SolidWorks 2014 Tool/Options Explained

Tools/ Options/ System Options

**GENERAL**

- **Open last used document(s) at startup: Never or Always**
  If you close SW with the files still open, then they will open back up when SW is re-started.

- **Input dimension value**
  When you add a dimension, do you want it to leave the dimension as it is or allow you to enter a number in the Modify Box?

- **Single command per pick**
  If you click on the icon, it will only operate once. If you are in this mode and double click on the icon (for example the Line icon), then you can temporarily over-ride the option.

- **Use Shaded face highlighting**
  With the option un-checked, then only the edges of the faces or features will highlight, otherwise the face(s) will highlight too.

- **Show thumbnail graphics in Windows Explorer**
  Isometric view of the part is displayed instead of regular block icons for the file. Having this option turned on can have some performance issues in folders with a large number of files.

- **Use system separator for dimensions**
  Some countries/standards call for use of something other than the decimal point for numbers, for example 325.44.

- **Enable Confirmation Corner**
  Displays icons in the top right corner to close a sketch or exit the sketch and lose my changes.

- **Auto-show PropertyManager**
  When un-checked, SW will not automatically show the PropertyManager when you select a sketch entity while editing a sketch. It will still show up if you simply edit the definition of a feature.

- **Auto-size PropertyManager when panels are split**
  If you already have the FM Tree split, then you can control the size of the PropertyManager when you have a need to edit a feature with it split.

- **Automatically edit macro after recording**
  Visual Basic for Applications automatically starts up after a macro is stopped recording so that you can edit it.

- **Stop VSTA debugger on macro exit**
  Allows for additional debugging when a macro operation is completed.
Enable FeatureXpert
Enables tools to help with errors with fillets and draft

Enable Freeze bar
Allows pull-down from the top of the FeatureManager that stops features from rebuilding to reduce rebuild time or prevent changes to features in the model above the line.

When rebuild error occurs: Prompt, Stop or Continue
If you edit a sketch and cause an error, you get the message to “Show the problem using Check Sketch for Feature” or “Exit the sketch and rebuild anyway”. If you chose the second option, do you want SW to give you the additional option to “Continue (Ignore Error)” or “Stop and Repair”.

Custom property used as component description: Description or other property
Deviate from normal description used in BOM or title block.

Show latest news feeds in task pane
Maybe turn this option off if you don’t have consistent internet access.

Disable additional weldment configuration description strings
Removed additional configuration descriptions for As Welded and As Machined configs when you turn a part into a weldment.

Enable sounds for SolidWorks event: Configure Sounds
Sounds can be assigned for some SW functions (for example: Animation Complete or Mesh Failure)

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**DRAWINGS**

Eliminate duplicate model dimensions on insert
Same option as option in Insert/ Model Items for eliminating duplicate dimensions

Eliminate duplicate model notes on insert
Same thing as dimensions except for Notes from Insert/ Model Items

Mark all part/assembly dimensions for import into drawings by default
Default setting for dimensions for Insert/ Model Items

Automatically scale new drawing views
SW tries to automatically scale views so that a Front, Top, Right and Iso view will fit for the part

Show contents while dragging drawing view
Do you want to see the item as you drag it? Sometimes it may help with performance on large drawings.

Display new detail circles as circles
Display circles or sketched shape
Select hidden entities
   At the drawing level, this may be troublesome at time (just like at the part and assembly level).

Allow auto-update when opening drawings
   Help says that un-checking the option may speed things up, but why wouldn’t you want it to update?

Disable note/dimension inference
   When dragging note, do you want it to snap in-line with other notes? You can hold down the Alt key to disable it temporarily.

Disable note merging when dragging
   Do you want to combine notes by dragging and dropping onto another, or not?

Print out-of-sync water mark
   Detached drawing out-of-sync label or not.

Show reference geometry names in drawings
   Not sure why you would want names of planes or other reference geometry to show up.

Automatically hide components on view creation
   If a part is inside the assembly, then it will get added to list of components to be hidden in the view. I have seen this hurt performance on larger assembly views.

Display sketch arc centerpoints
   Probably more useful at sketch level, but depends on your standards. Centermarks would probably be better to use.

Display sketch entity points
   Display endpoints of sketch lines at drawing level.

Display sketch hatch behind geometry
   Do you want hatch to cover up geometry lines?

Display sketch pictures on sheet behind geometry
   Brings sketch pictures to front or not.

Print breaklines in broken view
   Extend lines past geometry or not.

Save tessellated data for drawings with shaded and draft quality views
   Saves file size if cleared, but views won’t show in View-only mode or in eDrawings.

Automatically populate View Palette with views
   If cleared, then the Insert/ Model View method will be used instead of the View Palette.

Show sheet format dialog on add new sheet
   I like to use this option to insert additional sheets to a drawing, especially if you want to use a different title block for second and subsequent sheets.
- Reduce spacing when dimensions are deleted or edited (add or change tolerance, text etc...)
  When dimensions are deleted, SolidWorks will try to re-arrange other dimensions to use the space. I have seen people not like SolidWorks to move things around like this.
- Reuse view letters from deleted auxiliary, detail and section views
  Deleted letters become available to re-use. I guess it depends on your standards.
- Override quantity column name in Bill of Materials
  Uses the name you enter in Name to Use for quantity in BOM
- Detail view scaling: 2X
  Change it to whatever scale you want the detail views to default to.
- Custom property used as Revision: Revision or other property
  Pick a different property if you want to
- Keyboard movement increment: 10mm
  Pick on a view and use the arrow keys to move it over by a certain amount

**DRAWINGS – Display Style**

- Display style for new views
  Do you want to default to wireframe, HLV, HLR, Shaded with edges or Shaded
- Tangent edges in new views
  Options are Visible, Use Font (with or without hidden ends) or Removed
- Edge quality for wireframe and hidden views
  High quality (model resolved for greater precision) or Draft quality (faster performance for large assemblies)
- Edge quality for shaded edge views
  High quality (model resolved for greater precision) or Draft quality (faster performance for large assemblies)

**DRAWINGS – Area Hatch/Fill**

- None, Solid or Hatch
  Pick a style with scale and orientation angle
Colors

- Current color scheme
  Blue, Green or Orange highlight schemes (or pick a custom one)

- Color scheme settings
  Set colors for many different items (just be careful not to have any conflicting ones
  Popular ones: Inactive entities for color of title block lines
  Driven versus Imported dimensions

- Background appearance
  Document scene, Plain (Viewport background), Gradient or Image file

- Use specified color for drawings paper color (disable image in sheet background)
  You probably don’t want image on the production drawing level

- Use specified color for Shaded with Edges mode
  Maybe a different color besides black to emphasize edges

- Use specified colors when editing parts in assemblies
  Depends on which option for chose for display of other parts when editing one at the assembly level

- Use specified color for changed drawing dimensions on open
  New option from a few years ago to show what dimensions have changed since the drawing was last opened

- Envelopes: Semi transparent, Opaque, Do not change
  Similar to editing parts at the assembly level, but how do you want the envelopes to be displayed

Sketch

- Auto-rotate view normal to sketch plane on sketch creation
  Newer option that automatically rotates you normal to plane you are sketching on and changes to isometric when creating a feature

- Use fully defined sketches
  A good habit to have, but maybe not all the time

- Display arc centerpoints in part/assembly sketches
  Good to have on while sketching, but maybe not at the drawing level

- Display entity points in part/assembly sketches
  Controls visibility of endpoints of lines and arc, but I don’t know how people can work with this option off

- Prompt to close sketch
  May be a good option to turn off if you use open profile sketches a lot

- Create sketch on new part
  You normally need a sketch started at first, but it may not always need to be on the Front plane
Override dimensions on drag/move
Almost like Instant 3D, but at the sketch level

Display plane when shaded
Some people like to see front and back of planes

Line length measures between virtual sharps in 3D
Takes fillet into consideration when measuring lines

Enable spline tangency and curvature handles
Maybe remove some of the controls for novice users, but these are very powerful when left on

Show spline control polygon by default
Some like to use control polygon to define shape, but some find more difficult to use

Ghost image on drag
Preview of where you currently have the object moved to

Show curvature comb bounding curve
Spline option, but may be a good option to turn on as needed

Enable on screen numeric input on entity creation
Do you want to type in dimension value as you are dimensioning it

Create dimension only when value is entered
Do you only want the dimension to stay when you actually type something in

Over defining dimensions
Prompt to set driven state

Set driven by default
How do you want over-defining or references dimension to be show and what messages to pop up

Sketch – Relations/Snaps

Enable snapping
Why wouldn’t you want to snap (you can hold down Cntrl key to temporarily disable)

Snap to model geometry
Will still snap to sketch geometry but not underlying features

Automatic relations
If you don’t want horizontal and vertical type relations while sketching
Sketch snaps (most are self-explanatory)
- End points and sketch points
- Center points
- Mid-points
- Quadrant points
- Intersections
- Nearest
- Tangent
- Perpendicular
- Parallel
- Horizontal/vertical lines
- Horizontal/vertical to points
- Length – Really snaps to grid
- Grid – Will snap to grid even if visibility is turned off
- Angle – You can draw lines in any direction, but different angles will show up as a snap

Display/Selection

- Default bulk selection method: Lasso or Box
  Lasso is new to 2014
- Hidden edges displayed as: Solid/Dashed
  Hidden edges for me have always been dashed, but if you go with the solid option, then they are gray edges by default.
- Selection of hidden edges
  Allow select in wireframe and HLV modes
  Allow selection in HLR and shaded modes
  I like the first option where in Hidden Lines Visible and Wireframe, where it makes sense to pick the edges since you can see them, but the option to pick them when they aren’t visible like in Shaded and HLR is almost like selecting option in the Fillet tool to select hidden edges, but all the time.
- Part/Assembly tangent edge display
  As visible
  As phantom
  Removed
  My default is “As visible” so I can see the filleted edges, but I have seen a few people prefer the other two options.
Edge displayed in shaded with edges mode

HLR or Wireframe

I am not sure why you would want all of the edges displayed (wireframe option) when the model is shaded. New in 2014 is option for Optimize for thin parts to minimize bleed thru of edges.

Assembly transparency for in-context edit

Opaque assembly
Maintain assembly transparency
Force assembly transparency

I prefer leaving the assembly display as it is with the opaque option and hiding and showing components as needed.

Anti-aliasing

None
Anti-alias edges/sketches only
Full scene anti-aliasing

“Full scene anti-aliasing” is grayed out unless your video card supports full scene anti-aliasing and has passed a stability test. You must set the graphics card control panel settings for anti-aliasing so that the application has control. It applies anti-aliasing to the entire graphics area for parts and assemblies. Anti-aliasing is disabled during zoom, pan, and rotate.

Highlight all edges of features selected in graphics view

Sometimes, it is nice to see all the edges affected by the feature, but sometimes too much is highlighted to really tell much.

Dynamic highlight from graphics view

When you hover over the part in the FeatureManager tree, do you want to have a bounding box for the part to show up?

Show open edges of surfaces in different color

Makes it nice on imported assemblies to find parts that came in as surface bodies and not solids

Display shaded planes

Might as well shade them if you are going to show them

Enable selection through transparency

If a component is behind the transparent part, then you will get that part, but makes it a little tough if you really want to select the transparent one. Also, you can hold down the CNTRL key and then select the transparent part.

Display reference triad

Who doesn’t need the XYZ triad in the bottom left corner, even if you don’t use it to rotate the views.

Display scrollbars in graphics view

You have to close all open files to change this option

Display draft quality ambient occlusion

Only uses draft version when you turn on ambient occlusion
Display SpeedPak graphics circle
- It either shows the ghosting circle or displays the Hide/Show circle instead

Display pattern information tooltips
- Gives expanded information about pattern when you hover over feature in FM-Tree

Projection type for four view viewport
- You probably want Third Angle for Front, Right, Top and Trimetric

Display dimensions flat to screen
- Nice when editing a sketch in an isometric view

Display notes flat to screen
- Similar to the display of dimensions when you rotate the screen

Performance

Verification on rebuild (enable advanced body checking)
- Most of the time you may not need this, but it does catch some underlying issues

Ignore self-intersection for some sheet metal features
- Some common edge issues in flat patterns may not really be an issue

Transparency
- High quality for normal view mode – Higher quality when the file is standing still
- High quality for dynamic view mode – Better quality when the file is rotating
- Transparency in larger, complex files may really affect performance

Curvature generation
- Only on demand – Uses less memory, but initially slower
- Always (for every shaded mode) – Faster at first, but uses more memory as you go along

Level of detail:
- More (slower) to Less (faster)
  - Controls level of detail during dynamic rotating of the file

Assemblies

Automatically load components lightweight
- Less memory required to open files

Always resolve subassemblies
- Do you want a little more info about sub-assemblies loaded or not
Check out-of-date lightweight components:
  Don’t check
  Indicate
  Always  Resolve
  If components have been changed since the last time that the assembly has been opened, then you will have options about how to work with those components

Resolve lightweight components
  Prompt OR Always
  Some operations will need more information to be loaded for the file

Rebuild assembly on load
  Prompt
  Always
  Never
  Option to rebuild assembly so the components are up-to-date

Mate animation speed
  Off-Fast to slow
  Do you want to have the parts slide into place?

SmartMate sensitivity
  Off-Fast to slow
  How long do you need to hover over second part to activate SmartMate

Purge cached configuration data
  Can help reduce file size and save time

Update mass properties while saving document
  May slow things down with larger assemblies

Use shaded preview
  Solid looking and higher level of detail when you are creating a feature and haven’t accepted it yet

Use software OpenGL
  A good tool to check graphics card and driver combinations, but it can’t be checked if files are open

No preview during open (faster)
  Interactive preview when opening models that can slow things down

Assemblies

Move components by dragging
  Maybe we wouldn’t want this if we imported a large assembly and components weren’t mated together
Prompt before changing mate alignment on edit
Should SolidWorks ask if there are changes to parts that might need to change the alignment of a mate.

Save new components to external files
The idea of in-context parts being stored inside the actual assembly until ready to be saved to an actual file, or start them off as a file on your hard drive right from the beginning.

Update model graphics when saving files
Helps kept everything up-to-date when you save.

Large assemblies
Use Large Assembly Mode to improve performance whenever working with an assembly containing more than this number of components:
If every option is SolidWorks was this verbose, we would need another monitor for the Options. But this option is coupled with the check boxes below to select which things happen when you hit this component limit.

When Large Assembly Mode is active:
Do not save auto recover info
You may not want to do that with larger assemblies anyway. Save often.
Hide all planes, axes, sketches, curves, annotations, etc.
Almost like turning off accessories that you don’t need.
Do not display edges in shaded mode
Changes back to regular shaded mode
Do not rebuild when switching to assembly window
When switching back from editing a component

Use Large Design Review whenever working with an assembly containing more than this number of components:
Moves into the next level of large assembly editing, and examination tools when we hit a certain number of parts.

When Large Design Review is active:
Automatically check and update all components
Almost like performing a rebuild before we start looking at a snap-shot of the assembly.

Envelope Components
Automatically load lightweight
Loads smaller data subset for parts in envelope
Load read-only
Makes it where we can’t change envelope parts accidentally
External References

- Open referenced documents with read-only access
  You will only have write access to the top level assembly
- Don’t prompt to save read-only referenced documents (discard changes)
  One of several message to help you with read-only files
- Allow multiple contexts for parts when editing in assembly
  Allows for you to make in-context relations to more than one assembly
- Load referenced documents
  Prompt
  All
  None
  Changed Only
  Choose whether to load the referenced documents when you open a part that is derived from another document
- Search file locations for external references
  Searches Tools/ Options/ File Locations when check, but list will be ignored if de-selected
- Update out-of-date linked design tables to:
  Prompt
  Model
  Excel file
  What happens to linked values and parameters if the model and design table are out-of-sync

Assemblies

- Automatically generate names for referenced geometry
  When the option is off, you can mate to read-only parts and use Face IDs of the parts. Use this option if you plan to use component replacement
- Update component names when documents are replaced
  Click only if you want to manually define component name by going thru Component Properties
- Do not create references to the model
  Blocks the ability to create in-context relations
- Show “x” in feature tree for broken external references
  Just a graphical display in FM Tree if you want it
**Default Templates**

- File locations for which template to use for operations like File/Import, Mirror Part, adding a sheet by clicking on tab at the drawing level and other times when SolidWorks does not prompt for a template.
  - Radio button option to prompt user to select document template will skip the chosen template.

**File Locations**

- Add/change locations for storing anything from document templates, BOM, templates or even textures.
  - It may be safer to store custom files in some other location than C: \ Program Files \ SolidWorks so they won’t get deleted when SolidWorks is un-installed and new version is loaded.

**FeatureManager**

- Scroll selected item into view.
  - Nice sometimes when trying to identify feature or specific part in FM Tree, but not always desired if you are trying to keep the FM Tree from moving around.

- Name feature on creation.
  - If you know you are going to rename the feature anyway, then this will save time.

- Arrow key navigation.
  - Use arrow keys to scroll up and down thru FM Tree.

- Dynamic highlight.
  - Another way to identify features in the graphics area when you hover or features in FM Tree.

- Use transparent flyout FeatureManager tree in parts/assemblies.
  - Allows you to be able to see thru the flyout or leave it opaque.

- Enable FeatureManager tree filter.
  - Controls availability of search tool at top of FM tree, but will be grayed out if any files are open.
Spin Box Increments

- Length increments (both in English and metric units)
- Angle increments
- Time increments
  Default values for spin box that includes distance (for example inches or millimeters) degrees and seconds

View

- Reverse mouse wheel zoom direction
  For this people with experience with other CAD packages that use an opposite zoom standard
- Zoom to fit when changing to standard views
  Do you want to keep the zoom level when you change between different views

View rotation

- Arrow keys
  Default degrees of rotation
- Mouse speed: Slow to fast

Transitions:

- View transitions
  Off, Slow to fast
- Hide/show components
  Off, Slow to fast
- Isolate
  Off, Slow to fast
- View selector
  Off, Slow to fast
  How fast do you want each of these transitions to happen

Backup/Recover

Auto-recover

- Save auto-recover information every ? minutes
  Do you want and how often to save file information in-case SolidWorks or system craches
Backup

- Number of backup copies per document
- Backup folder location
- Save backup files in the same location as the original
- Remove backups older than ? days

  When file is saved after initial save, then original is copied over to defined folder and “Backup# of” is added to front of file name. If you haven’t needed the file after a certain period of time, then why not go ahead and delete it.

Save notification

- Show reminder if document has not been saved for ? minutes
- Automatically dismiss after ? seconds

  If you need help remembering to save your files, then you can also specify the amount of time that the message stays on the screen.

Touch

Rotate and Pan

- Rotate width
  - Off, Smaller to Larger
- Rotate versus pan threshold
  - Off, Smaller to Larger

  How do you want your touch pad to behave?

Hole Wizard/Toolbox

- Hole Wizard and Toolbox folder
- Make this folder the default search location for Toolbox components
  
  Set file storage location and whether this is the default location to search for Toolbox files
- Configure...
  
  Button to go in and customize sizes and setting for Toolbox and Hole Wizard files
**File Explorer**

- **Show in File Explorer view**
  - My Documents
  - My Computer
  - My Network Places
  - Recent Documents
  - Hidden referenced documents
  - Samples
- **Restore File Associations**
  - Use this option to reset file association for SolidWorks file to a specific version of SolidWorks

**Search**

- **Show SolidWorks search box**
  - Turns on/off search box in upper right corner of program

**File and Model search**

- **Search while typing (incremental search)**
- **Include 3D ContentCentral results**
- **Results per page**
- **Maximum results per data source (independent of 3D ContentCentral)**
  - Options for searching as you type and how the results are displayed
- **Indexing performance**
  - Index only when computer is idle
  - Always index (may slow SolidWorks)
  - People usually choose the first option and then schedule it below at a time when they know that the computer isn’t being used. Computer must remain turned on.

**Dissection**

- **Schedule dissection daily to automatically dissect files in search paths**
  - (Dissection does not modify or resave your files)
  - Daily start time
  - Daily stop time
  - Location
  - Dissection of parts will provide features of parts to also be included in search results
**Collaboration**

- Enable multi-user environment
  - Add shortcut menu items for multi-user environment
  - Check if files opened re-only have been modified by other users
    - Check files every ? minutes
  - Note: Lightweight components will not be checked
    - This is an interesting option if multiple people are working on the same assembly. If someone makes a change to a part and you have it opened as read-only, then you can get notified that they have changed and saved it.

**Messages/Errors/Warning**

- Show errors every rebuild
  - Normally a good option, but one that might be turned off if there are a large number of errors.
- Warn before saving documents with update errors
  - Nice option to keep you from accidentally missing an error
- Warn starting a sketch in the context of an assembly
  - Just a message in case you don’t want to make a part in-context
- Display FeatureManager tree warnings
  - Always, Never, All but top level
    - This is for warnings only and not errors
- Display circular references in equations
  - Everywhere, In equation dialog only, Never
    - Not sure why you wouldn’t want to be warned about a circular reference
- Display potential circular references in equations
  - Everywhere, In equation dialog only, Never
    - Options in equation tool may help you with re-ordering equations to avoid this problem
- Dismissed messages: Checked messages will be show again
  - Whenever you see a dialogue box and you check the box to “Not show again”, then SolidWorks will stop displaying those messages. If you want to see the messages again, you can bring them back to life by checking them in this box.