Solidworks 2006 Surface-modeling

(Tutorial 2-Mouse)

Surface-modeling

Solid-modeling

Assembly Design

Design with a Master Model

Written by Dickson Sham

Tutorial 2A

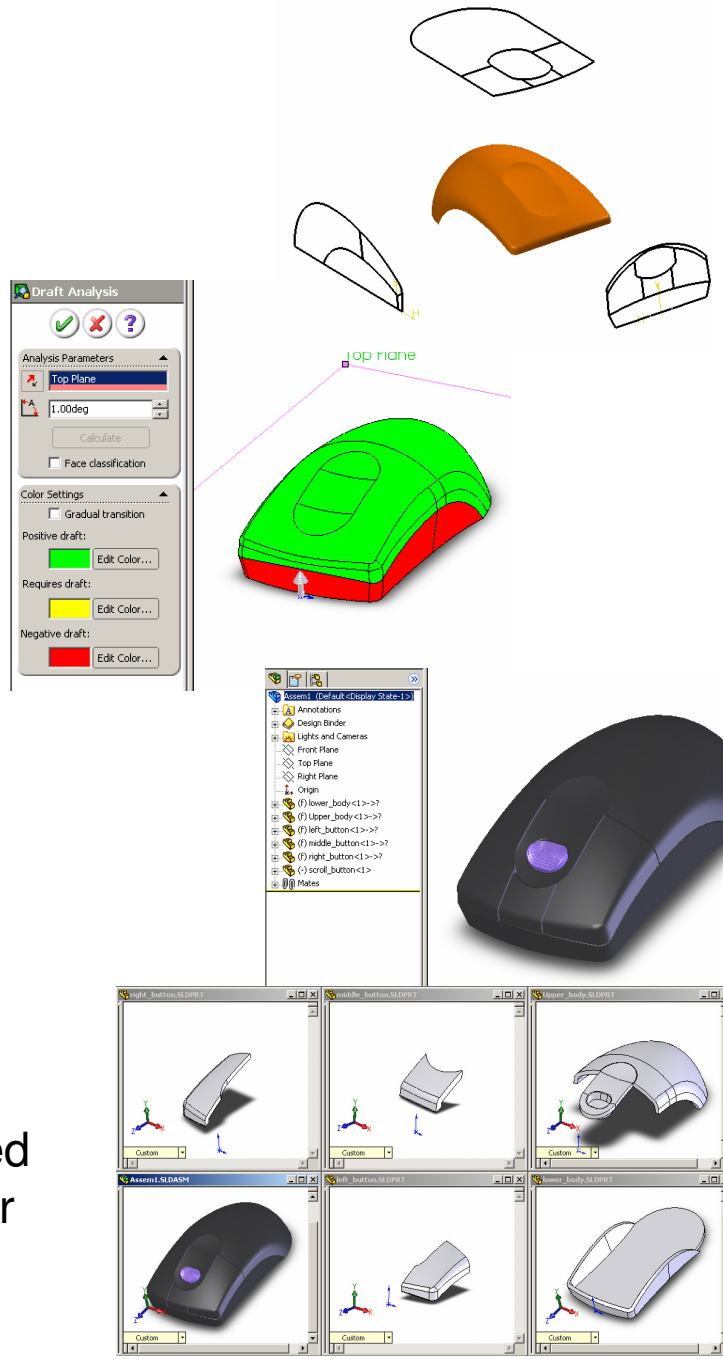
- Import 2D outline drawing into Solidworks2006
- Build 3D curves based on the imported drawing
- Build the upper surfaces of the mouse

Tutorial 2B

- Do the draft analysis to search any undercut portion on the upper surfaces
- Adjust the curvature of the problem surface manually
- Build the lower surfaces of the mouse
- Convert the surfaces into a solid

Tutorial 2C

- Build the parting surfaces based on the imported drawing
- Create components from the finished model
- Re-assemble the components into a product
- Modify the outlook of the master model and then get all components updated automatically

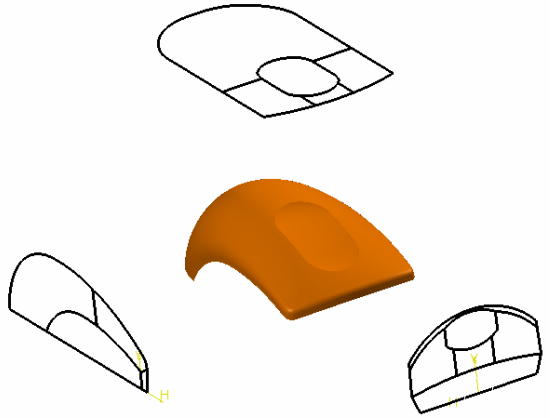


Please be reminded that this series of tutorials is designed to demonstrate a design approach with Solidworks, rather than the command itself.

Tutorial 2A

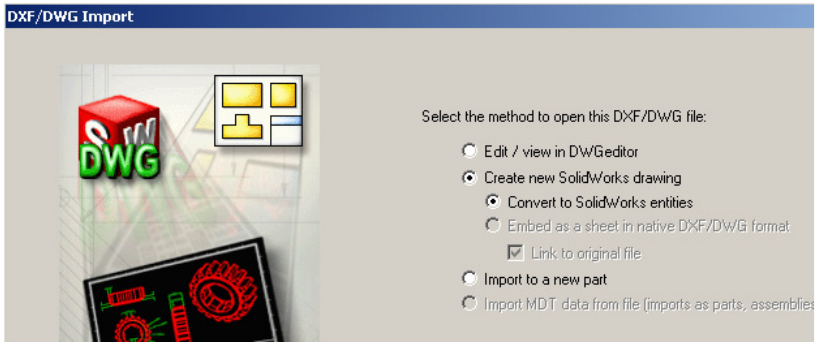
- Create a new project folder and copy this drawing file **(your DVD name):\Model\mouse_outline.dxf)** into the folder

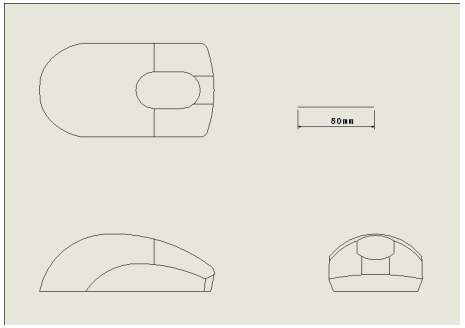
- Enter Solidworks2006 by double-clicking its icon on the desktop.



(BEFORE START, reset the layout of workbench if necessary)

- Select "**File/Open**" on the menu bar and select the drawing (**mouse_outline.dxf**)
- Select "Create new solidworks drawing"
- Select "Next"
- (You should have a preview); select "Next" again
- Select "Finish"





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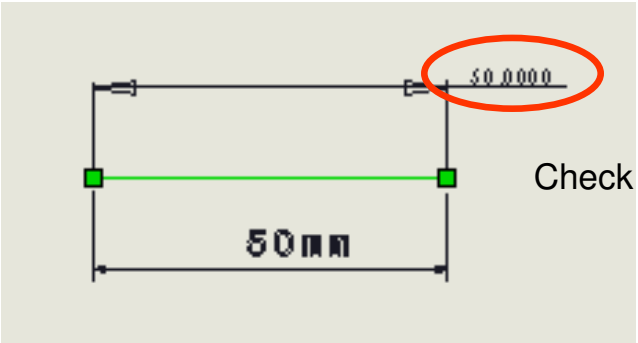
Tutorial 2A

To confirm that the size of the drawing is correct:-

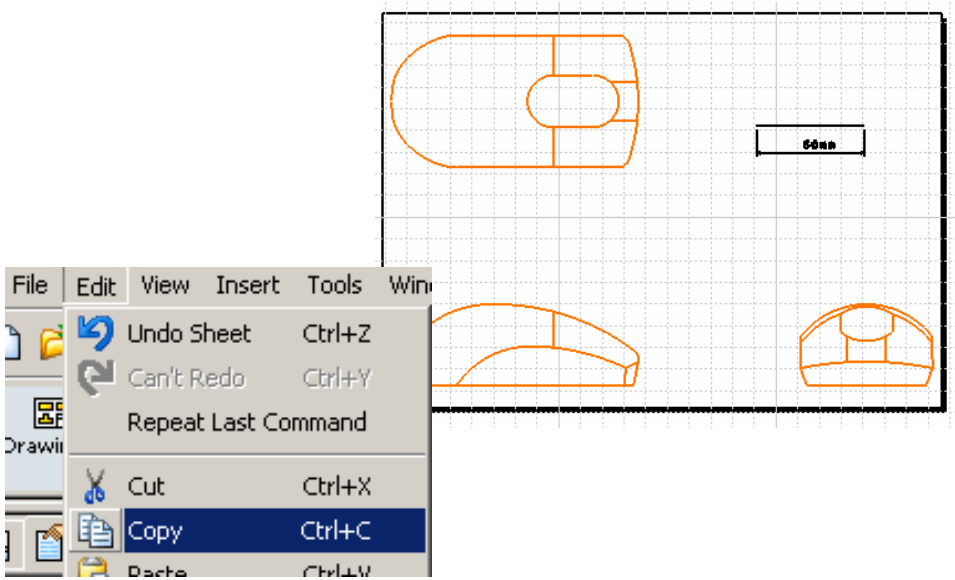
- Click "**Annotations**" icon
- Click "**Smart Dimension**" icon
- Click on the scale line of the drawing
- Check if the displayed dimension is 50mm; If not, we need to enlarge or shrink the drawing into the correct size.



Check if it is 50

To copy and paste the drawing into 3D space:-

- Multi-select all entities on the drawing, except the scale bar;
- Select "**Edit /Copy**" on the menu bar



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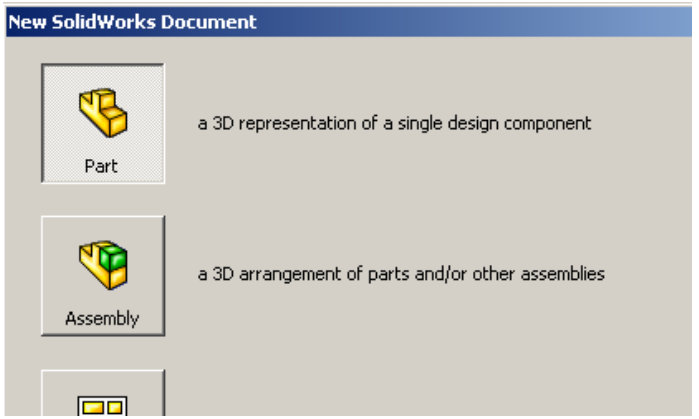
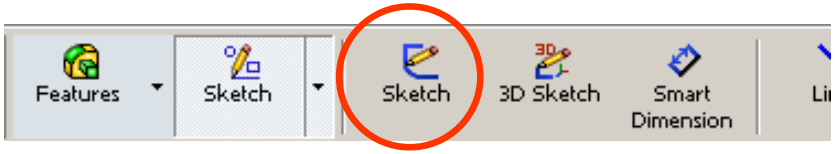
Tutorial 2A

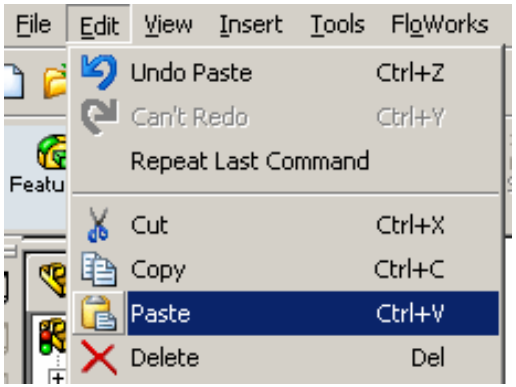
- Select “**File/New**” on the menu bar;
- Select “**Part**”
- Click OK



- Click “**Sketch**” icon and select **Front Plane**
- Select “**Edit/ Paste**” on the menu bar
- (All 2D elements are now pasted onto this sketch plane, Front Plane)



- Click “**Exit Sketch**” icon to exit



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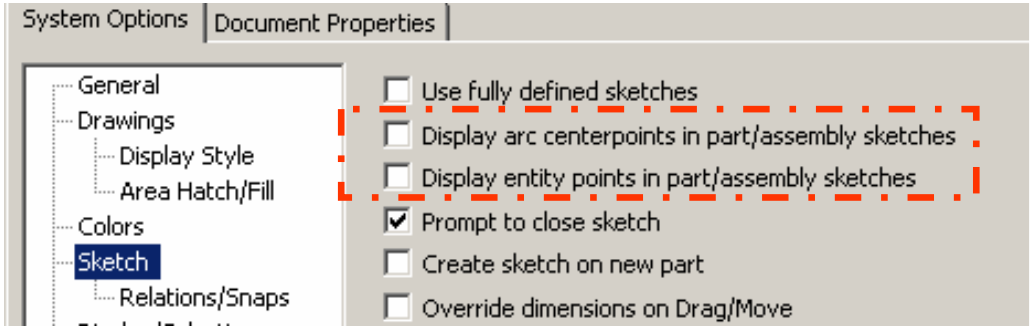
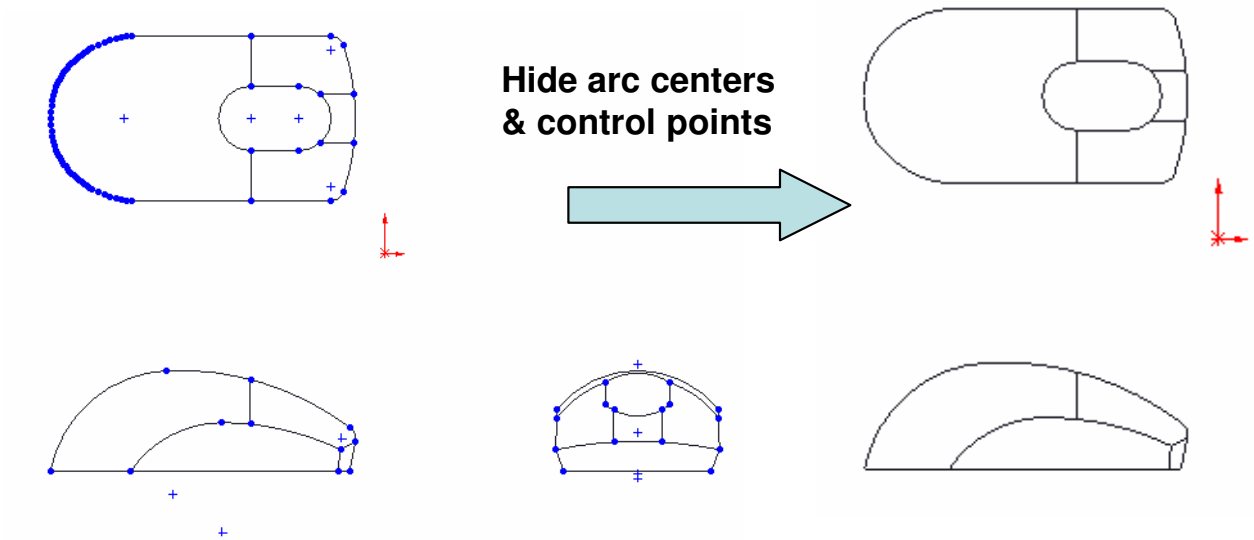
Tutorial 2A

To hide the arc centerpoints and control points of splines (optional):-

- Select "**Tools/ Options…**" on the menu bar
- On the first tab page "System Options", click on "Sketch"
- **Deselect** the two options
  - Display arc centerpoints in part/assembly sketches
  - Display entity points in part/assembly sketches
- Click ok to complete





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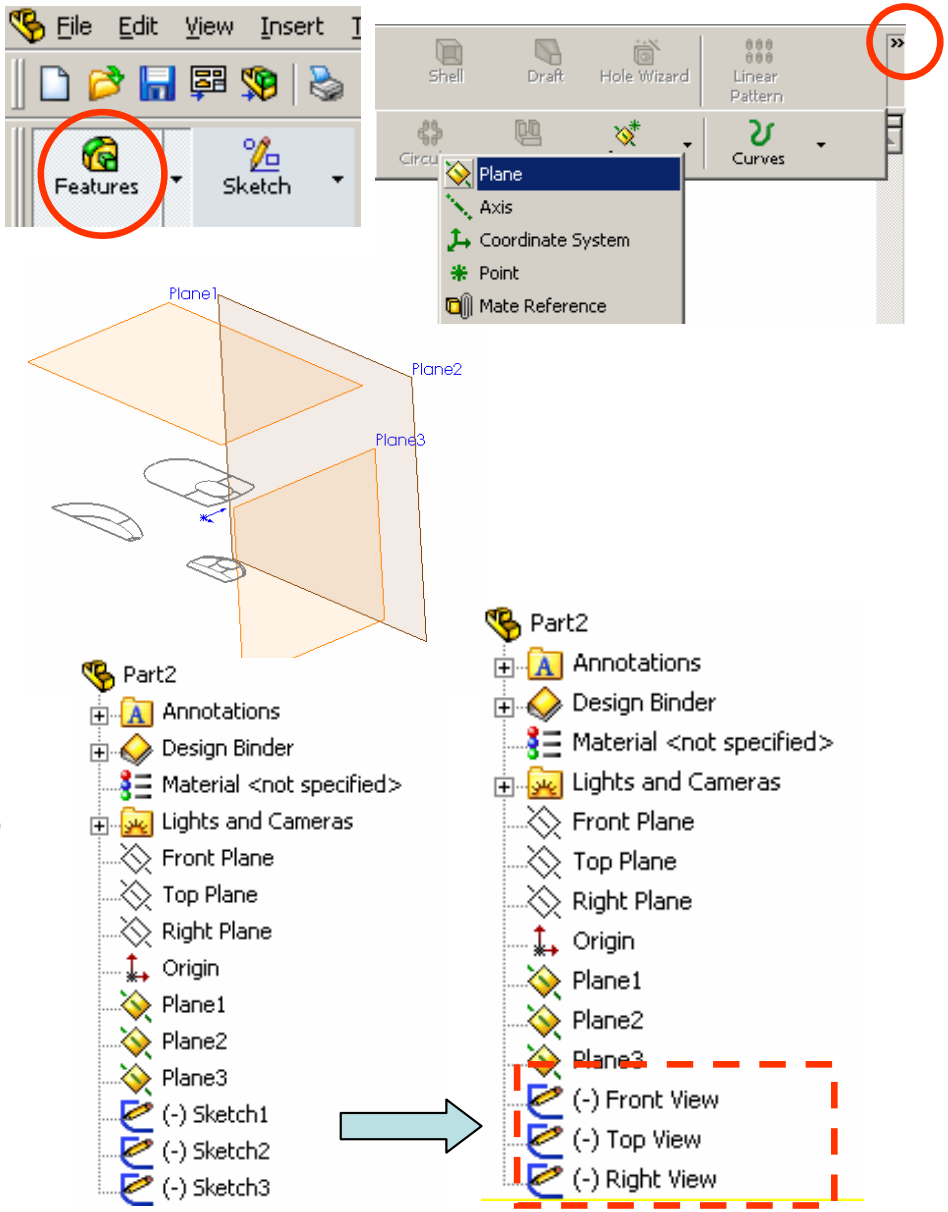
Tutorial 2A

To split the drawing into three individual views and position them:-

- Click “Features” icon
- Click >> icon at the rightmost
- Click “**Reference**” icon, then select “**Plane**”
- Select **Front Plane** on the tree
- Enter **150mm** as D1
- Press “Enter” key on the keyboard to complete

- Create an offset plane, 150mm from **Top Plane**
- Create an offset plane, 150mm from **Right Plane**

- Duplicate two more “Sketch1” by ***copy-and-paste*** function
- ***Drag*** “Sketch.1” to the position after “plane3” on tree
- Rename them as “Front View”, “Top View” & “Right View” (right-click the sketches and select “Feature Properties”)



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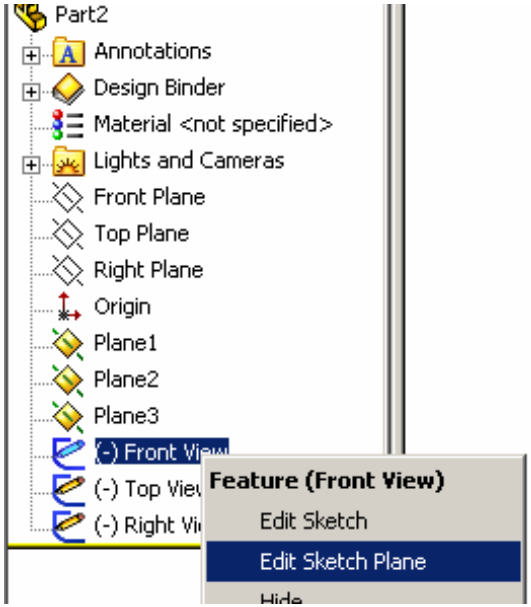
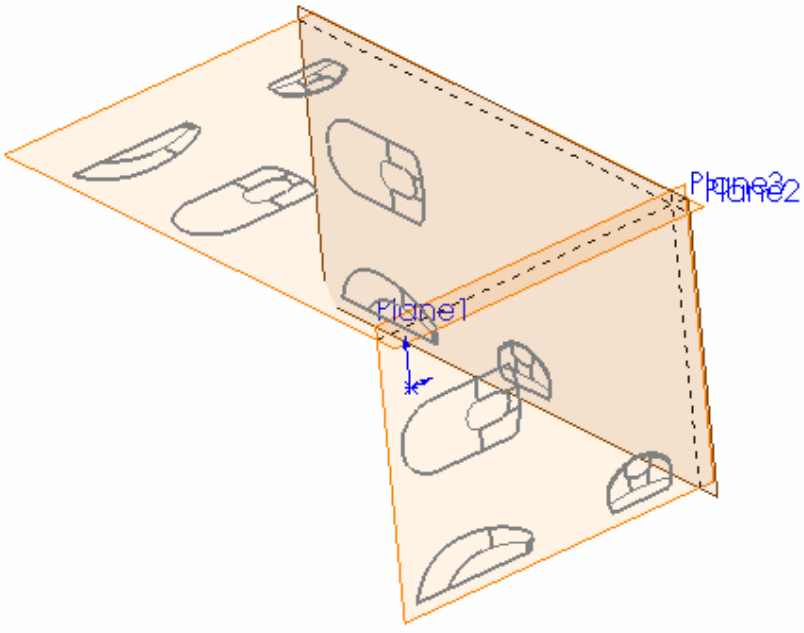
Tutorial 2A

- Right-click on “**Front View**” on the tree and select “**Edit Sketch Plane**”
- Select “**Plane.1**”
- Click ok to confirm



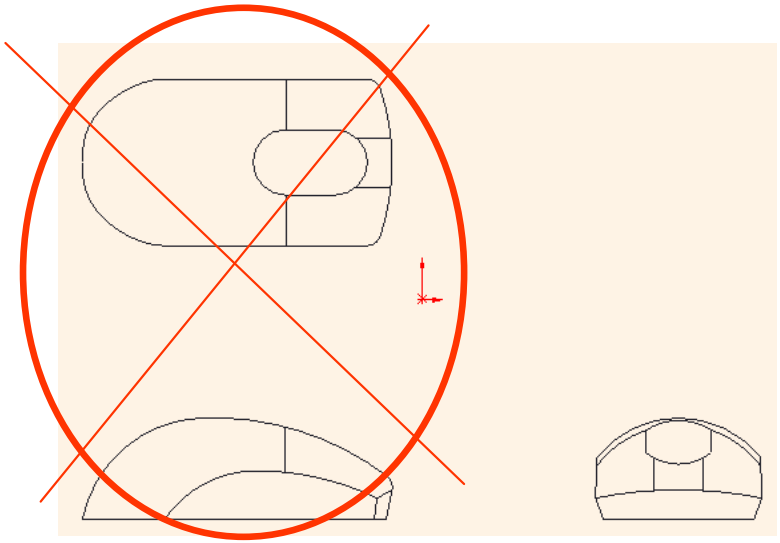
- Similarly, right-click “**Top View**” and select “**Edit Sketch Plane**”;
- Select “**Plane.2**”
- Click ok to confirm

- Similarly, right-click “**Right View**” and select “**Edit Sketch Plane**”;
- Select “**Plane.3**”
- Click ok to confirm

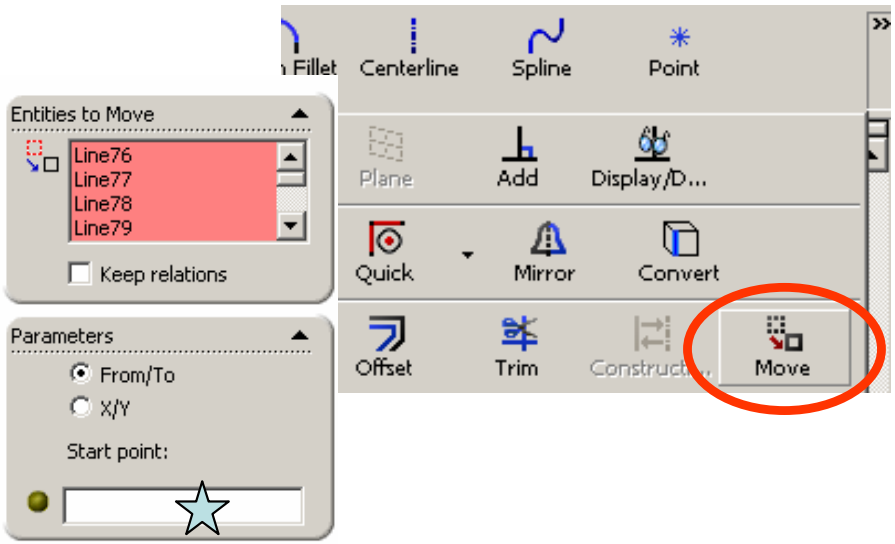


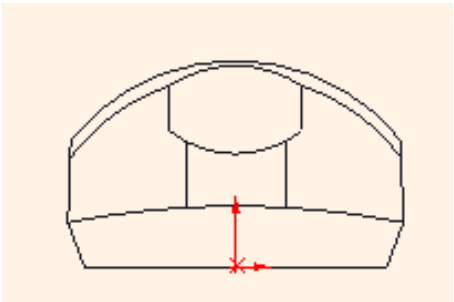
Tutorial 2A

- Right-Click "**Front View**" sketch on the tree and then select "**Edit Sketch**"
- Press "Space" key on the keyboard and then select "Normal to"
- Select and delete the curves not related to the Front view;



- Click >> at the rightmost
- Click "**Move**" icon
- Select all the remaining 2D lines/curves
- Select "From/to" in the box "Parameters"
- Click the entry box ☆
- Click the midpoint of the bottom line (the system can detect the midpoint when the mouse cursor is near the point)
- Right-Click "Quick Snap", then "Grid Snap"
- Click the **RED** origin
- Click "Exit Sketch" icon to complete





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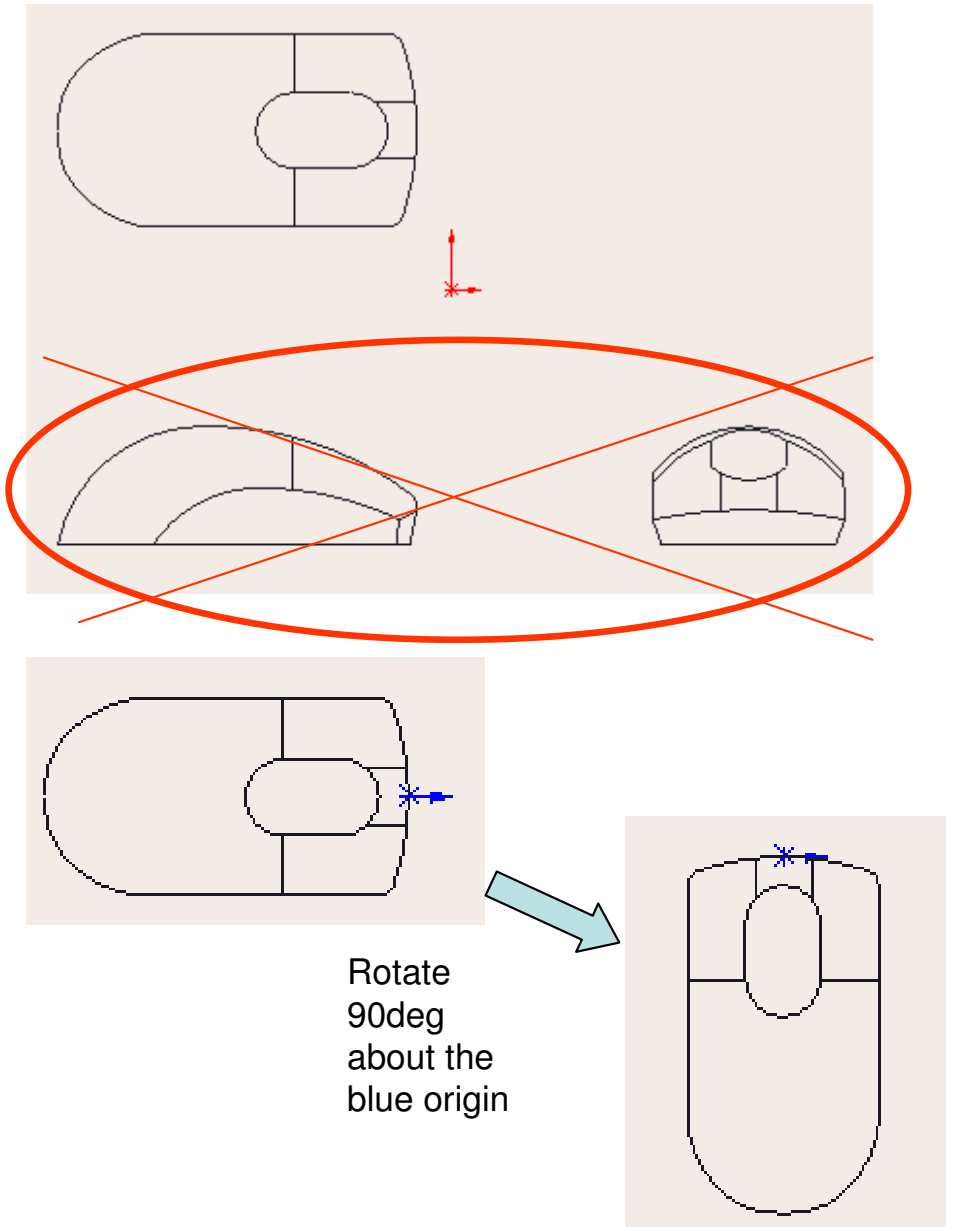
Tutorial 2A

- Right-Click "**Top View**" sketch on the tree and then select "**Edit Sketch**"
- Press "Space" key on the keyboard and then select "Normal to"
- Select and delete the curves not related to the Top view;

- Click >> at the rightmost
- Click "**Move**" icon
- Select all the remaining 2D lines/curves
- Select "From/to" in the box "Parameters"
- Click the entry box
- Click the midpoint of the right arc (the system can detect the midpoint when the mouse cursor is near the point)
- Right-Click "Quick Snap", then "Grid Snap"
- Click the **BLUE** origin (System origin)

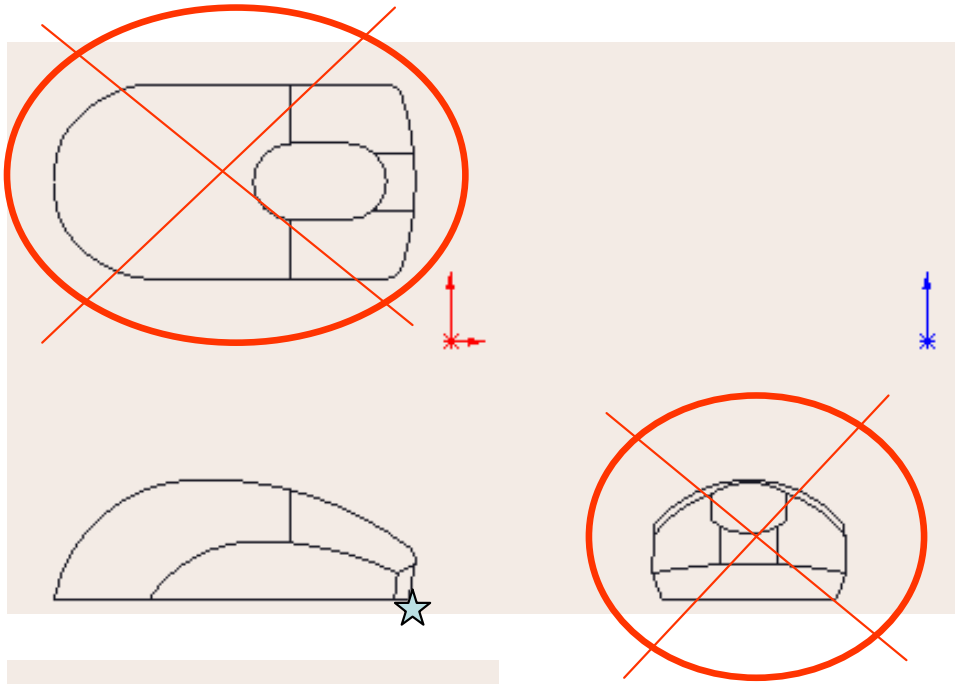
- Select "Tools / Sketch Tools / **Rotate**" on the menu bar
- Select all the remaining 2D lines/curves
- Click the entry box (center of rotation)
- Click the **BLUE** origin
- Enter 90 as Degree
- Press "Enter" key to complete

- Click "Exit Sketch" icon to complete

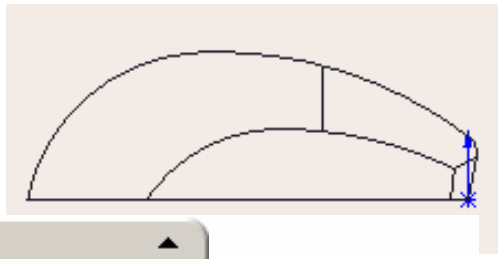


Tutorial 2A

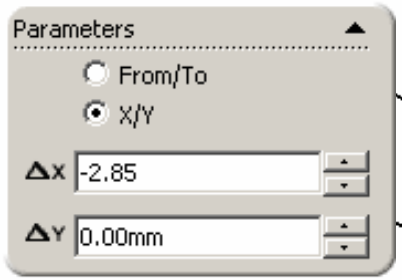
- Right-Click "**Right View**" sketch on the tree and then select "**Edit Sketch**"
- Press "Space" key on the keyboard and then select "Normal to"
- Select and delete the curves not related to the Right view;

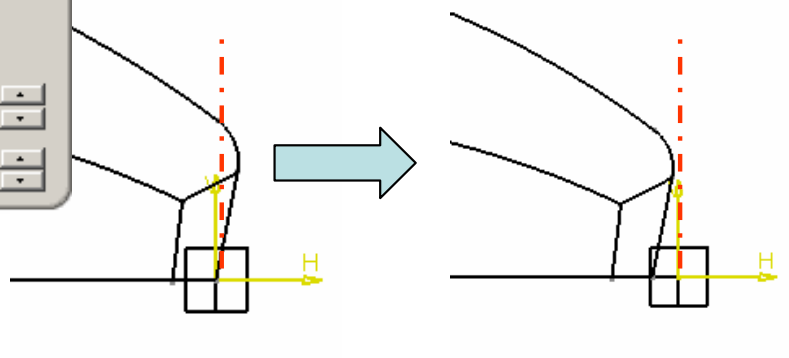


- Click >> at the rightmost
- Click "**Move**" icon
- Select all the remaining 2D lines/curves
- Select "From/to" in the box "Parameters"
- Click the entry box
- Click the point ☆
- Right-Click "Quick Snap", then "Grid Snap"
- Click the **BLUE** origin (System origin)



- Click "**Move**" icon again
- Select all 2D lines/curves
- Select "X/Y" in the box "Parameters"
- Enter -2.85 as △X
- Press Enter key to complete





- Click "Exit Sketch" to complete

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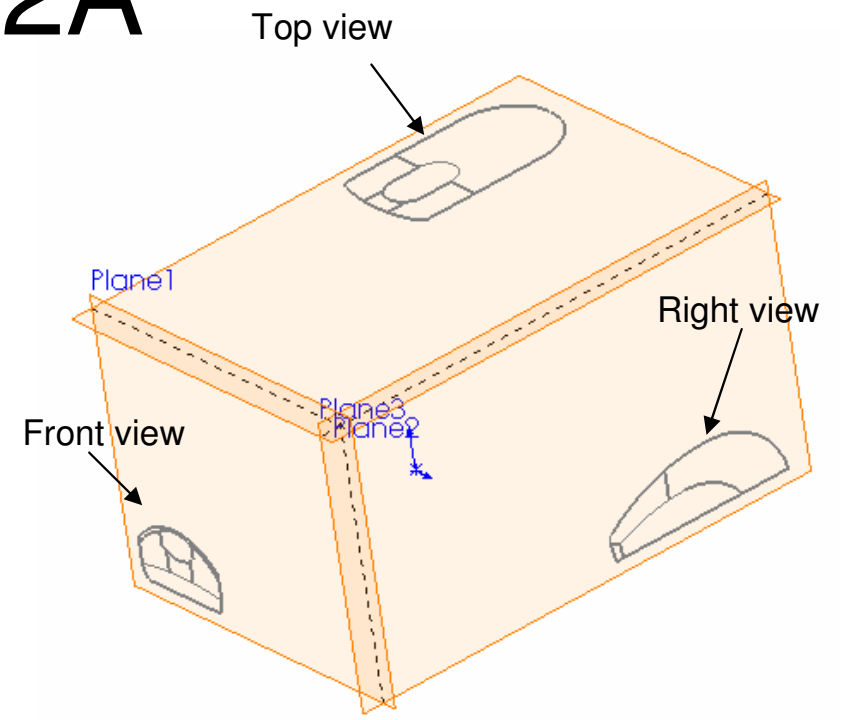
Tutorial 2A

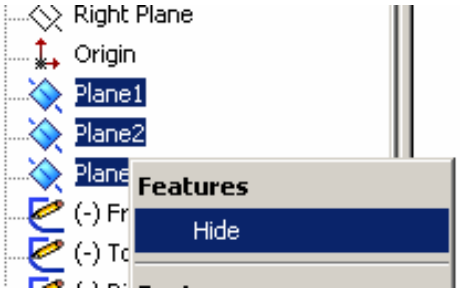
- Now we have positioned the three views at the correct places. These will be a good reference for us to build the 3D in the middle of the screen.

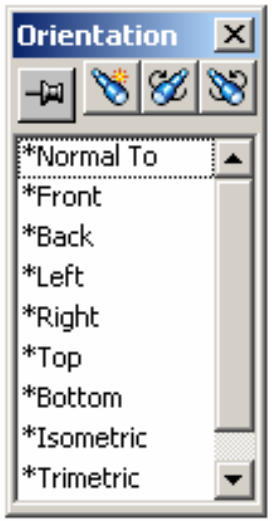
- You can click any standard view icon to change viewing direction so that you can compare your working 3D with the reference at a specific viewpoint.



To Hide all offset planes:-

- Multi-select Plane1, Plane2 & Plane3 on the tree
- Right-Click and select “Hide”





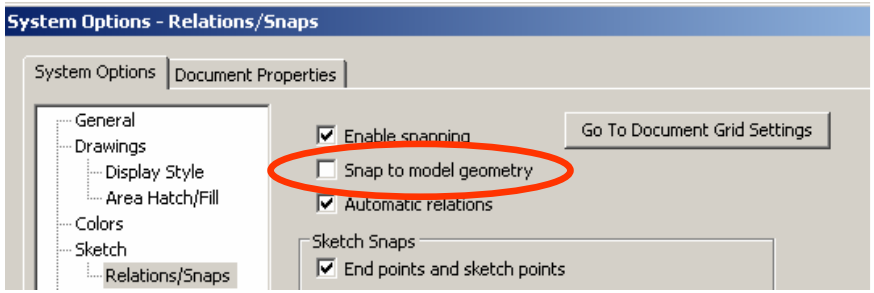
Anytime when we press “space” key on the keyboard, we can select a standard viewpoint

Tutorial 2A

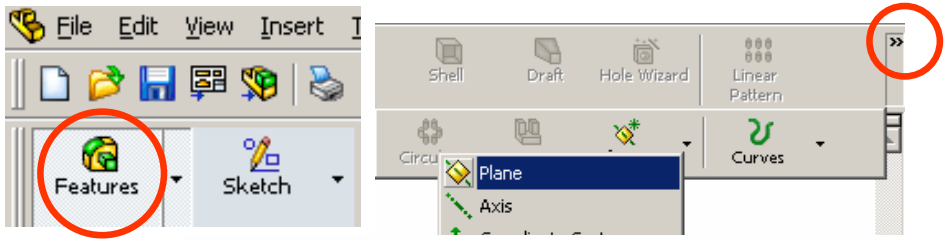
*** To disable auto-snapping to any geometry out of the working sketch:-**

- Select Tools/Options/System Options /Sketch/Relations
- Deselect “Snap to model geometry”

Remark: This setting can prevent us from making any relations between the reference sketches and other geometries. We should treat the reference sketches as the background images only.

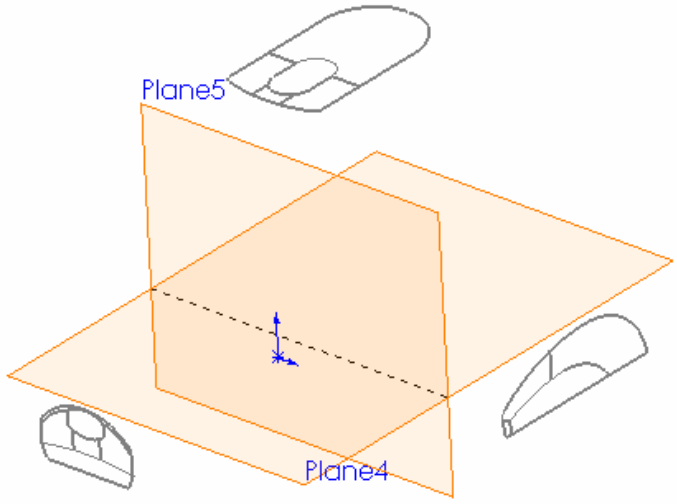


To create 3D curves from the reference sketches:-



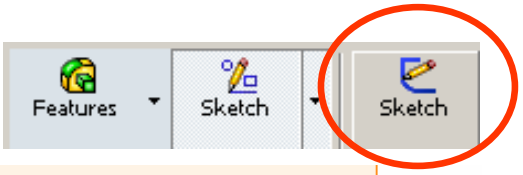
- Click “Features” icon
- Click >> icon at the rightmost
- Click “**Reference**” icon, then select “**Plane**”
- Select **Top Plane** and enter **50mm** as offset value
- Press “Enter” key to complete

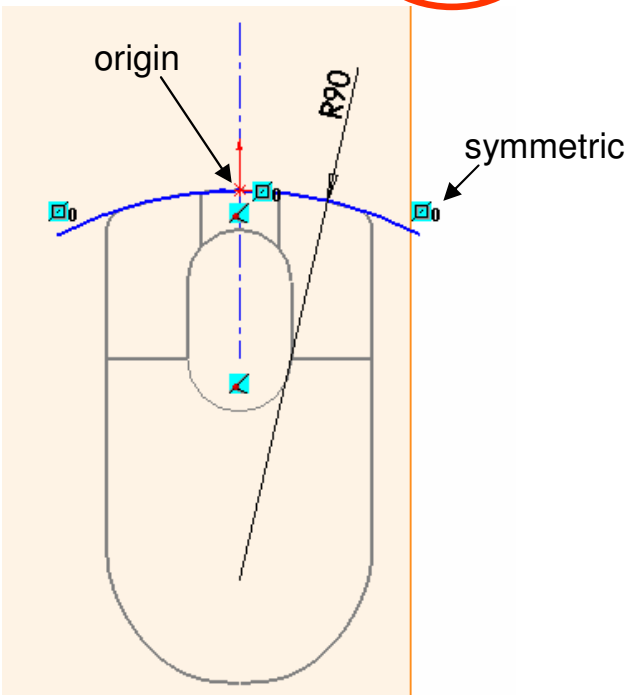
- Click “**Plane**” icon again
- Select **Front Plane** and enter **50mm** as offset value
- Press “Enter” key to complete



Tutorial 2A

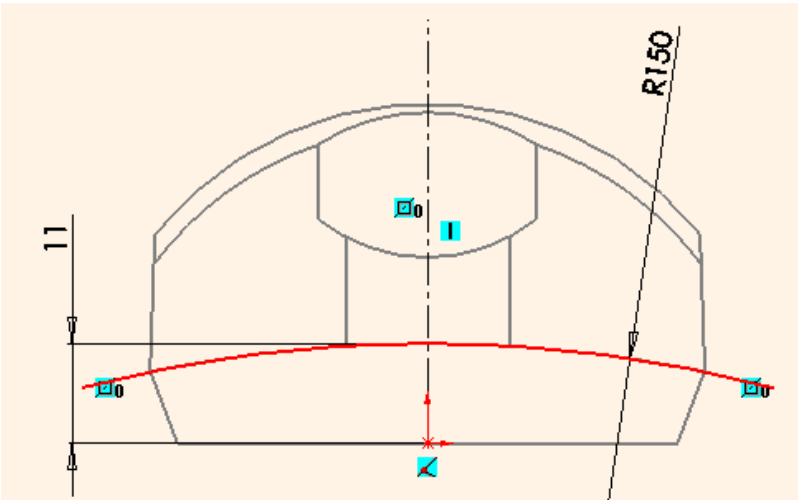
- Click “**Sketch**” icon and select “**Plane4**”;
- Press “Space” key on the keyboard and then select “Normal to”
- Draw an **Arc** (R90mm), with two ends symmetric about the y-axis and its midpoint is on the origin;
- (Remark: a centerline is needed to define the symmetry of two arc ends)
- (Remark: To define symmetry, multi-select the two arc ends first and then the centerline)
- (Remark: the arc is a little longer than the reference)
- Click “**Exit Sketch**” icon to complete



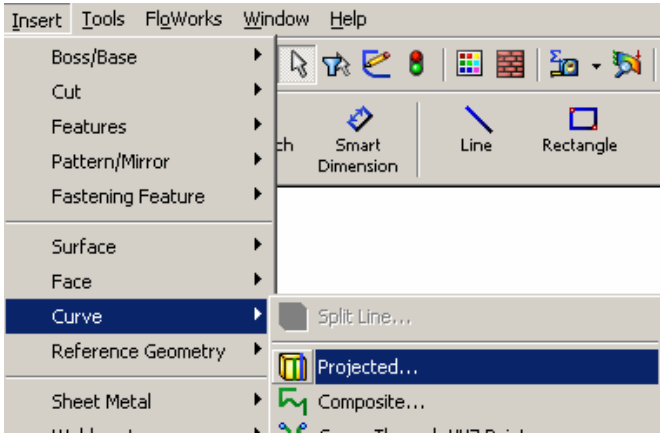


- **Click an open area to deselect the Sketch (Sketch4)**

- Click “**Sketch**” icon and select “**Plane5**”;
- Draw an **Arc** (R150mm), with two ends symmetric about the y-axis and the peak 11mm from the x-axis;
- Reminded that the arc should be a little bit longer than the reference;
- Click “**Exit Sketch**” icon to complete

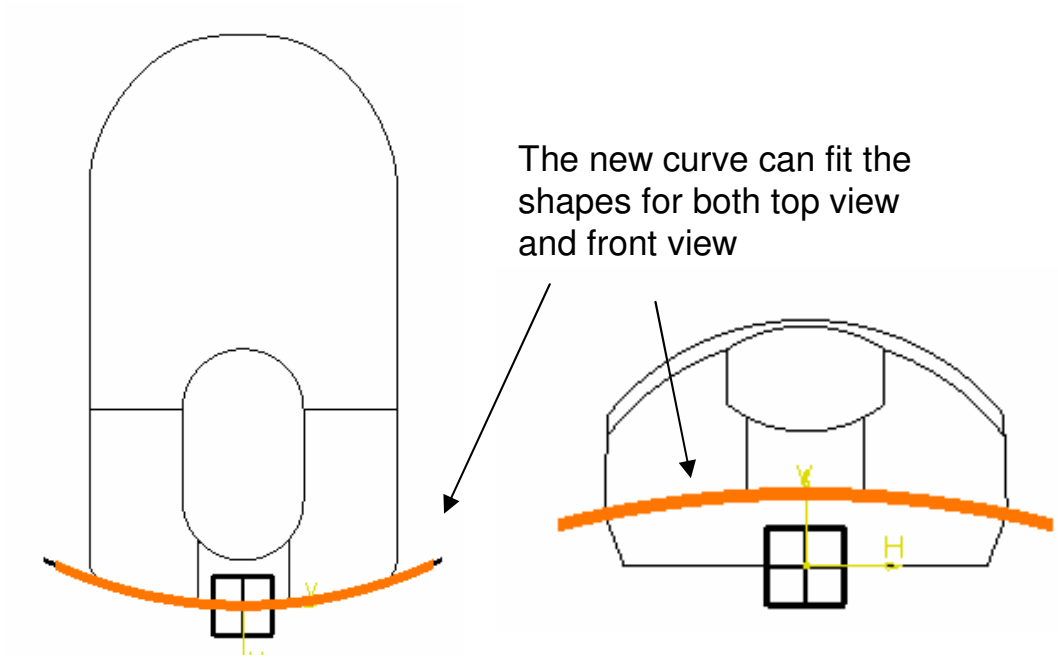


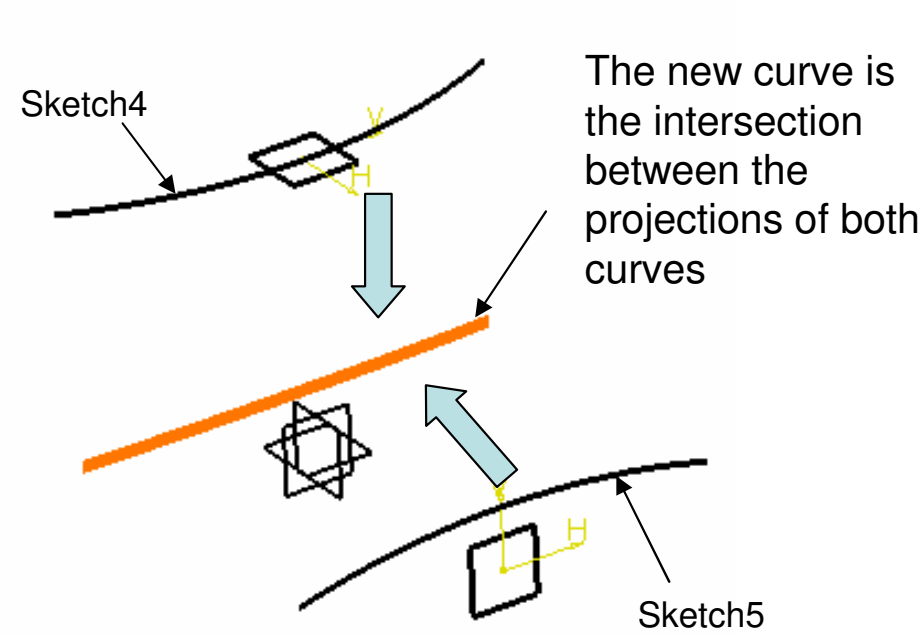
Tutorial 2A



- Select “**Insert** / **Curve** / **Projected…**” on the menu bar
- Select “**Sketch4**” & “**Sketch5**”;
- Press “Enter” key to complete
- (“Sketch.4” and “Sketch.5” are both hidden automatically after the projection)

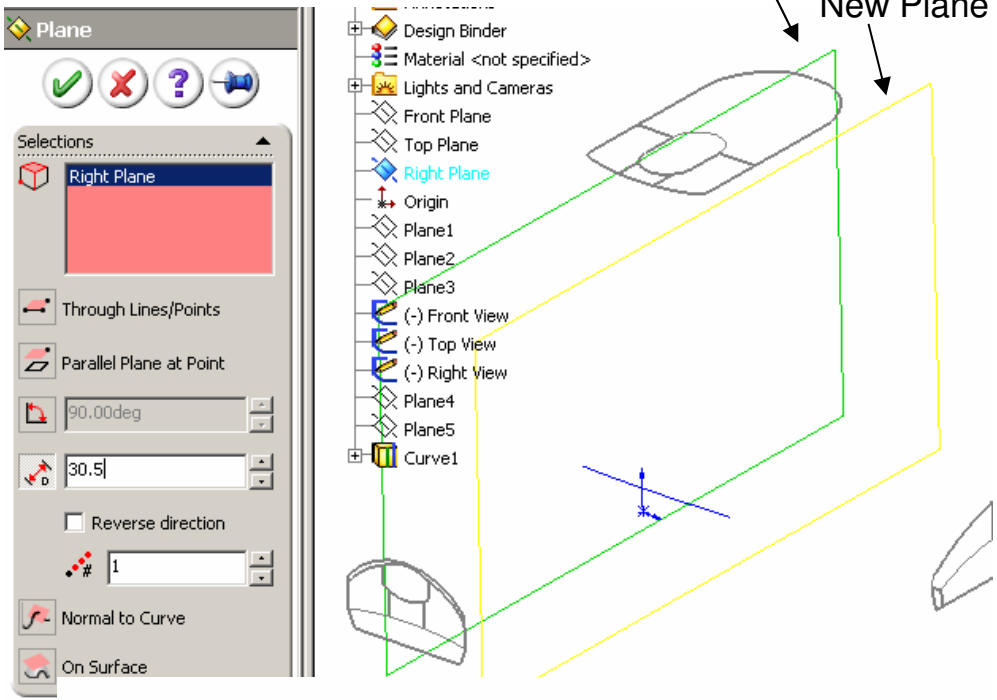
- **Hide “Plane4” & “Plane5”**



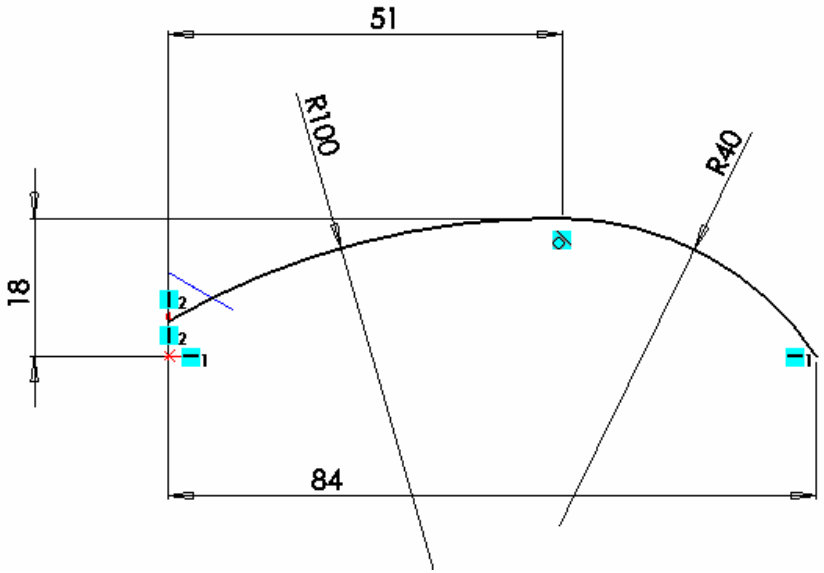


Tutorial 2A

- Click “Features” icon
- Click >> icon at the rightmost
- Click “**Reference**” icon, then select “**Plane**”
- Select **Right Plane** and enter **30.5mm** as offset value
- Press “Enter” key to complete



- Click “**Sketch**” icon and select the new plane “**Plane6**”
- Draw **two arcs** as shown
- Reminded that two arcs must be tangent to each other; one end of the small arc is touching the x-axis; one end of the bigger arc is just near y-axis
- (Remark: Do not make any relation with other sketch)
- Click “**Exit Sketch**” icon to complete.
- **Hide “Plane6”**



Tutorial 2A

To create surfaces from 3D curves:-

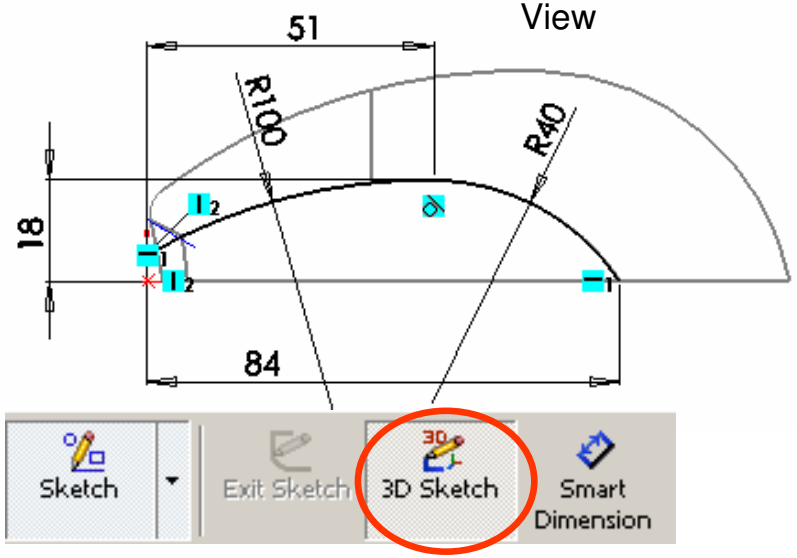
- Click "**3D Sketch**" icon
- Select "Curve1" (projected curve) on the tree
- Click "**Convert**" icon
- Click "3D Sketch" icon again to complete

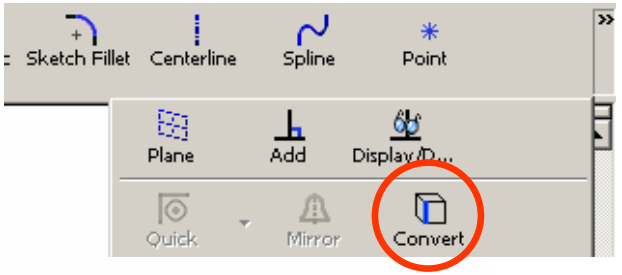
- Click "**3D Sketch**" icon
- Select "Sketch6" on the tree
- Click "**Convert**" icon
- Click "3D Sketch" icon again to complete

(This is a limitation of Solidworks2006 that we cannot create an extruded surface based on a projected curve; but after converting it into 3D Sketch, it is possible)

(This is another limitation of Solidworks2006 that we cannot create an extruded surface along the direction that is parallel to the sketch plane; but after converting it into 3D Sketch, it is possible)

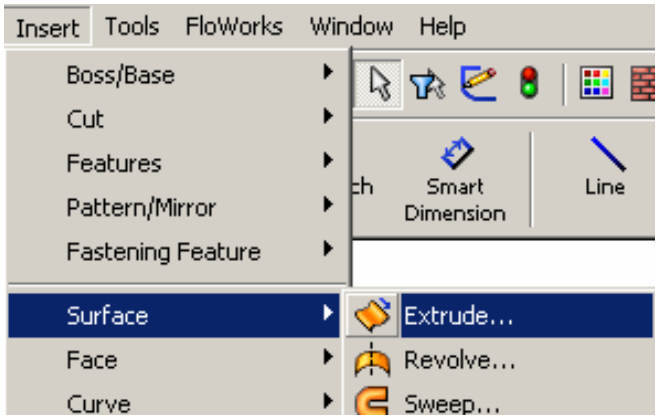
Sketch6 should match the profile in Right View





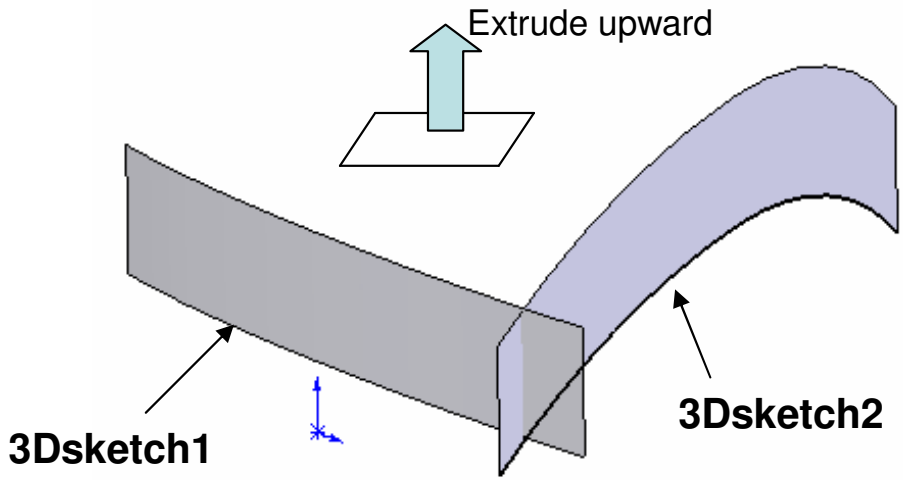
Tutorial 2A

- Hide “Curve1” & “Sketch6” on the tree **(we use their image, 3Dsketch1 & 3Dsketch2)**



- Select “**Insert** / **Surface** / **Extrude**” on the menu bar
- Select “3Dsketch1” as profile
- Select “Top Plane” as direction
- Enter **20mm** as D1
- Press Enter key to complete

- Select “**Insert** / **Surface** / **Extrude**” on the menu bar
- Select “3Dsketch2” as profile
- Select “Top Plane” as direction
- Enter **20mm** as D1
- Press Enter key to complete

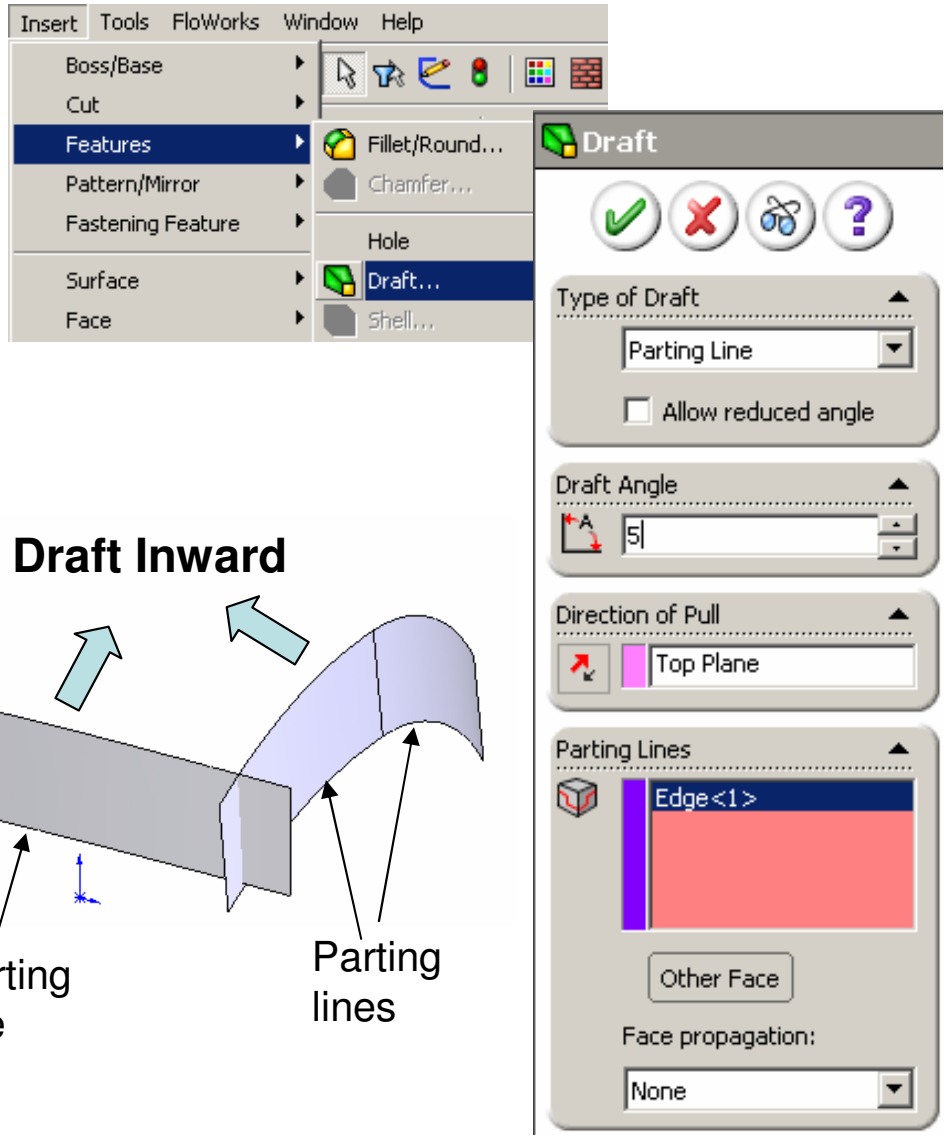


Tutorial 2A

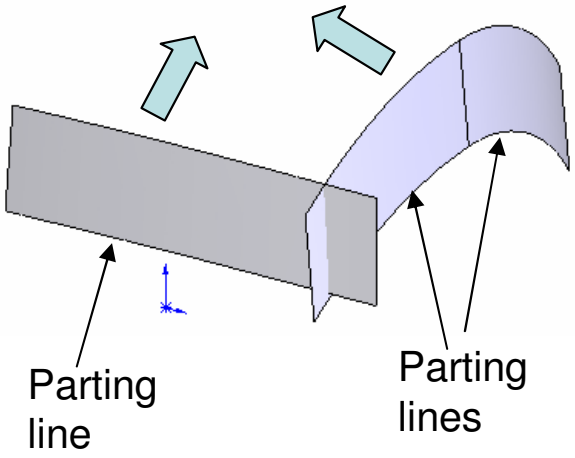
To add a draft onto extruded surfaces:-

- Select "**Insert** / **Features** / **Draft**" on the menu bar
- Select "Parting Line" as Type of Draft
- Select "Top Plane" as Direction of Pull
- Click "Reverse Direction" icon 
- Select the bottom edge of "Surface-Extrude1" as the parting line
- Enter **5deg** as Draft Angle
- Press Enter key to complete

- Select "**Insert** / **Features** / **Draft**" again
- Select "Parting Line" as Type of Draft
- Select "Top Plane" as Direction of Pull
- Select the TWO bottom edges of "Surface-Extrude2" as parting lines
- Enter **5deg** as Draft Angle
- Press Enter key to complete



Draft Inward

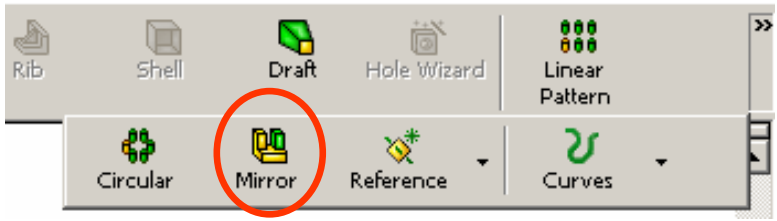


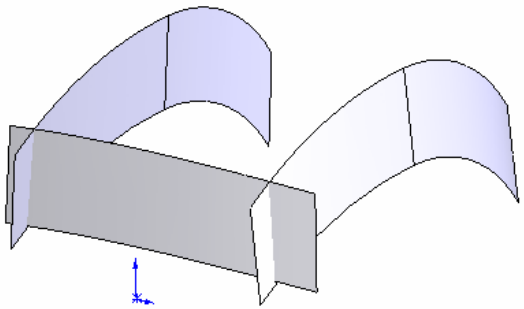
Written by Dickson Sham

Tutorial 2A

To duplicate a surface by mirroring:-

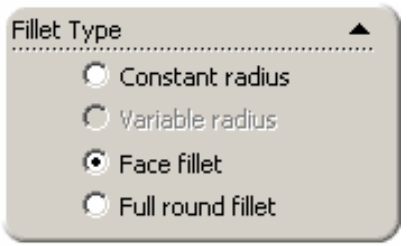
- Click “Features” icon
- Click >> icon at the rightmost
- Click “**Mirror**” icon
- Select “Right Plane” as Mirror Plane
- Click on “Bodies to Mirror”
- Select the surface “Draft2”
- Press Enter key to complete

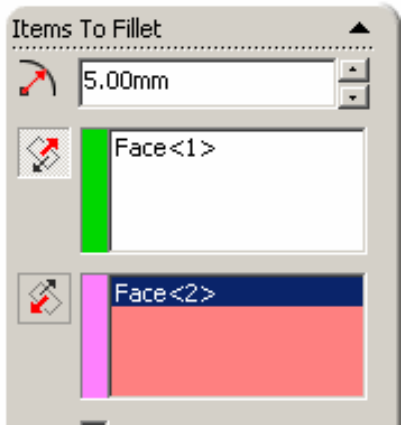


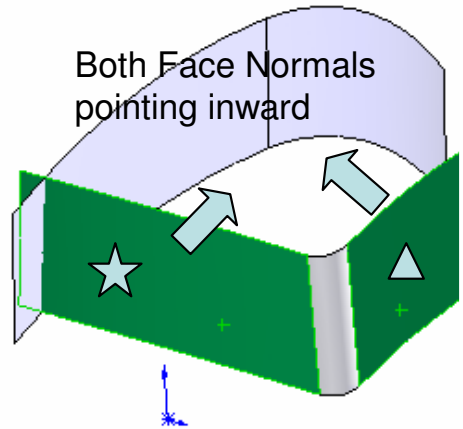


To add a fillet between two surfaces:-

- Click “**Fillet**” icon
- Select “Face Fillet” as Type
- Enter **<u>5mm</u>** as Radius
- Select the face ☆
- Click “Reverse Face Normal”
- Click the entry box “Face Set 2” (pink color)
- Select the face △
- Press Enter key to complete







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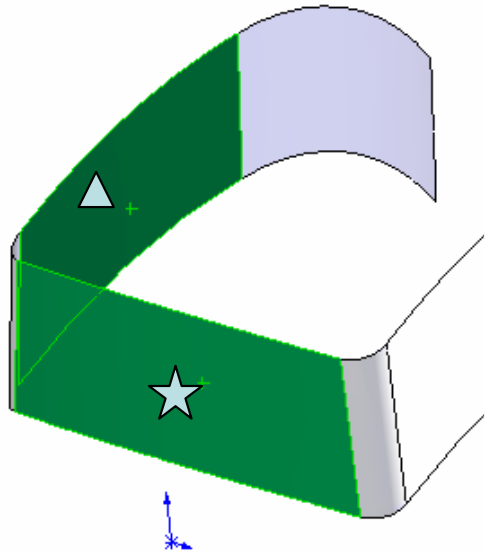
Tutorial 2A

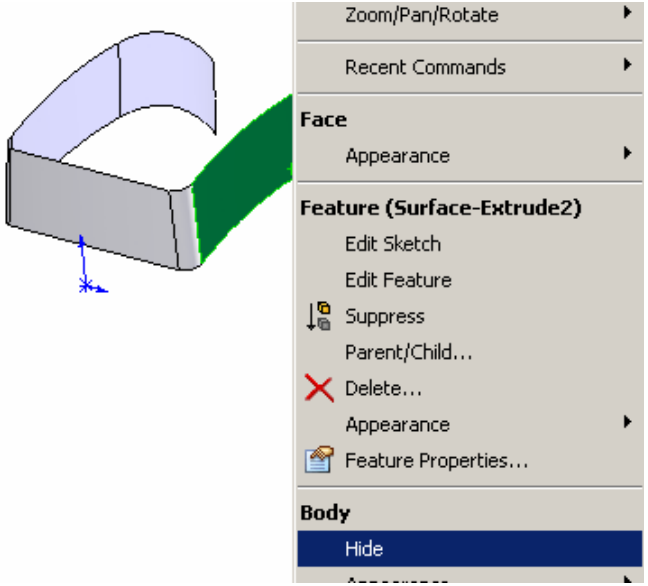
To add another face fillet :-

- Click "**Fillet**" icon again
- Select "Face Fillet" as Type
- Enter **5mm** as Radius
- Select the face ☆
- Click "Reverse Face Normal"
- Click the entry box "Face Set 2" (pink color)
- Select the face △
- Press Enter key to complete



To hide the surface:-

- Right- Click on any face of the surface
- Select "Body/ Hide" on the contextual menu

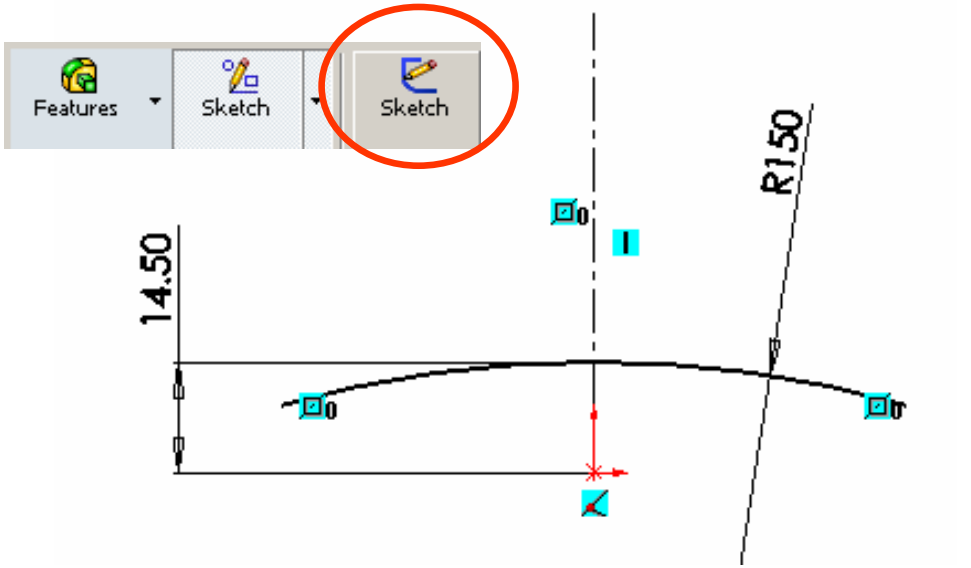


Written by Dickson Sham

Tutorial 2A

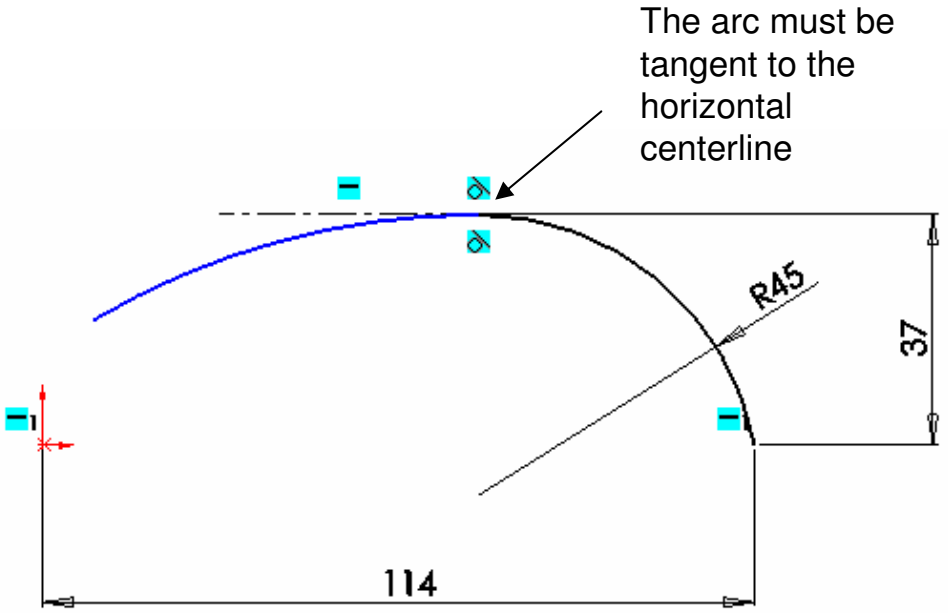
To Create a Sketch#7:-

- Click "**Sketch**" icon and select "**Front Plane**"
- Draw **an arc** as shown (R150, two ends symmetric about y-axis, distance between the midpoint and the origin is 14.5mm)
- Click "Exit Sketch" to exit



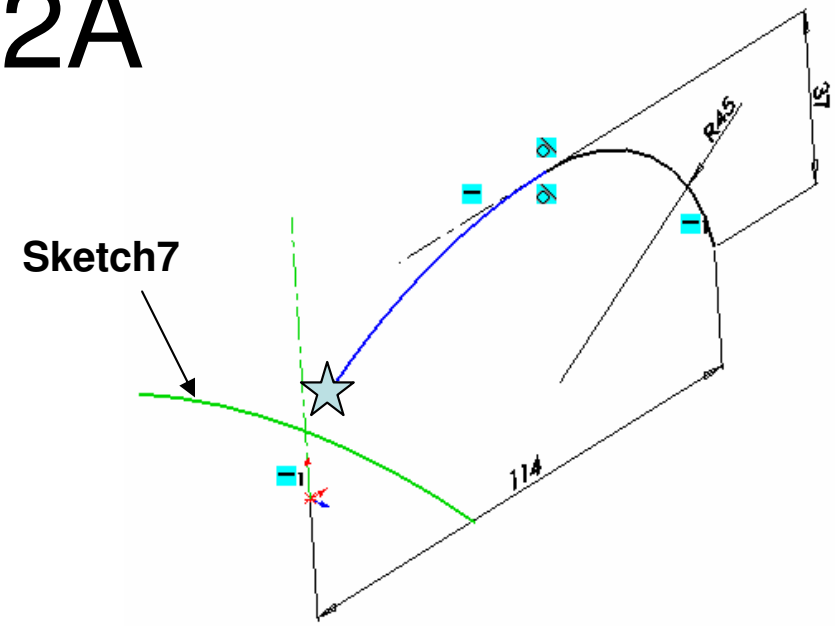
To create a sketch mating with an external sketch:-

- Click "**Sketch**" icon
- Select "**Right Plane**"
- Draw two arcs as shown
- Reminded that two arcs must be tangent to each other; one end of the small arc is touching the x-axis; one end of the bigger arc is just near y-axis
- (Remark: Do not make any relation with other sketch)
- (Remark: R45 should be tangent to a horizontal reference line so that the end point will be the highest point of the profile)



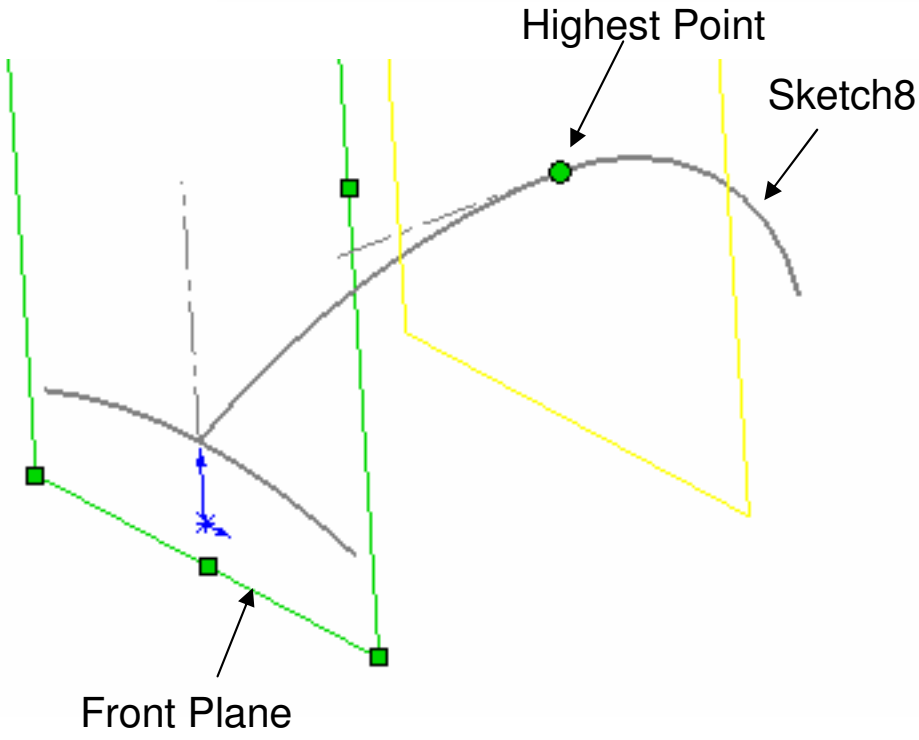
Tutorial 2A

- Multi-Select the endpoint ☆ and Sketch7 (external sketch)
- Select “**Pierce**” as relation
- (The endpoint is now touching Sketch7)
- Click “**Exit Sketch**” icon to complete.



To create a plane at the highest point:-

- Click “Features” icon
- Click >> icon at the rightmost
- Click “**Reference**” icon, then select “**Plane**”
- Select “Front Plane”
- Then select the **highest point** of Sketch8
- Press Enter key to complete

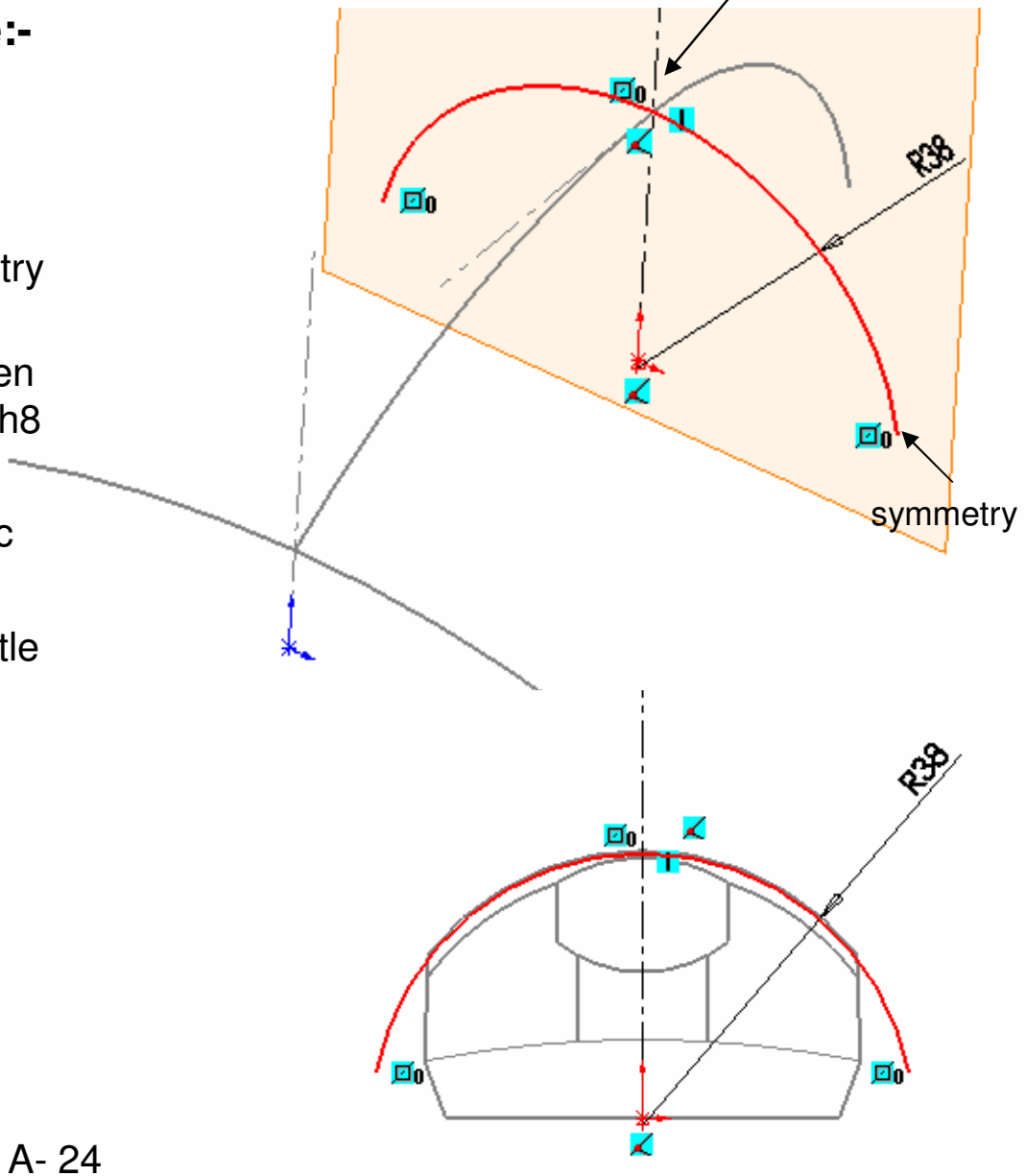


Tutorial 2A

To create a sketch on the new plane:-

(Sketch#9)

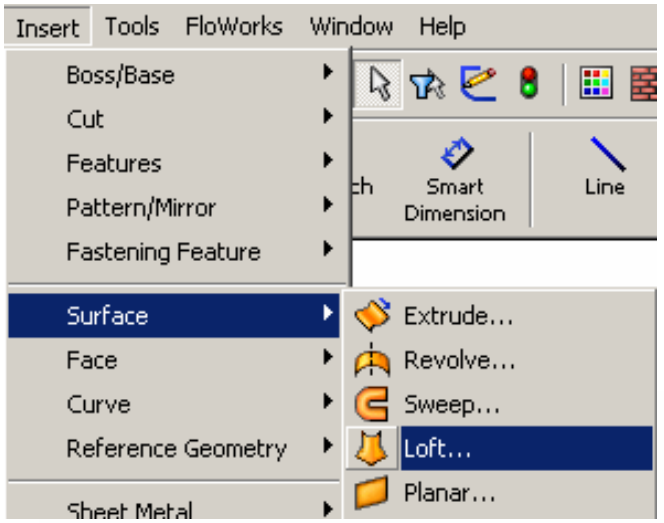
- Click "**Sketch**" icon;
- Select the new plane "**Plane7**";
- Draw an **arc** as shown; add a symmetry constraint onto the endpoints;
- Then add a coincident relation between the arc and the highest point of Sketch8
- Add a dimension **R38mm** onto the arc
- Remark: the endpoints should be a little bit out of the background image
- Click "**Exit Sketch**" icon to complete

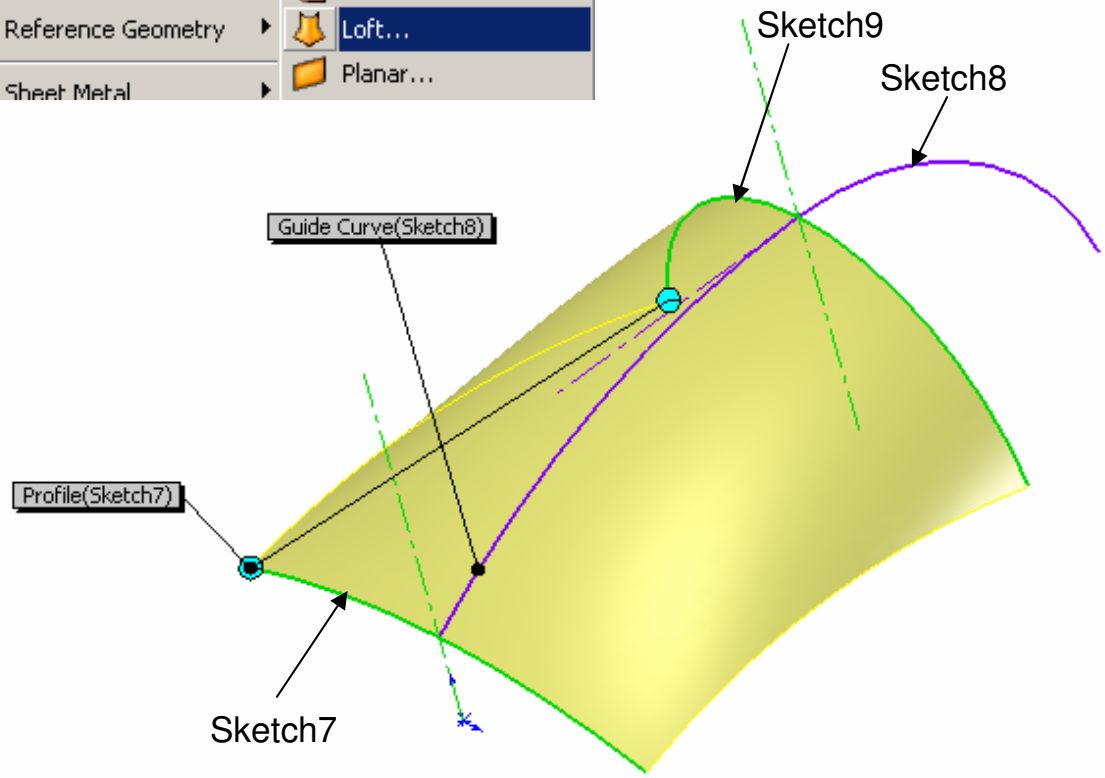


Tutorial 2A

To create a Loft Surface:-

- Select "**Insert** / **Surface** / **Loft**" on the menu
- Select "Sketch7" & "Sketch9" as **Profile**
- Click the entry box "Guide Curves"
- Select "Sketch8" as **Guide Curve**
- Press Enter key to complete



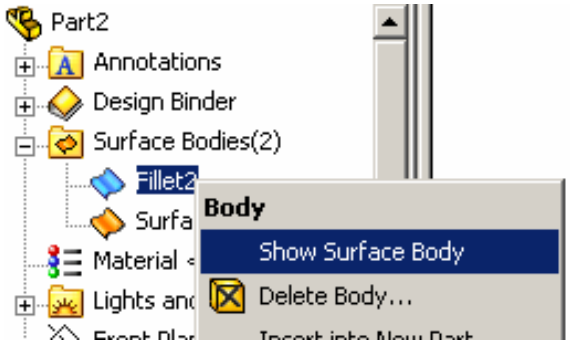


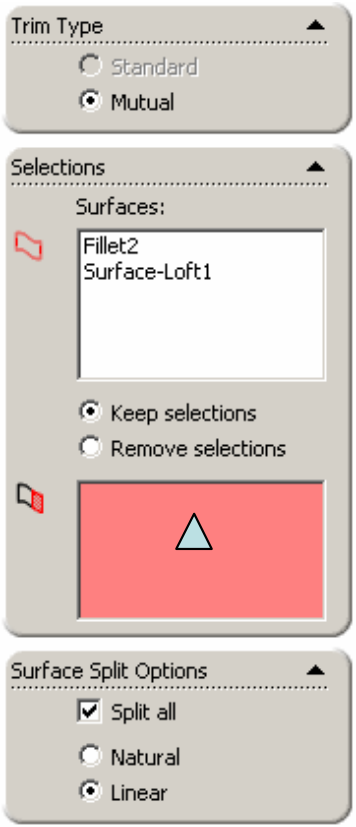
Hide "Sketch7", "Sketch8", "Sketch9" and "Plane7"

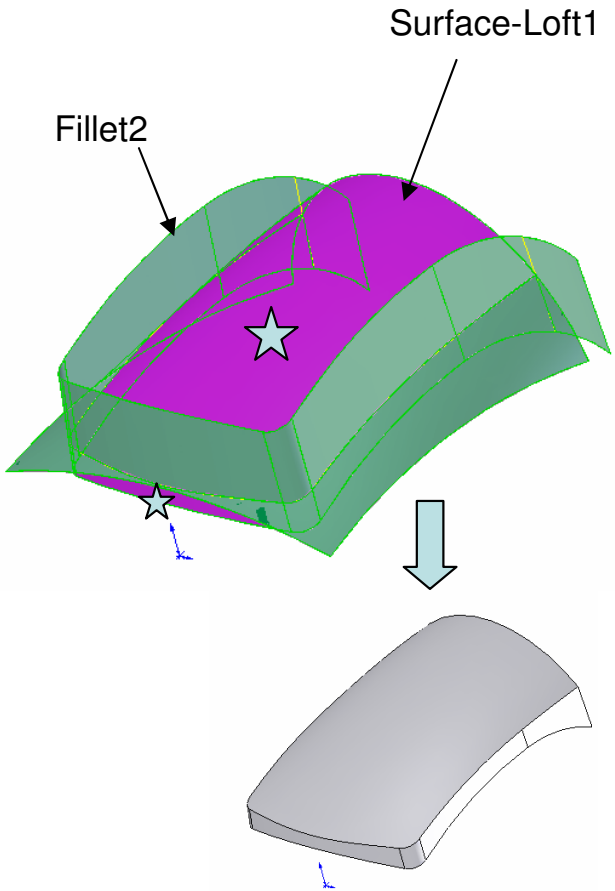
Tutorial 2A

To Trim surfaces and form a joined surface:-

- **Show** “Fillet2” on the tree
- Select “**Insert** / **Surface** / **Trim**” on the menu bar
- Select “Mutual” as Trim Type
- Select surfaces “Fillet2” & ‘Surface-Loft1”
- Select “Keep Selections”
- Select “Split all”
- Click the entry box (Pieces to keep)△
- Select the two faces ☆
- Press Enter key to complete





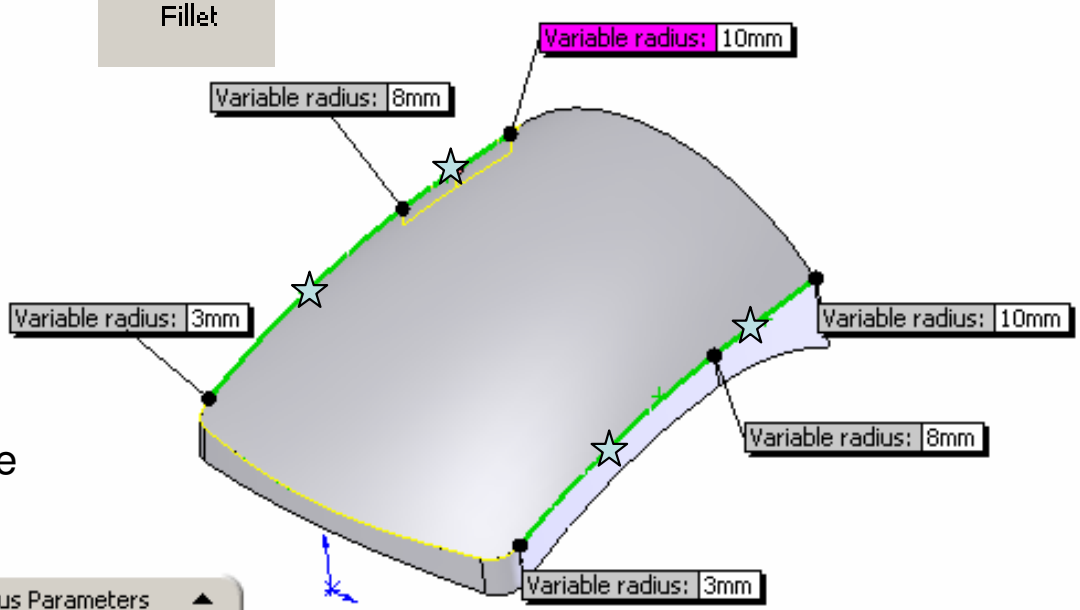


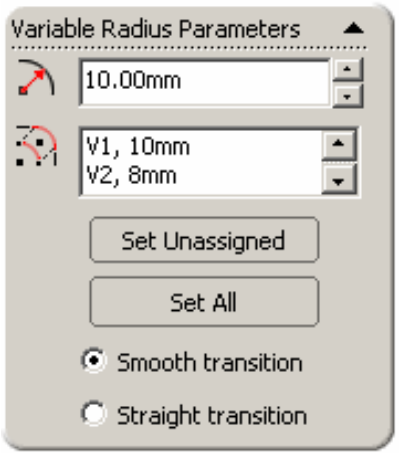
Written by Dickson Sham

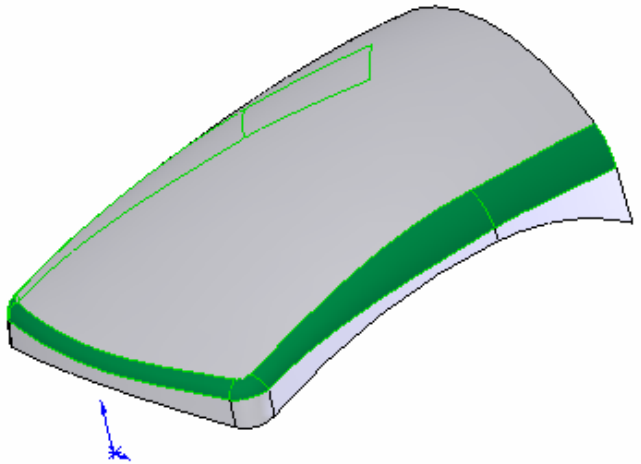
Tutorial 2A

To add a Variable Fillet along the edge:-

- Click “**Fillet**” icon again
- Select “Variable Fillet” as Type
- Select the four edges ☆
- Change “number of instances” to 1
- Select a vertex on the list and enter the value as shown; press “Enter” key
- Enter the value for the next vertex…
- Repeat the steps until the six vertices have been defined
- Press Enter key to complete







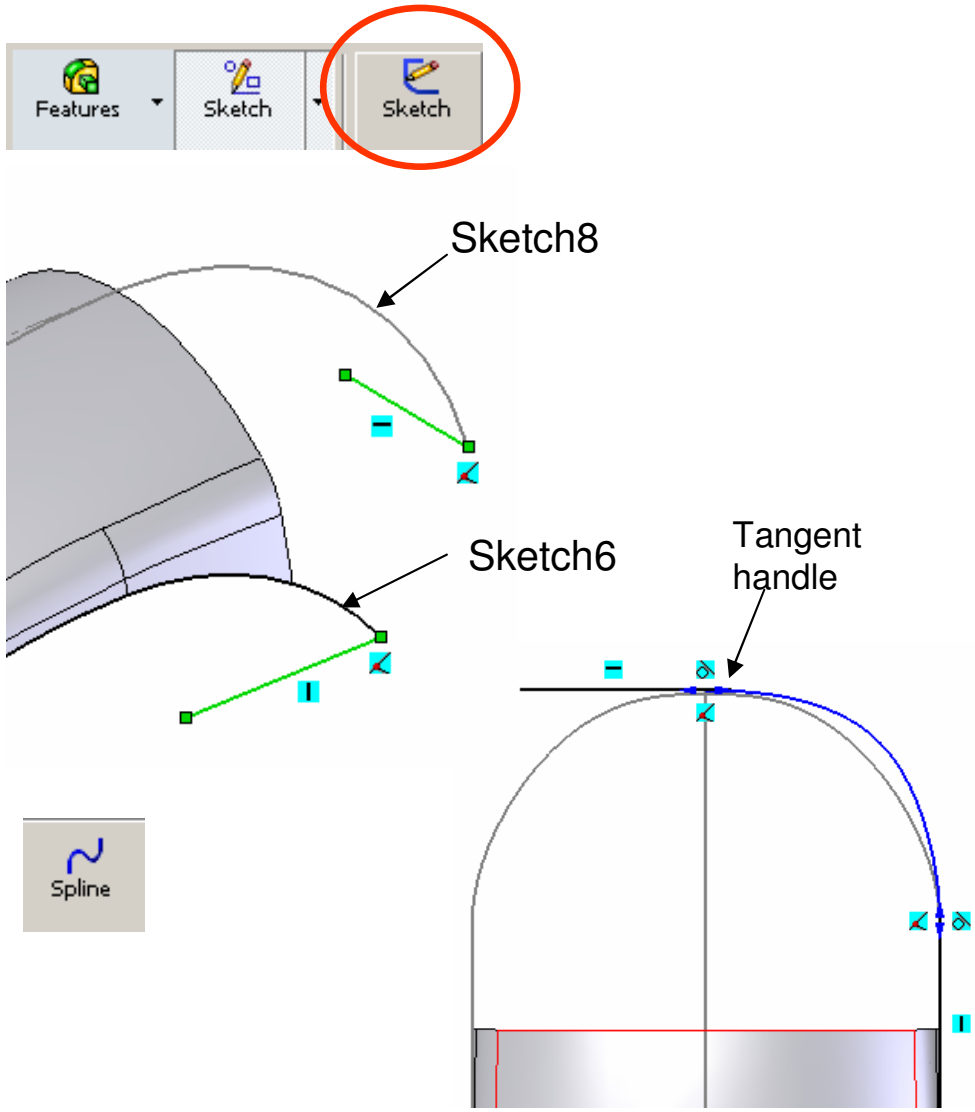
Tutorial 2A

Show "Sketch6" & "Sketch8"

To create a Sketch#10:-

- Click "**Sketch**" icon and select "**Top Plane**"
- Draw a vertical centerline, with one end starting from Sketch6
- Draw a horizontal centerline, with one end starting from Sketch8
- Draw a Spline with 3 control points, connecting the vertical line to the horizontal line
- Add a tangent relation between the spline and the horizontal line
- Similarly, add a tangent relation between the spline and the vertical line

- Drag the control point and also the tangent control handles to match the top view
- (Remark: To prevent making relations with other sketches, press and hold "CTRL" key)

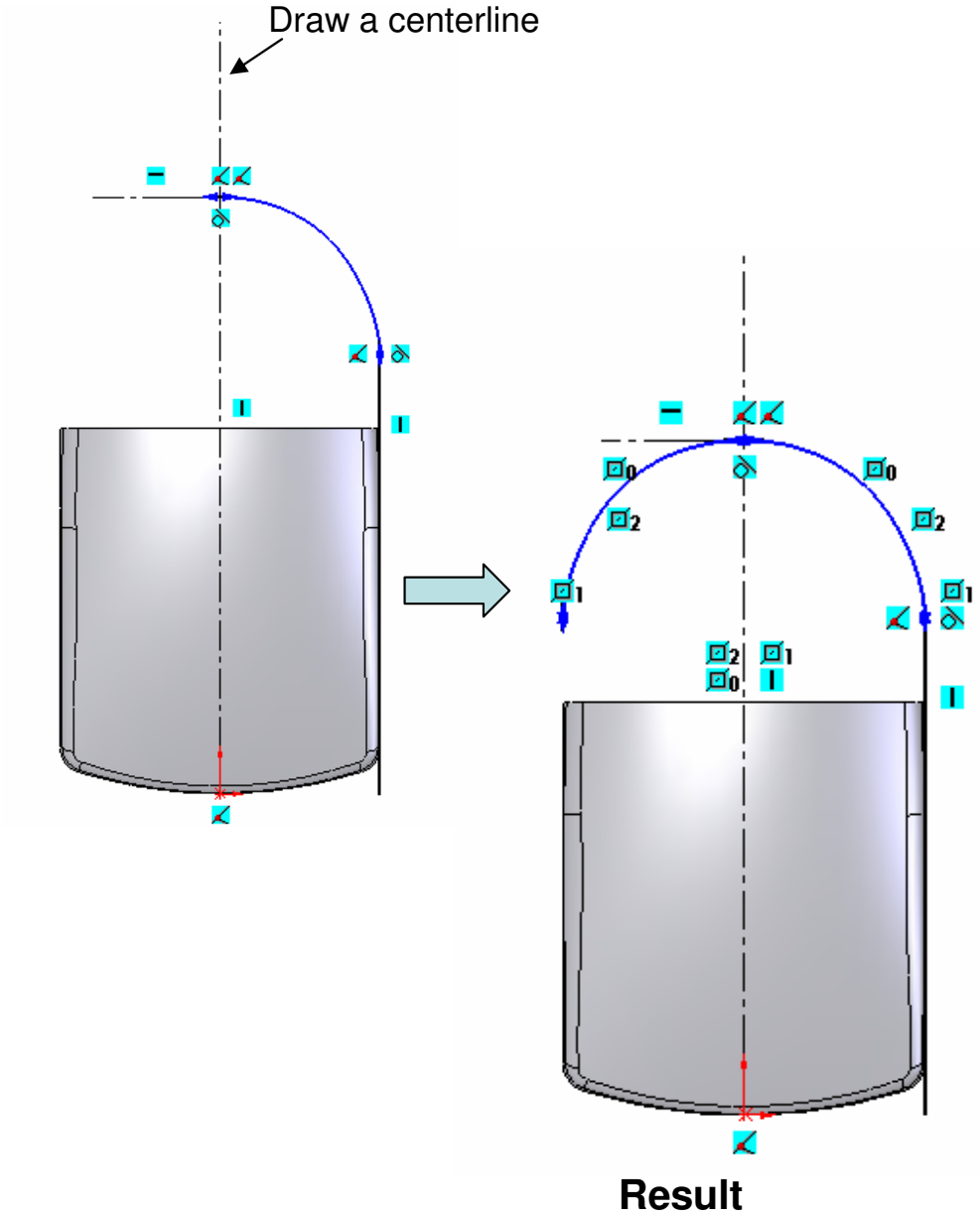


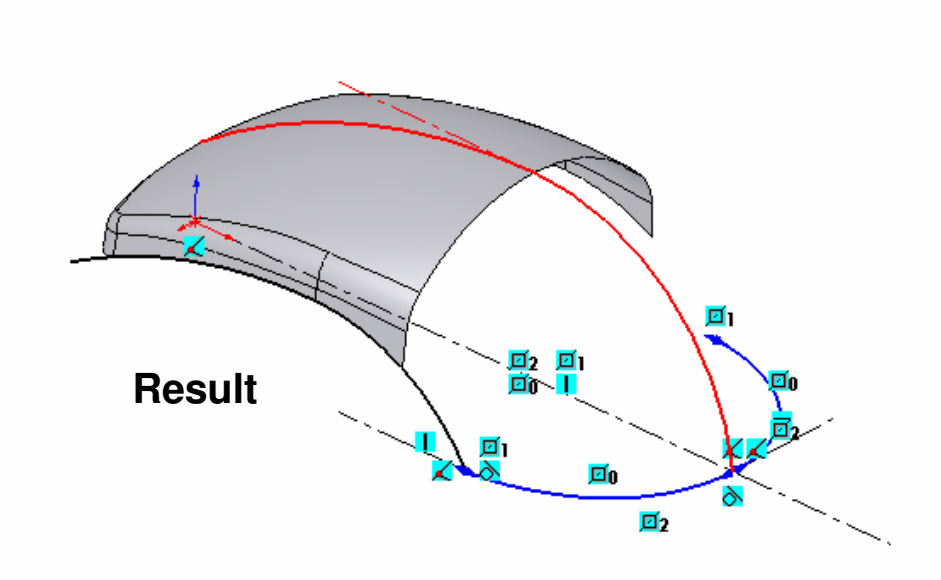
Tutorial 2A

- Draw a Vertical centerline, starting from the origin

- Click “Mirror” icon
- Select the Spline as “entities to mirror”
- Select the centerline as “mirror about”
- Click ok

- Click “**Exit Sketch**” icon to complete

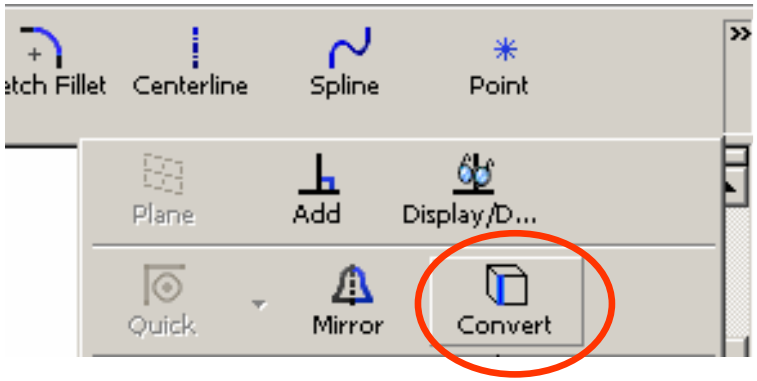




Tutorial 2A

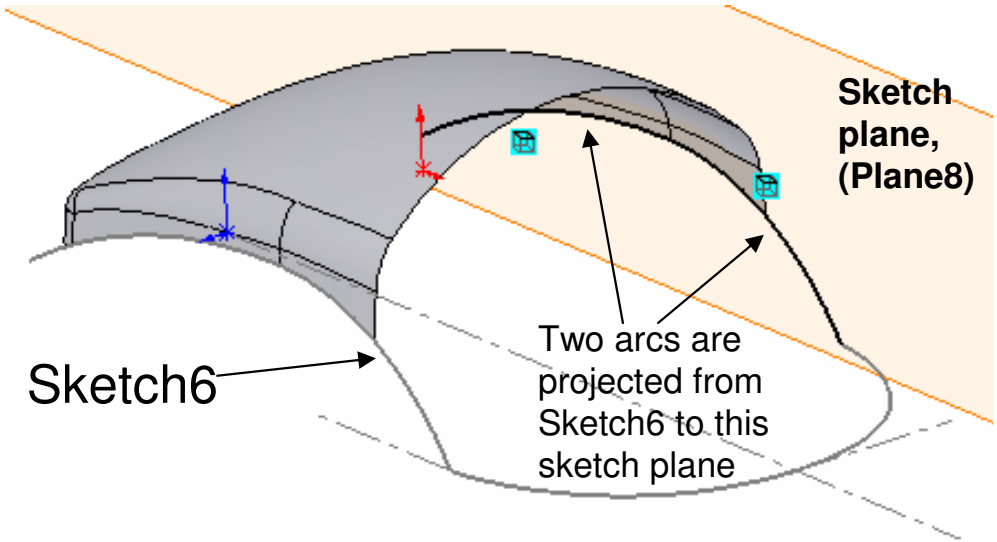
To duplicate Sketch6 about a mirror plane:-

- Click "**Reference**" icon, then select "**Plane**"
- Select **Right Plane** on the tree
- Enter 30.5mm as D1
- Select "Reverse Direction"
- Press "Enter" key to complete



- Click "**Sketch**" icon and select the new Plane
- Multi-select all arcs of Sketch6
- Click "**Convert**" icon
- Click "**Exit Sketch**" icon to complete
- Hide "Plane8"

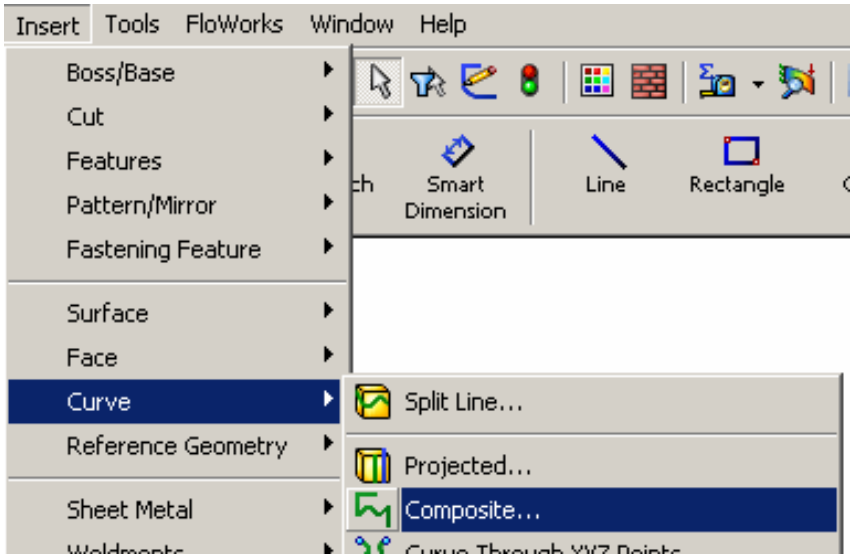
(This is a limitation of Solidworks2006 that sketches, 3d sketches or 3d curves cannot be mirrored in the 3D environment)

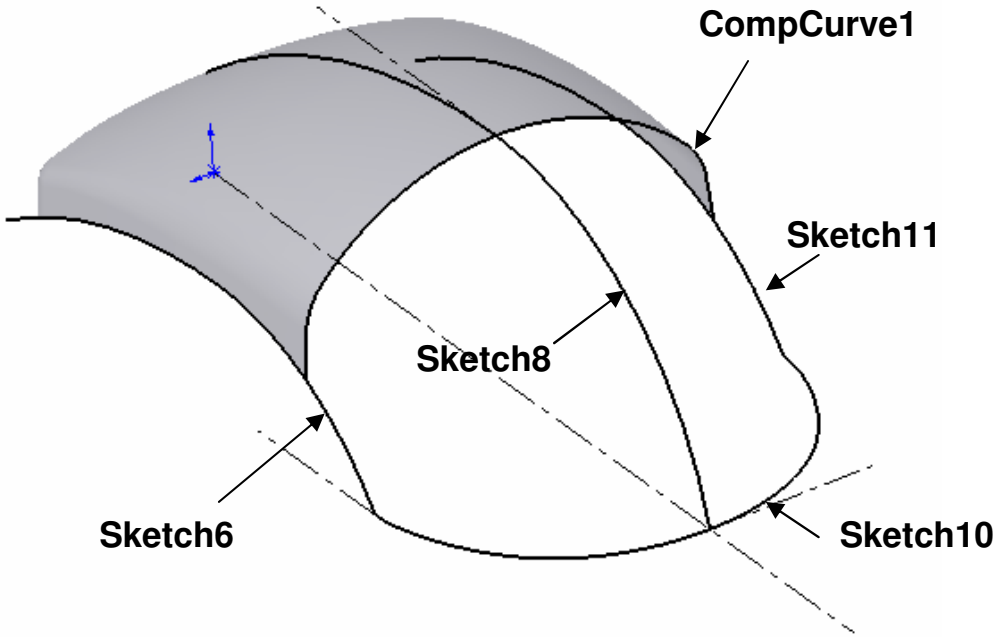


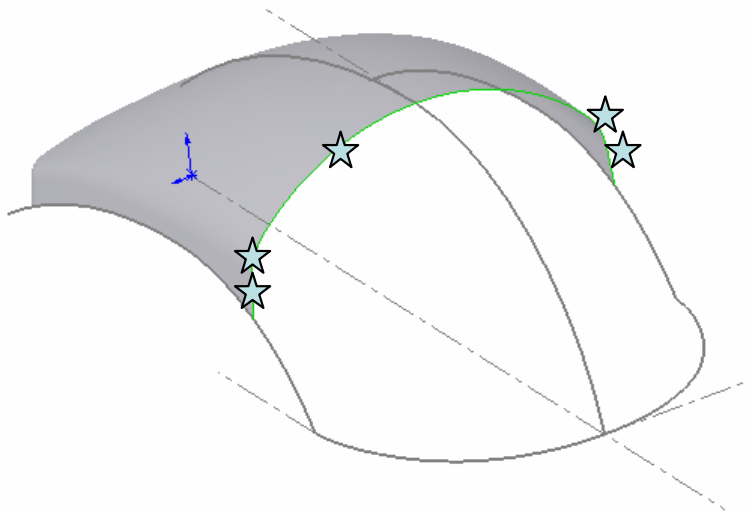
Tutorial 2A

To obtain a boundary of a surface (Loft1):-

- Select "**insert** / **Curve** / **Composite**" on the menu bar
- Multi-select the five edges ☆ of the Loft
- Press Enter key to complete



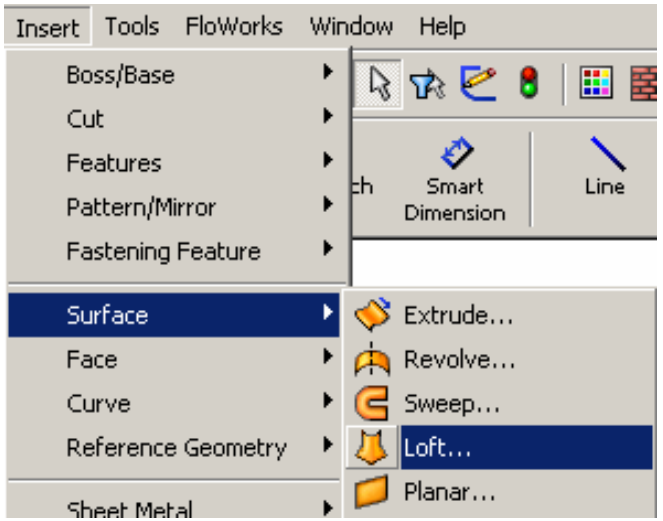




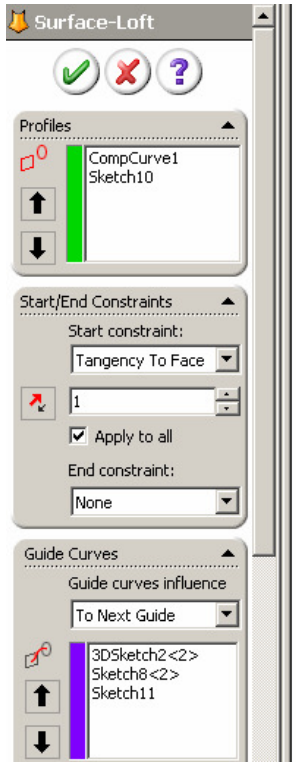
Tutorial 2A

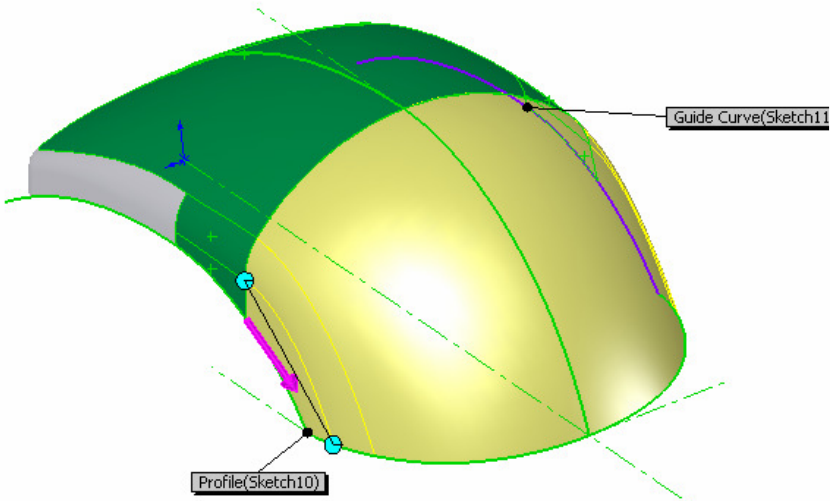
To create a Loft surface:-

- Select “**Insert** / **Surface** / **Loft**” on the menu
- Select “CompCurve1” & “Sketch10” as **Profile** (Ref P.31)
- Click the entry box “Guide Curves”
- Select “Sketch6”, “Sketch8” & “Sketch11” as **Guide Curve** (Ref P.31)
- Select “Tangency to Face” for Start Constraint (so that the new surface will be tangent to the existing surface)
- Press Enter key to complete
- Hide all curves/sketches



A-

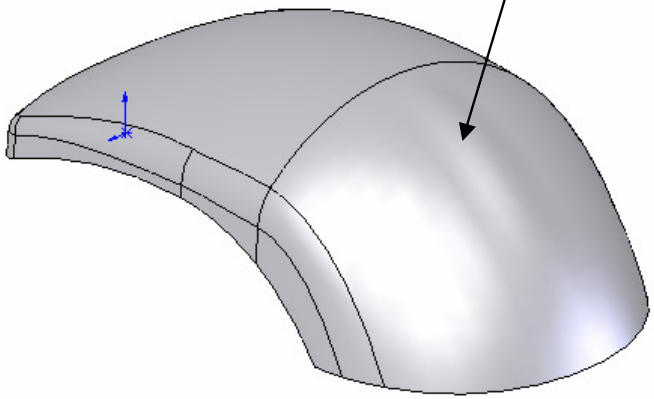




Written by Dickson Sham

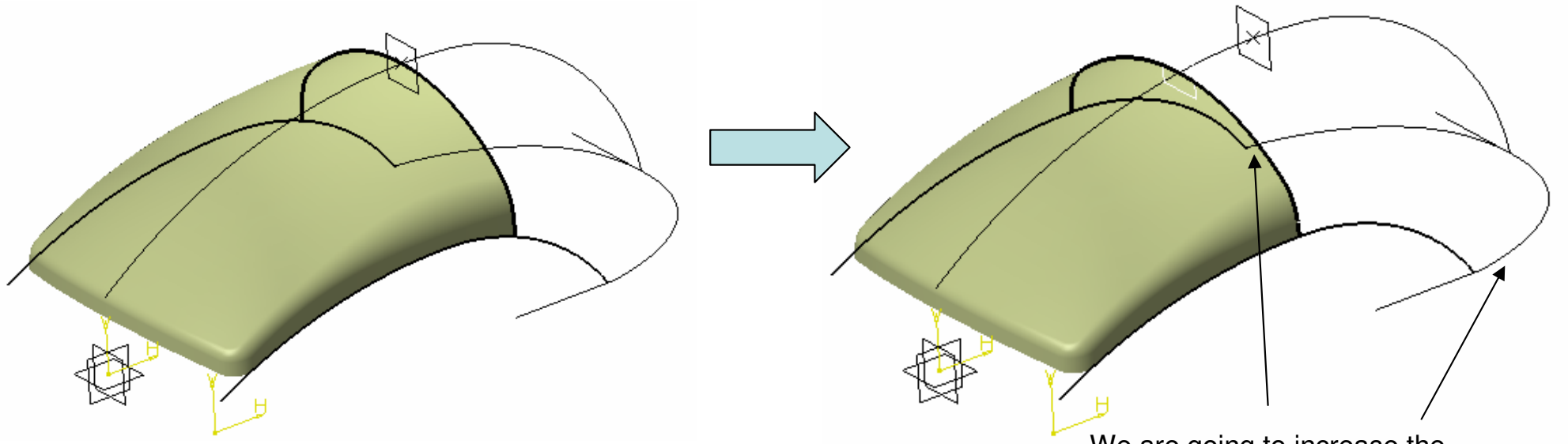
Tutorial 2A

The resultant surface has an obvious shrink mark, which needs further improvement



To improve the smoothness of a Loft Surface:-

One of the common strategies is to increase the distance between two sections so that there is more room to transform one section into the other one, thus decreasing the shrink marks on the resultant surface.

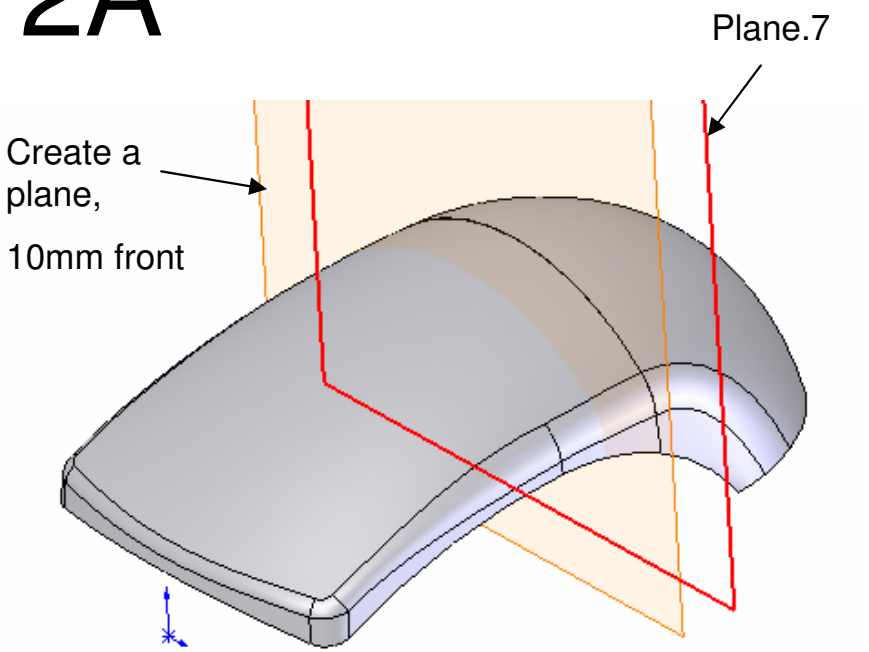


We are going to increase the distance between two sections

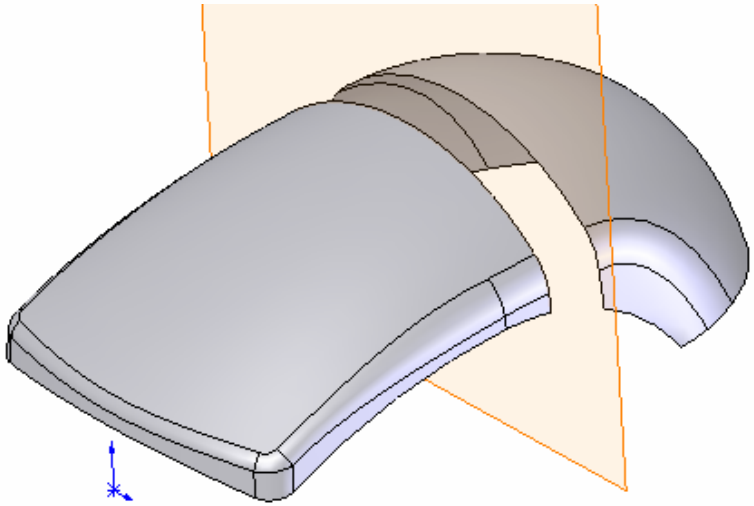
Tutorial 2A

To improve the smoothness of a Loft surface:-

- Click “**Reference**” icon, then select “**Plane**”
- Select “**Plane7”** and enter **10mm** as offset value
- Press “Enter” key to complete

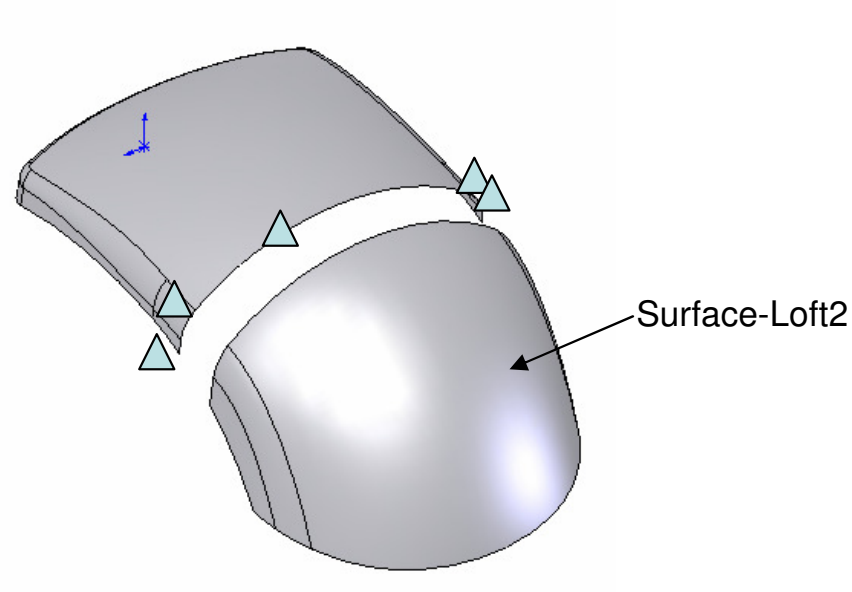


- Select “**Insert** / **Surface** / **Trim**” on the menu bar
- Select “Standard” as Trim Type
- Select the new plane as Trim Tool
- Click on the front portion of VarFillet1
- Press Enter key to complete



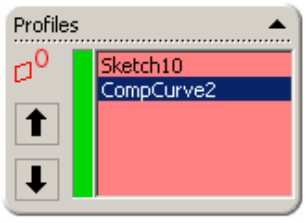
Tutorial 2A

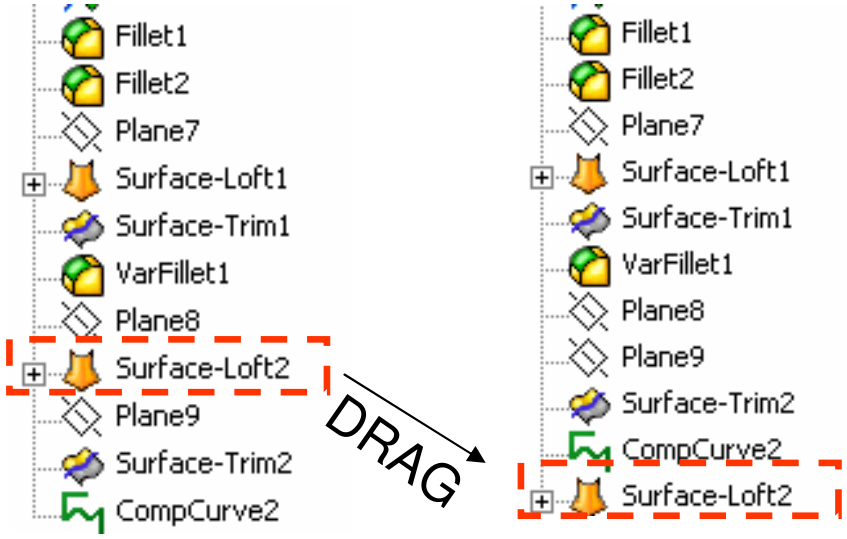
- Select "**insert** / **Curve** / **Composite**" on the menu bar
- Multi-select the five edges of the front Loft
- Press Enter key to complete
- (A new composite curve is created, CompCurve2)



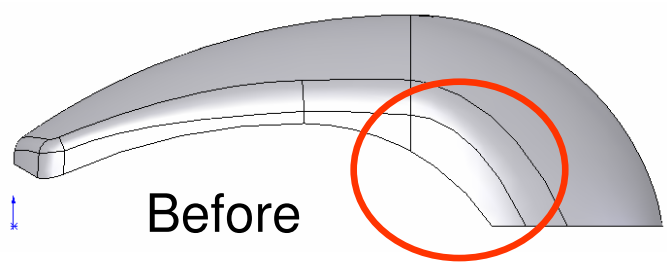
- Drag "**Surface-Loft2**" on the tree and Pull it after "**CompCurve2**"

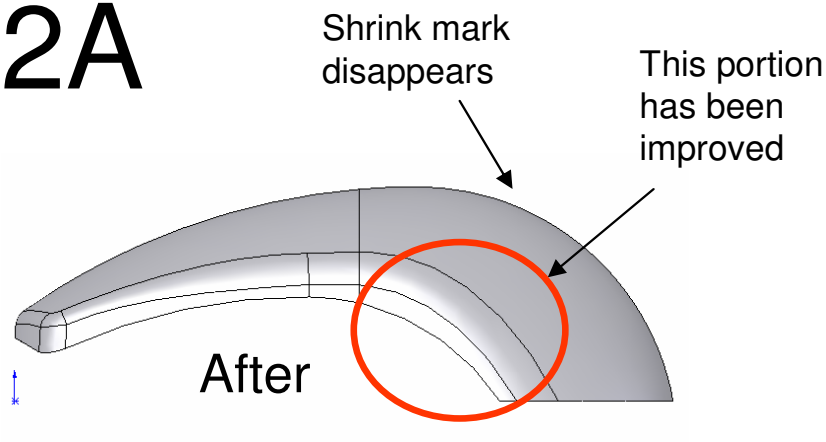
- Right-Click "Surface-Loft2" and select "**Edit Feature**"
- Delete "CompCurve1" on the list
- Select "CompCurve2" as Profile
- Select "Tangency to Face" for End Constraint
- Press Enter key to complete
- Hide "CompCurve2"





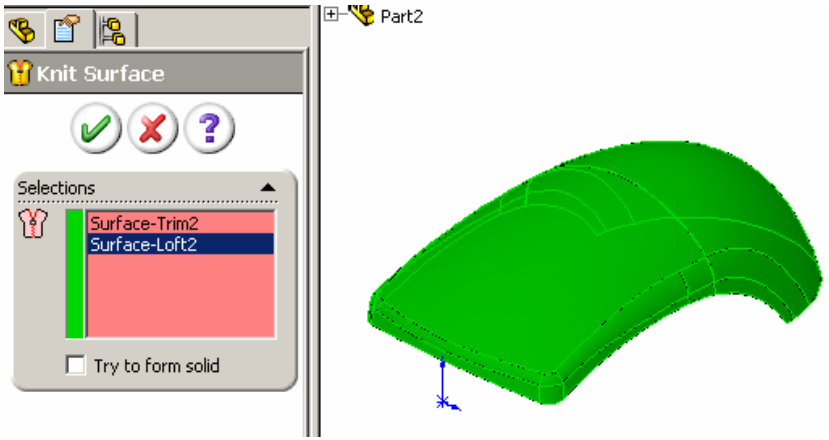
Tutorial 2A





To join two surfaces into one:-

- Select "**Insert/ Surface/ Knit**" on the menu bar
- Select Two Loft Surfaces
- Press Enter key to complete



To save the file:-

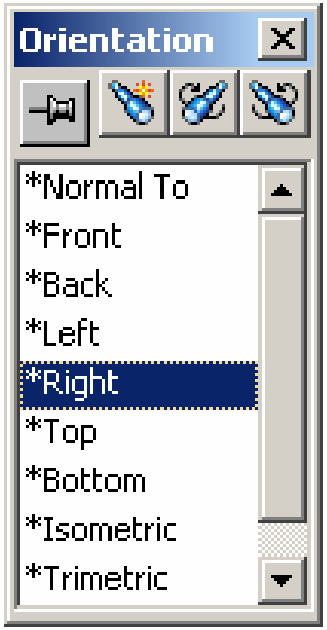
- Select **File/Save** on the menu bar;
- Select your project folder;
- Enter "**Mouse_mastermodel_a.sldprt**" as the file name.

END of Tutorial 2A

Written by Dickson Sham

Tutorial 2B

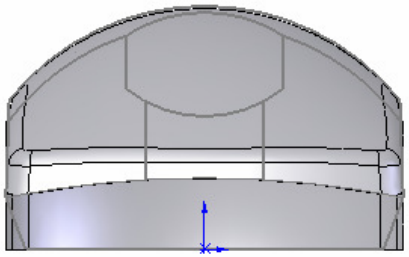
We continue to build the skin of the upper part. After that, we need to check if the whole skin can meet the required shape and has no undercut portion…

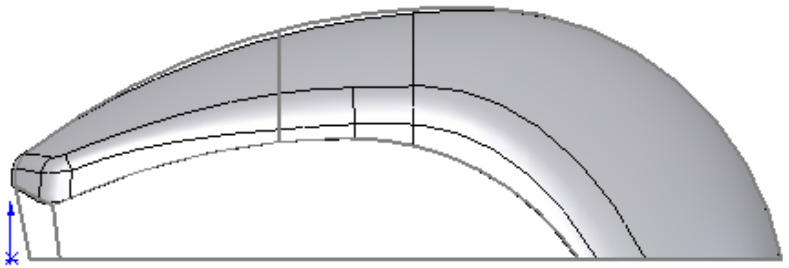


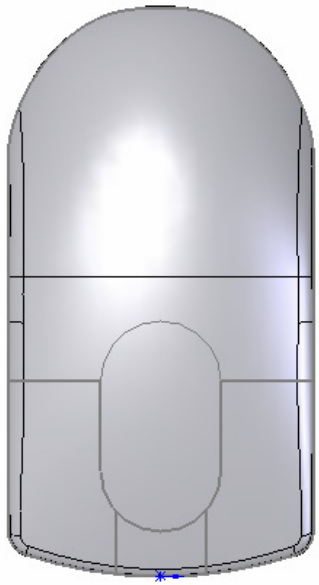
- *Reopen the file* "Mouse_mastermodel_a.sldprt"
- Press "Space" key to activate the orientation selection
- Click **"Front View"** icon to check the front view
- Click **"Top View"** icon to check the top view
- Click **"Right View"** icon to check the right view

  (Remark: the surface should match the three reference views)

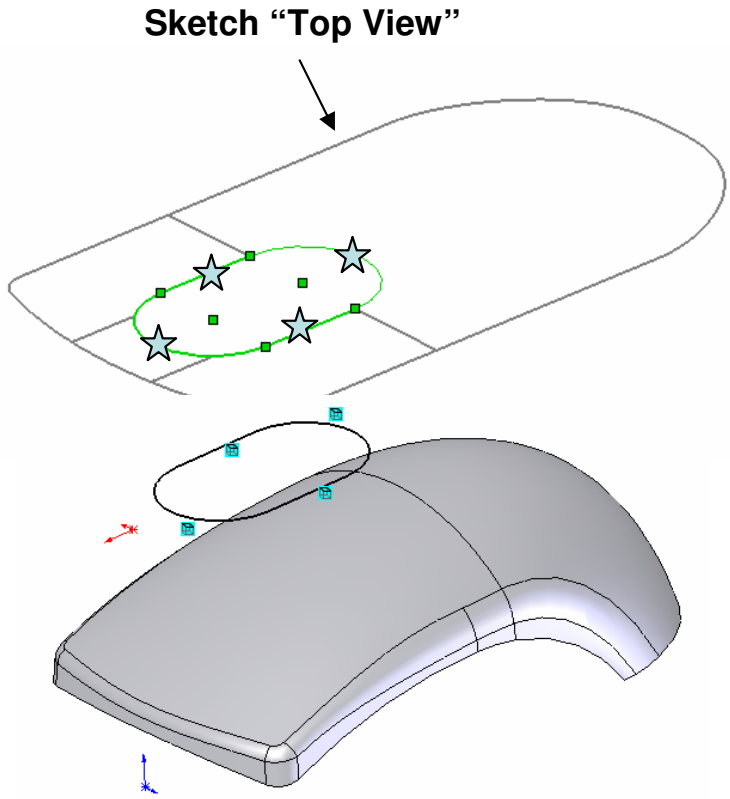
  (Remark: In the real case, the dimensions & locations of all control curves of the surfaces should be referred to these imported reference views)







Tutorial 2B

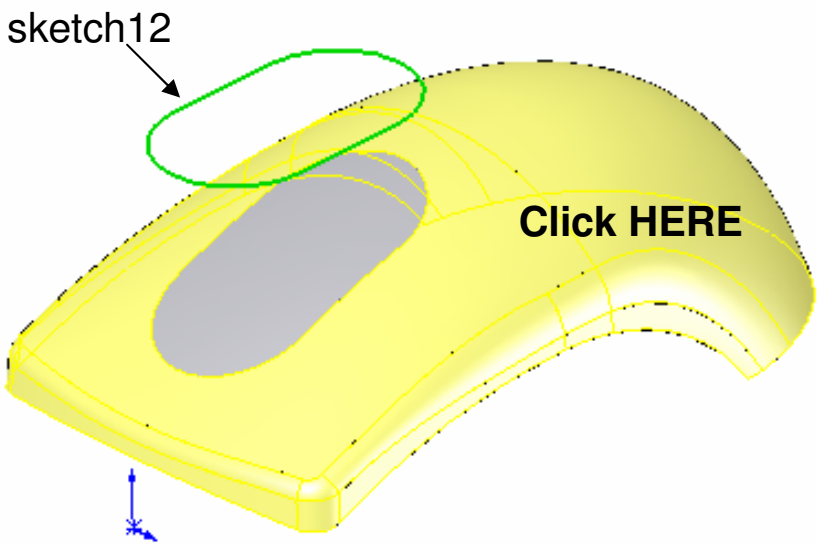


To make a sketch:-

- Click "**Sketch**" icon
- Select "**Plane4**"
- Multi-select the four curves ☆ on the sketch **"Top View"**
- Click "**Convert**" icon
- (We have 4 projected curves from "Top View")
- Click "**Exit Sketch**" icon to complete

To remove surface by the projection of a curve:-

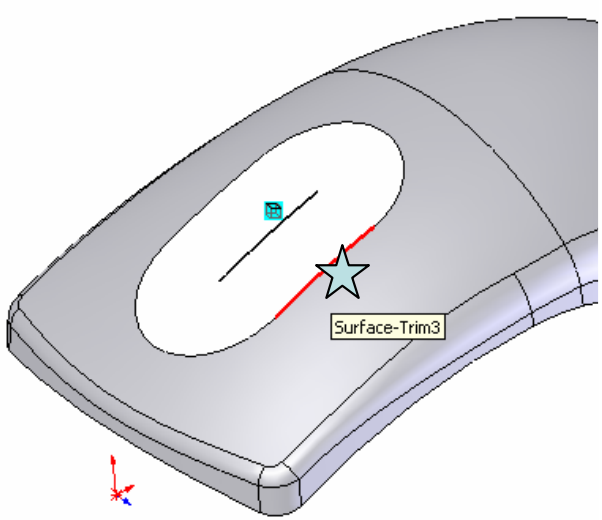
- Select "**Insert**/ **Surface** / **Trim**" on the menu bar
- Select "Sketch12" as Trim Tool
- Select "Keep Selections" and then click on the surface "Surface-Knit1"
- Press "Enter" key to complete
- ("Sketch12" is hidden automatically)

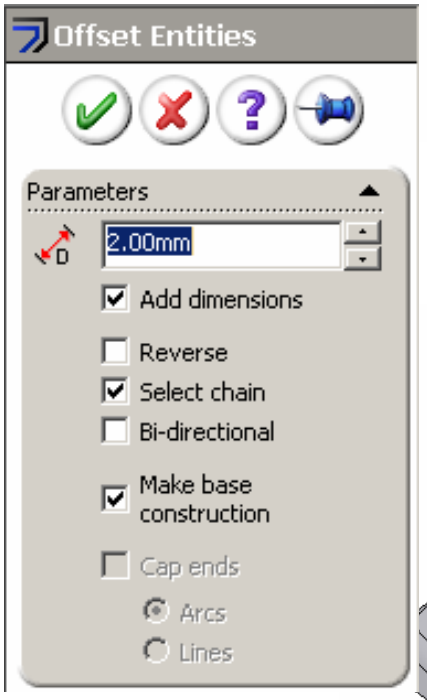


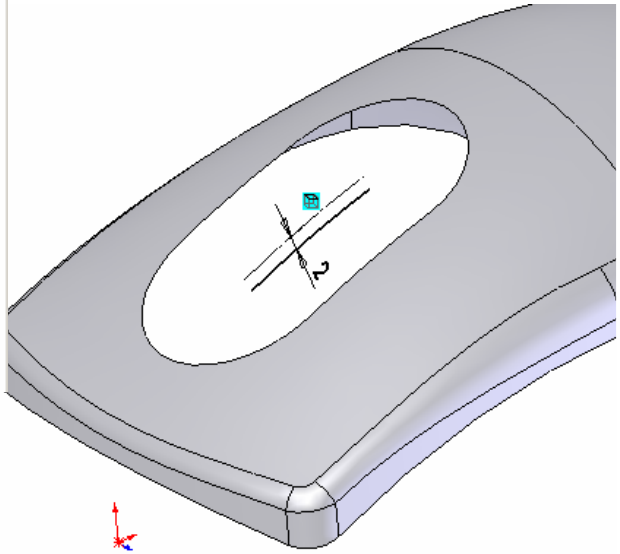
Tutorial 2B

To create a Loft Surface:-

- Click “**Sketch**” icon
- Select “**Right Plane**”
- Select the surface edge ☆
- Click “**Convert**” icon
- (We have a projected curve on the sketch plane)
- Select the projected curve
- Click “**Offset**” icon
- Enter 2mm as D (offset value)
- Select “Make base construction” (so that the projected curve will become a construction element after this command)
- (Remark: the offset direction should be downward; if not, click “reverse”)
- Press Enter key to complete
- Click “**Exit Sketch**” icon

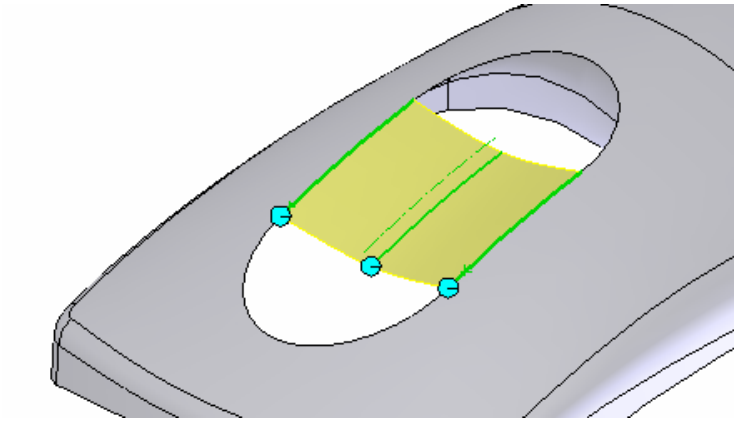


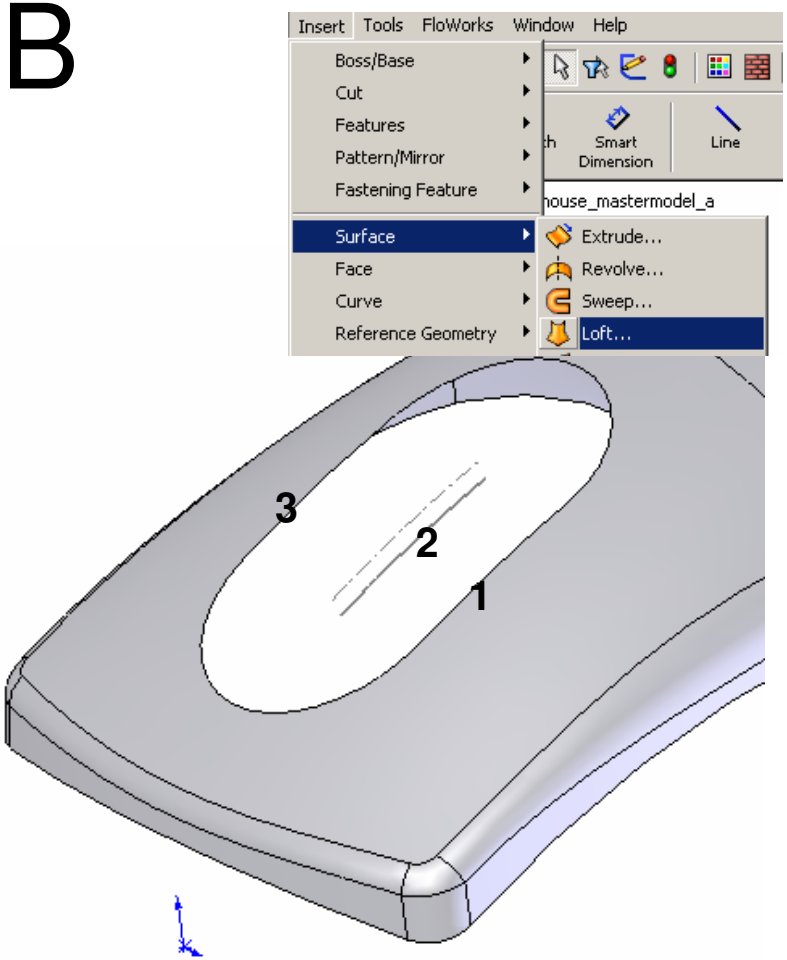




Tutorial 2B

- Select “**Insert** / **Surface** / **Loft**” on the menu bar
- Select the Surface Edge1
- Select “Sketch13”
- Select the Surface Edge3
- Press Enter key to complete



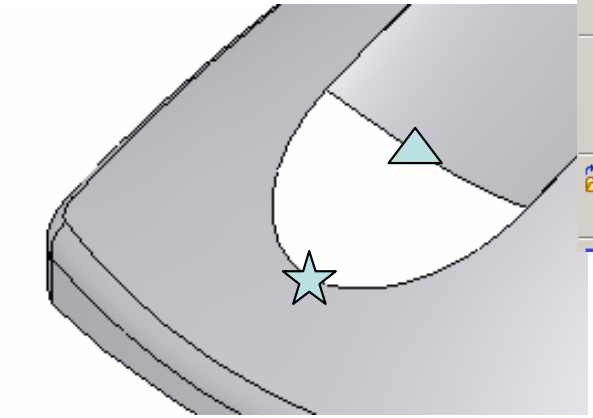


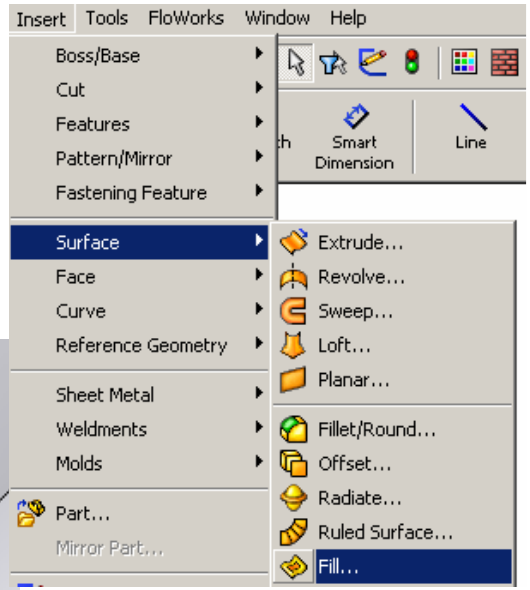
Tutorial 2B

To create a Fill Surface:-

- Select "**Insert** / **Surface**/ **Fill**" on the menu bar
- Select the edge △
- Select "tangent" as curvature control

- Select the edge ☆
- Select "contact" as curvature control
- Press Enter to complete

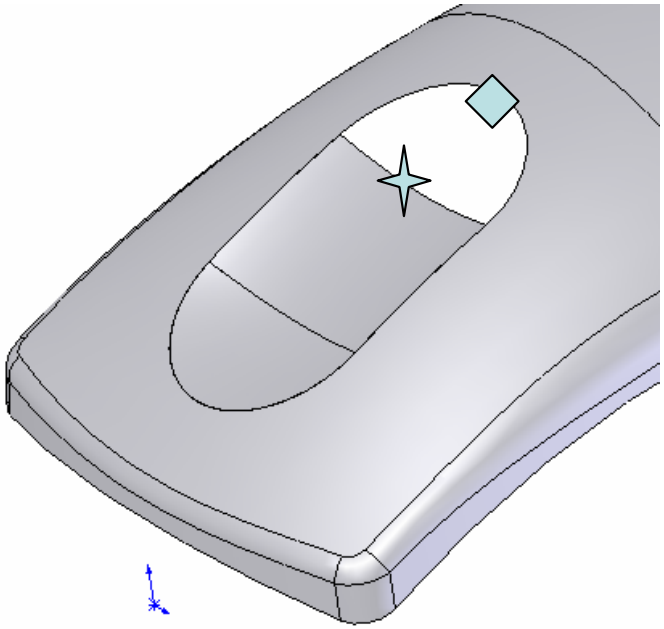




To create another Fill surface:-

- Select "**Insert** / **Surface**/ **Fill**" on the menu bar
- Select the edge ✦
- Select "tangent" as curvature control

- Select the edge ◇
- Select "contact" as curvature control
- Press Enter to complete



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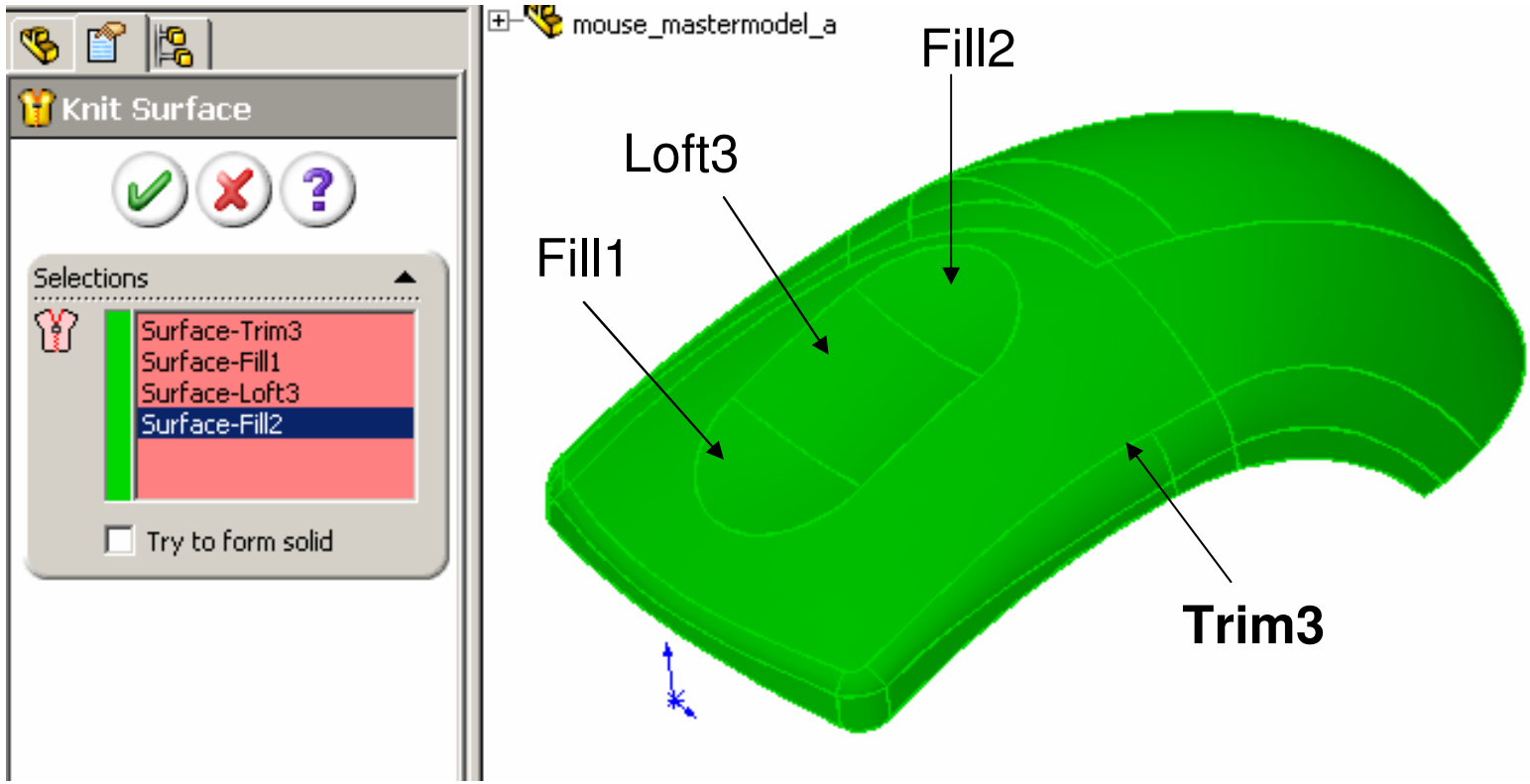
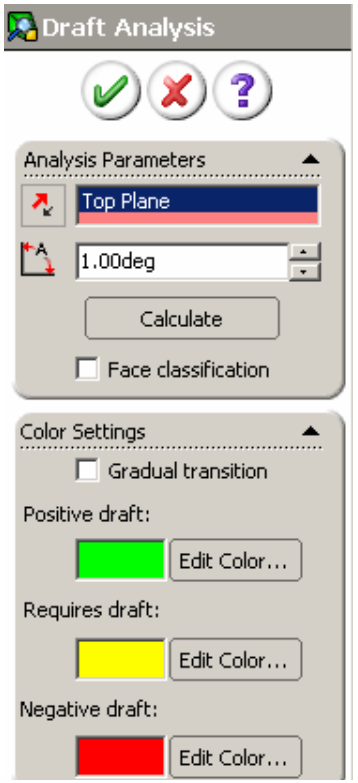
Tutorial 2B

To join surfaces as one:-

- Select "**Insert/ Surface/ Knit**" on the menu bar
- Select all surfaces
- Press Enter key to complete



Tutorial 2B



To do the draft analysis on the surface:-

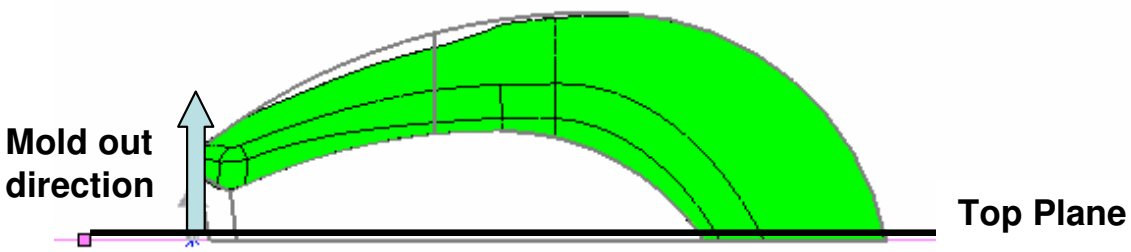
- Select "**Tools** / **Draft Analysis**" on the menu bar
- Select "Top Plane" on Direction of Pull
- Enter 1deg as A (Angle)
- Click "Calculate" icon

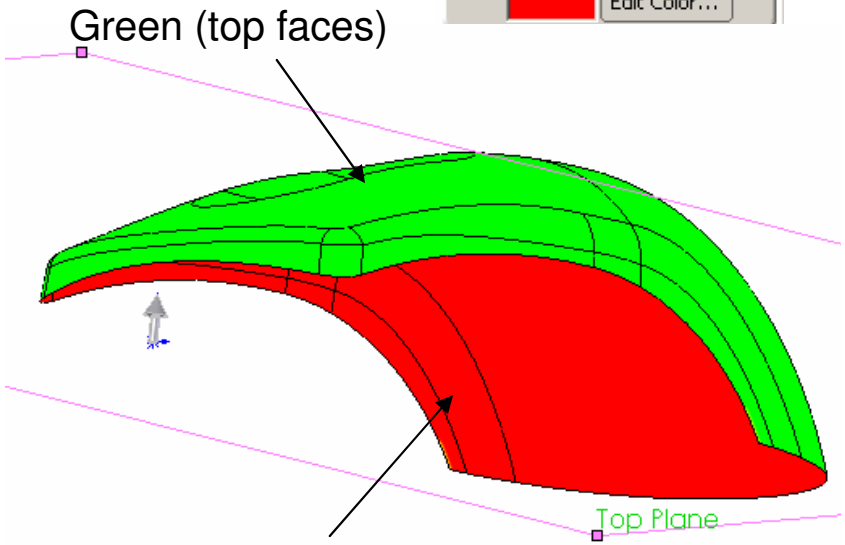
- If the big surface has no undercut, it should either all Green or all Red.

  Green, draft > 1deg

  Yellow, draft = 0deg

  Red, draft < -1 deg



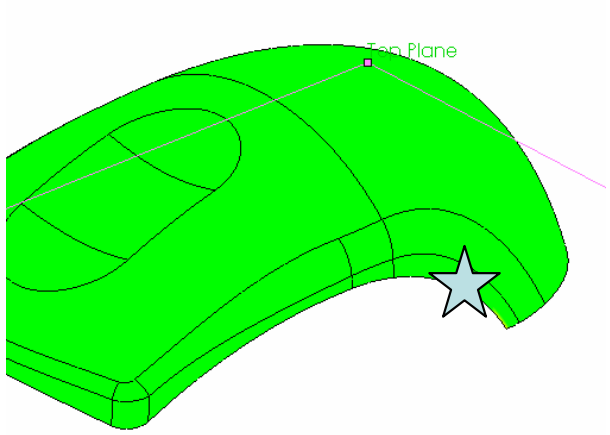


(Remark: the top faces are Green but the bottom faces are Red; that means they must be molded out in opposite directions)

Tutorial 2B

If the big surface has no undercut, it should either all Red or all Green.

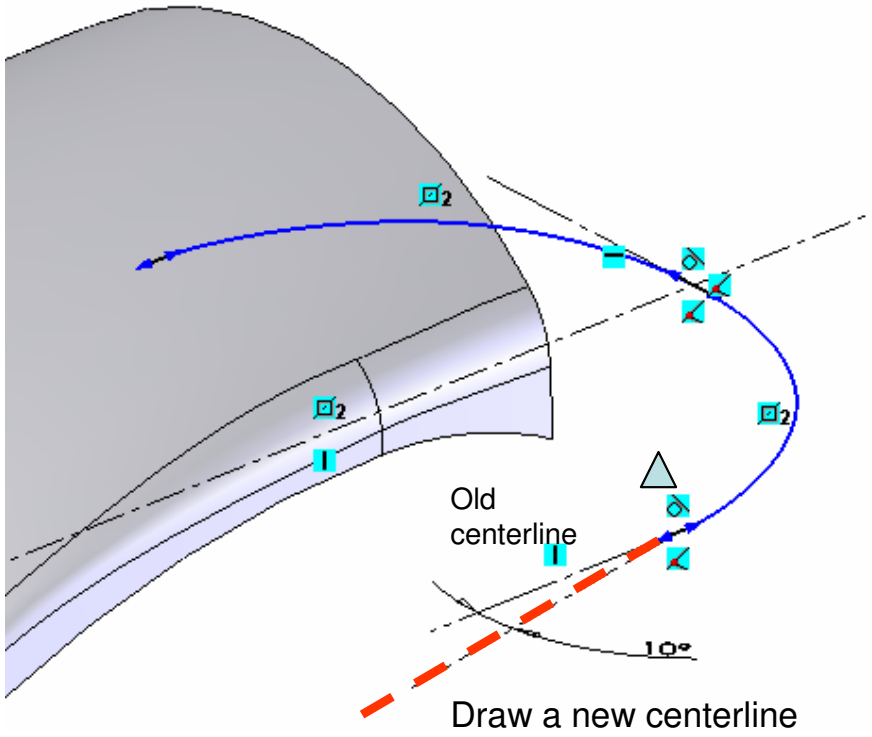
- We find a portion ☆ with Zero Draft (Yellow) along the parting line.
- Click **OK** to exit



To modify the curvature of the problem surface:-

- Right-Click “Sketch10” on the tree
- Select “Edit Sketch”
- Draw a new centerline (10deg from the existing vertical centerline) as shown
- Delete the tangent relation △ (between the spline and the vertical centerline)
- Add a tangent relation between the spline and the new centerline
- Click “Exit Sketch” icon to complete

(All surfaces will be updated automatically)



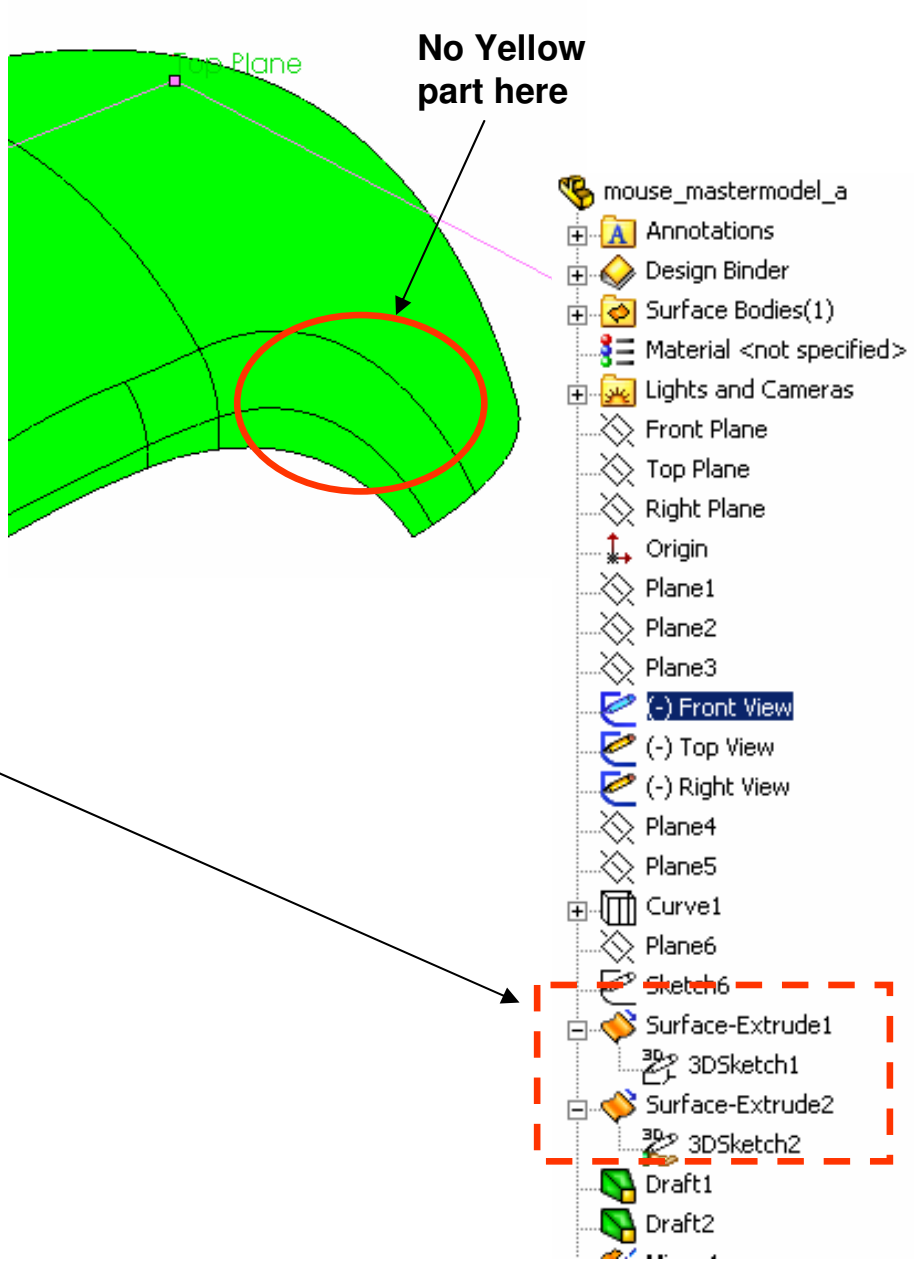
Draw a new centerline

Tutorial 2B

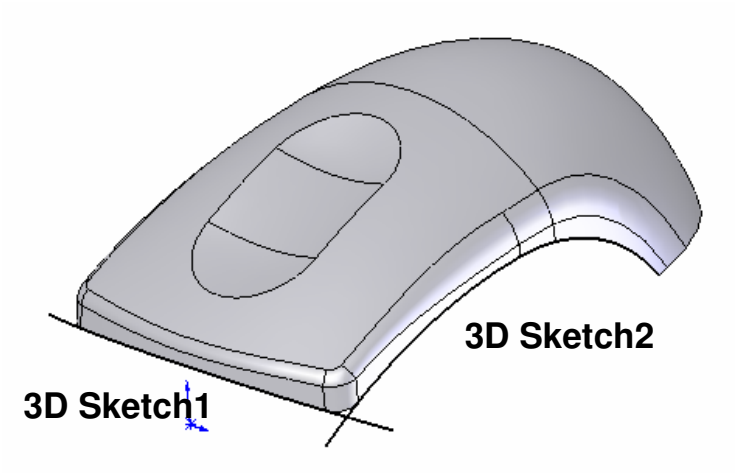
- Select " **Tools** / **Draft Analysis**" on the menu bar again to check if the Zero draft portion (Yellow portion) has been totally removed

- (There should be no Yellow Portion on the surface now)

Now, the upper skin has been completed.

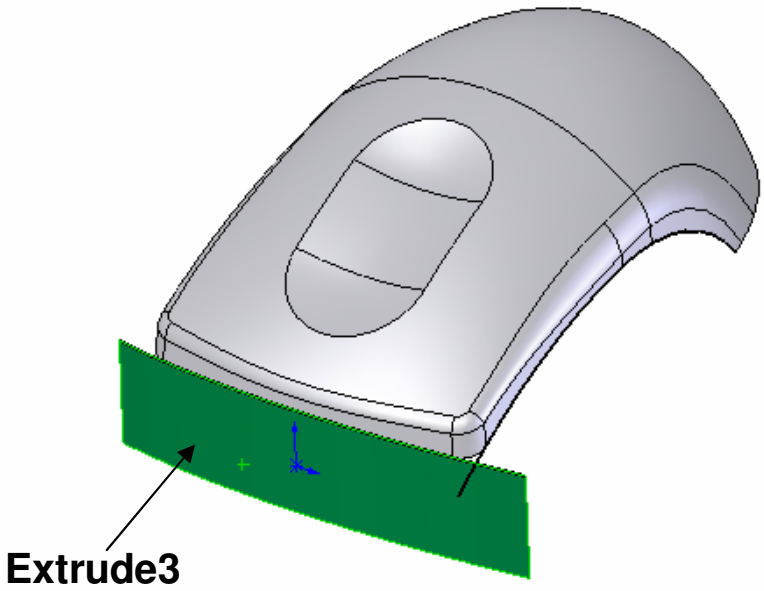


Show "3D Sketch1" & "3D Sketch2" on the tree



Tutorial 2B

We are going to build a lower skin…

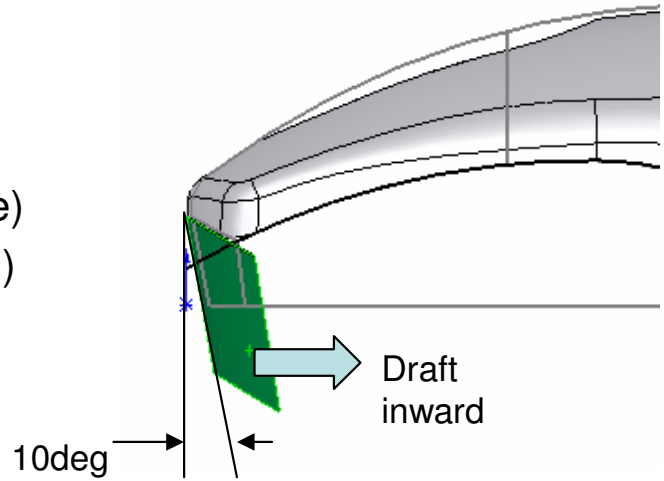


To create an extruded surface:-

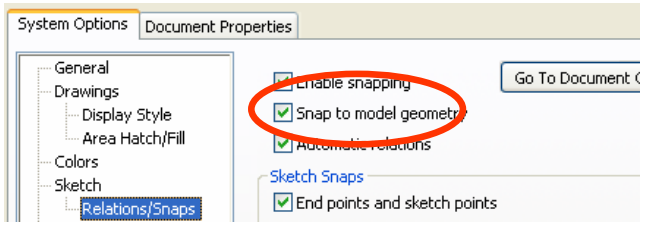
- Select "**Insert** / **Surface** / **Extrude**" on the menu bar
- Select "3Dsketch1" as profile
- Select "Top Plane" as direction
- Click "reverse direction" icon 
- Enter **20mm** as D1
- Press Enter key to complete
- **Hide "3Dsketch1"**

To add a draft onto the extruded surface:-

- Select "**Insert** / **Features** / **Draft**" on the menu bar
- Select "Parting Line" as Type of Draft
- Select "Top Plane" as Direction of Pull
- Select the top edge of "Surface-Extrude3" as the parting line
- (If "3Dsketch" has not yet hidden, you cannot select the surface edge)
- Click "Reverse Direction" icon  (so that the surface is tilted inward)
- Enter **10deg** as Draft Angle
- Press Enter key to complete



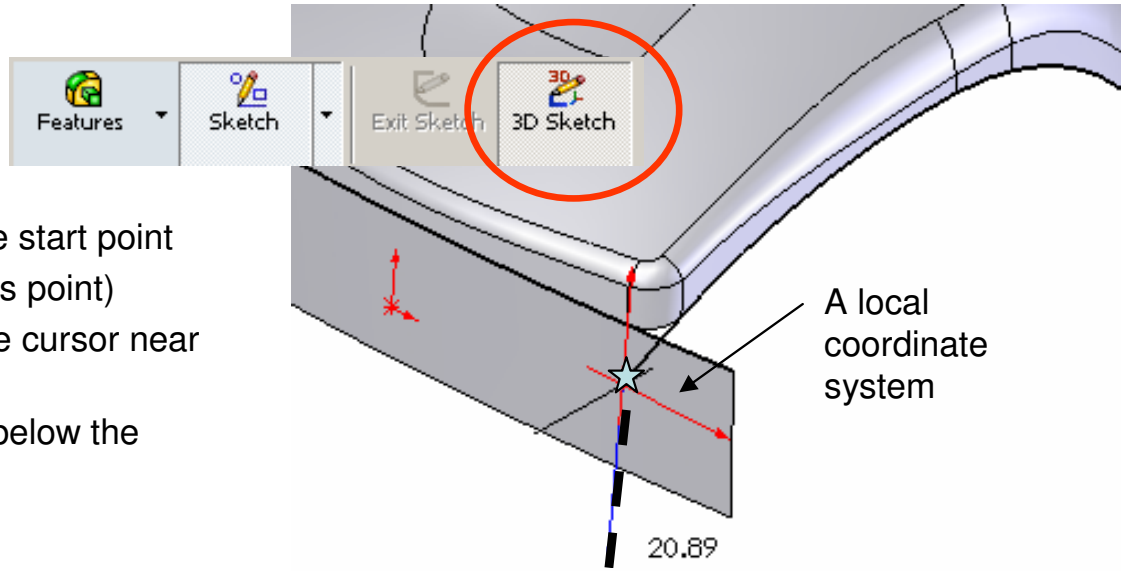
Tutorial 2B



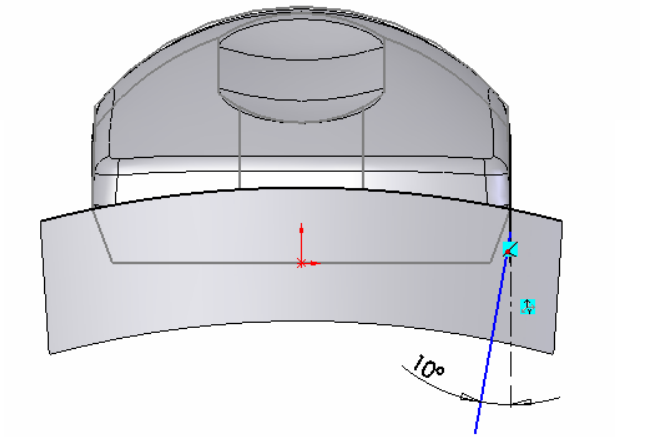
<u>To ENable auto-snapping to any geometry out of the working sketch:-</u>

- Select Tools/Options/System Options /Sketch/Relations
- **Select "Snap to model geometry"**

To draw 3D curves in 3D environment:-



- Click "**3D Sketch**" icon
- Click "Centerline" icon
- Select the endpoint ☆ of "3DSketch2" as the start point
- (a red local coordinate system appears at this point)
- To define the second point, move the mouse cursor near the y-axis until it is snapped onto the axis
- Click at the location where is around 20mm below the local origin
- (a vertical centerline is created)



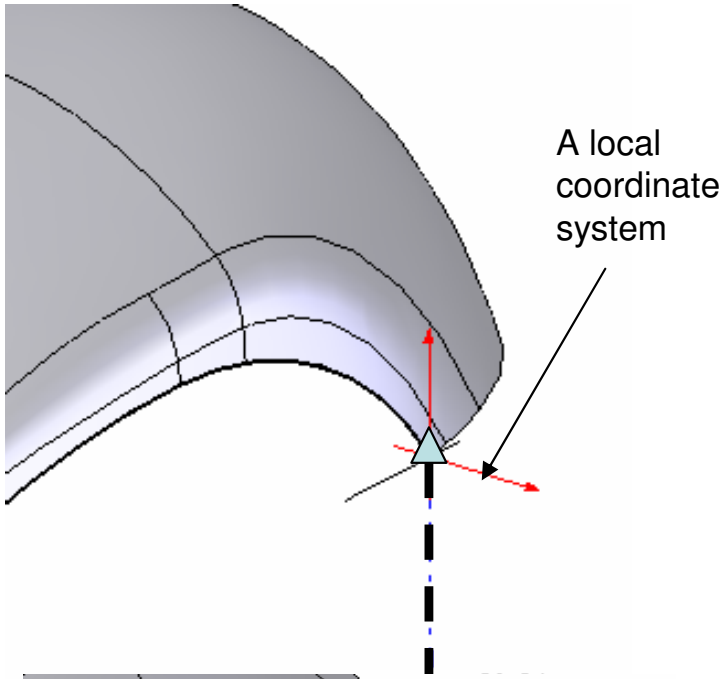
- Click "Line" icon
- Select the endpoint ☆ of "3DSketch2" as the start point
- Press "Space" key on the keyboard
- Select "Front" to switch the front view
- Draw an inclined line (~30mm Length)
- Add an angle dimension <u>10deg</u>

- Click "3D Sketch" icon again to complete

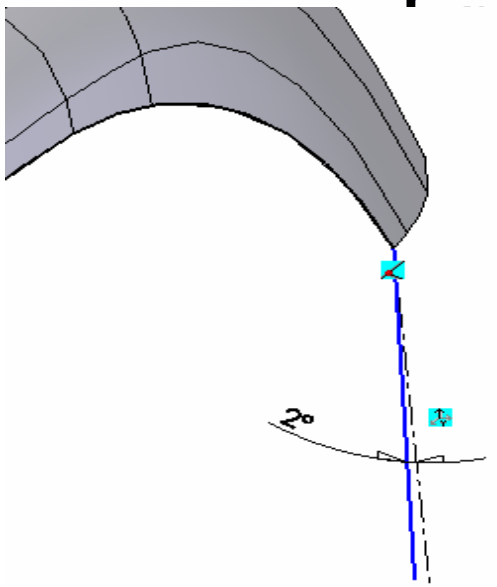
Tutorial 2B

Similarly,

- Click "**3D Sketch**" icon
- Click "centerline" icon
- Select the other endpoint △ of "3DSketch2" as the start point
- (a red local coordinate system appears at this point)
- To define the second point, move the mouse cursor near the y-axis until it is snapped onto the axis
- Click at the location where is around 20mm below the local origin
- (a vertical centerline is created)



- Click "Line" icon
- Select the endpoint △ of "3DSketch2" as the start point
- Press "Space" key on the keyboard
- Select "Front" to switch the front view
- Draw an inclined line (~30mm Length)
- Add an angle dimension 2deg

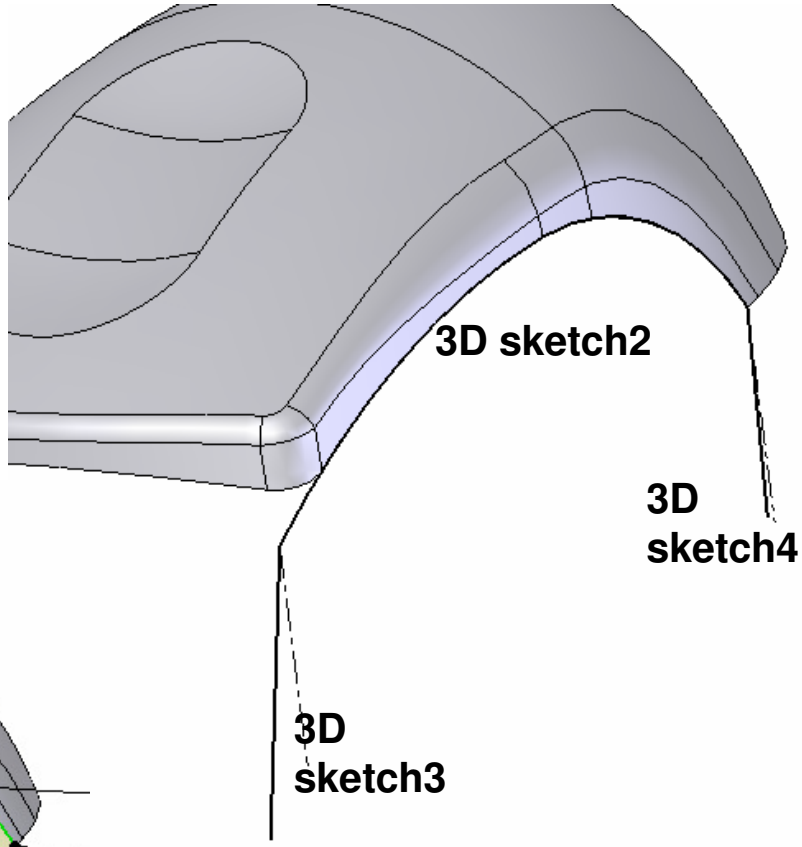


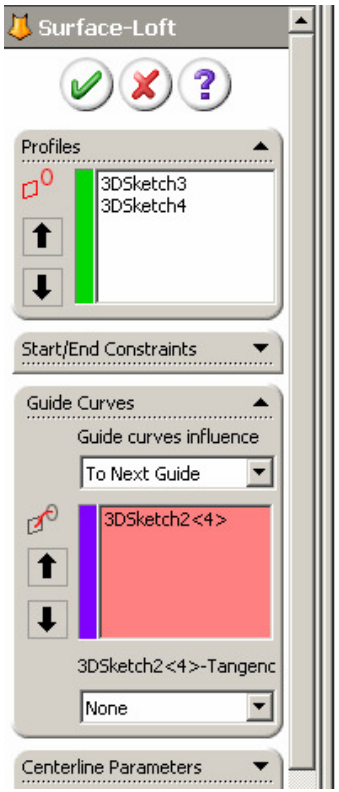
- Click "3D Sketch" icon again to complete

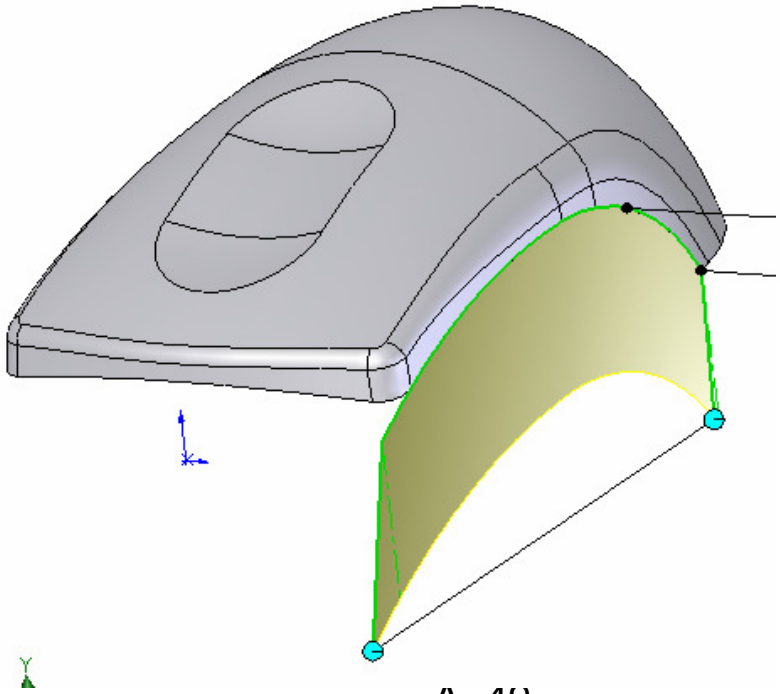
Tutorial 2B

To Create a Loft Surface:-

- Select "**Insert** / **Surface** / **Loft**" on the menu bar
- Select "3Dsketch3" & "3Dsketch4" as Profile
- Select "3Dsketch2" as guide curve
- Press Enter key to complete
- Hide "3Dsketch2" , "3Dsketch3" & "3Dsketch4"



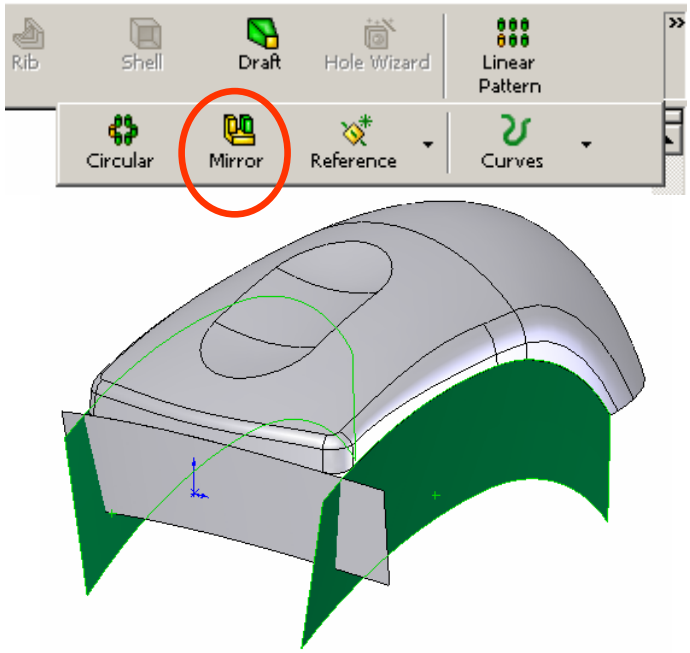


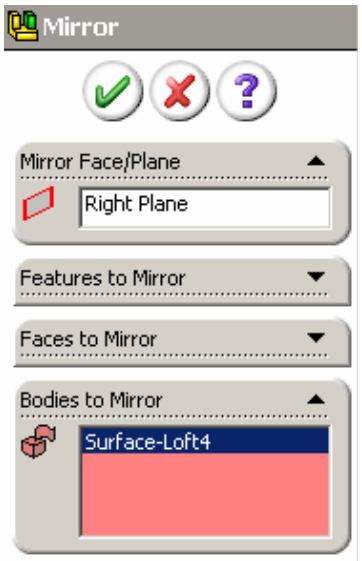


Tutorial 2B

To duplicate a surface by mirroring:-

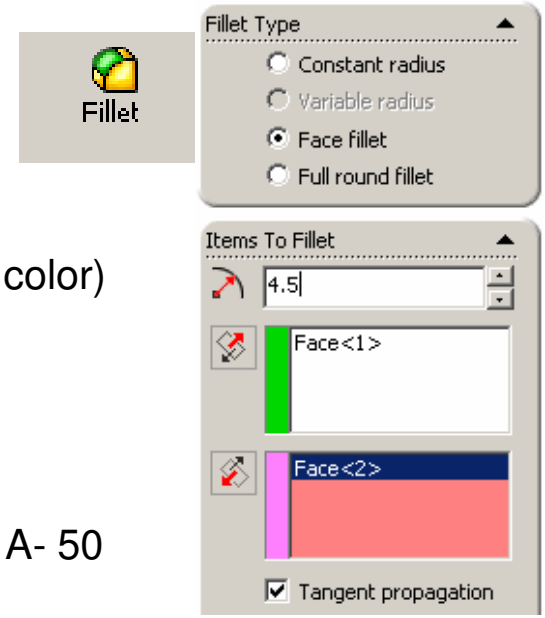
- Click "**Mirror**" icon
- Select "Right Plane" as Mirror Plane
- Click on "Bodies to Mirror"
- Select the surface "Loft4"
- Press Enter key to complete

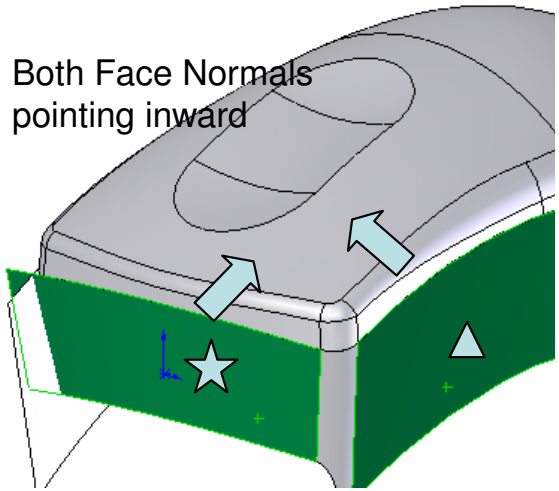




To add a fillet between two surfaces:-

- Click "**Fillet**" icon
- Select "Face Fillet" as Type
- Enter **4.5mm** as Radius
- Select the face ☆
- Click the entry box "Face Set 2" (pink color)
- Select the face △
- Press Enter key to complete



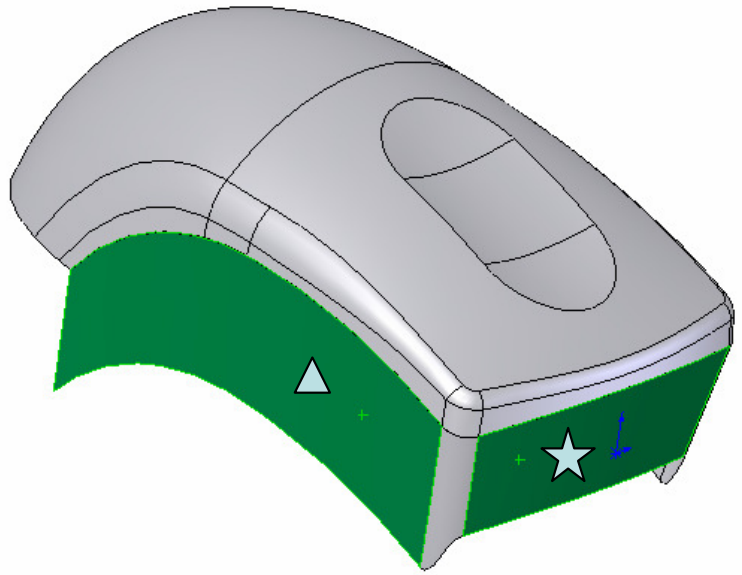


Written by Dickson Sham

Tutorial 2B

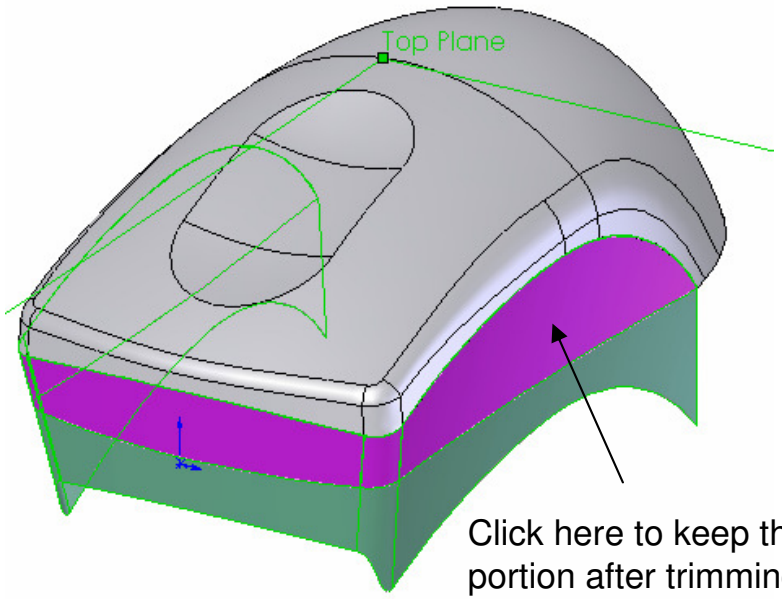
To add another face fillet :-

- Click "**Fillet**" icon again
- Select "Face Fillet" as Type
- Enter **4.5mm** as Radius
- Select the face ☆
- Click the entry box "Face Set 2" (pink color)
- Select the face △
- Press Enter key to complete



To Trim the surface:-

- Select "**Insert** / **Surface** / **Trim**" on the menu bar
- Select "Standard" as Trim Type
- Select "Top Plane" as Trim Tool
- Click on the front portion of "Fillet4"
- Press Enter key to complete



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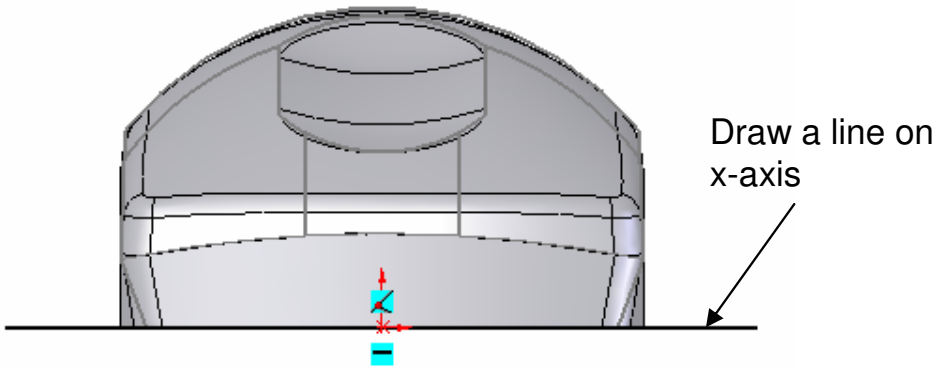
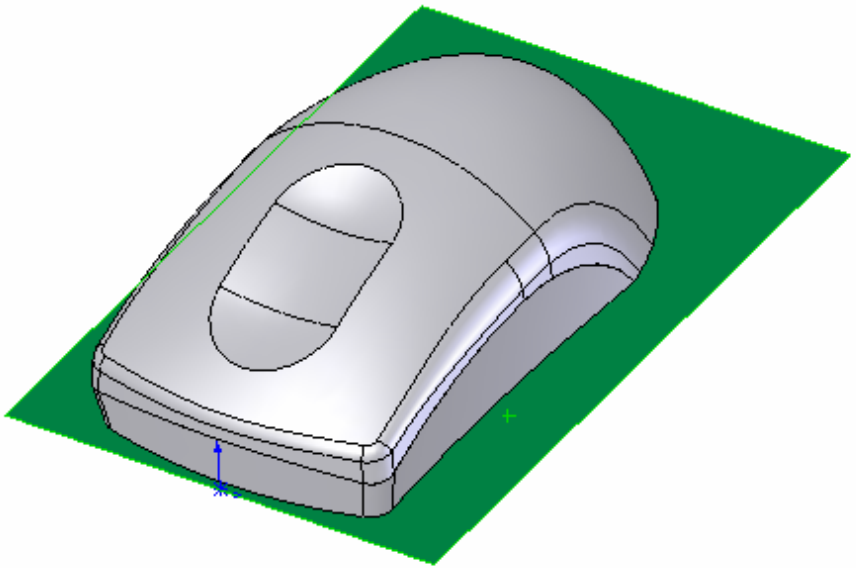
Tutorial 2B

To create a bottom surface:-

- Click "**Sketch**" icon
- Select "Front Plane"
- Draw a straight line on x-axis, which is long enough to go across the whole model
- Click "Exit Sketch" icon to exit



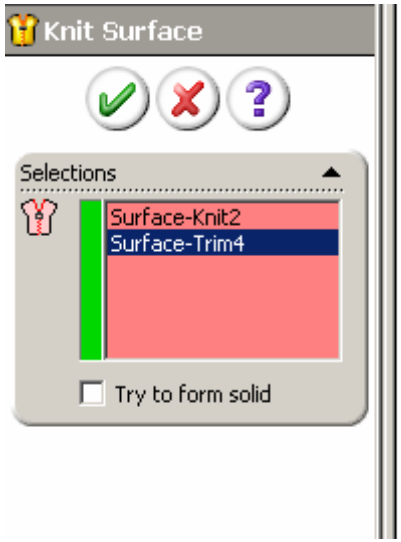
- Select "**Insert** / **Surface** / **Extrude**" on the menu bar
- Select the sketch as profile
- (Direction of extrusion is the normal of the sketch plane, by default)
- Click "Reverse Direction" icon
- Enter **120mm** as D1
- Press Enter key to complete



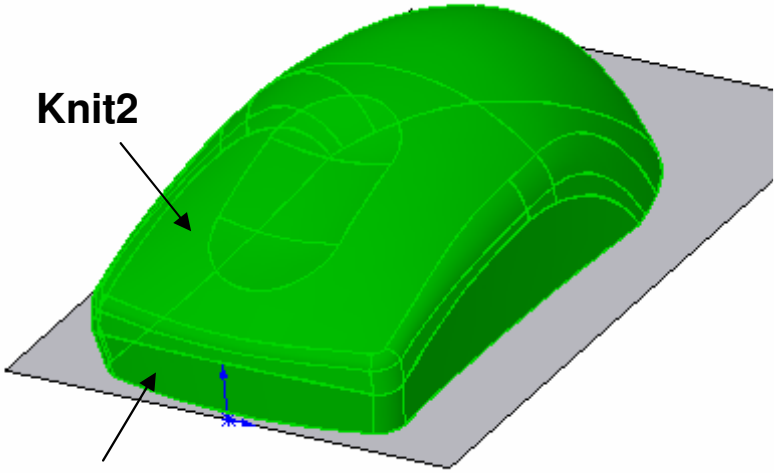
Tutorial 2B

To Trim the extruded surface by the upper surfaces:-

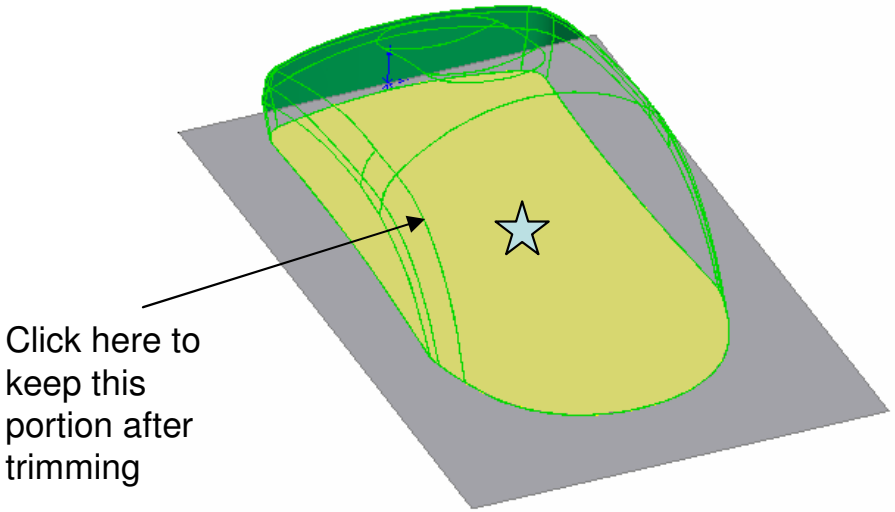
- Select "**Insert/ Surface/ Knit**" on the menu bar
- Select surfaces "Knit2" & "Trim4"
- Press Enter key to complete



Knit2 + Trim4 => Knit3



- Select "**Insert** / **Surface** / **Trim**" on the menu bar
- Select "Standard" as Trim Type
- Select "Surface-Knit3" as Trim Tool
- Click on the inner portion of the bottom extruded surface ☆
- Press Enter key to complete



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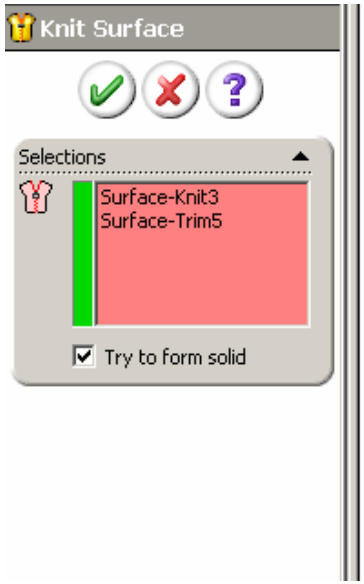
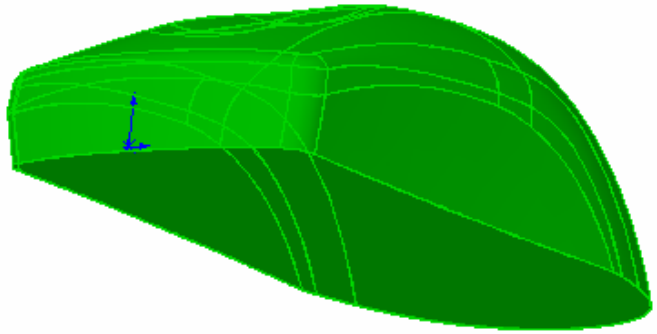
Tutorial 2B

To Join all surfaces and form into a Solid

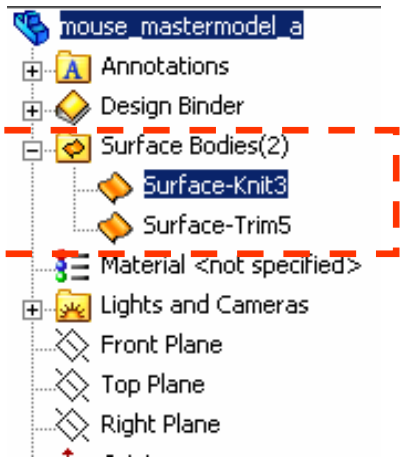
- Select "**Insert/ Surface/ Knit**" on the menu bar
- Select all Surfaces
- Select "**Try to form solid**"
- Press Enter key to complete





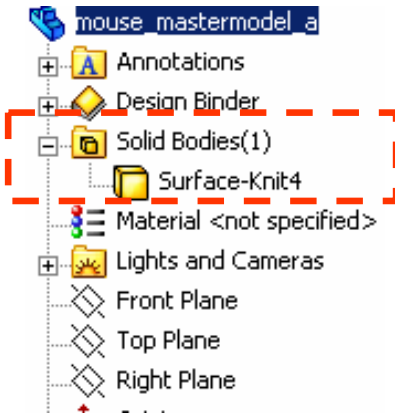
File/Save <u>Mouse_mastermodel_a.sldprt</u>

2 Surfaces existed; no solid bodies on the tree



Before

Join a group of surfaces without open ends



After

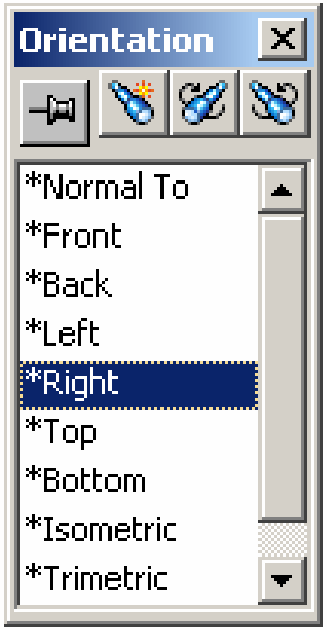
A solid is created; all surface bodies are removed from the tree

END of Tutorial 2B

Written by Dickson Sham

Tutorial 2C

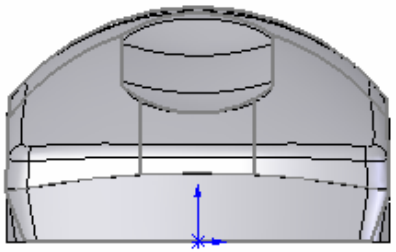
We've built the upper skin & the lower skin of the mouse. Now, we are going to create parting surfaces, transform the skin surface into a solid, and then split it into separate components.

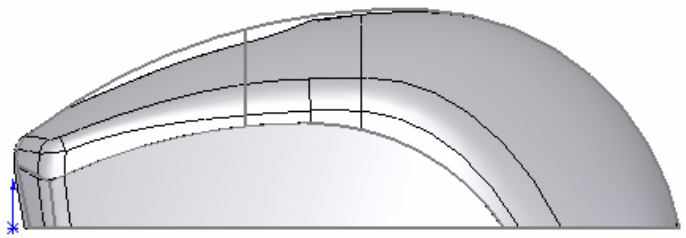


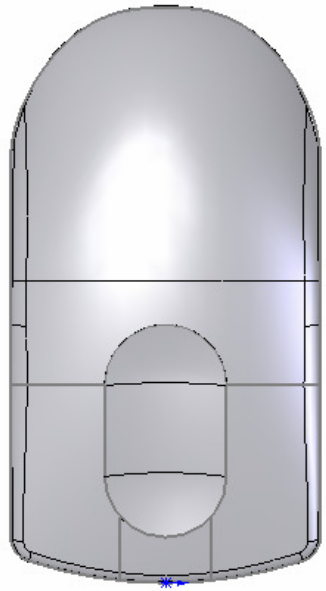
- *(Reopen the file* "Mouse_mastermodel_a.sldprt") ;
- Press "Space" key to activate the orientation selection
- Click **"Front View"** icon to check the front view;
- Click **"Top View"** icon to check the top view;
- Click **"Right View"** icon to check the right view;

  (Remark: the surface should match the three reference views)

  (Remark: In the real case, the dimensions & locations of all control curves of the surfaces should be referred to these imported reference views)



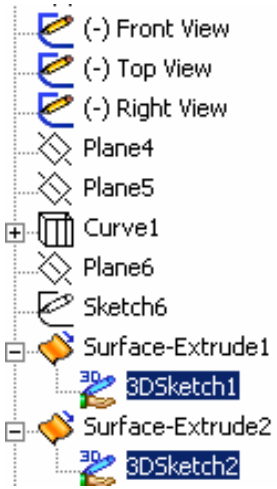


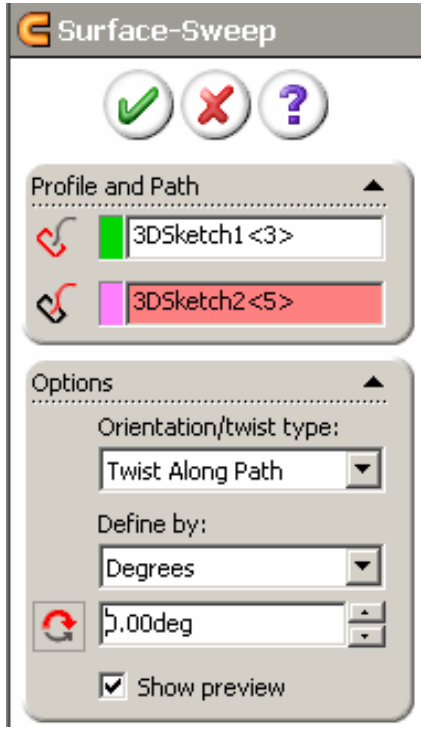


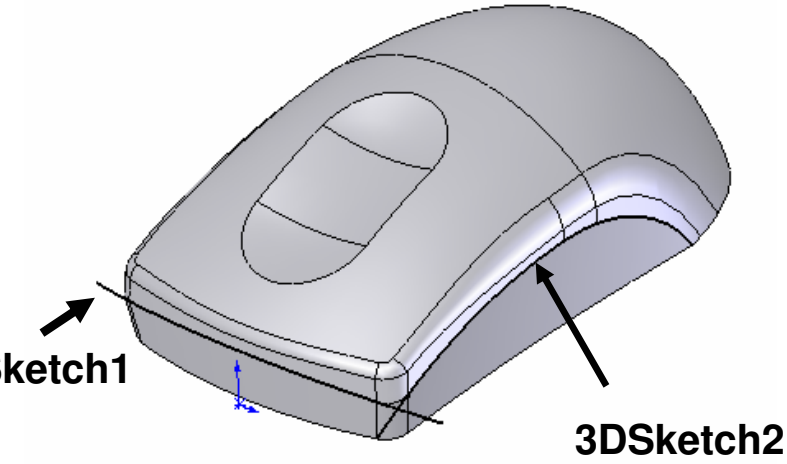
Tutorial 2C

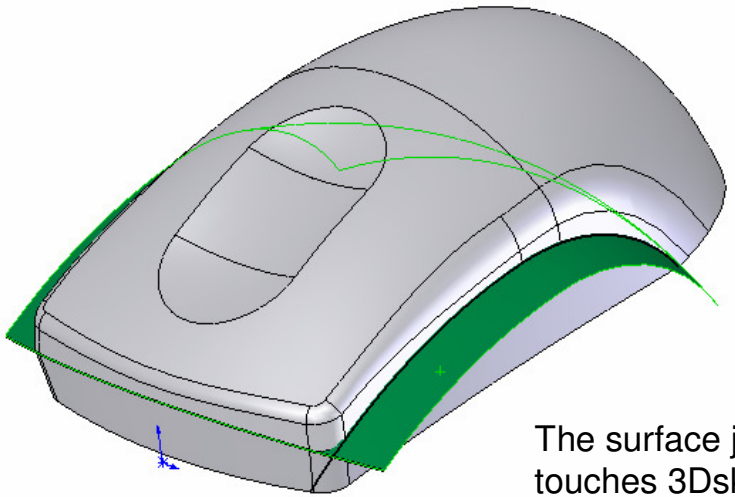
To create a swept surface:-

- Show “3DSketch1” & “3DSketch2”
- Select “**Insert/ Surface/ Sweep**” on the menu bar
- Select “3DSketch1” as profile
- Select “3DSketch2” as path
- Select “Twist along path” as orientation
- (Leave 0 deg as angle as default)
- Press Enter key to complete
- (Remark: A gap is allowed between Profile and Path, but the sweep surface must start from Profile)



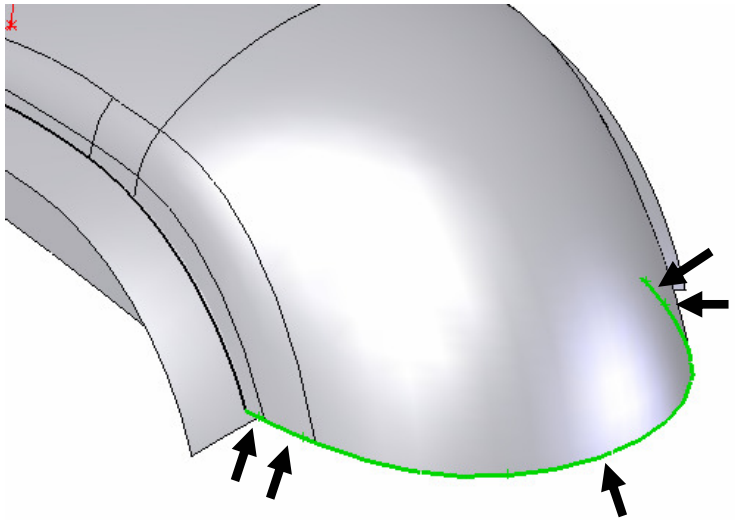






The surface just touches 3Dsketch1, but 3Dsketch2

Tutorial 2C

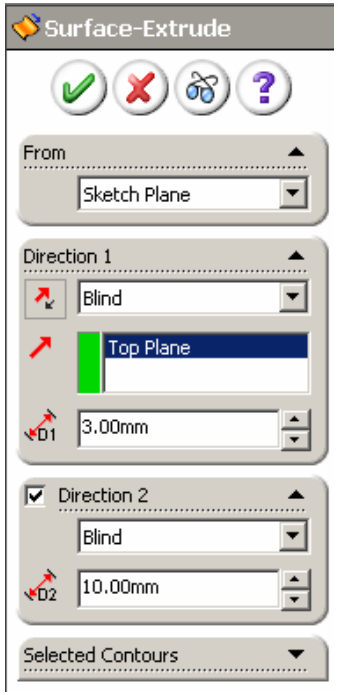


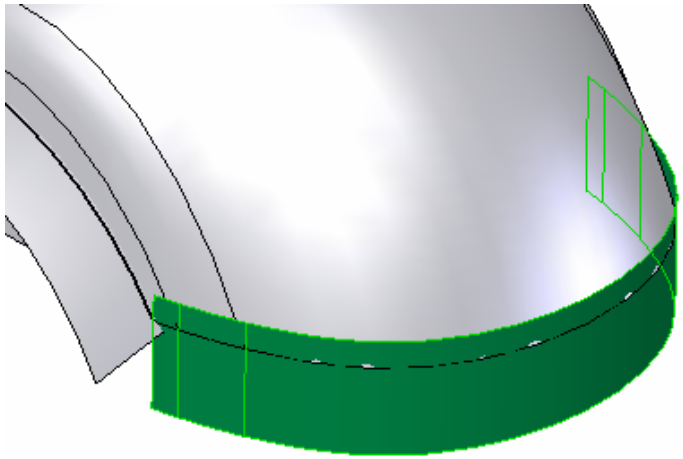
To extract a curve from the surface:-

- Click **" 3D Sketch"** icon
- Multi-select all tangent edges (5 edges)
- Click "**Convert**" icon
- Click " 3D Sketch" icon again to complete

To create an extruded surface:-

- Select "**Insert/ Surface/ Extrude**" on the menu bar
- Select the new 3DSketch ("3DSketch5")
- Select "Top Plane" as Direction
- Enter **3.0mm** as D1
- Activate Direction2
- Enter **10.0mm** as D2
- Press Enter key to complete

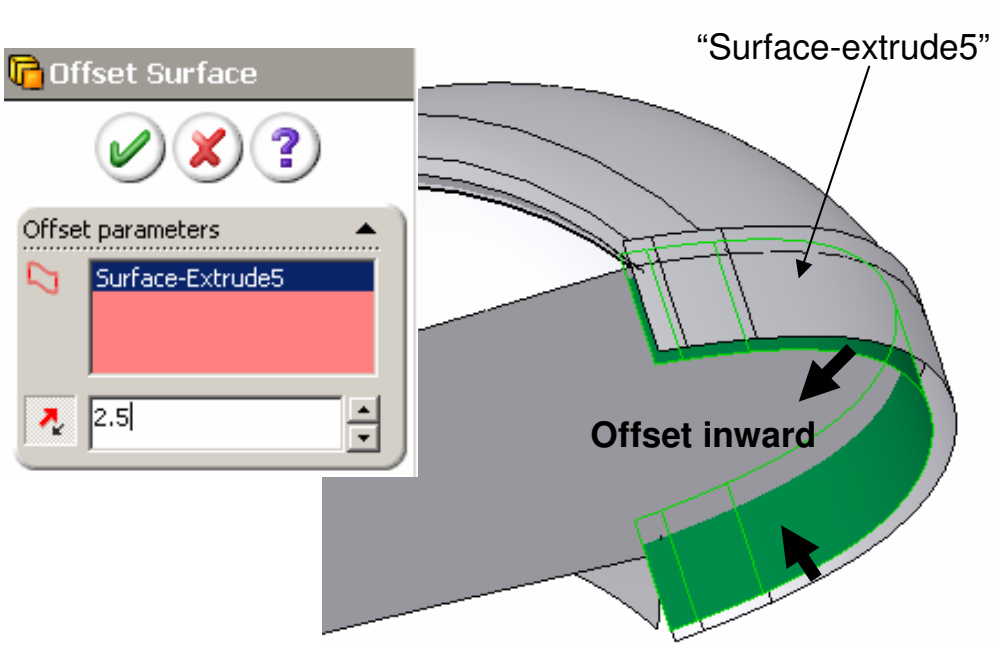




Tutorial 2C

To Create an Offset surface:-

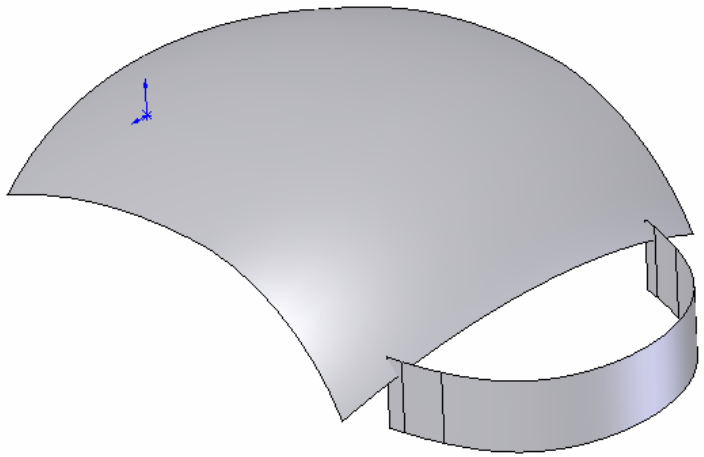
- Select "**Insert/ Surface/ Offset**" on the menu bar
- Select "Surface-Extrude5" (Click it on the tree to select all faces)
- Enter **2.5mm** as Offset value
- Click " Reverse Direction" to make the offset inward
- Press Enter key to complete



Hide "Surface-Extrude5"

Hide "3DSketch1" & "3DSketch2"

Hide the solid body

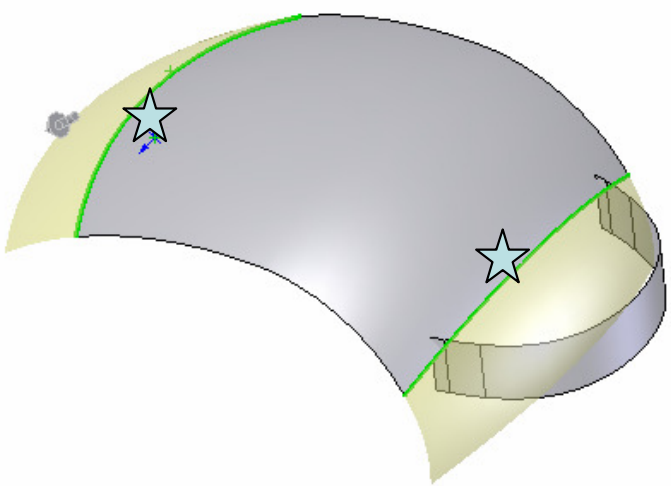


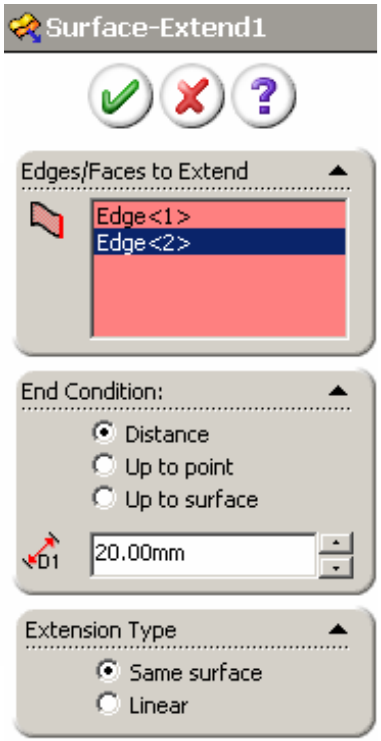
Result

Tutorial 2C

To extend a surface:-

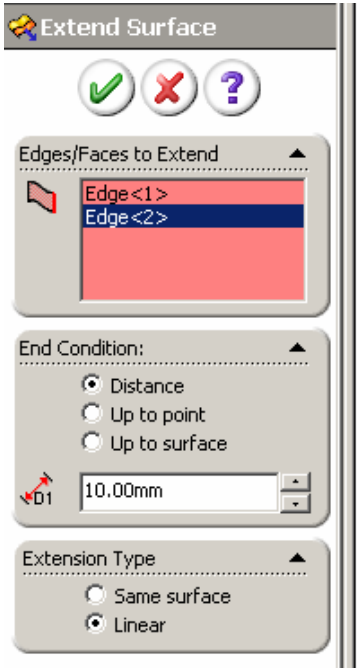
- Select "**Insert** / **Surface**/ **Extend**" on the menu bar
- Click on the edges ☆ of "Surface-Sweep1"
- Enter **20mm** as D1 (Extension Length)
- Select "**Same Surface**" as extension type (the surface will be extended according to its curvature trend)
- Press Enter key to complete

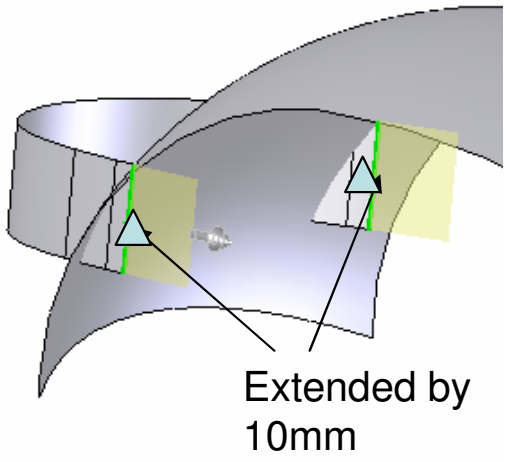




Similarly, extend the both edges of "Surface-Offset1" by 10mm:-

- Select "**Insert** / **Surface**/ **Extend**" on the menu bar
- Click on the edges △ of "Surface-Offset1"
- Enter **10mm** as D1
- Select "**Linear**" as extension type
- Press Enter key to complete

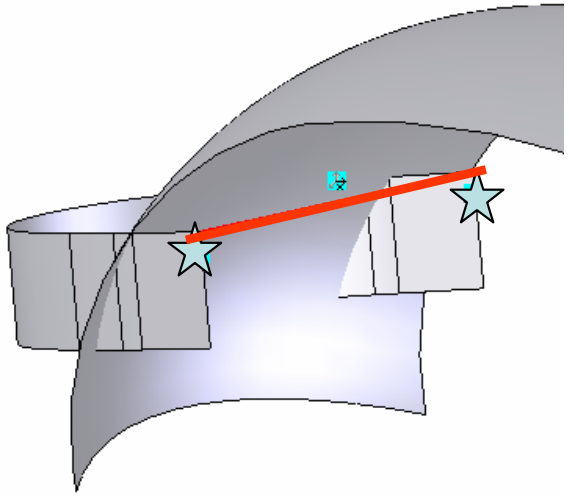




Tutorial 2C

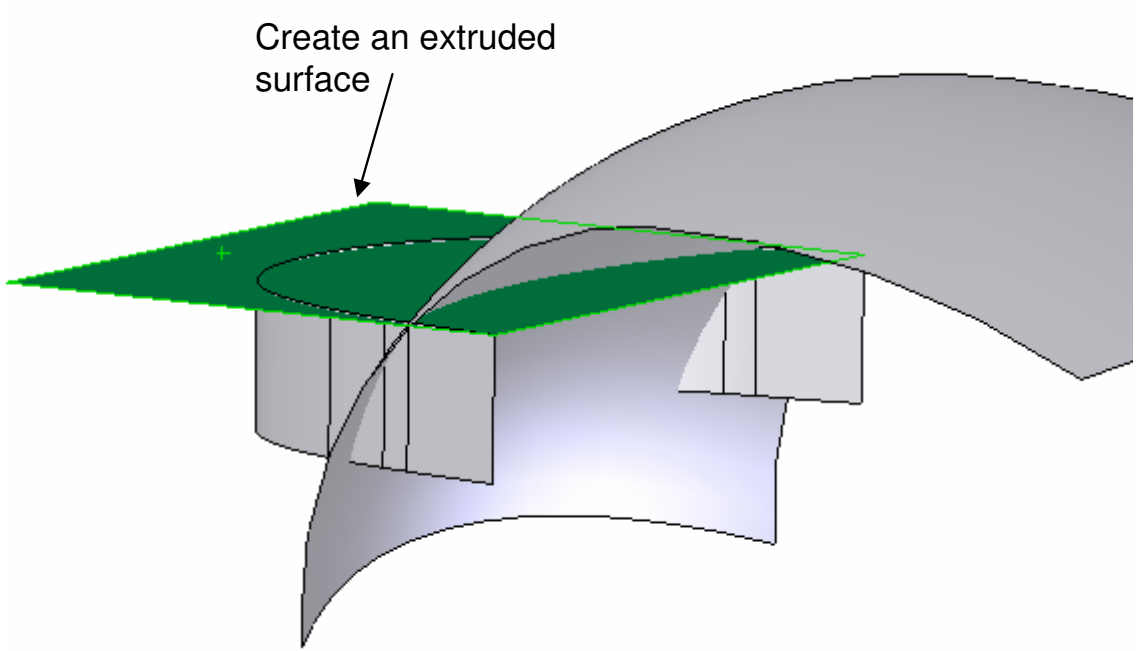
To create a line:-

- Click **" 3D Sketch"** icon
- Click "Line" icon
- Select the two end points ☆ of the surface
- Click " 3D Sketch" icon to complete



To create an extruded surface:-

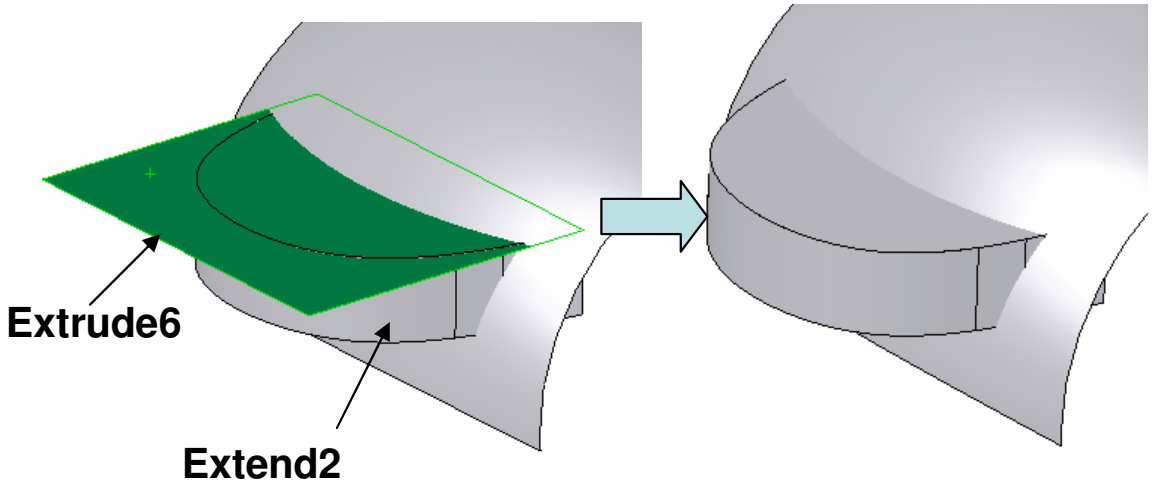
- Select "**Insert** / **Surface**/ **Extrude**" on the menu bar
- Select the new 3D Sketch as profile
- Select "Front Plane" as direction
- Click "Reverse Direction" icon
- Enter **50mm** as D1
- Press Enter key to complete



Tutorial 2C

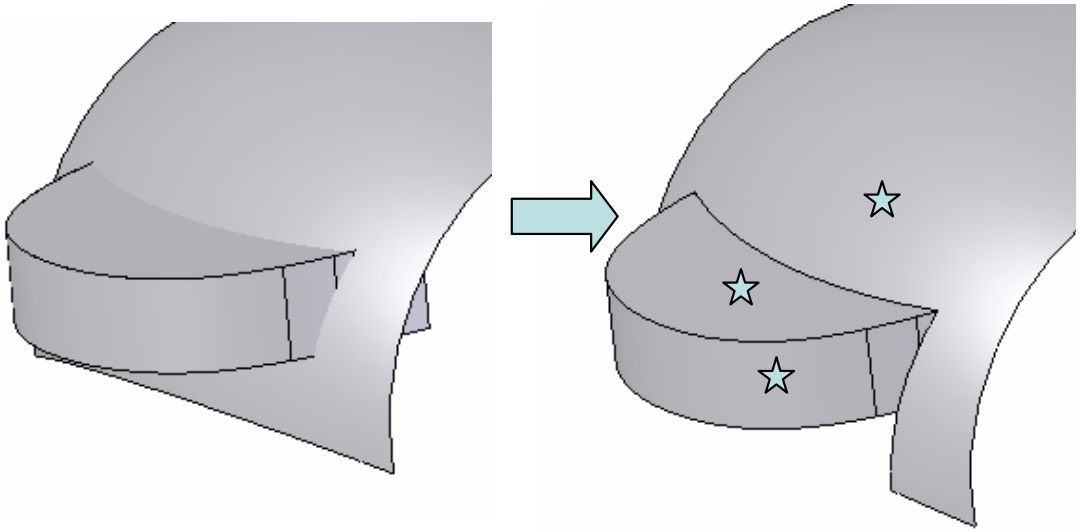
To Trim & Join surfaces:-

- Select "**Insert/Surface/Trim**" on the menu bar
- Select **"Standard"** as Trim type
- Select "Extend2" as Trim Tool
- Click on "Extrude6"
- Press Enter key to complete



Similarly

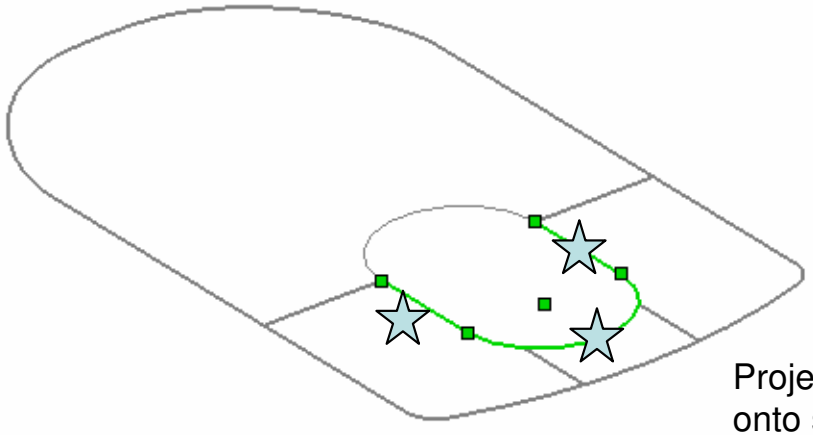
- Select "**Insert/Surface/Trim**" on the menu bar
- Select **"Mutual"** as Trim type
- Select all surfaces
- Click the entry box "Keep Selections"
- Click the three faces ☆
- Press Enter key to complete



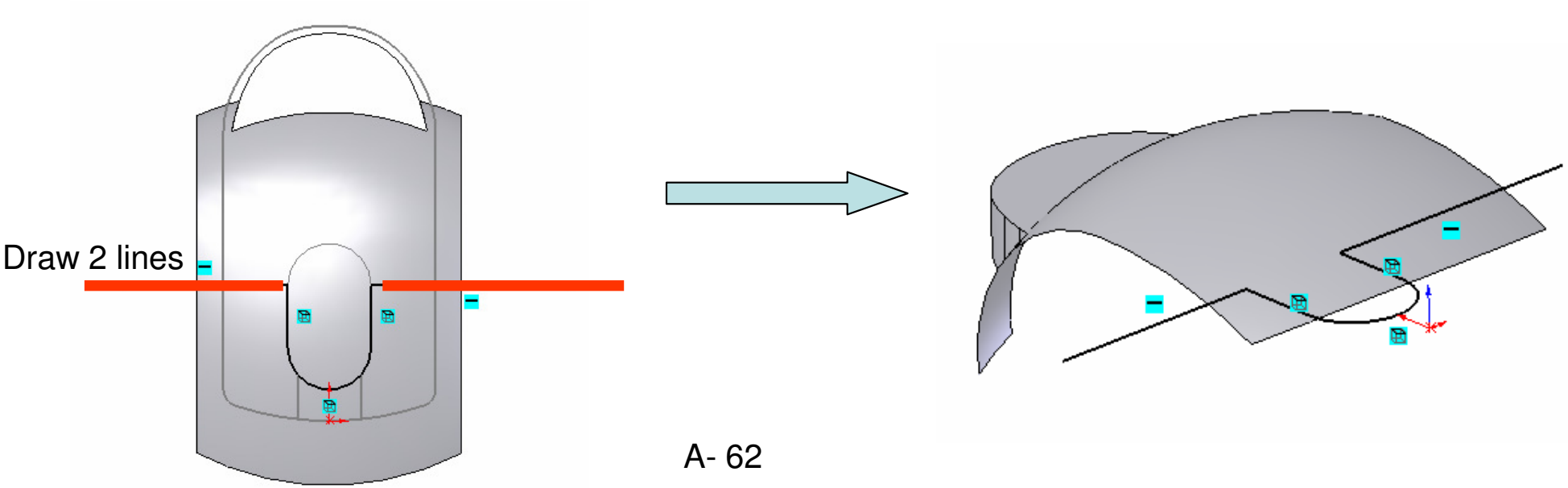
Tutorial 2C

To create 2nd parting surface:-

- Click "**Sketch**" icon
- Select "Top Plane"
- Multi-Select the three edges ☆ of the sketch "Top View"
- Click **"Convert"** icon
- Draw 2 horizontal lines to go across the model
- Click "**Exit Sketch**" icon to complete

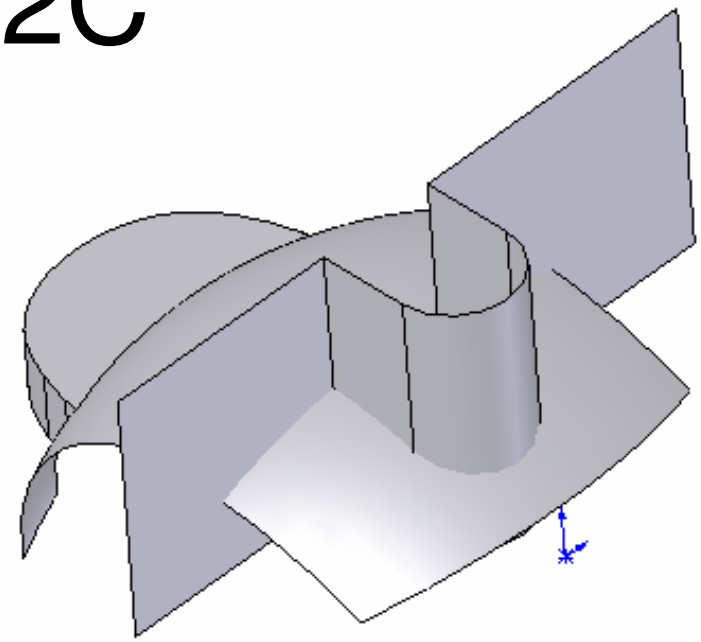


Project 3 curves onto sketch



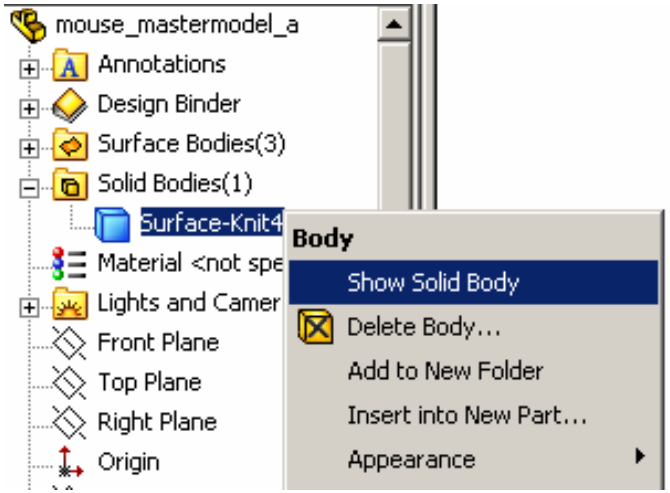
Tutorial 2C

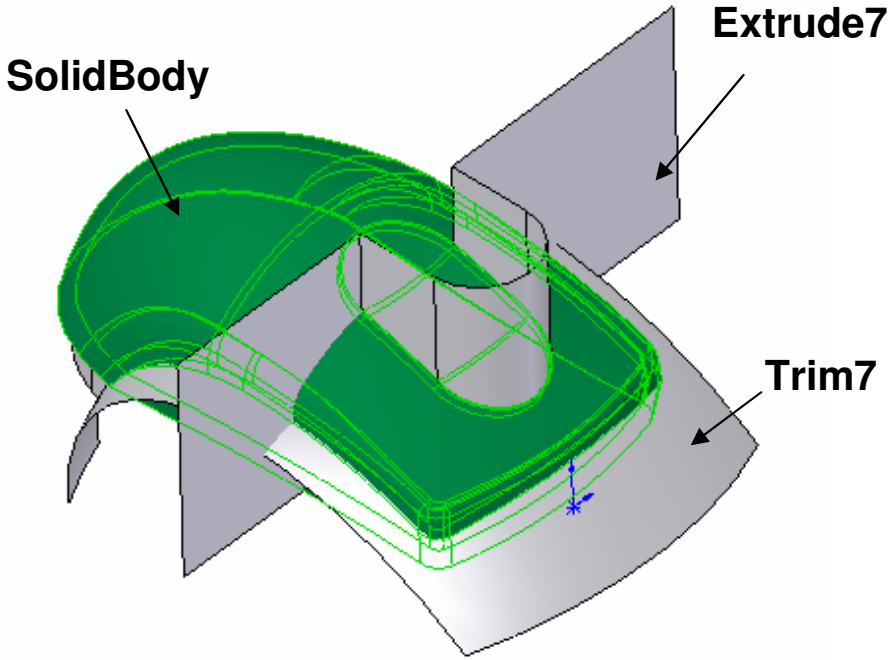
- Select "**Insert/ Surface/Extrude**" on the menu bar
- Select the new sketch as profile
- Enter **50mm** as D1
- Press Enter key to complete



SHOW the solid body

(Do it yourself)

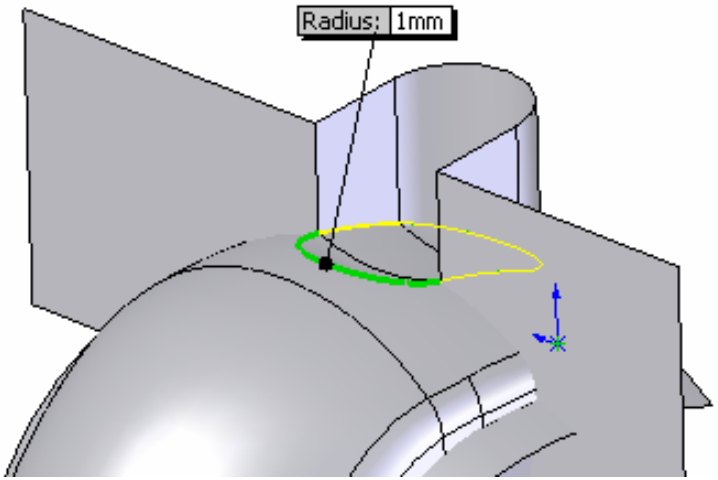




Tutorial 2C

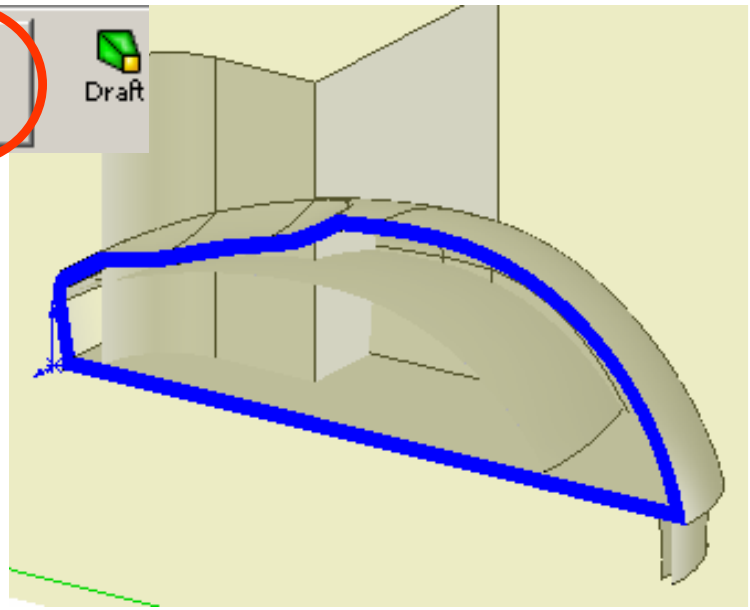
To add a fillet on the solid:-

- Click “Features” icon
- Click **“Fillet”** icon
- Select the edge
- Enter **1mm** as radius
- Select “Tangent Propagation”
- Press Enter key to complete



To hollow the solid:-

- Click “**Shell**” icon
- Enter **2.5mm** as D1
- (Do not pick any face of the solid)
- Press Enter key to complete



File/Save

(The MASTER model is finished, and we

are going to split it into separate parts)

Tutorial 2C

To create the upper body:-

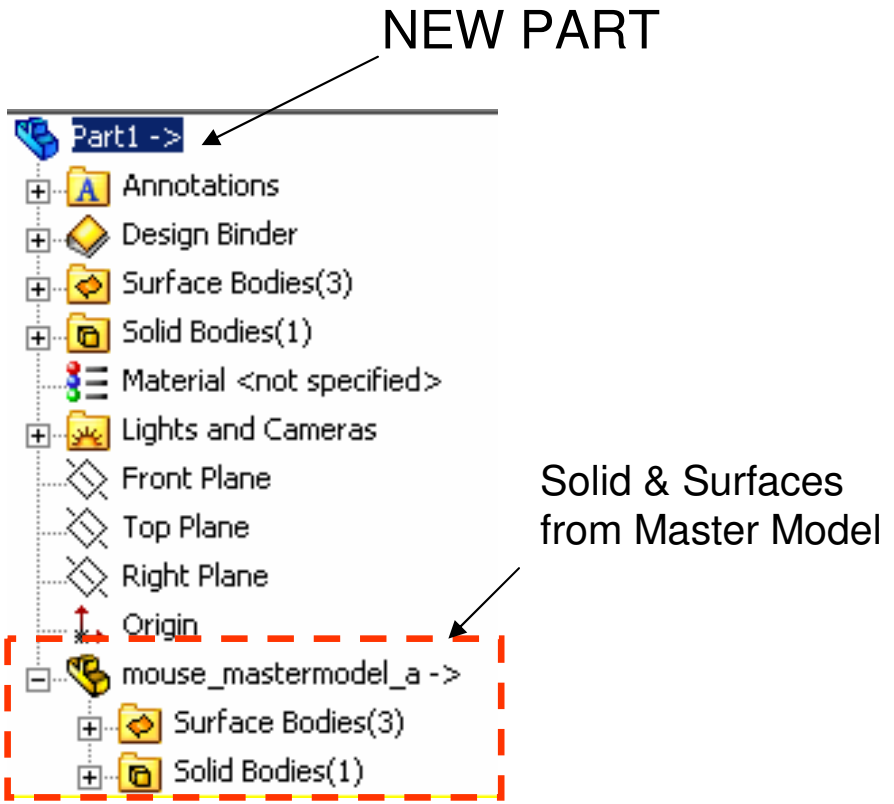
- Select **File/New**
- Select Part as type
- Click ok to complete

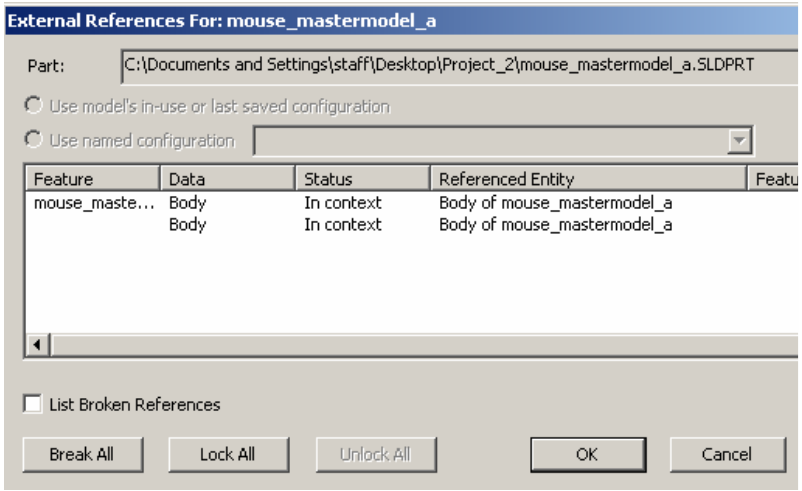
- **Insert/Part…**
- Select "Mouse_Mastermodel_a.sldprt"
- Click Open
- **Select "Surface" under the list of "Transfer"**
- Click ok (or Press Enter key to complete)

(Remark: <u>All Surfaces</u> of MasterModel will be copied and pasted onto this new part!)

(Remark: On the tree of the new part, we have three surfaces and a solid copied from the Master. If the master is changed, the four elements will be updated automatically!)

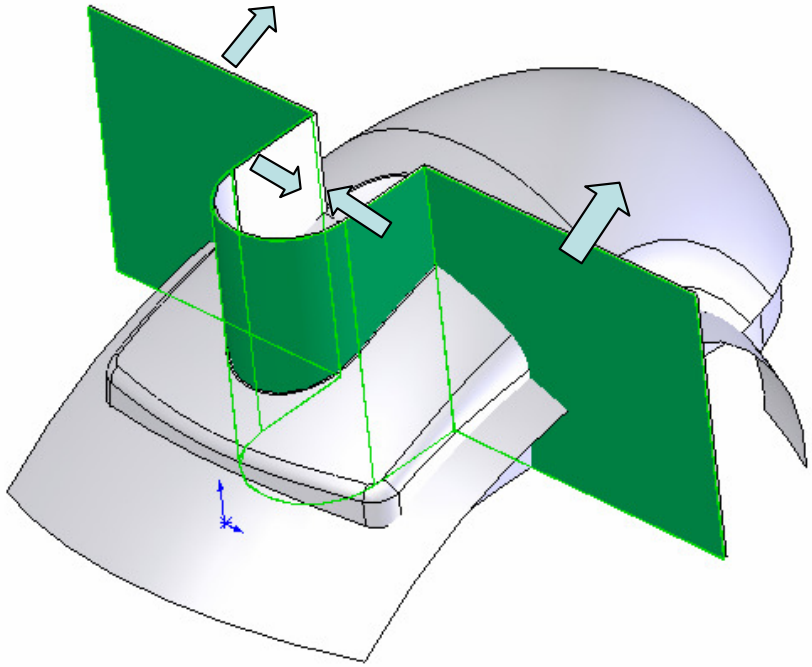
(Remark: If you right-click on "Mastermodel" and select "List External Refs…", you can see that the bodies have been linked to an external file "Mouse_Mastermodel_a.sldprt")



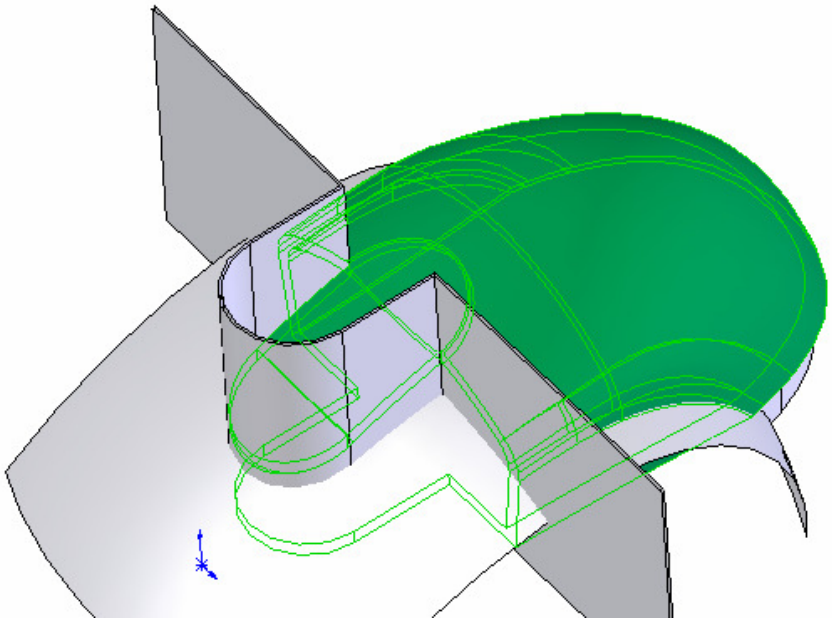


Tutorial 2C

- Select “**Insert** / **Surface**/ **Offset**” on the menu bar
- Select “Extrude7”
- Enter **0.5mm** as value
- Click “Reverse Direction” icon to make the offset inward
- Press Enter key to complete

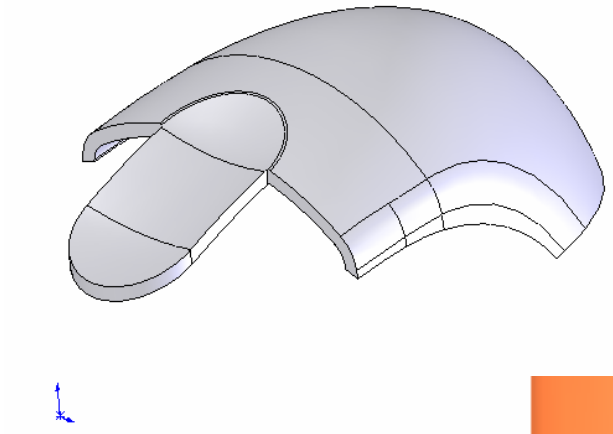


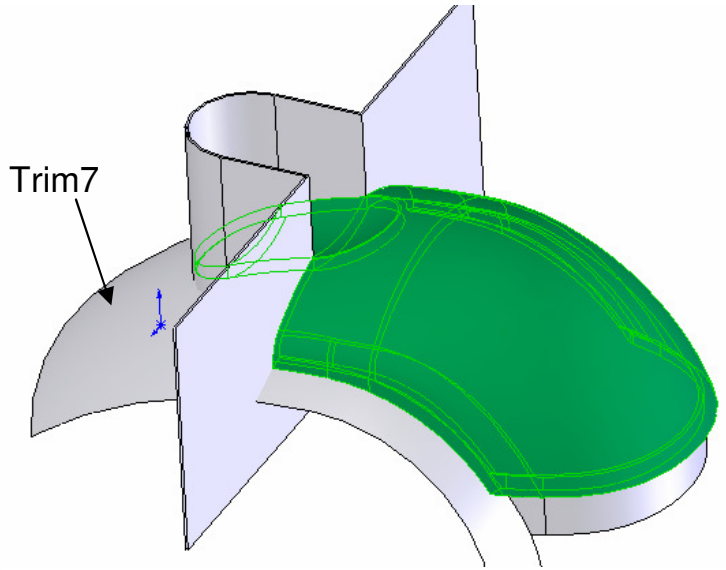
- Select “**Insert** / **Cut** / **With Surface**” on the menu bar
- Select “ Offset1”
- Click “Reverse Direction” icon
- Press Enter key to complete



Tutorial 2C

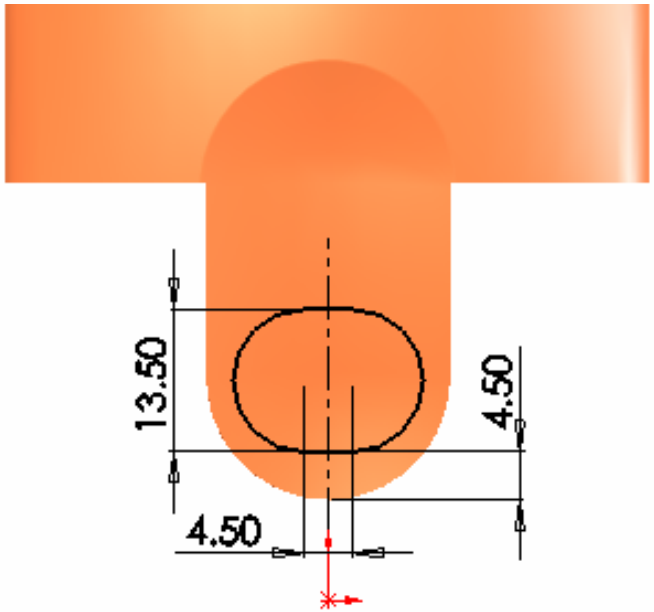
- Select “**Insert** / **Cut** / **With Surface**” on the menu bar again
- Select “Trim7”
- Click “Reverse Direction” icon
- Press Enter key to complete





Hide All Surfaces

- Click “**Sketch**” icon
- Select Top Plane
- Draw a profile as shown
- Click “Exit Sketch” to complete



Tutorial 2C

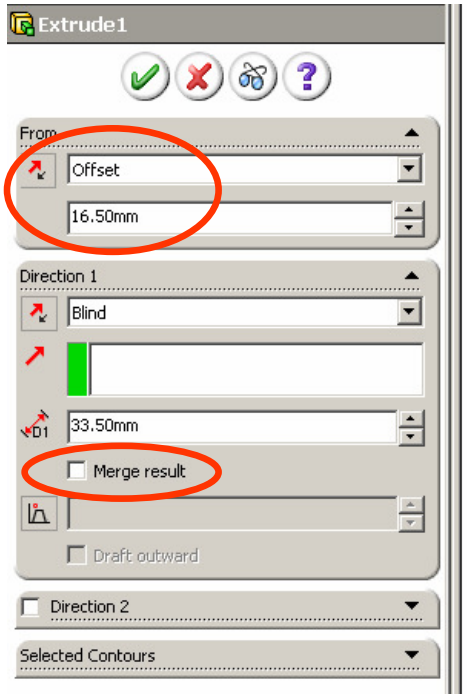
- Click “Features” icon
- Click “**Extruded Boss/Base**” icon
- Select the new sketch

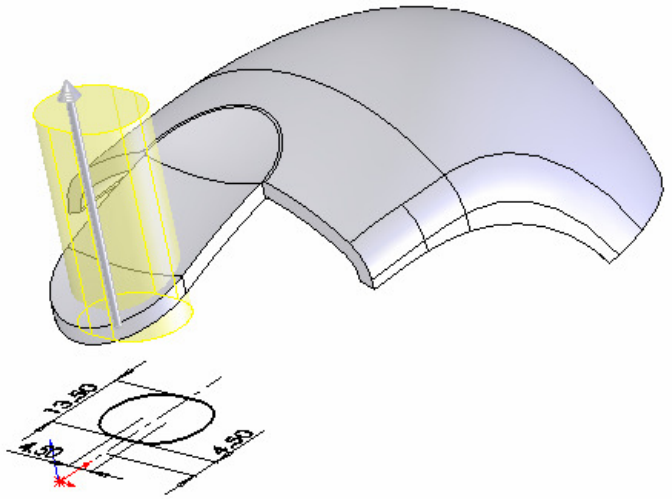
- Select “Offset” in the box “From”
- Enter **16.5mm**

- Enter **33.5mm** as D1 (Direction1)

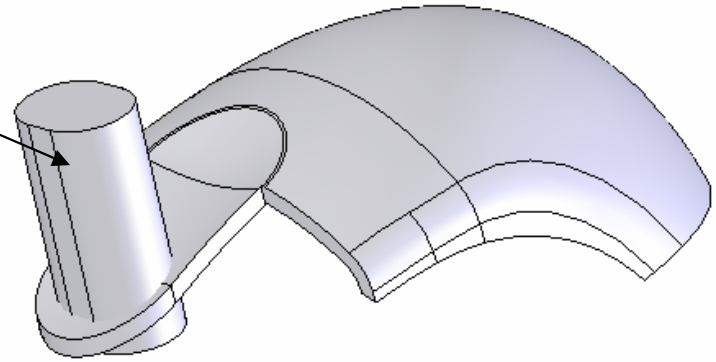
- **Deselect “Merge Result”**

- Press Enter key to complete



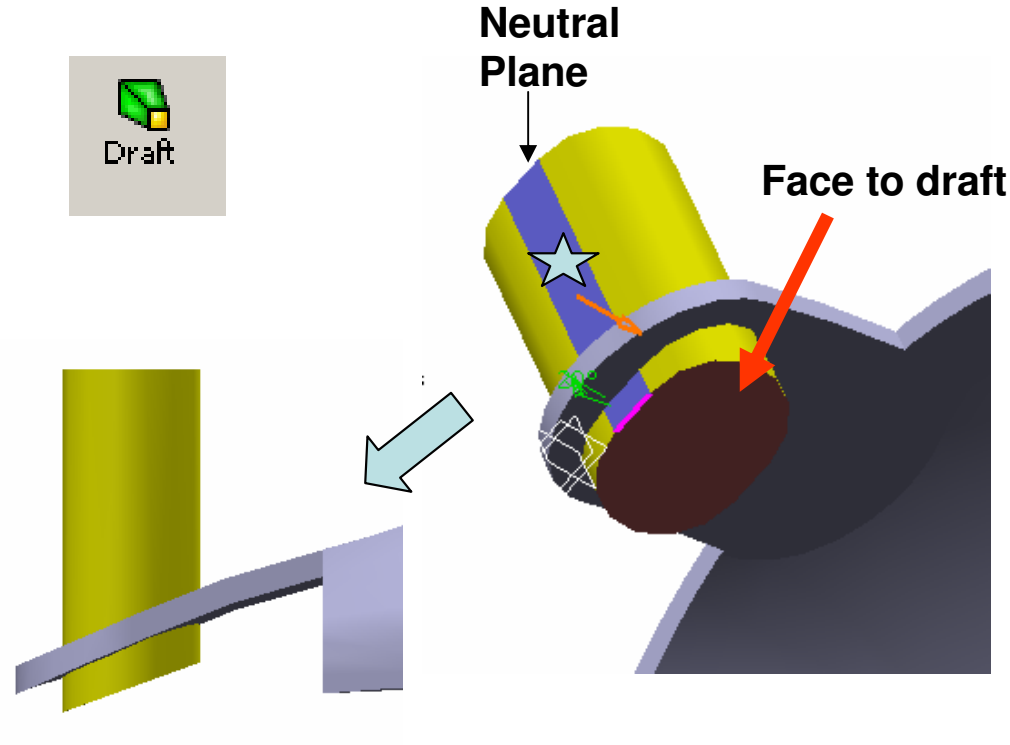


The Extruded solid body is not added into the main body

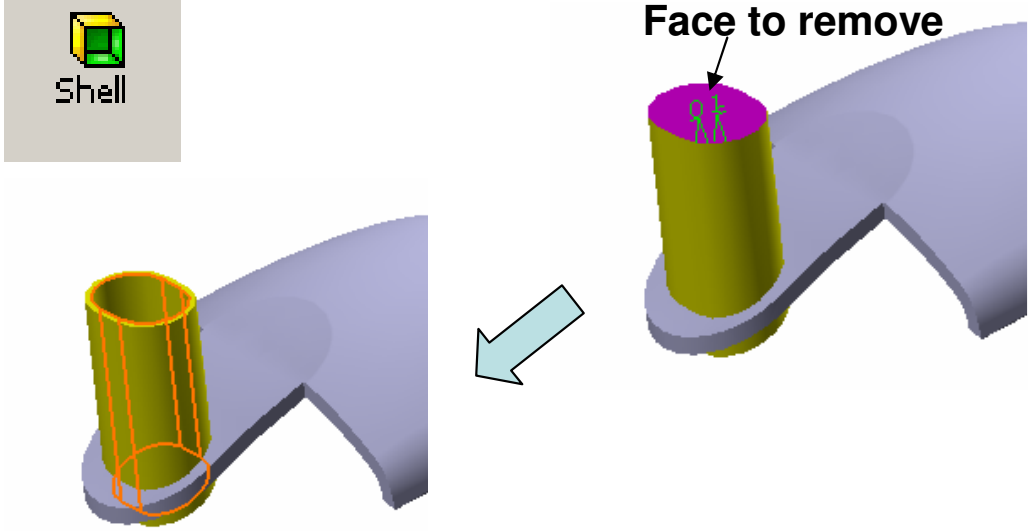


Tutorial 2C

- Click **"Draft"** icon
- Select "Neutral Plane" as Type of Draft
- Enter **20deg** as Angle
- Select the side planar face as **Neutral Plane** ☆
- Click "Reverse Direction " to make the arrow point backwards
- Select the bottom face as Face to Draft
- Press Enter key to complete

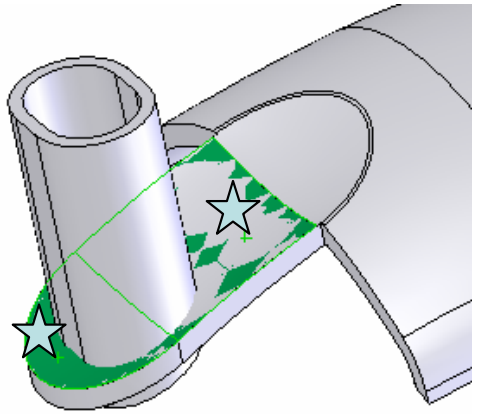


- Click **" Shell"** icon
- Enter **1.5mm** as D1 (wall thickness)
- Select the top face as **face to remove**
- Press Enter to complete

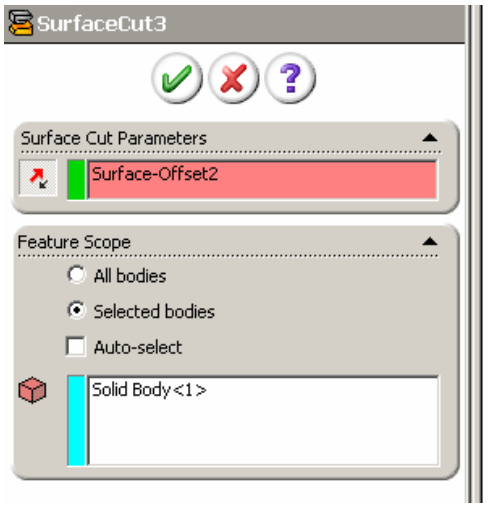


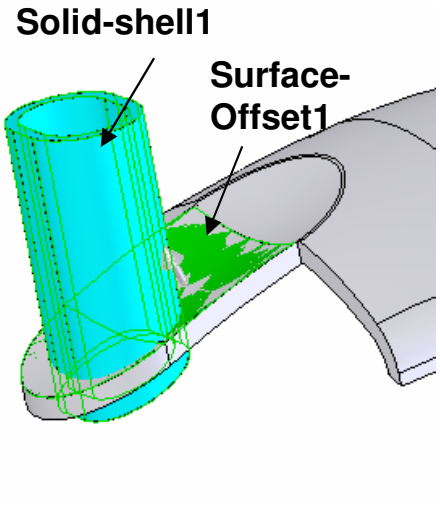
Tutorial 2C

- Select “**Insert** / **Surface** / **Offset**” on the menu bar
- Select the two faces of the main solid ☆
- Enter 0mm as offset direction
- Press Enter key to complete



- Select “ **Insert** / **Cut** / **With Surface**” on the menu bar
- Click “Reverse Direction” to make the arrow point upward
- Deselect “Auto-select”
- Select the solid “Shell1”
- Press Enter key to complete

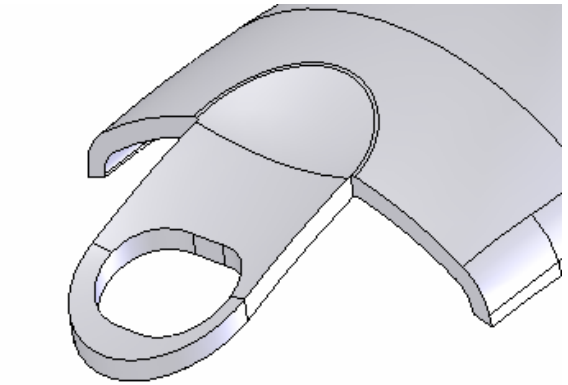




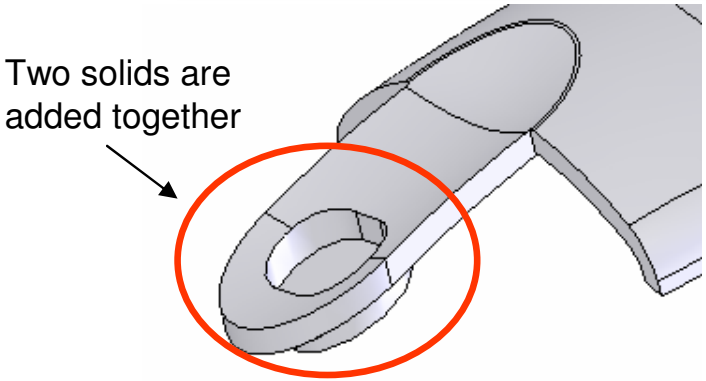
- Hide the cut solid & the offset surface

Tutorial 2C

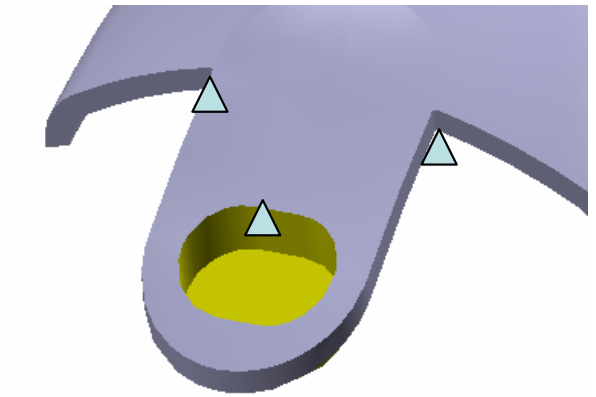
- Click **“ Extruded Cut”** icon;
- Select “Sketch1”
- Click “Reverse Direction” to make the arrow point upward
- Select “Through all” as D1
- Press Enter Key to complete



- Select “**Insert** / **Features**/ **Combine**” on the menu bar
- Select “Add” as Operation type
- Select the two solid bodies (one visible & one hidden)
- Press Enter key to complete



- Click “**Fillet**” icon
- Select the three sharp edges △
- Enter 1mm as radius
- Press Enter key to complete

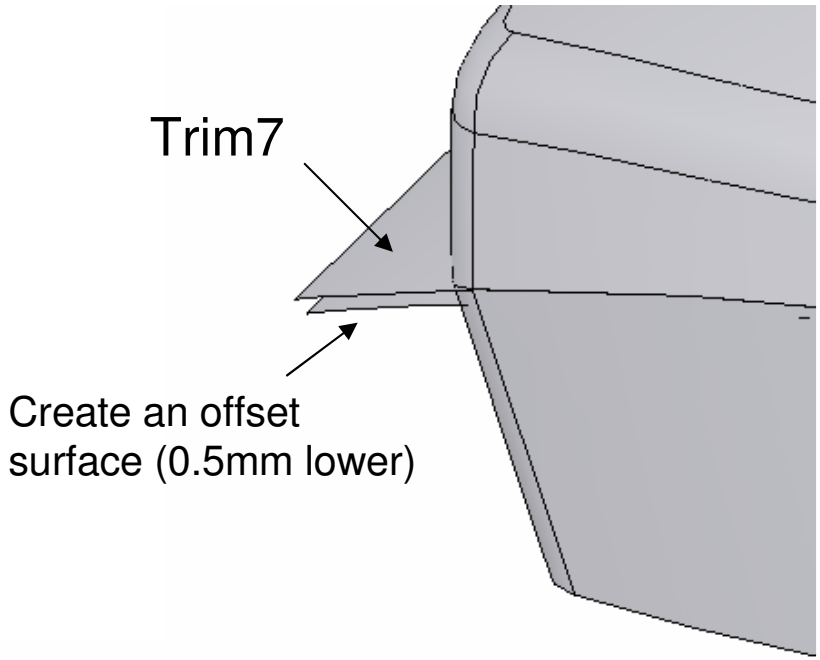


File/Save as “Upper_body_a.sldprt”
then File/Close

Tutorial 2C

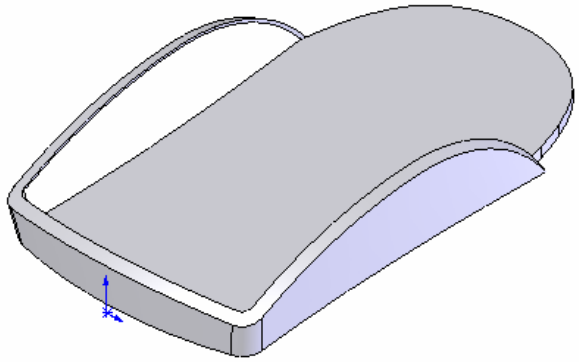
To create the lower body:-

- Select **File/New**
- Select Part as type
- Click ok to complete.
- **Insert/Part…**
- Select “Mouse_Mastermodel_a.sldprt”
- Click Open
- Select “Surface” under the list of “Transfer”
- Click ok (or Press Enter key to complete)



- Select “**insert/ Surface/ Offset**” on the menu bar
- Select “Trim7”
- Enter **0.5mm** as value (offset downwards)
- Press Enter key to complete

- Select “**insert/ Cut/ with surface**” on the menu bar
- Select the surface “Offset.1”
- Press Enter key to complete
- Hide all Surfaces

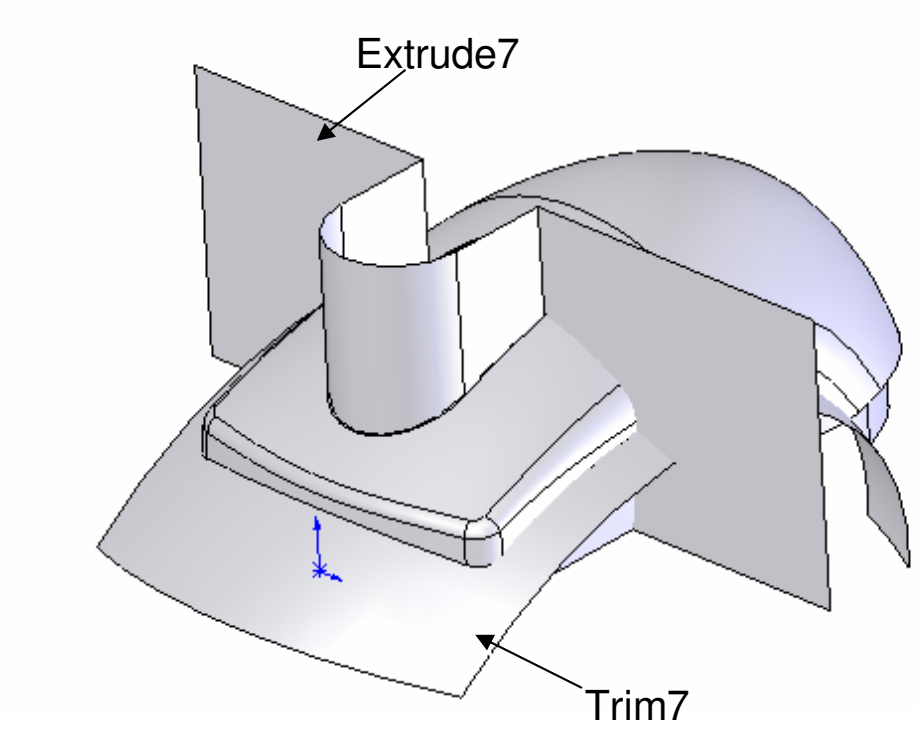


File/Save as “Lower_body_a.sldprt”
then File/Close

Tutorial 2C

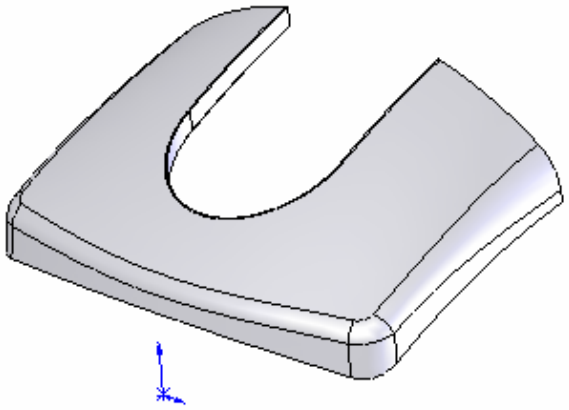
To create the Left Button:-

- Select **File/New**
- Select Part as type
- Click ok to complete
- **Insert/Part…**
- Select “Mouse_Mastermodel_a.sldprt”
- Click Open
- Select “Surface” under the list of “Transfer”
- Click ok (or Press Enter key to complete)



- Select “**insert/ Cut/ with surface**” on the menu bar
- Select the surface “Extrude7”
- (The arrow should point backwards)
- Press Enter key to complete

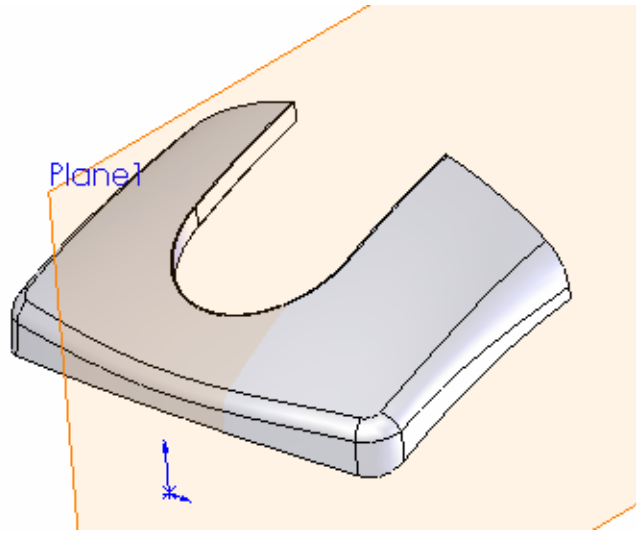
- Select “**insert/ Cut/ with surface**” again
- Select the surface “Trim7”
- (The arrow should point downwards)
- Press Enter key to complete



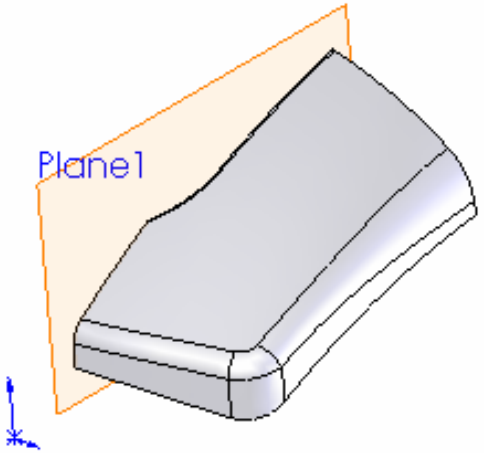
- Hide all Surfaces

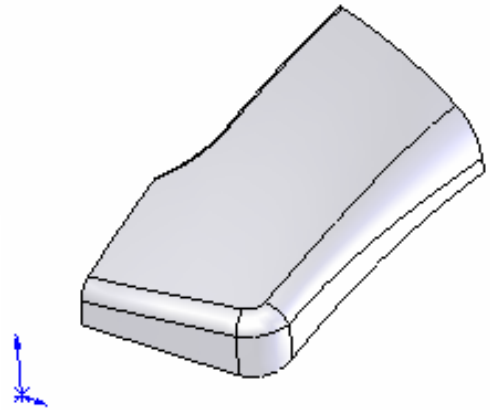
Tutorial 2C

- Click “Features” icon
- Click **“Reference”** icon, then “**Plane**”
- Select “Right Plane”
- Enter **9mm** as value
- (the arrow should point to +ve X-direction)
- Press Enter key to complete



- Select “**Insert/ Cut / with Surface**” on the menu bar
- Select “Plane1”
- Click on the arrow once if the direction is incorrect
- Press Enter Key to complete





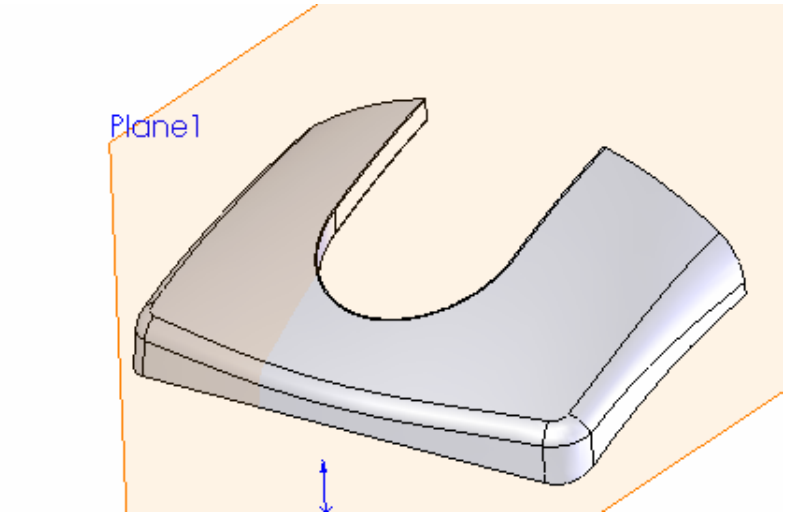
Hide “Plane1”

File/Save as “Left_Button_a.sldprt”
Then File/Close

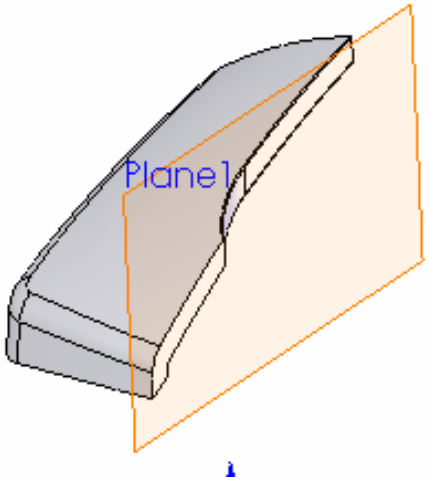
Tutorial 2C

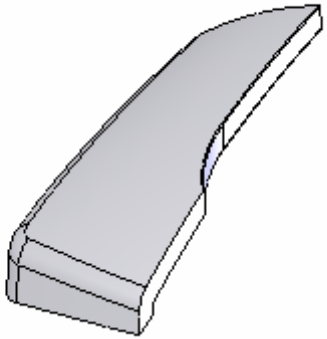
Similarly, create the Right button…

- Select **File/New**
- Select Part as type
- :
- (Ref to p.73, get the result as shown on the right)
- Click "**Plane**" icon
- Select "Right Plane"
- Enter **9mm** as value
- (the arrow should point to -ve X-direction)
- Press Enter key to complete



- Select "**Insert/ Cut / with Surface**" on the
- Select "Plane1"
- Click on the arrow once if the direction is incorrect
- Press Enter Key to complete





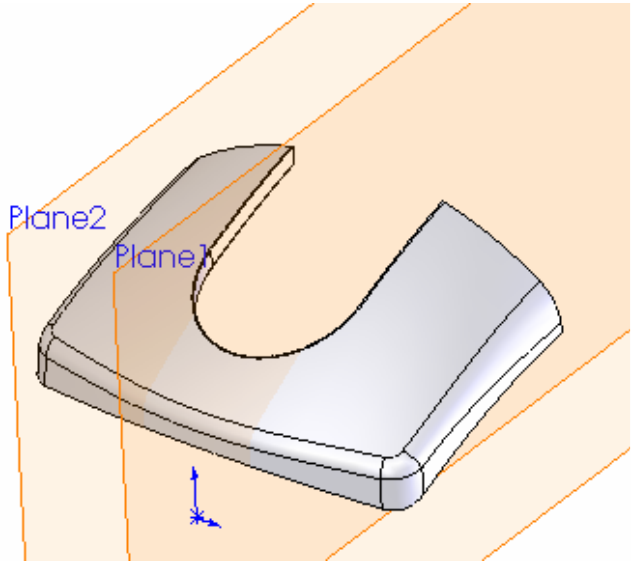
Hide "Plane1"

File/Save as "Right_Button_a.sldprt"
Then File/Close

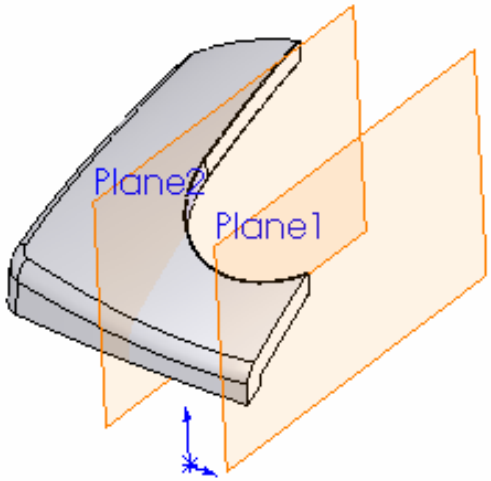
Tutorial 2C

Similarly, create the Middle button…

- Select **File/New**
- Select Part as type
- :
- (Ref to p.73, get the result as shown on the right)
- Click "**Plane**" icon
- Select Right Plane
- Enter **8.5mm** as value
- (the arrow should point to +ve X-direction)
- Press Enter key to complete

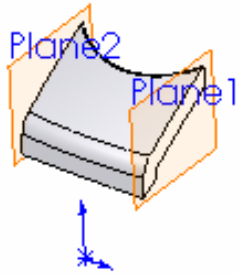


- Repeat the steps to create another offset plane (8.5mm, on the negative side)



- Cut the solid by these two offset planes

Hide "Plane1" & "Plane2"



File/Save as "Middle_Button_a.sldprt"
Then File/Close

Tutorial 2C

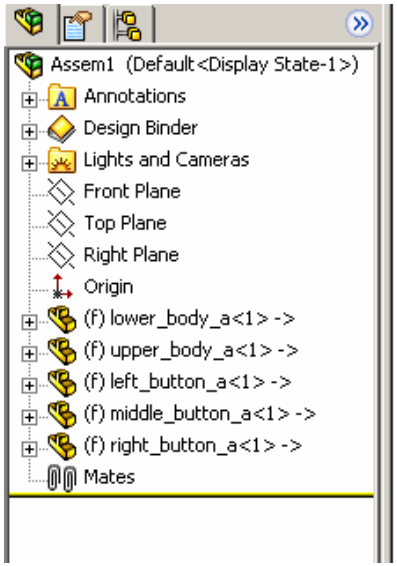
We have split the master into separate parts. We should always follow the rule that one file contains one part.

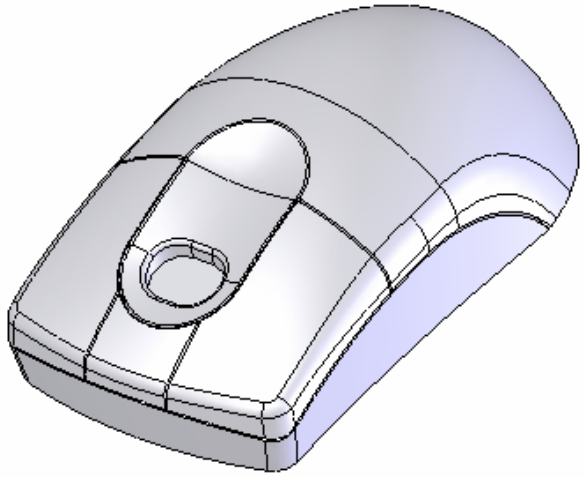
Now we are going to assemble the parts into a product

- Select **"File/New…"** on the menu
- Click "**Assembly**" and then OK

To insert existing parts into the assembly:-

- Click "Browse…"
- Select the file "Lower_body_a.sldprt"
- Click "Open"
- Press Enter key to complete

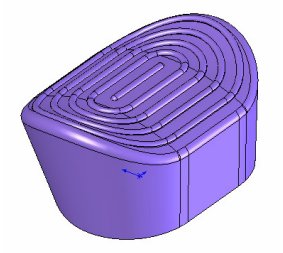


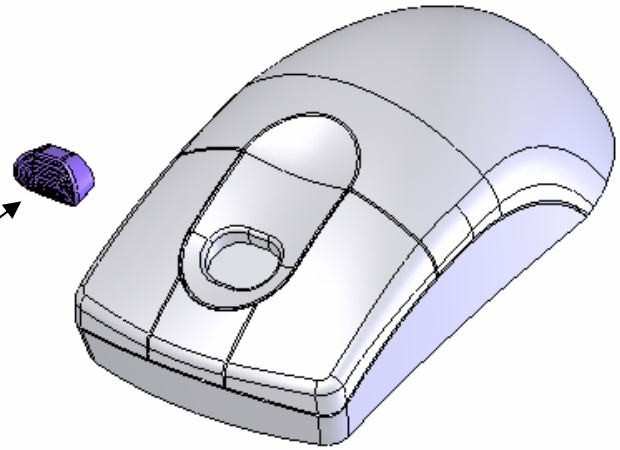


- Click "Insert Components" icon, repeat insert other parts:
  - Upper_body_a.sldprt
  - Left_button_a.sldprt
  - Middle_button_a.sldprt
  - Right_button_a.sldprt

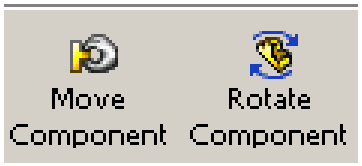
Tutorial 2C

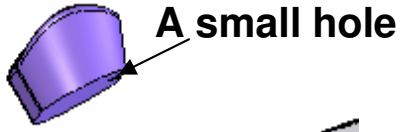
- Copy the part **(Your DVD drive) :\Model\ Scroll_button_a.CATpart** , and Paste it into your project folder.



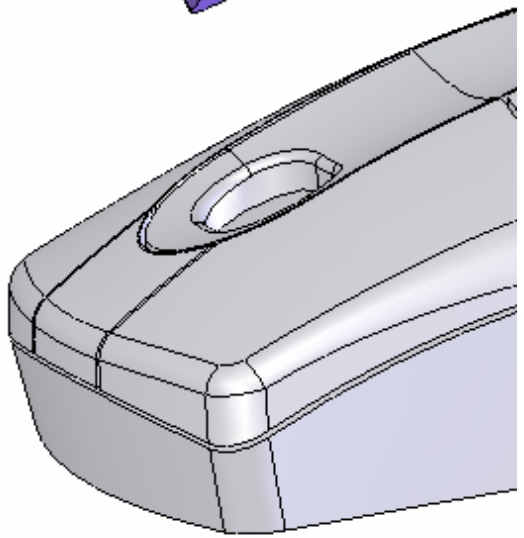


- Click “**Insert Components**” icon
- Select the file, click **Open**
- Drop the part next to the assembly as shown



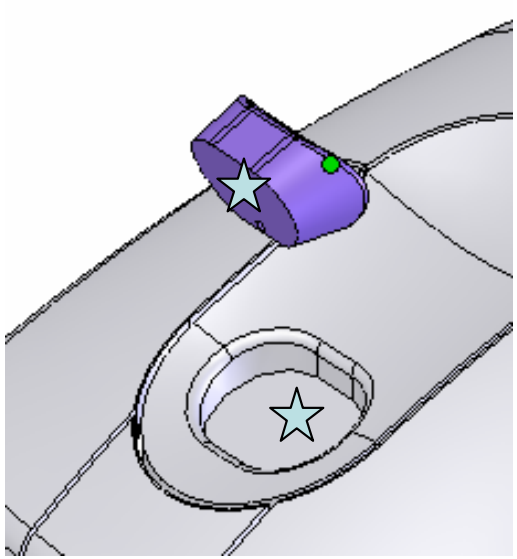


- Drag the part above the assembly by using these two commands “Move component” and “Rotate Component”
- (Remark: the scroll button is not symmetric, the side with a hole should be closer to the center of the mouse)



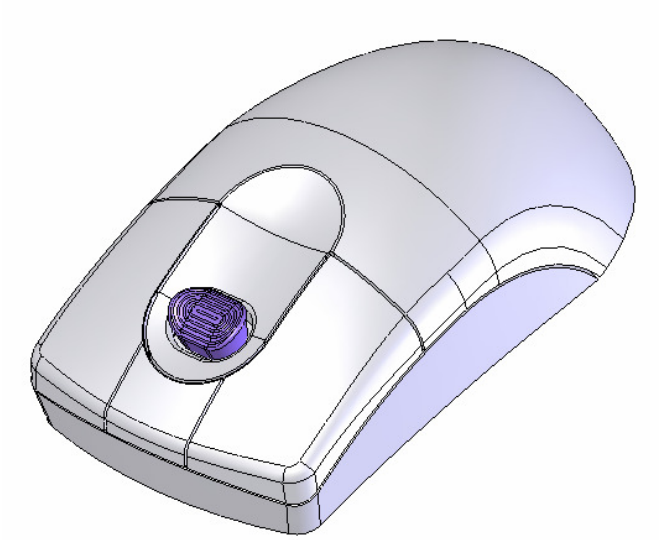
Tutorial 2C

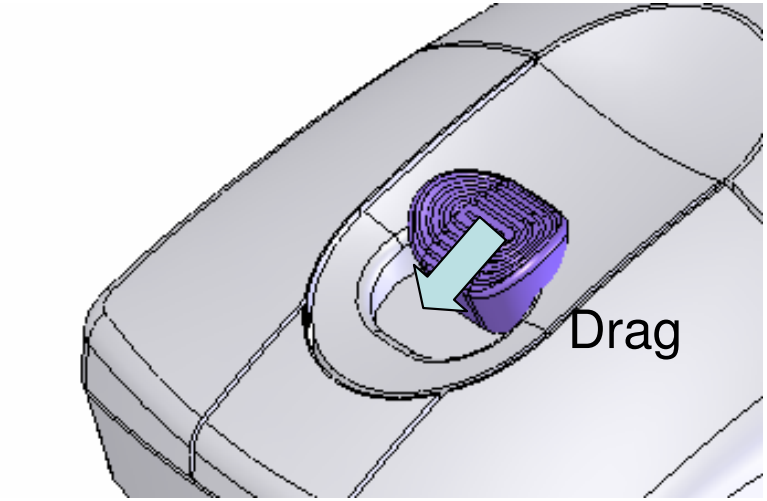
- Click **“Mate”** icon;
- Select “Right Plane” of Scroll Button;
- Select “Right Plane” of Upper_body;
- Press Enter key to confirm



- Select bottom face ☆ of Scroll Button
- Select bottom face ☆ of the pocket of Upper_body
- Press Enter key to confirm

- Drag the Scroll Button to the center of the pocket

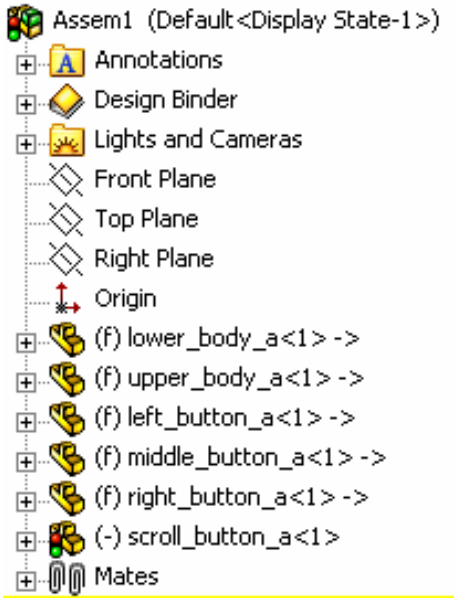


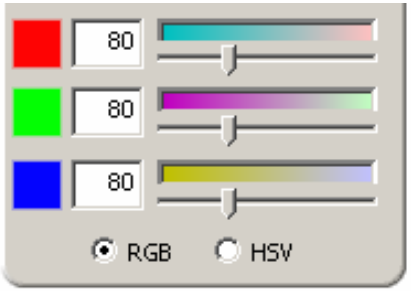


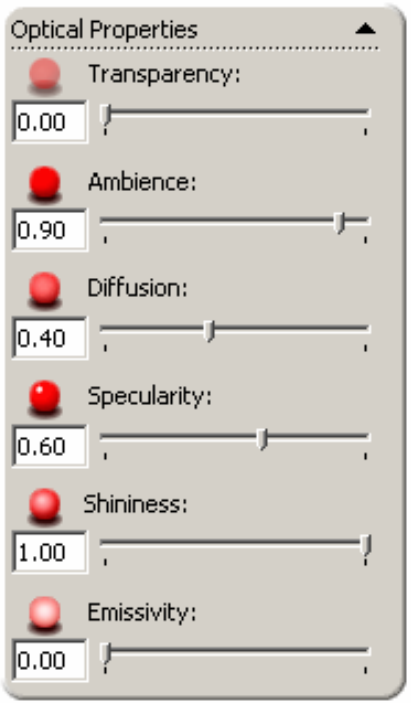
Tutorial 2C

To add material properties onto the parts:-

- Right-click “lower_body” on the tree
- Select “**Appearance /Color**” on the contextual menu
- Enter Color Red 80; Green80; Blue80
- Change the optical properties as below:
  - Transparency 0.0
  - Ambience 0.9
  - Diffusion 0.4
  - Specularity 0.6
  - Shininess 1.0
  - Emissivity 0.0
- Press Enter to confirm







- Repeat the steps onto other components except the scroll button.

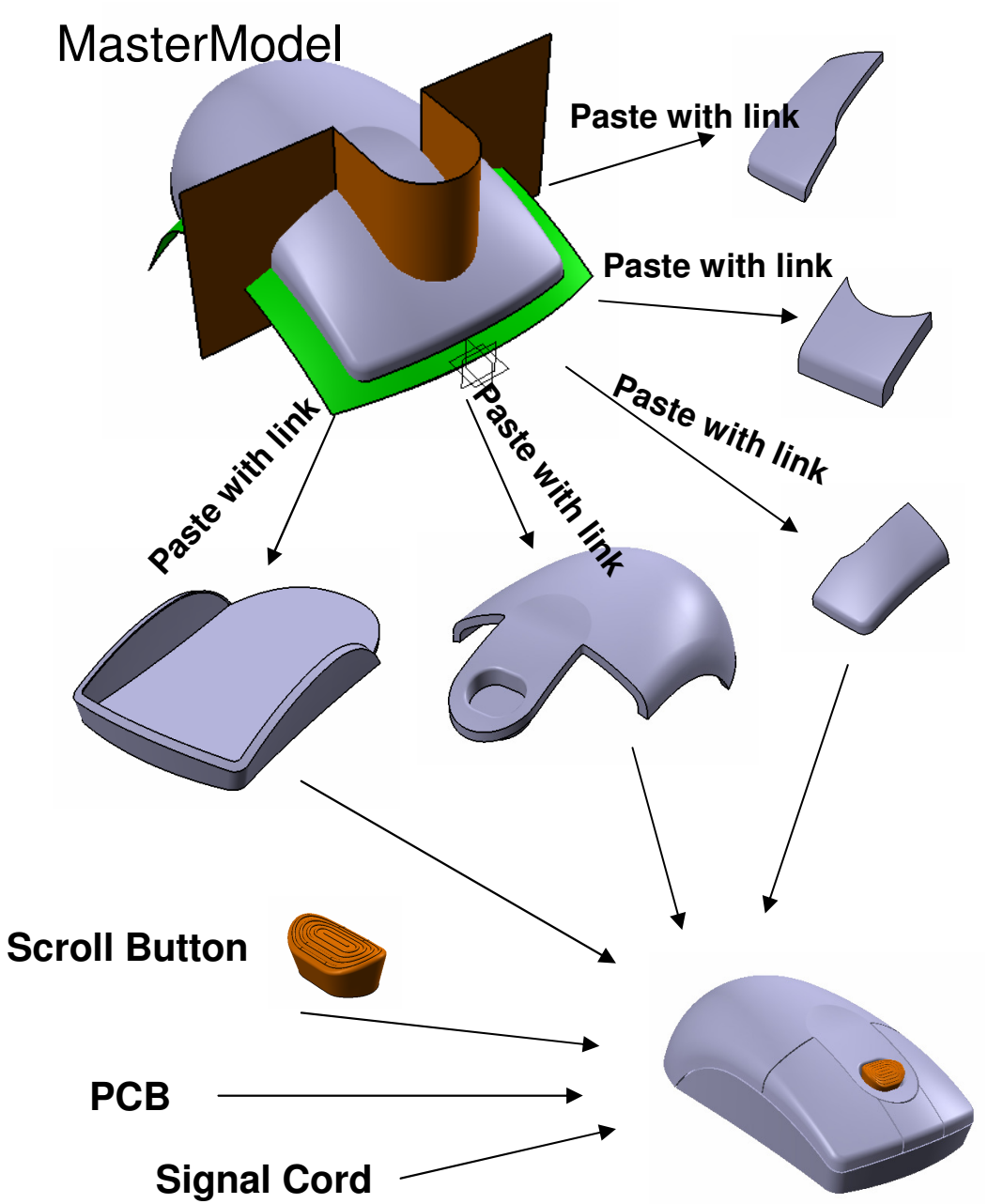
File /Save all

- Enter “**Mouse_assm_a.sldasm**” as the file name of the assembly

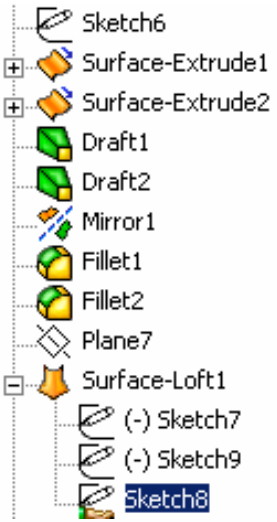
Tutorial 2C

The outlook of the mouse is controlled by the Master Model. If we make any change on it, the linked parts will be updated automatically. Also, because all components are created from one model, their surfaces & boundaries can match among themselves when assembled together.

Now we are going to modify the master model and see what will happen on the corresponding parts…

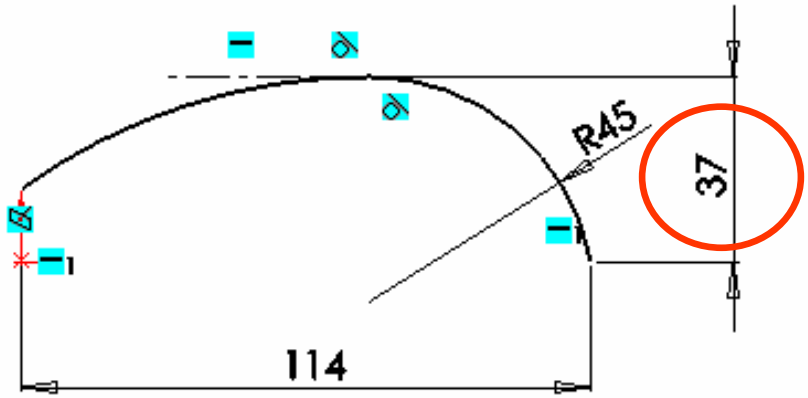


Tutorial 2C

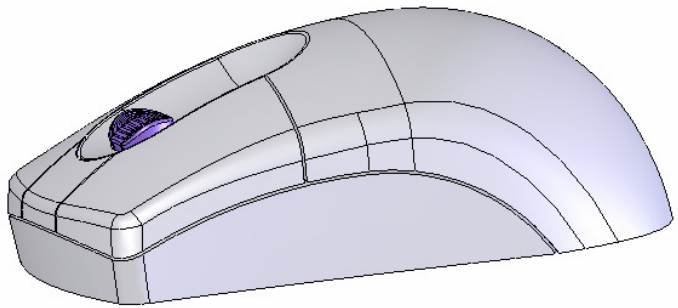


- File/Open... **Mouse_mastermodel_a.sldprt**

- Right-Click “Sketch.8” on the tree (which is under “Surface-Loft1”
- Select “Edit Sketch”
- **Change “37mm” to “40mm”**
- Click ‘Exit Sketch” icon to confirm
- (Now the mastermodel will be updated)



- Select “Window/Mouse_assm_a.sldasm”
- Click Yes to rebuild the assembly
- (All parts will then be updated automatically)



END of Tutorial 2C

Written by Dickson Sham

For enquiries, please contact:

Mr. Dickson S.W. Sham
Department of Mechanical Engineering,
The Hong Kong Polytechnic University

Tel : (852) 2766 4507
Email : mmdsham@polyu.edu.hk
Website : http://myweb.polyu.edu.hk/~mmdsham