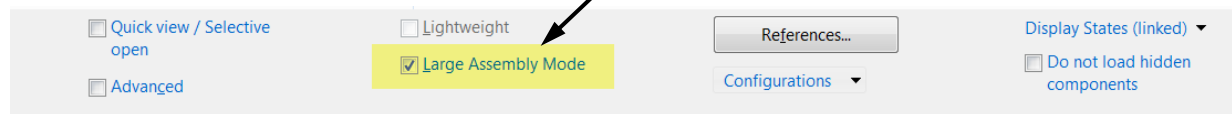


When checked, a new component added during an assembly will not become a Virtual Component by default. It will ask for name and save location. (How it worked prior to Virtual components)

The new part appears in the FeatureManager design tree. Externally saved parts appear with a name in the form **Part*n***. Virtual components appear with a name in the form **[Part*n*^*assembly_name*]**.

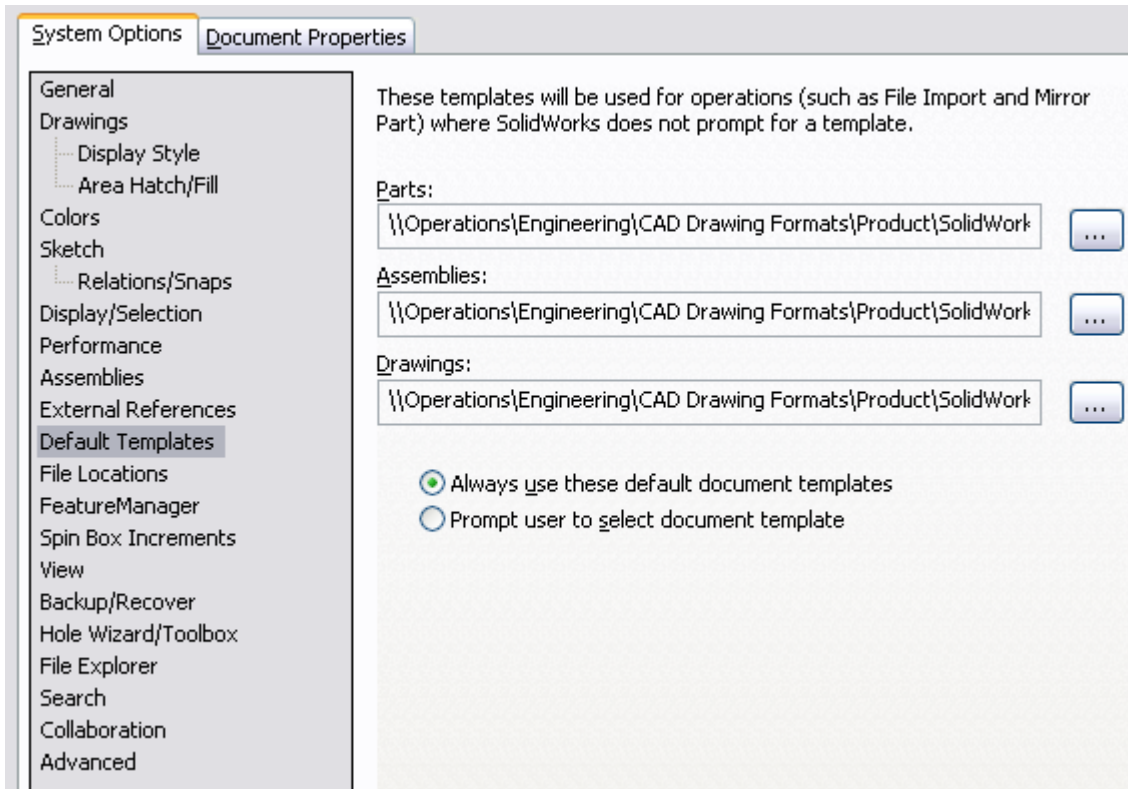
All the Large Assembly options are there to help increase the **performance** of your assembly with a greater component count than what you specify.

You can choose not use Large assembly mode during file open if needed



This table is taken directly from the Help

System Options page or Toolbar	Option	Status when Large Assembly Mode is on
Drawings	Show contents while dragging drawing view	Off. Only the view boundary is shown while dragging a drawing view.
	Smooth dynamic motion of drawing views	Off. Dynamic operations to drawings, such as panning and zooming, do not display smoothly.
	Automatically hide components on view creation	On. Any components of an assembly not visible in a new drawing view are hidden and listed on the Hide/Show Components tab of the Drawing View Properties dialog box.
Drawings - Display Style	Display style for new views	Hidden lines removed is set as the default display style for new views.
	Display quality for new views	Draft quality. Only minimum model information is loaded into memory. Some edges may appear to be missing, and print quality may be slightly degraded.
Display/Selection	Dynamic highlight from graphics view	Off. Model faces, edges, and vertices are <u>not</u> highlighted when you move the pointer over a sketch, model, or drawing.
	Anti-alias edges	Off. Jagged edges in Shaded With Edges , Wireframe , Hidden Lines Removed , and Hidden Lines Visible modes are <u>not</u> smoothed out.
	Assembly transparency for in context edit	Maintain assembly transparency. Components not being edited retain their individual transparency settings.
FeatureManager	Dynamic highlight	Off. The geometry in the graphics area (edges, faces, planes, axes, and so on) is <u>not</u> highlighted when the pointer passes over the item in the FeatureManager design tree.
Performance	Transparency	High quality for normal view mode. While the part or assembly is not moving or rotating, the transparency is high quality. When moved or rotated with the pan or rotate tools, the application switches to low-quality transparency, enabling you to rotate the model faster.
	Curvature generation	Only on demand. Initial curvature display is slower, but uses less memory.
	Level of detail	Minimum. The level of detail is minimal during dynamic view operations (zoom, pan, and rotate) in assemblies, multi-body parts, and draft views in drawings.
	Check out-of-date lightweight components	Don't Check. Loads assemblies without checking for out-of-date lightweight components.
	Update mass properties while saving document	Off. Does not recalculate the mass properties on save. The next time you access the mass properties, the system will need to recalculate them.
View toolbar and menu	Shadows in Shaded Mode	Off.
	RealView Graphics	Off.

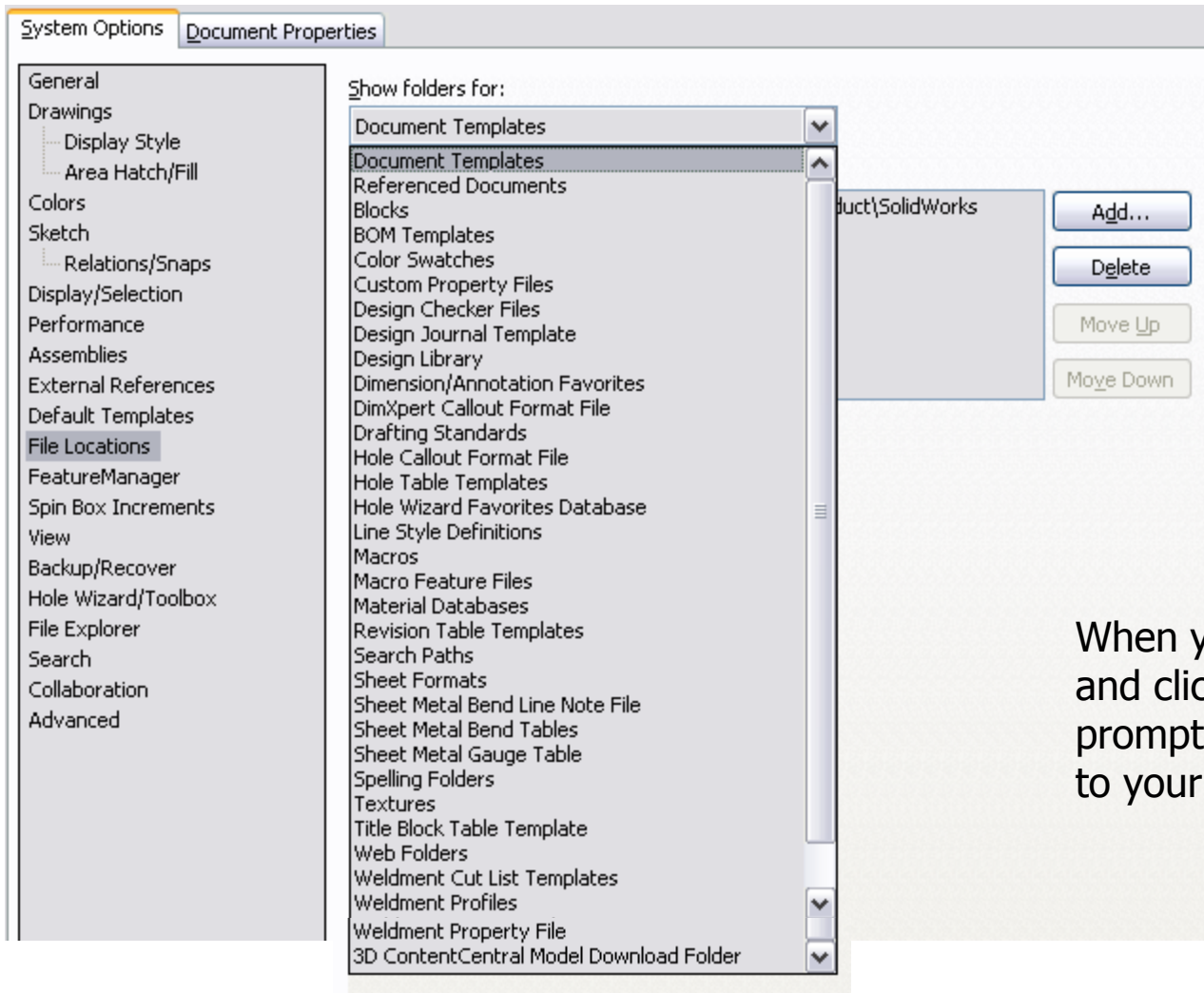


For those files that are generated automatically, SolidWorks will look here to grab the template.

Usually these will be your company templates.

I recommend using the root network path and avoid using Mapped network drive letters.

This is not related to the templates when creating a new part, assembly, or drawing file



Use full network paths. No mapped drives!

Much of this will depend on you company standards

When you add a new location, and click OK, you'll be prompted to add the location to your search paths

System Options | Document Properties

General

Drawings

- Display Style
- Area Hatch/Fill

Colors

Sketch

- Relations/Snaps

Display/Selection

Performance

Assemblies

External References

Default Templates

File Locations

FeatureManager

Spin Box Increments

View

Backup/Recover

Hole Wizard/Toolbox

File Explorer

Search

Collaboration

Advanced

Scroll selected item into view

Name feature on creation

Arrow key navigation

Dynamic highlight

Use transparent flyout FeatureManager tree in parts/assemblies

Display warnings: Always

Hide/show tree items

Blocks	Automatic	Equations	Automatic
Design Binder	Automatic	Material	Show
Annotations	Show	Default Planes	Show
Lights, Cameras, and Scene	Show	Origin	Show
Solid Bodies	Show	Mate References	Automatic
Surface Bodies	Automatic	Design Table	Automatic
Tables	Automatic	Sensors	Automatic

If you like to name your features immediately after creation, this will put you in the edit mode right after.

Part1 (Default <<Default>_Dis

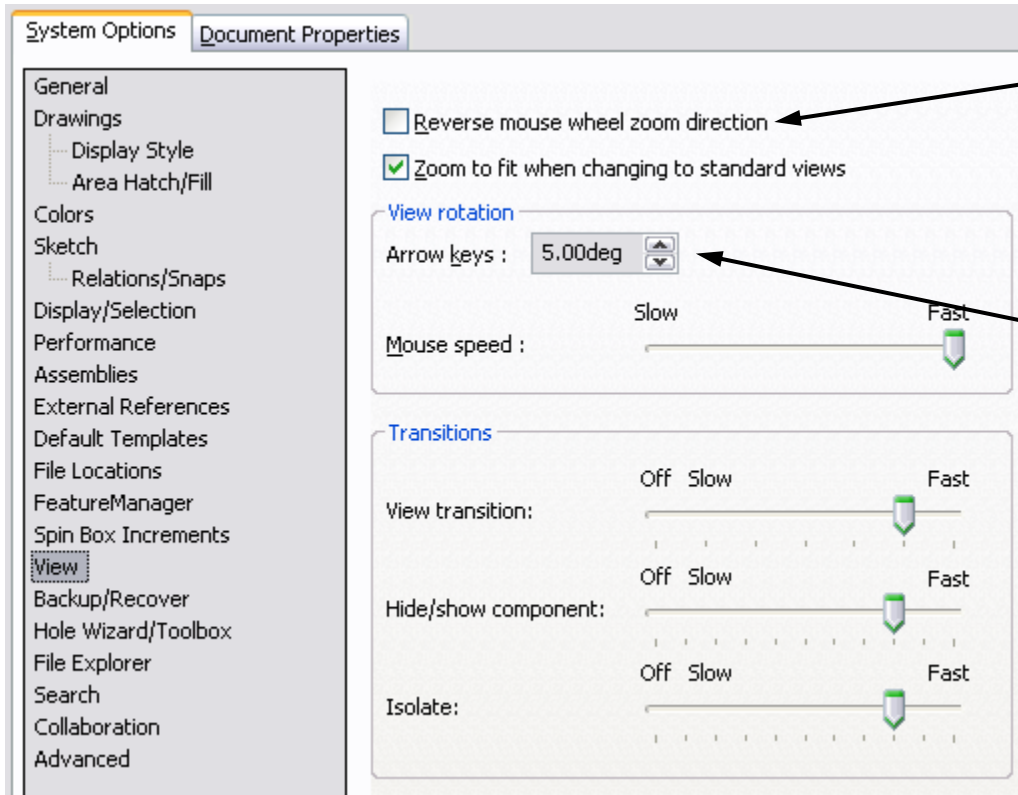
- Sensors
- Annotations
- Solid Bodies(1)
 - Boss-Extrude1
- Lights, Cameras and Scene
- Material <not specified>
- Front Plane
- Top Plane
- Right Plane
- Origin

Set to Show

aw_spider

- Annotations
- Design Binder
- Material <not specified>
- Lights and Cameras
- Solid Bodies(1)
 - Plane1
 - Plane2
 - Plane3
- Origin
- Base-Extrude
- Boss-Extrude1
- Boss-Extrude2
- Boss-Extrude3
- Boss-Extrude4
- Boss-Extrude5
- Cut-Extrude2
- Boss-Extrude6
- Cut-Extrude1

Transparent flyout FeatureManager design tree



Changes the direction of the mouse wheel for zooming in and out

For keyboard arrow keys, remember holding down the ALT+left arrow or ALT+right arrow keys allows rotation about an axis normal to the screen.

The screenshot shows the 'System Options' dialog box with the 'Document Properties' tab selected. The 'Backup/Recover' section is highlighted in the left-hand navigation pane. The settings are as follows:

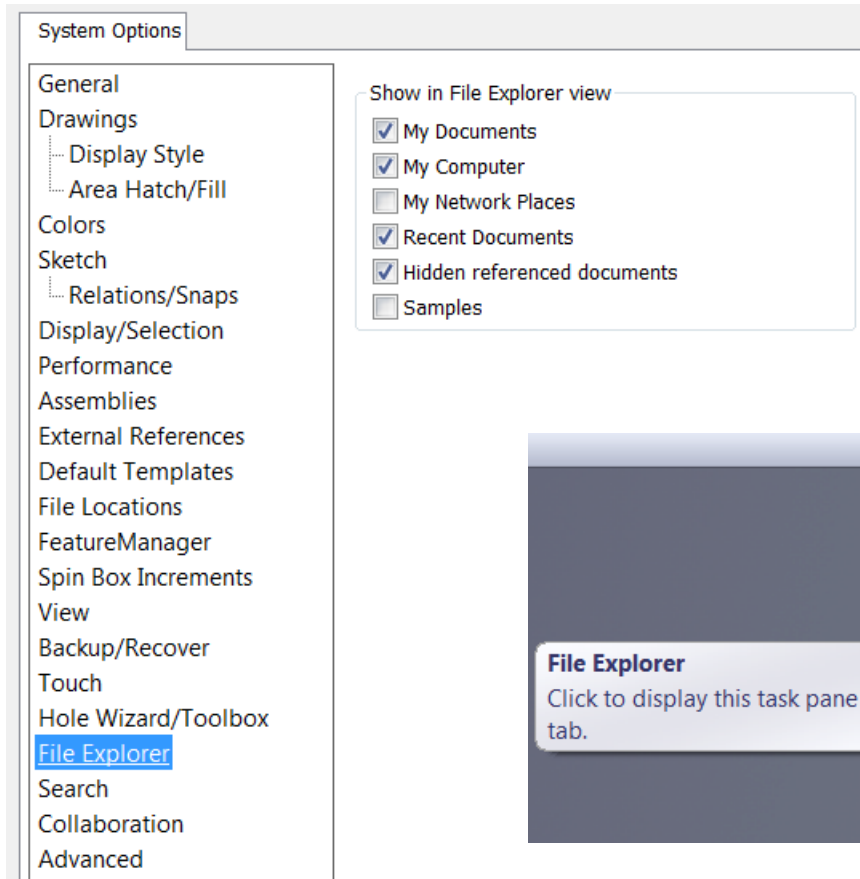
- Auto-recover:**
 - Save auto-recover info every: 0 minutes
 - Auto-recover folder: C:\DOCUME~1\JMATRI~1\LOCAL5~1\TempSWBacku
- Backup:**
 - Number of backup copies per document: 1
 - Backup folder: C:\DOCUME~1\JMATRI~1\LOCAL5~1\TempSWBacku
 - Save backup files in the same location as the original
 - Remove backups older than 7 days
- Save notification:**
 - Show reminder if document has not been saved for: 15 minutes

An arrow points from a text box to the 'Show reminder if document has not been saved for' checkbox.

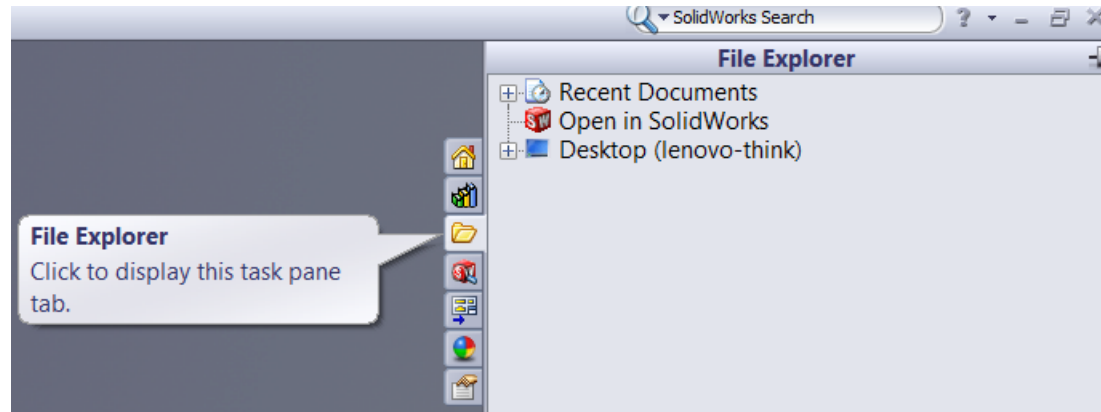


At a minimum, this is a good one to have.

Here is where you set the frequency and location of your auto-recovery and Backups, and how often SolidWorks reminds you to save



What folder do you want to see in the Task Pane?



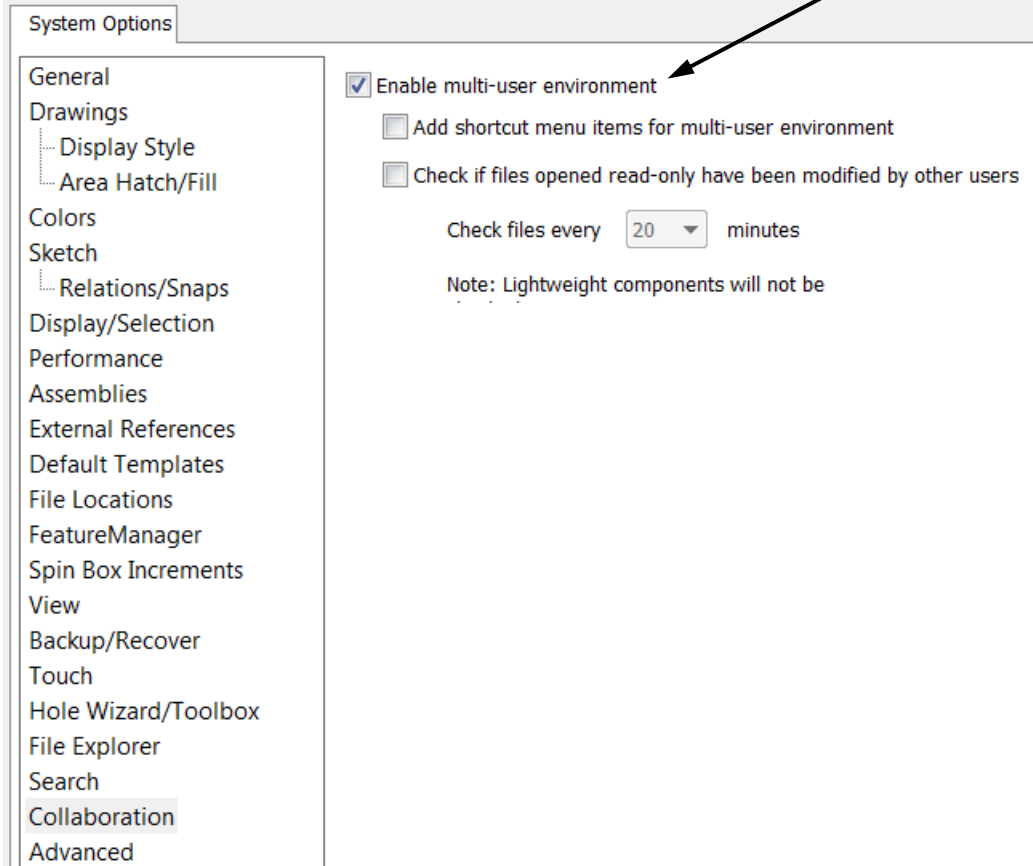
The screenshot shows the 'Document Properties' dialog box for Search. The left sidebar lists various options, with 'Search' selected. The main area contains several settings:

- Search while typing (incremental search)
- Include 3D ContentCentral results
- Results per page: 10
- Maximum results per data source: 1000
- Indexing performance**
 - Index only when computer is idle
 - Always index (may slow SolidWorks)
- Dissection**
 - Schedule dissection daily to automatically dissect files in search paths (Dissection does not modify or resave your files)
 - Daily start time: 11:00:00 PM
 - Daily stop time: 5:00:00 AM
 - Location: C:\Documents and Settings\jmatrishon\Application Data\SolidW

When SolidWorks says it may slow it down, it is a good idea to listen...

Dissection is a COOL function, but can hog up resources on an older computer. It's something to think about and research a little more.

This will enable the other options



*This is not a substitute for a Product Data Management (PDM) system.

System Options | Document Properties

General
 Drawings
 └─ Display Style
 └─ Area Hatch/Fill
 Colors
 Sketch
 └─ Relations/Snaps
 Display/Selection
 Performance
 Assemblies
 External References
 Default Templates
 File Locations
 FeatureManager
 Spin Box Increments
 View
 Backup/Recover
 Hole Wizard/Toolbox
 File Explorer
 Search
 Collaboration
Advanced

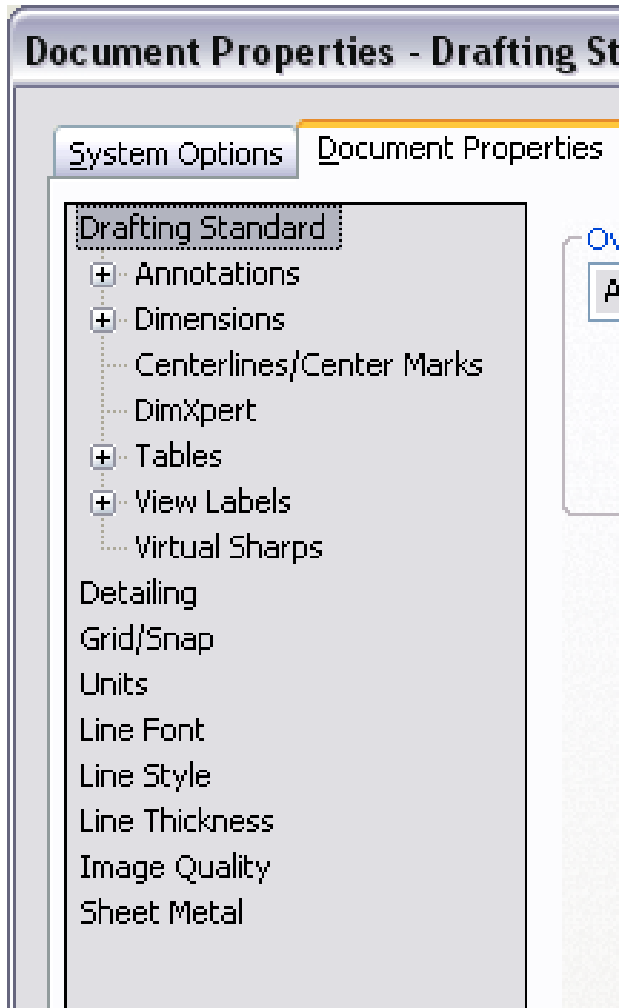
Dismissed messages (checked messages will be shown again)

- "Models contained within the assembly have changed. Would you like to re..."
- "Feature ____ failed to rebuild, which may cause subsequent features to f..."
- "Choose display state to convert."
- "Do you really want to delete the following item(s)?"
- "Adding this dimension will cause the sketch to become over defined or un..."
- "Deleting the offset dimension will remove the offset relations from the..."
- "Do you want to discard all changes to the sketch?"
- "This part has not been rebuilt since the last change. It needs a rebuild..."
- "The internal ID of document ____ does not match the internal ID saved wi..."
- "Unable to locate the file _____. Would you like to find it yourself?"
- "MSG_EXTERN_REFS_CHANGED"
- "The following models referenced in this document have been modified. The..."
- "Saving will convert the following files to the current version:"
- "The automatic relations inferred by this operation would over define the..."
- "____ needs updating. It has not been rebuilt successfully since the las..."
- "____ has not been rebuilt successfully since the last feature change or..."
- "____ has one or more update errors. Save anyway?"
- "____ is being referenced by other open documents. "Save As" will replac..."
- "MSG_SAVEAS_REFWARNING_OPEN_IN_OWN_WINDOW"
- "Because you used "Selective Open", hidden components are not loaded into..."
- "MSG_SHOW_HATCHING_WHILE_PRINTING"
- "You are about to make changes to an older version file. Saving will conv..."
- "This assembly contains unsaved virtual components which must be saved."

Remember all those dialog boxes that you selected "don't ask me again", here they are, in case to need them back!



WHEW!
THAT'S A LOT
STUFF



- Settings, they're here too!
- What do they control?
 - Particular to the document parts, assemblies, and drawings
- Do I need to change them?
 - If your templates are set up correctly, than no...most of the time
- Heavy hitters, which ones to pay attention too

Document Property Differences

System Options | Document Properties

- Drafting Standard
 - Annotations
 - Balloons
 - Datums
 - Geometric Tolerances
 - Notes
 - Surface Finishes
 - Weld Symbols
 - Dimensions
 - Angle
 - Arc Length
 - Chamfer
 - Diameter
 - Hole Callout
 - Linear
 - Ordinate
 - Radius
 - Virtual Sharps
 - Tables
 - Bill of Materials
 - Title Block Table
- Detailing
- Grid/Snap
- Units
- Colors
- Material Properties
- Image Quality
- Plane Display
- DimXpert
 - Size Dimension
 - Location Dimension
 - Chain Dimension
 - Geometric Tolerance
 - Chamfer Controls
 - Display Options

35 tabs

PART

System Options | Document Properties

- Drafting Standard
 - Annotations
 - Balloons
 - Datums
 - Geometric Tolerances
 - Notes
 - Surface Finishes
 - Weld Symbols
 - Dimensions
 - Angle
 - Arc Length
 - Chamfer
 - Diameter
 - Hole Callout
 - Linear
 - Ordinate
 - Radius
 - Virtual Sharps
 - Tables
 - Bill of Materials
 - Title Block Table
- Detailing
- Grid/Snap
- Units
- Colors
- Image Quality
- Plane Display

27 tabs

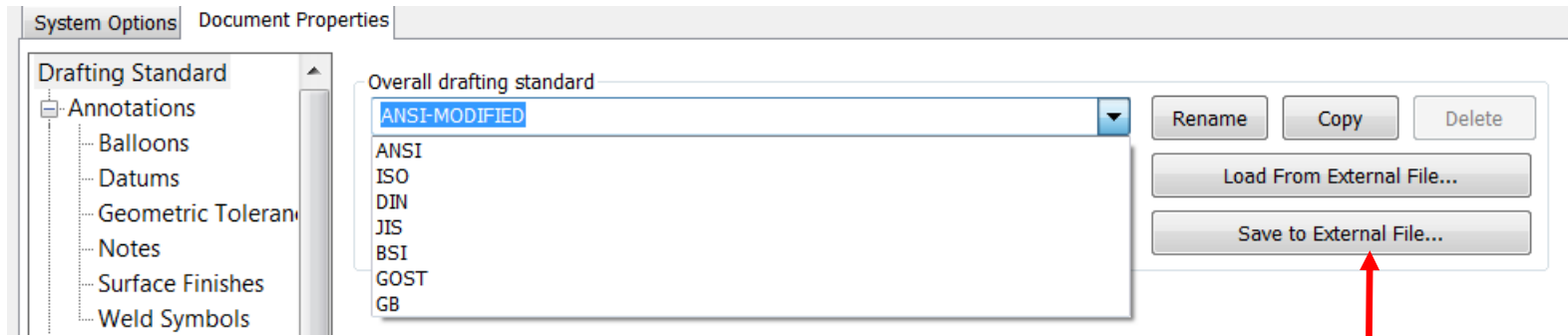
ASSEMBLY

System Options | Document Properties

- Drafting Standard
 - Annotations
 - Balloons
 - Datums
 - Geometric Tolerances
 - Notes
 - Surface Finishes
 - Weld Symbols
 - Dimensions
 - Angle
 - Arc Length
 - Chamfer
 - Diameter
 - Hole Callout
 - Linear
 - Ordinate
 - Radius
 - Centerlines/Center Marks
 - DimXpert
 - Tables
 - Bill of Materials
 - General
 - Hole
 - Revision
 - View Labels
 - Auxiliary
 - Detail
 - Section
 - Virtual Sharps
- Detailing
- Grid/Snap
- Units
- Line Font
- Line Style
- Line Thickness
- Image Quality
- Sheet Metal

DRAWING

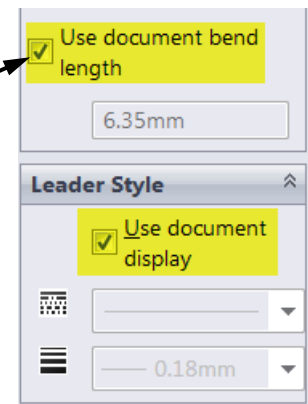
37 tabs



Use a known good part, assembly, and drawing file to first "Save to External file" to a directory that everyone has access to.

*Note if you have drawing formats with different specs, you'll need a file for each size drawing that is different.

If these type of boxes are unchecked, then the change will not take affect for that entity



If you have an older file, you can update it to your current standards by selecting the "Load from External File..."

Where were you in 1982?

System Options Document Properties

Drafting Standard

- Annotations
 - Balloons
 - Datums
 - Geometric Tolerances
 - Notes
 - Surface Finishes
 - Weld Symbols
- Dimensions
- Virtual Sharps
- Tables
- Detailing
- Grid/Snap
- Units
- Colors
- Material Properties
- Image Quality
- Plane Display
- DimXpert
 - Size Dimension
 - Location Dimension
 - Chain Dimension
 - Geometric Tolerance
 - Chamfer Controls
 - Display Options

Overall drafting standard
ANSI

Base datums standard
ANSI

Leader style

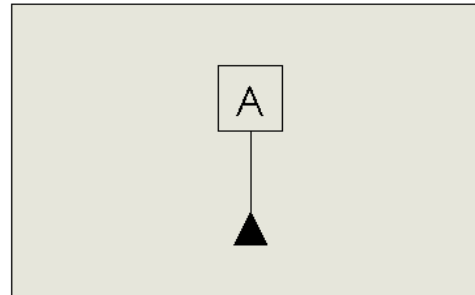
Frame style

Text
Font... Century Gothic

Datum features
Display type: Per Standard
Next label: A

Anchor style

Display datums per 1982



Display datums per 1982

Overall drafting standard has been changed to "ANSI-MODIFIED"

Note that the overall drafting standard note has popped up when you change a document property.

Dimension stuck in the center?

System Options | Document Properties

Drafting Standard

- Annotations
 - Balloons
 - Datums
 - Geometric Tolerances
 - Notes
 - Surface Finishes
 - Weld Symbols
- Dimensions**
 - Angle
 - Arc Length
 - Chamfer
 - Diameter
 - Hole Callout
 - Linear
 - Ordinate
 - Radius
 - Virtual Sharps
- Tables
- Detailing
- Grid/Snap
- Units
- Colors
- Material Properties
- Image Quality
- Plane Display
- DimXpert
 - Size Dimension
 - Location Dimension
 - Chain Dimension
 - Geometric Tolerance
 - Chamfer Controls
 - Display Options

Overall drafting standard
ANSI-MODIFIED

Text
Font... Century Gothic

Dual dimensions
 Dual dimensions display Show units for dual display
 Top Bottom Right Left

Primary precision **Dual precision**
 .12 .12
 Same as nominal Same as nominal

Fractional display
 Style: x/x x/x x/x x/x Stack size: 100%
 Show double prime mark x/x'' x/x

Bent leaders
 Leader: 0.25in

Leading Standard
Trailing Smart

Add parentheses by default
 Center between extension lines
 Include prefix inside basic tolerance box
 Display dual basic dimension in one box

Tolerance...

Arrows
 0.04in
 0.13in
 0.25in
 Scale with dimension height
 Style:

Offset distances
 0.23622in
 0.393701in

Radial/Diameter leader snap 15deg

When that dimension keeps snapping back to the center, and you can't seem to override it

System Options | Document Properties

Drafting Standard

- Annotations
 - Balloons
 - Datums
 - Geometric Tolerances
 - Notes
 - Surface Finishes
 - Weld Symbols
- Dimensions
 - Centerlines/Center Marks
 - DimXpert
- Tables
- View Labels
- Virtual Sharps
- Detailing**
- Grid/Snap
- Units
- Line Font
- Line Style
- Line Thickness
- Image Quality
- Sheet Metal

Display filter

- Cosmetic threads
- Datums
- Datum targets
- Feature dimensions
- Reference dimensions
- DimXpert dimensions
- Shaded cosmetic threads
- Geometric tolerances
- Notes
- Surface finish
- Welds
- Display all types

Text scale: 1:1

- Always display text at the same size
- Display items only in the view in which they are created
- Display annotations
- Use assembly setting for all components
- Hide dangling dimensions and annotations

Import annotations

- From entire assembly

Auto insert on view creation

- Center marks-holes
- Center marks-fillets
- Center marks-slots
- Centerlines
- Balloons
- Dimensions marked for drawing

Cosmetic thread display

- High quality

Area hatch display

- Show halo around annotation:

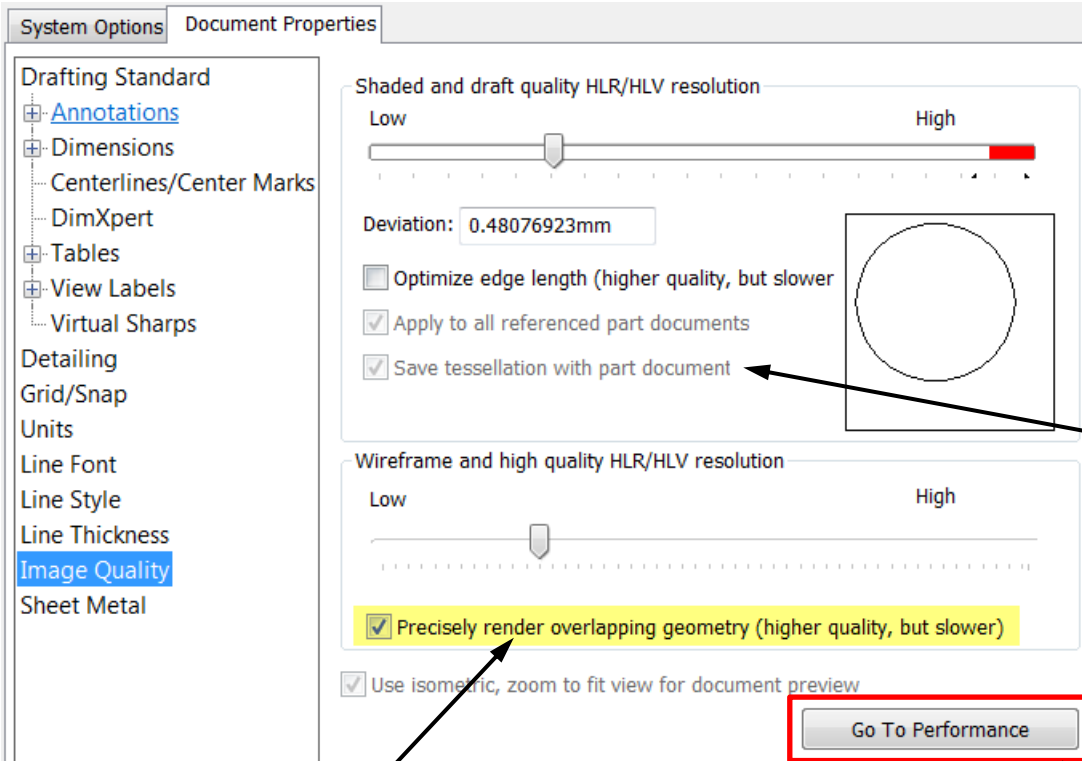
View break lines

Gap: 12.7mm

Extension: 3.175mm

In a drawing, what symbols are created automatically





Higher resolution = more accurate curves, but slower model rebuilding

Save tessellation with part document (parts only). Saves display information. When cleared, file size is reduced (often significantly), but the model is not displayed when the file is opened in **view-only** mode, the SolidWorks Viewer, or eDrawings. The display data is regenerated when the file is opened again in SolidWorks.

For drawings only. If you have a drawing with many tiny facets, you may want to uncheck this box to improve performance

This is a toggle back to System Options/Performance because the two are so closely related

Questions?
Thank you!!