Ben Gerber
21 November, 2017

Solidworks 2012 Assembly Features Reference Sheet

For all:
1. Select the part you want to perform the operation *on.*
2. Click "Edit Component"

- Round/Fillet – Rounds of a face
   1. Navigate to "Insert > Features > Round/Fillet"
   2. Select face to round. Edges and vertexes throw errors, but they can be fixed by envoking the "FilletXpert" on the left side at the top of the Fillet pane.
   3. Adjust edge radius to desired arc
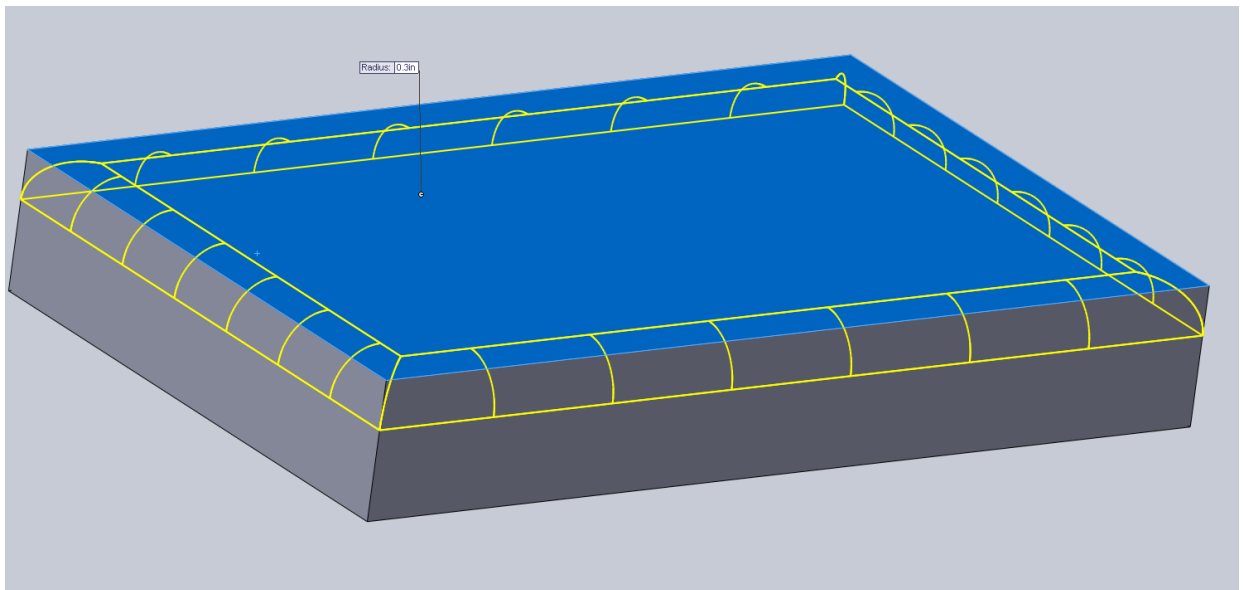
   - NOTE: Clicking "Full Preview" will render a wireframe preview of the portions you are rounding.
   4. Click green arrow in upper left to complete rounding
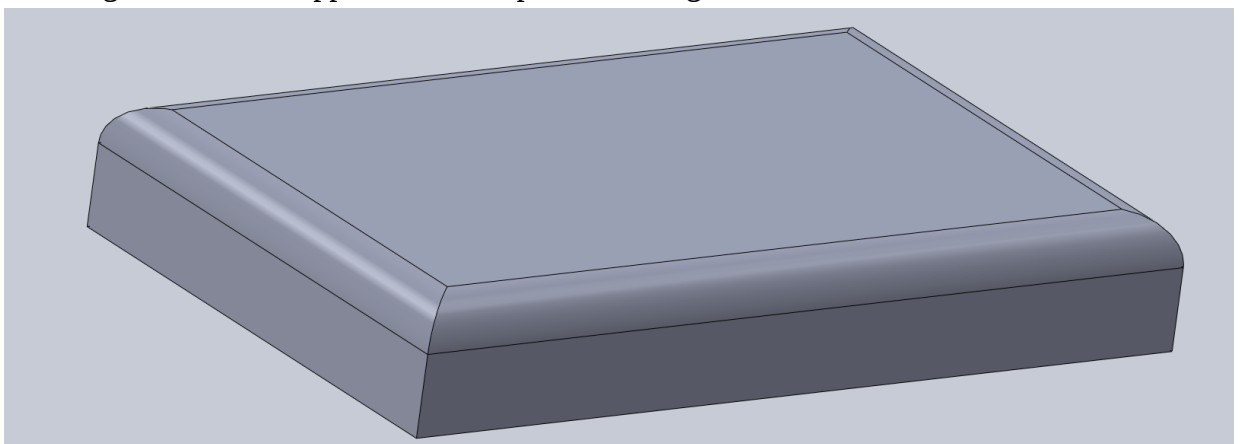
- Chamfer – Cuts a plane out of a shape
  1. Navigate to "Insert > Features > Chamfer"
  2. Select Edge or face to perform chamfer on
  3. Adjust distance on left side, and adjust angle of cut to desired angle
  4. Click green check to complete cut

- Hole – Creates a circular hole through an object aligned with a face
  1. Navigate to "Insert > Features > Hole > Simple Hole"
  2. Click on the place on the face on which you want to create the hole.
  3. Adjust hole depth and radius from left hand side
  4. Click green check to complete hole
- Draft – Pulls a face out with relation to another face
  1. Navigate to "Insert > Features > Draft"
  2. Select two faces which you want to pull in relation to each other. The first selection is your non-moving face, and the second is the face to be pulled.
 3. Press Apply to complete the draft. You can do multiple drafts in one session.
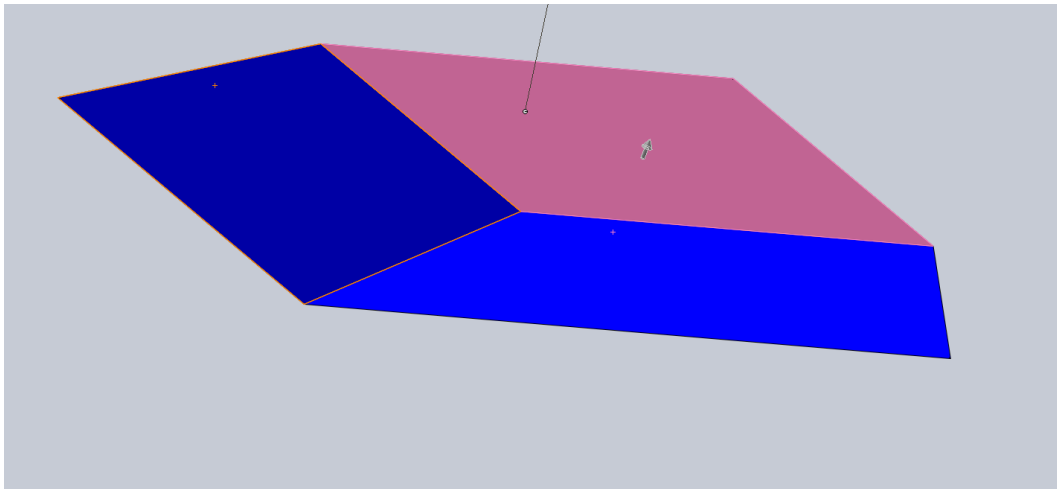
  - ○ NOTE: All drafts will by applied with "Apply," not just previewed.
  4. Press green check on left to complete drafting

- Shell – Hollows out a part
  1. Select your shape that you wish to hollow out
  2. Navigate to "Insert > Features > Shell"
  3. Select "Show Preview" on the left hand side
  4. At this point you can select faces. Whatever faces you select will be hollowed all the way to the edge as in figure.
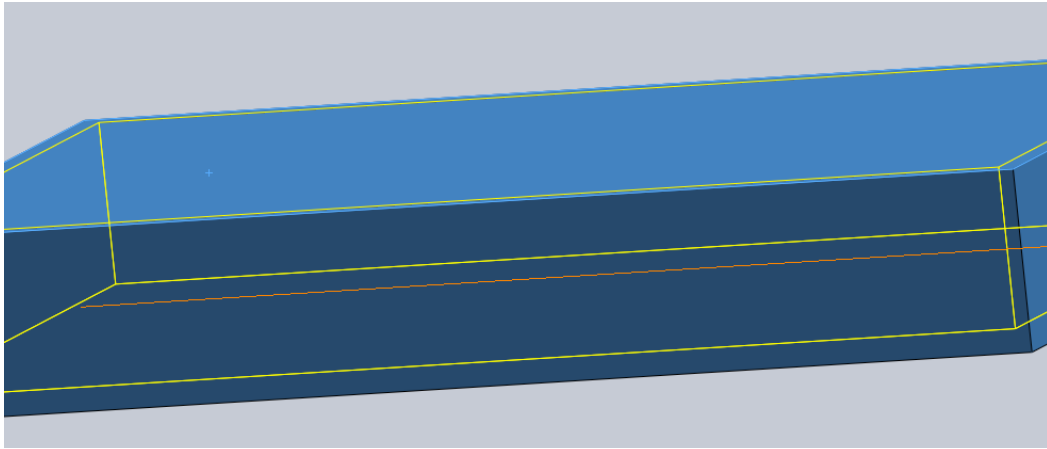
5. You can select "Shell Outward" on the left to create a shell around the part. Selecting a face here will not create a shell on that face as shown in figure.
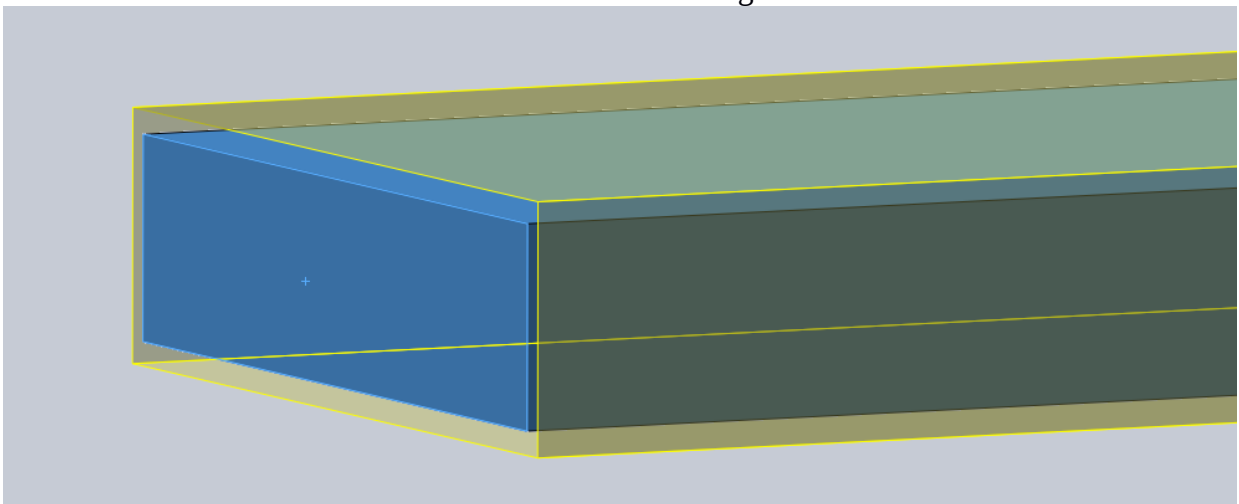
6. Click the green check on left to complete shell as shown in figure.
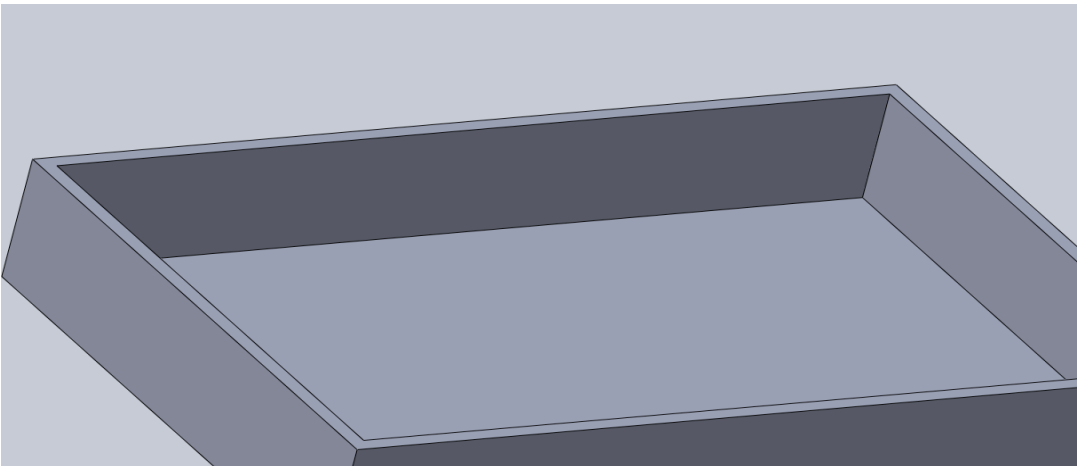
- Rib – Creates a support rib to help with structural integrity of part of assembly.
    1. For this, you need to sketch an external line in a plane perpendicular to whatever you want to put the rib on
    2. Navigate to "Insert > Features > Rib"
    3. Use this guide to set correct properties:
       http://help.solidworks.com/2012/English/SolidWorks/sldworks/HIDD_FEAT_RIB.htm

- Scale – Precisely increase or decrease overall size of part without changing geometry
  1. Click on the component you want scaled on the left hand side of the screen.
  2. Navigate to "Insert > Features > Scale"
  3. Type in your scaling constant (0-1 for smaller, >1 for bigger).
  4. Click green check mark to finish scaling

- Dome – Creates a uniform curved surface on a given object face
  1. Navigate to "Insert > Features > Dome"
  2. Click on the face(s) to be domed
  3. Type in value of dome apex distance from face origin
     NOTE: You can also specify constrain point and direction of the dome
  4. Click green check to finish dome

- Freeform – Creates a grid that can be used to create polygons, curved, etc on a given face to create complex ergonomic shapes.
  - This is a complex feature that is best describes here:
   http://help.solidworks.com/2012/English/SolidWorks/sldworks/Freeform.htm

- Deform – A Complex feature that allows you to deform a complex surface in multiple ways
  - This complex feature is best described here:
    http://help.solidworks.com/2012/English/SolidWorks/sldworks/HIDD_DVE_DEFORM.htm

- Indent – Similar to cavity except the indent takes away everything past a certain point
  1. Navigate to "Insert > Features > Indent"
  2. Select the target body (body receiving the indent)
  3. Click on the tool body (part doing the indenting)
  4. Check "Cut to preview the indent in wireframe
   NOTE: You can choose reversing parameters
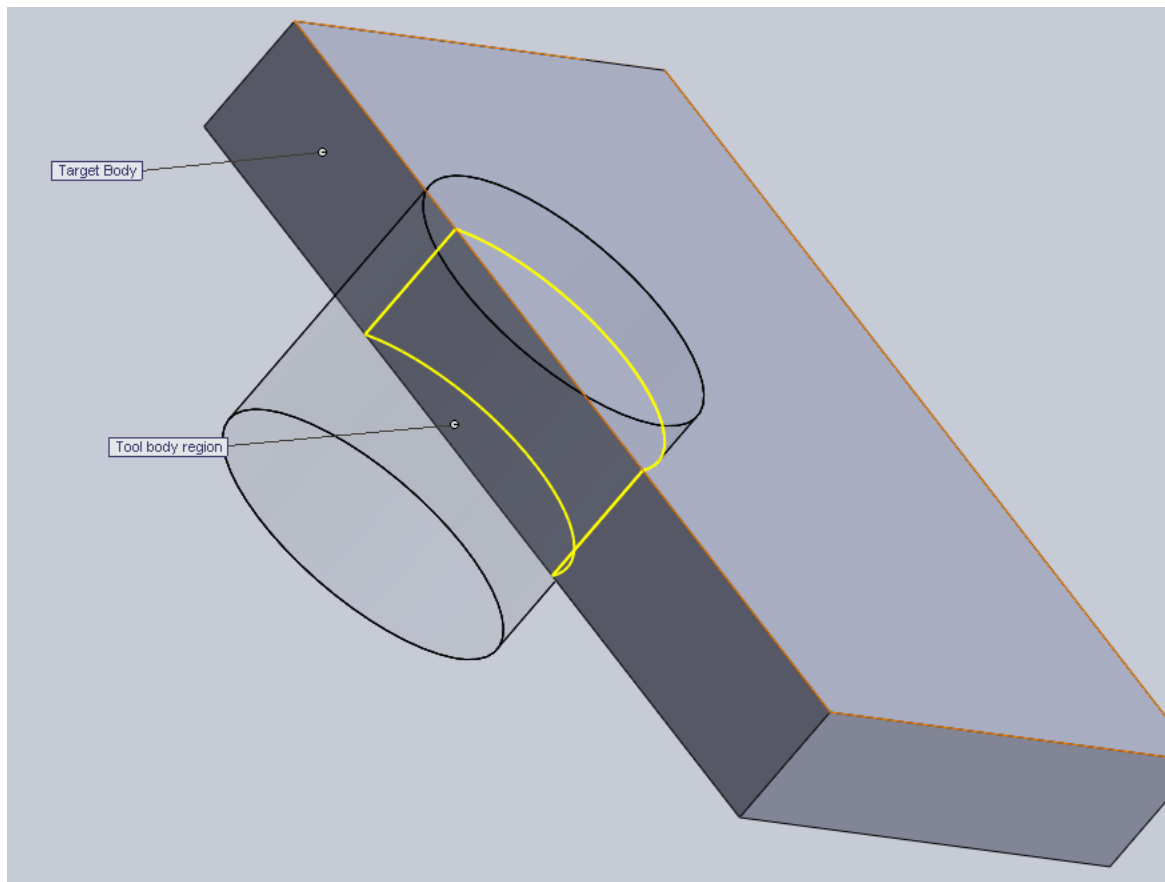
5. Click green check to complete indent

- Flex
  1. Click on the item to be flexed
  2. Navigate to "Insert > Features > Flex"
  3. Select a Category on the left
  4. Bending
     - The bend goal point is denoted by a red solid plane (Trim Plane 2)
   - Move green solid plane (Trim Plane 1) to define bend start point
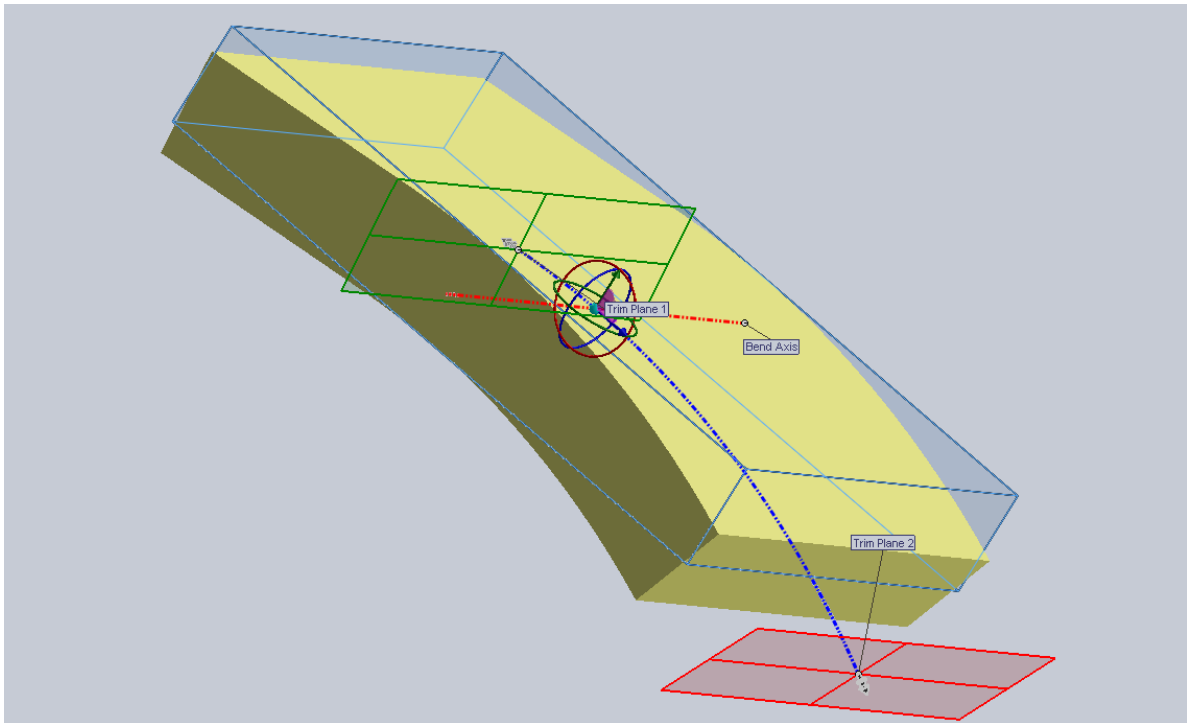
- ○ Use angle and distance to arc to change how it bends
5. Twisting
    - ○ The Twist goal point is denoted by Trim Plane 2
    - ○ Start point defined by green plane (Trim Plane 1)
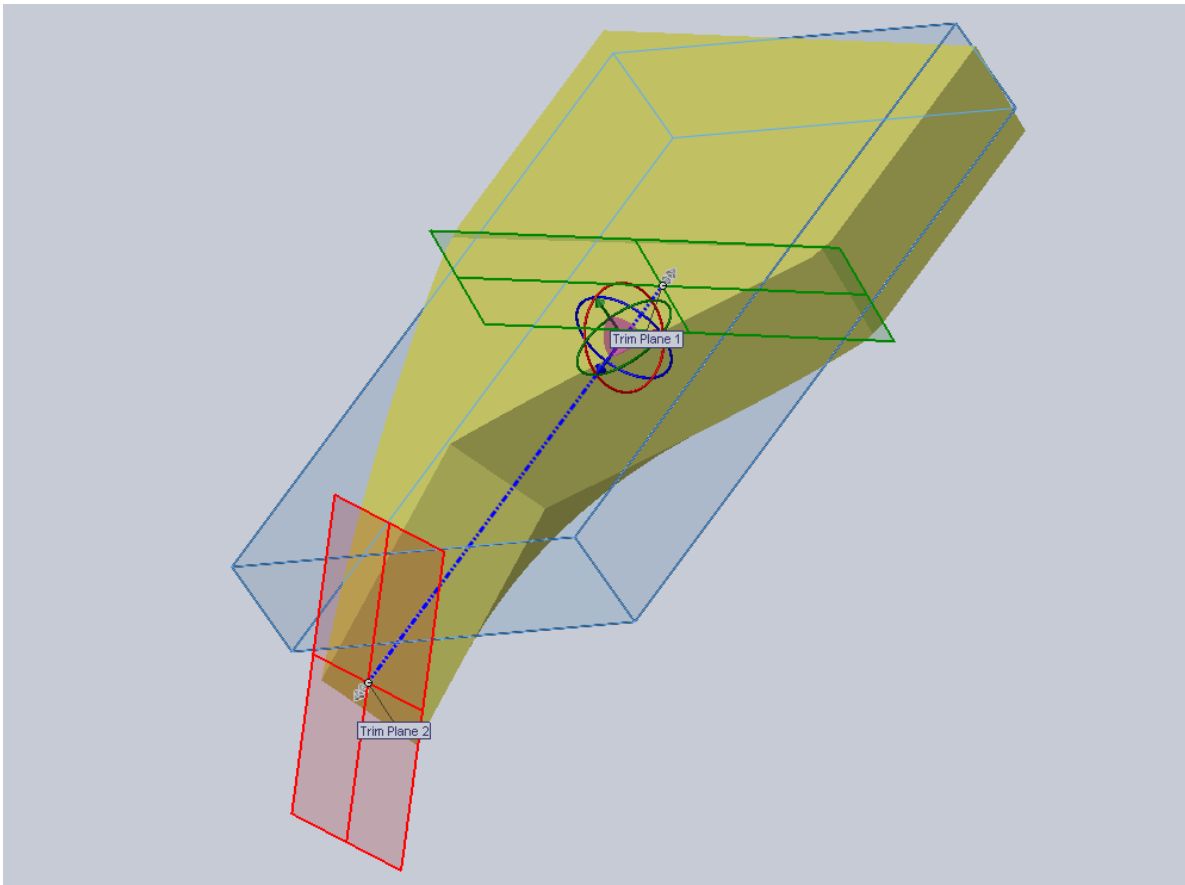
- ○ Use angle to change the twist angle and move the end point to change properties
6. Tapering
   - ○ Use Trim Plane 2 to adjust end of taper
   - ○ Use Trim Plane 1 To adjust start of taper
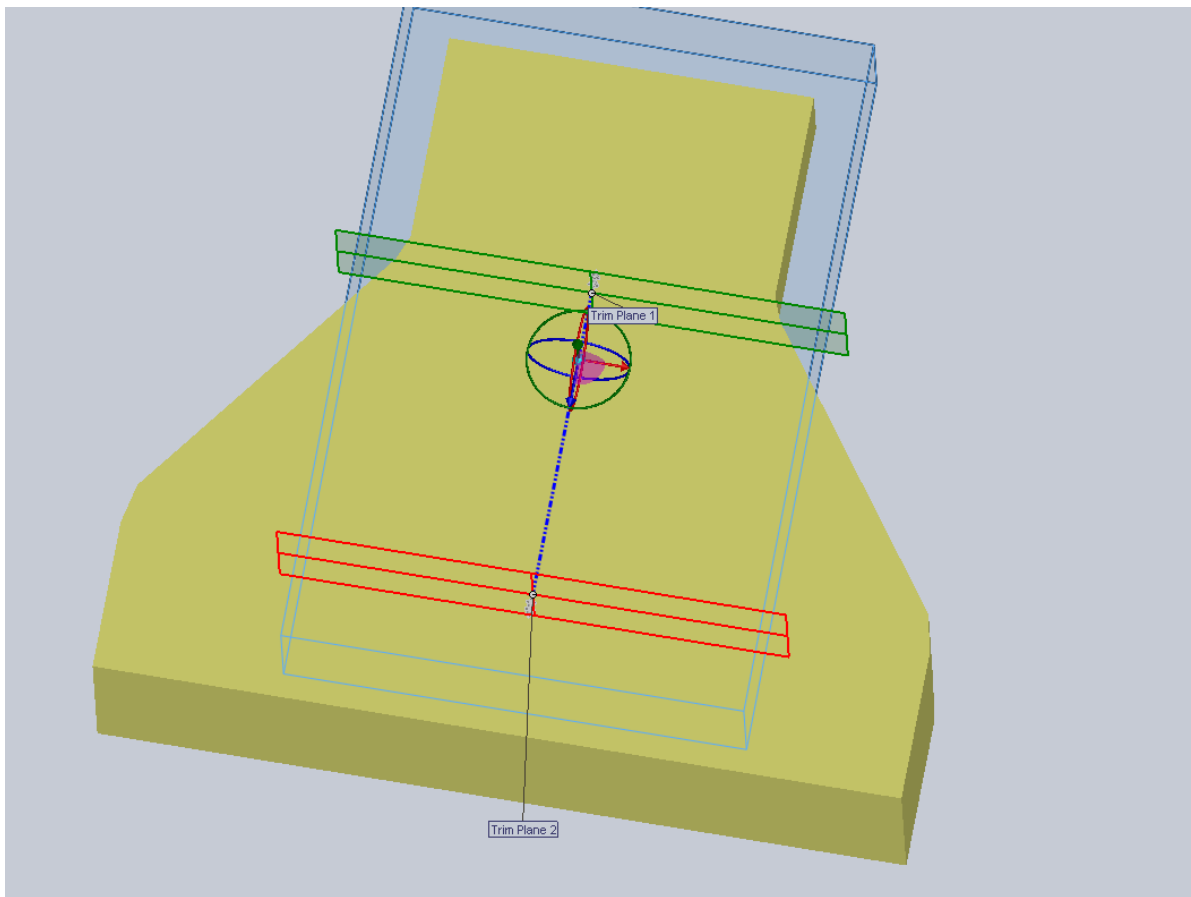
- ○ Use delta D to change taper thicknesses
7. Stretching
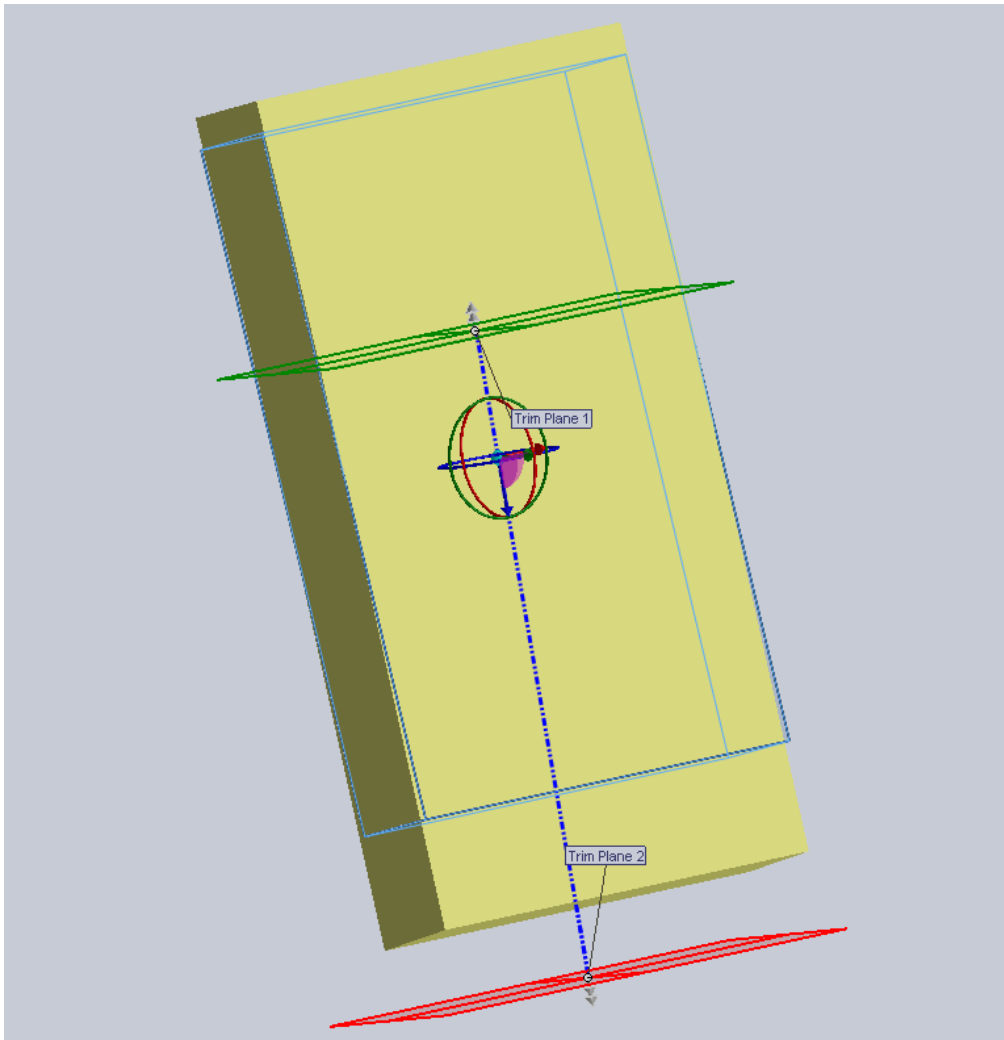
- Use the two trim planes' movements to adjust stretch distance of opposing faces
8. Press green check to complete flex
   - NOTE: Any of these can be fine-tuned with angle and distance adjustments in the bottom of the left pane.

- Wrap
  - Wrap Feature is best defined here: http://help.solidworks.com/2012/English/SolidWorks/sldworks/HIDD_DVE_SURF_WRAPPING_SKETCH.htm?id=bc27a946a6044b5db933523bfd78ce83#Pg0

- Cavity (Subtract)
  1. Insert the two parts into an assembly via "File > Make Assembly from Part" and "Insert Components"
  2. Use "Move Part" under the "Features" tab on the ribbon to line up part with part you want to create a cavity or mold in.
  3. Select the part in which the cavity will be and click on the component in the navigation pane on the left. Now go to the ribbon under "Features" and click "Edit Component"
  4. Click on the part that will be the shape of the cavity in the Design Tree and make sure the entire part is highlighted.
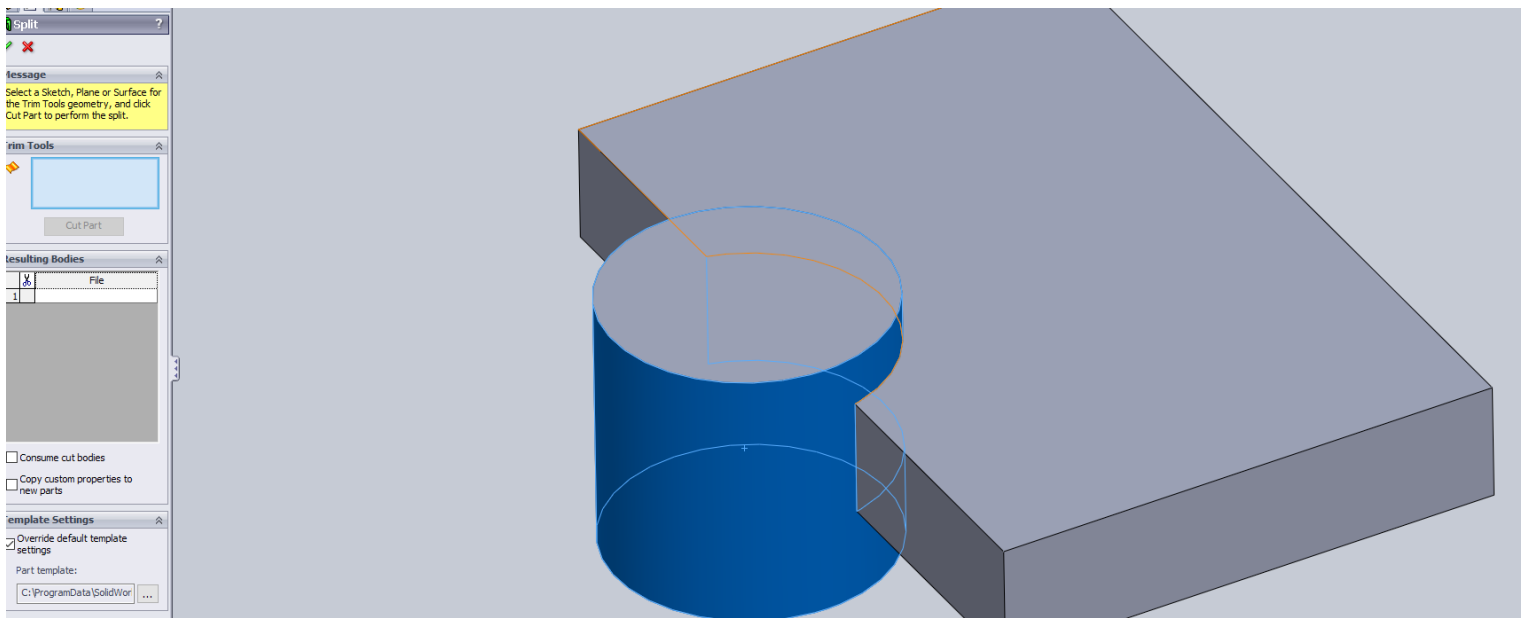 5. Navigate to "Insert > Features > Cavity..."
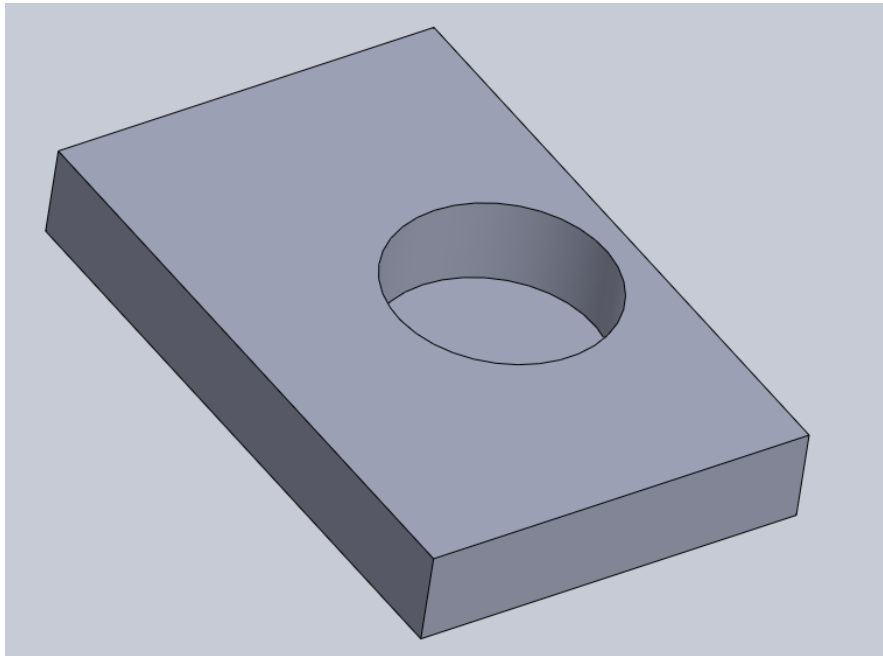
6. Click on the green check on the left hand side of the screen to complete the cavity.
7. Now click on "Edit Component" again to exit.
8. If you don't need the part that made the cavity anymore, right click on it in the Design Tree and go to the third picture option at the top of the menu. The tooltip (mouse hover text) should be labeled "Hide Component." This means that it won't be exported in the .stl file needed to print.

- Join
   1. Insert the two parts into an assembly via "File > Make Assembly from Part" and "Insert Components"
   2. Use "Move Part" to adjust parts to needed joining place
   3. Select the base part in the left navigation pane
   4. Navigate to "Insert > Features > Join"
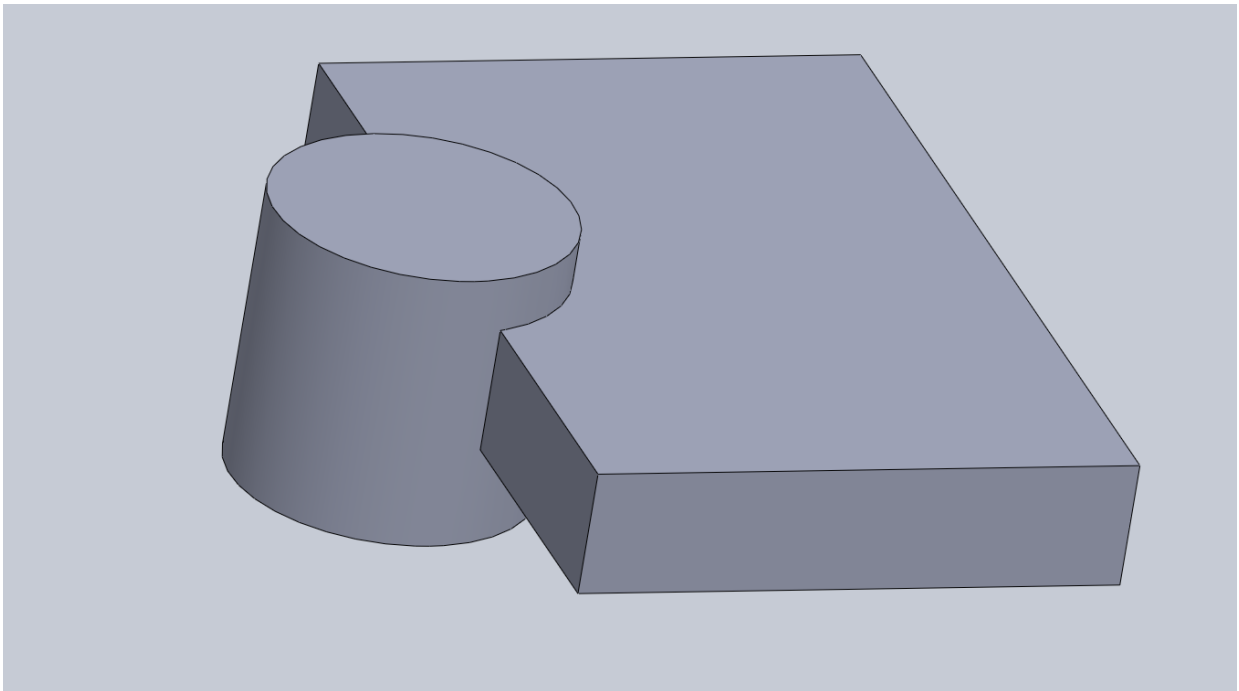
5. Select second part to join
6. Press the green check to complete
   NOTE: This will not remove the second part from the assembly, but will still modify the first part. This can create issues if second part isn't hidden.

- Split
  1. Draw whatever lines you want to split part by using Sketch
  2. Make sure whatever faces you want to split by are selectable

  3. Navigate to "Insert > Features > Split"
  4. Select faces/sketches/etc to split apart.

Body 3: <None>

Body 1: <None>

Body 2: <None>
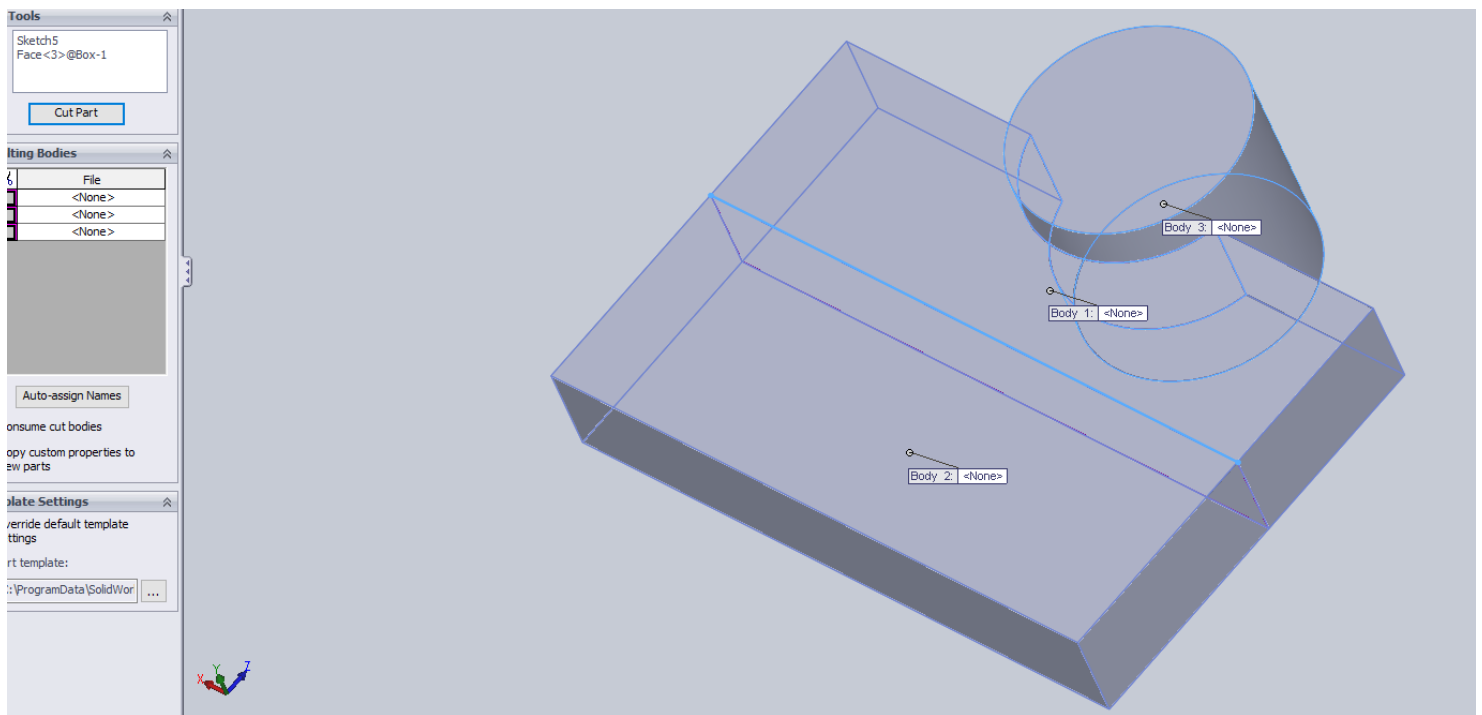
5. Press "Cut Part" on the left hand side
6. Result as in figure. Press green check to finish splitting.