

SolidWorks Tips & Tricks Part 1



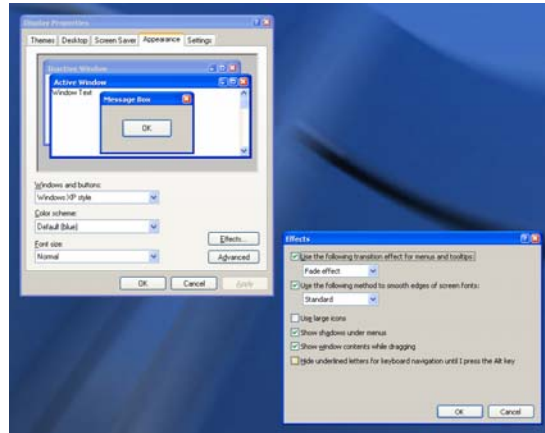
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Special Thanks To:
Adam Smith
Product Creation Studio



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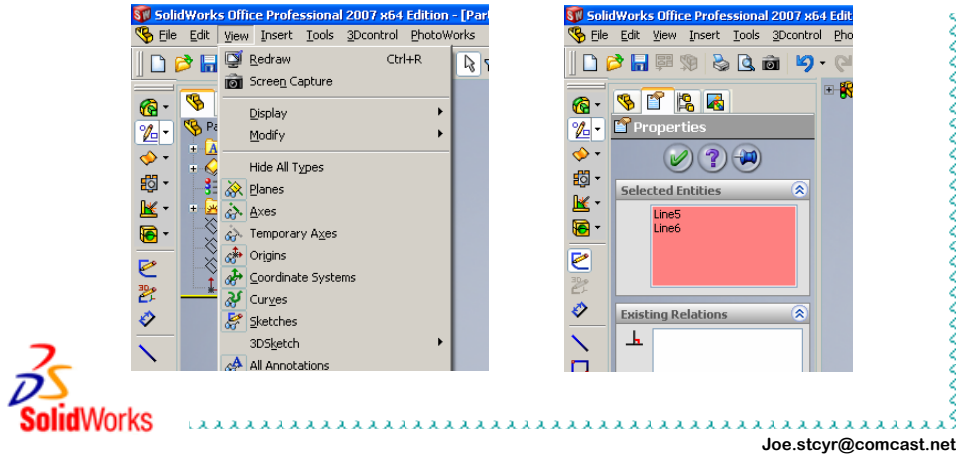
- Uncheck “Hide underlined” in Windows Display Properties
 - Shows keyboard shortcuts in most applications



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R-Click Windows Desktop - Properties - Appearance - Effects

- Alt + Keyboard Strokes
 - Quick access to underlined commands
 - Great for R-click menu (r-click then press letter key only)
 - Quick sketch relations



Hold Alt Key down while typing shortcut keys to menu selections

- How Do I Select My Stuff?
 - Box selection from left to right
 - Cross selection from right to left
 - Shift selects everything within the box
 - Ctrl inverts the current selection in box



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Left Mouse Drag to create selection box. Right to left drag creates selection within drag box. Left to Right drag creates selection within and crossing box.

- Dynamic Line/Arc Creation
 - Use the line tool to create lines and arcs
 - Hover to the beginning of the active segment to switch to arc
 - Press “A” key to alternate between line and arc dynamically



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Using line tool, Draw line segment then start second segment but hover back to beginning of second segment to automatically start arc creation. Hover away in the direction of tangency required. Pressing the "A" key while creating a segment Alternates between line and arc creation

- Merge Arc into Circle or Partial Ellipse into Ellipse
 - No need to delete arc and create new circle
 - Maintain downstream relations



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Select open endpoints of Arc or Partial ellipse. Add merge relationship to make complete

- Quick sketch group copy or move
 - Window or Ctrl-select sketch data
 - Hold Ctrl then left drag vertex to start copy
 - Release vertex to copy
 - Release Ctrl then vertex to move



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hold Ctrl before and while dragging sketch segment to copy. Release Ctrl before releasing Segment to move freely without copy

- Wake Up Sleeping Geometry for Dynamic Sketch Relations
 - Create parallel, perpendicular, etc. relations to remote geometry



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While creating sketch segment, hover over geometries to wake up automatic relations to hibernating geometries

- Momentary Inference Free Sketching
 - Create or drag geometry without snapping
 - Hold Ctrl before creating or after dragging to disable inferencing



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Hold Ctrl while creating sketch geometry, or hold ctrl after dragging but before dropping sketch geometry, to eliminate inferencing

- Quick Sketch Drag Undo
 - Left drag geometry, then right click while left dragging to cancel



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r-click while left dragging sketch geometry to let it snap back into original position

- Spline Handlebars for Smooth Edit
 - Horizontal and vertical construction lines on spline points
 - Allows no inference precision dragging of spline points



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Place Horizontal and Vertical Construction Lines on spline points to create drag handlebars for smooth control. (drag lines not points)

- Quick Slot or Cam Profiles
 - Create slot centerline
 - Use offset entities, bi-directional, cap ends to create closed profile
 - Select multiple centerline geometries to speed creation



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Use Offset entities - Bidirectional - Cap ends - in sketches to create slot shapes from lines, arcs and splines.

- Easy Sketch Geometry Manipulation
 - Select geometry to move, copy, rotate, etc.
 - R-click create block



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Create sketch blocks to group geometries for easy moving or copying

- Save Now, Save Often
 - Save while in rollback
 - Save while in sketch
 - Did I mention you should save?



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You can save while in rollback mode only in SW 2007

- That's What I Wanted to Do / Undo
 - The undo barrier has been removed



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You can undo sketch changes after exiting a sketch.

- Hollow model creation
 - Create shell feature without removing a face



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- Split Part with Parting Surface into Multiple Bodies
 - Create appropriate lip and groove parting surface
 - Use split feature to separate the part into logical pieces



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Use parting surface to split bodies into multiple pieces without material removal, do not use save section of split feature, just split part

- Quick and easy assembly clearance
 - Build or split pieces nominal
 - Use move face to create appropriate clearances



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Use Move face feature to create manufacturing tolerance between parts

- Robust Multi-Body Part Break-Out
 - Create a new part with the appropriate properties (part no. etc)
 - Insert part within this part as the first feature
 - Delete bodies to isolate the appropriate body



NOTE Save New Parts A/B



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Save Bodies feature is not fool proof!! Do not use the Save bodies feature for downstream data creation. Instead create a new part and use insert part to use the master part as the first feature, then immediately use delete bodies feature to eliminate the extra bodies. This method can be edited and managed without data loss. The Save bodies feature cannot.

- Quick Multi-Body Re-Assembly
 - Drag and drop components onto graphics origin (show origins)
 - Drag and drop components into feature tree
 - Insert component into feature tree



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For Split parts (either method) origins match in geometry space. For quick reassembly of split parts drag parts onto origin from windows. (be sure to show origins so you can see where to drop your parts in the assembly.)

- Variable Chamfer
 - Create a variable radius fillet
 - Loft between the fillet boundaries
 - Replace the fillet faces with the new lofted surface



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Use a variable radius to create the boundaries for a variable chamfer. Then delete the radius faces and replace them with a loft surface using replace face or extend and knit into solid

Thank You!



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