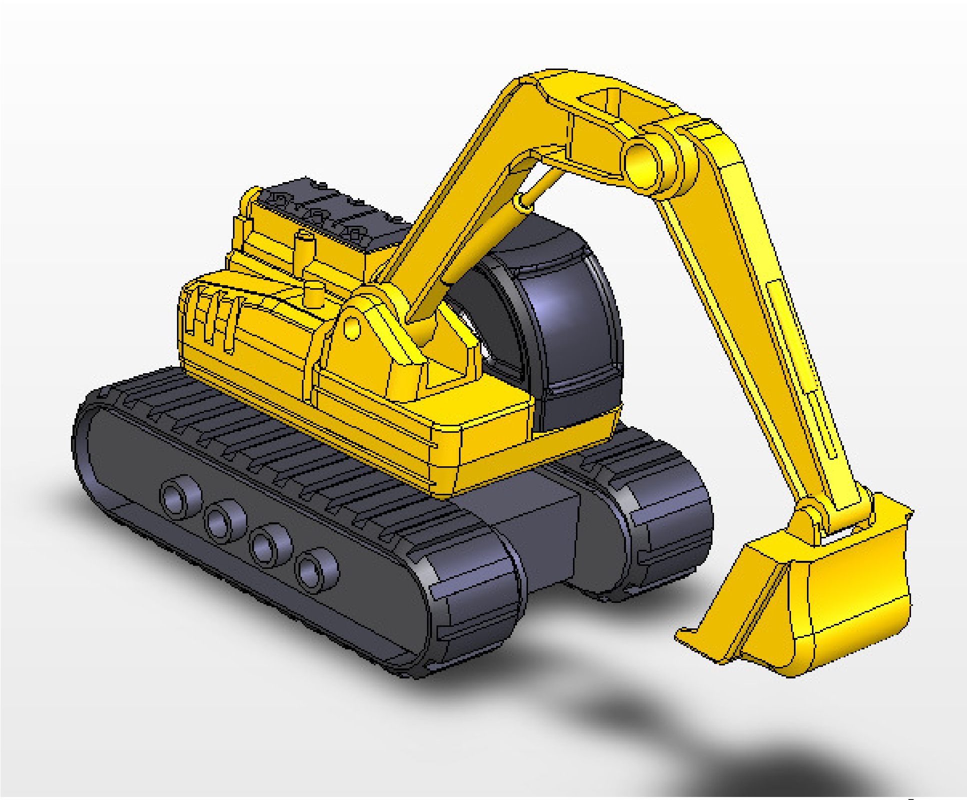
Solidworks 2006 Overview

(Tutorial 1-Toy Excavator)



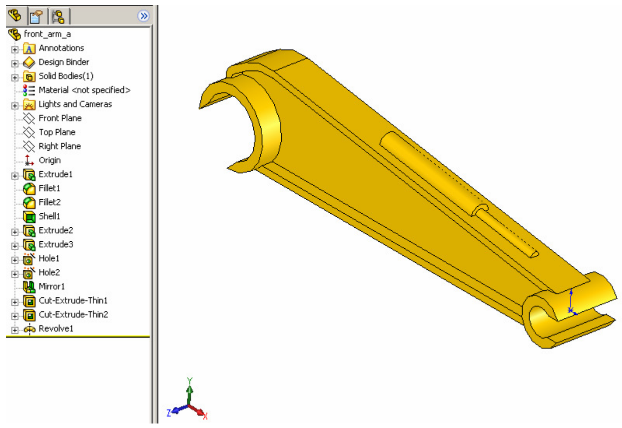
A- 1

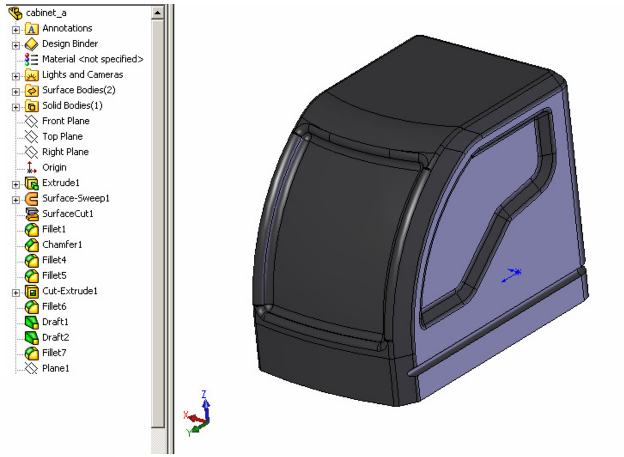
- Infrastructure
- Sketch
- Solid Features
- 2D-Drawing
- Surfaces
- Assembly Design

Written by Dickson Sham

Solidworks Overview

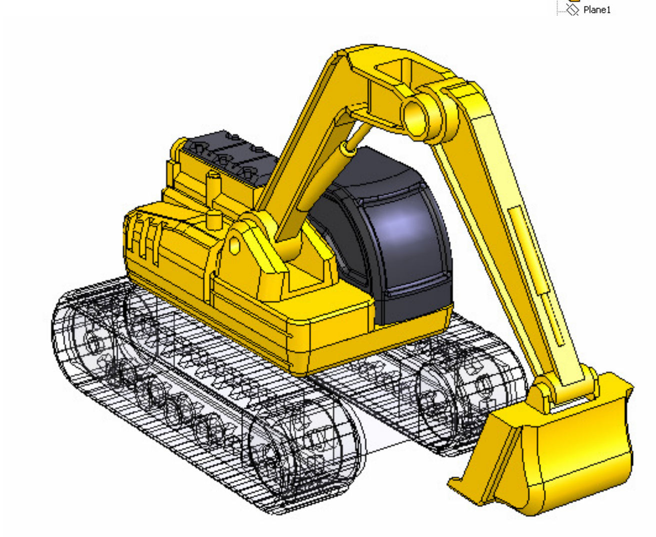
Tutorial 1A
- Solidworks Infrastructure
- Sketch
- Solid Features
- 2D-Drawing (projected from 3D)
- Auto-Update





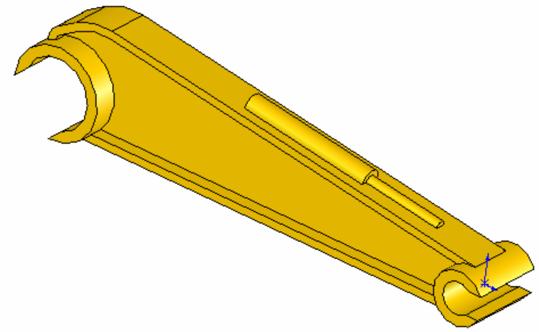
Tutorial 1B
- Solid Features
- Surfaces
- Real-time rendering & Material Mapping

Tutorial 1C
- Assembly Design
- Clash Detection & Part Modification

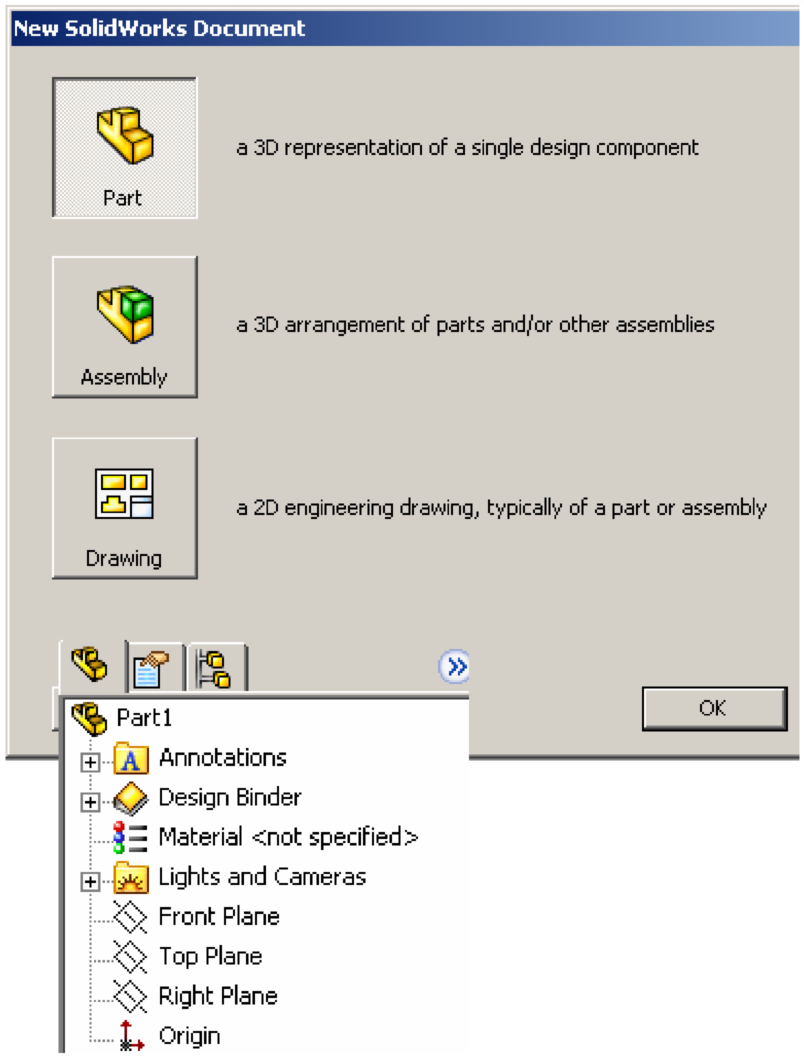


Please be reminded that this series of tutorials is designed to demonstrate a design approach with Solidworks, rather than the command itself.

Tutorial 1A



- Enter Solidworks 2006 by double-clicking its icon on the desktop.
- Select “**File**/**New…**” on the menu
- Click “**Part**” and then OK
- (A new file is created; an empty part tree appears on the left; Front, Top & Right Planes are hidden by default)
- (A blue dot appears on the working area, which is the system origin)



Change the view with the mouse

A. **Rotating** enables you to rotate the model around a point. Click and hold the middle mouse button, then drag the mouse.

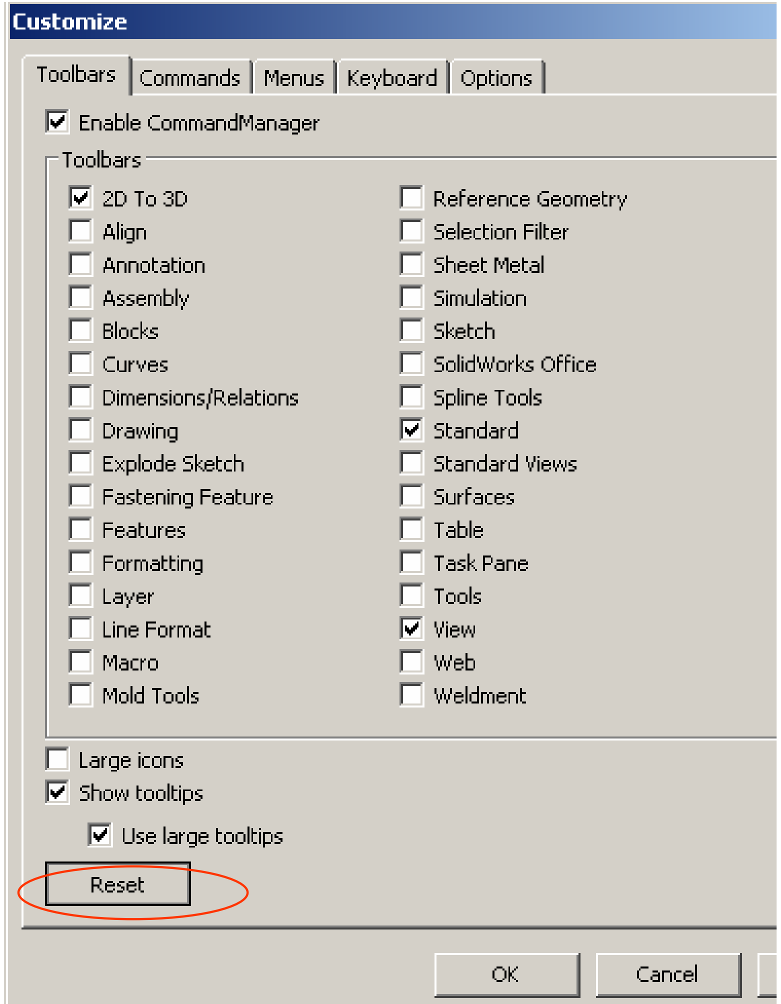
B. **Panning** enables you to move the model on a plane parallel to the screen. <u>Press and hold “Ctrl” key,</u> then click and hold the middle mouse button, then drag the mouse.

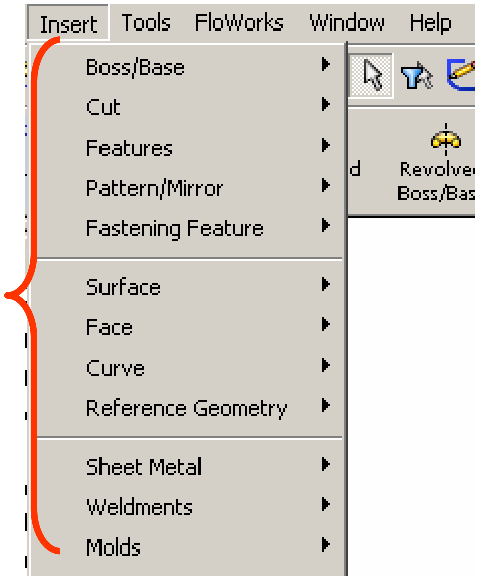
C. **Zooming** enables you to increase or decrease the size of the model. <u>Press and hold “Shift” key</u>, then click and hold the middle button, then drag the mouse up or down.

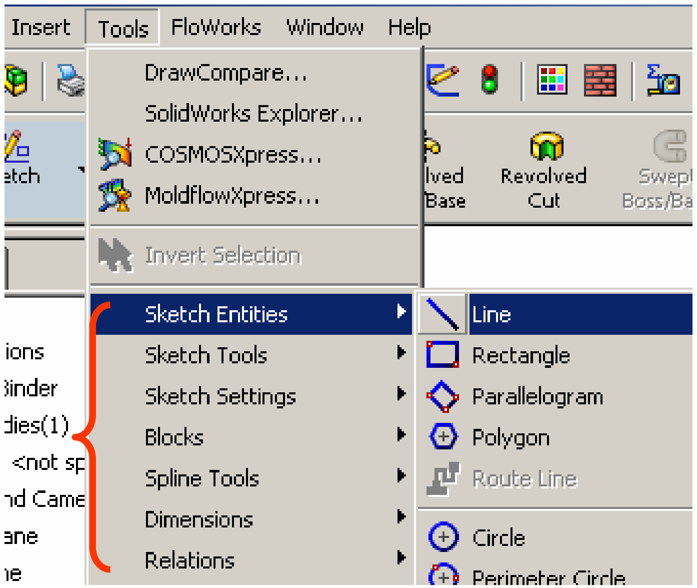
Tutorial 1A

To reset the layout of workbench:-

- Sometimes the workbench may not be tidy before you use; some toolbars are missing and some are at wrong positions. To reset the layout, select "**View/Toolbars/Customize…**" on the menu bar and select "**Reset**" on the first tab page.
- Check "**Enable CommandManager**"
- (Visible toolbars have a tick next to them)
- (The toolbar "Surfaces" is hidden by default; we can find all commands on the menu bar)







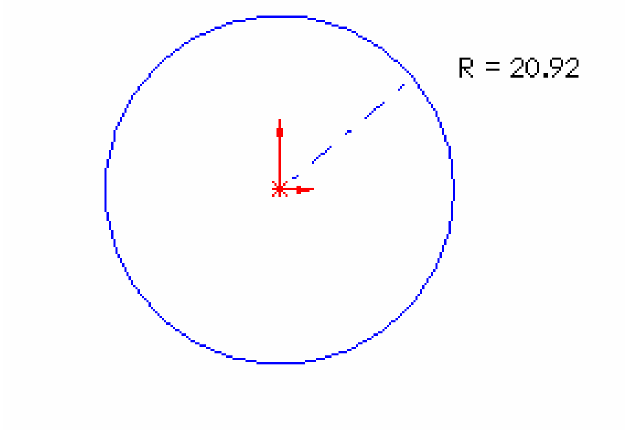
Tutorial 1A

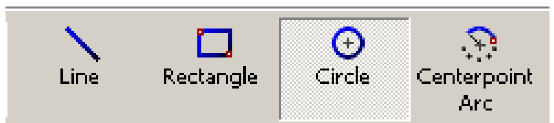
Features ON

Sketch ON

To build 1st sketch:-

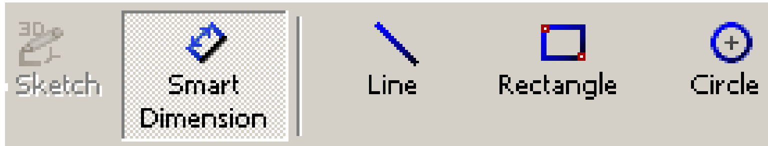
- Click “Sketch” (the second icon) to show all common icons related to “Sketch”
- click “**Sketch**” icon (3rd icon) and select **Front Plane on the tree.** (you can see the plane in the working area. You do not need to make it visible)
- Now the display is temporarily switched to the sketcher mode, in which you can draw 2D elements on the selected plane. (The viewpoint is changed automatically so that the plane’s normal is pointing directly to you)
- **Draw** a circle at the origin. 1st click is to define the centre and 2nd click is to define the radius. (no need to care too much about the position of 2nd click, we will define the radius later)



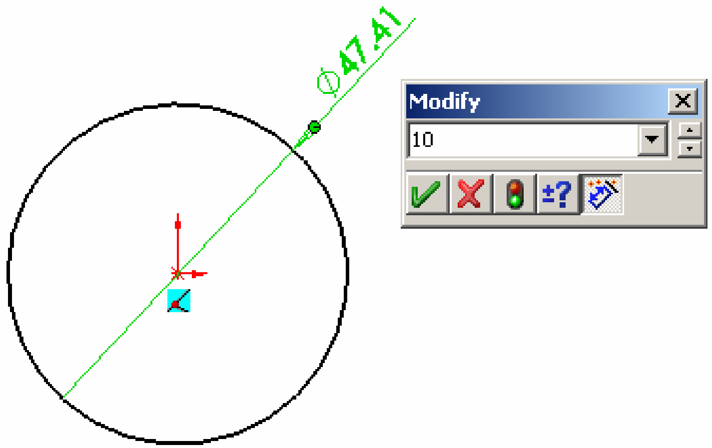


Tutorial 1A

- **To deactivate** “Circle” **command**, press “**Esc**” on the keyboard

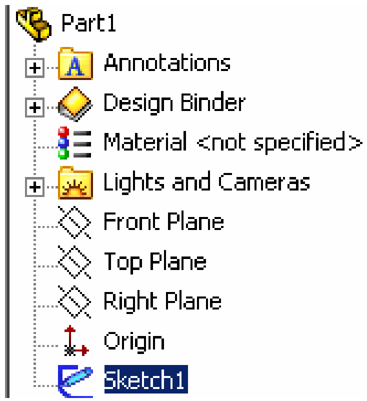


- (A coincident relation is created automatically and appears at the circle center; if you cannot see it, select “View/Sketch Relations” to make it visible)



- Add a dimension onto the circle by clicking “**Smart Dimension**” icon and then selecting the circle.
- Click on an open area to place the dimension
- Enter 10mm in the entry box and then press “Enter” on the keyboard; the circle will be resized automatically.

- Exit the sketch mode by clicking “**Exit Sketch**” icon
- Now, you are back to the 3D environment and “Sketch1” is created on the tree.

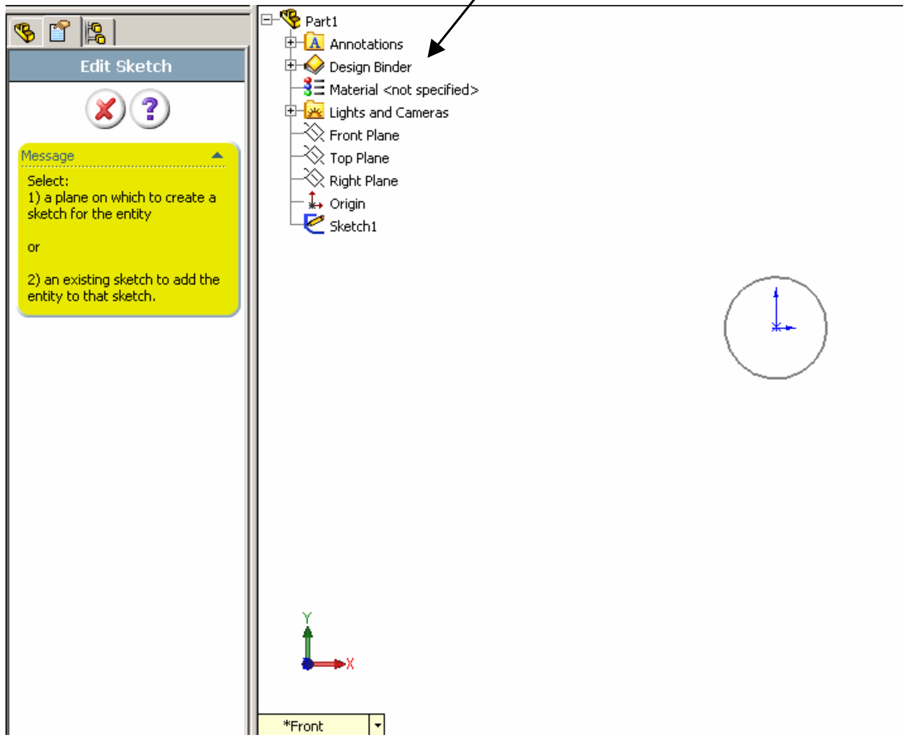


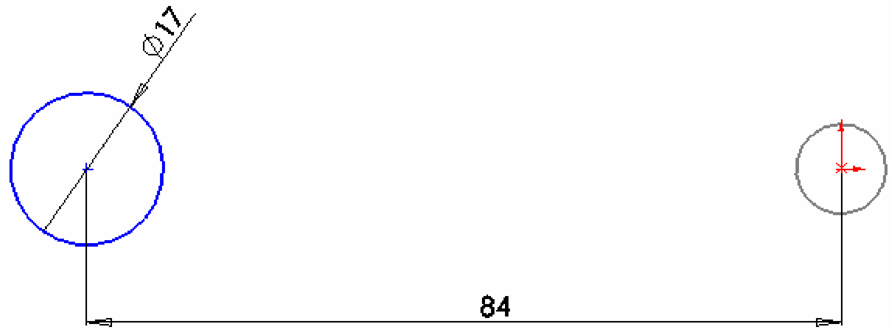
Tutorial 1A

To build 2nd sketch:-

- Click somewhere near the circle to deselect Sketch1. (The circle color is then changed from green to gray)
- Click “**Sketch**” icon again and select **Front Plane** again to draw another sketch.
- **Draw** a circle on the left of the previous circle. With the help of auto-detection, you can define the center on the x-axis. (no need to care too much about the size and the position, we will define later).
- **To deactivate** “Circle” **command**, press “**Esc**” on the keyboard
- Click “**Smart Dimension**” icon and select the circle. **Modify** its diameter as <u>17mm</u> and then press “Enter” on the keyboard to confirm.
- Select the two circle centers. Modify the distance as <u>84mm</u>. (You will see that only the current circle will move correspondingly. Remark: you cannot modify any elements that do not belong to the sketch.)
- Exit the sketch mode by clicking “**Exit Sketch**” icon
- You can see “Sketch2” on the tree.

Tree appears in working area



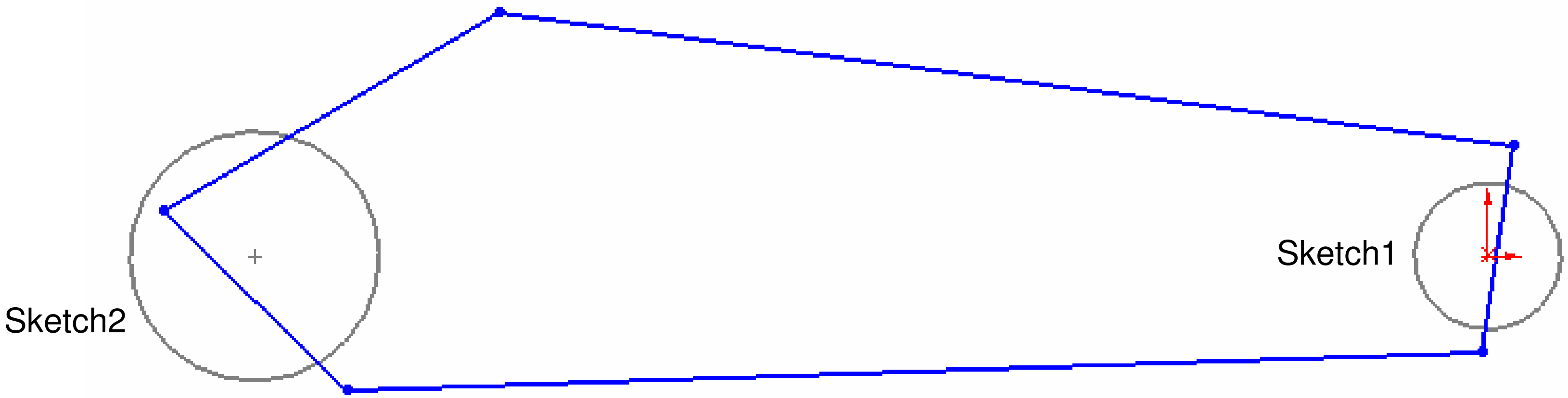


Tutorial 1A

To build 3rd sketch:-

- Click somewhere near the 2nd circle to deselect Sketch2.
- Click "**Sketch**" icon and select **Front Plane** again to draw another sketch.
- **Draw** a profile as below (Five straight lines forming a closed profile).

- REMARK: To temporarily deactivate the auto-creation of relations or auto-snapping, press "CTRL" key on the keyboard while drawing the profile. To snap the last point onto the first point of the profile, remember release the "CTRL" key.
- There should be no geometrical relations on the profile. If one exists, select and delete it.

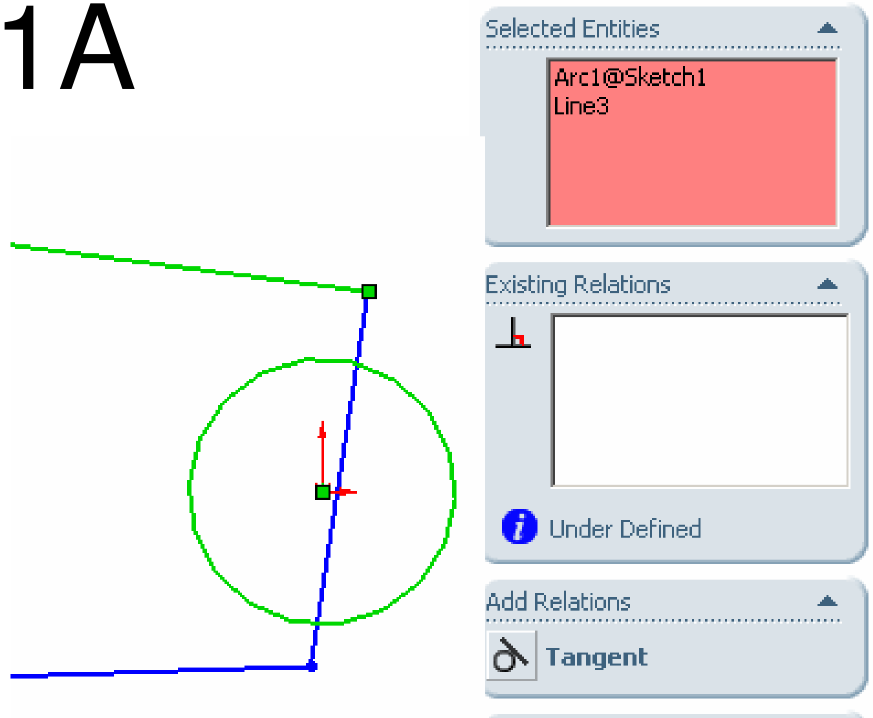


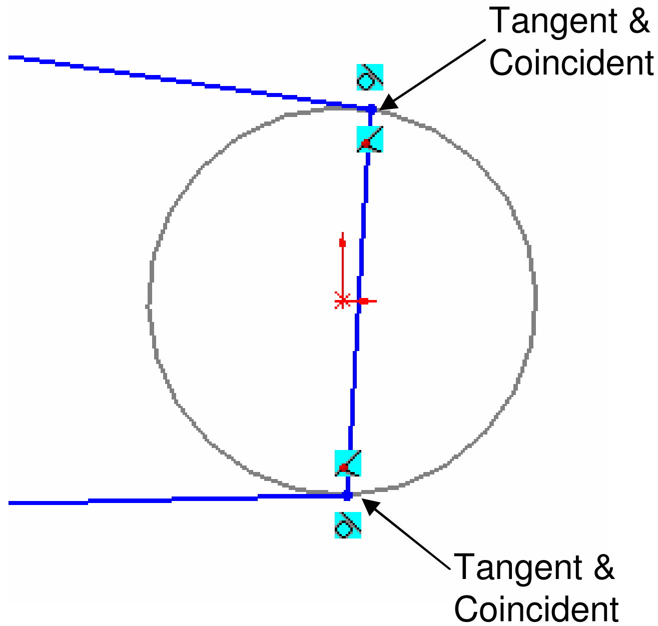
Tutorial 1A

- (To ensure the lines are tangent to the small circle, we need to add a tangency relation between them)

- **Multi-select** the upper line and the small circle by pressing and holding “**ctrl**” key on the keyboard. Select “**Tangent**”.
- **Multi-select** the endpoint of the line and the small circle by pressing and holding “**ctrl**” key on the keyboard. Select “**Coincident**”.
- (No need to click “Close Dialog”)

- **Multi-select** the lower line and the small circle by pressing and holding “**ctrl**” key on the keyboard. Select “**Tangent**”.
- **Multi-select** the endpoint of the line and the small circle by pressing and holding “**ctrl**” key on the keyboard. Select “**Coincident**”.
- (No need to click “Close Dialog”)





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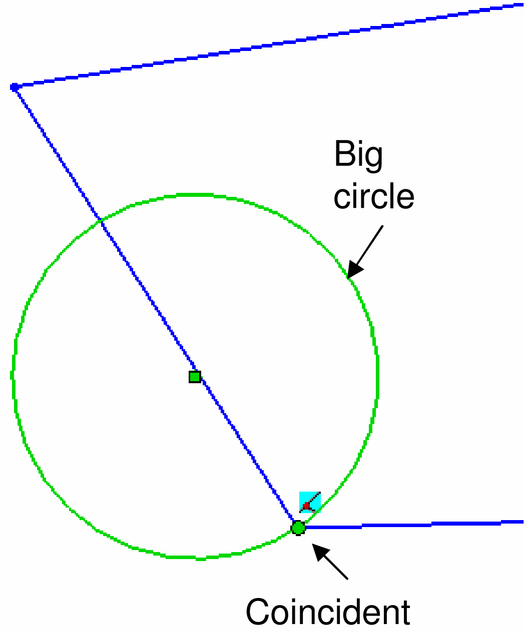
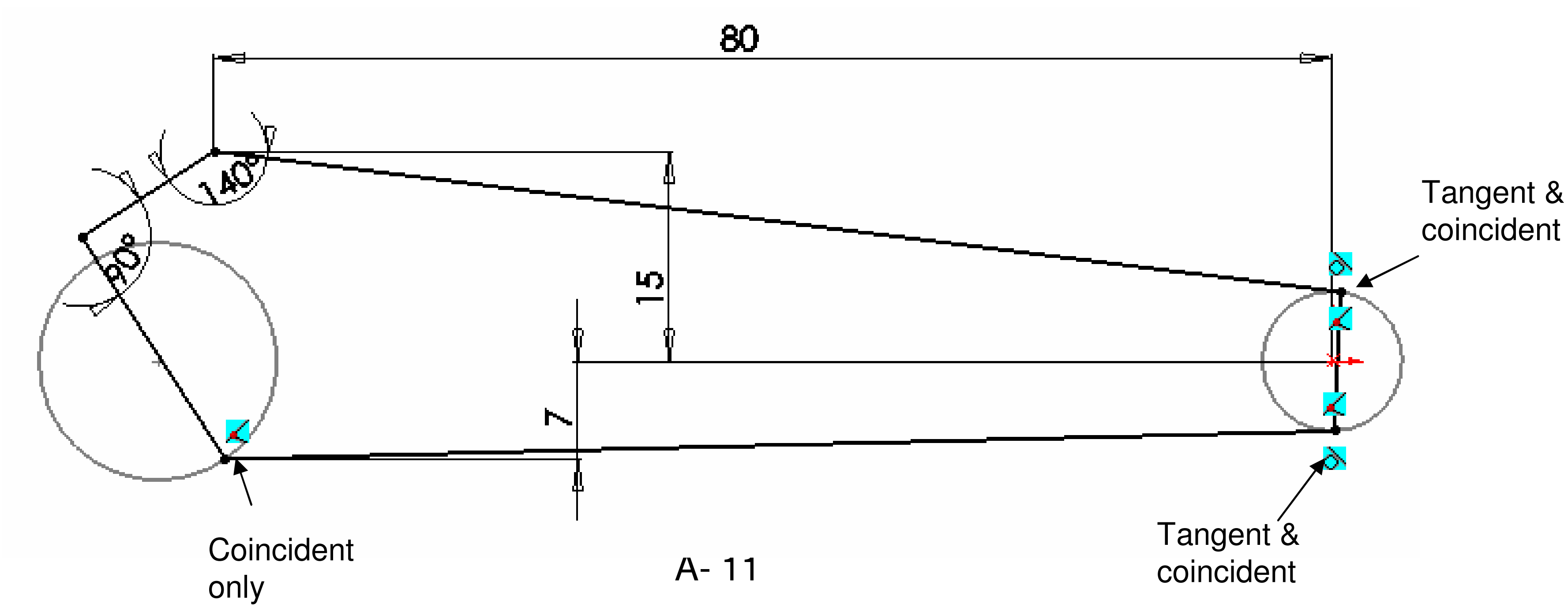
Tutorial 1A

- **Multi-select** the endpoint of lower line and the big circle by pressing and holding “**ctrl**” key on the keyboard.
- Select “Coincident”.
- (No need to click “Close Dialog”)



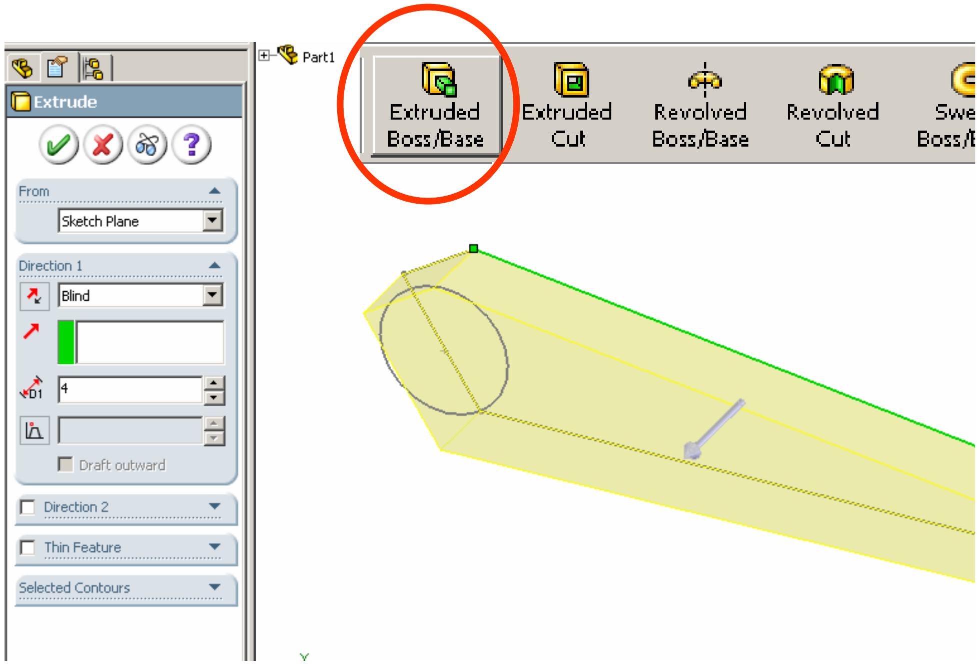
- Continue to add the dimensions as shown so that the profile is fully-constrained.
- Exit when it is complete.
- Now, you should see Sketch1, Sketch2 and Sketch3 on the tree.



Tutorial 1A

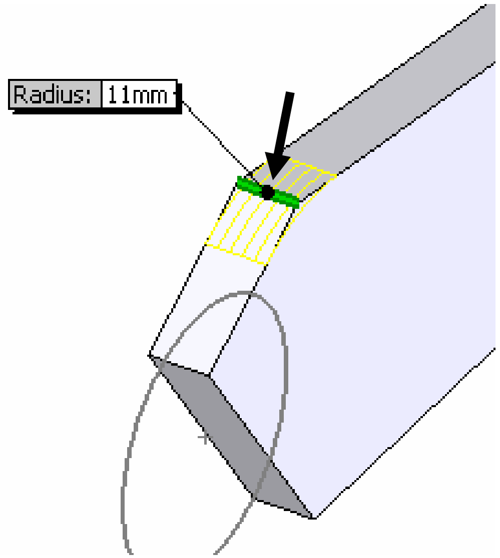
To build a solid:-

- Click “Features” icon (First icon) to show the solid feature icons
- Select “Sketch3” on the tree / directly click on the geometry
- Click “**Extrude Boss/Base**” icon.
- Enter 4mm as D1 (First Limit)
- Press ‘Enter” key on the keyboard to complete.
- A solid is created.
- (Normal of profile is chosen as the direction of extrusion by default)



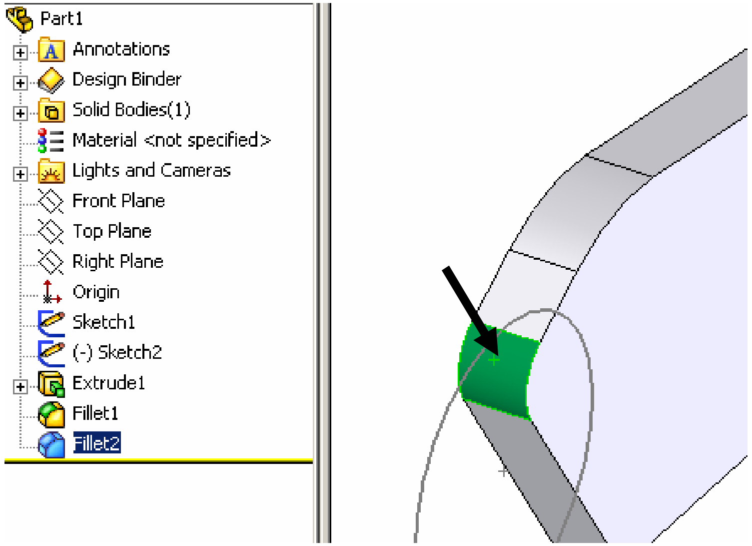
To round the sharp edge:-

- Click “**Fillet**” icon
- Select “Constant Radius” as type
- Enter 11mm
- Select the edge
- Press ‘Enter” key to complete.



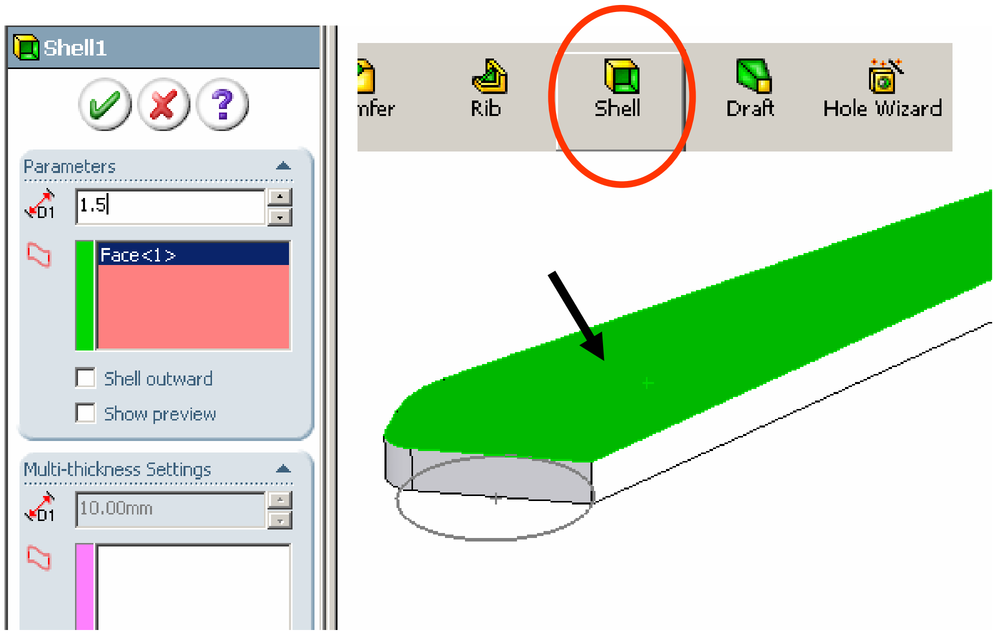
Tutorial 1A

- Add another “**Fillet**”
  - Constant Radius
  - R3mm.



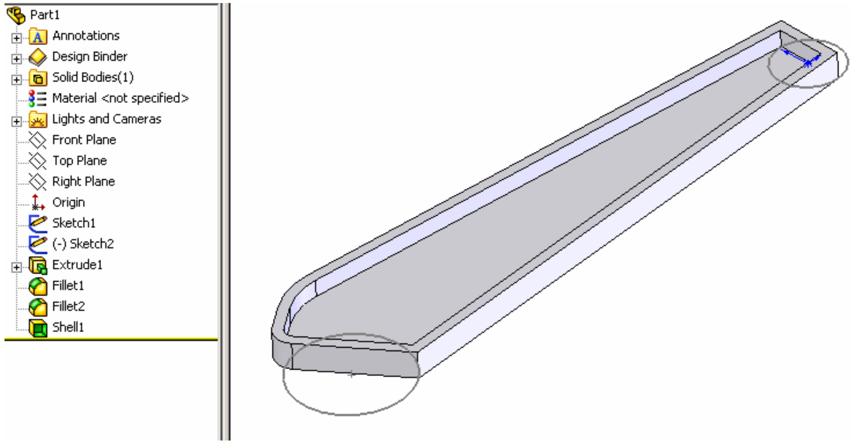
To make the solid hollow:-

- Click “**Shell**” icon.
- Enter 1.5mm as “D1 thickness”.
- Do not select “Shell outward”
- Select the top surface of the solid, which is considered as “Face to remove”
- Press ‘Enter” key to complete.



Tutorial 1A

- You should now have a model as shown on the right; all the wall thickness is 1.5mm, and the top cover is removed.

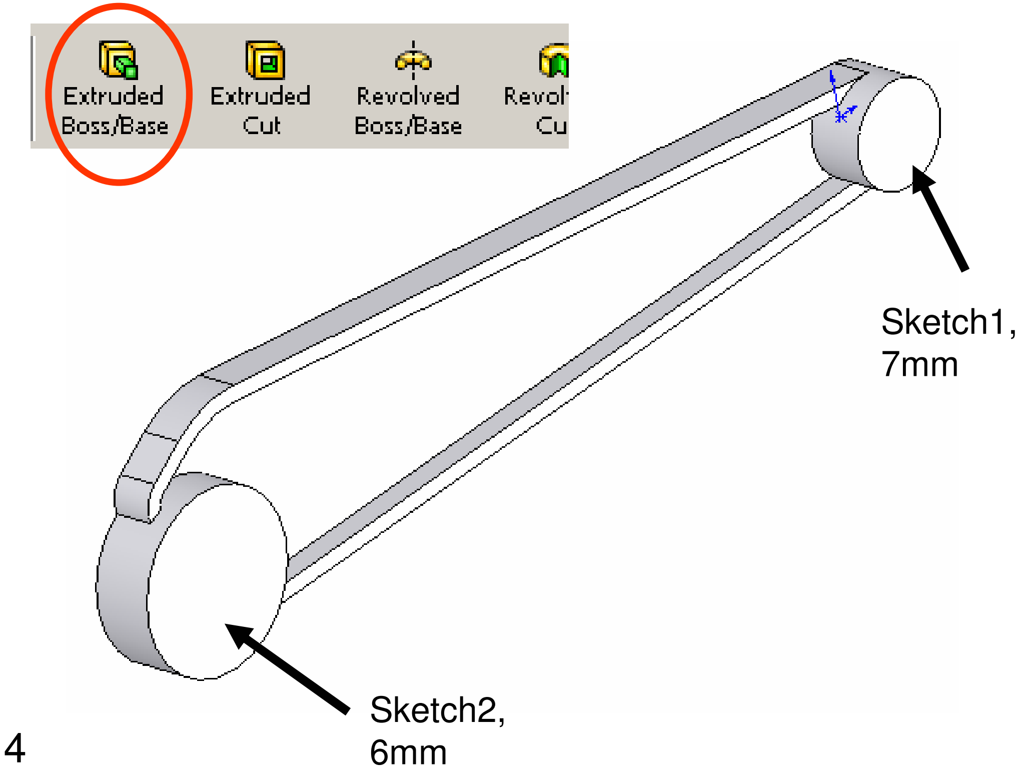


To build 2 more Extruded Bosses:-

- Click "**Extruded Boss/Base**" icon
- Select "Sketch1"
- Enter 7mm as D1
- Press "Enter" key to complete.

Similarly,

- Click "**Extruded Boss/Base**" icon again
- Select "Sketch2"
- Enter 6mm as D1.
- Press "Enter" key to complete.

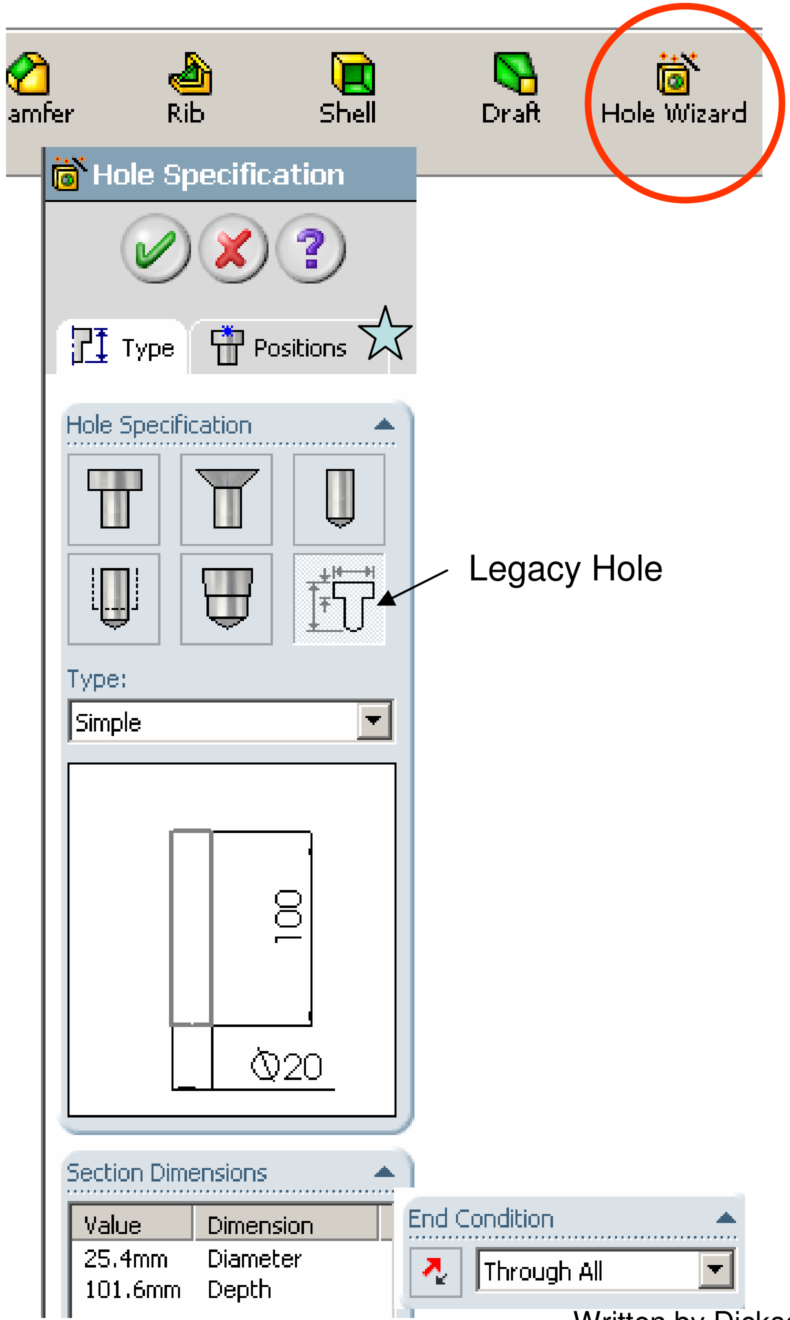


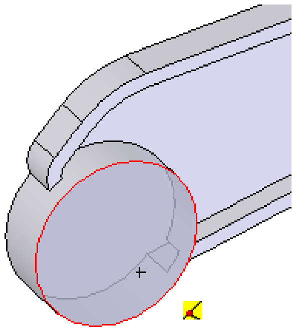
Tutorial 1A

To make a hole:-

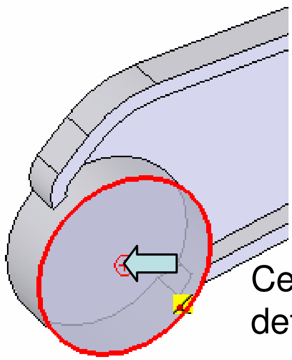
- Click "**Hole Wizard**" icon.
- Select "Legacy Hole" as Hole Specification
- Select "Simple" as Type
- Double-click the value of Diameter and change it to 13mm
- Select "Through All" as End Condition

- Select the tab page "Positions" ☆
- Move the mouse cursor onto the top surface of the bigger cylinder until it is snapped onto the circle center. Click once to accept
- Click "OK" icon to complete





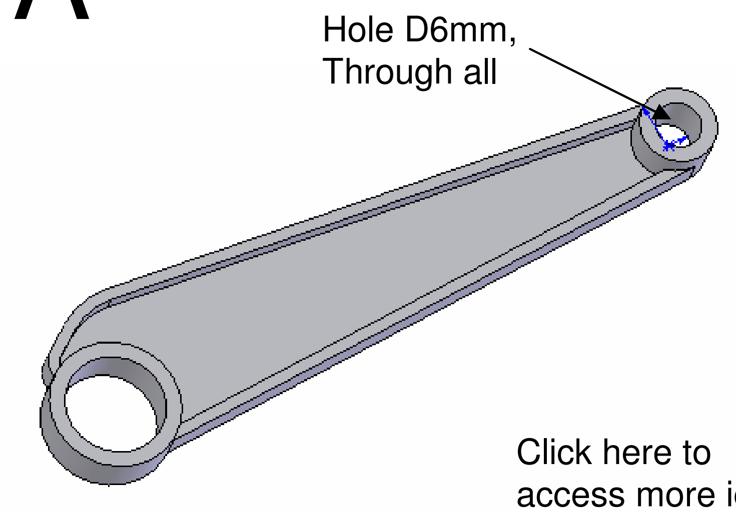
Center is NOT detected



Center is detected

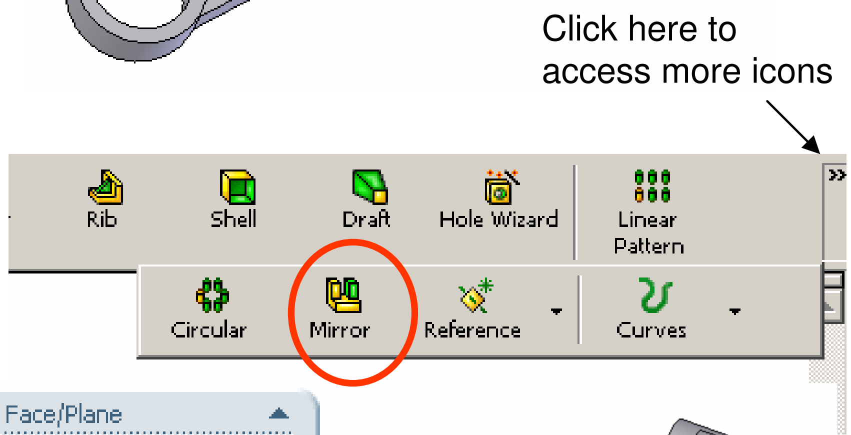
Tutorial 1A

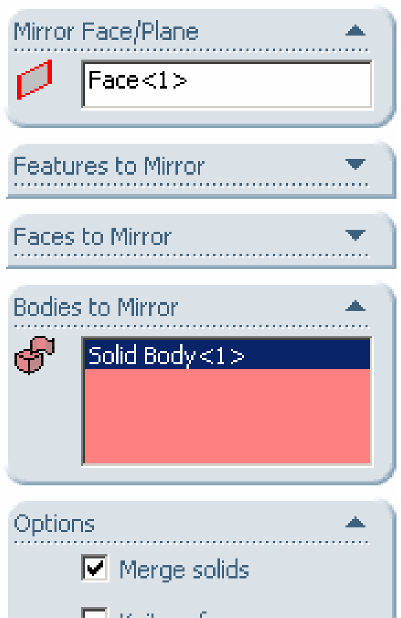
- Make another hole Dia6mm on the smaller cylinder in the same way…

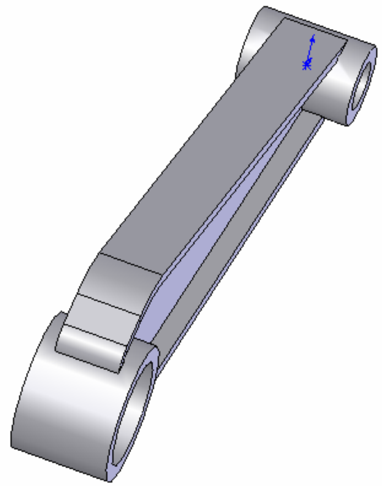


To Duplicate another half:-

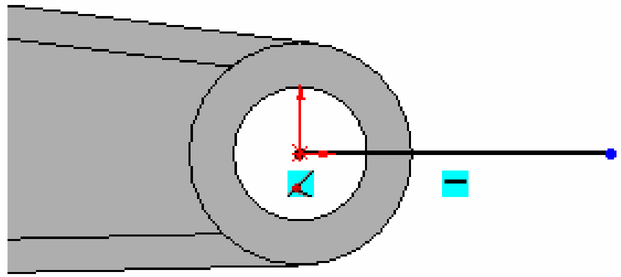
- Click >> on the rightmost to access more icons
- Click "**Mirror**" icon and select **Front Plane or the bottom planar face** as Mirror Plane
- Click "Bodies to Mirror"
- Select the solid
- Press "Enter" key to complete







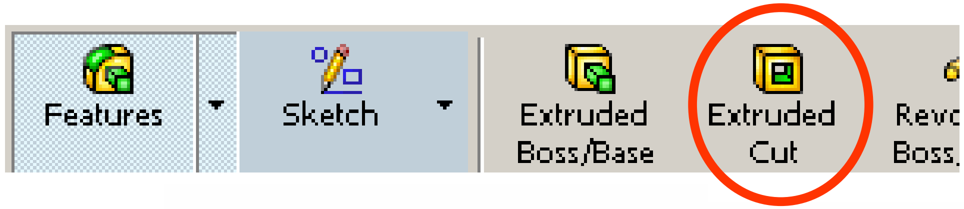
Tutorial 1A



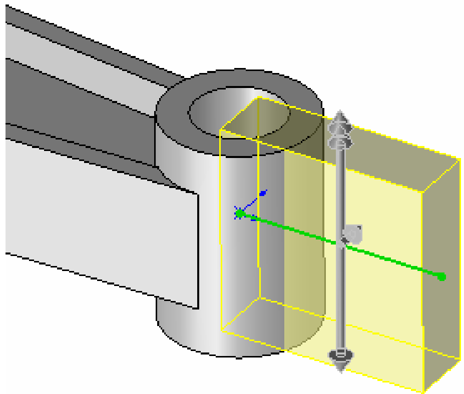
To build a sketch (open profile):-

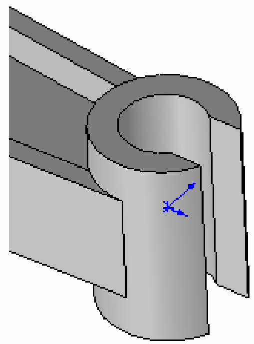
- Click “**Sketch**” icon and select “**Front Plane**”
- Press “Space” key on the keyboard and double-click “Normal to” on the popup list
- Draw a horizontal line, w/ one end at center of the small circle and the other outside it.
- No need to specify its length.
- Click “Exit Sketch” icon to exit.

To remove material with an open profile:-



- Click “Features” icon and then click “**Extruded Cut**” icon
- Select “Through All” for both first direction & second direction
- Select “Thin Feature” and then “Mid-Plane”
- Enter <u>4.4mm</u> as T1
- Press “Enter” key to complete

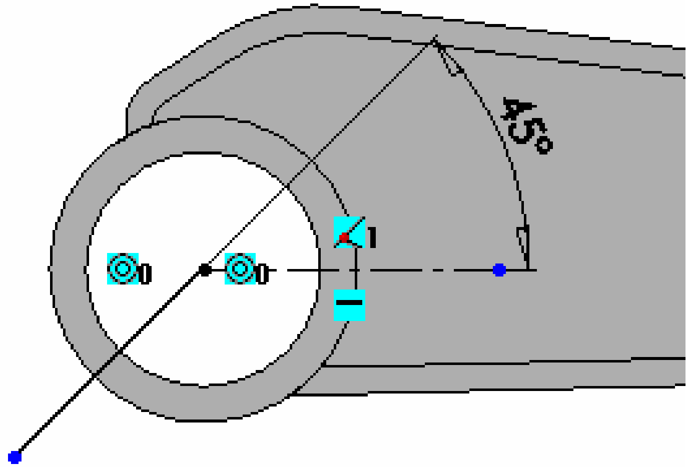




Tutorial 1A

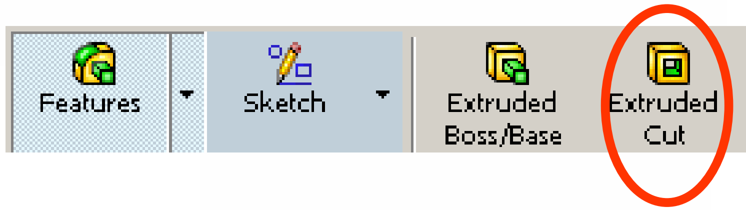
Similarly, to build another sketch (open profile):-

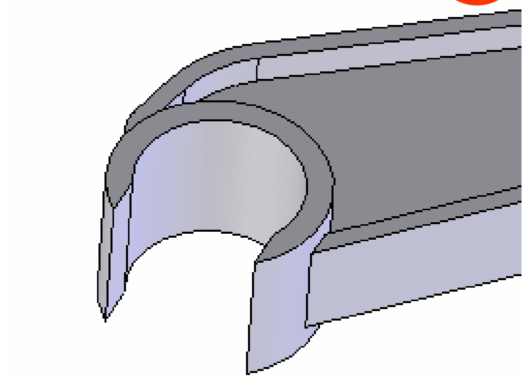
- Click “**Sketch**” icon and select “**Front Plane**”
- Draw a inclined line, w/ one end near center of the big circle and the other outside it
- Add a “concentric” relation to ensure that the endpoint is at the circle center
- Draw a horizontal centerline from the circle center
- Inclined angle =45 deg from the axis.
- No need to specify its length.
- Click “Exit Sketch” icon to exit.



To remove material with an open profile:-

- Click “Features” icon and then click “**Extruded Cut**” icon
- Select “Through All” for both first direction & second direction
- Select “Thin Feature” and then “Mid-Plane”
- Enter <u>11.6mm</u> as T1
- Press “Enter” key to complete

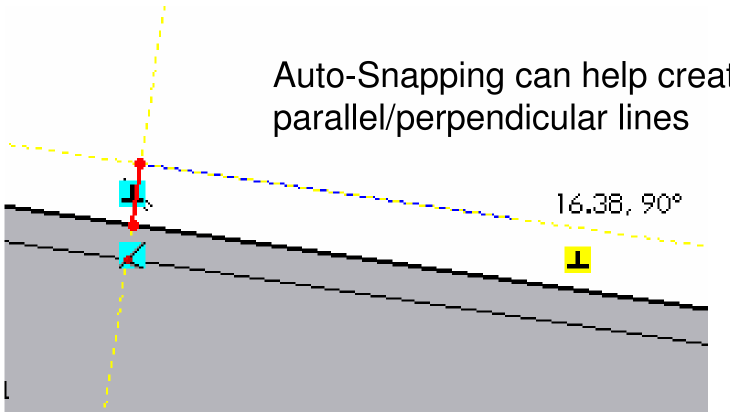




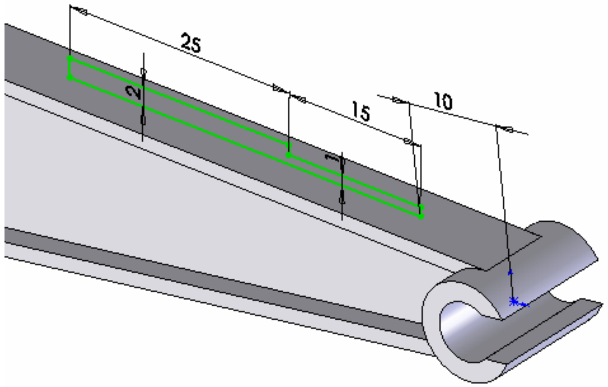
Tutorial 1A

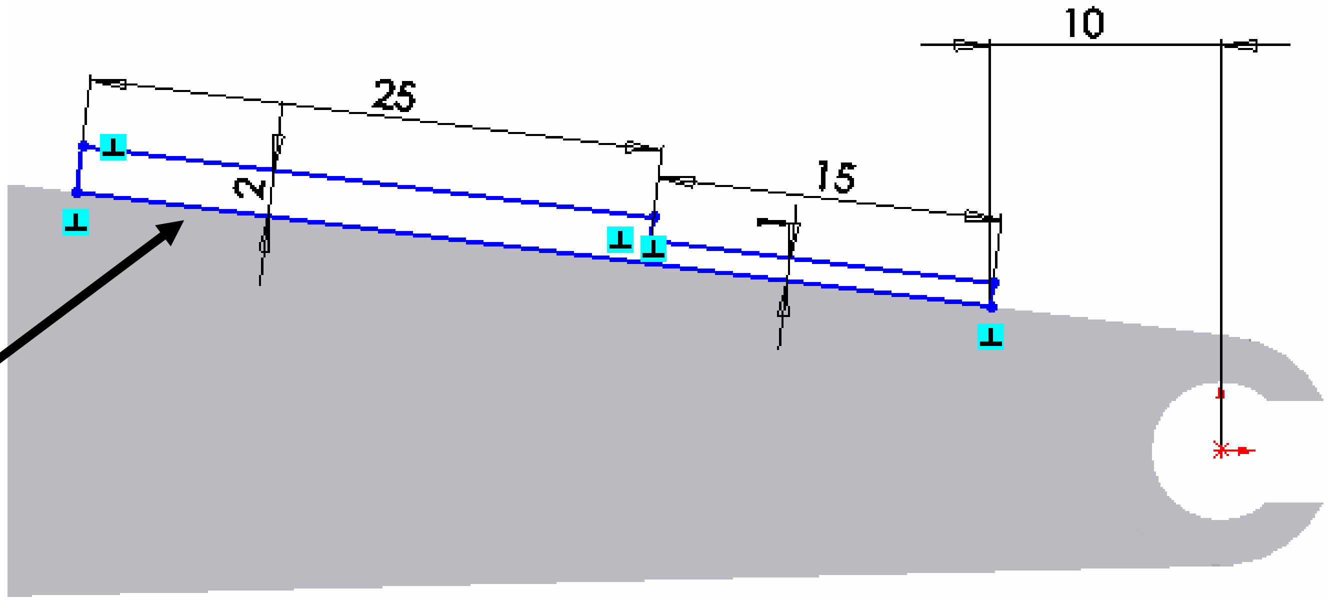
To build a new sketch:-

- Click “**Sketch**” icon and select “**Front Plane**”
- Draw the profile as shown.
- Click “Exit Sketch” to complete.



Auto-Snapping can help creating parallel/perpendicular lines



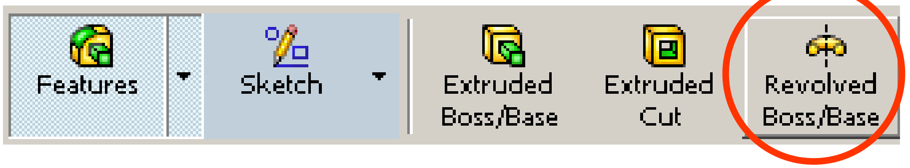


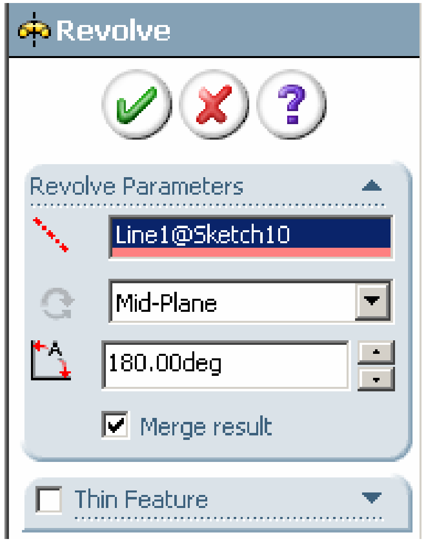
The solid line is coincident with the solid surface

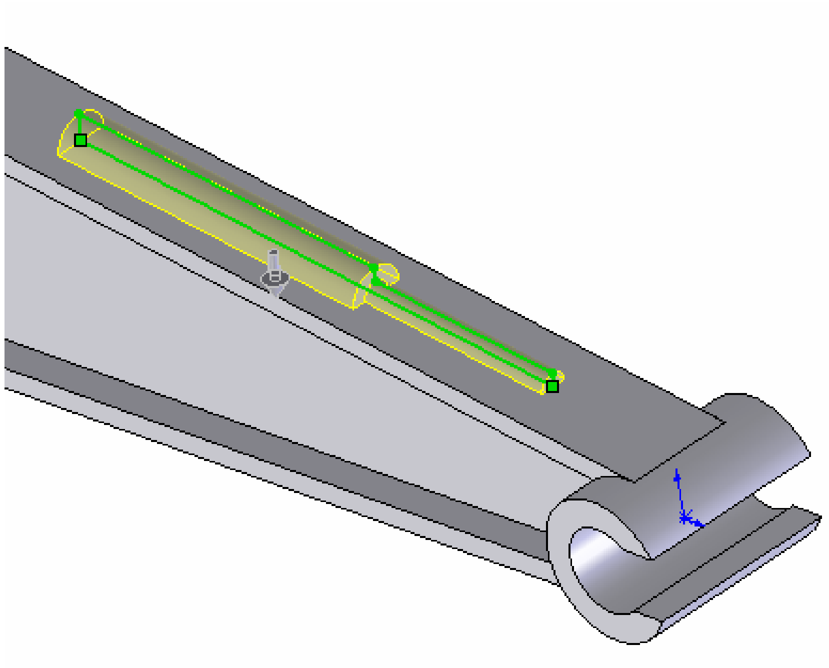
Tutorial 1A

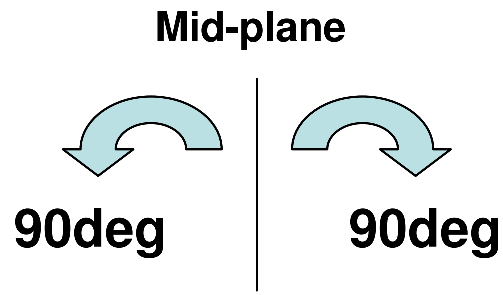
To add material by rotating a sketch:-

- Click “**Revolved Boss/Base**” icon to add material by rotation.
- Select the bottom line as the axis of revolution
- Select “Mid-Plane” as type
- Enter <u>180deg</u> as A (angle)
- Press “Enter” key to complete





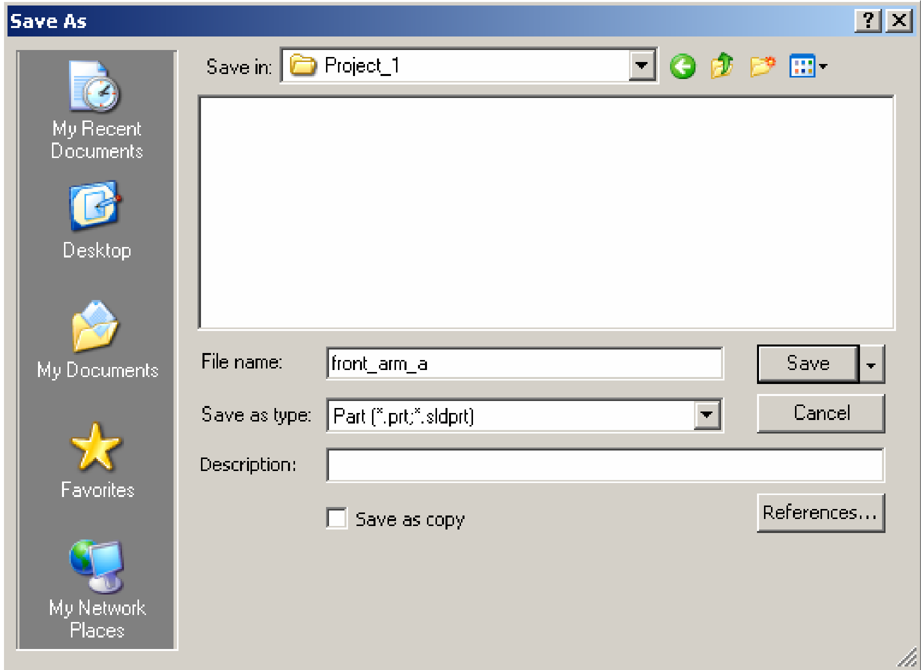




Tutorial 1A

To save the new part in a Project Folder:-

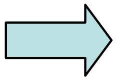
- It is a good practice to store all part files of a product in one specific folder.
- Create a folder wherever you can save (by MS window technique).
- Save your current part as “**front_arm_a.sldprt**” into the folder.
- Add “a” after its name to remind us its version. For example, I sent you the part with version “a” some days ago. But now I modify the part and resend you with version “b”. When you see both files, you know which is the latest one.

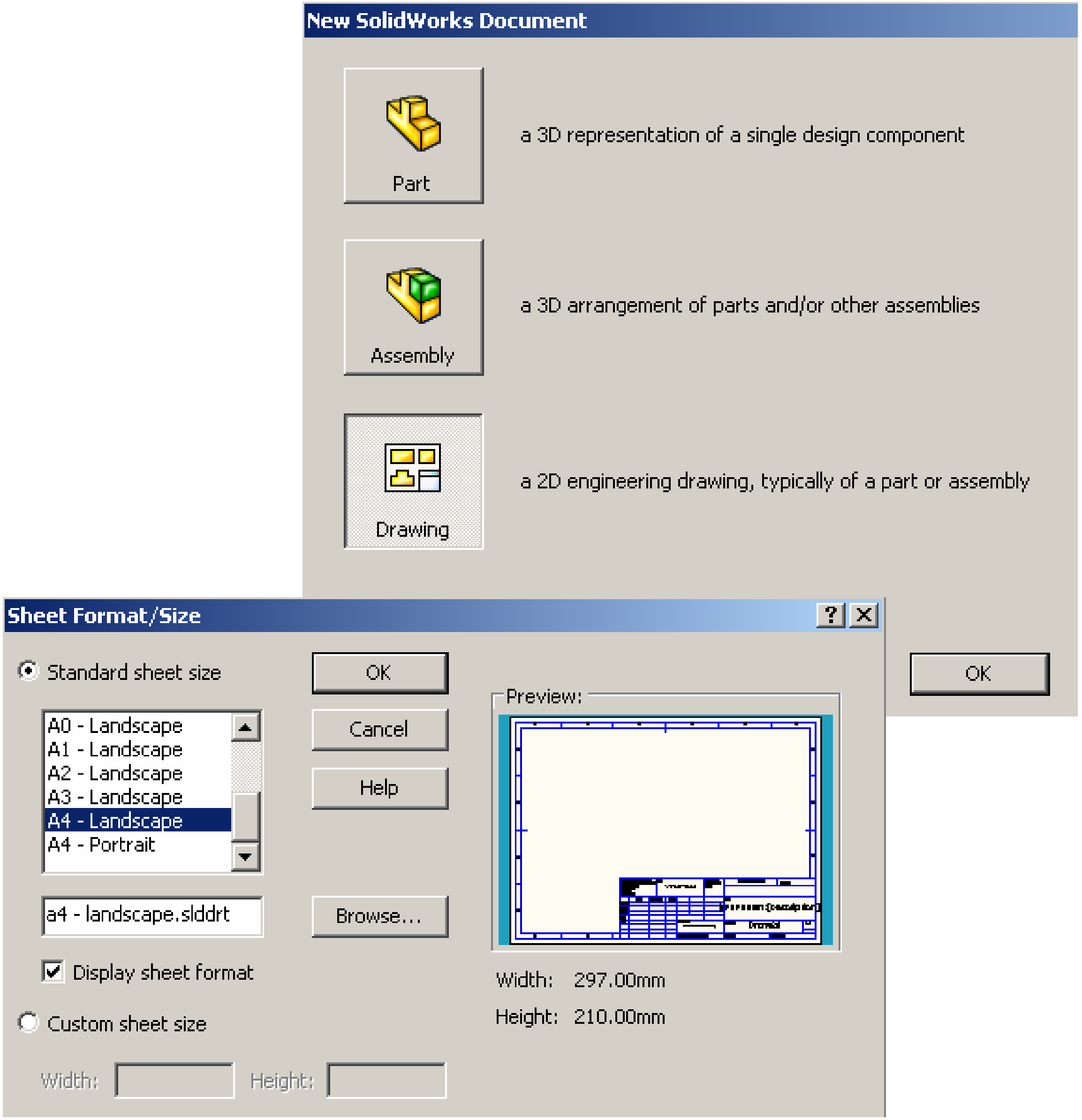


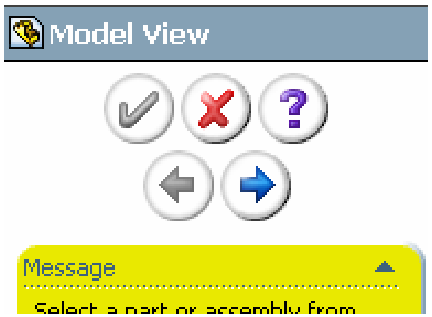
- (Remark: while saving the file, the system will capture the current viewpoint as the file preview.)

Tutorial 1A

To create a 2D drafting:-

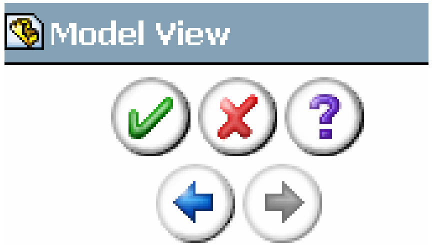
- Select **“File/New…”** on the menu
- Click “**Drawing**” and then OK
- Select “A4- Landscape” as sheet size
- Click ok
- (“front_arm_a” should be selected)
- Click  to proceed

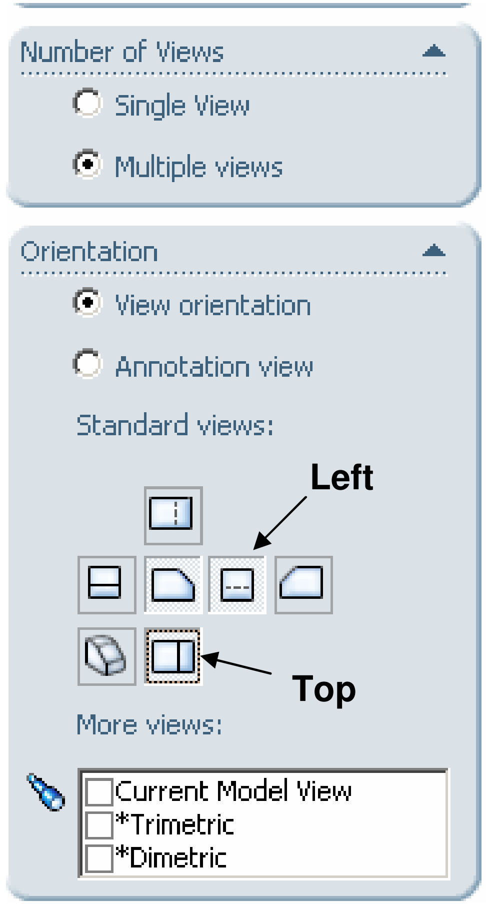




Tutorial 1A

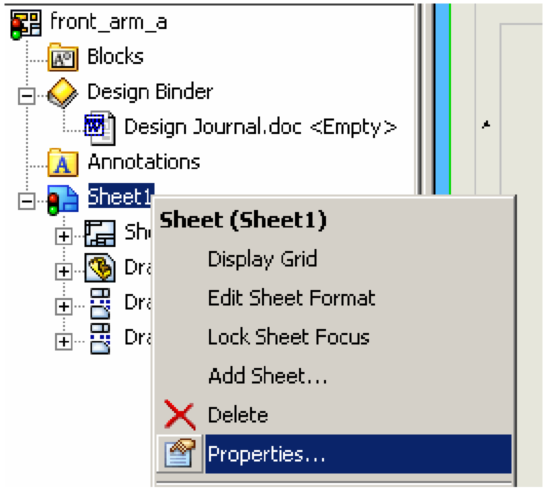
- Select “Multiple Views”
- Click the icons “Left” & “Top”
- Click ok to complete (Green Arrow)





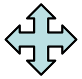
To define the projection angle:-

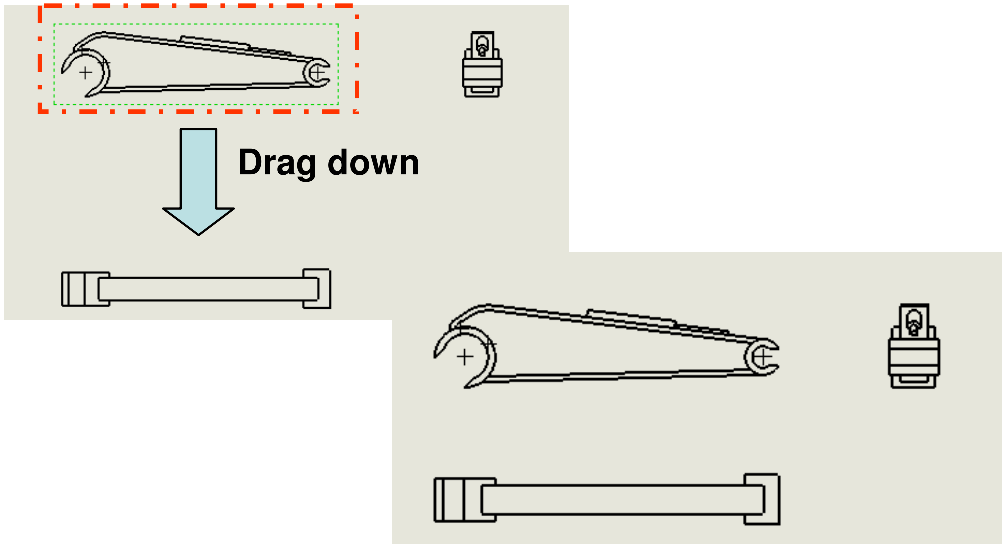
- Right-click “sheet1” on the tree
- Select “Properties…”
- Select “**Third Angle**” as Type of Projection
- Click ok to complete
- (All views are updated, according to the new projection)



Tutorial 1A

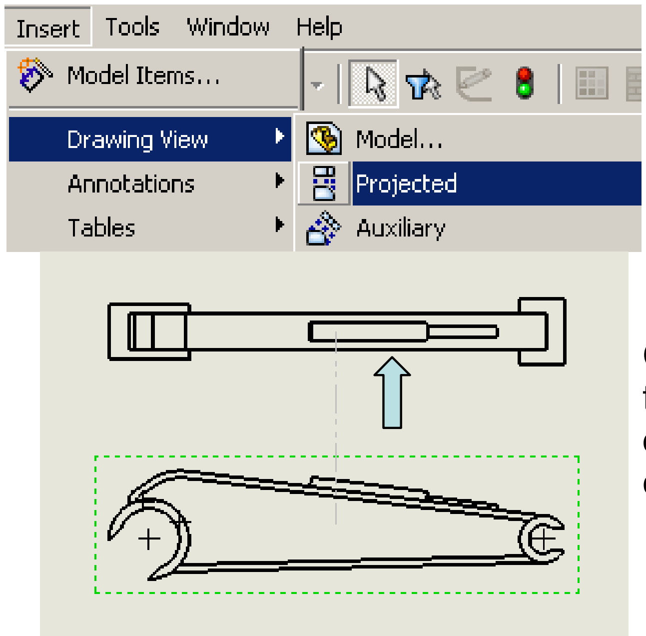
To move a view:-

- Move the mouse cursor near the view until you can see this symbol 
- Drag the view to the desired location
- (The related views will also be moved automatically so that the projection directions are kept unchanged)



To add a projection view:-

- Select "**Insert** / **Drawing View** / **Projected**" on the menu bar
- Click on the front view and then drag upward to create a projection view on the top



Click on the front view and then drag upward to create a projection view on top

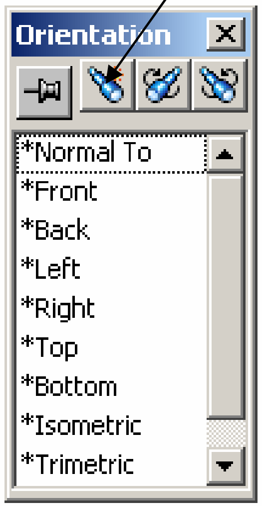
Tutorial 1A

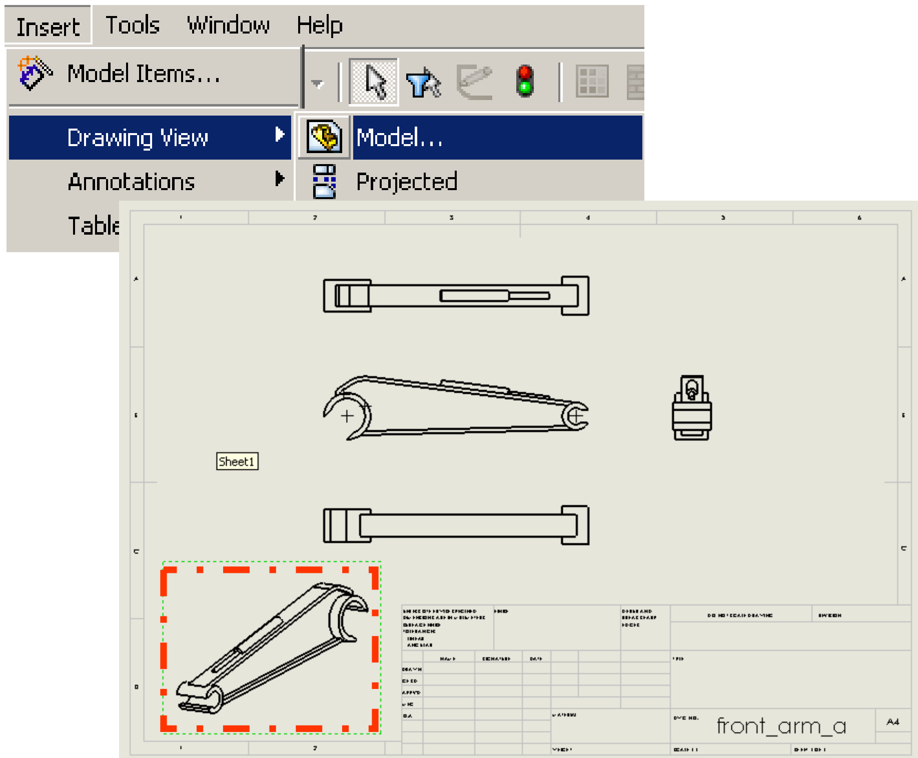
To add an isomeric view (user-defined):-

- Select "**window/front_arm_a.sldprt**" to view the 3D part.
- Rotate the part to the desired orientation
- Press "Space" key on the keyboard
- Click "**New View**" icon to save the current orientation
- Enter a View Name and click ok

- Select "**window/front_arm_a – SHEET1**" to view the 2D drawing
- Select "**Insert / Drawing View / Model…**" on the menu bar
- Click "Next" icon
- Select "Single View"
- Select the newly-created view (under more views…)
- Click an empty space on the drawing
- Click yes to accept true iso dimensions
- Click "Close Dialog" icon to complete

New View





Tutorial 1A

Select File/Save…

Enter "Front_arm_a.slddrw"

Click yes to save the 3D file too

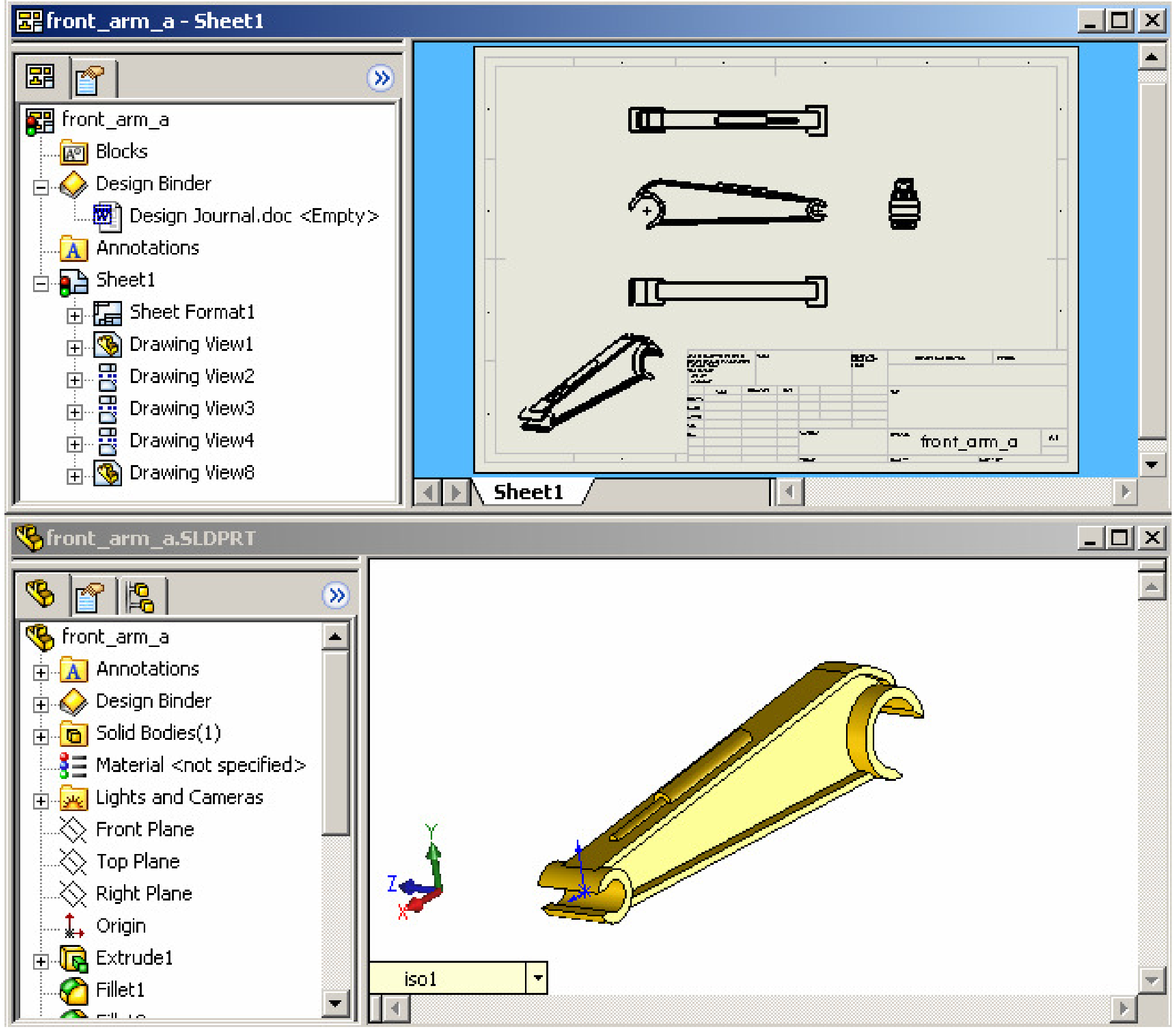
Now you have two files:-

- Front_arm_a.sldprt
- Front_arm_a.slddrw

- The drawing is created from the part file, and so if the part is changed, the drawing will change automatically.

- Now try to modify the 3D.
- Go back to the drawing to see whether the views have been updated correspondingly.

- Close both files without saving.

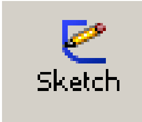


END of Tutorial 1A

Summary of Tut-1A

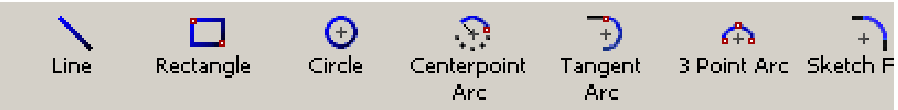
Build a Sketch:-

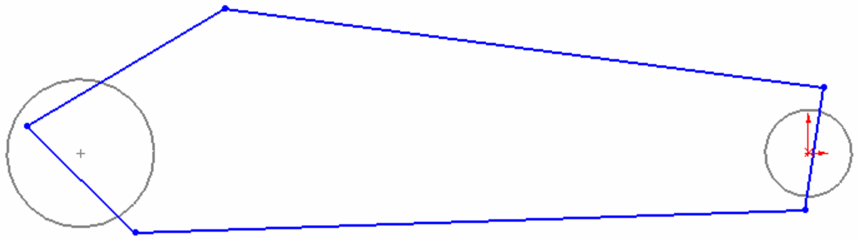
1. Click “**Sketch**” Icon



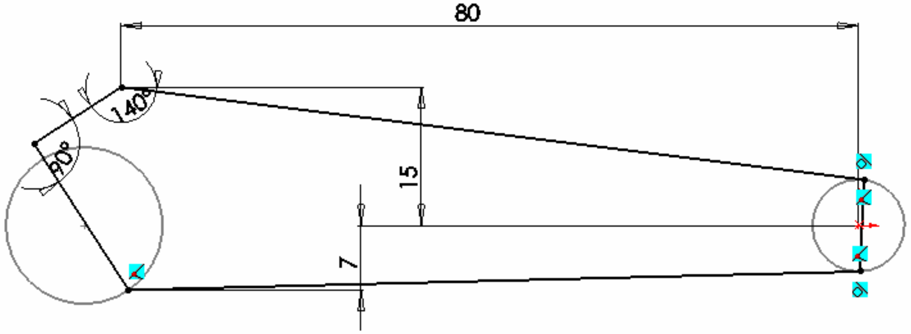
2. Select a **plane**

3. Draw a **profile** (with lines, curves and/or centerlines)





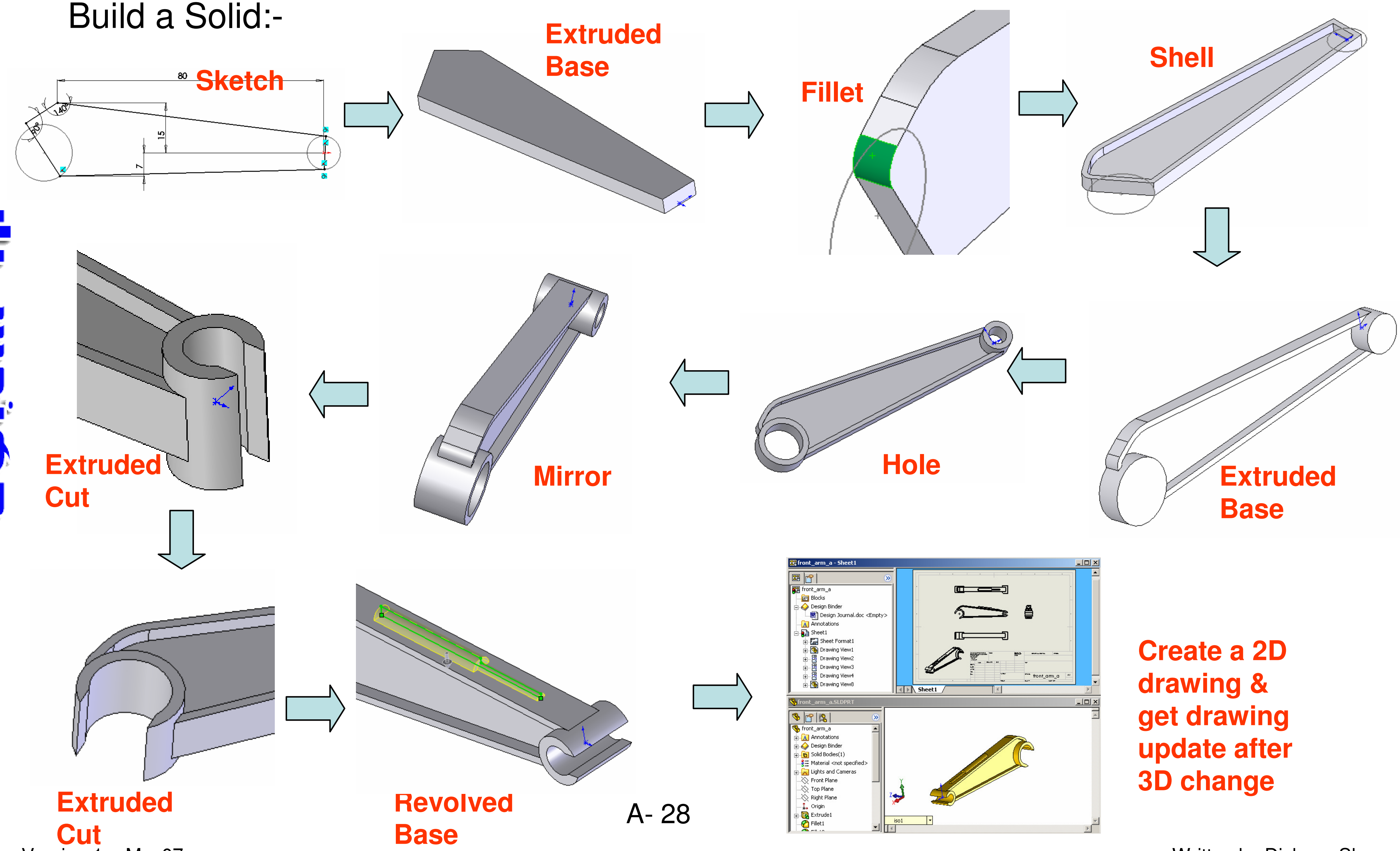
4. Add **geometrical constraints (relations)**
5. Add **dimensional constraints & modify the values**



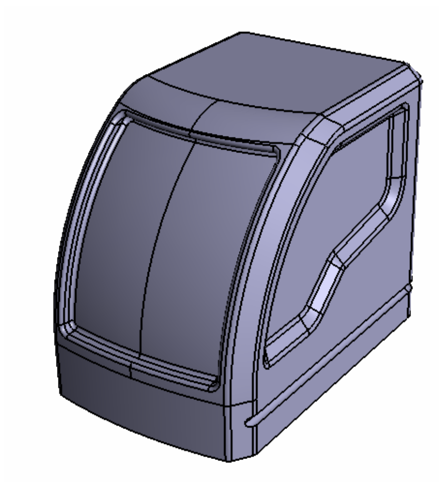
6. Click “**Exit Sketch**” icon

Summary of Tut-1A

Build a Solid:-



Tutorial 1B

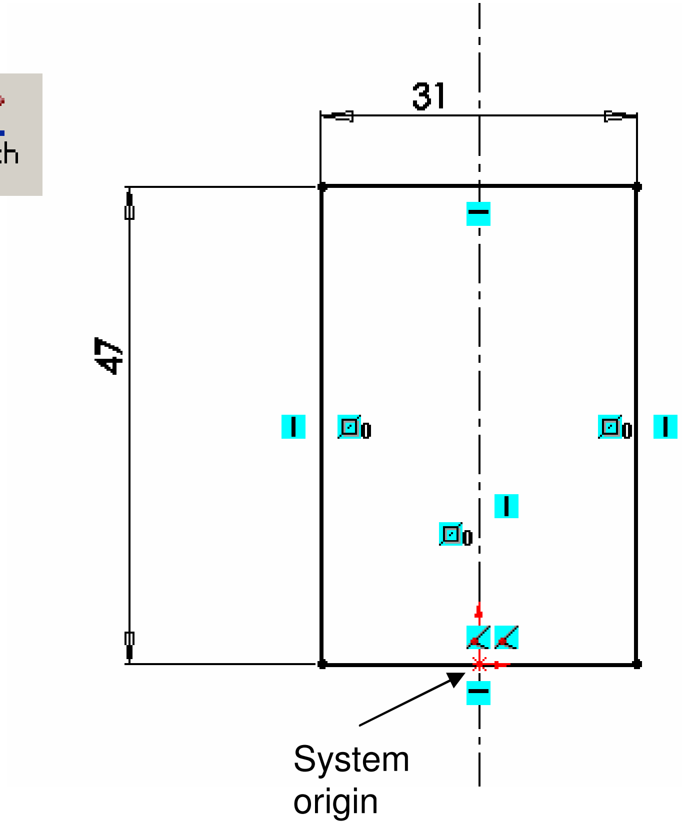


Continuing what we learnt in Tutorial 1A, we are going to build the cabinet by the solid-modeling technique plus some surface modeling technique…

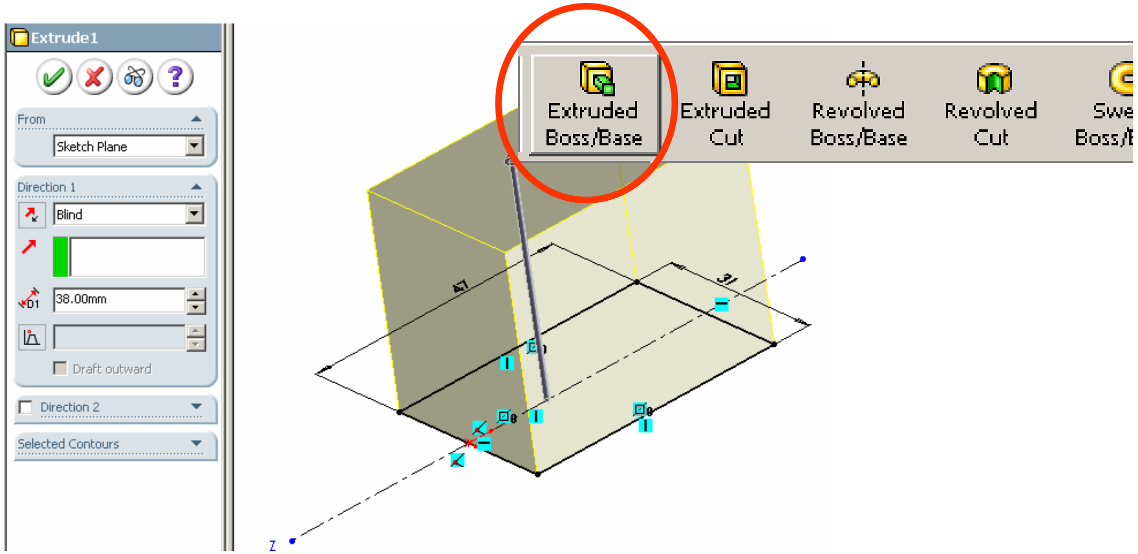
- Enter Solidworks 2006
- Select **"File/New…"** on the menu
- Click "**Part**" and then OK

To build a sketch:-

- Click "**Sketch**" icon and select **Front Plane.**
- Draw a rectangle (47mm x 31mm) as shown; one edge is aligned on x-axis
- Draw a centerline along y-axis
- Multi-select two vertical sides and then the centerline by pressing and holding "CTRL" key
- Select "Symmetric" relation
- Exit to complete.



Tutorial 1B

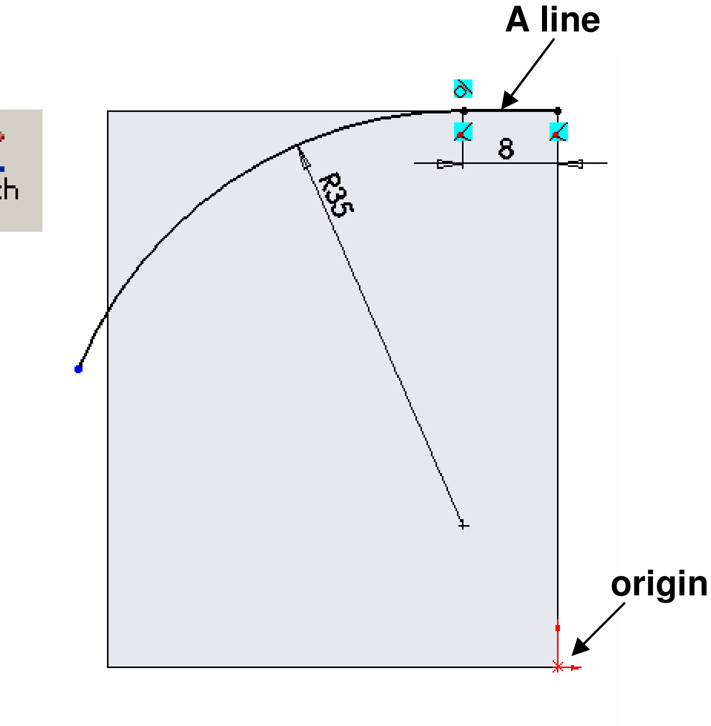


To build a solid:-

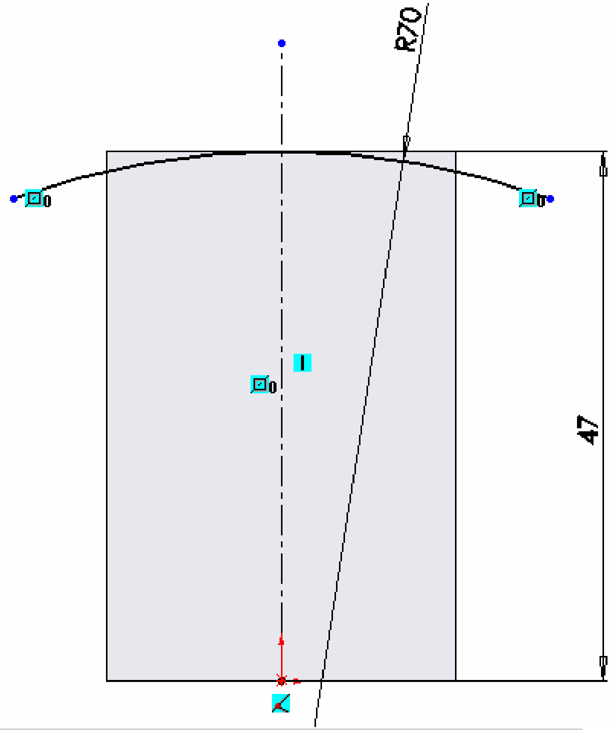
- Click “**Extruded Boss/Base**” icon.
- Enter 38mm as D1 (First Limit).
- Click ok to complete.

To build 2nd sketch:-

- Click “**Sketch**” icon and select **Right Plane.**
- Draw an arc R35 & a line as shown; They are tangent to each other; The line is aligned onto the solid edge and one endpoint touches x-axis.
- Exit to complete.
- Click the open area near the solid to deselect “Sketch2”

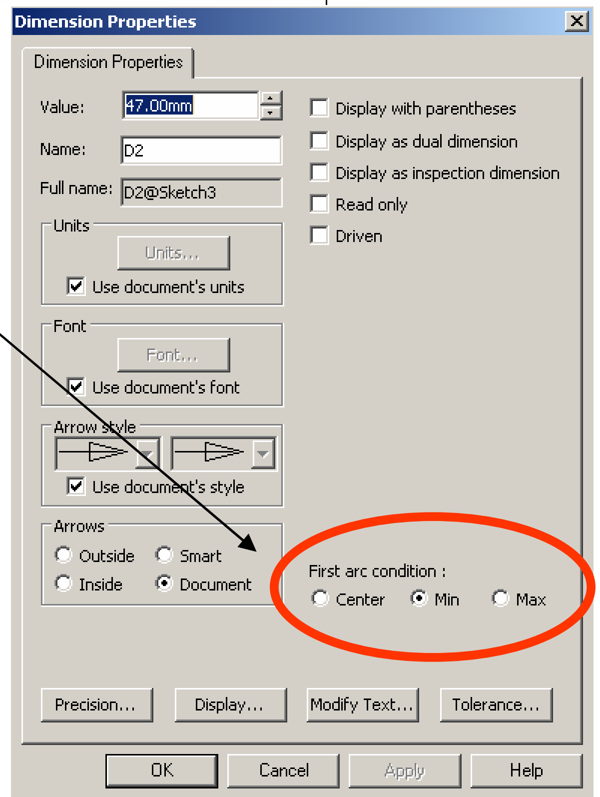


Tutorial 1B



To build 3rd sketch:-

- Click “**Sketch**” icon and select **Front Plane.**
- Draw an arc <u>R70</u> as shown; The endpoints should be symmetric about the y-axis (while pressing “ctrl” on keyboard, select both endpoints then the centerline, and finally select “symmetric”)
- (Remark: To define the distance (47) between the extremum point of the arc and the origin, right-click the dimension to access its properties, and then select “Min” as First Arc Condition”)

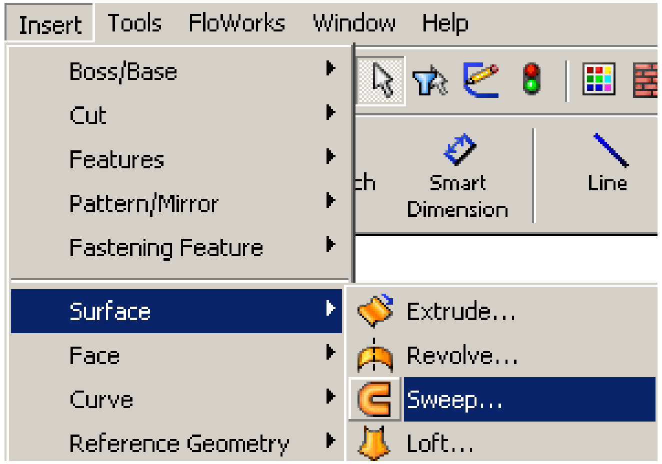


- Exit to complete.
- Click the open area near the solid to deselect “Sketch3”

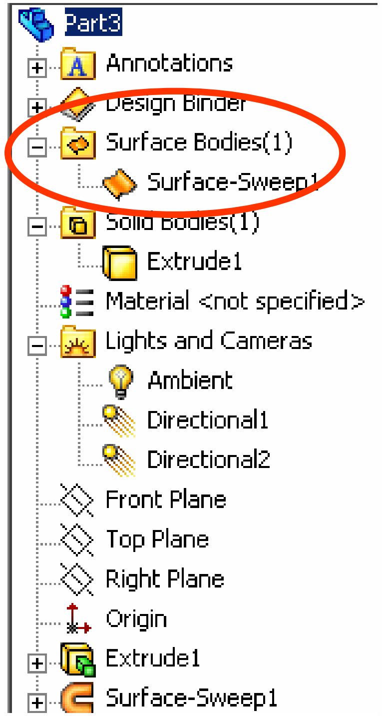
Tutorial 1B

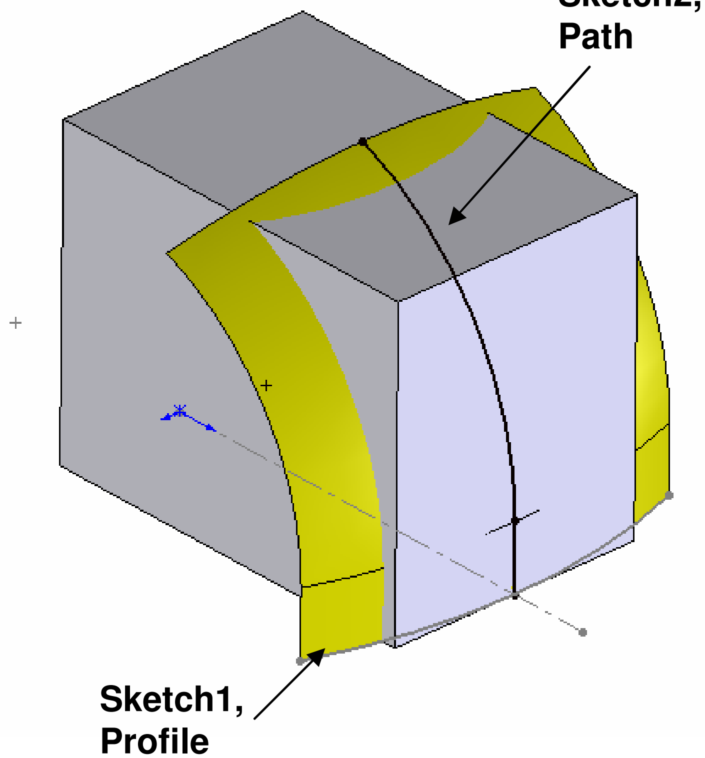
To build a SURFACE:-

- Select "**Insert** /**Surface**/ **Sweep**" on the menu bar
- Select "Sketch3" as **Profile**
- Select "Sketch2" as **Path**
- Press "Enter" key to complete



On the tree, this surface is stored in "Surface Bodies", so it will not be mixed with solids.

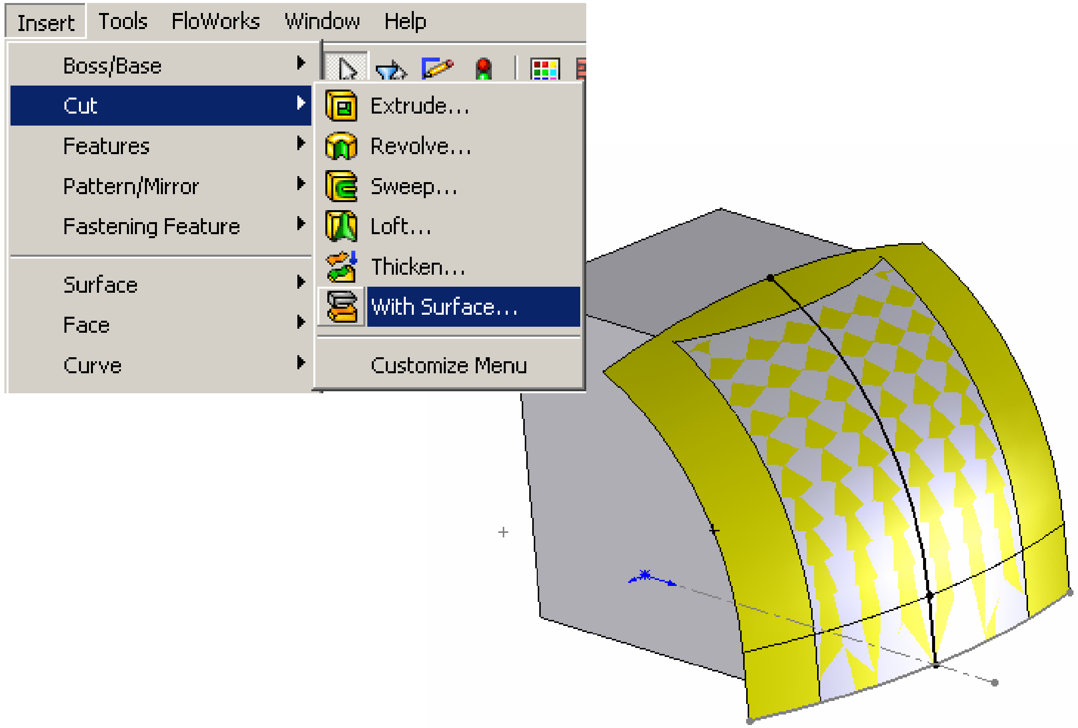




Tutorial 1B

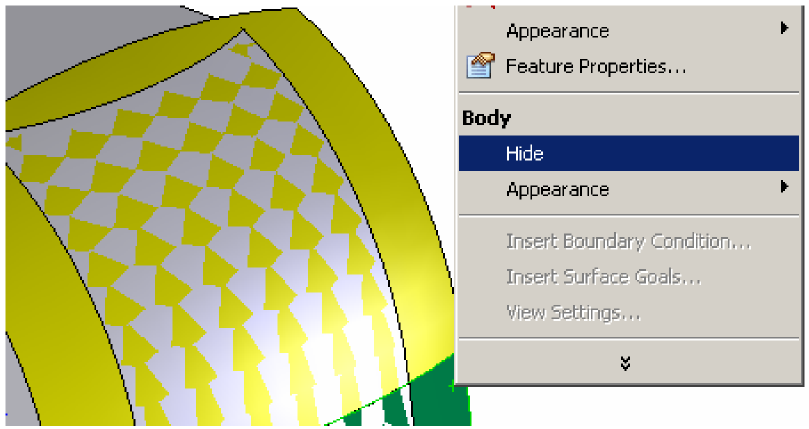
To cut the solid with this SURFACE:-

- Select “**Insert /Cut /With Surface**” on the menu bar
- Select the Surface “Sweep1”
- (Arrow is pointing to the material which will be removed)
- Click on the arrow if it is pointing inwards.
- Press “Enter” key to complete

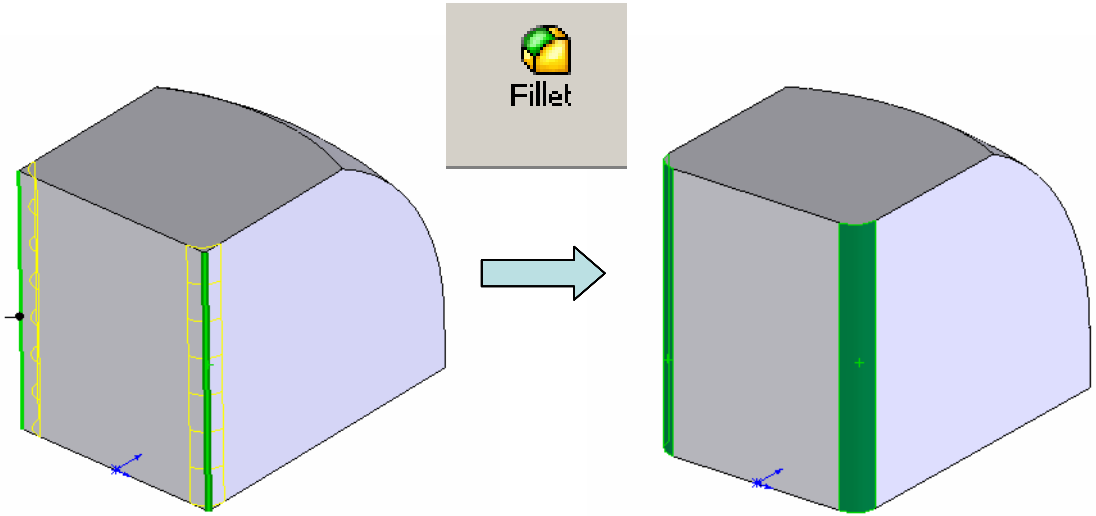


To hide the surface & its curves:-

- Right-click the surface and select “hide”
- Hide “Sketch2” & “Sketch3” too.

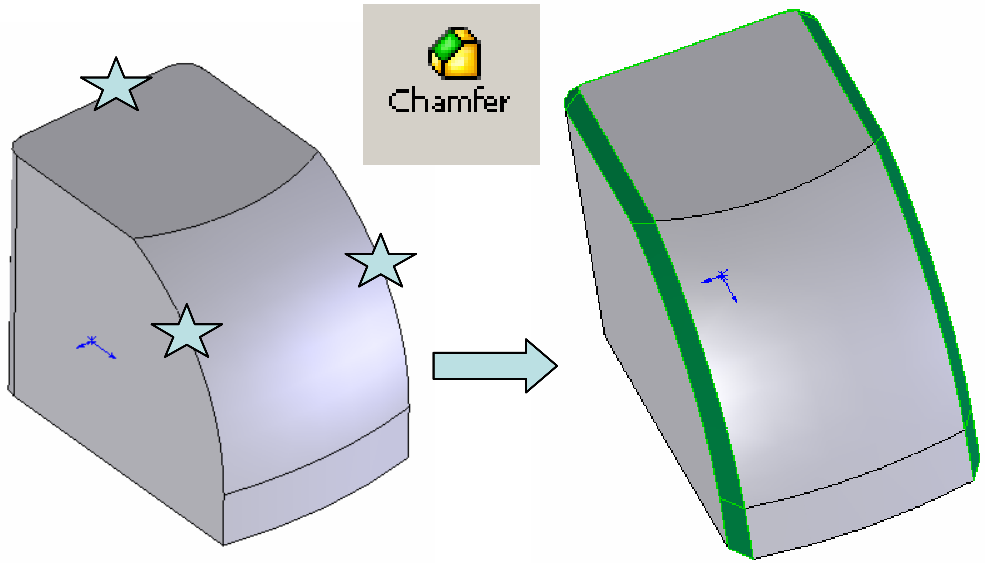


Tutorial 1B



Add a "Fillet" <u>R3mm</u> as shown:-

- Select the two vertical edges
- Select "Constant Radius" as type
- Enter <u>3mm</u> as radius

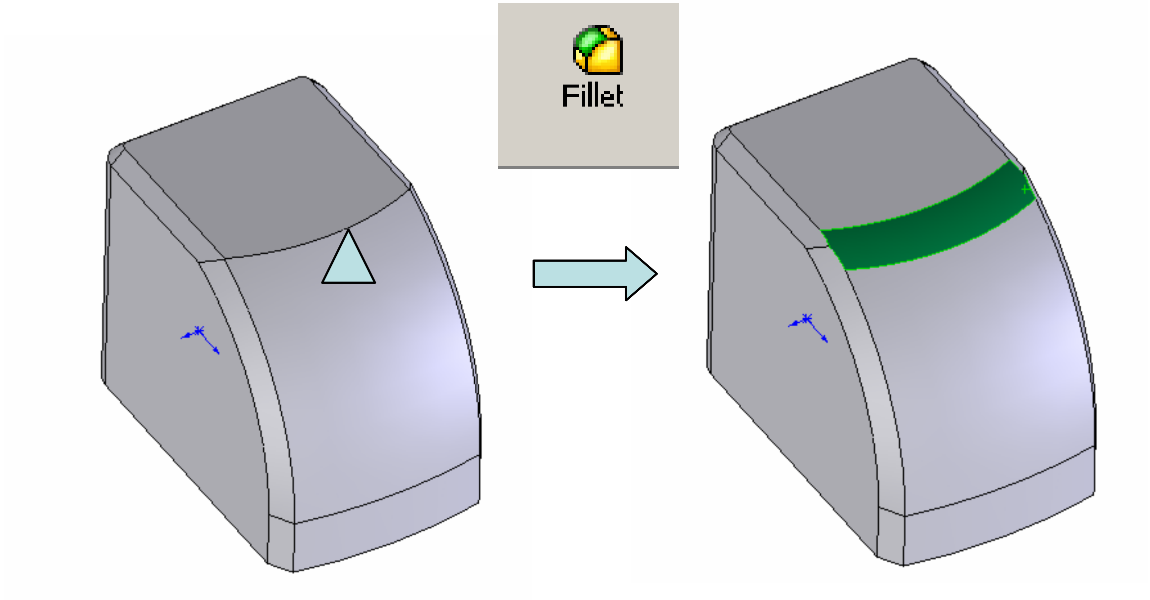


Add a "Chamfer" onto the edges as shown:-

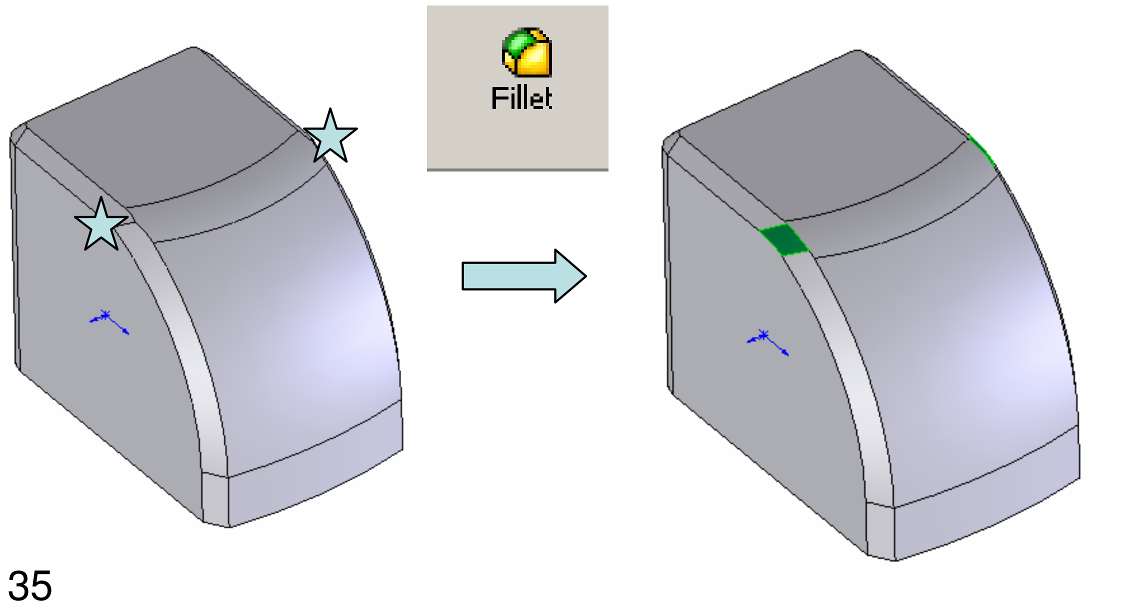
- Select "Angle Direction" as mode.
- Enter <u>2mm</u> as D (Distance)
- Enter <u>45deg</u> as A (Angle)
- Select "Tangency Propagation"
- Click 3 edges at ☆

Tutorial 1B

Add a “**Fillet**” <u>R10mm</u> on the edge △



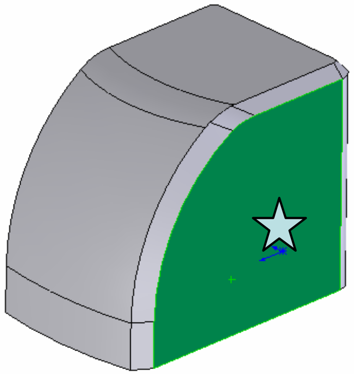
Again, add a “**Fillet**” <u>R10mm</u> on the edges on both sides ☆

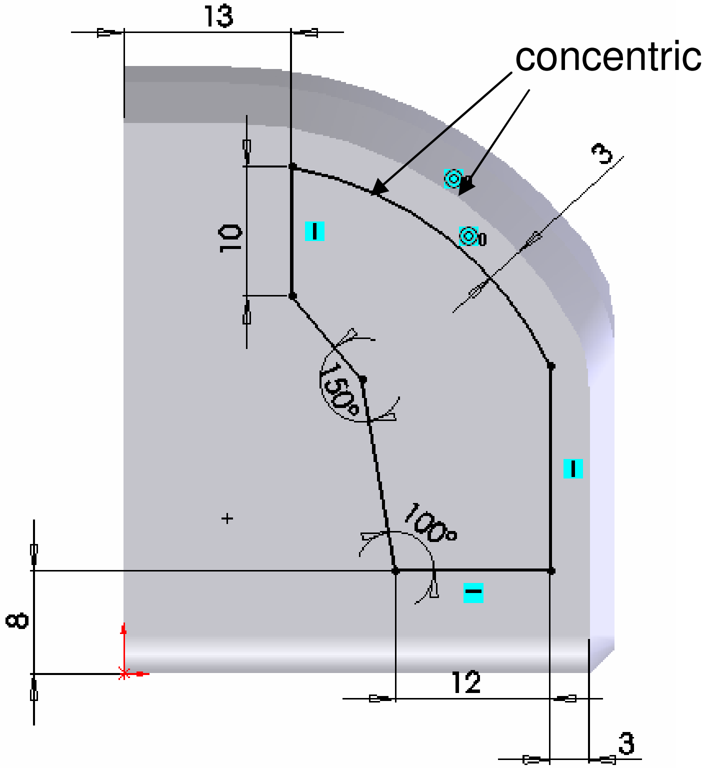


Tutorial 1B

To build a sketch:-

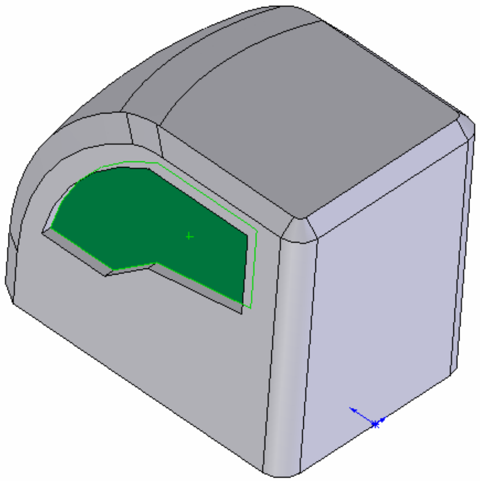
- Click "**Sketch**" and select **Planar Face**
- Draw a profile as shown; The profile should be fully-constrained.
- Exit to complete.



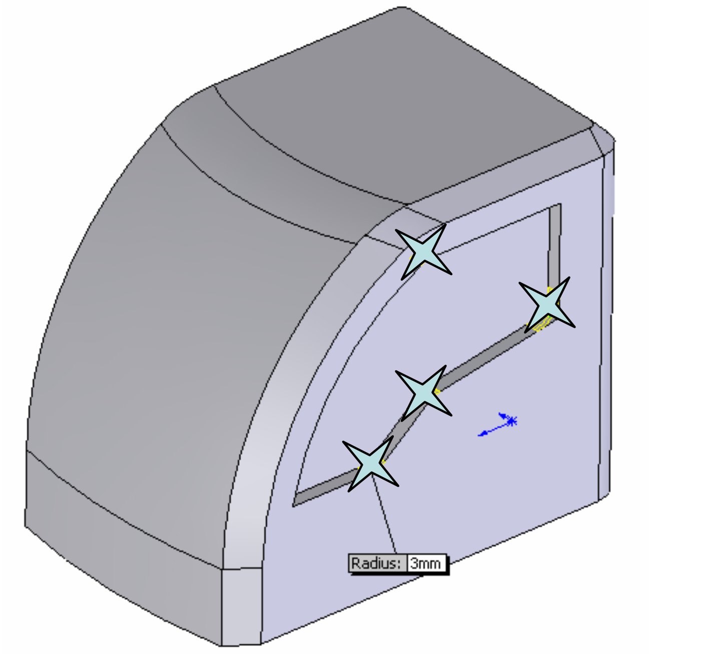


To make an extruded cut:-

- Click "**Extruded Cut**" icon.
- Enter <u>1.5mm</u> as D1.
- Press Enter key to complete.



Tutorial 1B

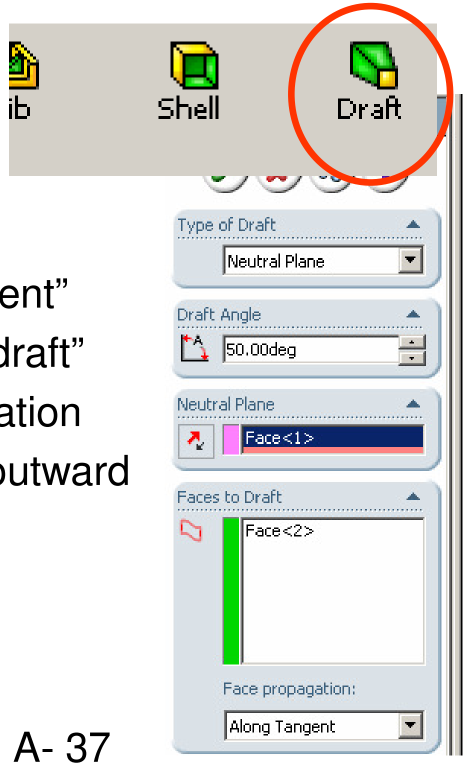


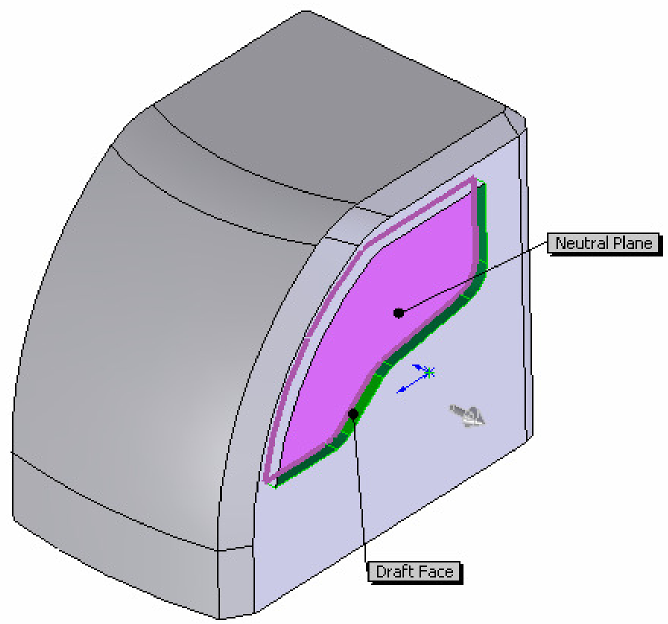
Add "Fillet" <u>R3mm</u> on the four edges of the Pocket:-

- Add fillets at ✧ positions.

To add a Draft onto the side faces of the cut:-

- Click "**Draft**" icon
- Select "Neutral Plane" as Type
- Enter <u>50deg</u> as Draft Angle
- Select the bottom face as "Neutral Element"
- Select the lower side face as "Faces to draft"
- Select "Along Tangent" as Face Propagation
- Click the arrow once if it is not pointing outward
- Press Enter key to complete

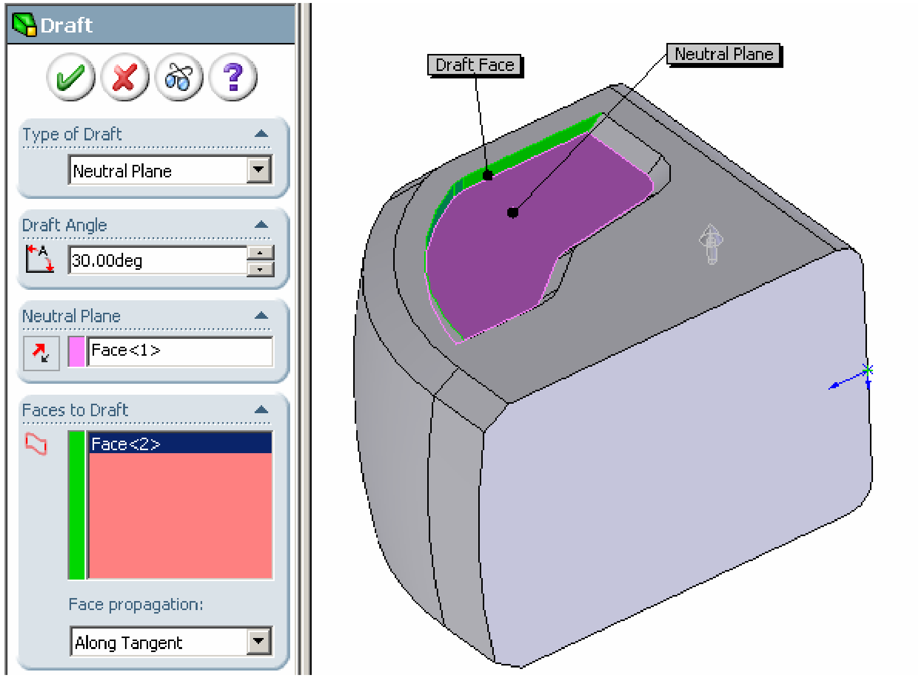




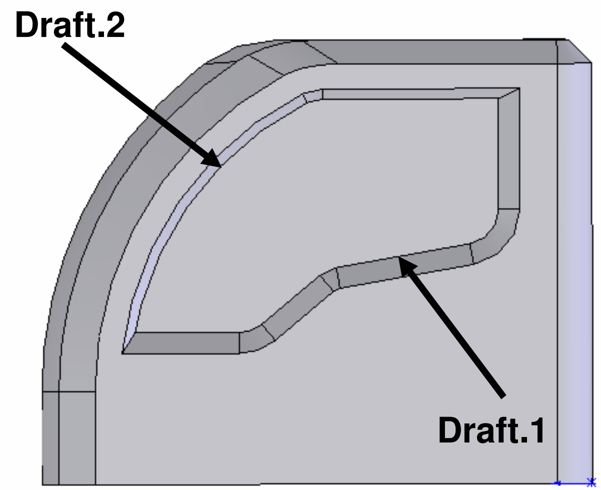
Tutorial 1B

To add another Draft onto the side faces of the cut:-

- Click “**Draft**” icon.
- Select “Neutral Plane” as Type
- Enter 30deg as Draft Angle
- Select the bottom face as “Neutral Element”
- Select the upper side face as “Faces to draft”
- Select “Along Tangent” as Face Propagation
- Click the arrow once if it is not pointing outward
- Press Enter key to complete

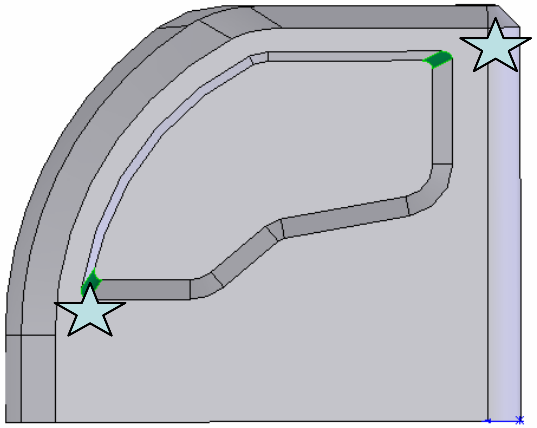


Now you should have two drafts on the cut.



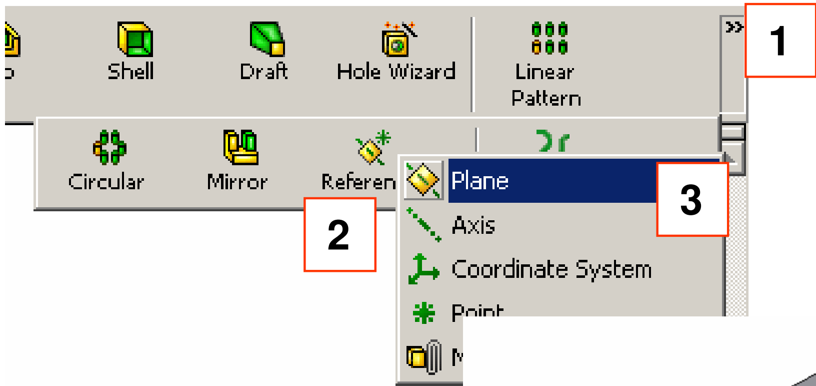
Tutorial 1B

Add "Fillet" R1mm on the remaining two edges of the Cut at ☆ positions.

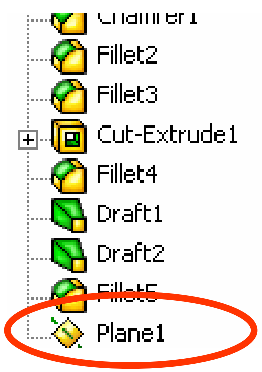


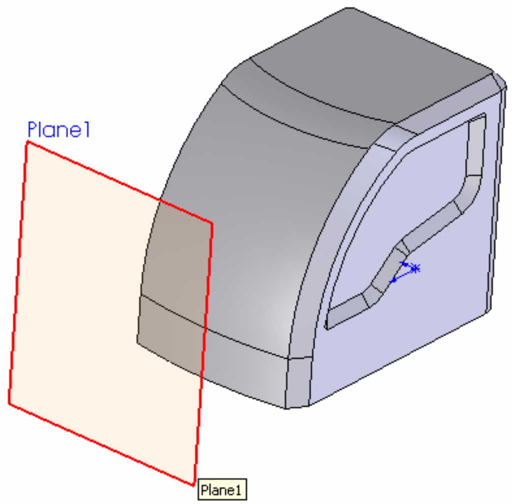
To create an offset reference plane:-

- Click >> on the rightmost to access more icons
- Click "**Reference**" icon, then "**Plane**"
- Select **Top Plane**
- Enter 70mm as D (Offset Value)
- Press Enter key to complete



Now a plane is created in front of the solid, which is stored on the tree.

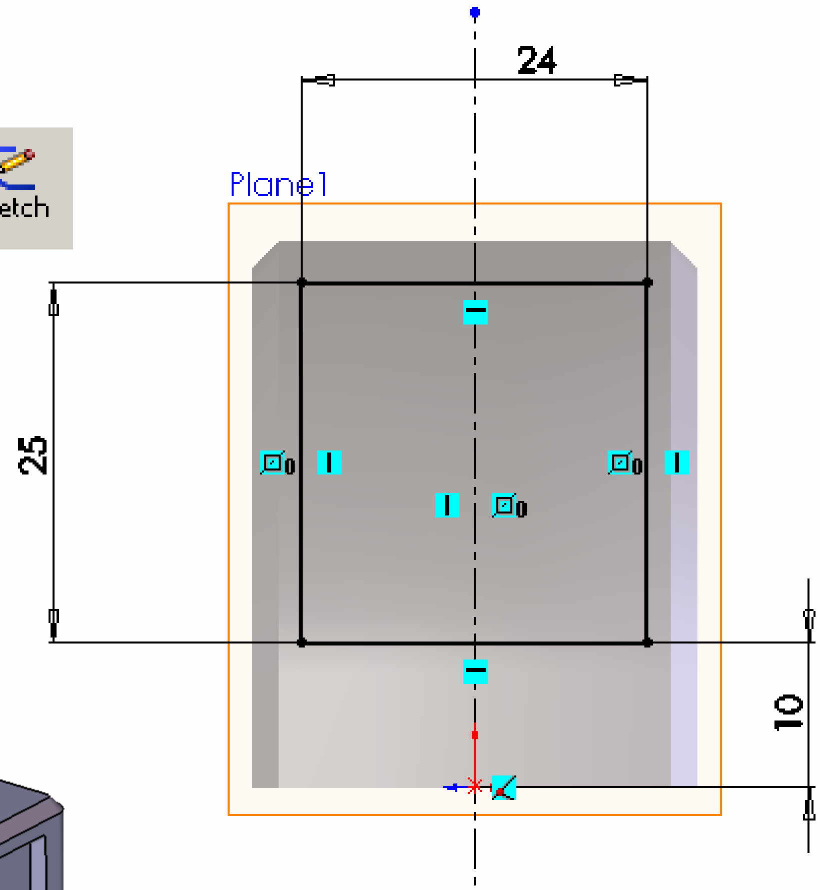


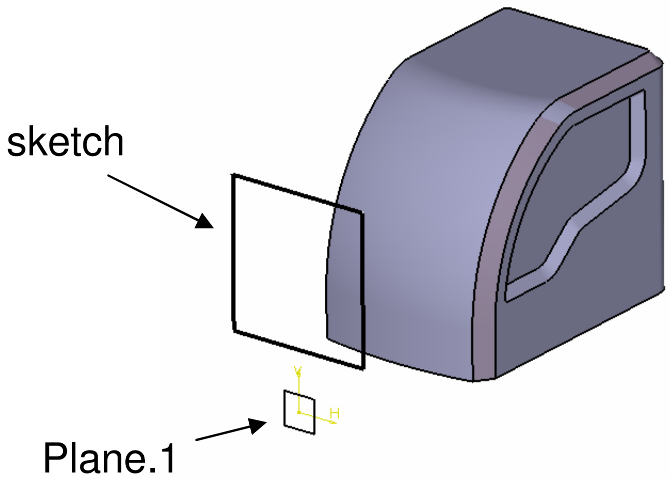


Tutorial 1B

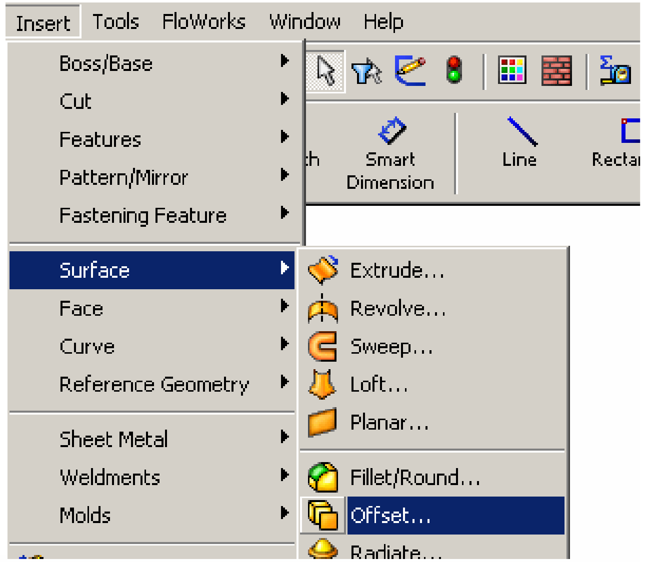
To build a sketch on the offset plane:-

- Click “**Sketch**” icon and select “Plane1”.
- Draw a rectangle (24x25) and position it as shown
- Draw a centerline along y-axis
- Multi-select two vertical sides and then the centerline by pressing and holding “CTRL” key
- Select “Symmetric” relation
- Exit to complete.





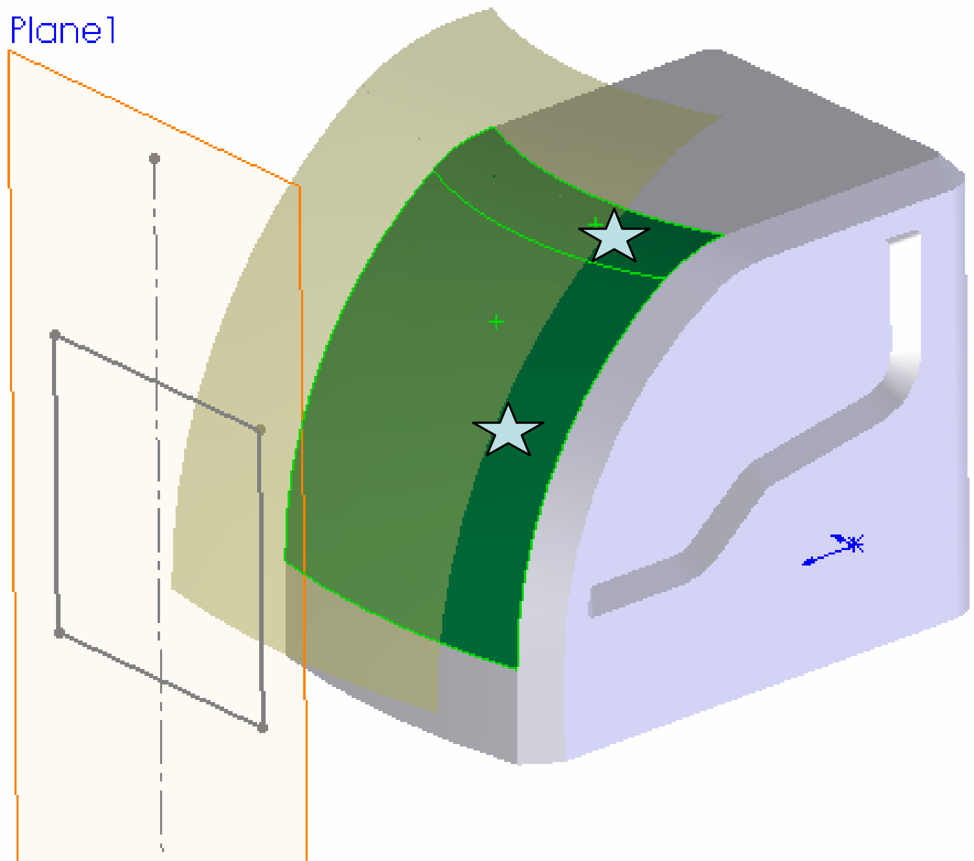
Tutorial 1B

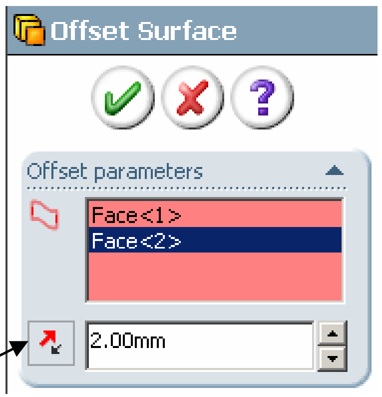


To create an offset surface from the solid:-

- Select "Insert / Surface / Offset…" on the menu bar
- Select the two front faces ☆
- Enter 2mm as Offset Value
- Click "Flip Offset Direction" icon
- Press Enter Key to complete

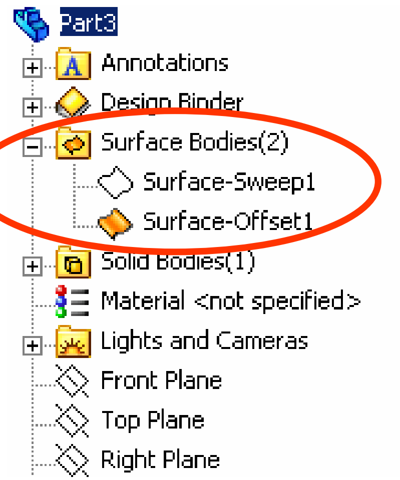
(We can now have an offset surface inside the solid)





Flip Offset Direction

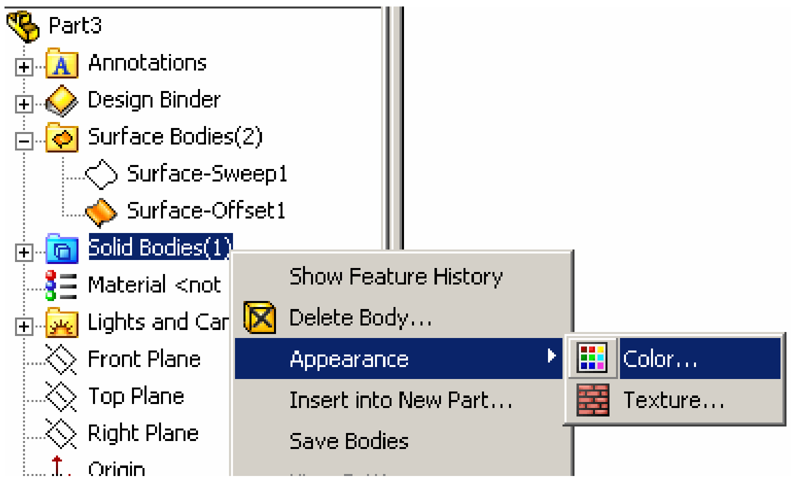
Tutorial 1B



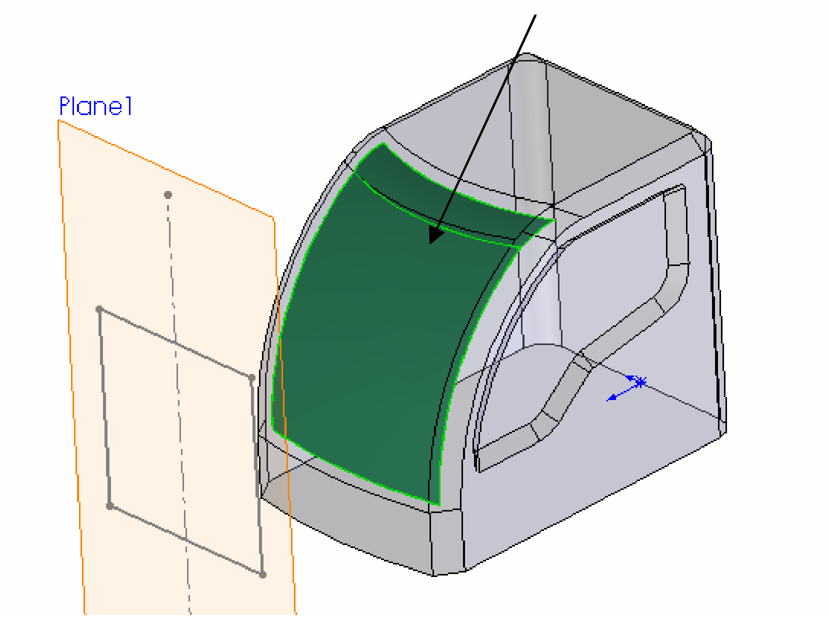
- The offset surface is created ***inside the solid***, and it is stored in "Surface Bodies" on the tree

To visualize the offset surface:-

- Right-Click "Solid Bodies" on the tree
- Select "**Appearance/Color**"
- Change Transparency to 0.7
- Press Enter Key to complete



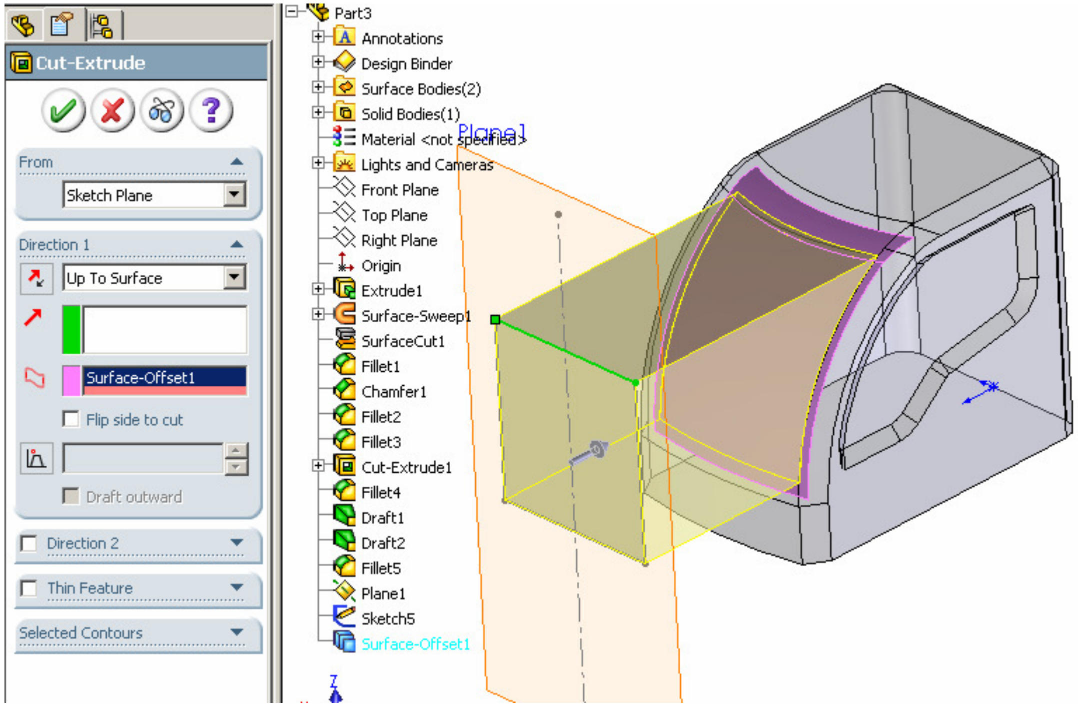
The offset surface will appear when the solid is semi-transparent



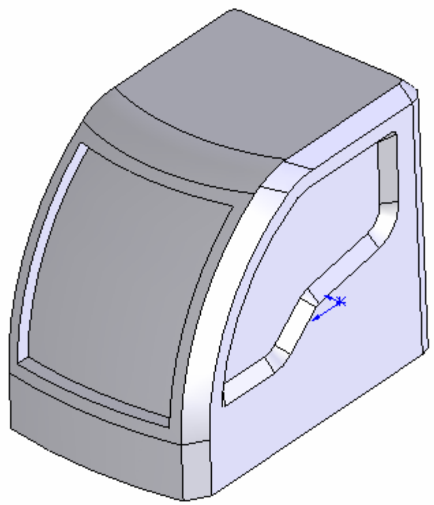
Tutorial 1B

To make a Cut (Up to Surface):-

- Click “**Extruded Cut**” icon.
- Select “Sketch5” (sketch on offset reference plane)
- Change “Blind” to “**Up to Surface**” for Direction1
- Select “Surface-Offset1” on the tree as Limit Surface
- Press Enter key to complete

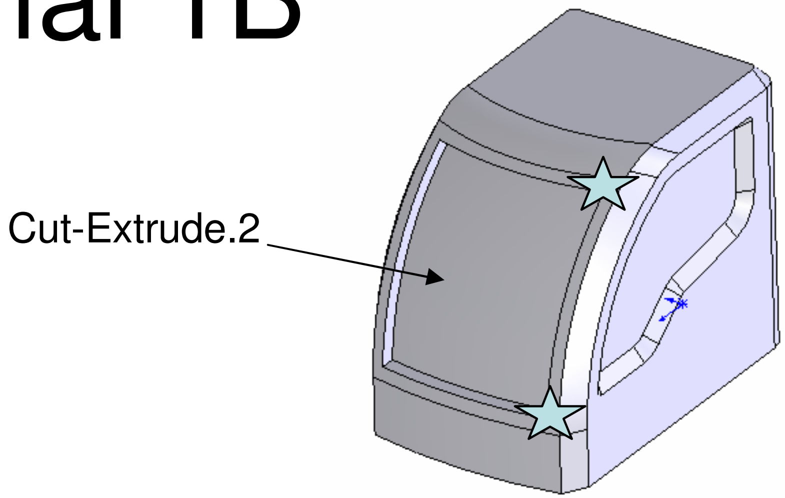


- Hide “Surface- Offset1” & “Plane1”
- Change Transparency Value of “Solid Bodies” back to 0



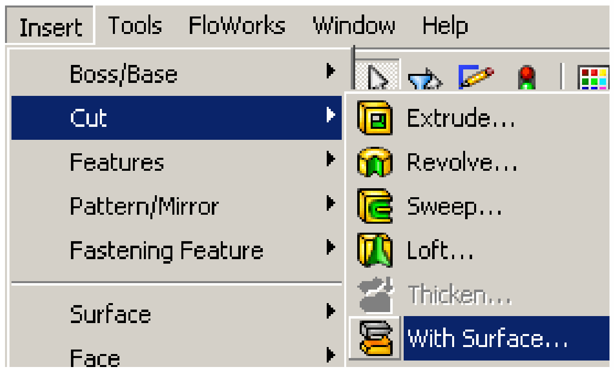
Tutorial 1B

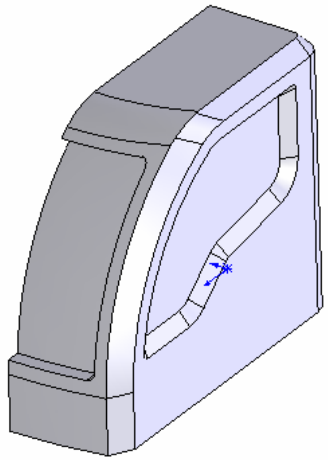
Add "Fillet" R1mm at 2 corners of "Cut-Extrude.2" at ☆ positions



To split the solid into a half:-

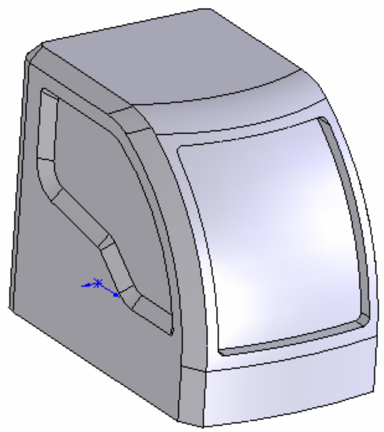
- Select "**Insert/ Cut /With Surface**" on the menu bar
- Select "Right Plane"
- (Arrow should be pointing the side without a cut; If not, click on it to change the direction)
- Press Enter key to complete





To create another half by mirroring:-

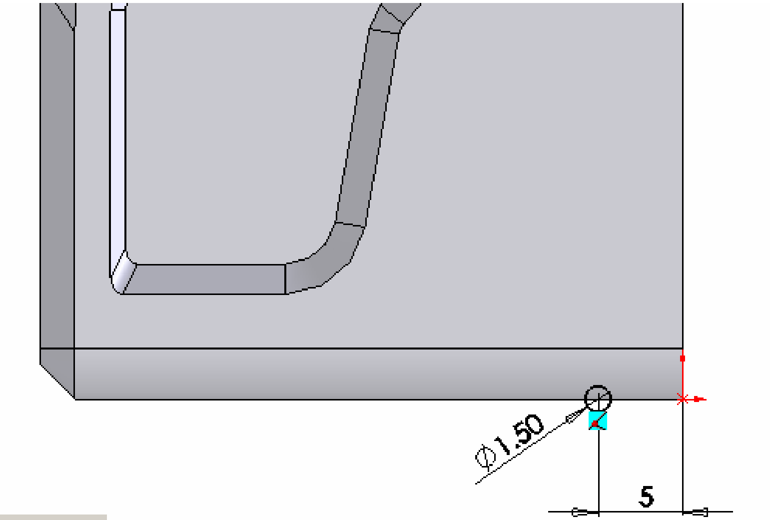
- Click "**Mirror**" icon.
- Select the cut face as Mirror Plane
- Click on "Bodies to mirror"
- Select the solid
- Press Enter key to complete

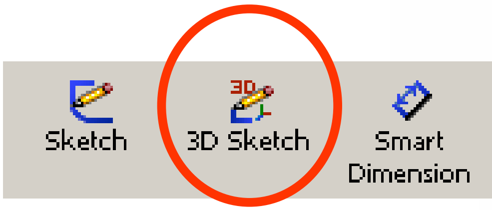


Tutorial 1B

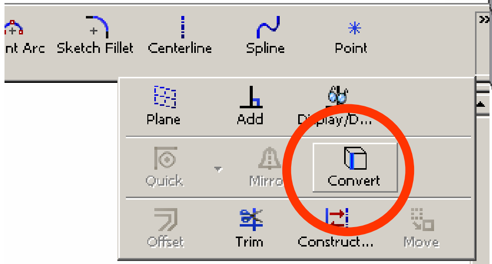
To remove material along a guide:-

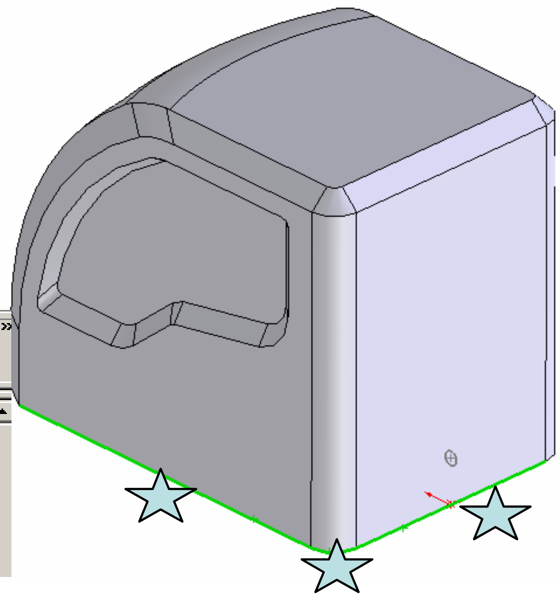
- Click "**Sketch**" icon and select **Right Plane**
- Draw a circle D1.5mm; 5mm above the base, & circle center is aligned on the axis
- Exit to complete





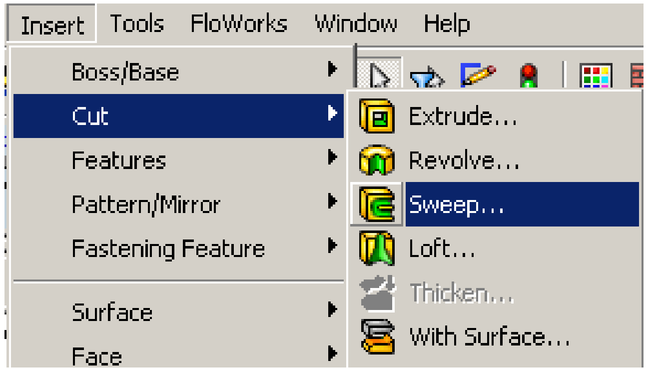
- Click "**3D Sketch**" icon
- Multi-select the three edges ☆
- Click "**Convert**" icon (which will appear if clicking >> on the rightmost)
- Click "3D Sketch" icon to complete





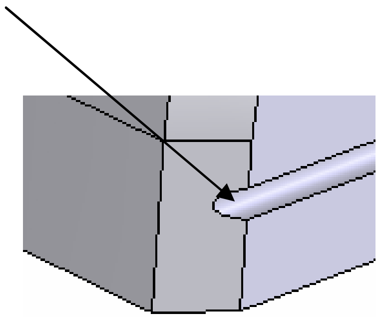
Tutorial 1B

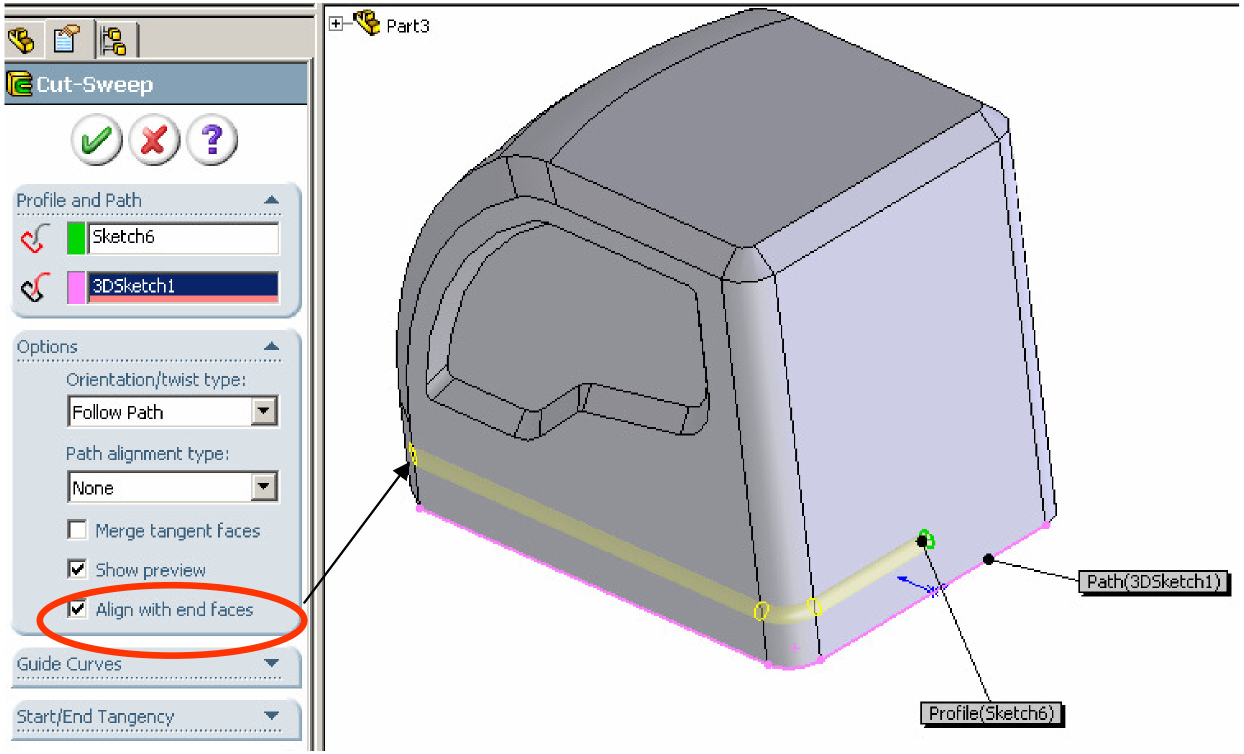
- Select "**Insert** / **Cut** / **Sweep…**" on the menu bar
- Select "Sketch6" as **Profile**
- Select "3DSketch1" as **Path**
- ("**aligned with end faces**" should be selected by default, under "Options")



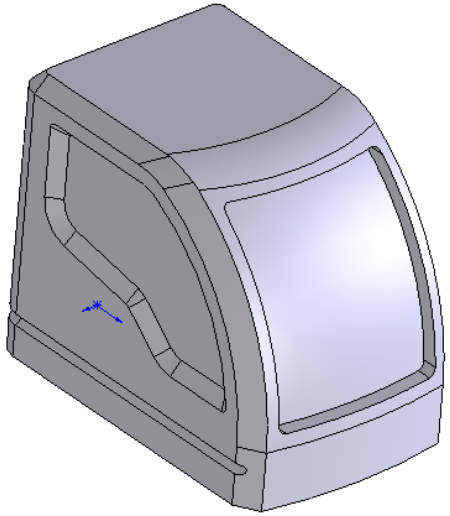
(Remark: This is a limitation of solidworks 2006 that the cut must always start from "Profile". Also, sweeping only happens in one direction)

(Remark: When "aligned with end faces" is selected, the cut will go through the solid, even though the path is not long enough)



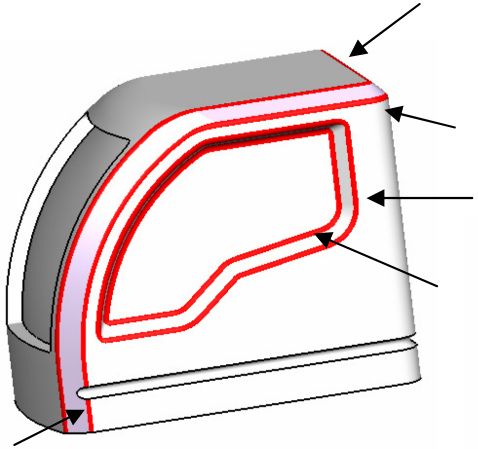


Tutorial 1B



To create another half by mirroring:-

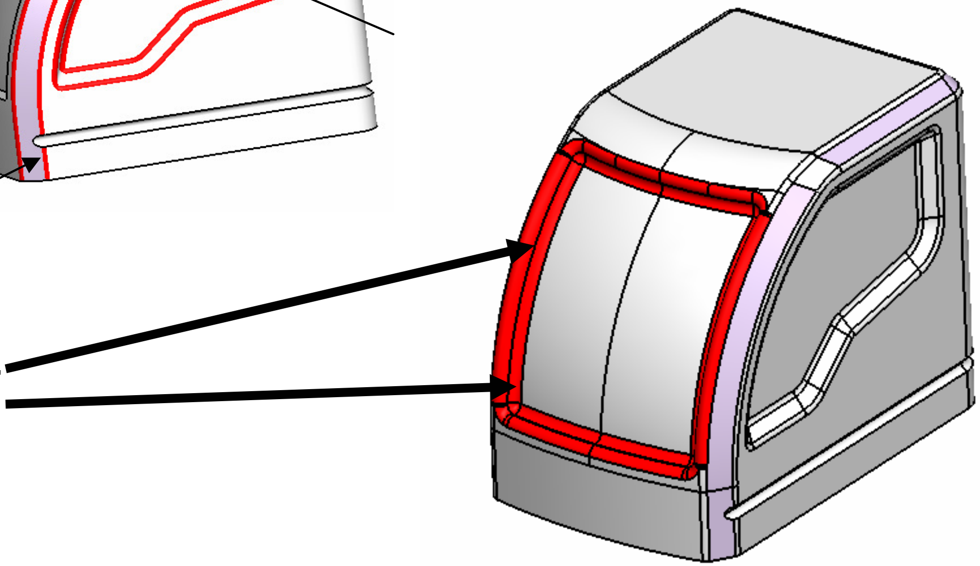
- Click “**Mirror**” icon.
- Select “Right Plane” as Mirror Plane
- Click on “**Features to mirror**”
- Select “Cut-Sweep1”
- Press Enter key to complete



Add “Fillet” <u>R1.0mm</u> on the edges on both sides, except those of the front cut.

Add “Fillet” <u>R1.0mm</u> on the edges of the front cut.

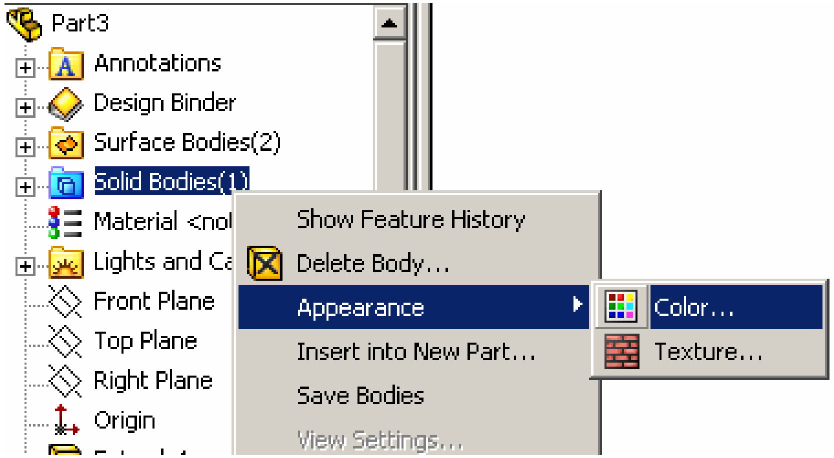
- Sometimes, we need to build fillets separately when the sharp edges are too close to each other. We need to build a fillet on one edge first, and then build another one on top of it.

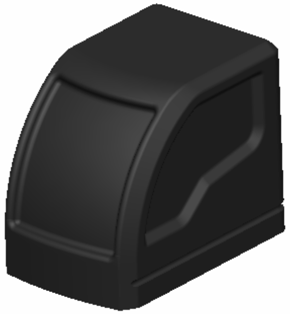


Tutorial 1B

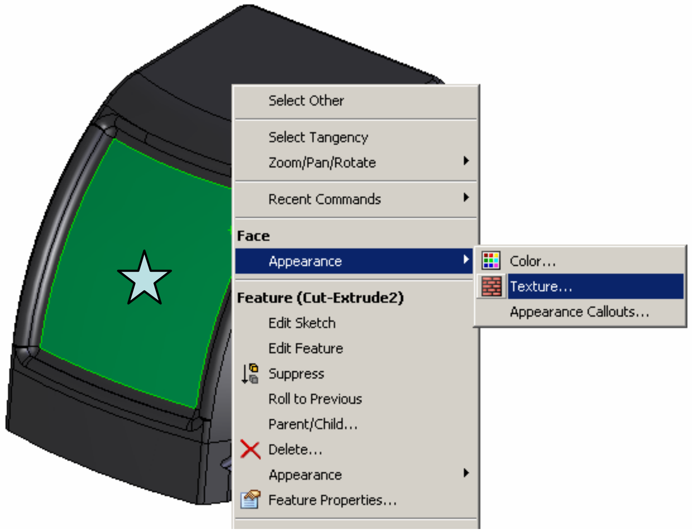
To change color properties of the model:-

- Right-Click “Solid Bodies” on the tree
- Select “**Appearance /Color…**”
- Change color as Red 60, Green 60, Blue 60
- Change “Ambience” to 1.0
- Change “Diffusion” to 1.0
- Change “Specularity” to 1.0
- Change “Shininess” to 0.3
- Press Enter key to complete





- Right-Click on the front face
- Select “**Appearance /Texture…**”
- Select “**Metal /Polish / Chrome1**”
- Press Enter key to complete

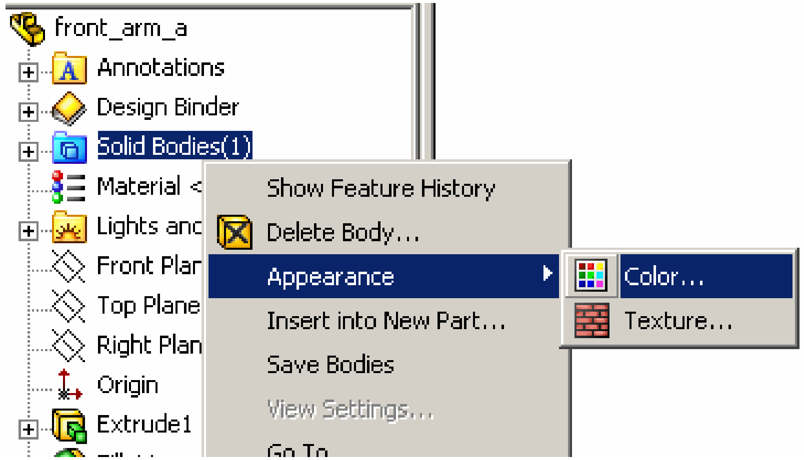


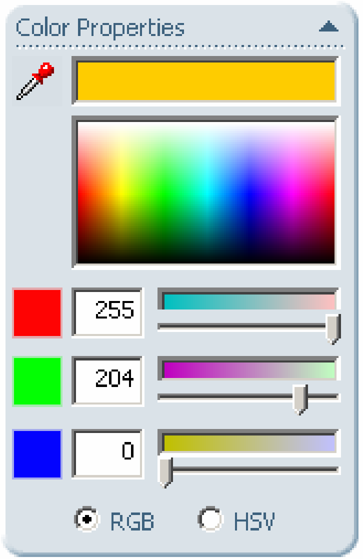
Save the file as “Cabinet_a.sldprt” in your project folder and then close it.

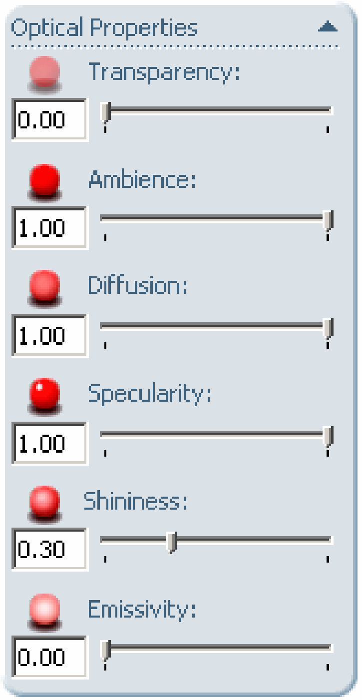
Tutorial 1B

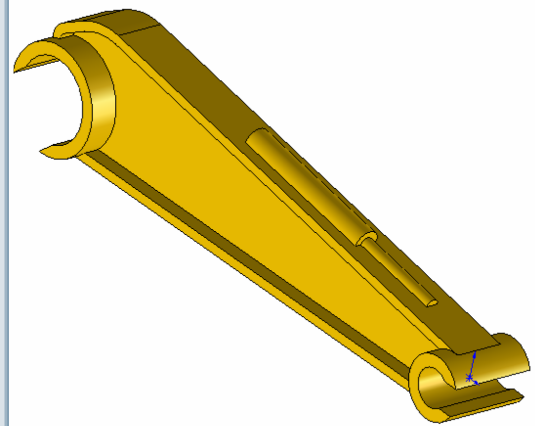
To change color properties of “front_arm”:-

- File/Open… “front_arm_a.sldprt”
- Right-Click “Solid Bodies” on the tree
- Select “**Appearance /Color…**”
- Change color as Red 255, Green 204, Blue 0
- Change “Ambience” to 1.0
- Change “Diffusion” to 1.0
- Change “Specularity” to 1.0
- Change “Shininess” to 0.3
- Press Enter key to complete









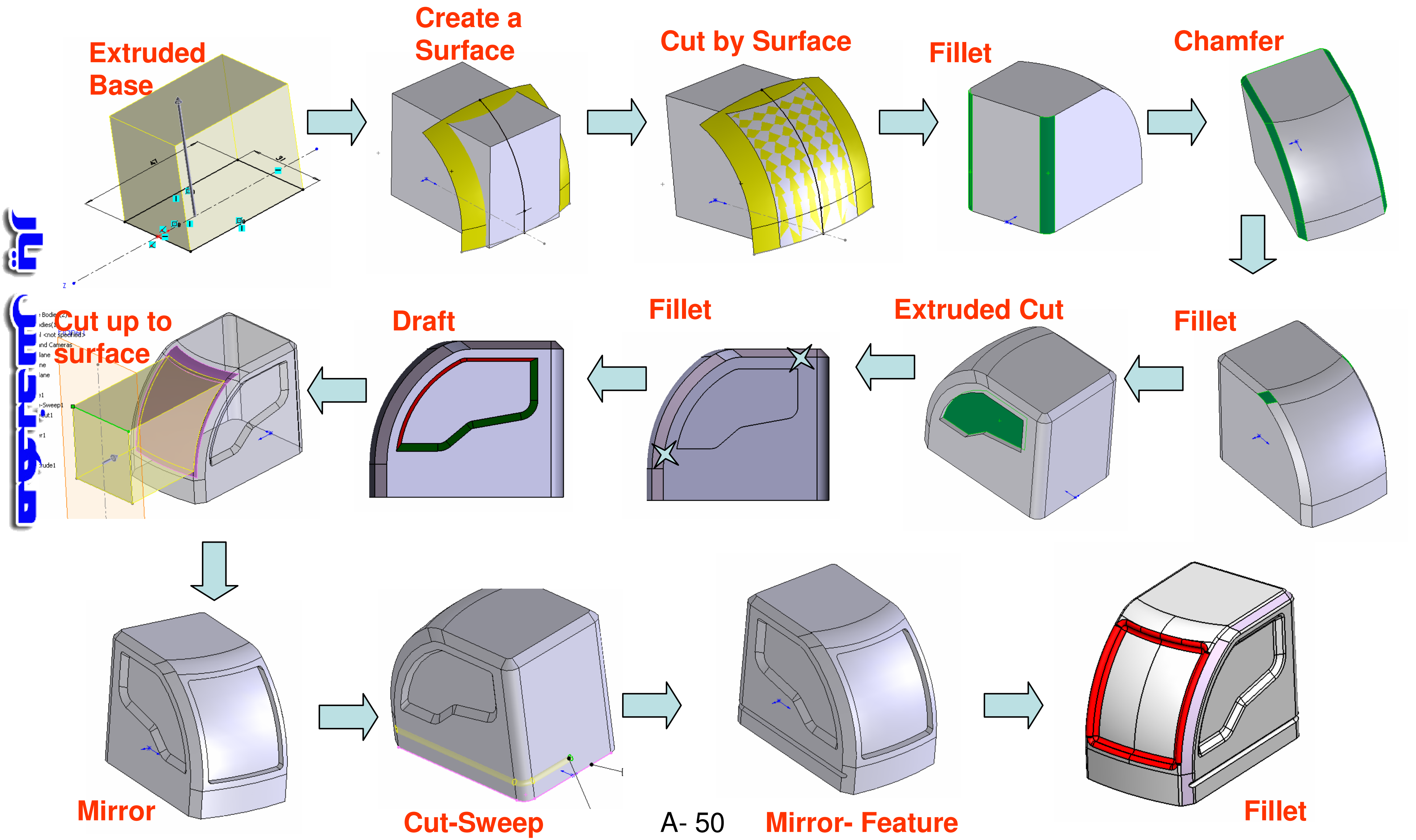
Save and Close the file

END of Tutorial 1B

Written by Dickson Sham

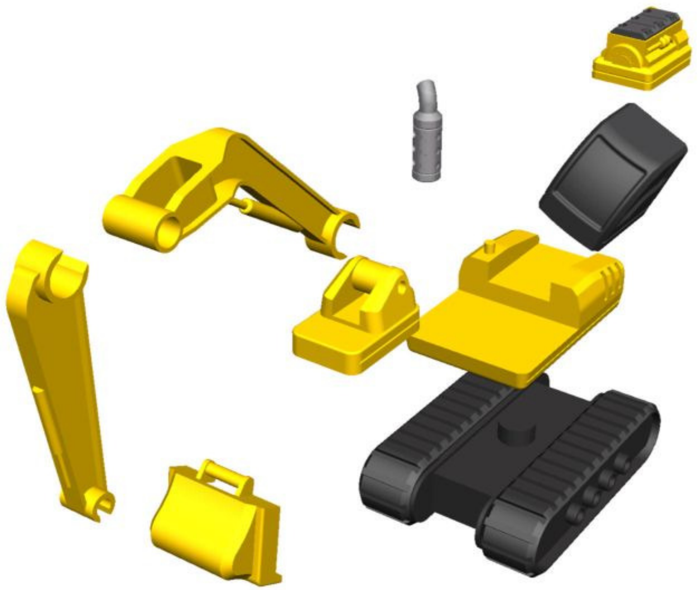
Summary of Tut-1B

Build a Solid:-



Tutorial 1C

In Tutorial 1A &1B, we have learnt some basic modeling technique to create parts. Now it's time to assemble them together…



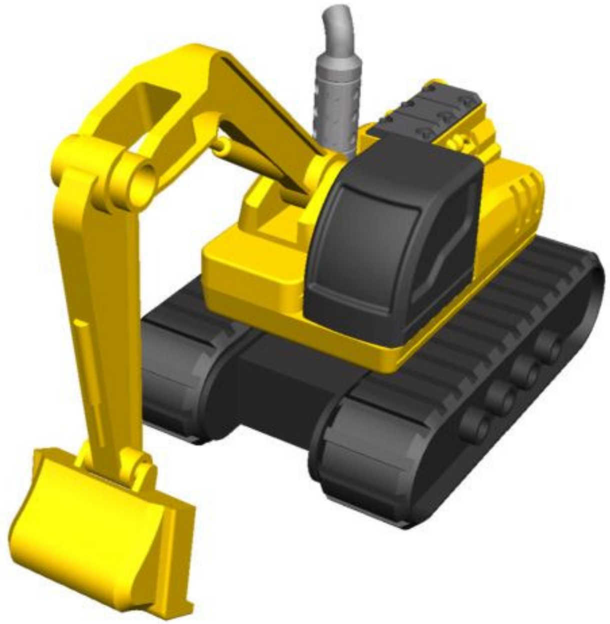
To collect all component files into your project folder:-

- In the folder, you should have two part files;
  - Front_arm_a.sldprt
  - Cabinet_a.sldprt

- For the rest, you can find in this folder:

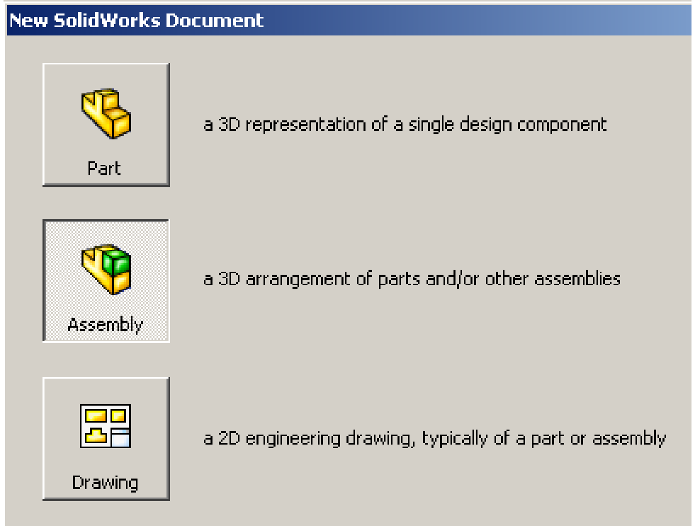
  <u>(Your DVD name):\Model</u>

  - Base_a.sldprt
  - Body_a. sldprt
  - Arm_support_a.sldprt
  - Engine_a. sldprt
  - Back_arm_a. sldprt
  - Bucket_a. sldprt
  - Exhaust_a. sldprt



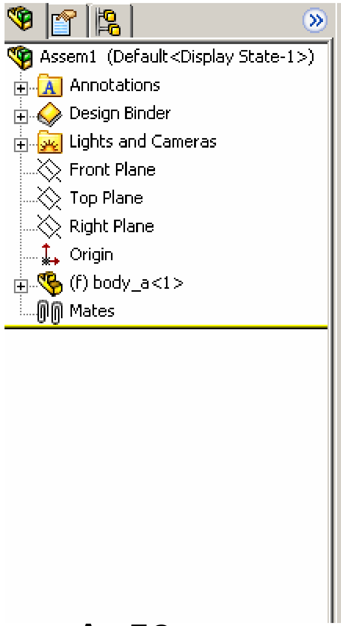
Tutorial 1C

- Enter Solidworks
- Close all files
- Select "**File**/**New…**" on the menu
- Click "**Assembly**" and then OK

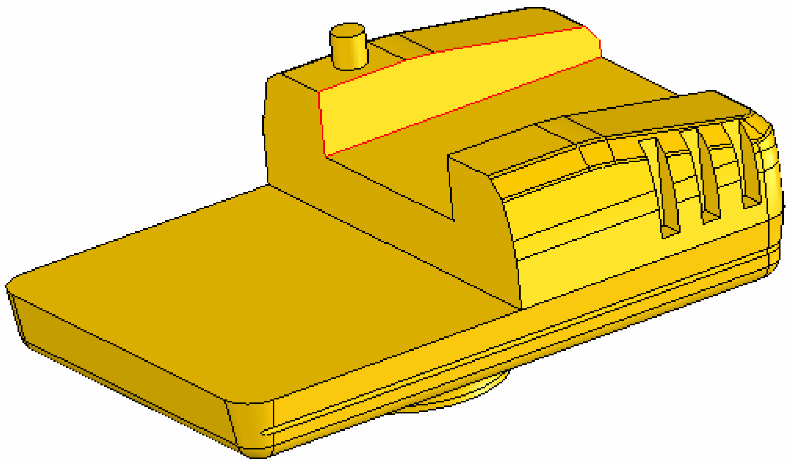


To insert existing parts into the assembly:-

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



The first component is fixed in position by default; (f) = (fixed)



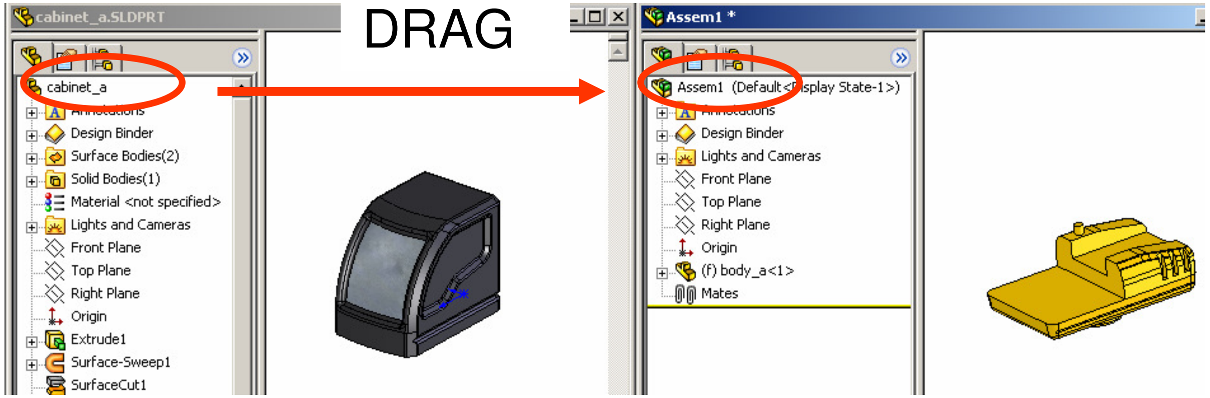
Tutorial 1C

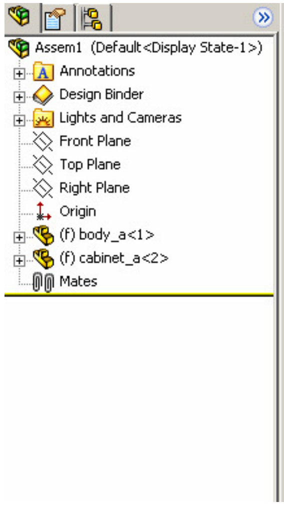
To insert existing parts into the assembly (Cont'):-

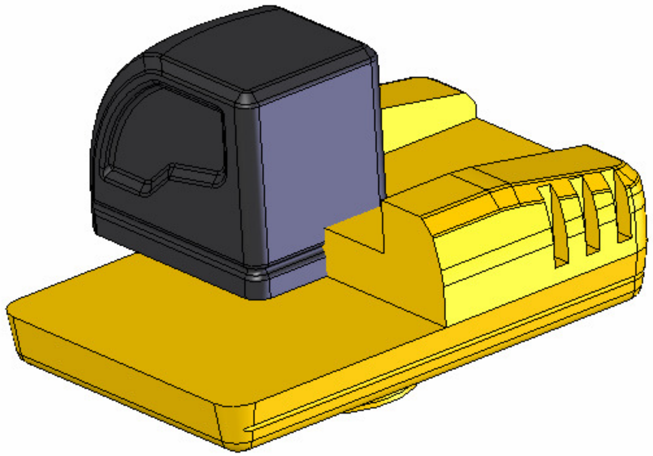
- File/Open…"Cabinet_a.sldprt".
- Select "Window/Tile vertically".
- Click and hold the highest icon of the part tree, and then drag it onto the assembly tree.

OR

- (NO need to open a part file)
- Click "**Insert Component**" icon
- Click "Browse…" icon
- Select "Cabinet_a.sldprt"
- Press Enter key to complete
- (If pressing Enter key or clicking ok to complete, the component will be aligned onto the assembly origin and will be fixed in position)
- To make the component free-to-move, right-click it and select "float"





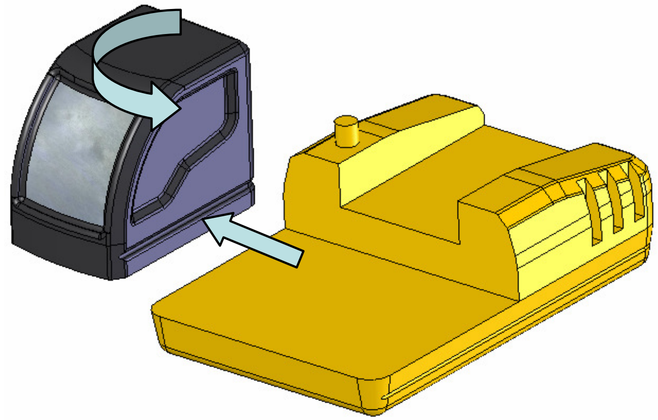


Tutorial 1C

To move a part

(1) By Direct Dragging:-

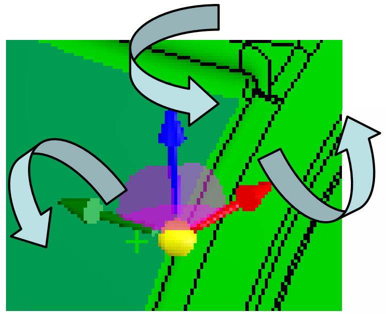
- Click on the part directly and drag it to the desired location
- To rotate the part, click "Rotate Component" icon and drag the part

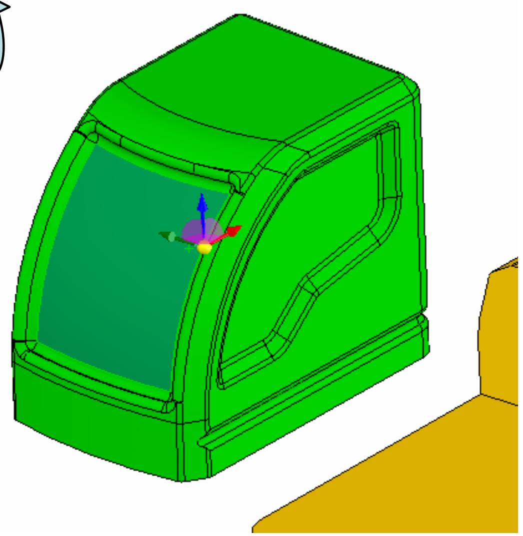


OR

(2) By "Triad":-

- Right-click "cabinet_a" (on the tree /on the model)
- Select "**Move with Triad**"
- (A local coordinate system appears; place the mouse cursor onto any axis to move the part along the selected axis)
- (Remark: Left Button for translation; Right Button for rotation)
- (Remark: To change the origin location, drag the yellow dot and then drop onto the desired place)



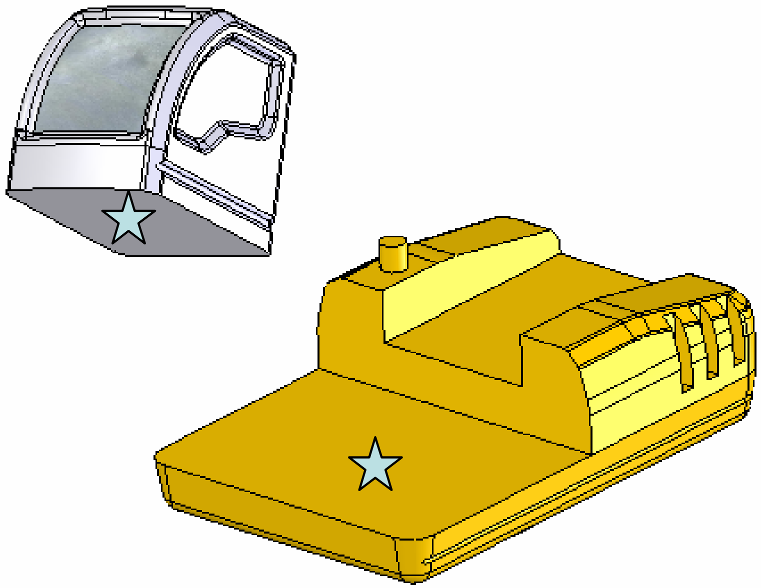


Tutorial 1C

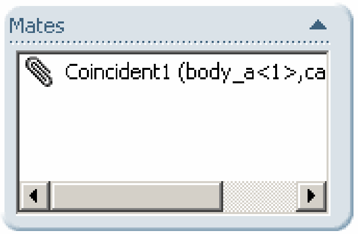
To link "Cabinet" to "Body" by adding constraints:-

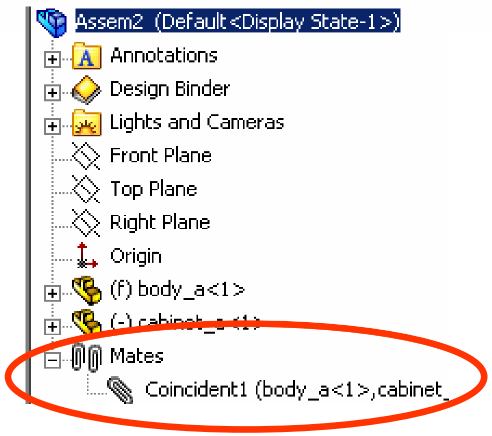
- By default, "body_a" (the first component) is fixed in position

- Click "**Mate**" icon
- Select the bottom face ☆of "Cabinet" and then select the face ☆of "Body"
- "Cabinet" is snapped onto "Body"
- Press Mouse Right Button to accept
- A "coincident" mate is created



If you want to delete a constraint, just select the constraint either on the list (during mating) or on the tree (after mating), and then press "Delete" key on the keyboard.



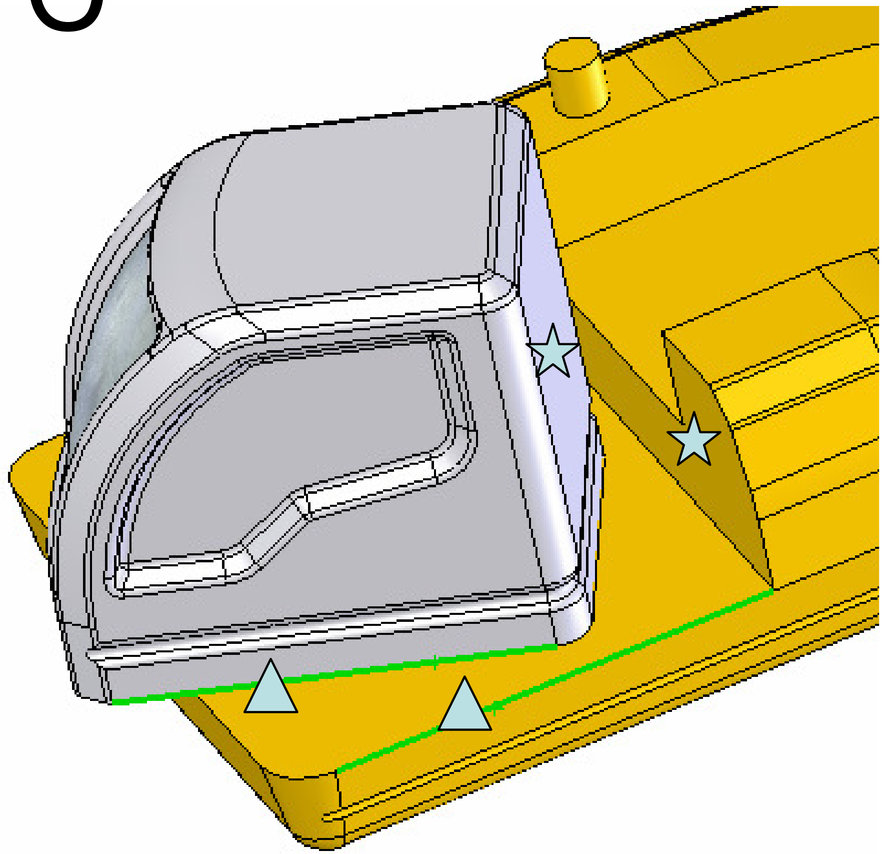


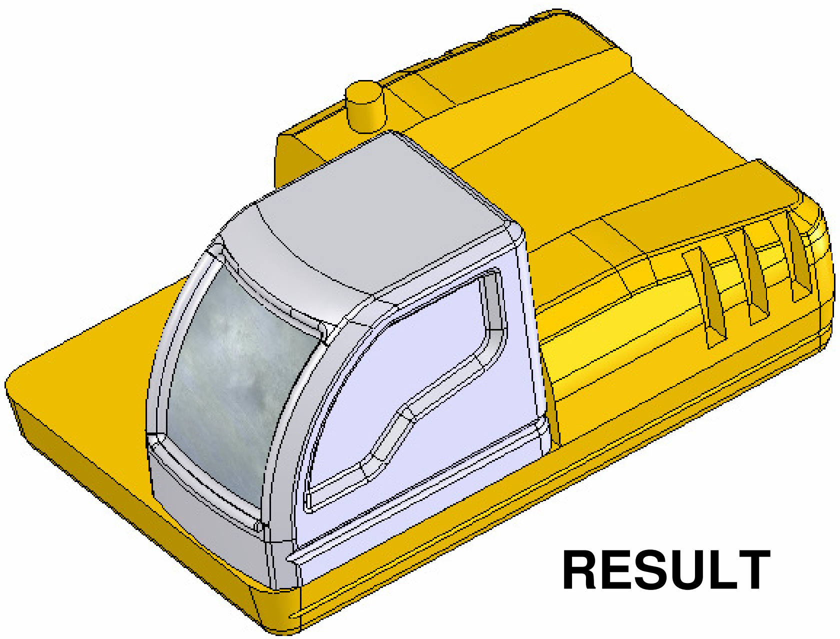
Tutorial 1C

To link “Cabinet” to “Body” (cont’)

- Add another “**Coincident**” mate between the faces marked with ☆

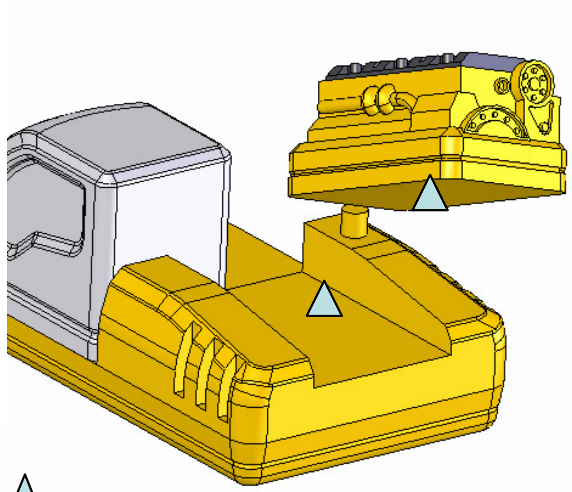
- Add a “**Coincident**” mate between the edges marked with △

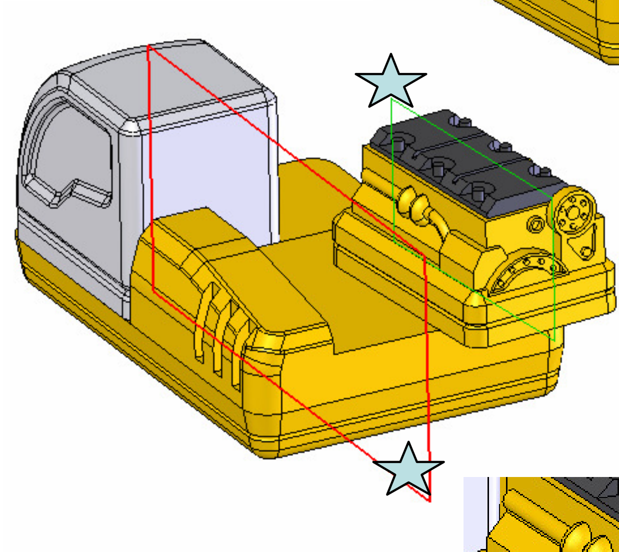


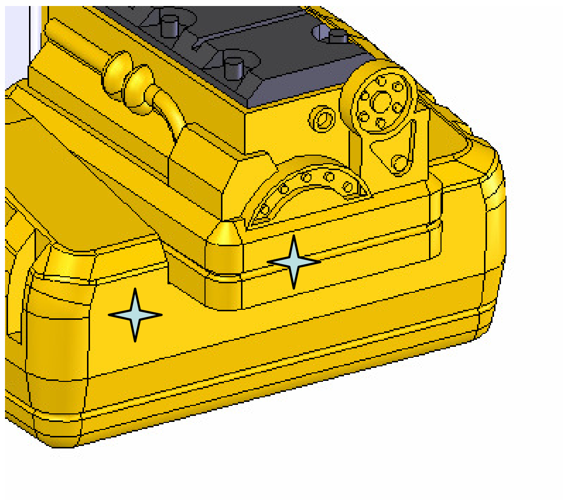


RESULT

Tutorial 1C







To insert "Engine" into the assembly

- Click "**Insert Component**" icon
- Click "Browse…" icon
- Select "Engine_a.sldprt"
- Click an open area near the assembly
- (The component will be dropped onto the selected location. Different from the previous component, it has been "float")

Drag "Engine" out so that we can see the whole model

To link "Engine" to "Body"

- Add a "**Coincident**" mate between the bottom face of Engine and the top face of Body △
- Add a "**Coincident**" mate between the Right plane of Engine and the Top plane of Body ☆
- Add a "**Coincident**" mate between the faces marked with ✧

Written by Dickson Sham

Tutorial 1C

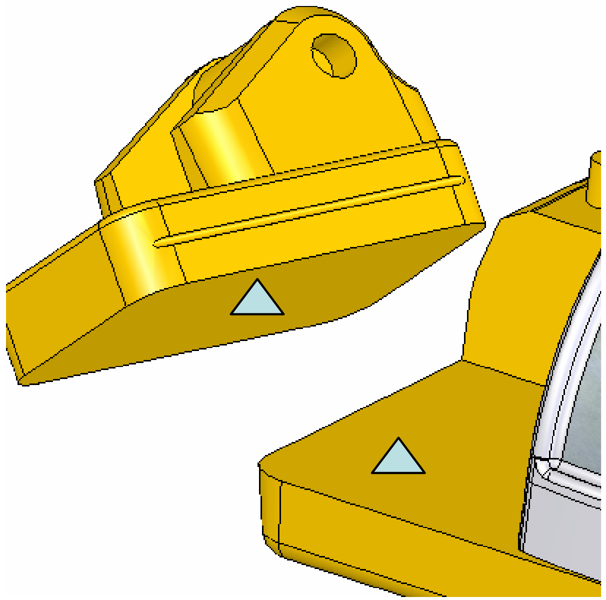
Insert "Arm_support" into the assembly

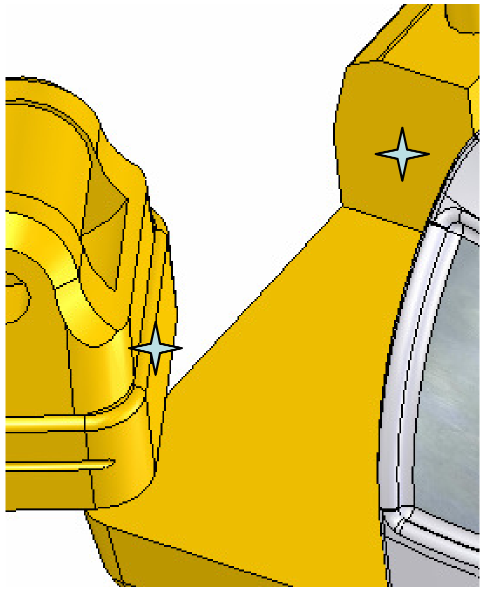
Drag "Arm_support" out so that we can see the whole model

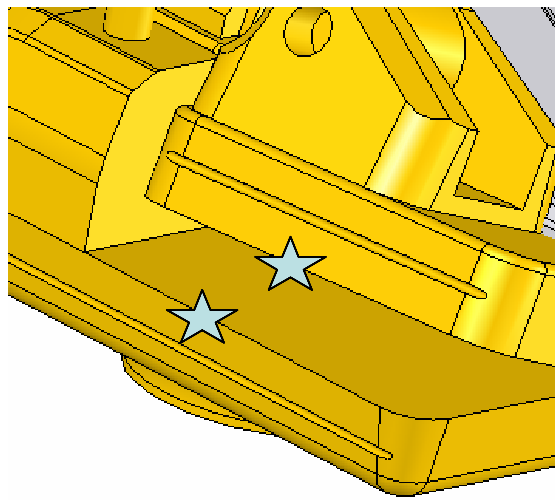
To link "Arm_support" to "Body"

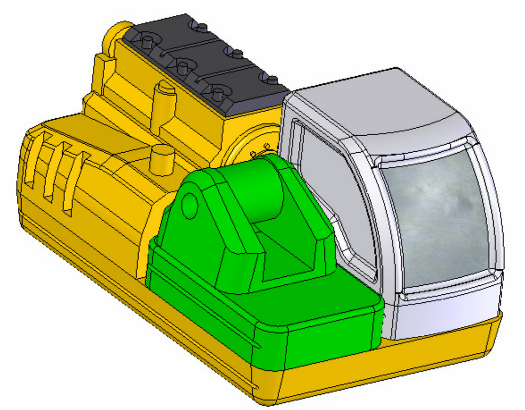
- Coincident (Face to Face) △
- Coincident (Face to Face) ✧
- Coincident (Edge to Edge) ☆

- Remark: We cannot add "Coincident relation" between the faces with ☆ because they are not parallel. Therefore they can only add the constraint between their edges









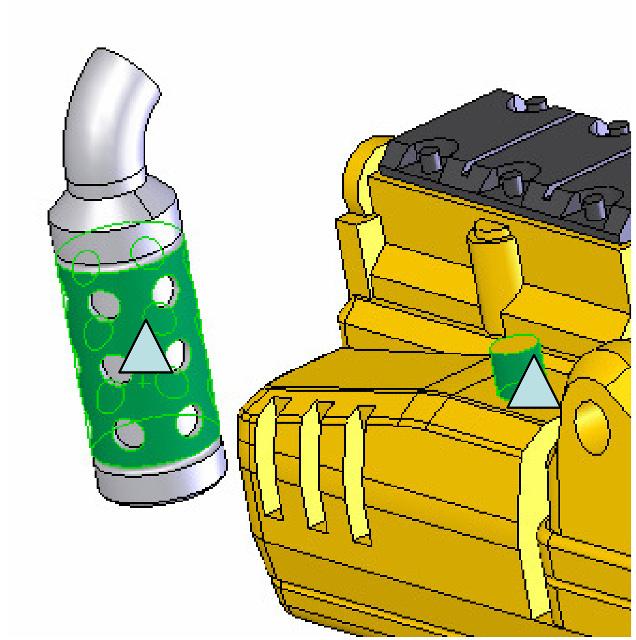
Tutorial 1C

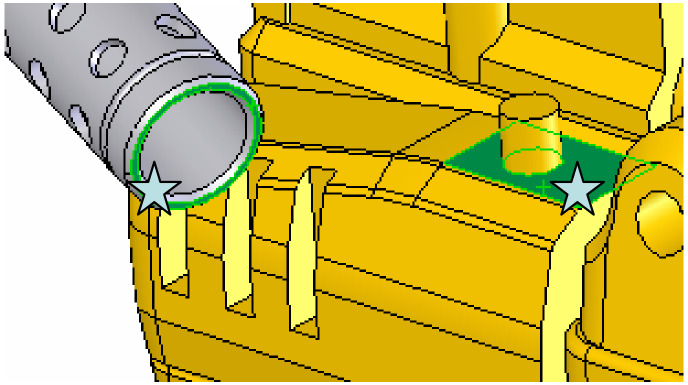
Insert "Exhaust" into the assembly

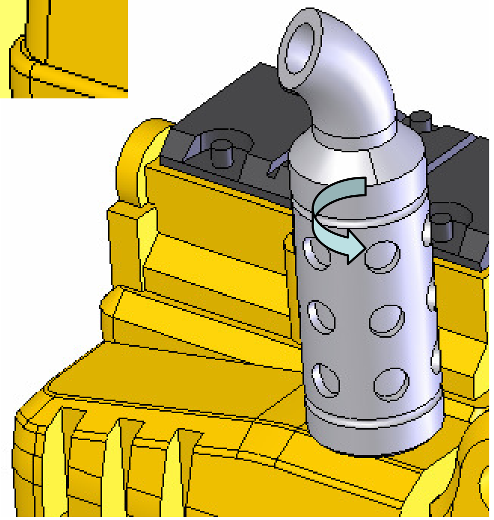
Drag "Exhaust" out so that we can see the whole model

Link "Exhaust" to "Body"

- Add a "**Concentric**" mate between the two circular faces (Their axes are then coincided)
- Add a "Coincident" mate (Face to face) as shown.
- The angular orientation is not important in this case, but you may rotate the part by dragging directly







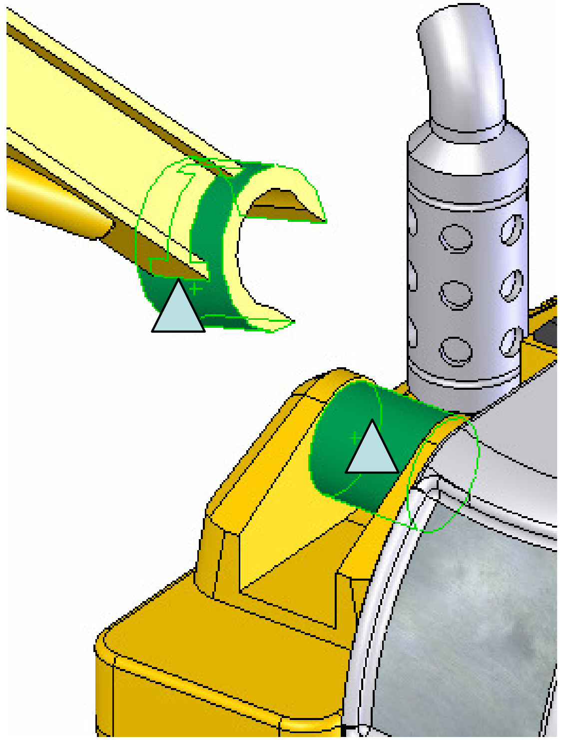
Tutorial 1C

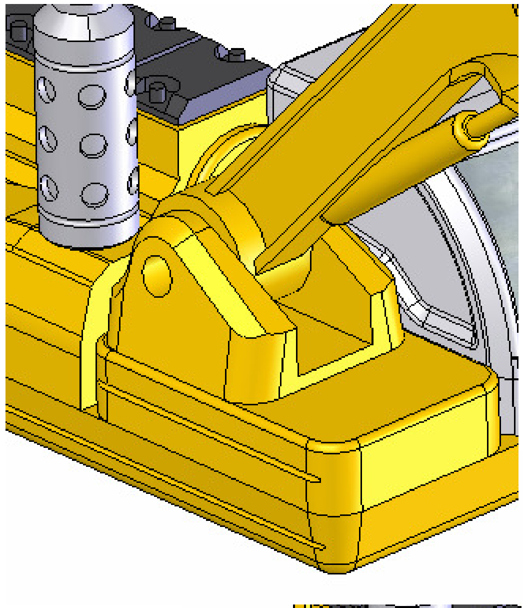
Insert "Back_arm" into the assembly

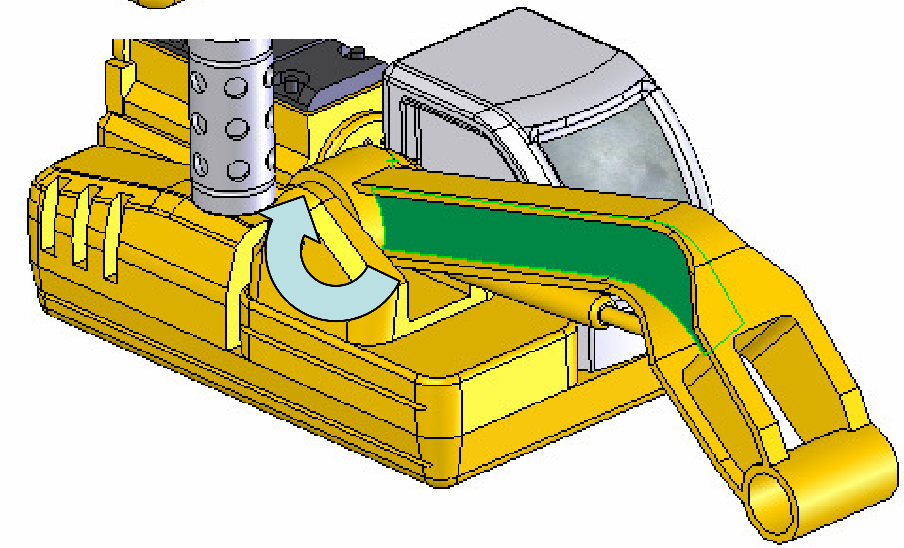
Drag "Back_arm" out so that we can see the whole model

Link "Back_arm" to "Arm_support"

- Add a "**Concentric**" mate between the two circular faces △
- Add a "**Coincident**" mate between Front Plane of Back_arm and Right Plane of Arm_support
- The angular orientation is not important in this case, you may rotate it by dragging directly







Tutorial 1C

Insert "Front_arm" into the assembly

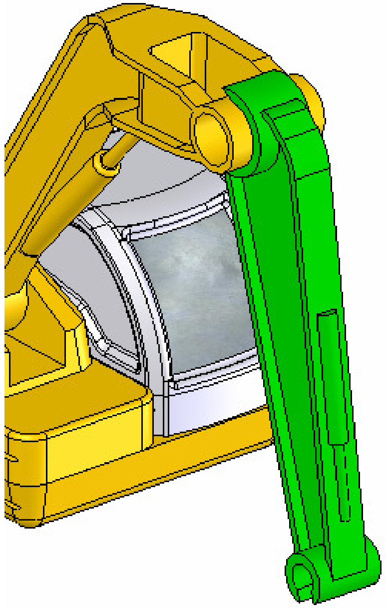
Drag "Front_arm" out so that we can see the whole model

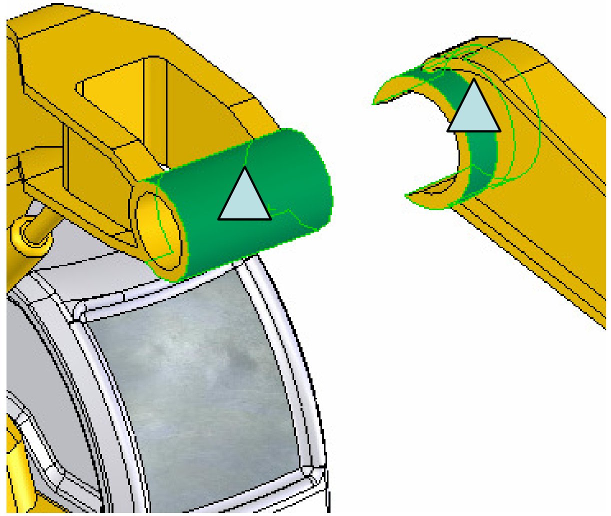
Link "Front_arm" to "Back_arm"

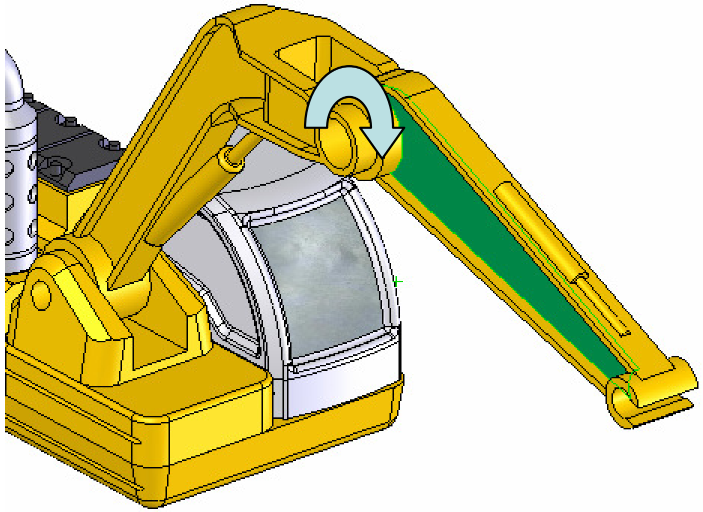
- Add a "**Concentric**" mate between the two circular faces

- Add a "**Coincident**" mate between Front Plane of front_arm and Front Plane of back_arm

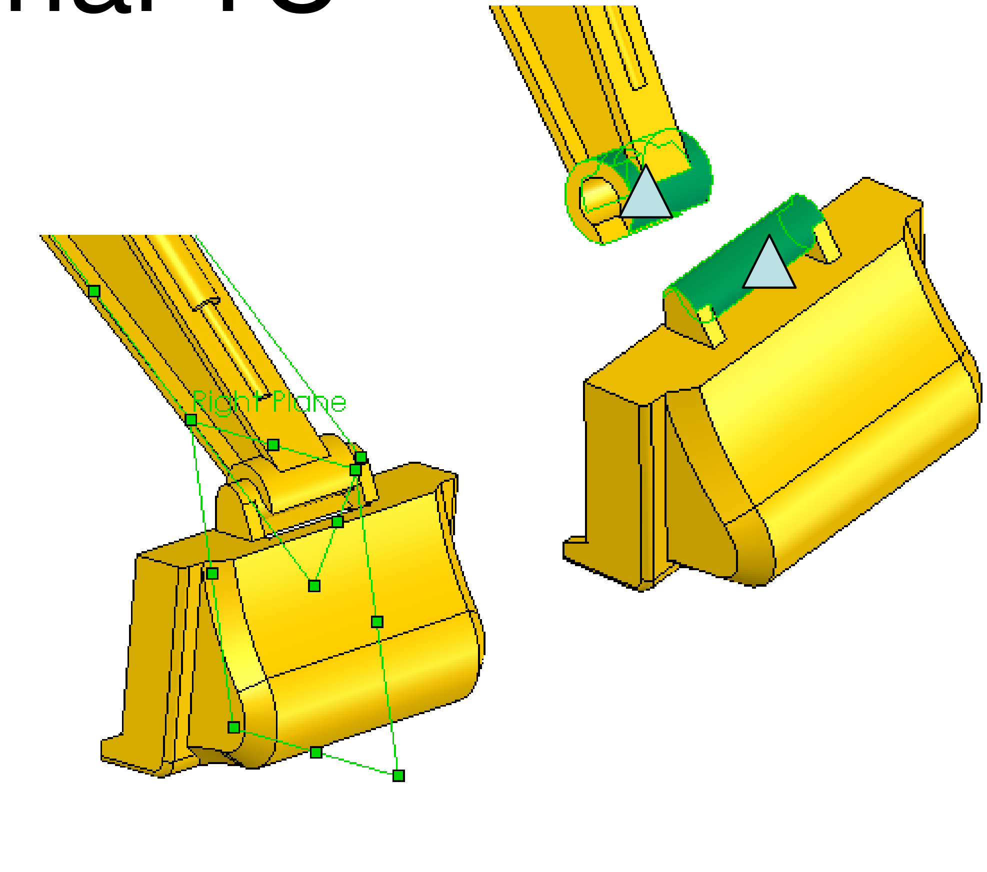
- The angular orientation is not important in this case, you may rotate it by dragging directly







Tutorial 1C

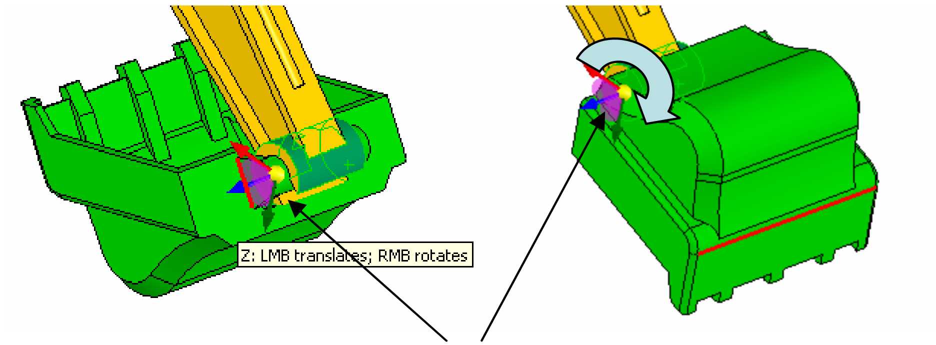


Insert "Bucket" into the assembly

Drag "Bucket" out so that we can see the whole model

Link "Bucket" to "Front_arm"

- Add a "**Concentric**" mate between the two circular faces △
- Add a "**Coincident**" mate between Right Plane of bucket and Front Plane of front_arm
- The angular orientation is not important in this case, you may rotate it by dragging directly or moving with triad



Snap the triad onto the axis and then rotate the bucket along the axis

Tutorial 1C

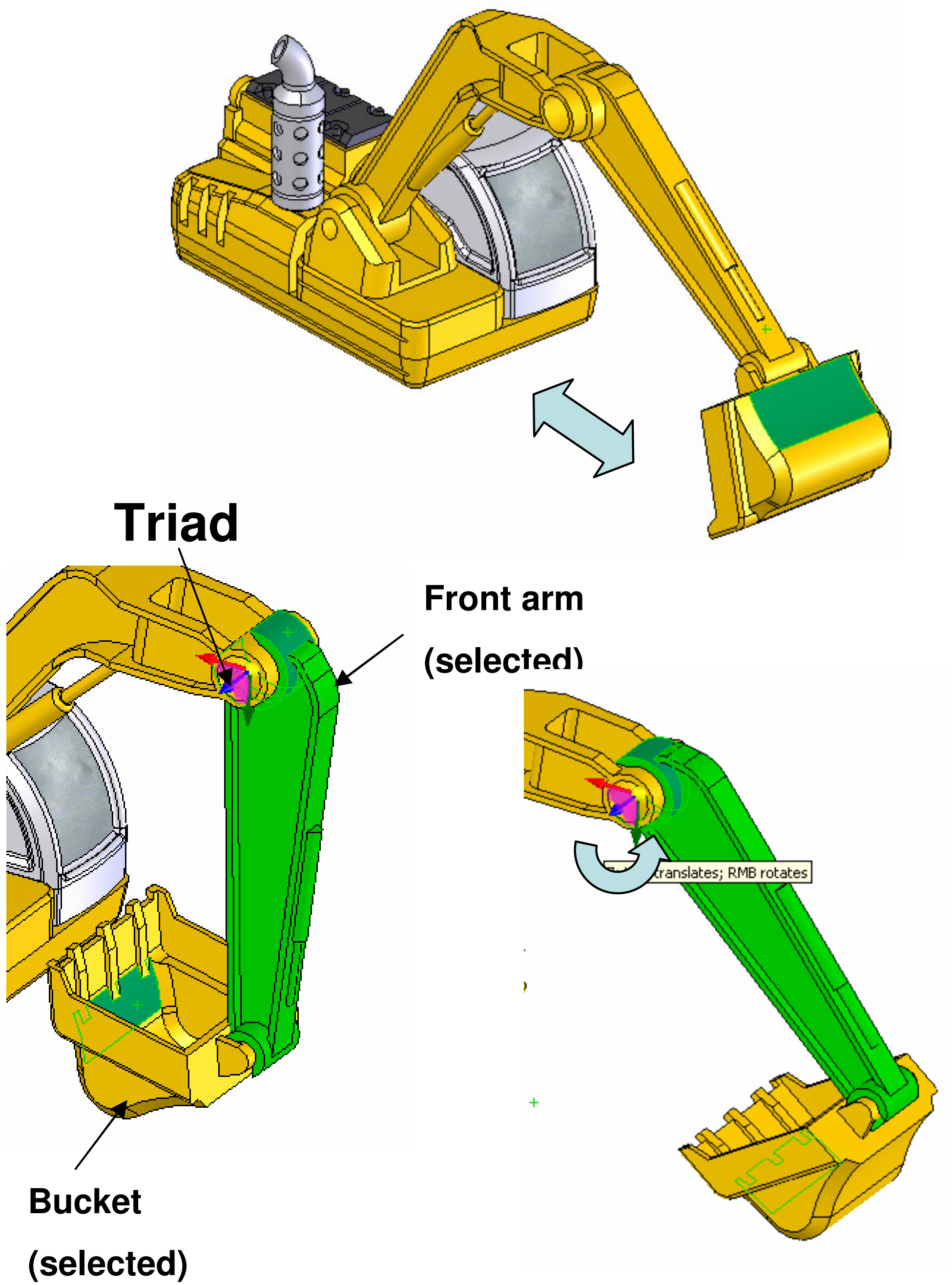
To simulate the motion of the machine arm:-

- Because the angular orientation at the joints is not constrained, we can change the angular positions of the arm and the bucket by dragging directly

- Or dragging with triad

- (Remark: To snap the origin of the triad onto the axis of rotation, drag it onto the corresponding circular face)

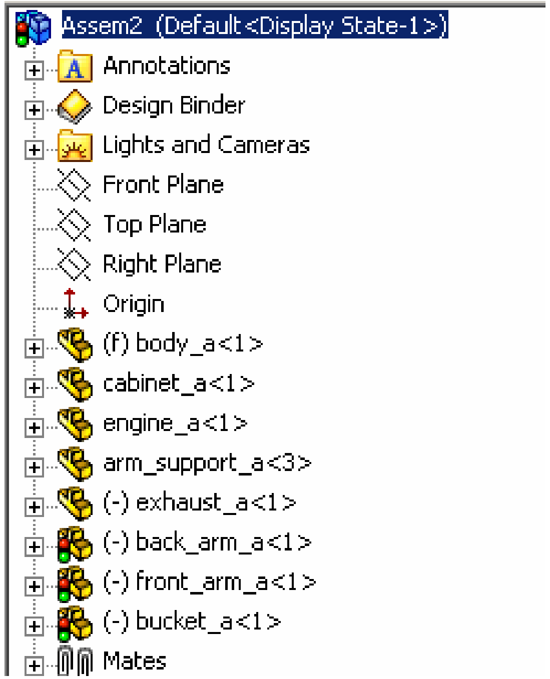
- (Remark: To rotate the front arm with triad, we need to multi-select “front_arm” and “bucket” first; when rotating, their relative angle can be kept changed)



Tutorial 1C

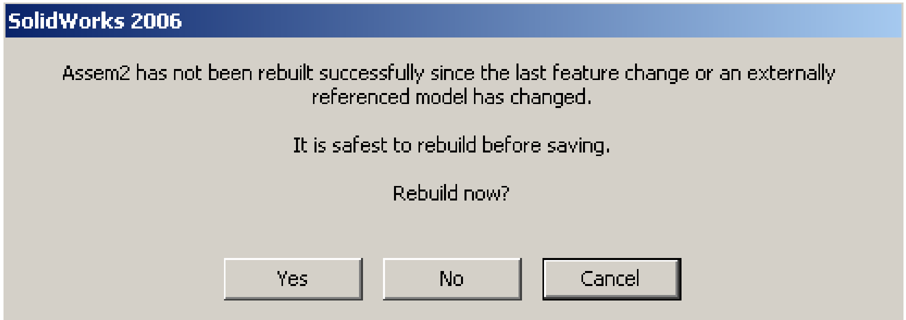
From the assembly tree, we should have:

(f) Body	fixed in position
Cabinet	fully constrained
Engine	fully constrained
Arm_support	fully constrained
(-) Exhaust	not fully constrained
(-) Back_arm	not fully constrained
(-) Front_arm	not fully constrained
(-) Bucket	not fully constrained



To save the file:-

- Select “**File/Save all**” on the menu bar
- Click Yes on the popup window
- Click “Save As…” icon
- Enter “Upper_assm_a.sldasm” as filename and save it in your project folder.
- **Close All files.**

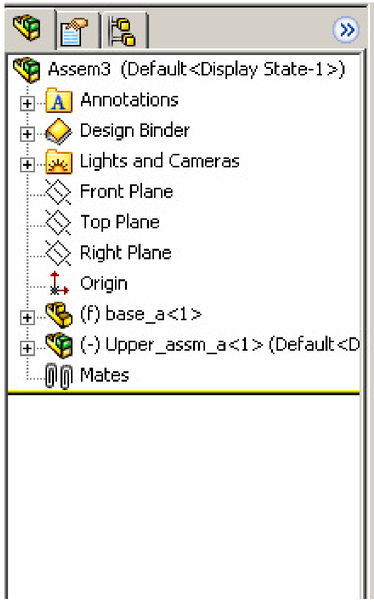


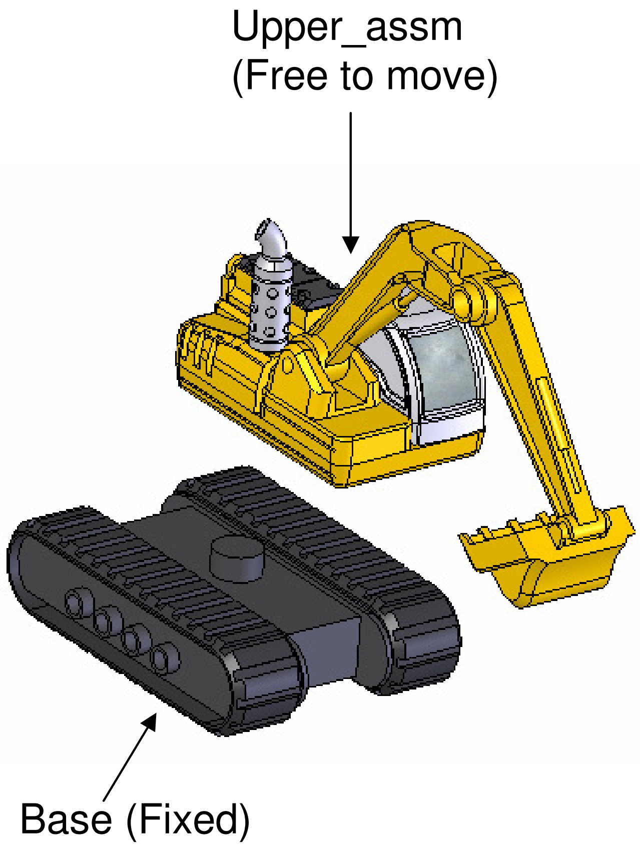
Tutorial 1C

To assemble the upper assembly to the base:-

- Select **“File/New…”** on the menu
- Click “**Assembly**” and then OK
- Click “Browse…”
- Select the file “Base_a.sldprt”
- Click “Open”
- Press Enter key to complete

- Click “**Insert Component…**” icon
- Click “Browse…”
- Change Files of Type to “Assembly (*.asm, *.sldasm)”
- Select the file “upper_assm_a.sldasm” in your folder
- Click an open area near the Base (so that it is free-to-move after inserted)

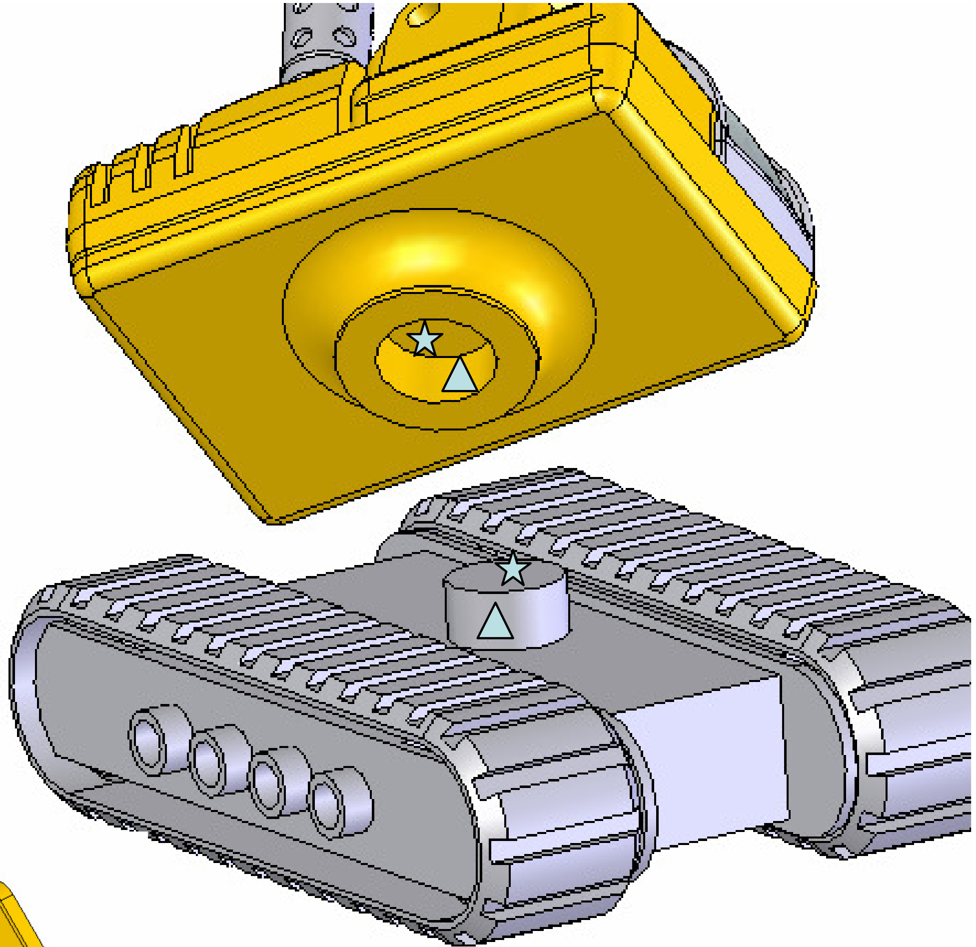




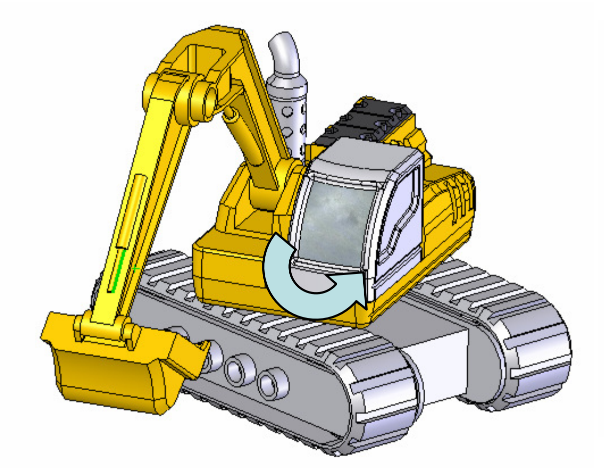
Tutorial 1C

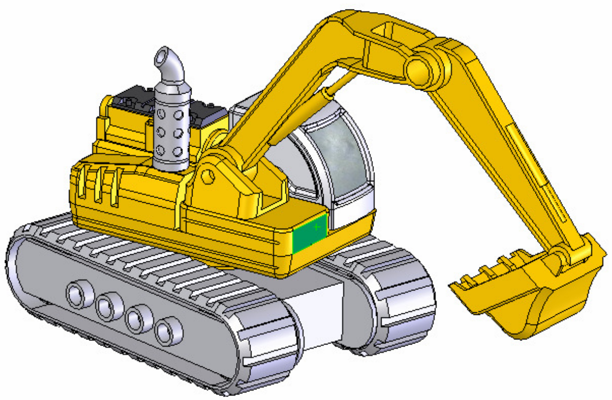
Link “Upper_assm” to “Base”

- Add a “**Concentric**” mate between the two circular faces △
- Add a “**Coincident**” mate between the faces with ☆



To simulate the angular motion of the upper assembly by dragging <u>ANY</u> component of the upper assembly



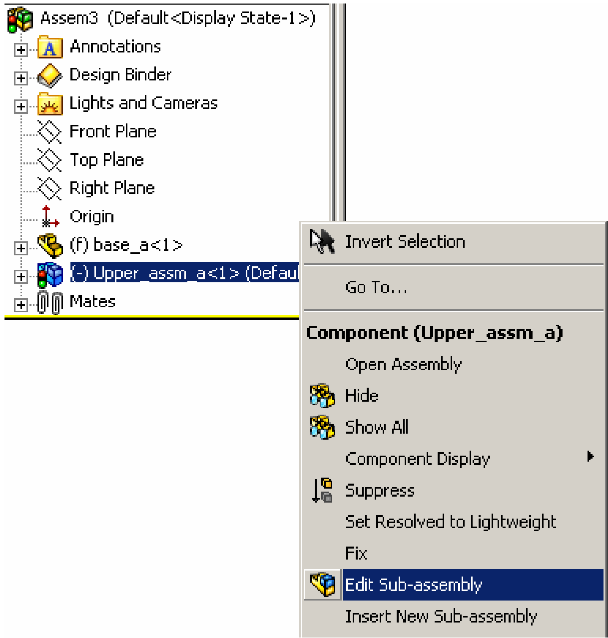


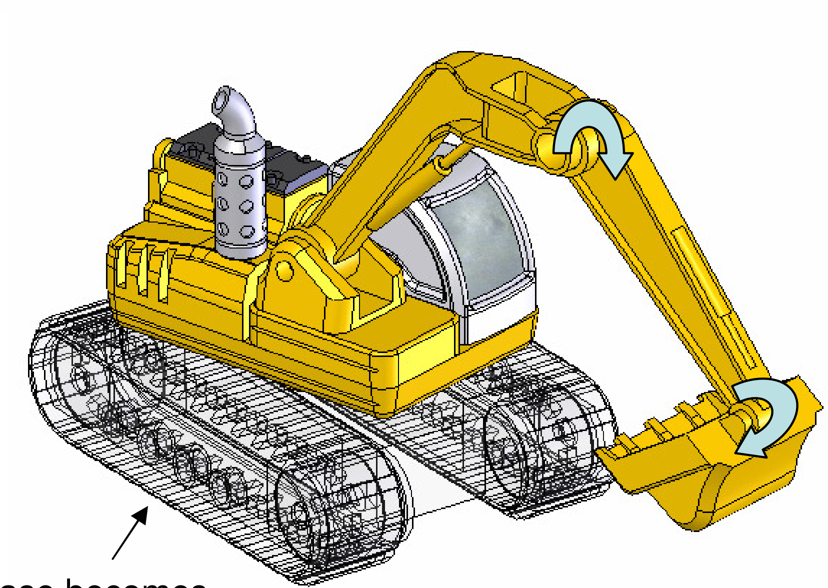
Remark: the relative positions of the components of the upper assembly will not be changed

Tutorial 1C

To simulate the angular motion of the machine arm:-

- Right-Click “Upper_assm” on the tree and then select **“Edit Sub-assembly”** on the contextual menu
- (The base will become transparent)
- Now, you can move the arm by dragging directly
- Back to original, just right-click the topmost of the assembly tree, i.e. Assem3, and select **“Edit Assembly”**



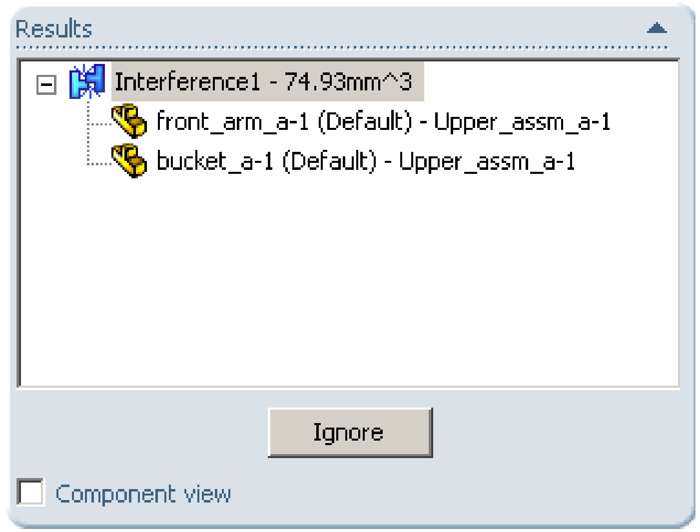


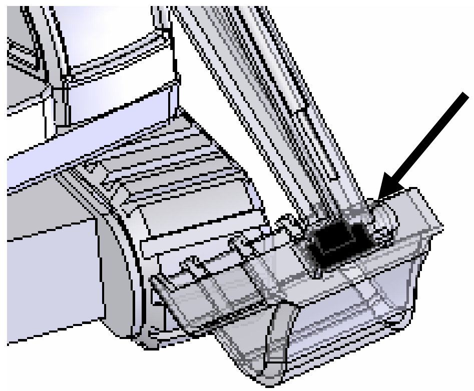
Base becomes transparent

Tutorial 1C

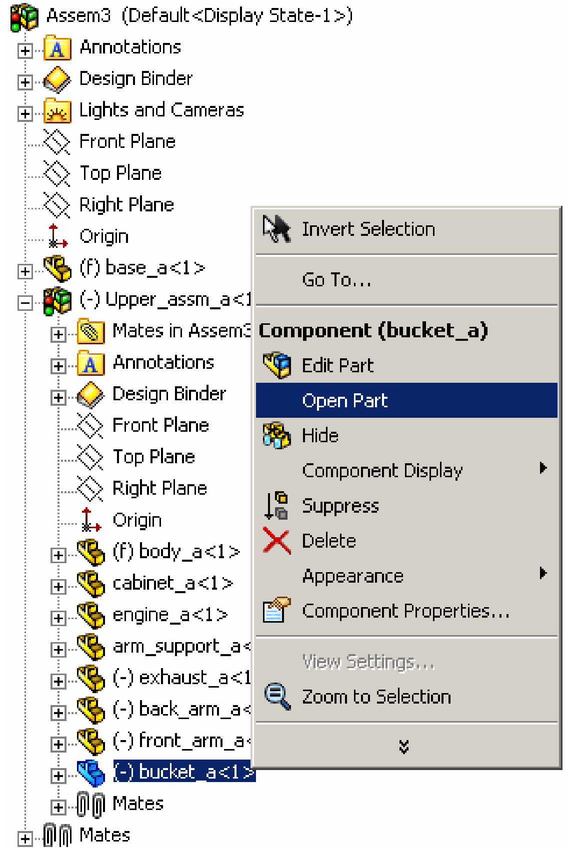
To check any interference among parts:-

- Click “**Interference Detection**” icon
- Click “Calculate” icon
- The system finds an interference (between “front-arm” and “bucket”)
- (the overlapping portion is highlighted on the model)
- Press “Esc” key on the keyboard to exit
- Now we know where the problem is and we are going to correct the bucket design…





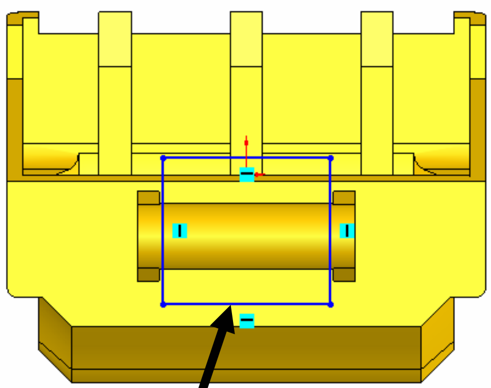
Tutorial 1C



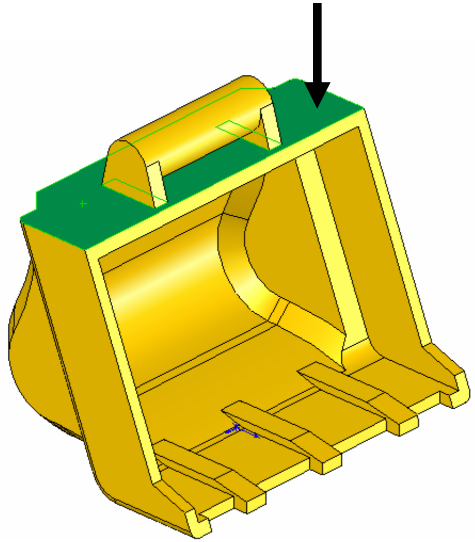
To modify the error part:-

- Extend the Assembly Tree and locate the error part “Bucket”
- Right-click on it and select “**Open Part**” on the contextual menu
- Select “**Sketch**” icon and select the planar face under the joint
- Draw a rectangle in the middle

Create a sketch on this face

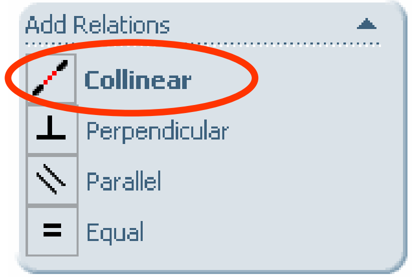


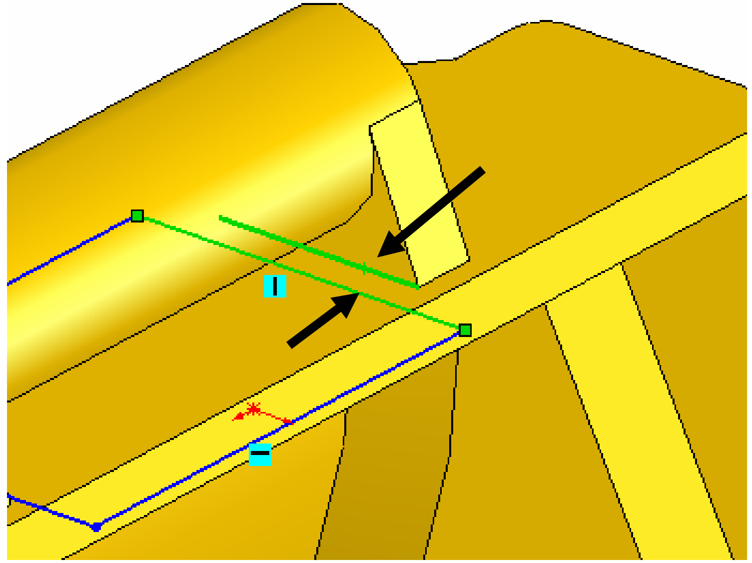
Draw a rectangle here



Tutorial 1C

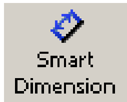
- Add a “**collinear**” relation to align an edge of the rectangle onto the solid edge nearby.



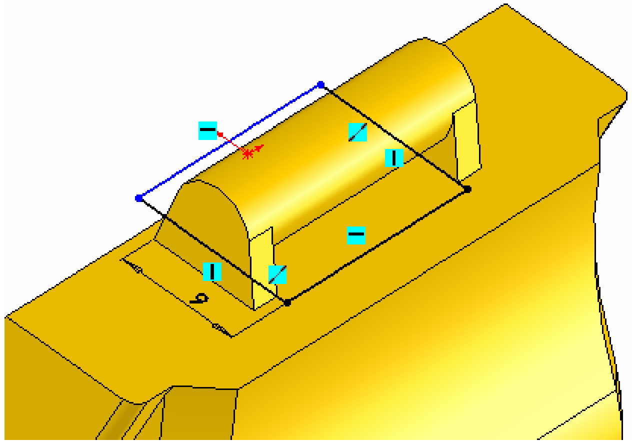


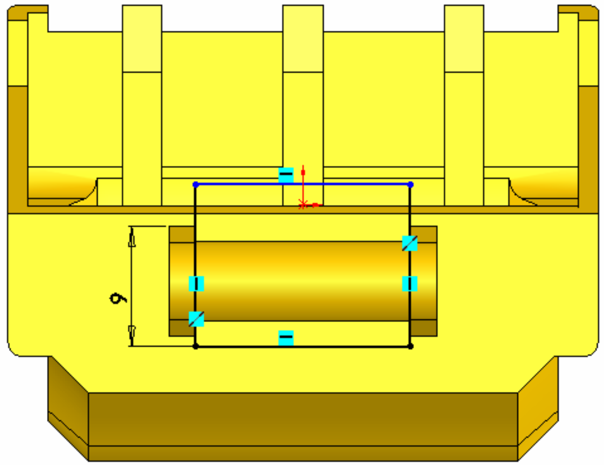
- Similarly, align the edge on the opposite side.

- Add a **dimensional** <u>9mm</u> as shown



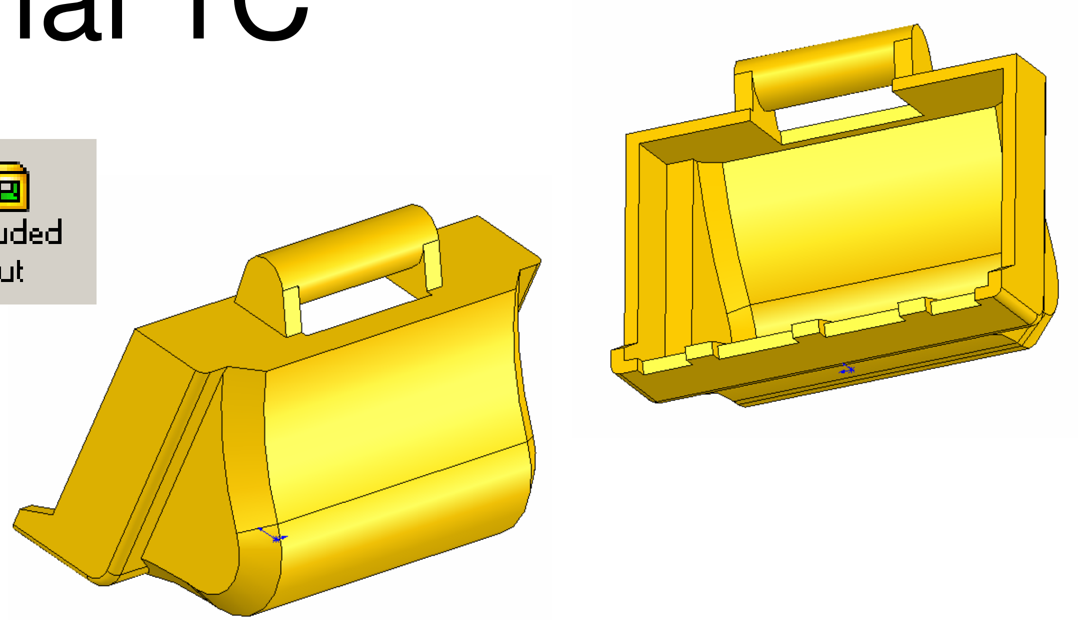
- Exit to complete





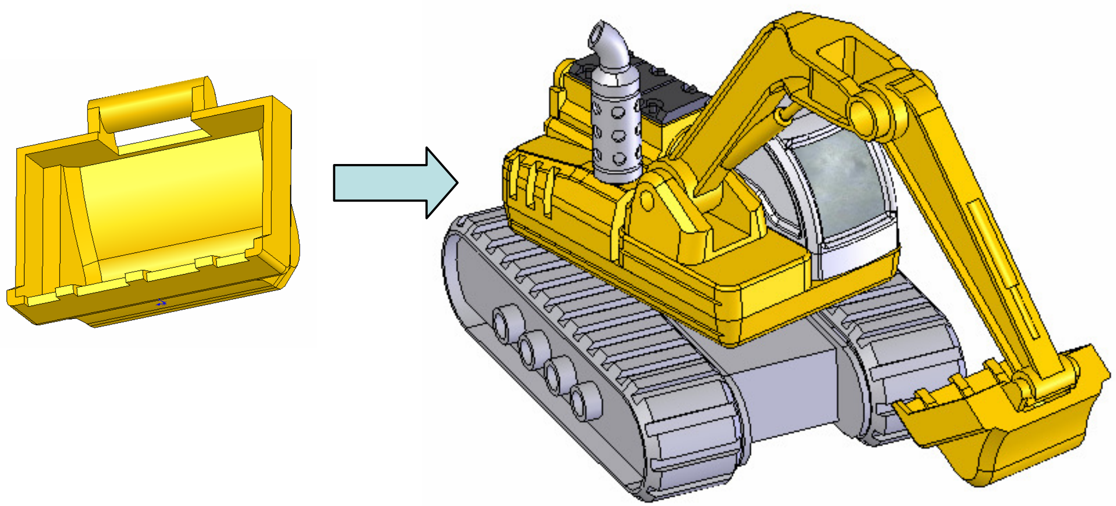
Tutorial 1C

- Click “**Extruded Cut**” icon
- Enter 5mm as D1
- Press Enter key to complete



To re-do the interference check of the whole assembly:-

- Select “ Window/Assem3” on the menu bar to switch the display back to the whole assembly.
- Click yes to rebuild
- You can see that the “Bucket” of the assembly has been updated with an opening

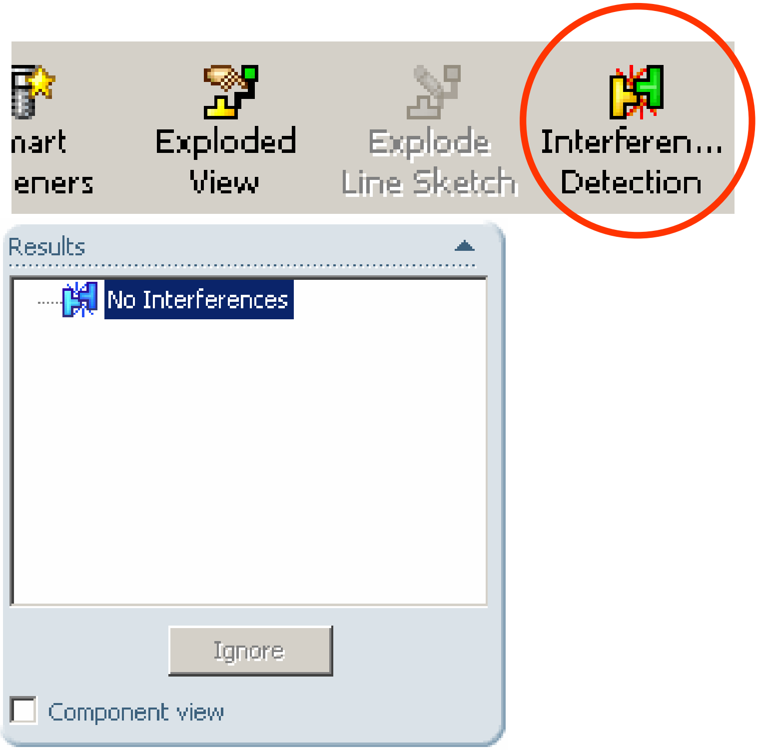


The bucket has been updated

Tutorial 1C

- Click “**Interference Detection**” icon
- Click “Calculate” icon

- There should be no clash error on the list now
- Press “Esc” key on the keyboard to exit



To save the file:-

- Select “**File/Save all**”
- Enter “Excavator_a.sldasm” as filename and save it in your project folder.

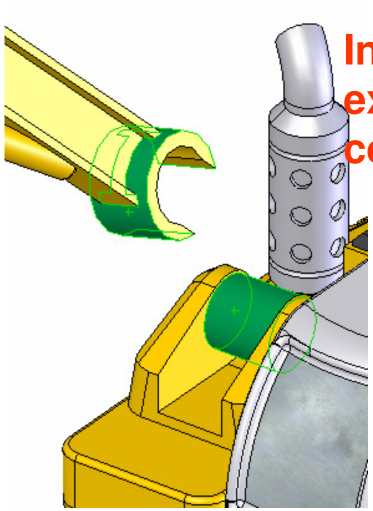
- **Close All files.**

END of Tutorial 1C

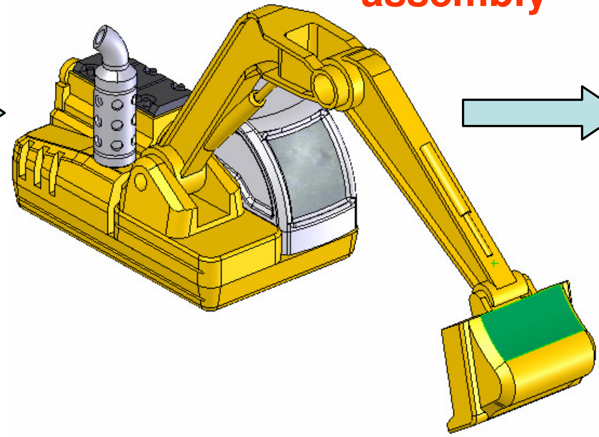
Summary of Tut-1C

Assemble parts:-

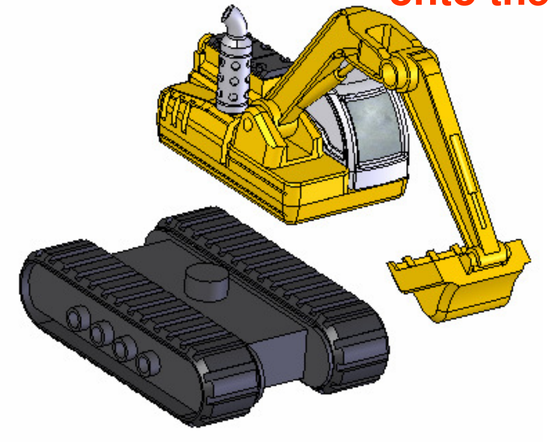
Insert existing components

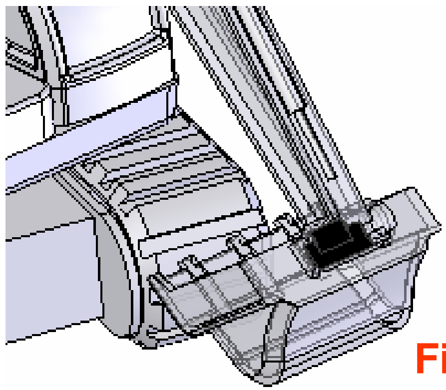


Build an upper assembly



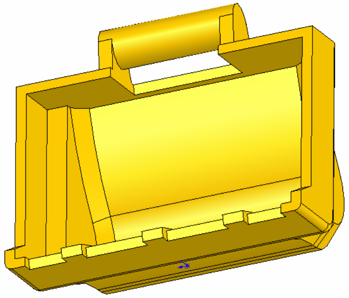
Put the upper assembly onto the base



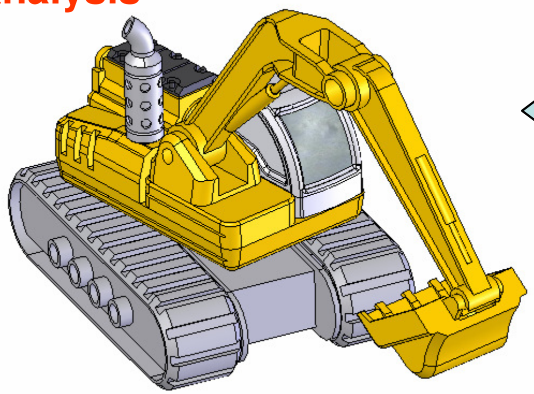


Find a clash between parts

Modify the error part



Redo the clash analysis



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