# SOLIDWORKS 2019 Keyboard/Mouse Shortcut Reference

## SOLIDWORKS Commands

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## Windows Commands

- New: Ctrl + N
- Open: Ctrl + O
- Print: Ctrl + P
- Save: Ctrl + S
- Close: Ctrl + W
- Copy: Ctrl + C
- Paste: Ctrl + V
- Cut: Ctrl + X
- Redo: Ctrl + Y
- Undo: Ctrl + Z
- Close File: Ctrl + F4
- Close SW: Alt + F4
- Select All: Ctrl + A
- Find/Replace: Ctrl + E
- Swap Window: Ctrl + Tab

## Mouse Scroll Wheel and Button (MMB)

- Zoom to Pointer: MMB Scroll
- Rotate (Pan Drawing): MMB
- Pan: MMB + CTRL
- Zoom to Center: MMB + Shift
- Rotate Normal to Screen: MMB + Alt

## Mouse Gestures: Drag RMB in one of 2 to 12 Directions

- Part/Assembly
- Sketch
- Drawing

To Modify Keyboard Shortcuts and Mouse Gestures: RMB any toolbar, click Customize..., Keyboard tab
LMB = Left Mouse Button, RMB = Right Mouse Button, MMB = Middle Mouse Button (press down the scroll wheel)
*Command Search Option can be enabled in System Options > General

**Potentially Problematic**
- Filter Edges: E
- Filter Vertices: V
- Filter Faces: X
Order of Operations for Creating a Feature:
1. Pick a Plane and Select it
2. Start the Sketch (Sketch Command)
3. Orient ‘Normal To’ the sketch (helps to see what you are doing)
4. Sketch the general Shape
5. Fully Define the Sketch
   a. Add Relations
   b. Add Dimensions
   c. If still under-defined, pull blue end points to discover freedoms and inaccurate movement. Repeat.
6. Create Feature, checking the direction in an isometric view
7. Edit according to the specific issue: Sketch
   a. If there are Dangling Entities:
      i. Click the dangling sketch entities
      ii. Look for small read square
      iii. Drag red square to new appropriate entity
      -OR-
      iv. Delete dangling references and recreate them
   b. If Sketch is Over-Defined
      i. Click on ‘Over-Defined’ at the bottom right of the SOLIDWORKS window to start the SketchXpert
      ii. Click ‘Diagnose’ to search for possible solutions
      iii. Browse the suggestions and choose the best option
      iv. Accept the solution and click OK
   c. If the Sketch is Invalid (open and closed contour for instance)
      i. Repair Sketch on the sketch toolbar
      ii. Search for problems, use the magnifying glass
   d. If the sketch is invalid for the feature type
      i. Tools > Sketch Tools > check Sketch for Feature

Generalized Order of Operations for Creating a Model (finish each step before proceeding to the next):
1. Add – Add all material needed for the part
2. Cut – Cut away excess material to get the final overall shape
3. Holes – Add holes to the finished shape
4. Fillets – Add detail features, such as fillets and chamfers

NOTE: This is an efficient order of operations, but there will always be exceptions to the rule

Fixing Rebuild Errors
1. Open the part that is problematic
2. Start at the top of the feature tree, work down in order
3. Right click the feature – What’s Wrong?
4. Read the error, find out the following:
   a. Where is the problem: Feature, Sketch, or Sketch Plane?
5. Edit according to the specific Issue: Feature
   a. Redefine the feature if selections are lost
   b. Verify the feature properties
   c. Verify no invalid conditions exist
6. Edit according to the specific issue: Sketch Plane
   a. Redefine the Sketch Plane

I’m STUCK! What do I do now?
Help File – Useful to determine known limitations and intended functionality (Full of useful information, seriously!)
   - Start your command or show the dialog, then click ‘Help’ or the ‘?’
   - Hit ‘F1’ and search for keywords

SOLIDWORKS.com Knowledge Base – Useful for errors and possible bugs
   - Go to SOLIDWORKS.com
   - Click ‘Login’ for the Customer Portal
   - Login or create an account (use your serial number to register your account)
   - Click Knowledge Base
   - Search for keywords, such as ‘error’ or ‘message text.’

My.SOLIDWORKS.com – Single platform for community interaction and tools.
   - Get models from 3D Content Central
   - Video Tutorials
   - SOLIDWORKS Forums
   - SOLIDWORKS News

SOLIDWORKS Forums – Useful to get ideas from the large user base
   - Forum. SOLIDWORKS.com
   - Log in with your Customer Portal account to participate!

MLC CAD YouTube Channel – Great for how-to videos, informational videos, webinars, and tech tips.

myMLCservices.com – Great for how-to videos, news from the MLC CAD Team, and much more!

MLC CAD SOLIDWORKS Support – Great for how-to advice, bugs, stability issues, anything related to SOLIDWORKS
   - Send an email to SOLIDWORKSsupport@mlc-cad.com, you should receive an automated response email and someone will call you back
   - Call 800-364-1652 Option 8
   - Call or email your local office if you prefer, www.mlc-cad.com has contact information