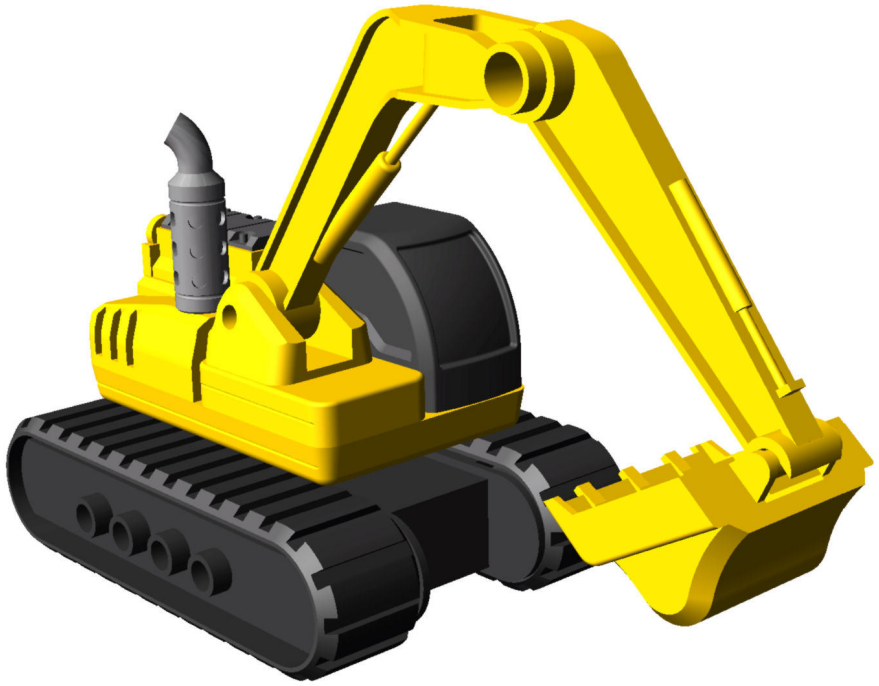
CATIA V5 Overview

(Tutorial 1-Toy Excavator)



Infrastructure

Sketcher

Part Design (Solid-modeling)

2D-Drafting

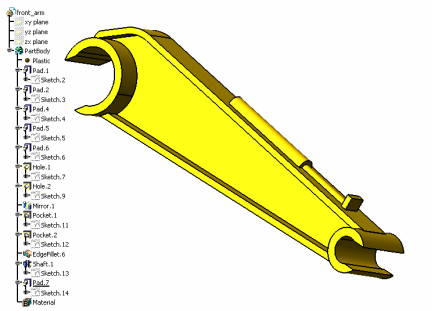
GSD (Surface-modeling)

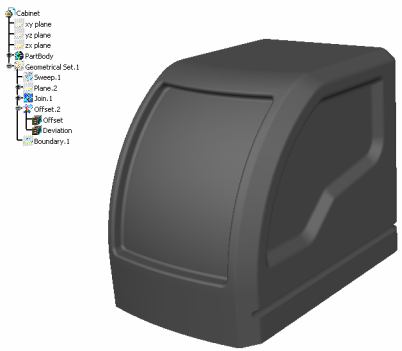
Assembly Design

By Dickson Sham (ME dept, HKPU)

Tutorial 1A

- CATIA Infrastructure
- Sketcher
- Part Design (Solid-modeling)
- 2D-Drafting
- Auto-Update



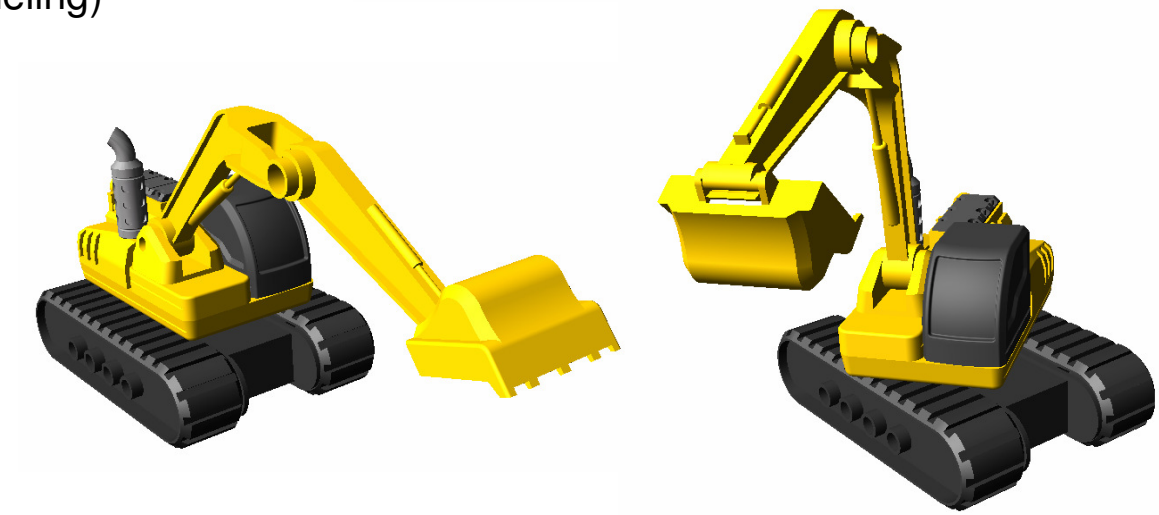


Tutorial 1B

- Part Design (Solid-modeling)
- Generative Shape Design (Surface-modeling)
- Real-time rendering & Material Mapping

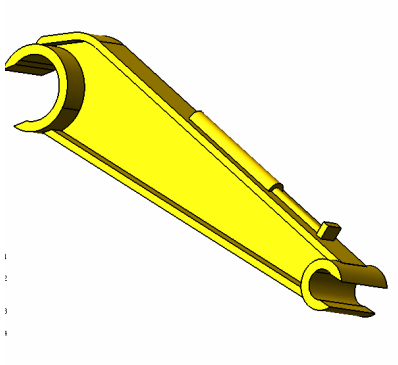
Tutorial 1C

- Assembly Design
- Clash Detection & Part Modification

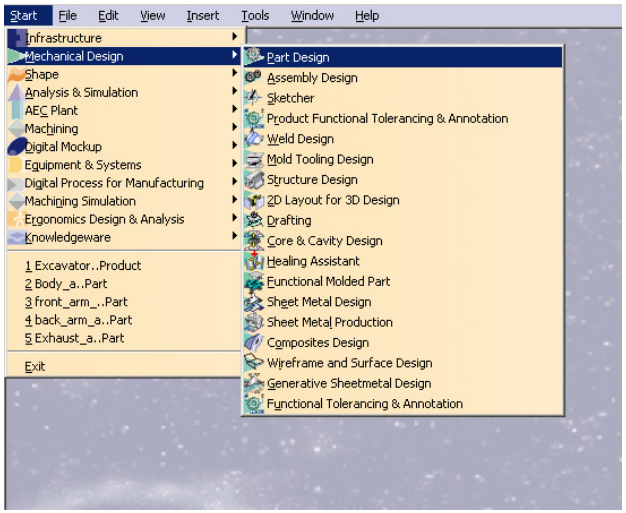


Please be reminded that this series of tutorials is designed to demonstrate a design approach with CATIA, rather than the command itself.

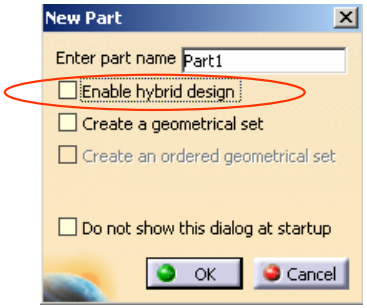
Tutorial 1A



- Enter CATIA by double-clicking its icon on the desktop.
- (If a license menu pops up), select **ED2** and close CATIA. Then reopen again.
- By default, a empty “Product” file is created. But now, you don’t need this, just select “**File/Close**” on the menu.



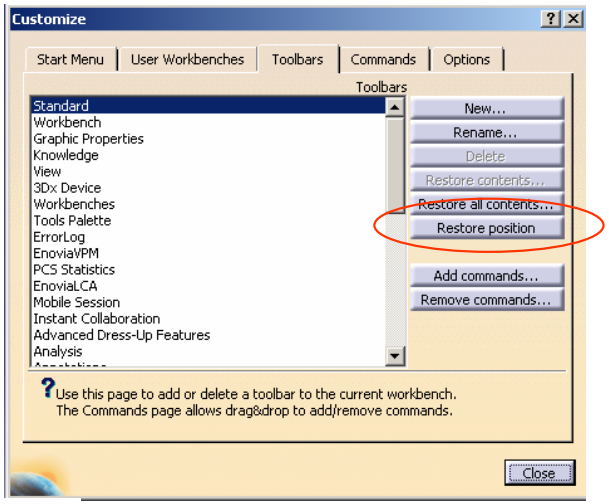
- You are going to draw a machine arm as below:-
- Select ‘**Start/Mechanical Design/Part Design**”on the menu bar.
- If you’re using Catia V5R16, uncheck “Enable Hybrid Design” and then click “ok”.
- An empty part is now created on “Part Design” workbench. You can see a specification tree at the upper left-hand corner and xyz datum planes in the middle of the screen .



Tutorial 1A

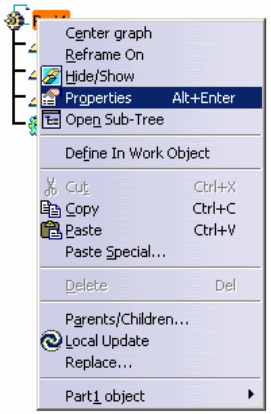
To reset the layout of workbench:-

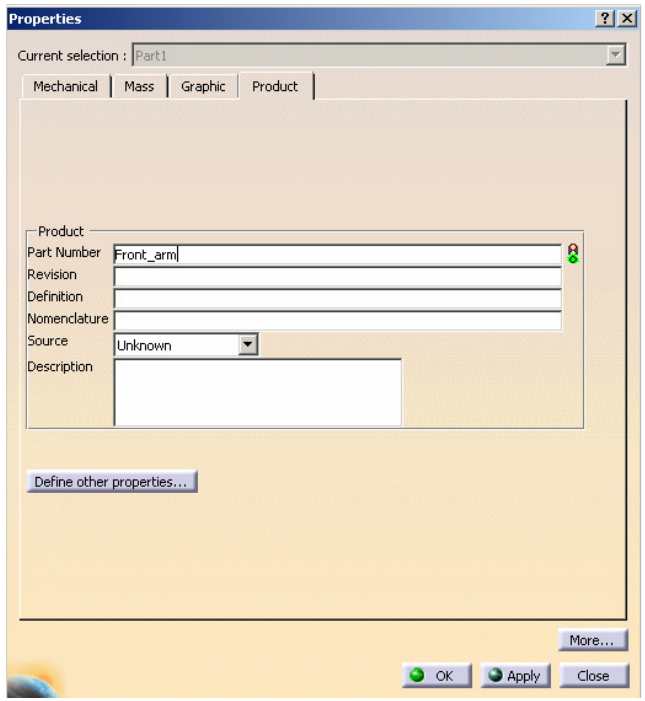
- Sometimes the workbench may not be tidy before you use; some toolbars are missing and some are at wrong positions. To reset the layout, select "**View/Toolbars/Customize**" and select "**Toolbar/restore position**" on the pop-up window; Close and exit.



To rename the tree:-

- Single-click "Part1" on the tree, right-click it, and then select "**Properties**".
- Modify Part Number as "front_arm" on the tab page "Product".
- Select "ok" to exit .



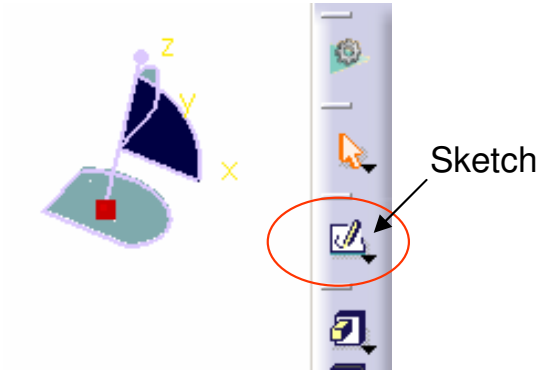


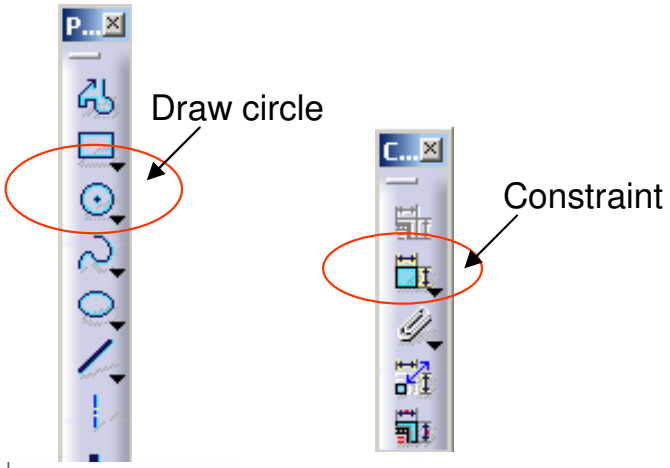
By Dickson Sham (ME dept, HKPU)

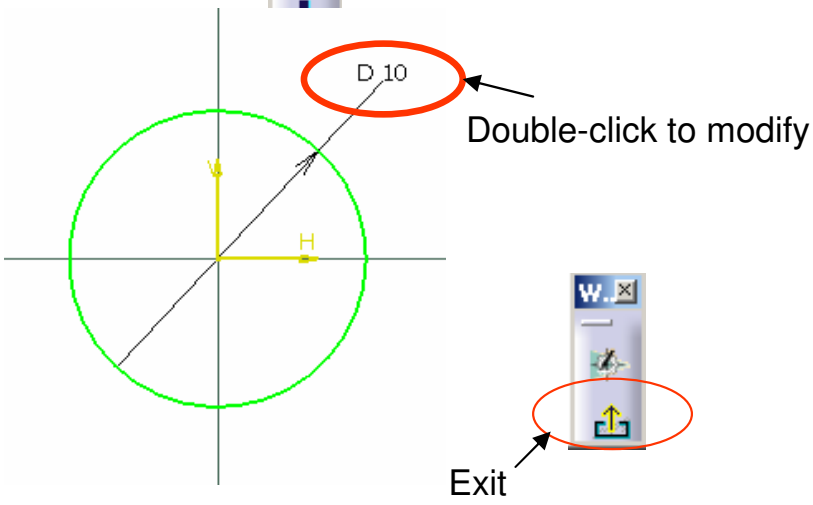
Tutorial 1A

To build 1st sketch:-

- click "**Sketch**" icon and select **xy plane.**
- Now the display is temporarily switched to a new workbench, Sketcher, in which you can draw 2D elements on the selected plane.
- **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)
- Add a dimension **constraint** onto the circle by clicking "constraint" icon and then selecting the circle.
- Double-clicking on the dimension and **modify** the diameter as 10mm; the circle will be resized automatically.
- Exit the workbench by clicking "**Exit**" icon.
- Now, you are back to Part Design Workbench (3D environment) and "Sketch.1" is created on the tree.





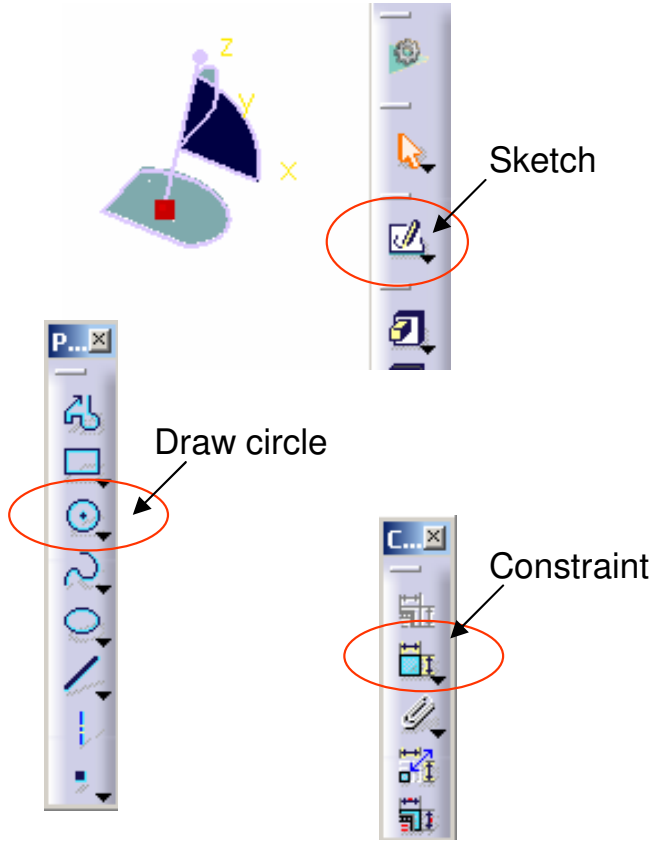


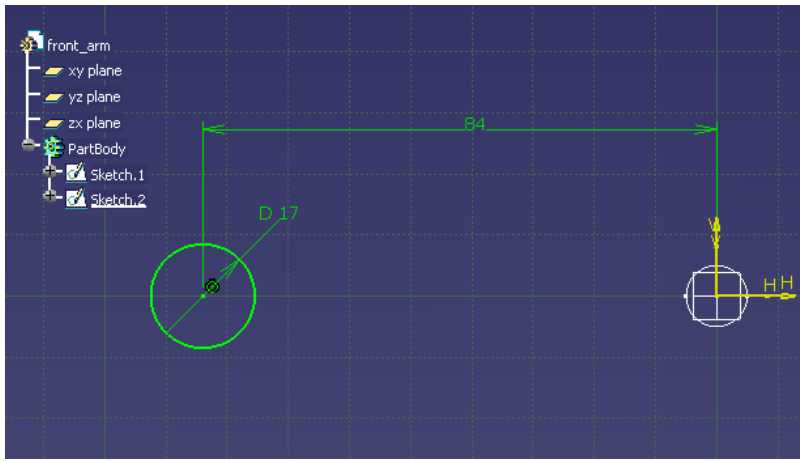
By Dickson Sham (ME dept, HKPU)

Tutorial 1A

To build 2nd sketch:-

- Click somewhere near the circle to deselect Sketch1.
- Click "**Sketch**" icon again and select **xy 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).
- Add a dimension **constraint** onto the circle and **modify** its diameter as 17mm.
- To define their distance, click "Constraint" icon and select their centers. Modify it as 84mm. (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 workbench.
- You can see Sketch.2 on the tree.



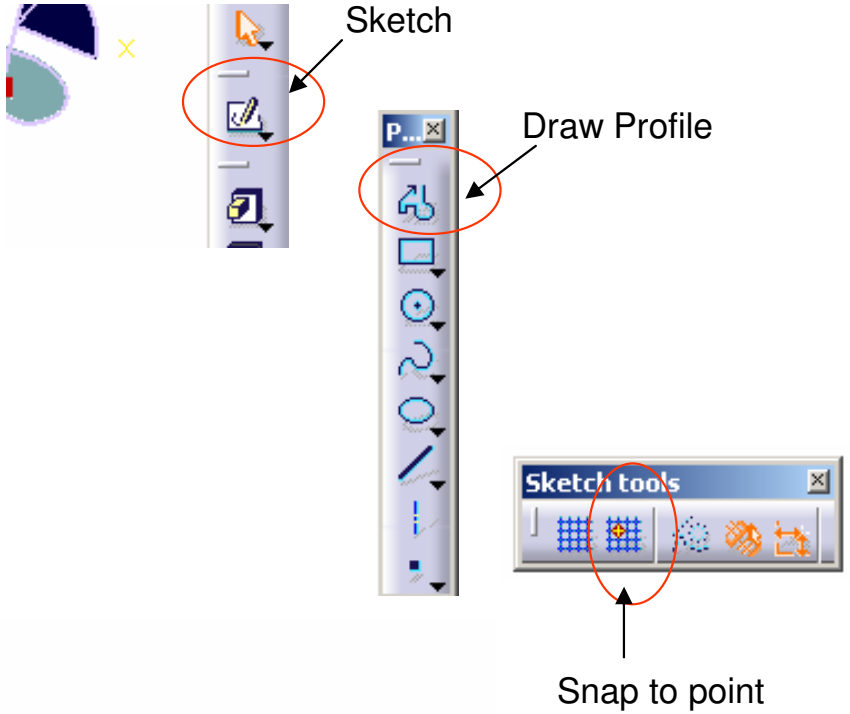


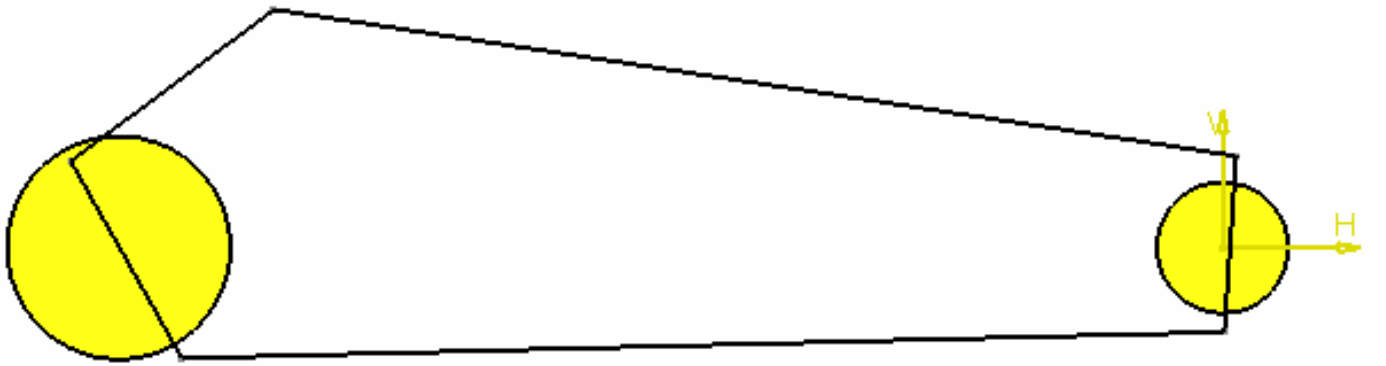
By Dickson Sham (ME dept, HKPU)

Tutorial 1A

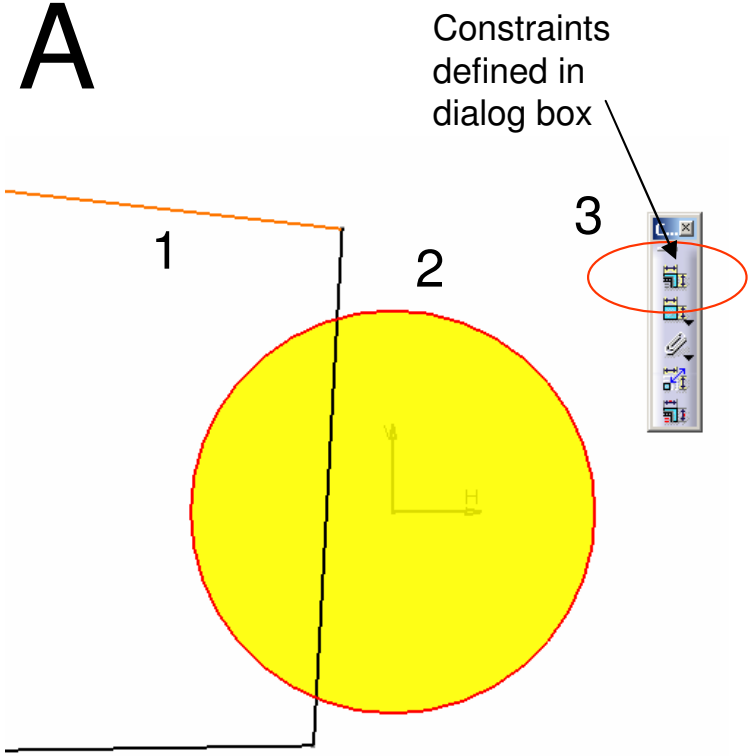
To build 3rd sketch:-

- Click somewhere near the 2nd circle to deselect Sketch2.
- Click "**Sketch**" icon and select **xy plane** again to draw another sketch.
- **Draw** a profile as below(Five straight lines forming a closed profile).
- Switch off "**Snap to Point**" so that you can draw the lines easily.





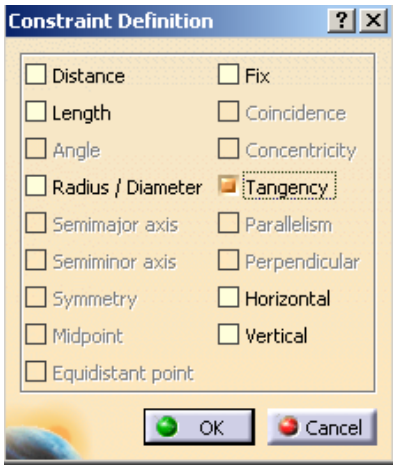
Tutorial 1A

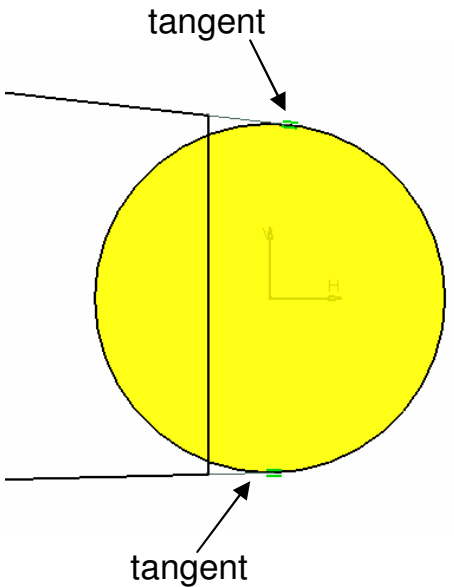


To build 3rd sketch (Cont'):-

- To ensure the lines are tangent to the small circle, we need to add a geometrical constraint:-
- **Multi-select** the line and the small circle by pressing and holding "**ctrl**" key on the keyboard.
- Then select "**Constraints defined in dialog box**" icon.
- Select "Tangency" and "ok".

- Repeat the same steps for the other line…

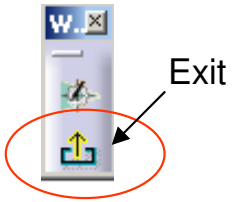


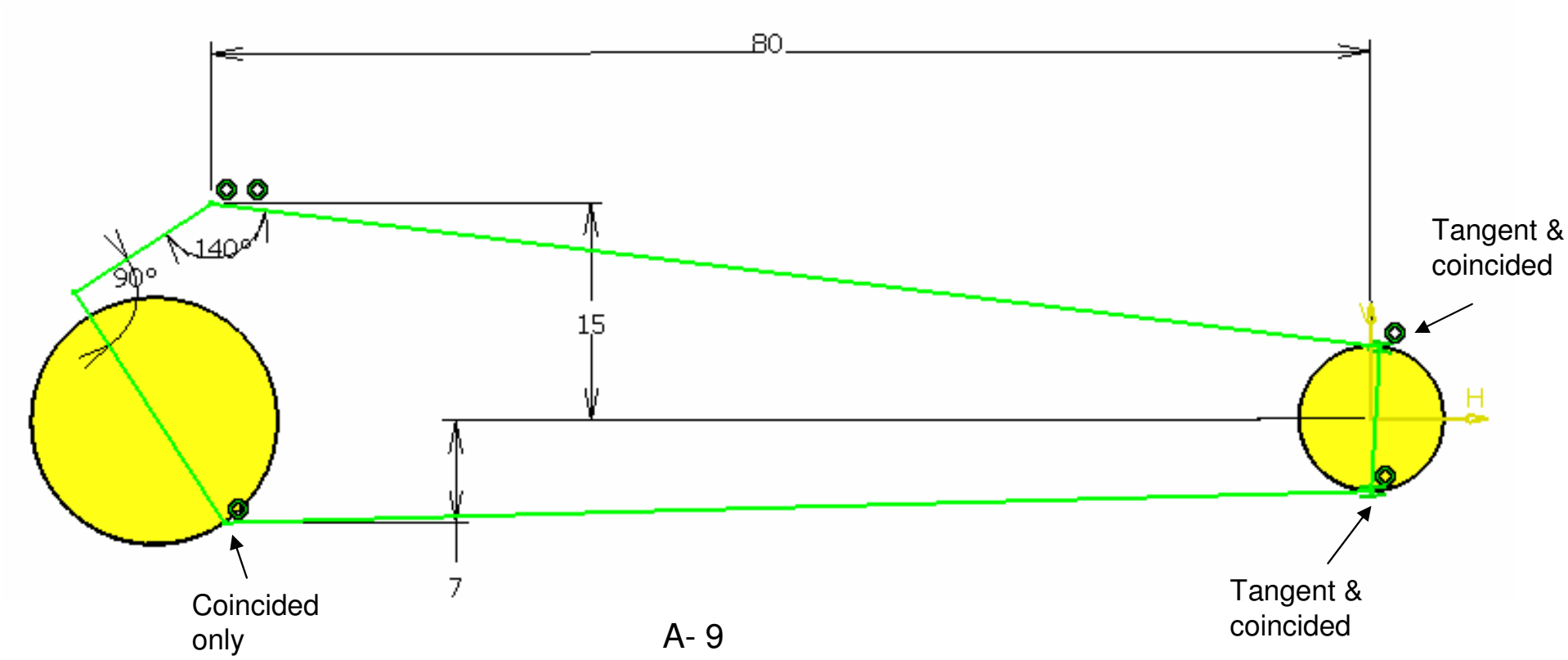


Tutorial 1A

To build 3rd sketch (Cont'):-

- Continue to add the remaining constraints until the sketch turns green, which is fully-constrained.
- Exit when it is complete.
- Now, you should see Sketch1, Sketch2 and Sketch3 on the tree.

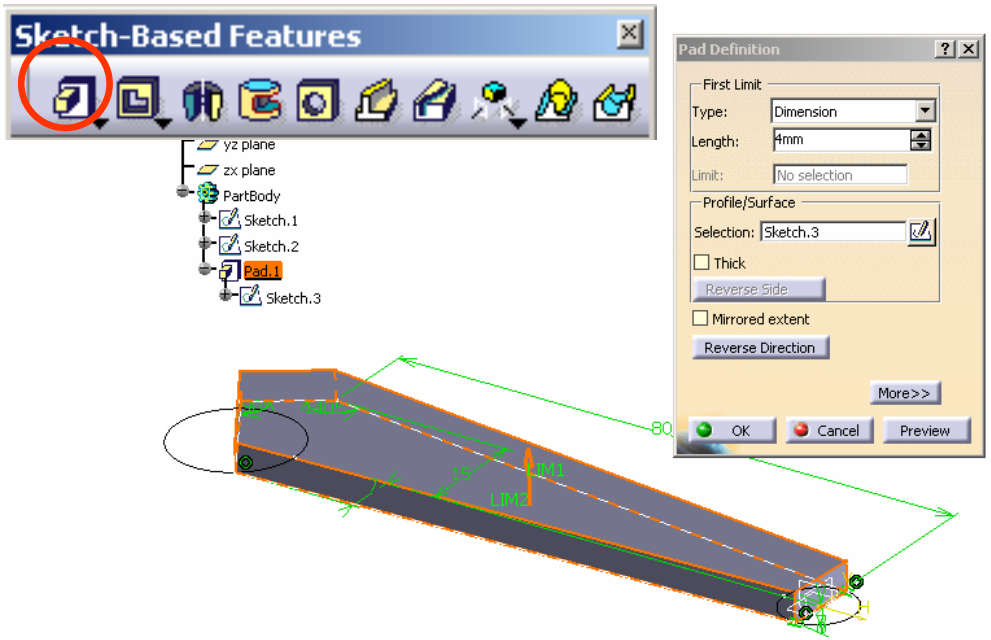




Tutorial 1A

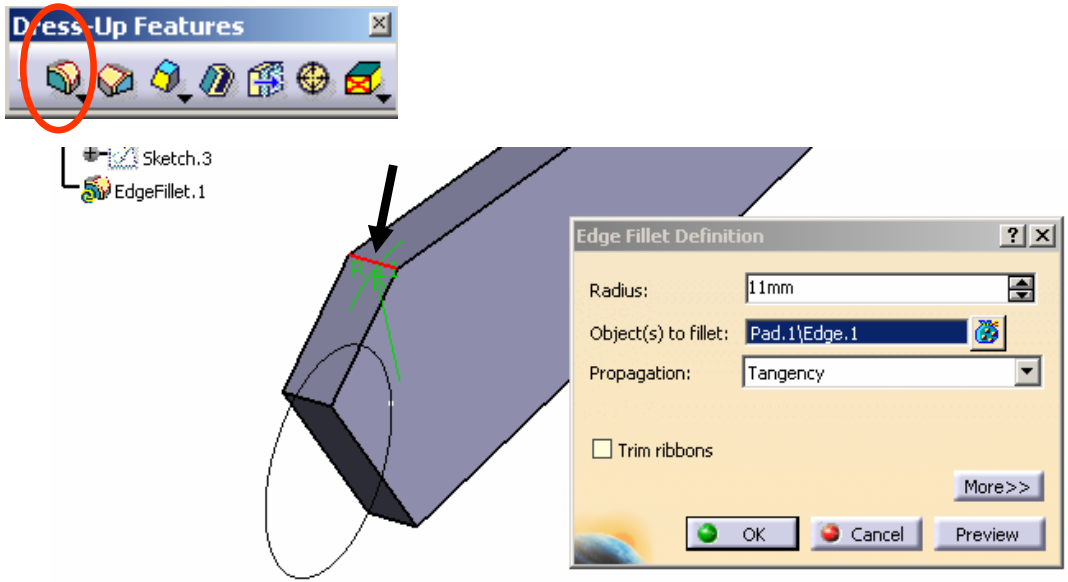
To build a solid:-

- Select “Sketch.3” on the tree / directly click on the geometry .
- Click “**Pad**” icon.
- Enter 4mm as the length of First Limit.
- Click “ok”.
- A solid is created.



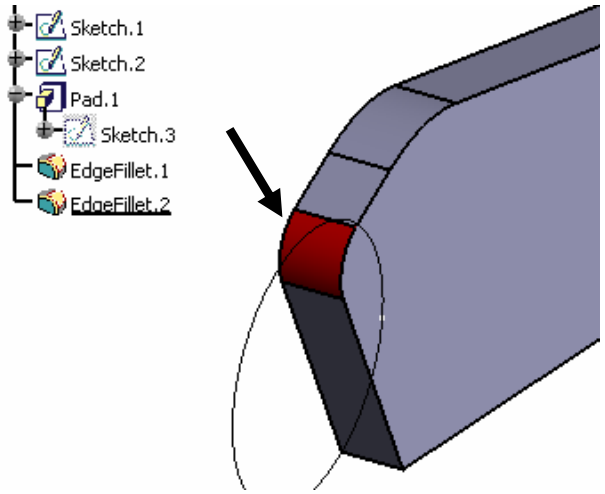
To round the sharp edge:-

- Add a “**Edge Fillet**” R11mm onto the uppermost corner of Pad1.



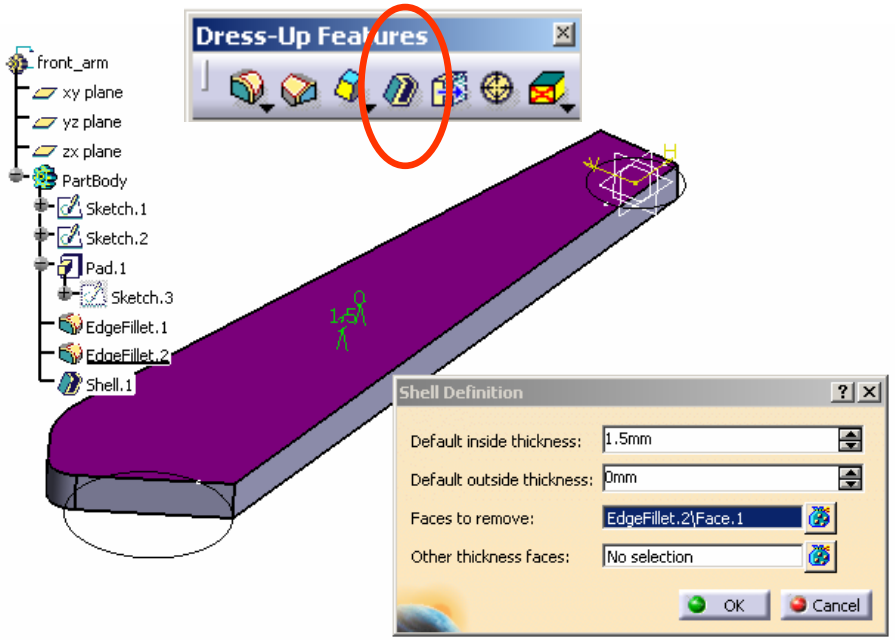
Tutorial 1A

- Add another "**Edge fillet**" R3mm.



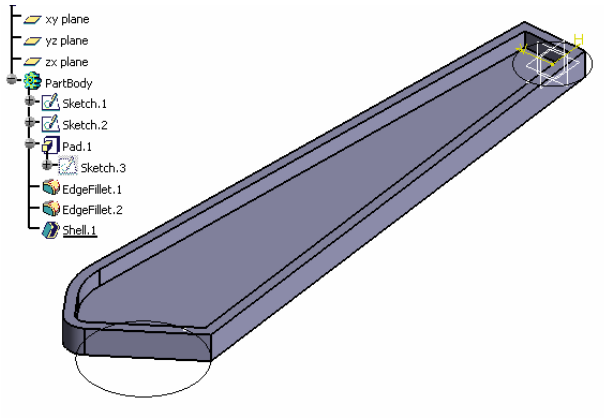
To make the solid hollow:-

- Click "**Shell**" icon.
- Enter 1.5mm as "Default inside thickness".
- Select the top surface of the solid, which is considered as "Face to remove".
- Click "ok" 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.

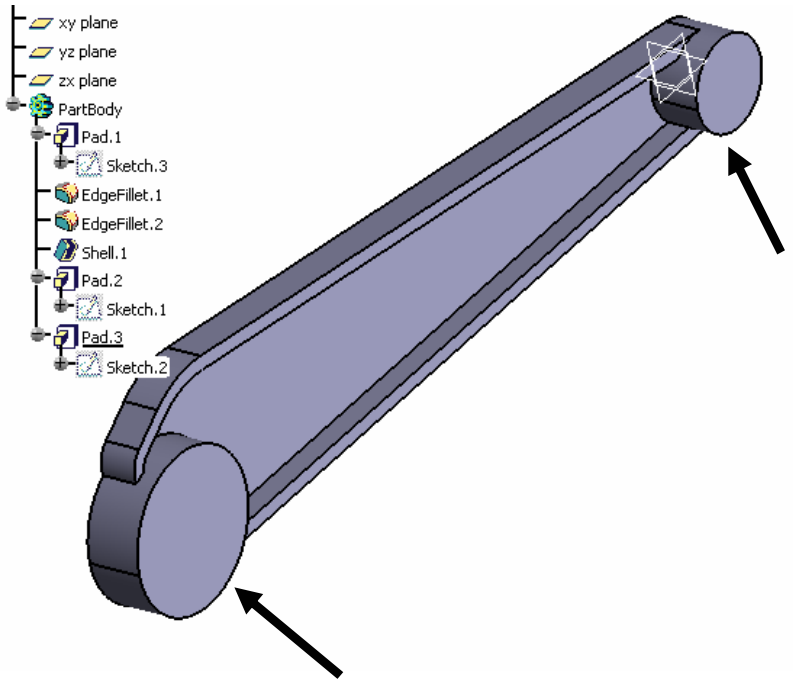


To build 2 more pads:-

- Click "**Pad**" icon.
- Select Sketch.1
- Enter 7mm as First Limit.
- Click Ok to complete.

Similarly,

- Click "**Pad**" icon.
- Select Sketch.2
- Enter 6mm as First Limit.
- Click Ok to complete.

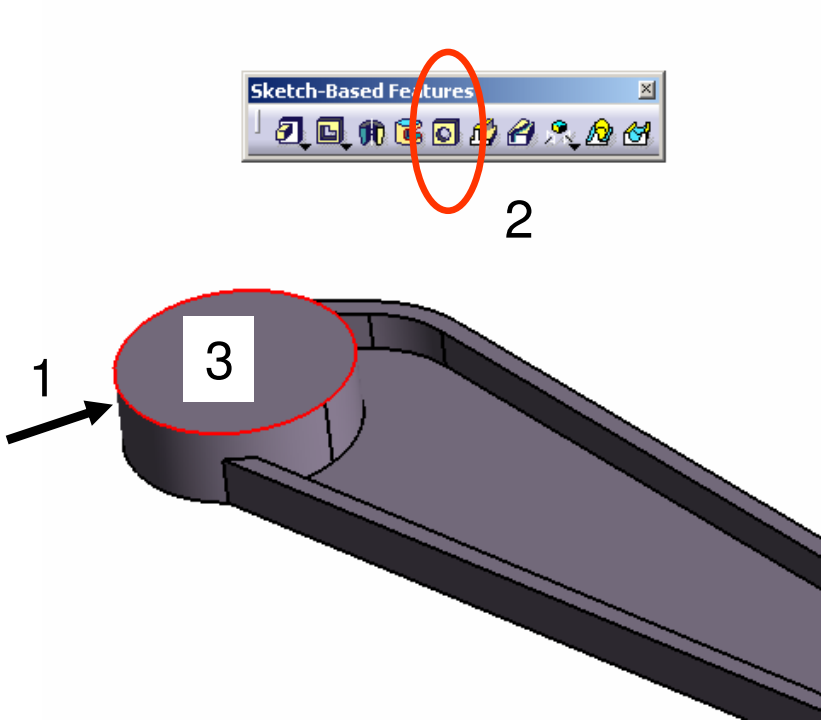


Tutorial 1A

To make a hole:-

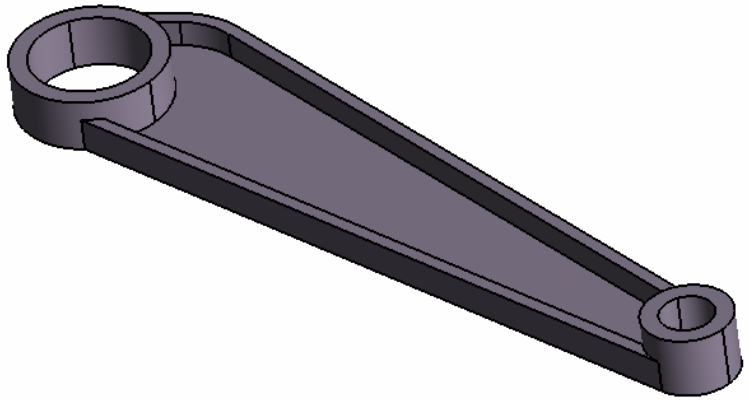
- Select the circular edge of the bigger cylinder.
- Click "**Hole**" icon.
- Select the top surface of the cylinder.

(w/ the steps, the hole and cylinder are concentric.)



- Select "Up to Last" to have an infinite depth.
- Enter 13mm as Diameter.
- Click "ok" to complete.

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

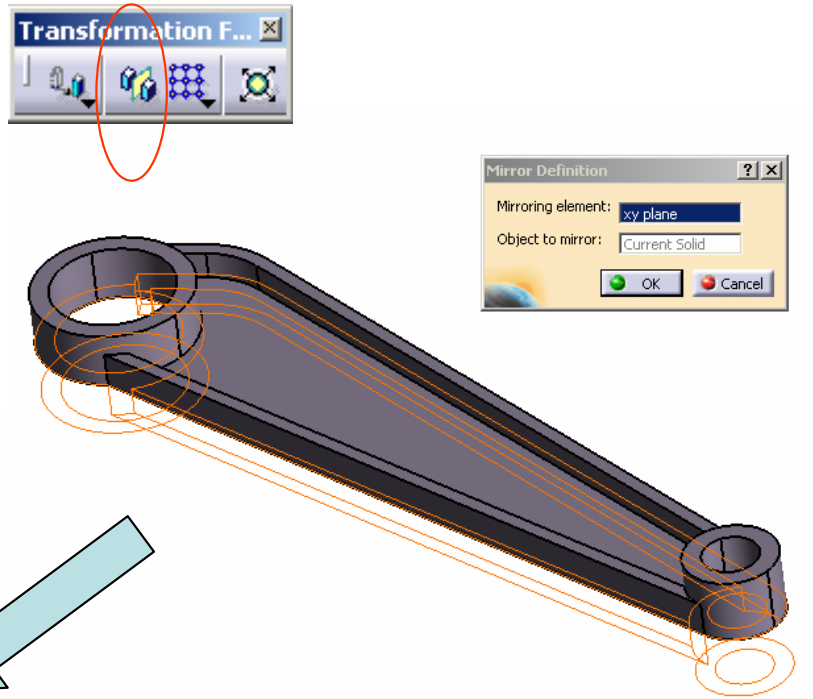


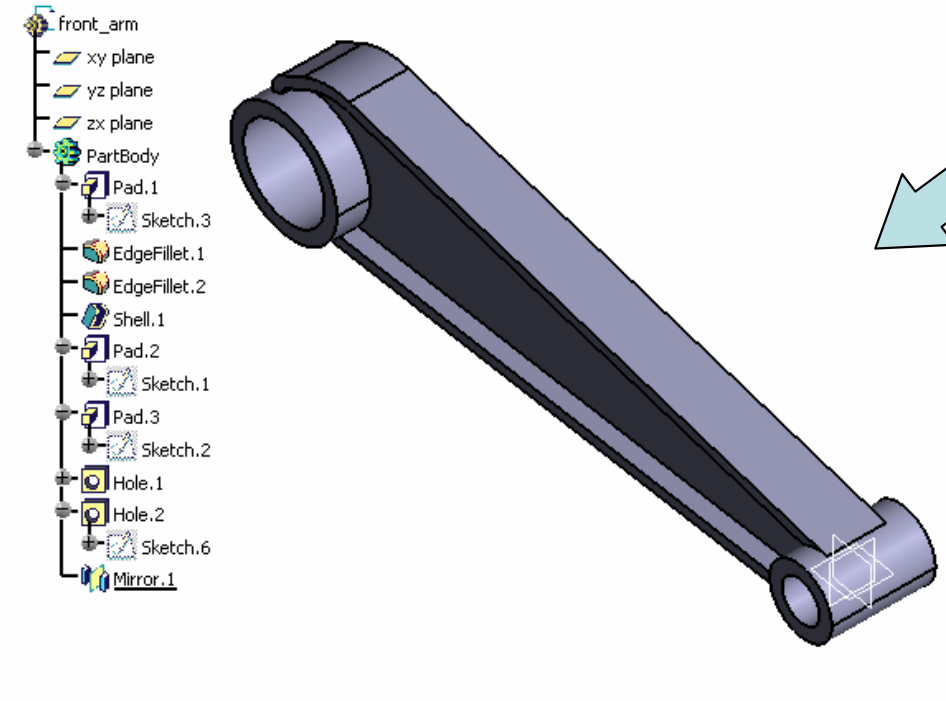
By Dickson Sham (ME dept, HKPU)

Tutorial 1A

To duplicate another half:-

- click "**Mirror**" icon and select xy plane as the mirroring element.

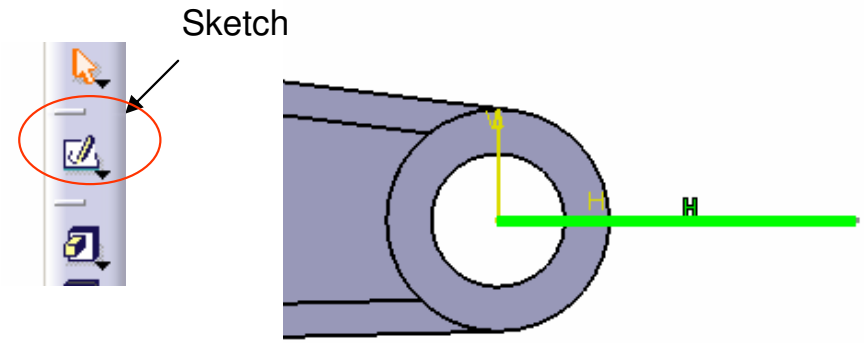




Tutorial 1A

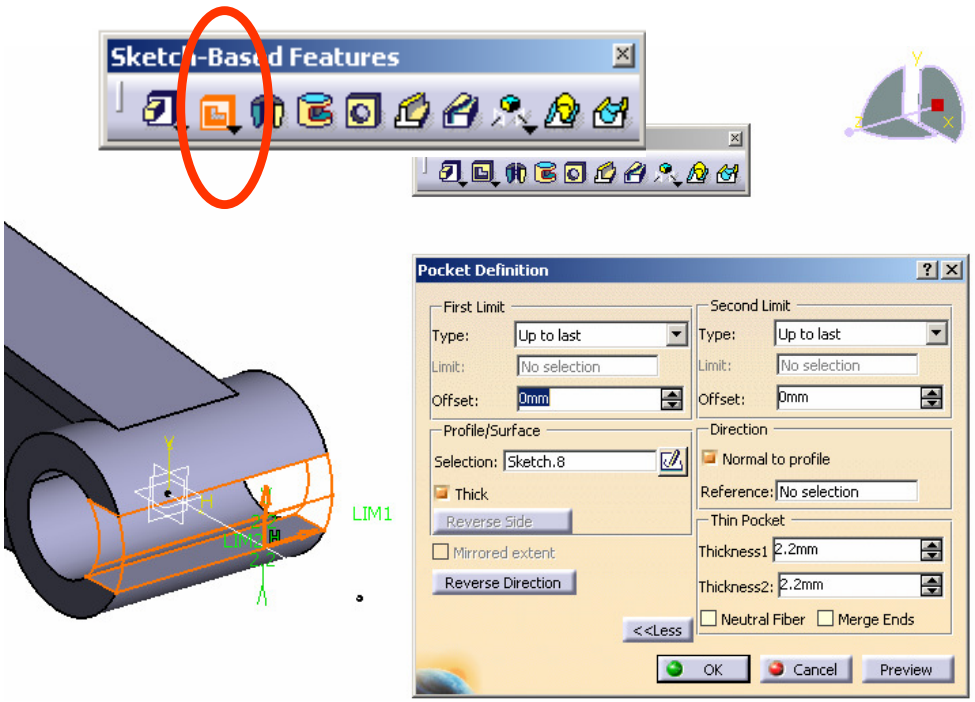
To build a sketch (**open profile**):-

- Click "**Sketch**" icon and select xy plane.
- 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" icon to exit.



To remove material with an open profile:-

- Click "**Pocket**" icon.
- Select "Thick" on the menu.
- Enter 2.2mm for both thickness1 &2.
- Select "Up to Last" for both first limit & second limit.
- Click ok to complete.

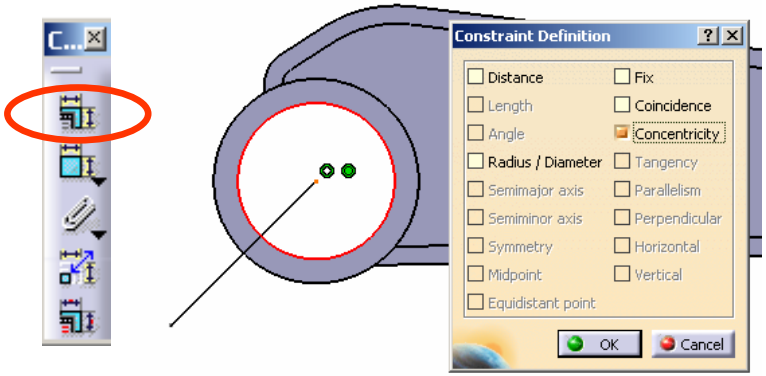


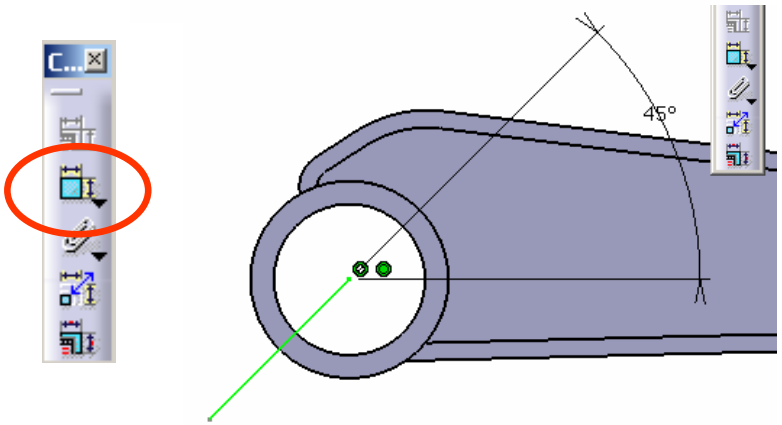
By Dickson Sham (ME dept, HKPU)

Tutorial 1A

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

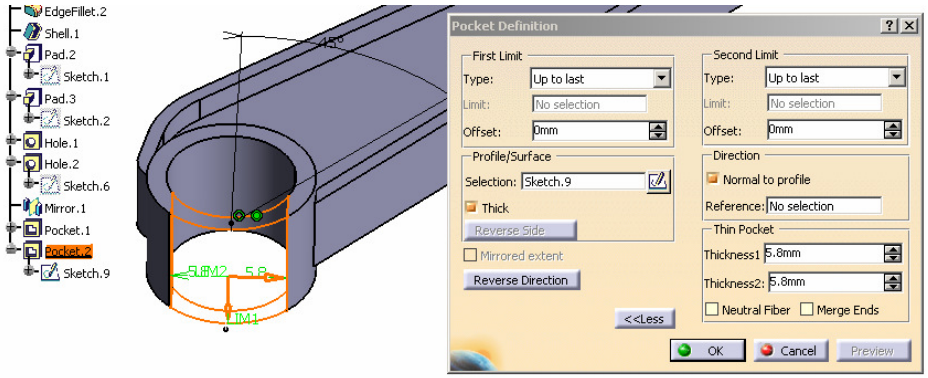
- Click "**Sketch**" icon and select xy plane.
- Draw a inclined line, w/ one end near center of the big circle and the other outside it.
- Add a concentricity constraint to ensure the endpoint is at the circle center.
- Inclined angle =45 deg from the x-axis.
- No need to specify its length.
- Click "Exit" icon to exit.





To remove material with an open profile:-

- Click "**Pocket**" icon.
- Select "Thick" on the menu.
- Enter 5.8mm for both thickness1 &2.
- Select "Up to Last" for both first limit & second limit.
- Click ok to complete.



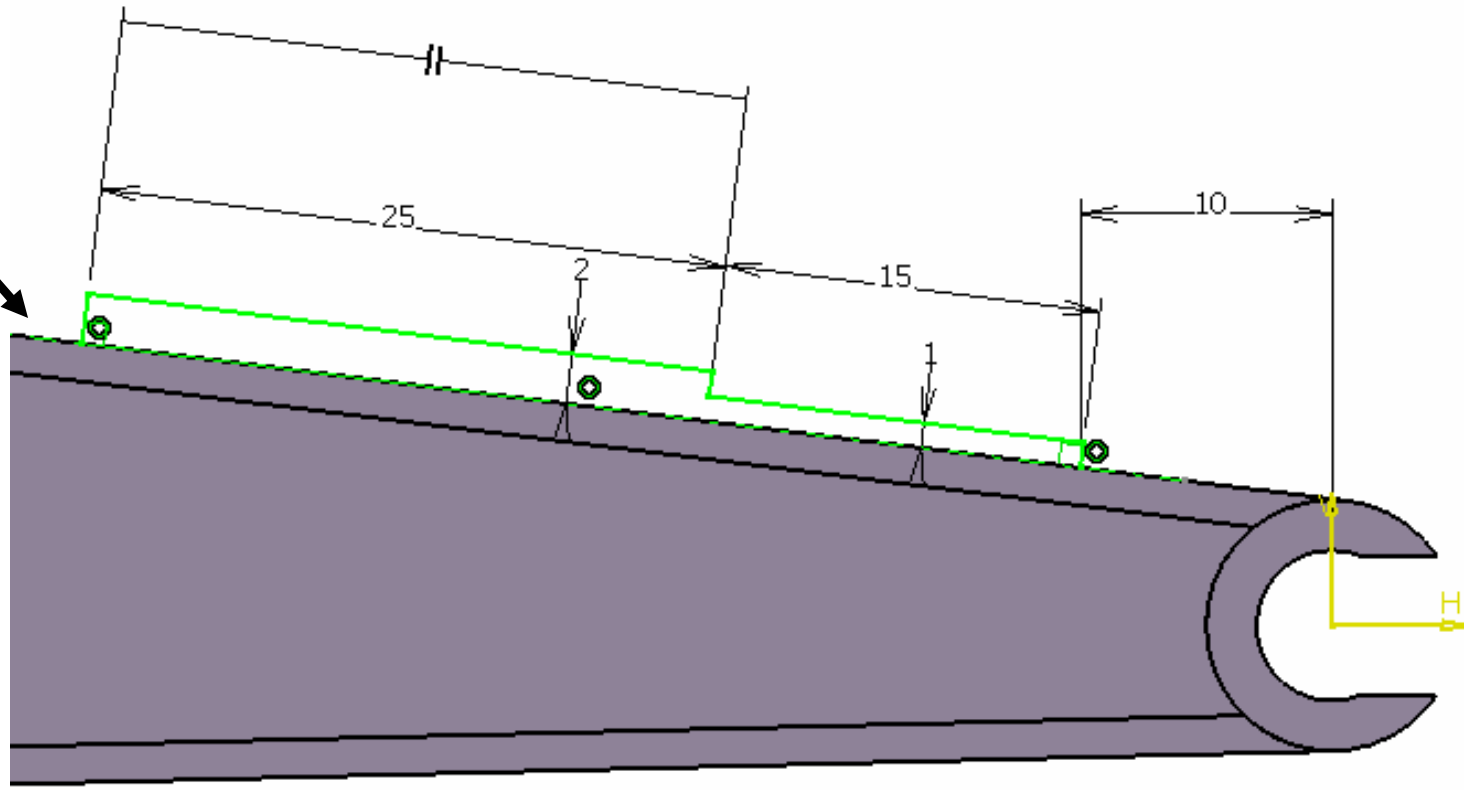
By Dickson Sham (ME dept, HKPU)

Tutorial 1A

To build a new sketch:-

- Click "**Sketch**" icon and select xy plane.
- Draw the profile as shown.
- Exit to complete.

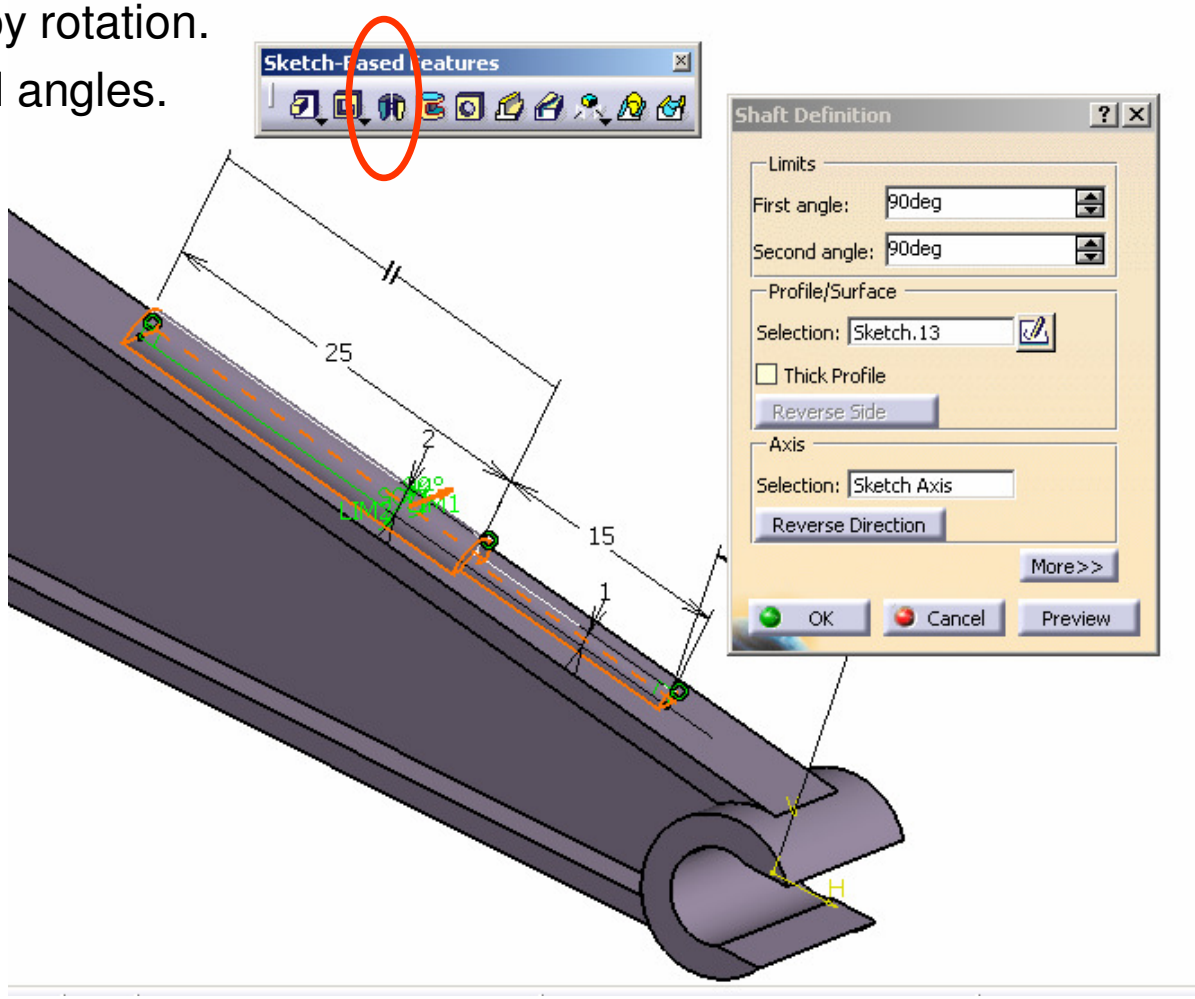
An axis is coincident with the solid surface

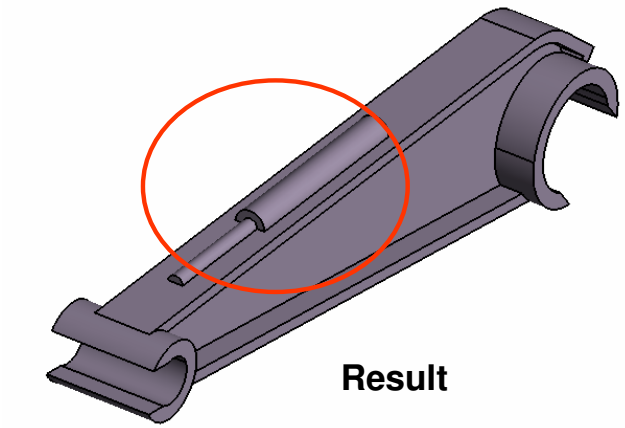


Tutorial 1A

To add material by rotating a sketch:-

- Click "**Shaft**" icon to add material by rotation.
- Enter 90deg for both first & second angles.
- Click "ok" to complete.



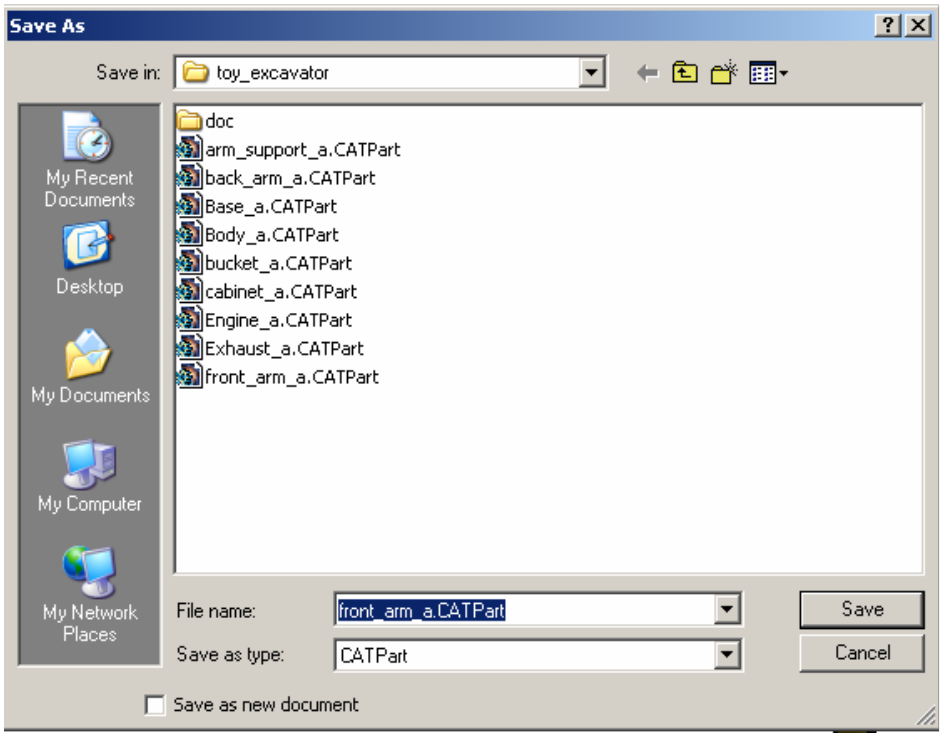


Result

Tutorial 1A

To save the new part in a Project Folder:-

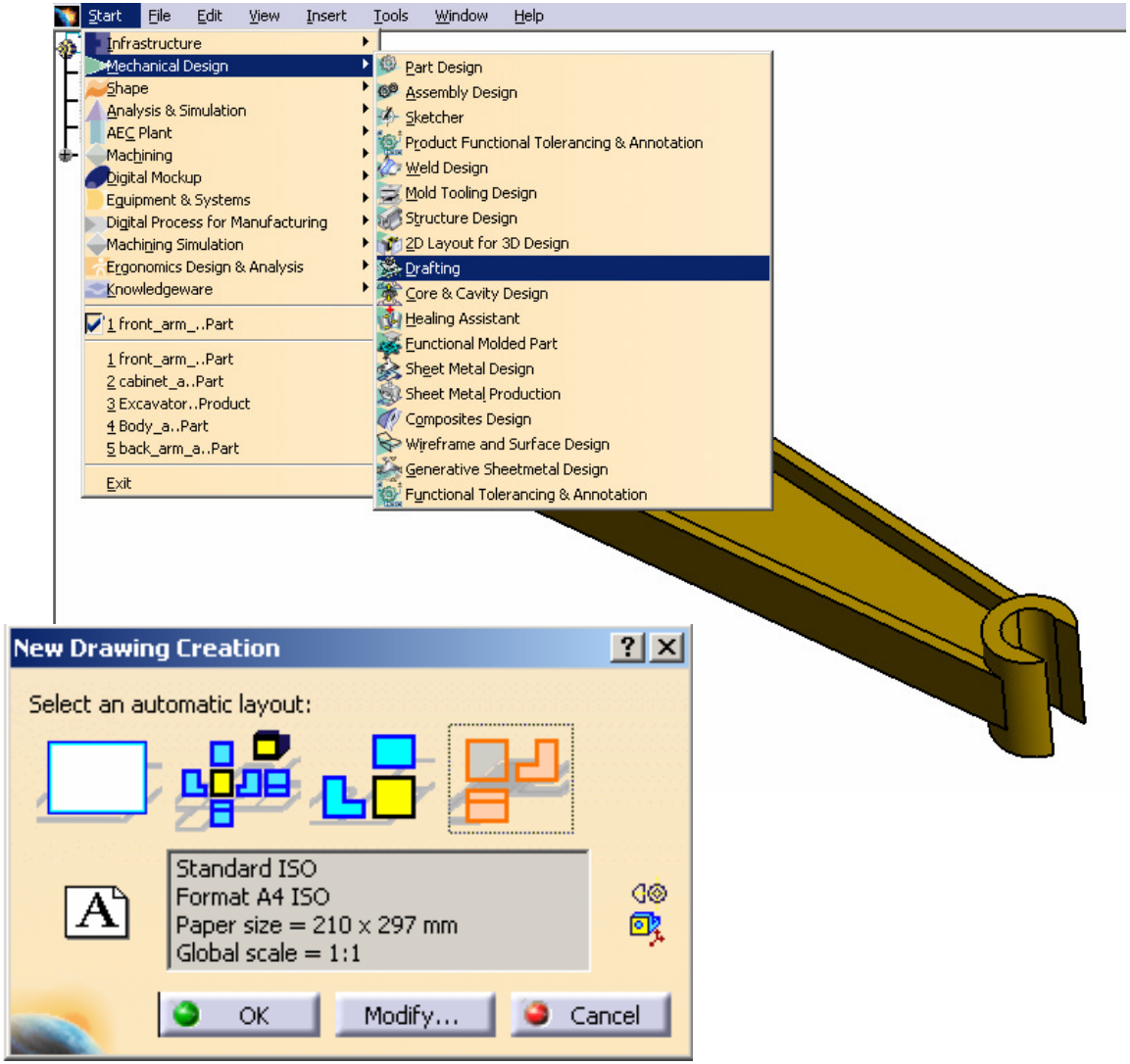
- It is a good practice to store all part files of a product in one specific folder.
- Create a folder wherever you can save (by MS window technique).
- Save your current part as "**front_arm_a.CATPART**" into the folder.
- Add "a" after its name to remind us its version. For example, I sent you the 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.



Tutorial 1A

To create a 2D drafting:-

- Select “**Start/Mechanical Design/Drafting**”.
- Select “A4 ISO” as paper format.
- Select “Front, Top, Left” as layout.
- Click “OK” to complete.

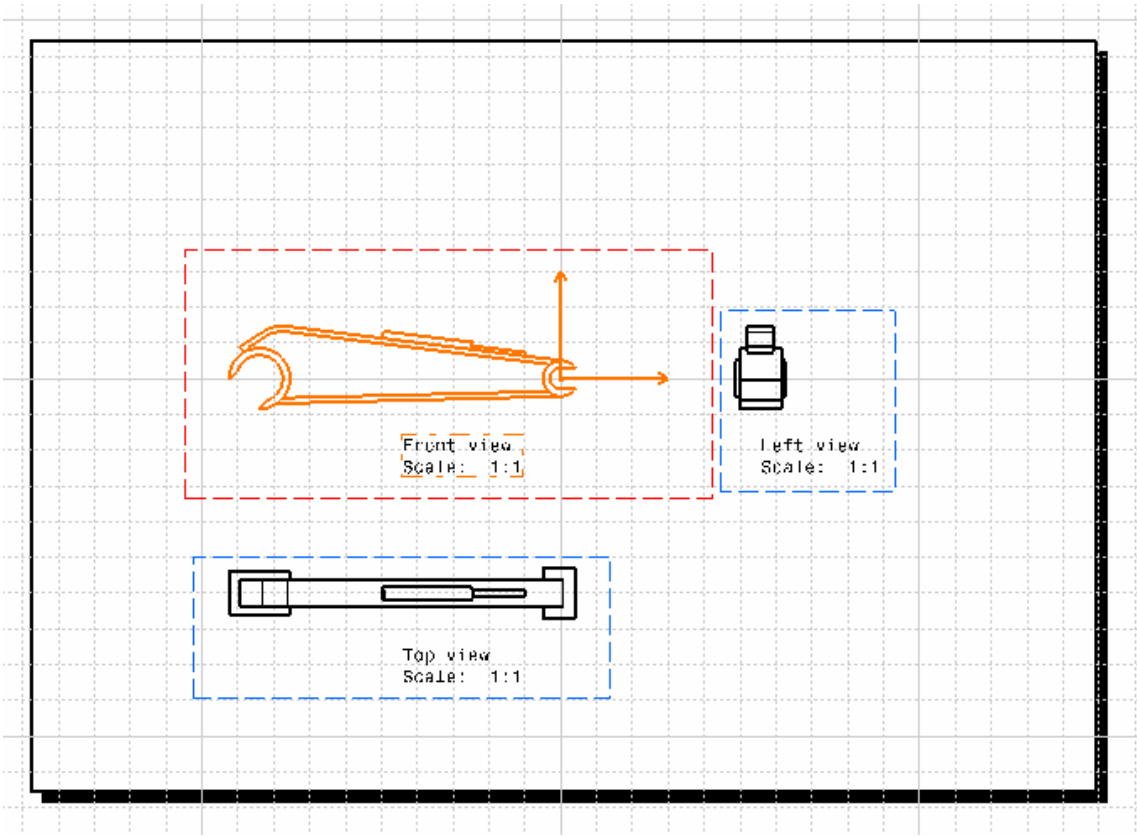


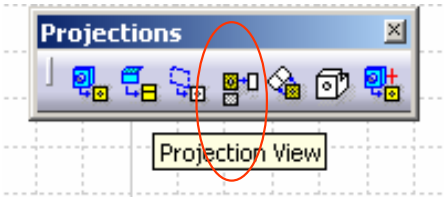
By Dickson Sham (ME dept, HKPU)

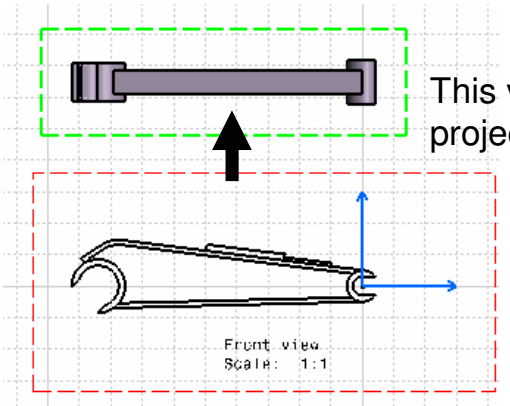
Tutorial 1A

To add & modify views:-

- Click & Drag the dotted rectangle of a view to move it to a desired position.
- You can also add additional views by clicking “Projection view” icon.





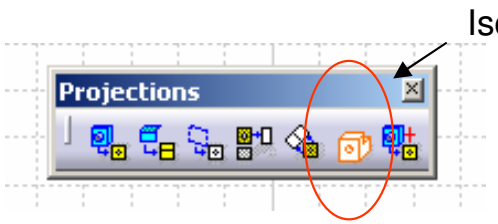


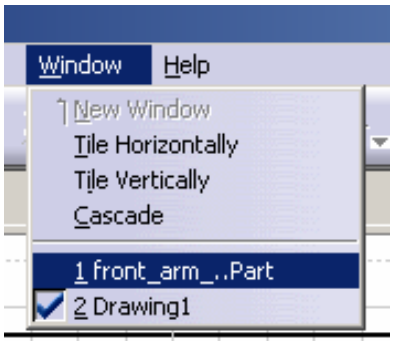
This view is created from the projection from the active view.

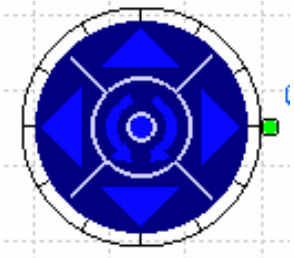
Tutorial 1A

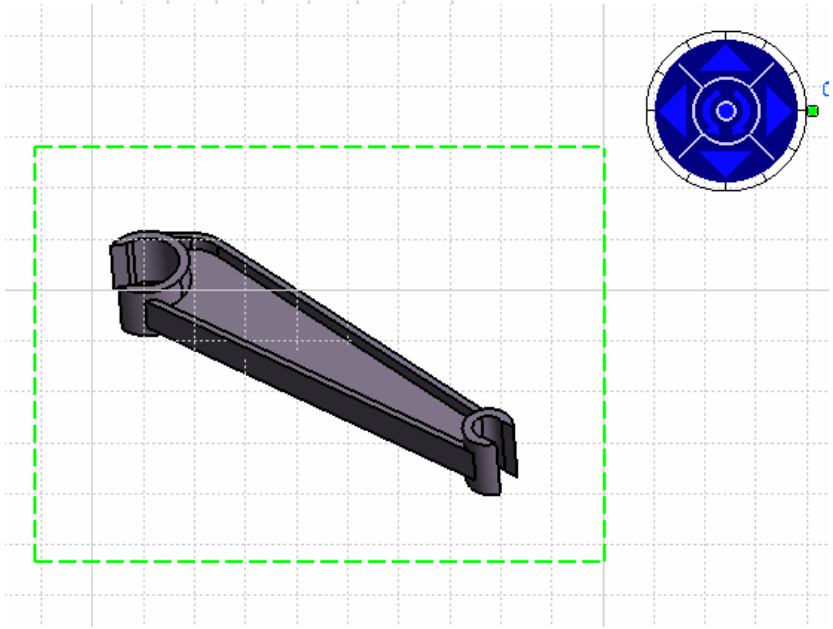
To add an isomeric view:-

- Click "**Isomeric view**" icon;
- Select "window/select/front_arm" to view the 3D part.
- Select xy plane or any other planes of the 3D part.
- Then the system will go back to the drafting mode; you will see the 3D part on the drawing and a blue circular panel at the upper right-hand corner.
- Click any button on the blue panel to select the favorite orientation.
- Click anywhere on the drawing to complete.









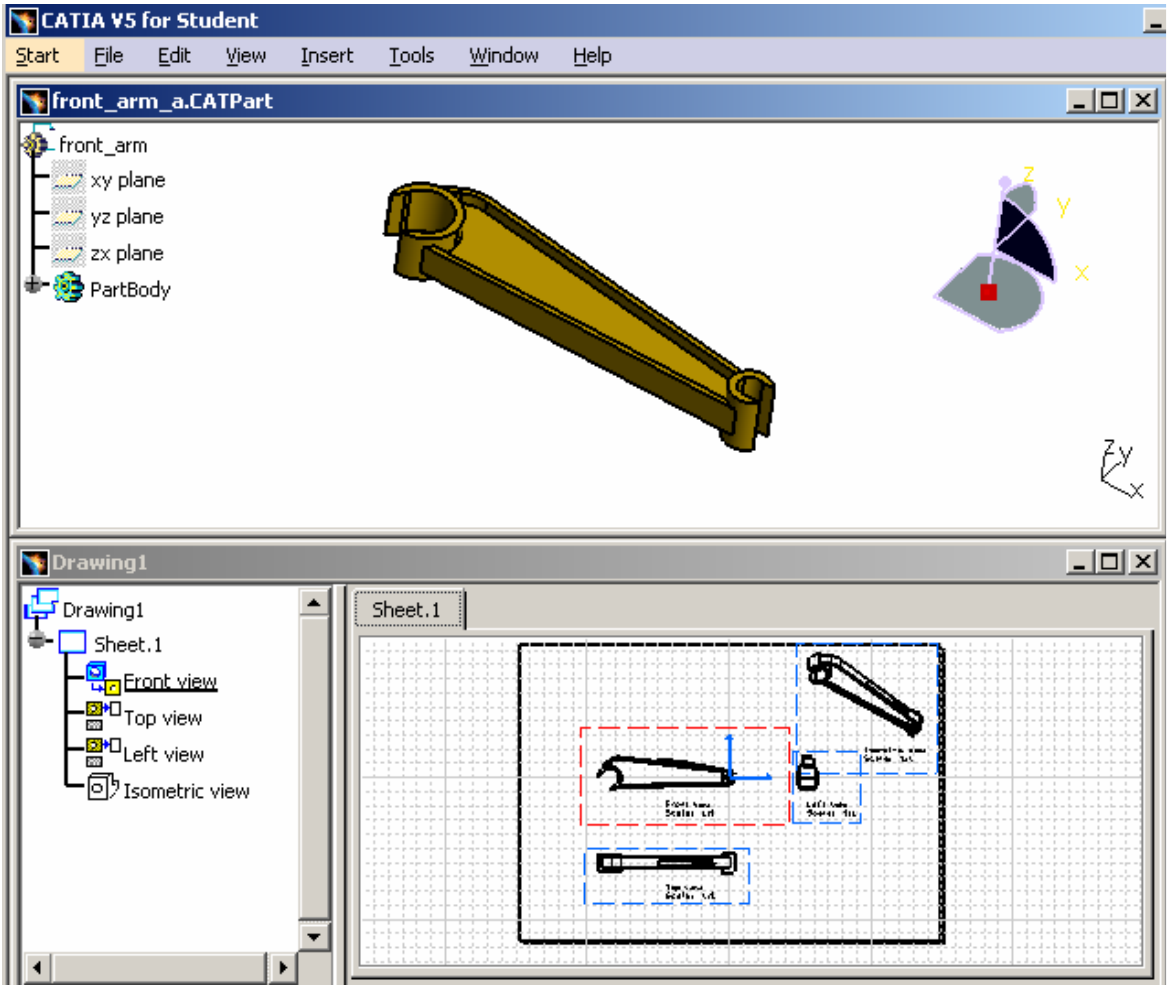
Tutorial 1A

Now you have two files:-

- Front_arm_a.CATPART
- Drawing1.CATDrawing

- 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.
- Click “Update” icon to update the drawing. 



- Close both files without saving.

END of Tutorial 1A

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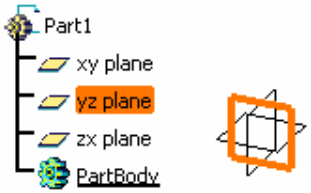
Summary of Tut-1A

Build a Sketch:-

- Click "**Sketch**" Icon

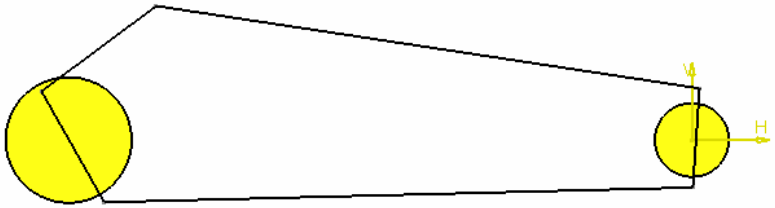


- Select a **plane**

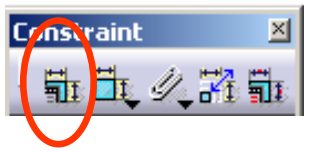


3. Draw a **profile** (with lines, curves and/or axis)





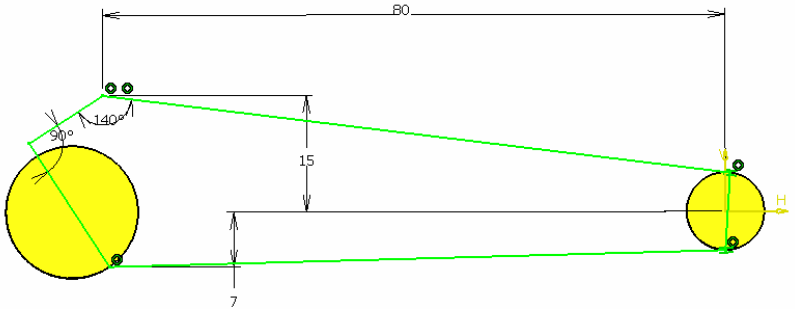
4. Add **geometrical constraints**



5. Add **dimensional constraints**

   & modify the values



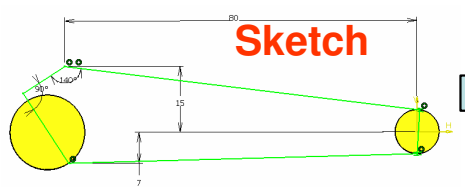


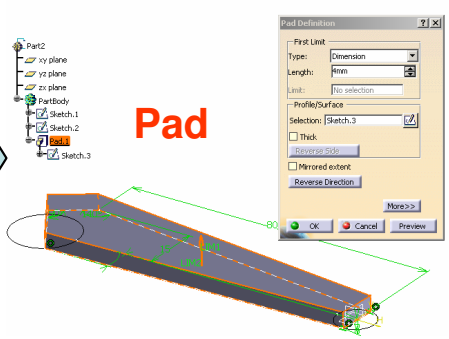
6. Click "**Exit**" icon

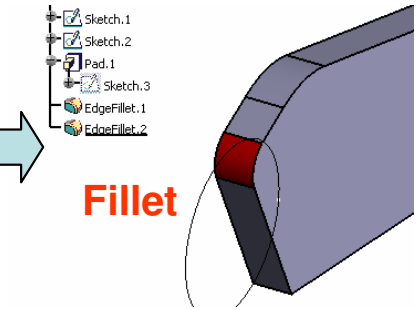


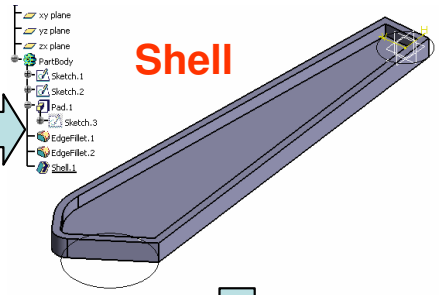
Summary of Tut-1A

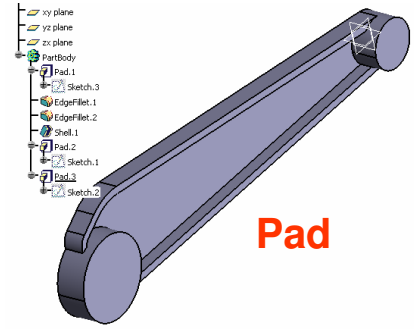
Build a Solid:-

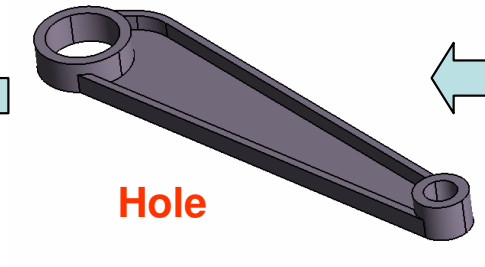


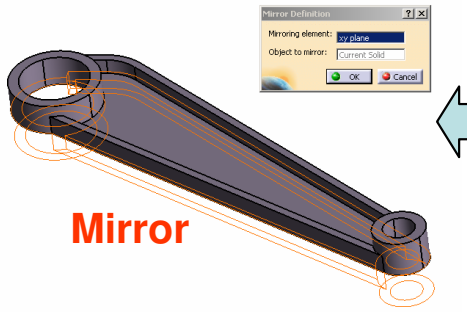


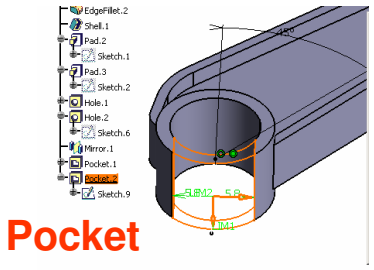


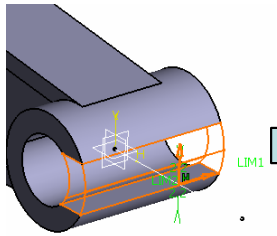


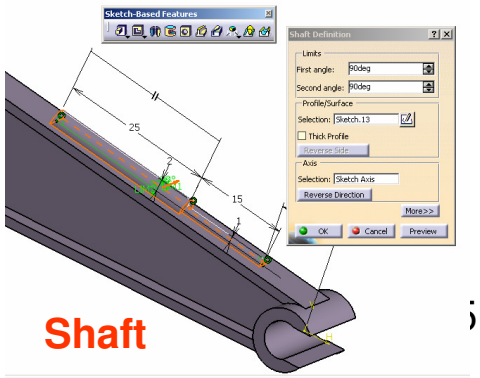


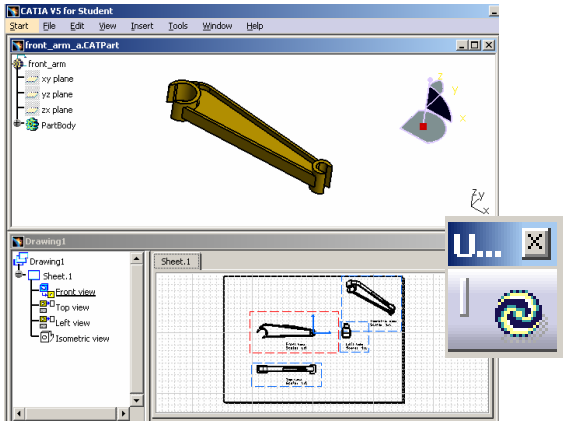








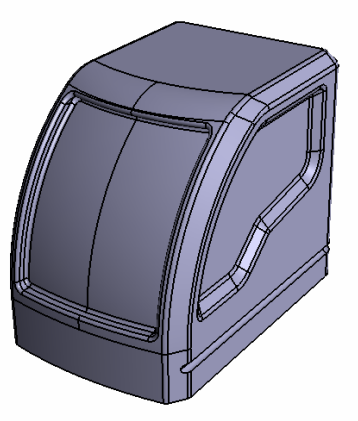




Create a 2D drawing & get drawing update after 3D change

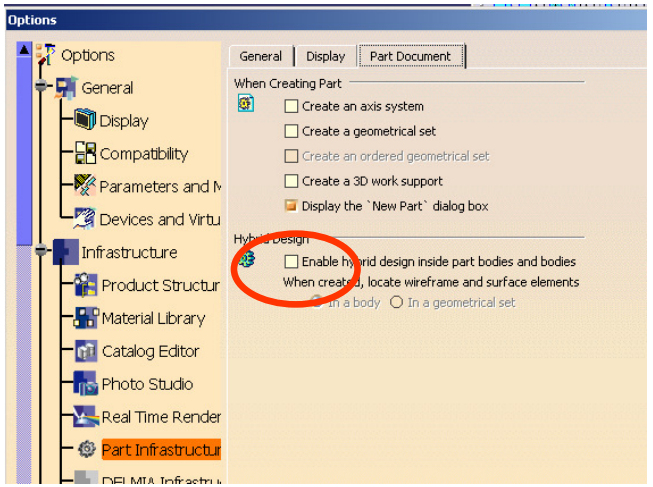
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Tutorial 1B



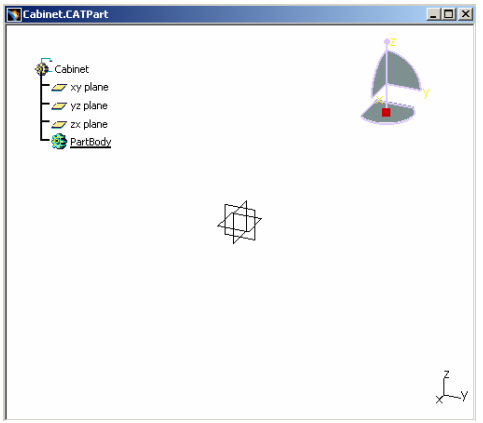
Continuing what we learnt in Tutorial 1A, we are going to build the cabinet by the solid-modeling technique plus some surface modeling technique…

- Enter CATIA.
- Close all files.
- Select '**Start/Mechanical Design/Part Design**" on the menu bar.
- If you're using Catia V5R16, uncheck "Enable Hybrid Design" and then click "ok".
- Select Tools/Options/infrastructure/Part Infrastructure… then **deselect the option "Enable Hybrid Design inside part bodies and bodies"**



To rename the tree:-

- Single-click "Part1" on the tree, right-click it, and then select "Properties".
- Modify Part Number as "cabinet" on the tab page "Product".
- Select "ok" to exit .

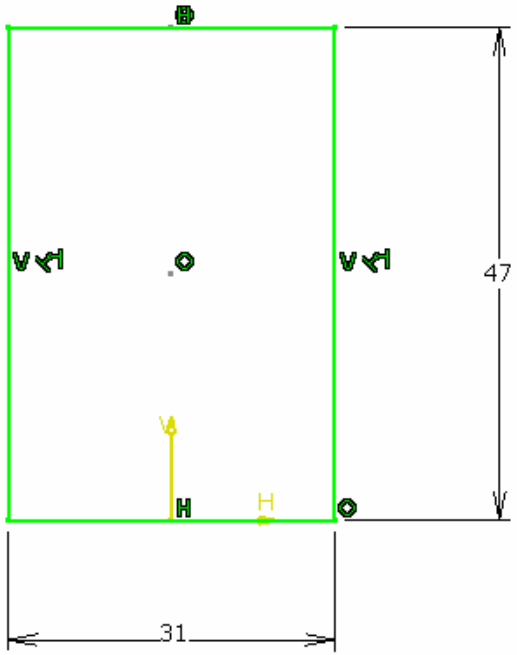


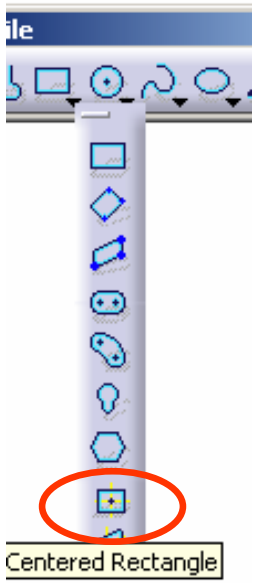
By Dickson Sham (ME dept, HKPU)

Tutorial 1B

To build a sketch:-

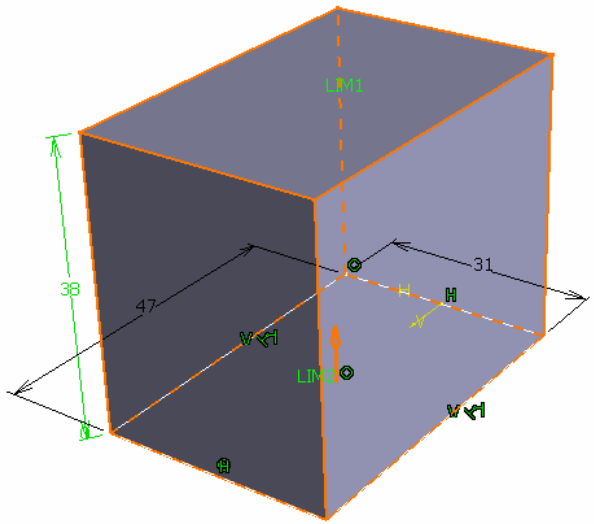
- Click "**Sketch**" icon and select **XY plane.**
- Draw a rectangle (47x31) as shown; the centre aligned on y-axis & one edge aligned on x-axis (you need to add a symmetry constraint/ or you may use "centered rectangle")
- Exit to complete.

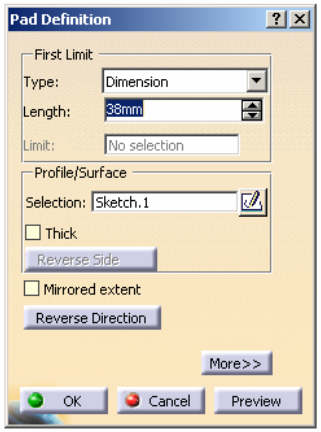




To build a solid:-

- Click "**Pad**" icon.
- Enter 38mm as First Limit.
- Click ok to complete.

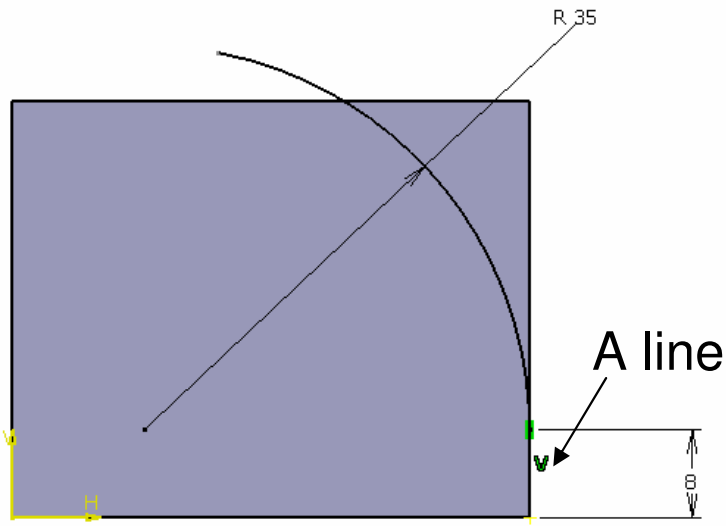




Tutorial 1B

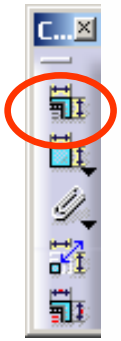
To build 2nd sketch:-

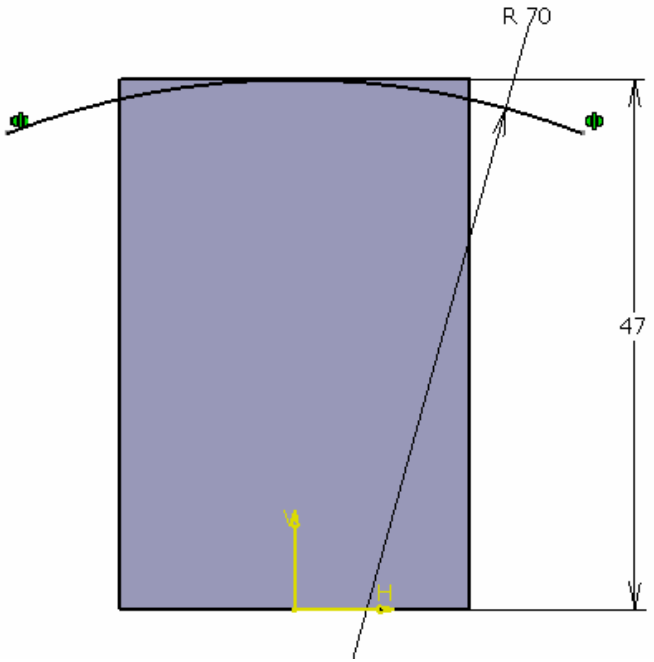
- Click "**Sketch**" icon and select **YZ plane.**
- Draw an arc R35 & a line as shown; They are tangent to each other; The line is aligned onto the solid edge and one endpoint touches x-axis.
- Exit to complete.
- Click the open area near the solid to deselect Sketch.2



To build 3rd sketch:-

- Click "**Sketch**" icon and select **XY plane.**
- Draw an arc R70 as shown; The endpoints should be symmetric about the y-axis (while pressing "ctrl" on keyboard, select both endpoints then the y-axis, then click "constraints defined in dialog box" icon)
- Exit to complete.
- Click the open area near the solid to deselect Sketch.3



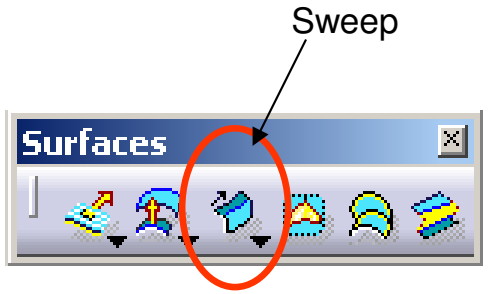


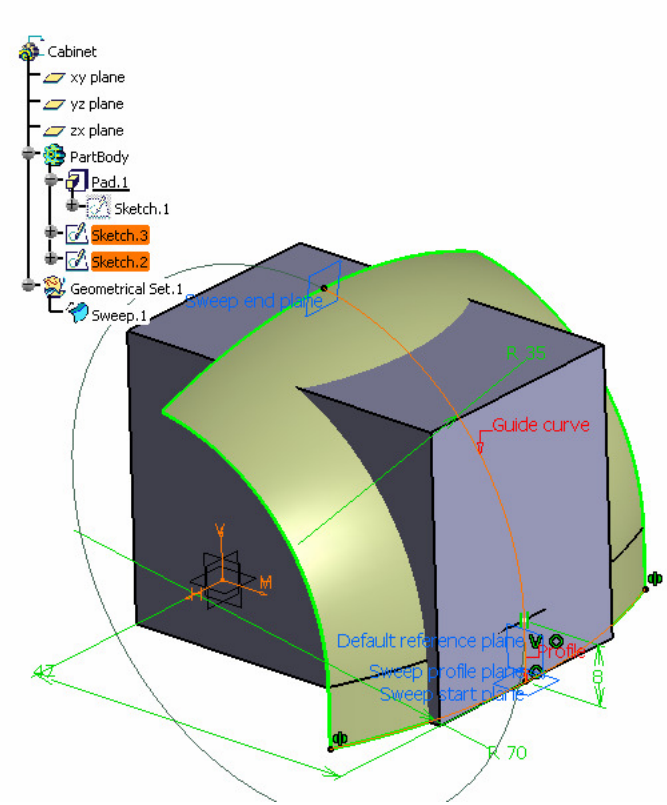
Tutorial 1B

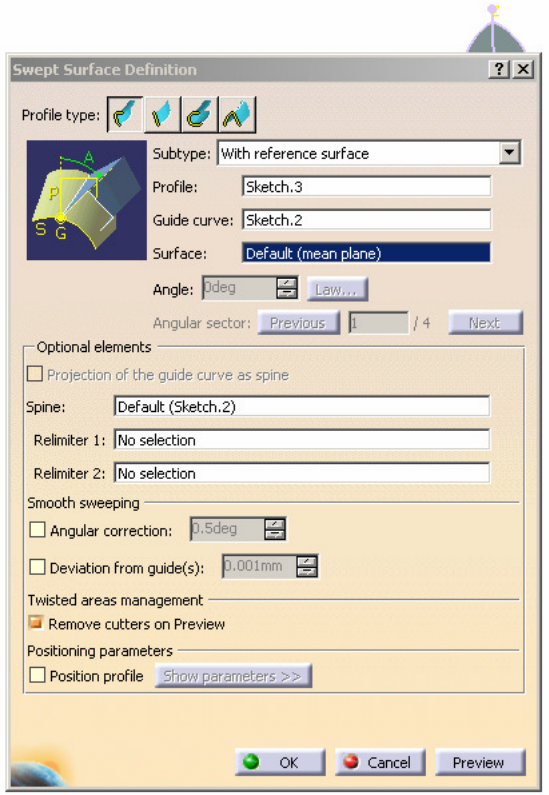
To build a SURFACE:-

- Select '**Start/Shape/Generative Shape Design**" on the menu bar; and now we are moved to a surface-modeling workbench.
- If necessary, reset the layout to make it tidy.
- Click "**Sweep**" icon
- Select "Explicit" as Profile Type
- Select "Sketch.3" as Profile
- Select "Sketch.2" as Guided Curve
- Click ok to complete

On the tree, this surface is stored in "Geometrical Set.1", so it will not be mixed with solids.



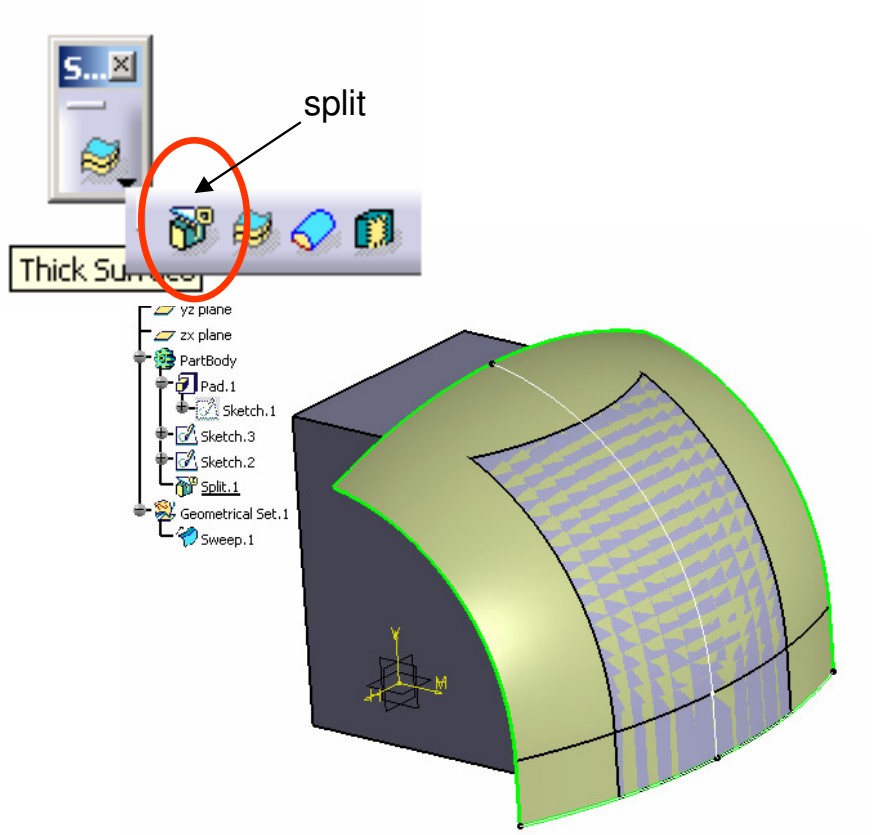




Tutorial 1B

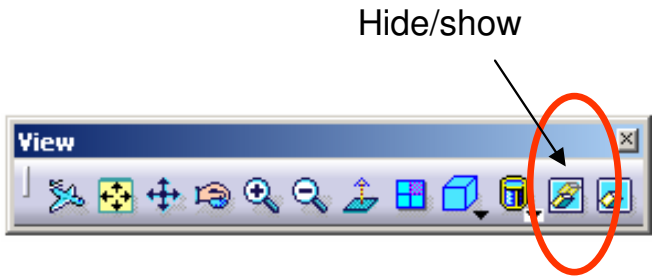
To cut the solid with this SURFACE:-

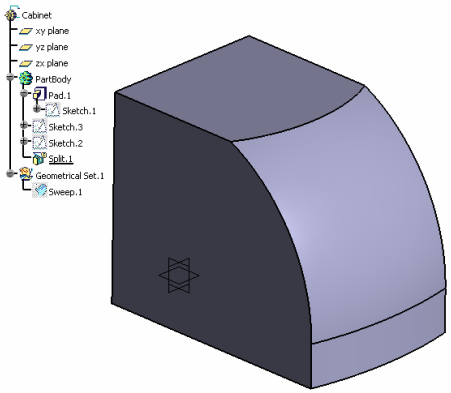
- Select '**Start/Mechanical Design/Part Design"** on the menu bar to go back to the solid-modeling environment.
- Click "**Split**" icon.
- Click OK on the warning window.
- Select the Yellow Surface "Sweep.1"
- Click on the arrow if it is pointing outwards.
- Click ok to complete



To hide the surface & its curves:-

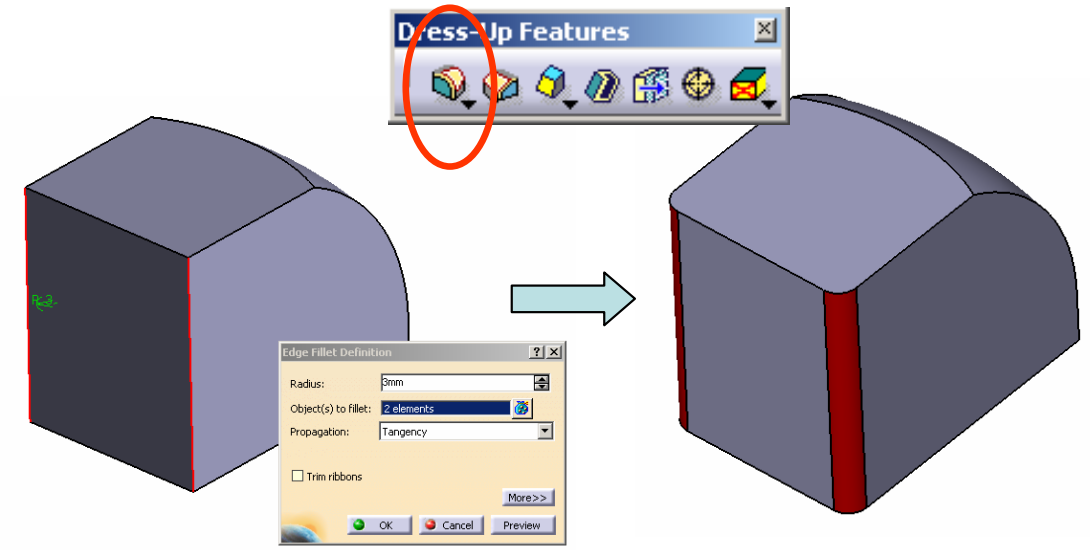
- Select the surface and click "hide/show" icon.
- Hide Sketch.2 & Sketch.3 too.





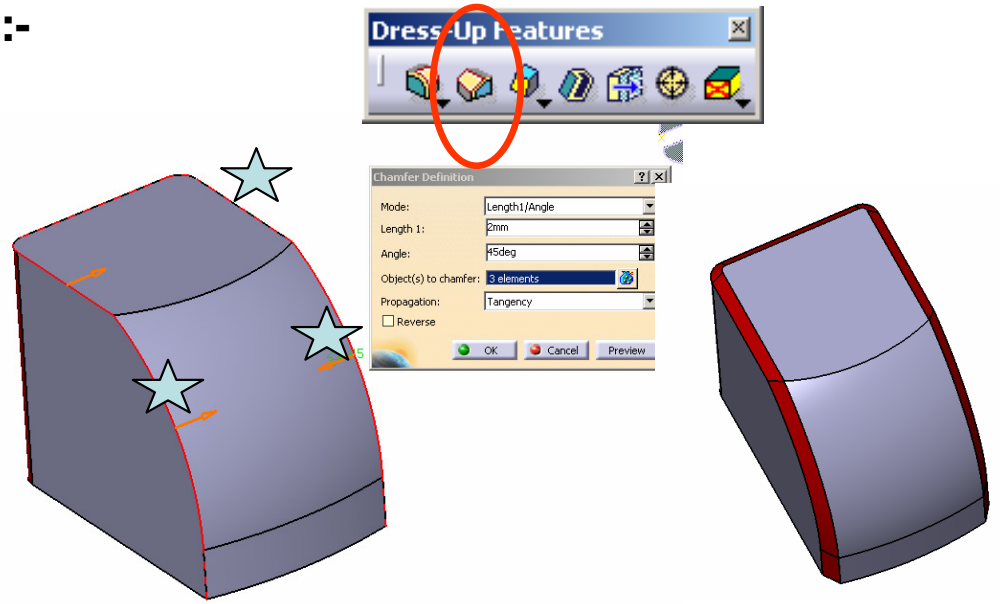
Tutorial 1B

Add a "Edge Fillet" R3mm as shown.



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

- Select "Length1/Angle" as mode.
- Enter 2mm as Length1.
- Enter 45deg as Angle.
- Select "Tangency" as Propagation.
- Click 3 edges at ☆
- Click ok to complete.

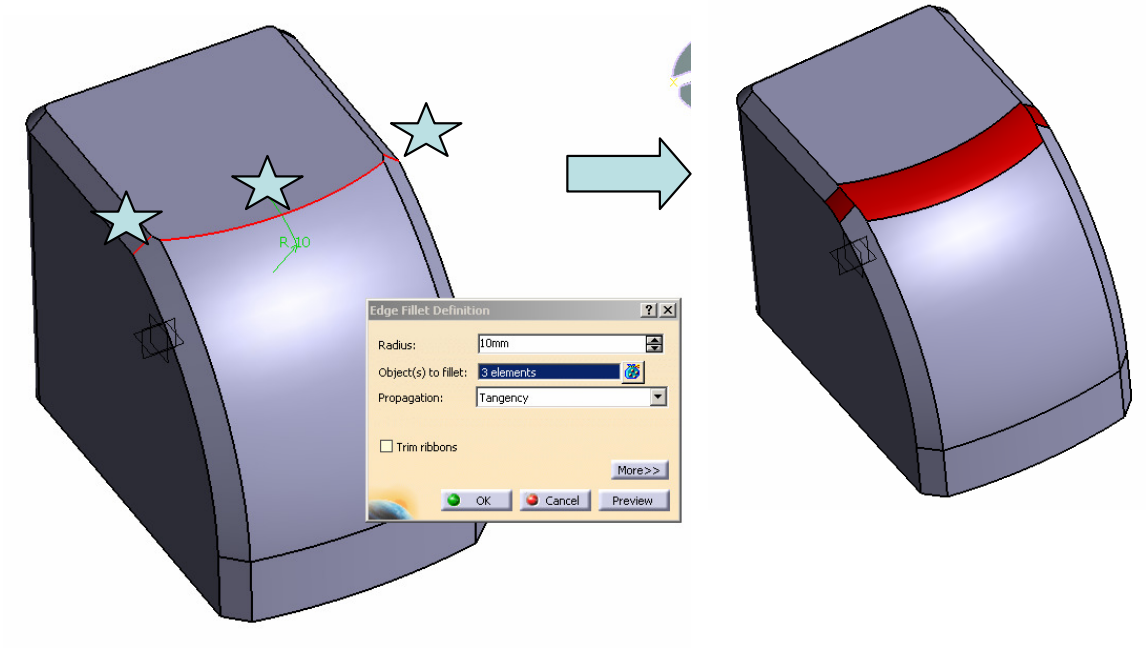


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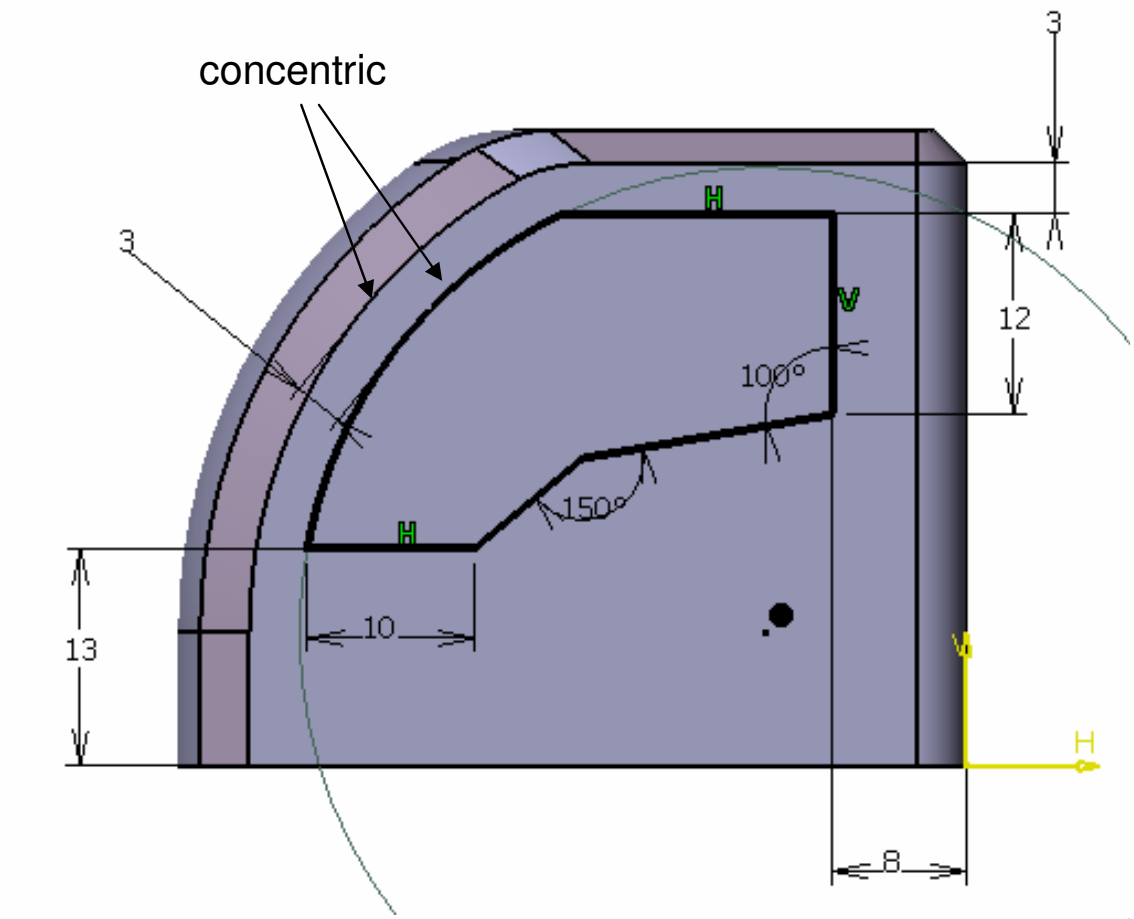
Tutorial 1B

Add a "Edge Fillet" <u>R10mm</u> on the 3 edges at ☆





Tutorial 1B



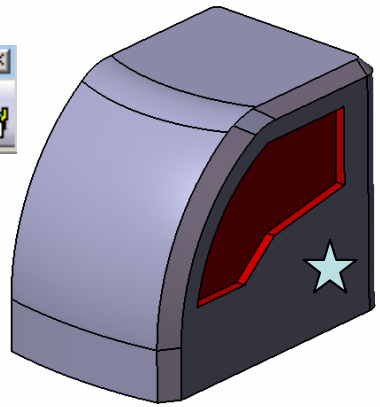
To build a sketch:-

- Click "**Sketch**" and select **Plane** ☆ **.**
- Draw a profile as shown; The profile must be fully-constrained.
- Exit to complete.

To make a pocket:-

- Click "**Pocket**" icon.
- Enter <u>1.5mm</u> as First Limit.
- Click ok to complete.

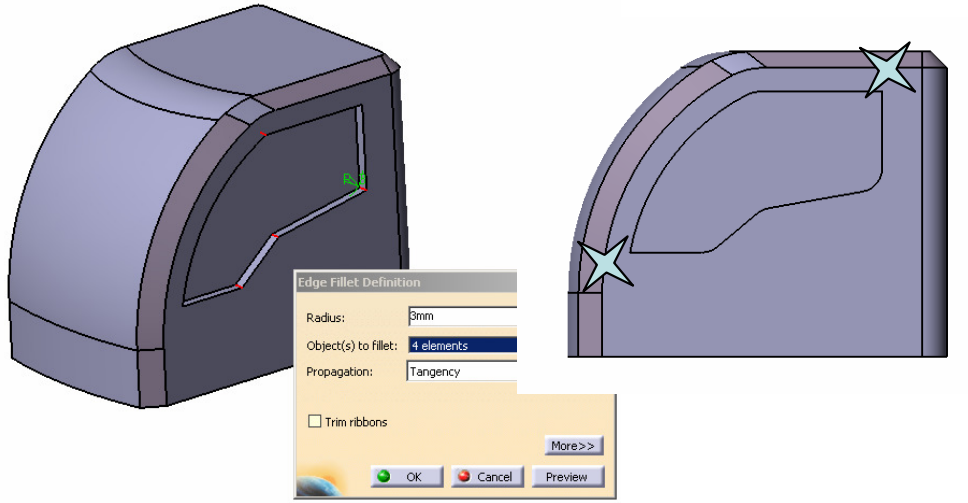




Tutorial 1B

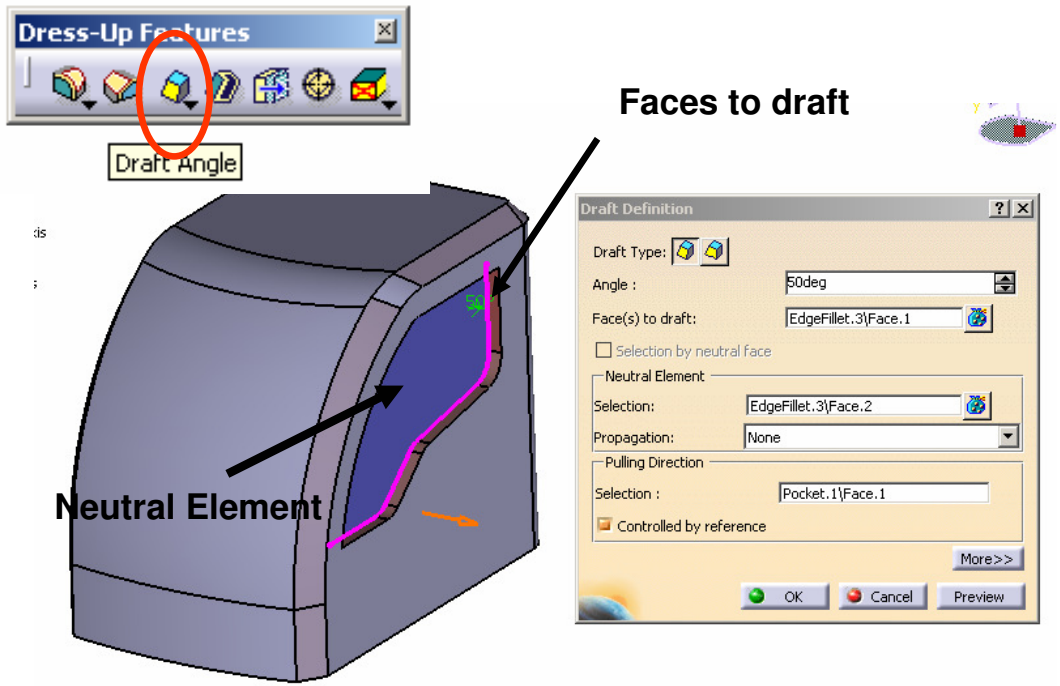
Add "Edge Fillet" <u>R3mm</u> on the four edges of the Pocket:-

- Do not add fillets at ✕ positions.



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

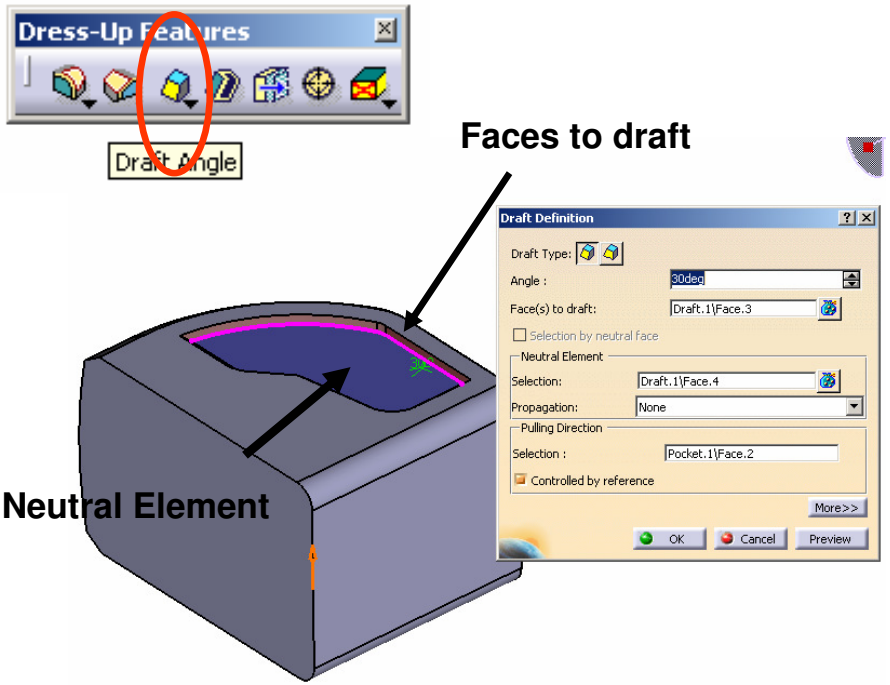
- Click "**Draft Angle**" icon.
- Select "Constant" as Draft Type.
- Enter <u>50deg</u> as Angle.
- Select the lower side face as "Faces to draft".
- Select the bottom face as "Neutral Element".
- Click the arrow once if it is not pointing outward.
- Click ok to complete.



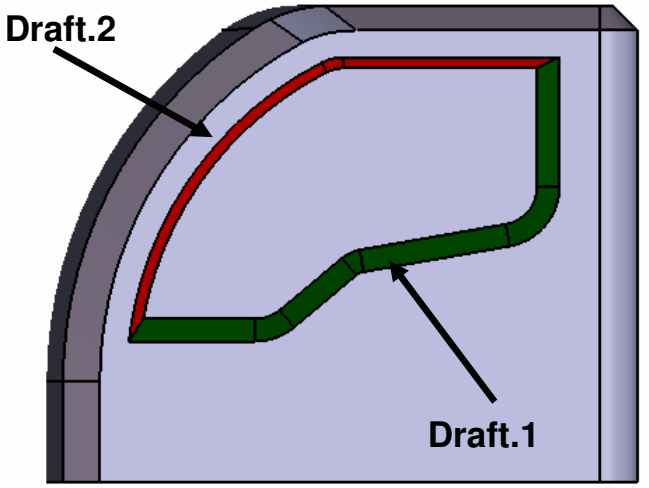
Tutorial 1B

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

- Click "**Draft Angle**" icon.
- Select "Constant" as Draft Type.
- Enter 30deg as Angle.
- Select the upper side face as "Faces to draft".
- Select the bottom face as "Neutral Element".
- Click the arrow once if it is not pointing outward.
- Click ok to complete.



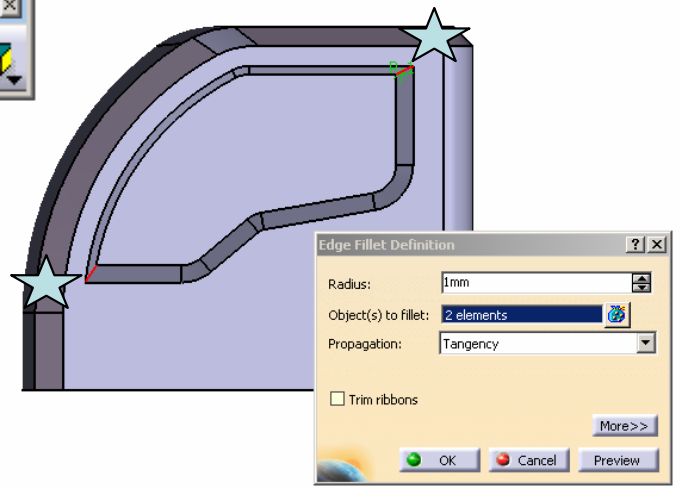
Now you should have two drafts on the pocket.



Tutorial 1B

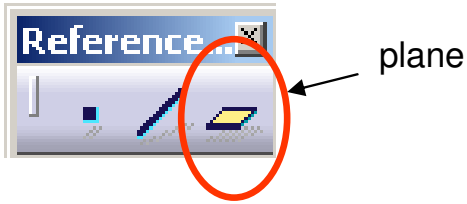
Add "Edge Fillet" R1mm on the remaining two edges of the Pocket at ☆positions.

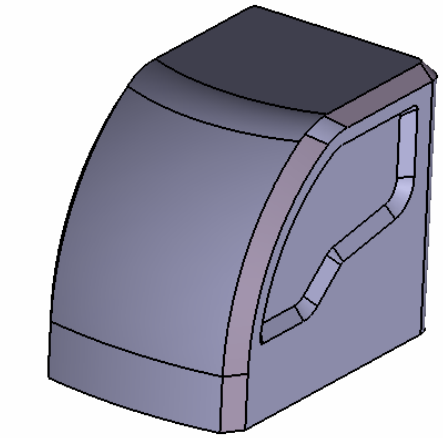




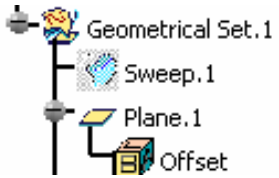
To create an offset plane:-

- Click "**Plane**" icon
- Select "**Offset from plane**" as type.
- Select **ZX** plane
- Enter 70mm as Offset (Value)
- Click ok to complete





Now a plane is created in front of the solid, which is stored in "Geometrical Set.1" on the tree.

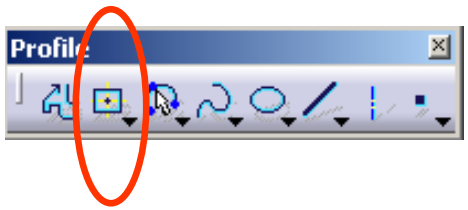


Tutorial 1B

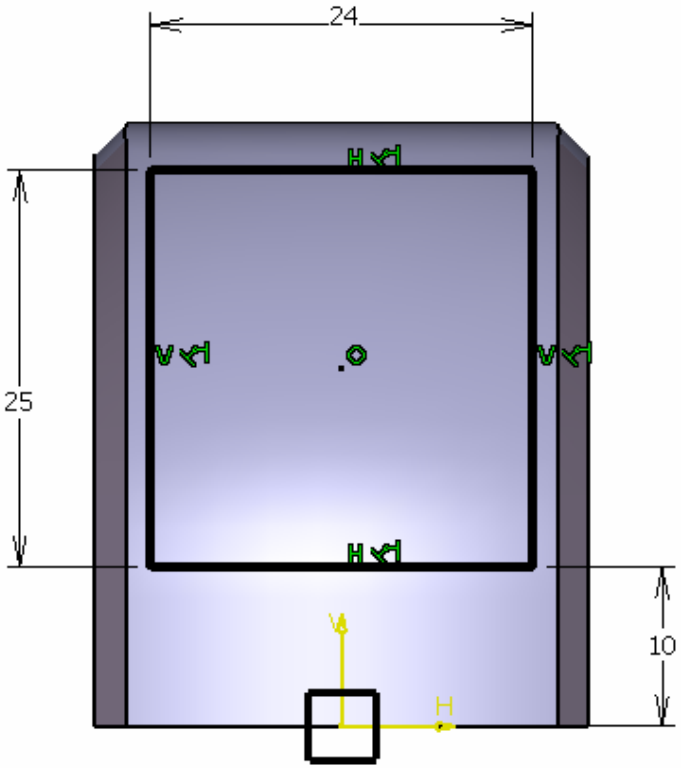
To build a sketch on the offset plane:-

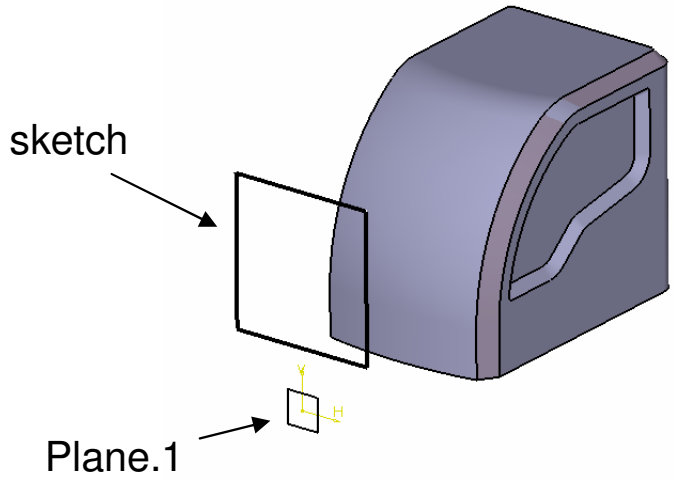
- Click "**Sketch**" icon and select "Plane.1".
- Draw a rectangle (24x25) and position it as shown. (you may use "centered rectangle" for convenience)





- Click ok to complete

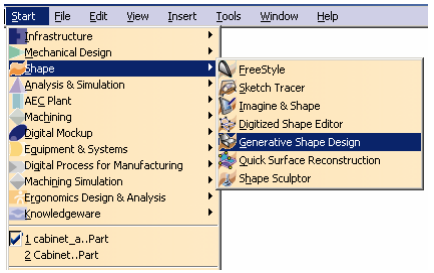




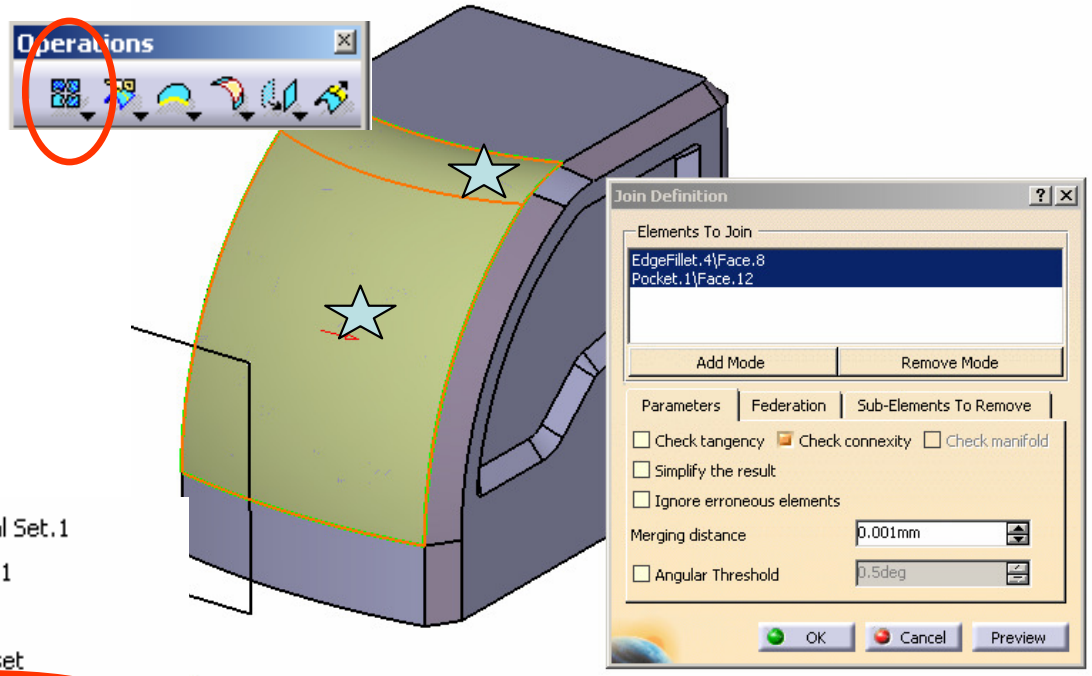
Tutorial 1B

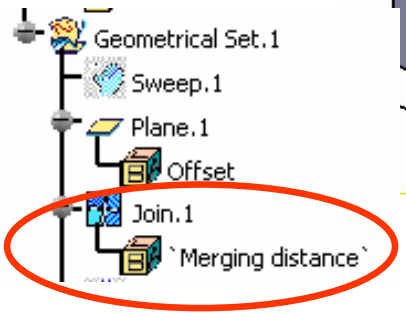
To create an offset surface from the solid:-

- Select “**Start/Shape/Generative Shape Design**” on the menu bar.



- Click “**Join**” icon.
- Select the two surfaces at ☆
- Click Ok to complete
- A new group of surfaces is created on the tree.

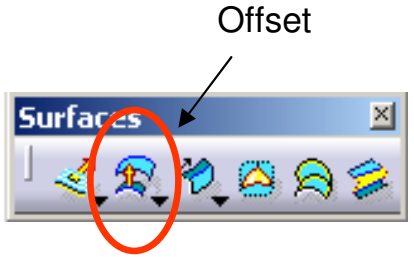


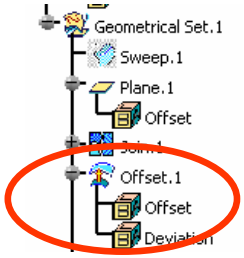


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Tutorial 1B

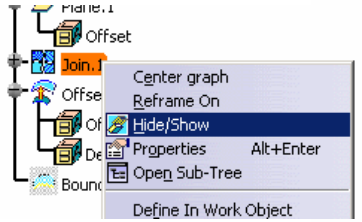
- Click “**Offset**” icon.
- Select “Join.1” on the tree.
- Enter 2mm as Offset (Value) .
- Click the red arrow once or click “Reverse Direction” icon on the menu, if it is not pointing inwards.
- Click Ok to complete
- A offset surface is created ***inside the solid***, and it is stored in “Geometrical.Set.1”.

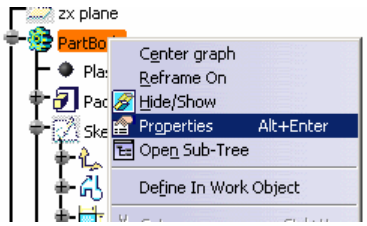




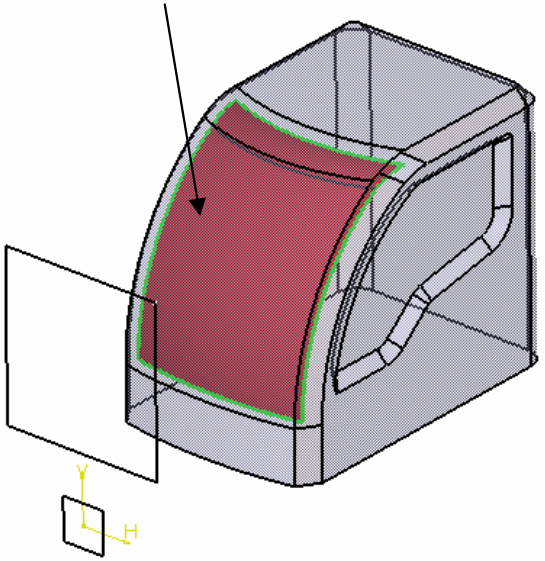
To visualize the offset surface:-

- Hide “Join.1” (click it on tree, right-click to show the contextual menu, then select **Hide/Show**)
- Make “PartBody” semi-transparent (click it on tree, right-click to show the contextual menu, select **Properties**, then set transparency to 100)





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

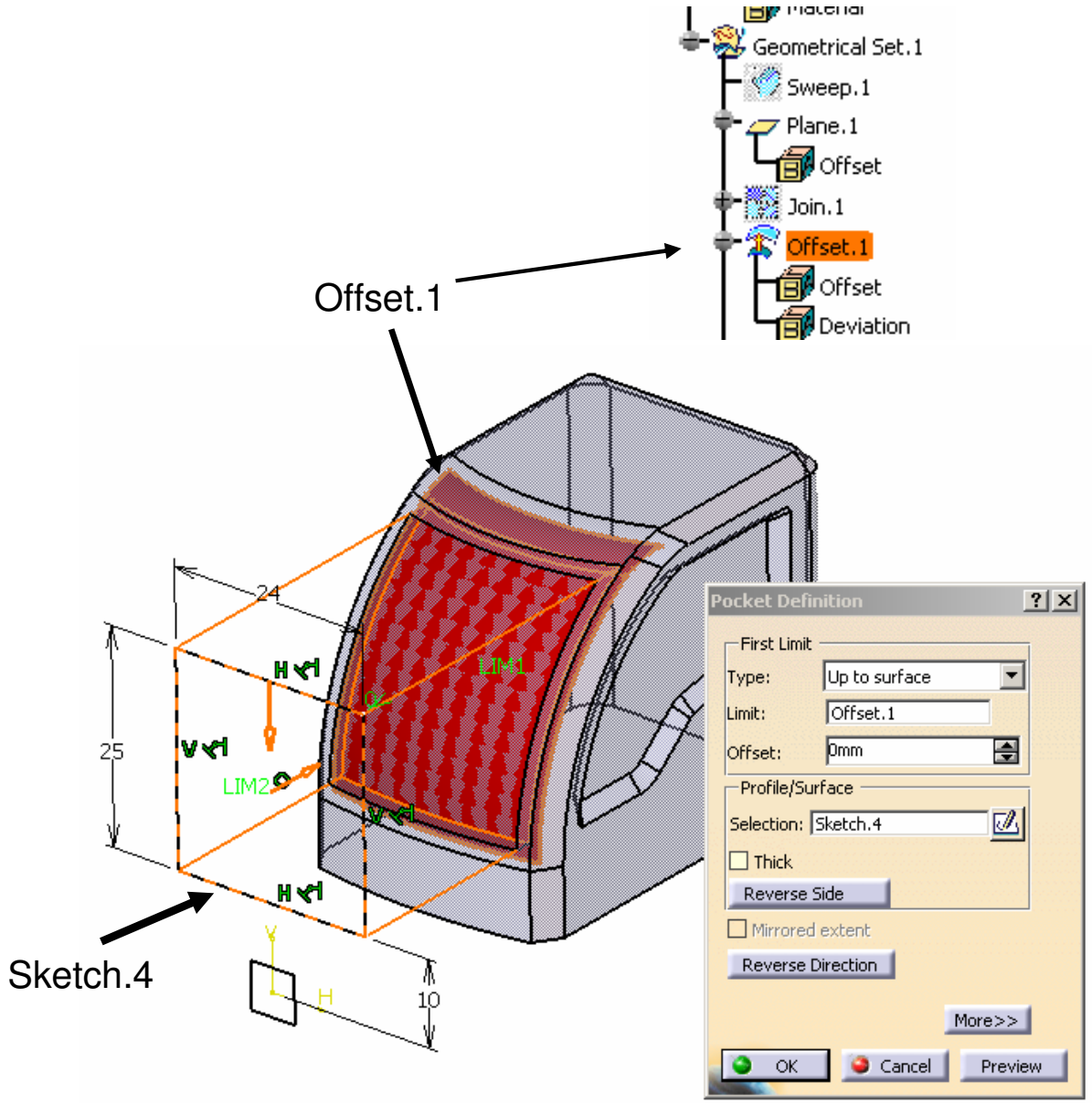


Tutorial 1B

To make a pocket:-

- Go back to Part Design workbench (select "**Start/Mechanical Design/Part Design**")
- Click "**Pocket**" icon.
- Select "Sketch.4" as Profile.
- Select " Up to surface" as Type of First limit.
- Select "Offset.1" on tree as Limit.
- Click ok to complete

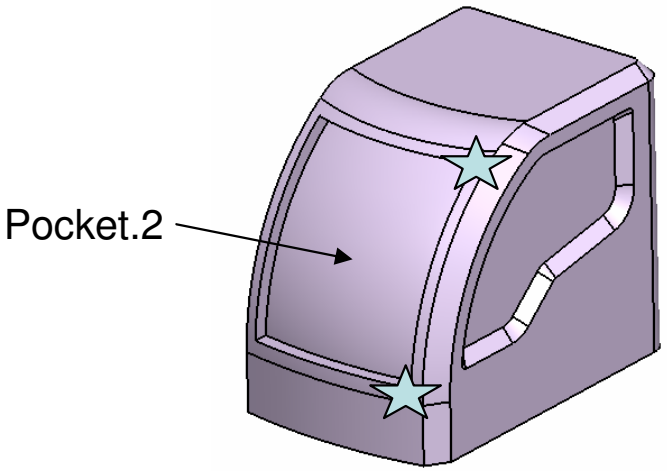
- Hide "Offset.1" & "Plane.1"
- Reset Transparency of "PartBody" to 1



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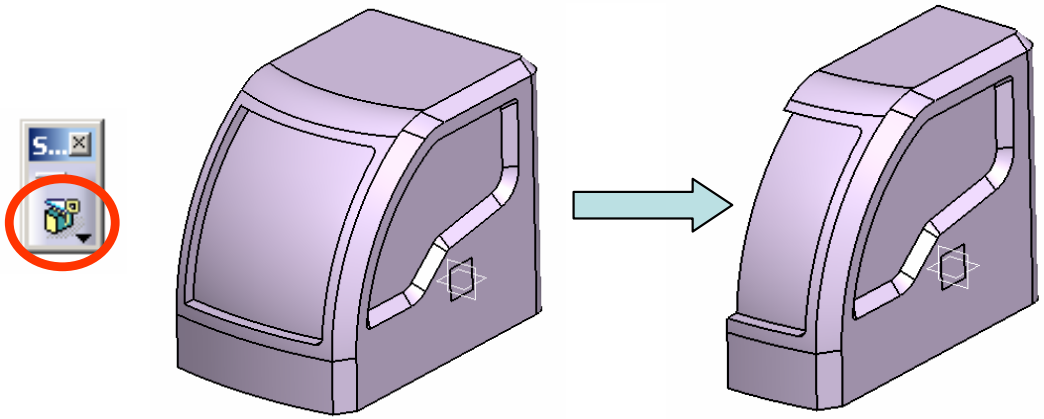
Tutorial 1B

Add "Edge Fillet" R1mm at 2 corners of "Pocket.2" at ☆positions



To split the solid into a half:-

- Click "**Split**" icon.
- Select YZ plane.
- Click the arrow once if it is not pointing to the pocket side.
- Click ok to complete



To copy another half by mirroring:-

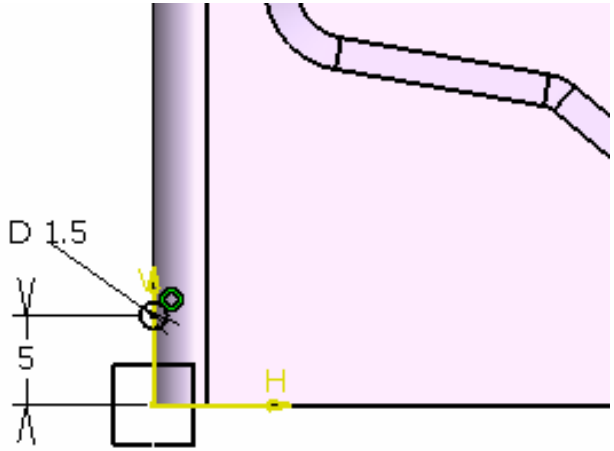
- Click "**Mirror**" icon.
- Select yz plane.
- Click ok to complete.



Tutorial 1B

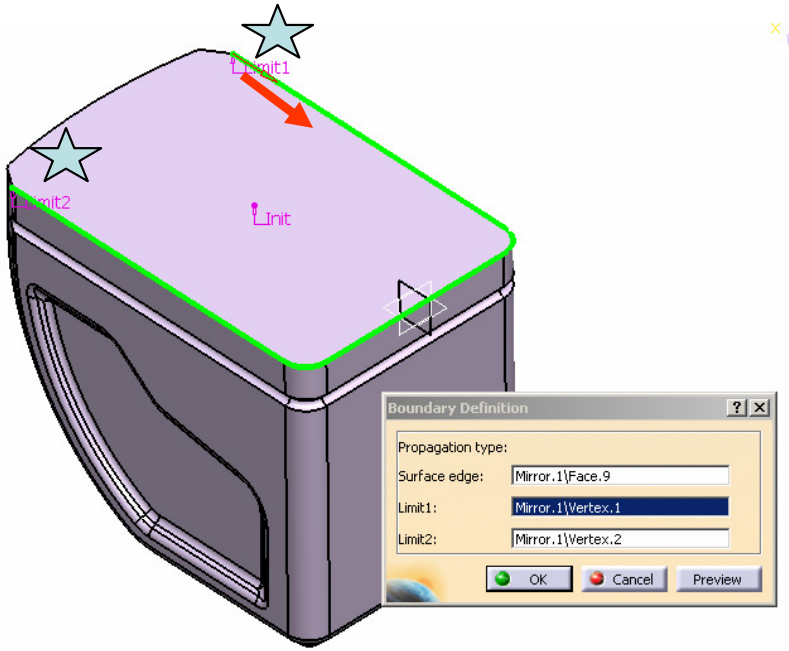
To remove material along a guide:-

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



- Select "**Start/Shape/Generative Shape Design**" on the menu bar
- Click "**Boundary**" icon
- Select the bottom surface of solid
- Select the endpoints at ☆ position as Limit 1 & Limit 2.
- Click the red arrow once if it is not choosing the longer path
- Click ok to complete

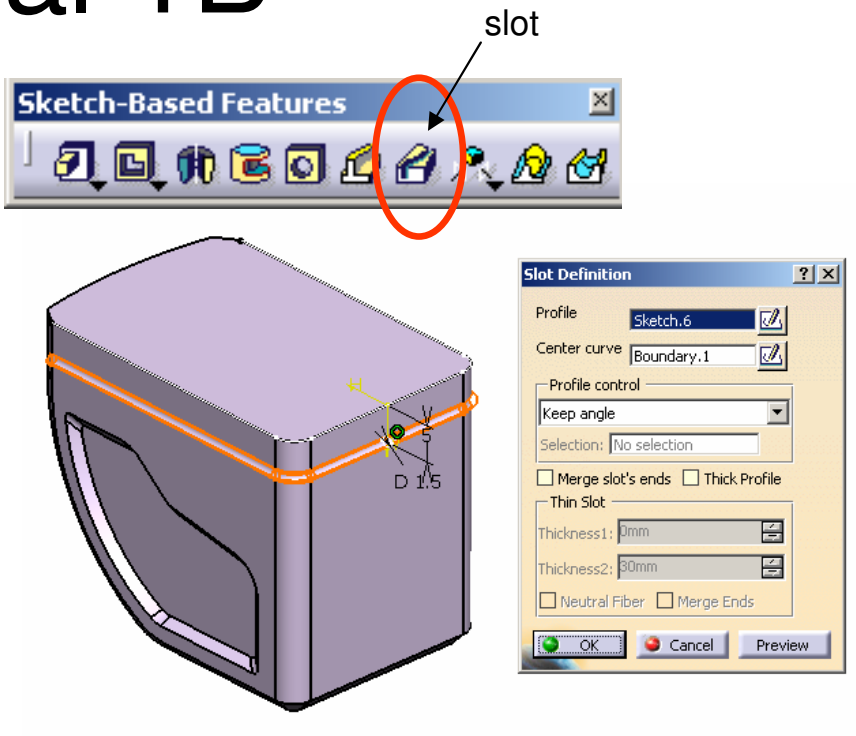




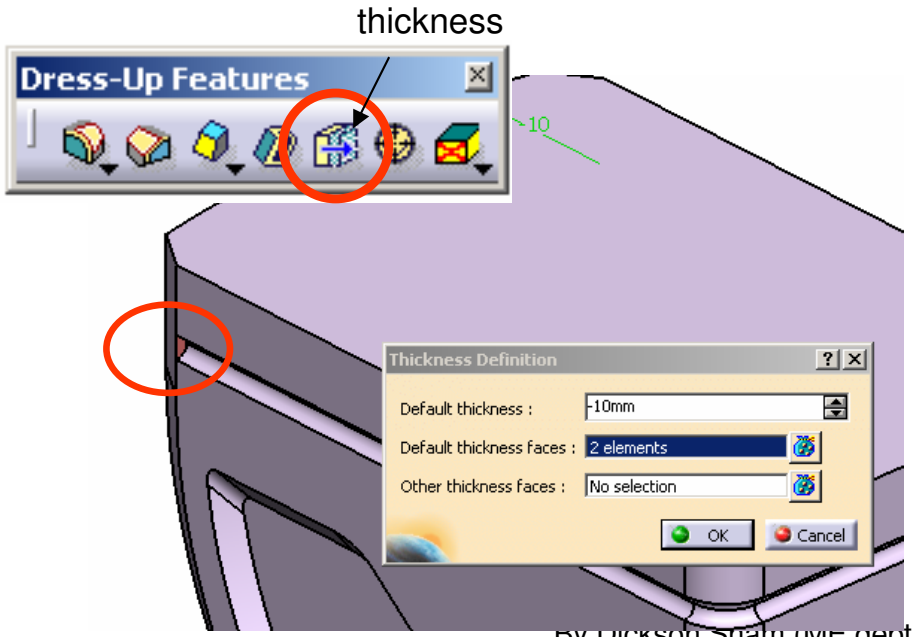
By Dickson Sham (ME dept, HKPU)

Tutorial 1B

- Go back to Part Design Workbench (Select “Start/Mechanical Design/Part Design”)
- Click “**Slot**” icon.
- Select “Sketch.6” as Profile.
- Select “Boundary.1” as Center Curve.
- Click ok to complete

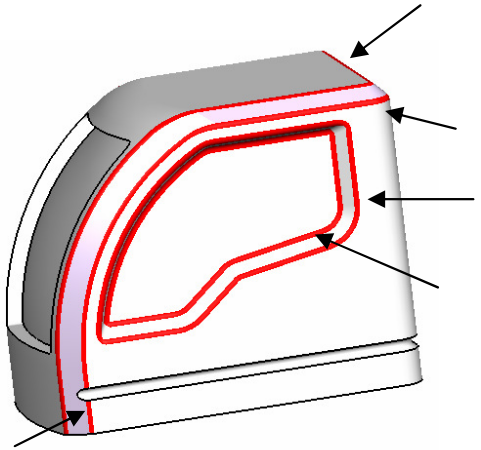


- Click “**Thickness**” icon
- Select the faces at the both ends of the slot
- Enter -10mm as Default thickness
- Click ok to complete
- Now you can see the slot has open ends



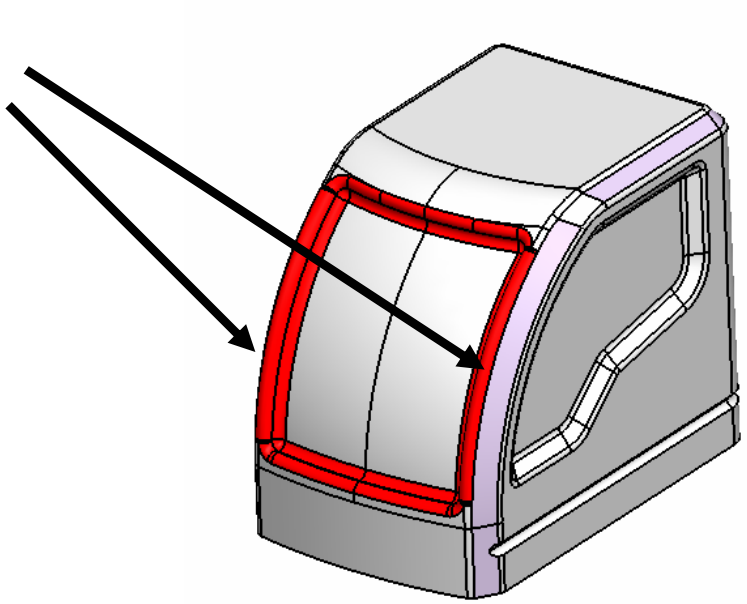
Tutorial 1B

Add "Edge Fillet" R1.0mm on the edges of both sides, except those of the front pocket.



Add "Edge Fillet" R1.0mm on the edges of the front pocket.

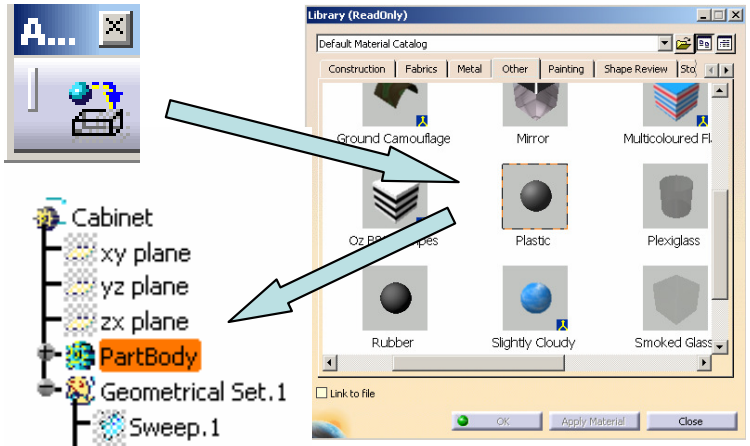
- Sometimes, we need to build fillets separately when the sharp edges are too close to each other. We need to build a fillet on one edge first, and then build another one on top of it.



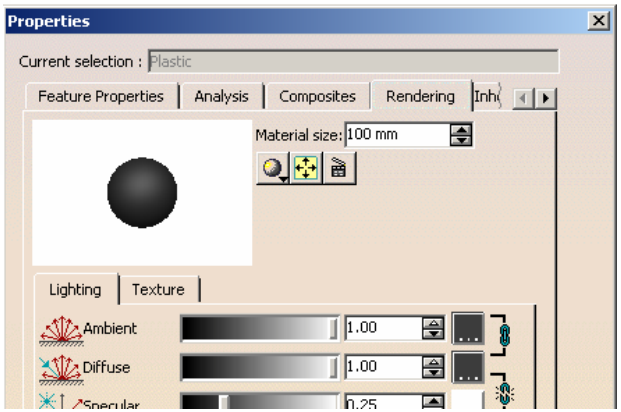
Tutorial 1B

To apply material properties on the model:-

- Click “**Apply Material**” icon.
- Select “Plastic” on the tab-page ”Other”
- Click “PartBody” of the tree
- Click Ok to complete

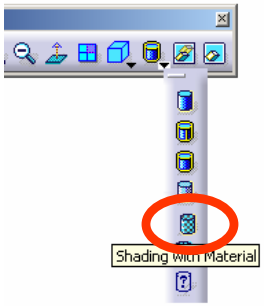


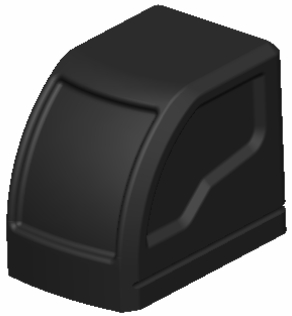
- Double-Click “Plastic” on the tree
- Select “Rendering” Tab-page
- Change “Ambient” to 1.00
- Change “Diffuse” to 1.00
- Change color as Red 60, Green 60, Blue 60
- Click ok to complete



To view the material rendering:-

- Click “**Shading with material**” icon





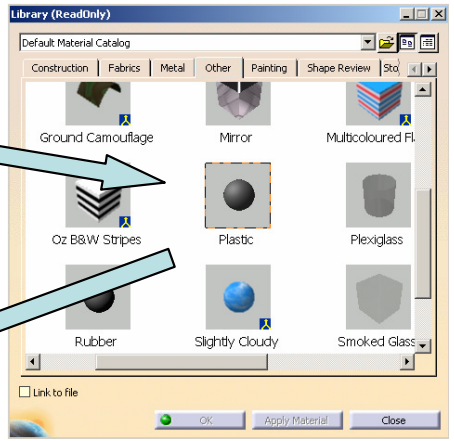
Save the file as “Cabinet_a.CATPART” in your project folder and then close it.

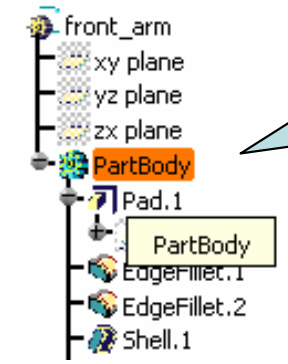
Tutorial 1B

To apply material properties on "front_arm":-

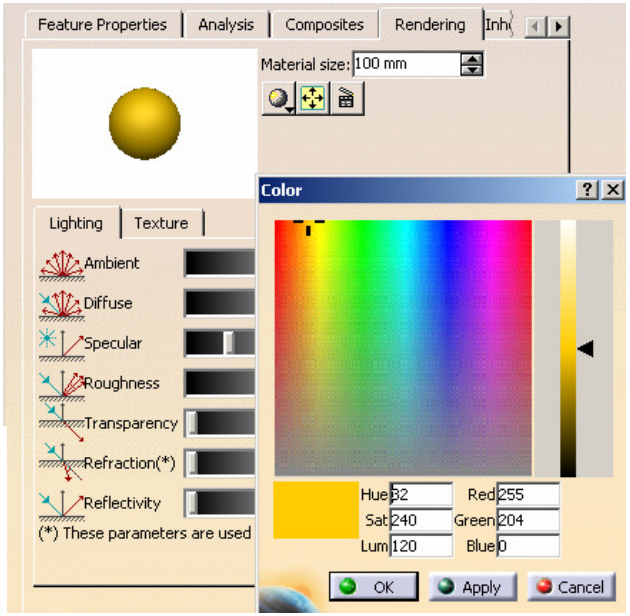
- File/Open… "front_arm_a.CATPART
- Click "**Apply Material**" icon.
- Select "Plastic" on the tab-page "Other".
- Click "PartBody" of the tree.
- Click Ok to complete.

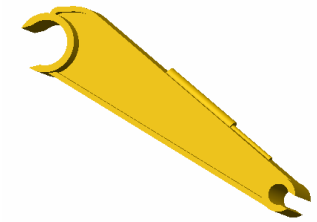






- Double-Click "Plastic" on the tree.
- Select "Rendering" Tab-page.
- Change "Ambient" to 1.00
- Change "Diffuse" to 1.00
- Change color as Red 255, Green 204, Blue 0
- Click ok to complete.





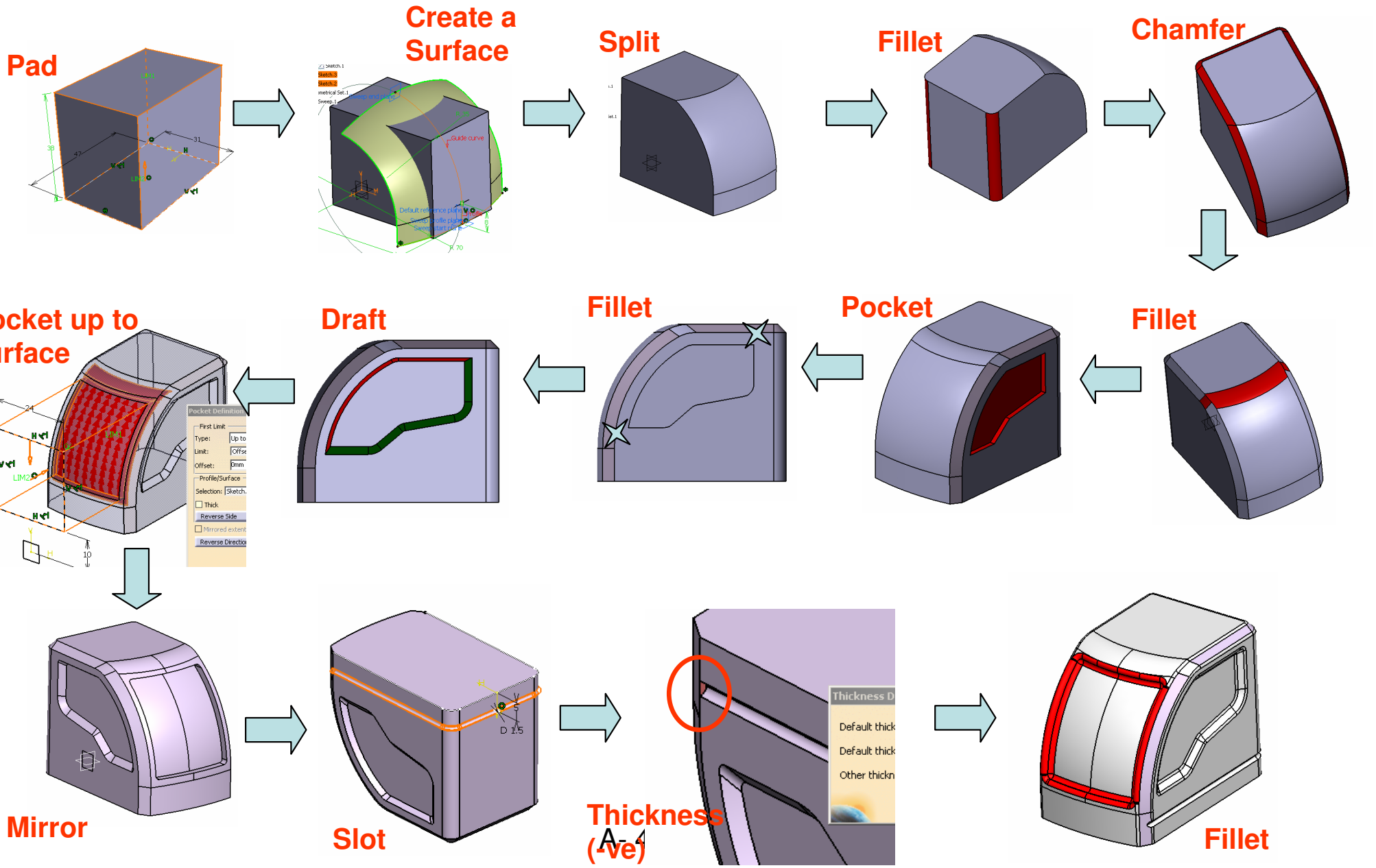
Save and Close the file

END of Tutorial 1B

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Summary of Tut-1B

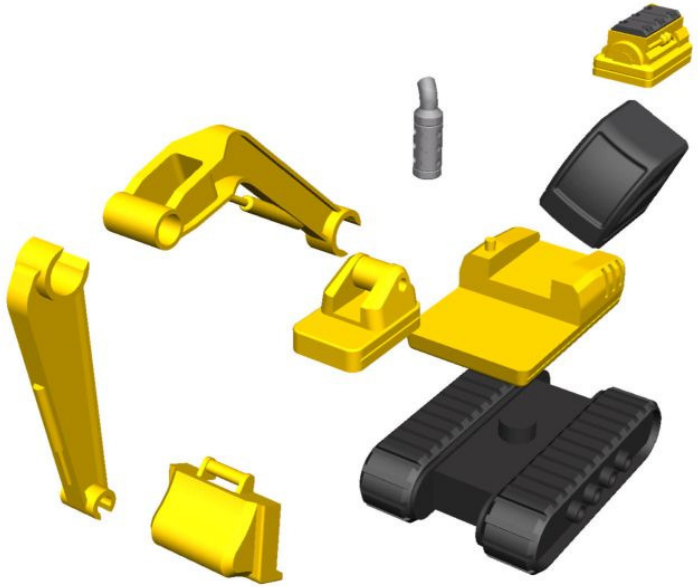
Build a Solid:-



By Dickson Sham (ME dept, HKPU)

Tutorial 1C

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



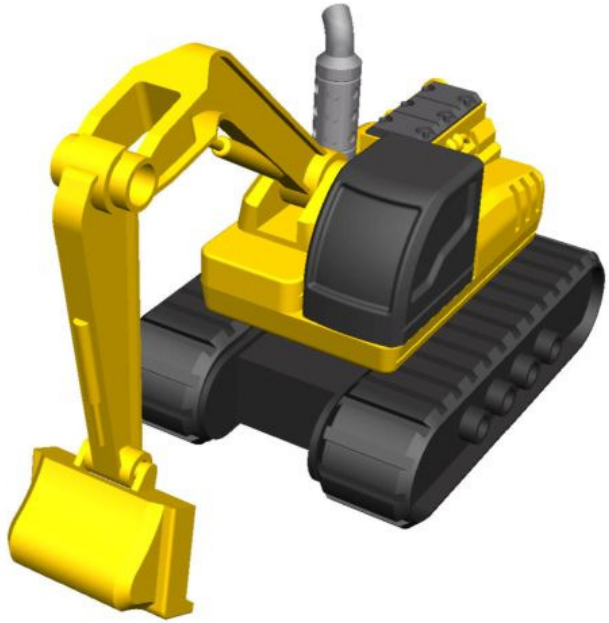
To collect all component files into your project folder:-

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

- For the rest, you can download from the web:

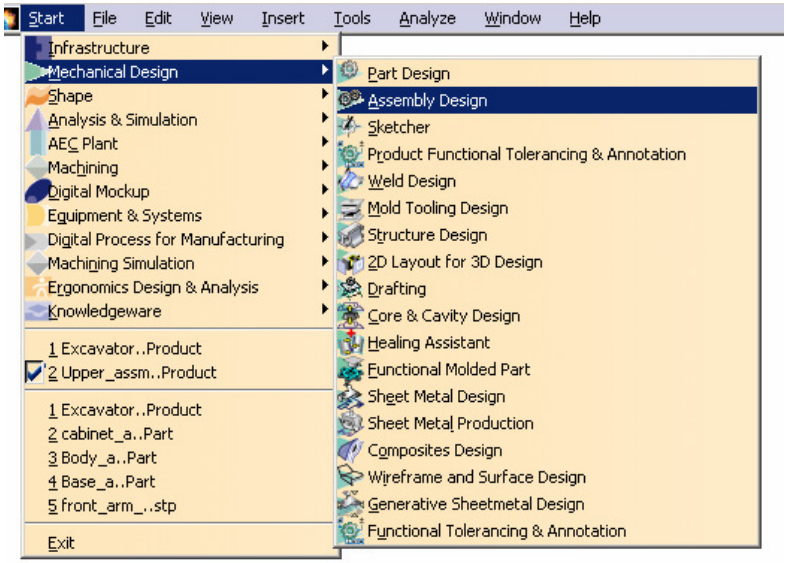
http://myweb.polyu.edu.hk/~mmdsham/training%20material.htm

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



Tutorial 1C

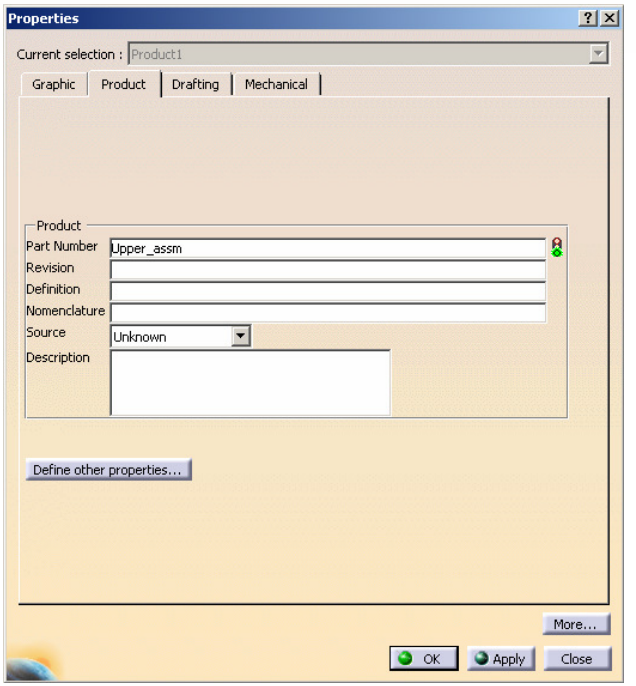
- Enter CATIA.
- Close all files.
- Select '**Start/Mechanical Design/Assembly Design**" on the menu bar.
- You may need to reset the layout of the toolbars if the workbench isn't tidy.



To rename the tree:-

- Single-click "Product1" on the tree, right-click it, and then select "**Properties**".
- Modify Part Number as "Upper_assm" on the tab page "Product".
- Select "ok" to exit .

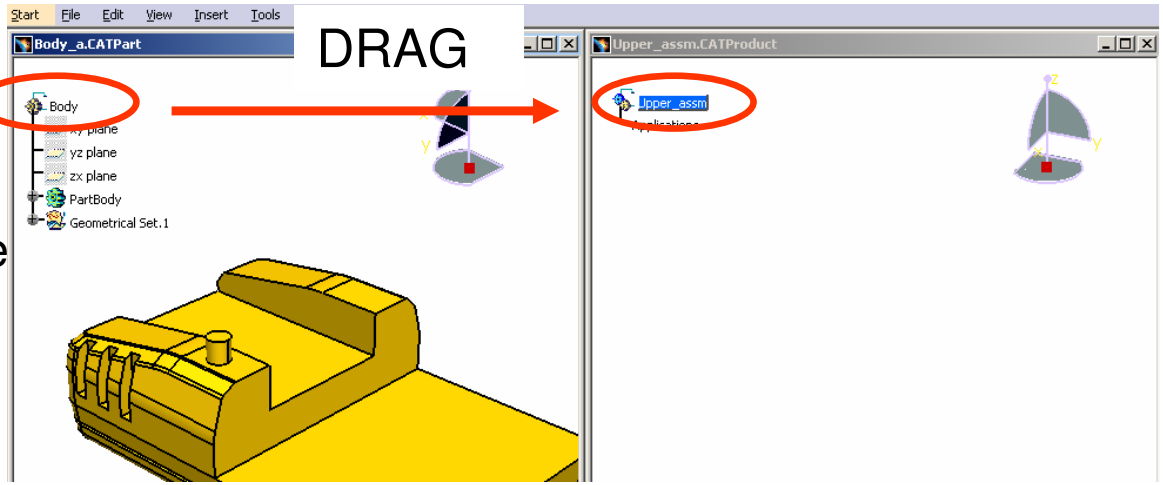




Tutorial 1C

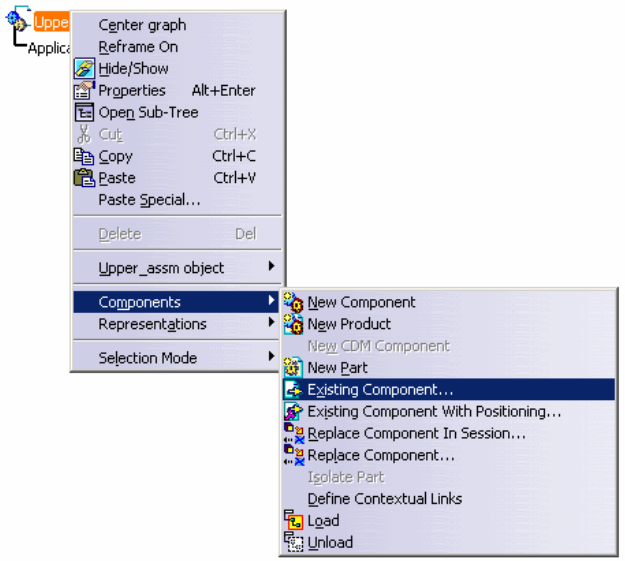
To insert existing parts into a product:-

- File/Open…"Body_a.CATPART".
- Select "Window/Tile vertically".
- Click and hold the highest icon of the part tree "BODY" and then drag it onto the product tree.



OR

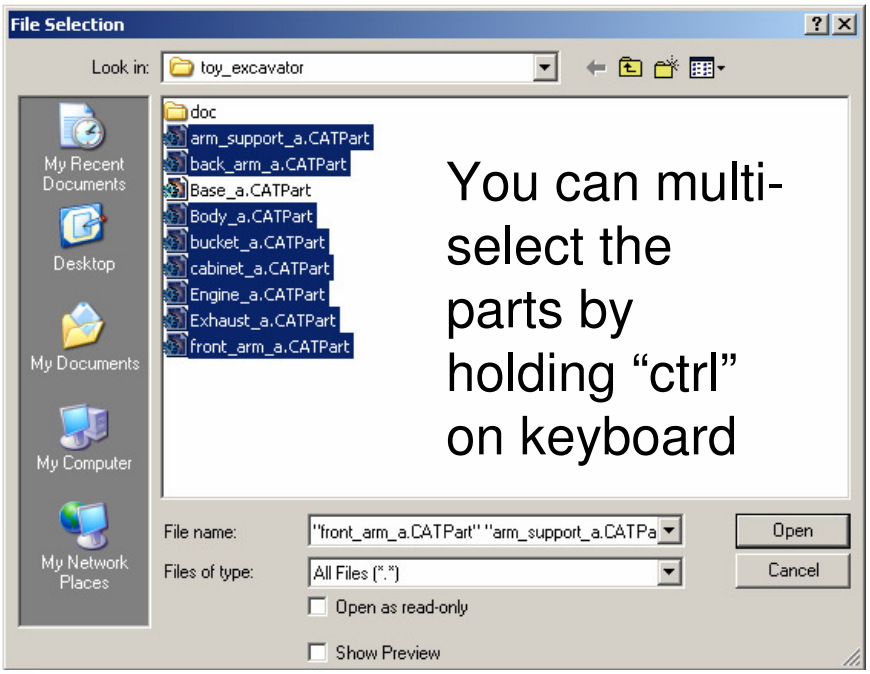
- (NO need to open a part file)
- Right-click the highest icon of the Product tree "Upper_assm", then select **"Components/Existing component…"**
- Select "Body_a.CATPART"
- Click "open"

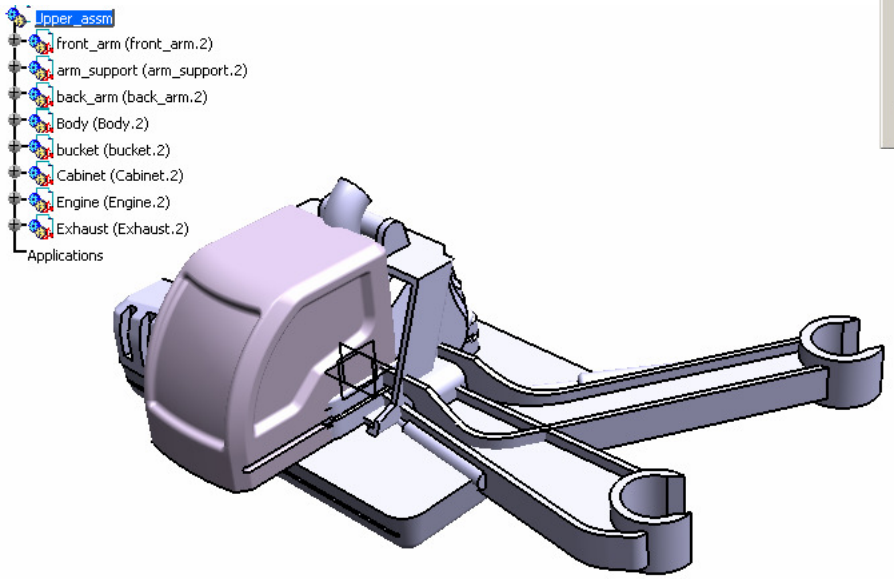


Tutorial 1C

- Similarly, insert all parts EXCEPT “Base_a.CATPART”

- You should see all inserted parts are mixed together and at wrong positions. It is normal because the system puts all the parts’ origins onto the product’s origin.

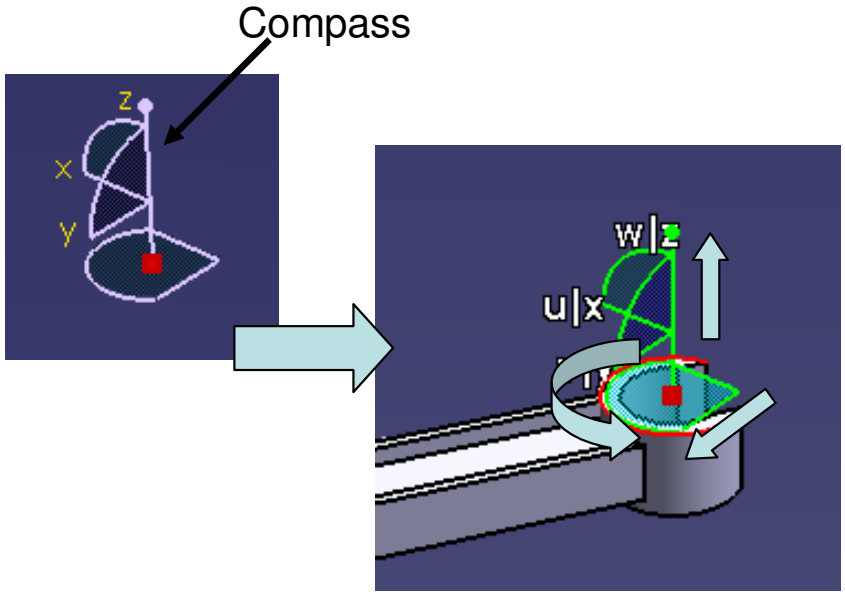




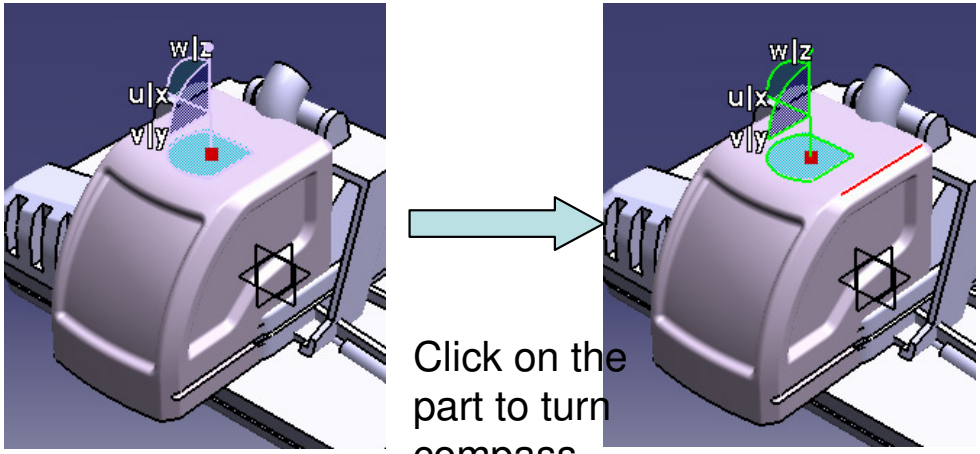
Tutorial 1C

To move a part by "Compass":-

- Click and hold the **RED** dot of the compass
- Drag it onto the part that you want to move.
- The compass will then turn into green and its axis labels will be v-u-w.
- Drag along the green lines/arcs of the compass to move the part to a desired position.



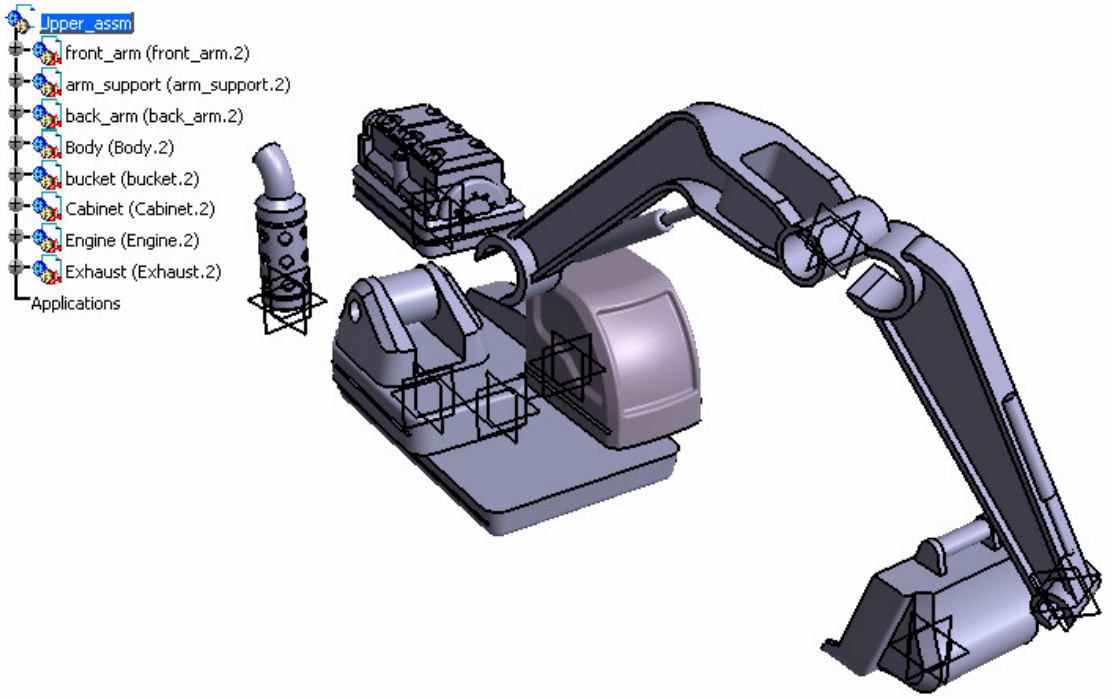
- After moving one part, drag the compass onto the other part.
- Click the 2nd part once so that the compass turns green again. Now the compass can move the 2nd part.



Click on the part to turn compass GREEN

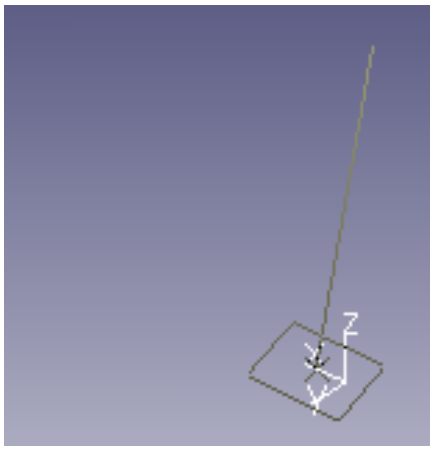
Tutorial 1C

- Repeat the steps so that all parts are NEARLY at a desired positions.
- Now the parts are separated. It is easier for us to select part features later.



To reset "Compass" as original:-

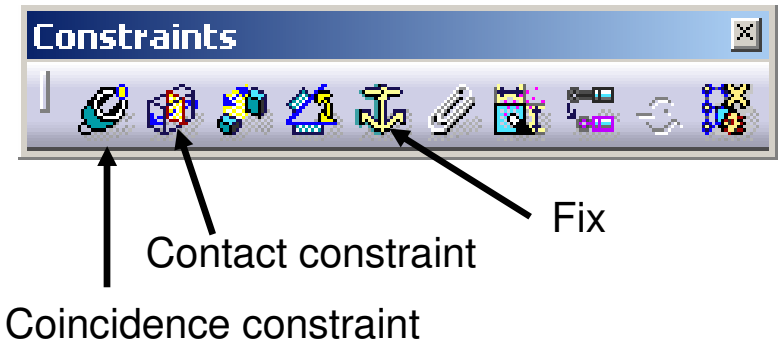
- Click and hold the red dot of the compass.
- Drag it onto the coordinate system at lower right-hand corner of the window and then release.
- It will be auto-reset.



Tutorial 1C

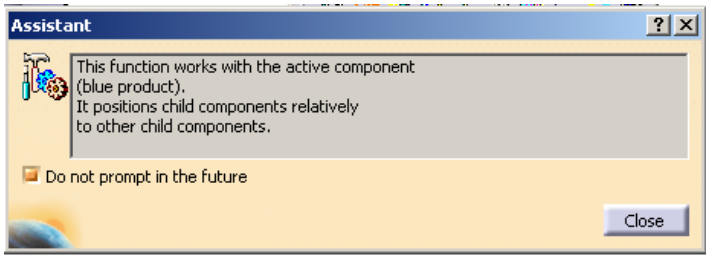
To assemble parts by adding constraints:-

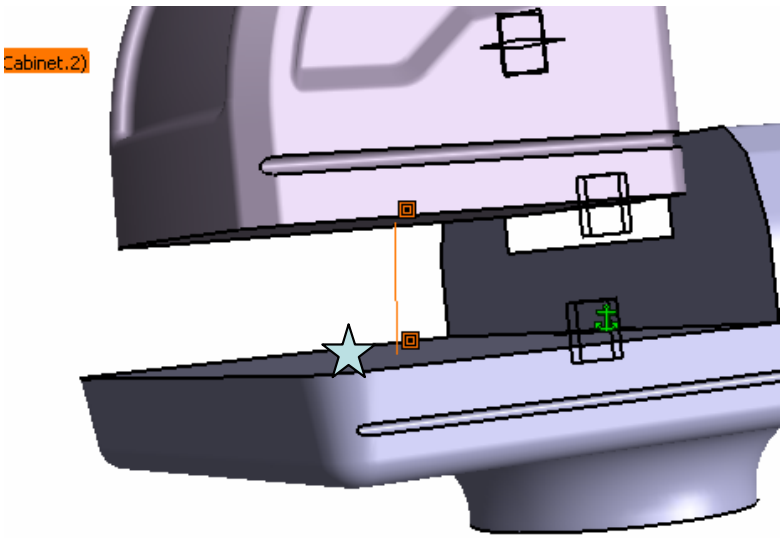
- Click “**Fix**” icon
- Select “Body” on tree; Now the part “Body” is fixed in position.



(1) Link “Cabinet” to “Body”

- Click “**Contact**” icon
- Check “Do not prompt in future” and click “close” to close the message box.
- Select the bottom face of “Cabinet” and then select the face ☆of “Body”
- A constraint is created, although “Cabinet” hasn’t snapped onto “Body”.



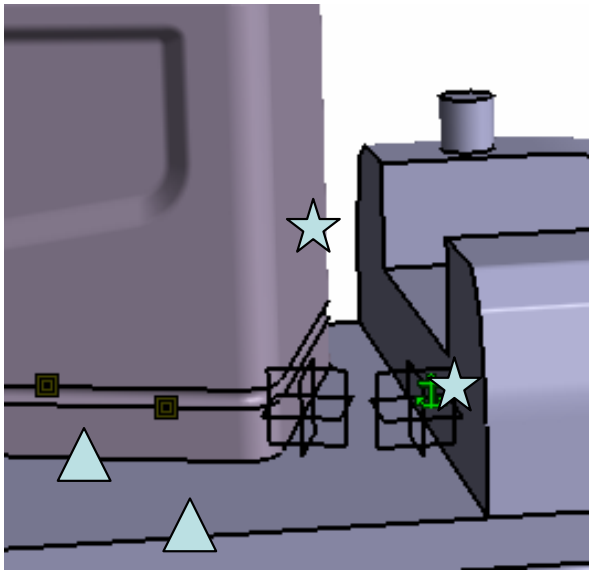


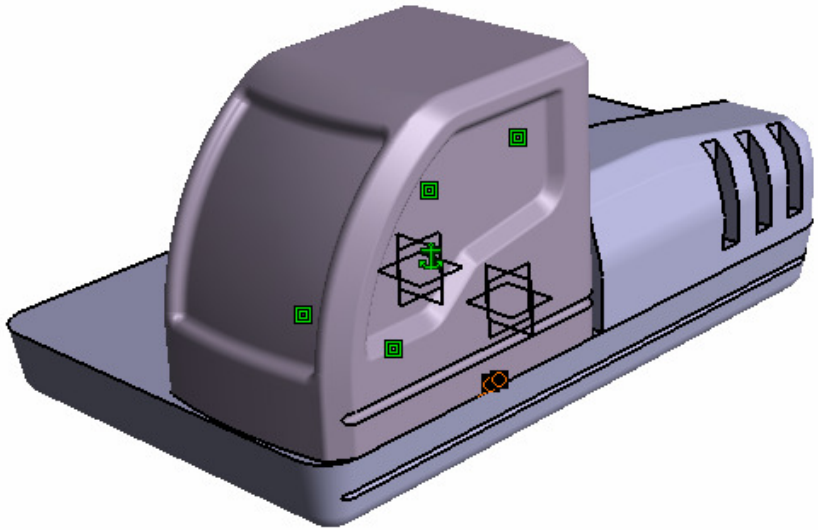
If you want to delete a constraint, just click the constraint either on the model or on the tree, and then press “Delete” key on keyboard.

Tutorial 1C

Link "Cabinet" to "Body" (cont')

- Add another "**Contact**" Constraint between the faces marked with ☆
- Add a "**Coincidence**" Constraint between the edges marked with △
- Click "**Update**" Icon  to update the position.

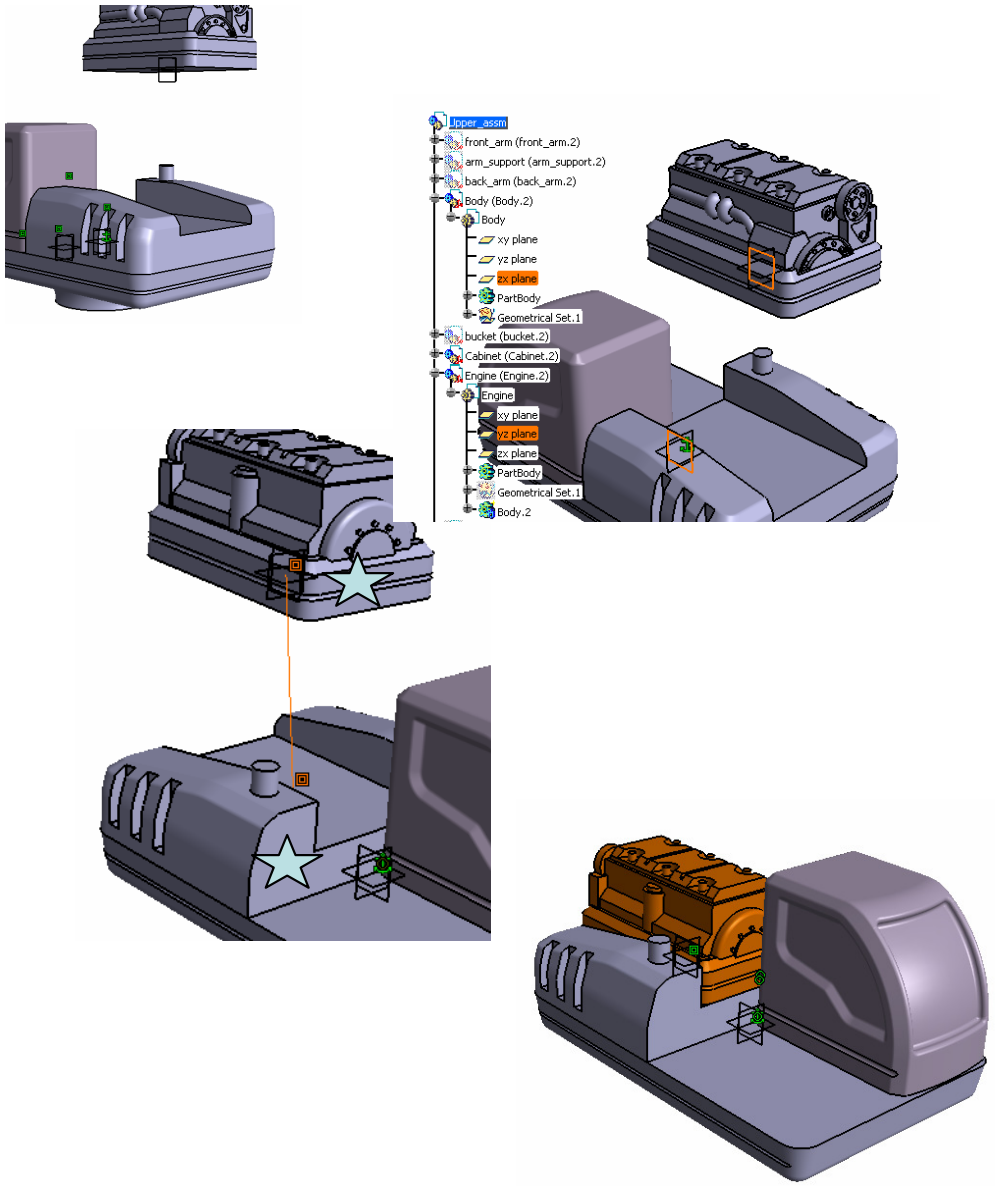




Tutorial 1C

Link "Engine" to "Body"

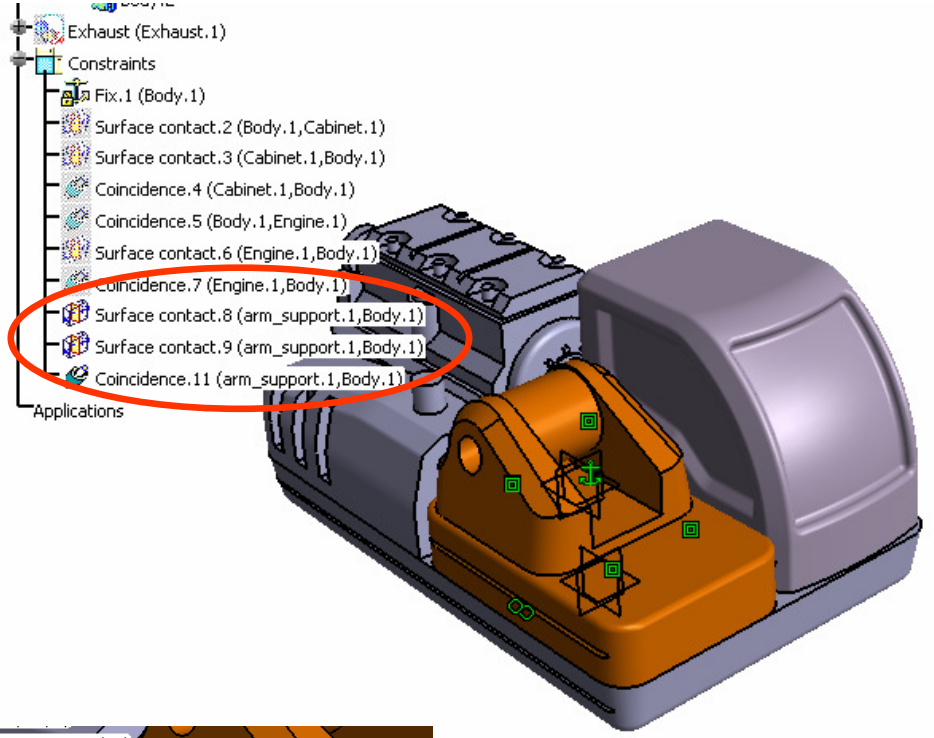
- Add a "**Contact**" Constraint between the bottom face of Engine and the top face of Body
- Add a "**Coincidence**" Constraint between the yz plane of Engine and the zx plane of Body
- Add a "**Coincidence**" Constraint between the faces marked with ☆
- Click "**Update**" Icon  to update the position.

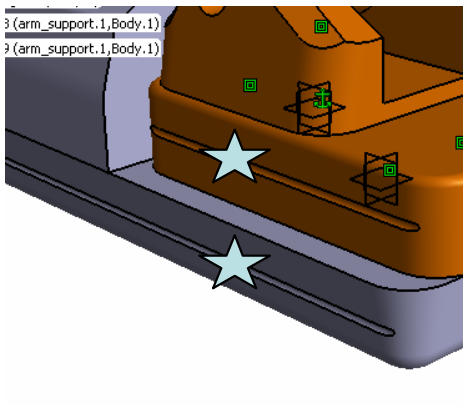


Tutorial 1C

Link "Arm_support" to "Body"

- Add two **contact** constraints and one **coincidence** constraint.
- Click "**Update**" Icon  to update the position.
- Remark: We cannot add "Coincidence constraint" between the faces with ☆ because they are not parallel. Therefore they can only add the constraint between their edges



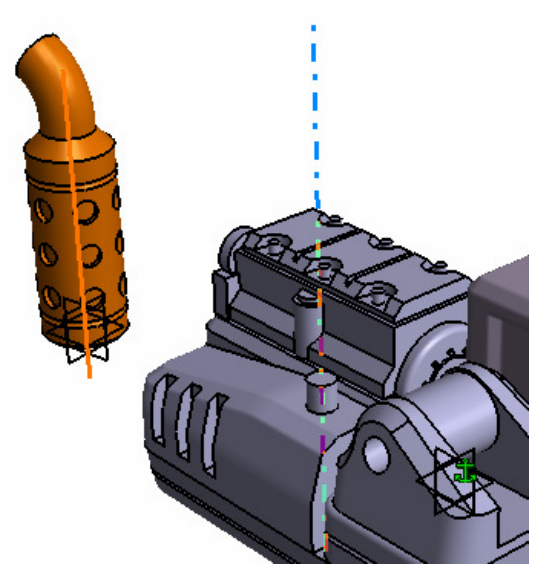


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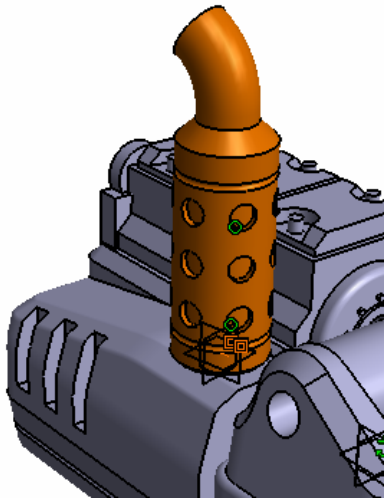
Tutorial 1C

Link "Exhaust" to "Body"

- Add a "**Coincidence**" constraint between the axes. (Remark, when the mouse cursor is on the circular surface of the cylinder, the axis is auto-detected)



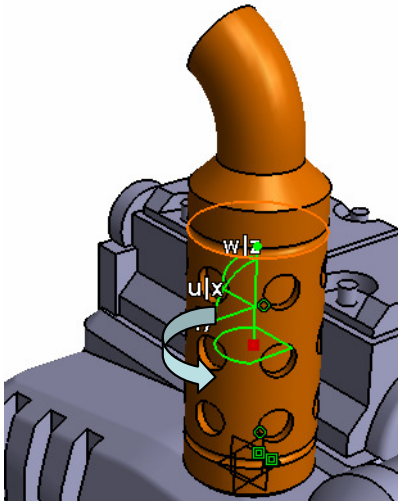
- Add a "**Contact**" constraint as shown.



- Click "**Update**" Icon 

- The angular orientation is not important in this case, but you may use the compass to change it…

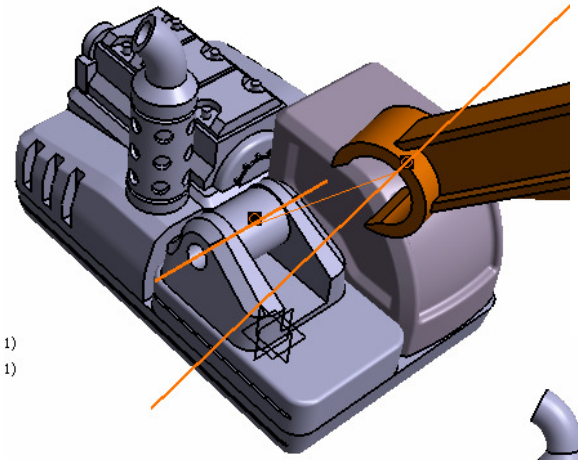
  (place the compass on the circular surface of a cylinder, the compass will snap onto the axis)



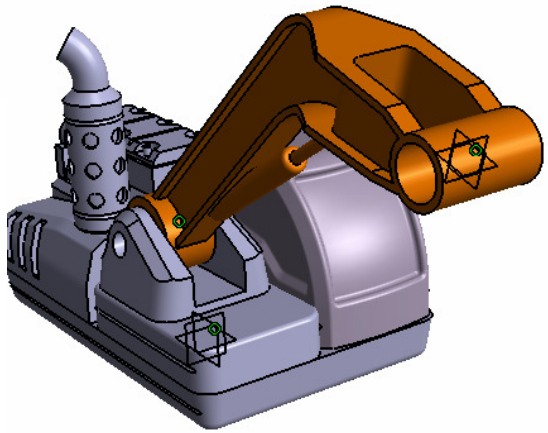
Tutorial 1C

Link "Back_arm" to "Arm_support"

- Add a "**Coincidence**" constraint between the axes. (Remark, when the mouse cursor is on the circular surface of the cylinder, the axis is auto-detected)

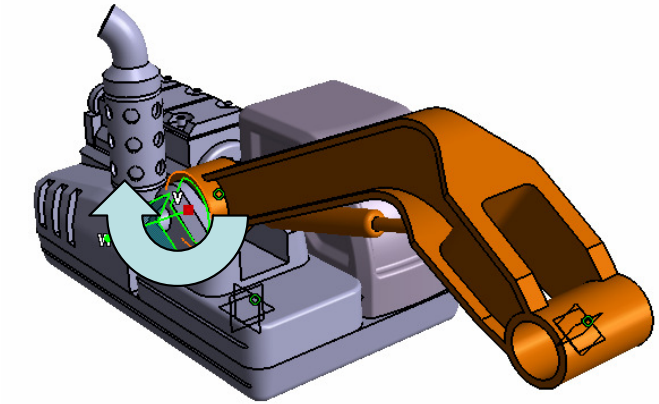


- Add a "**Coincidence**" constraint between xy plane of Back_arm and yz plane of Arm_support



- Click "**Update**" Icon 

- The angular orientation is not important in this case, you may use the compass to change it…

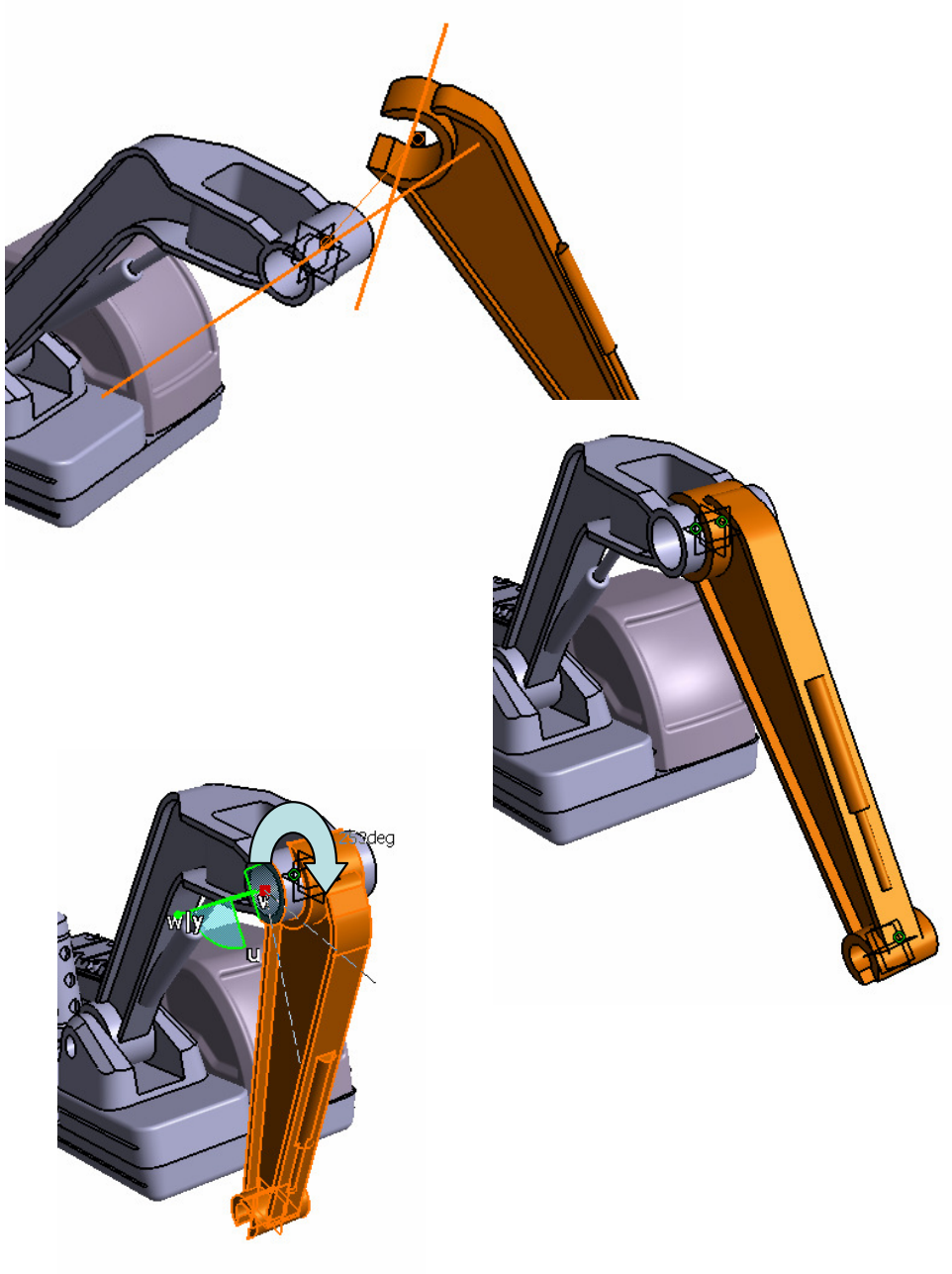


Tutorial 1C

Link "Front_arm" to "Back_arm"

- Add a "**Coincidence**" constraint between the axes. (Remark, when the mouse cursor is on the circular surface of the cylinder, the axis is auto-detected)
- Add a "**Coincidence**" constraint between xy plane of front_arm and xy plane of back_arm
- Click "**Update**" Icon 

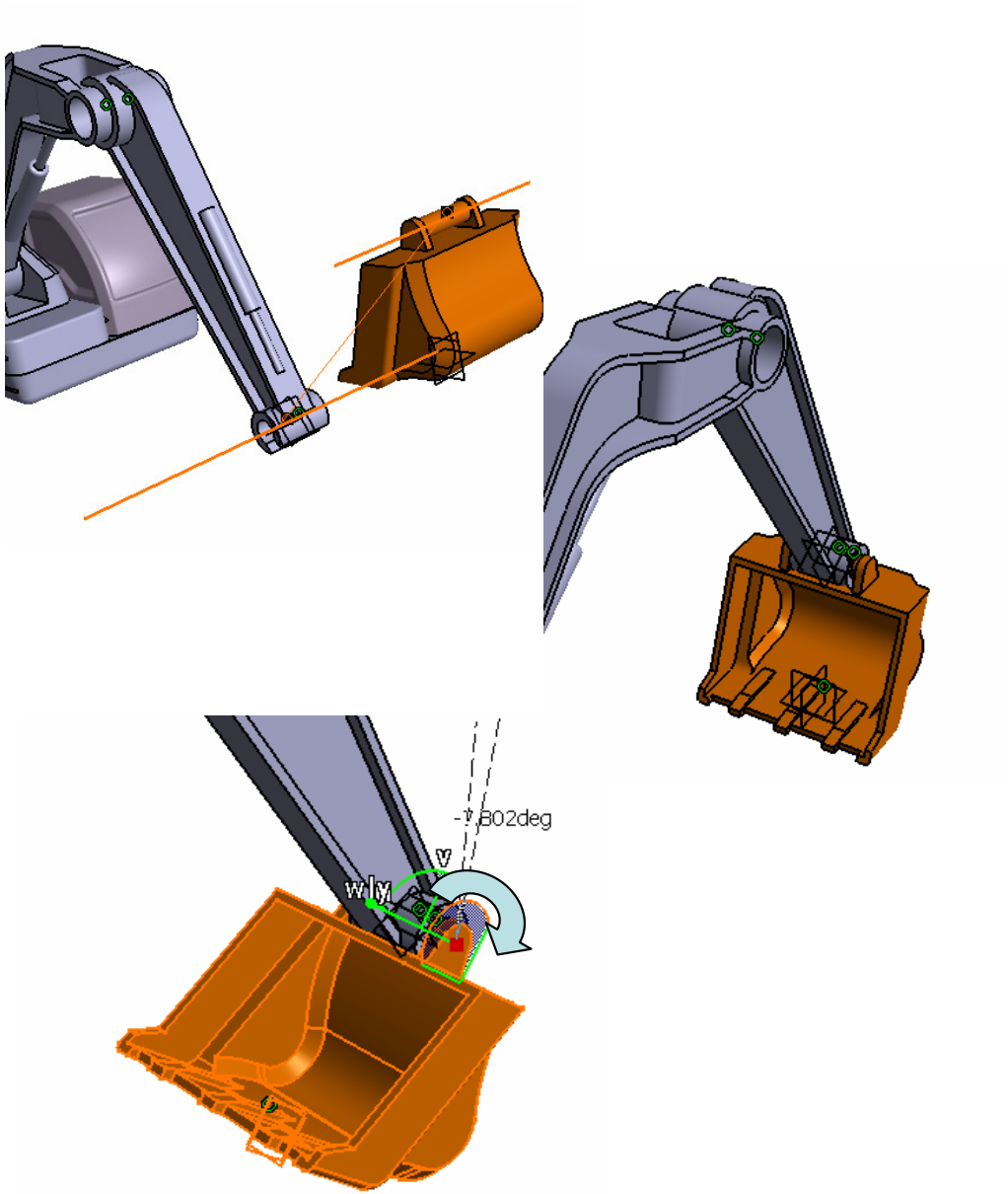
- The angular orientation is not important in this case, you may use the compass to change it…



Tutorial 1C

Link "Bucket" to "Front_arm"

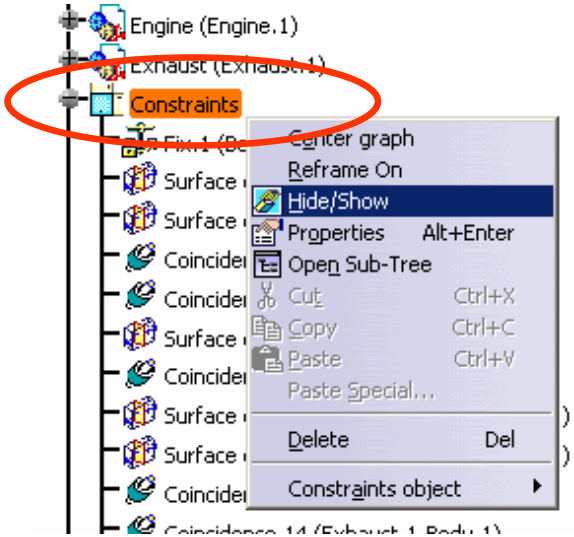
- Add a "**Coincidence**" constraint between the axes. (Remark, when the mouse cursor is on the circular surface of the cylinder, the axis is auto-detected)
- Add a "**Coincidence**" constraint between yz plane of bucket and xy plane of front_arm
- Click Update Icon 
- The angular orientation is not important in this case, you may use the compass to change it…



Tutorial 1C

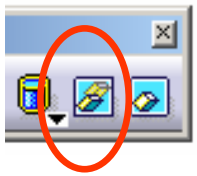
To hide all constraints:-

- Just single-click “Constraints” on the tree and right-click to show the contextual menu; then select “Hide/Show”.



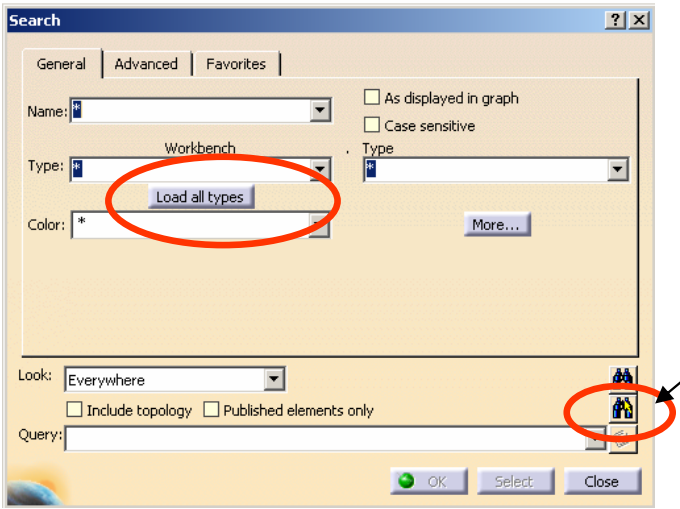
To hide all datum planes:-

- You can multi-select all planes and click “hide/show” icon.



OR

- Select “Edit/Search..” on the menu bar and then click “Load all type” icon.
- Select “Plane” as Type.
- Click “Search & Select” icon.
- Click “Hide/Show” icon.

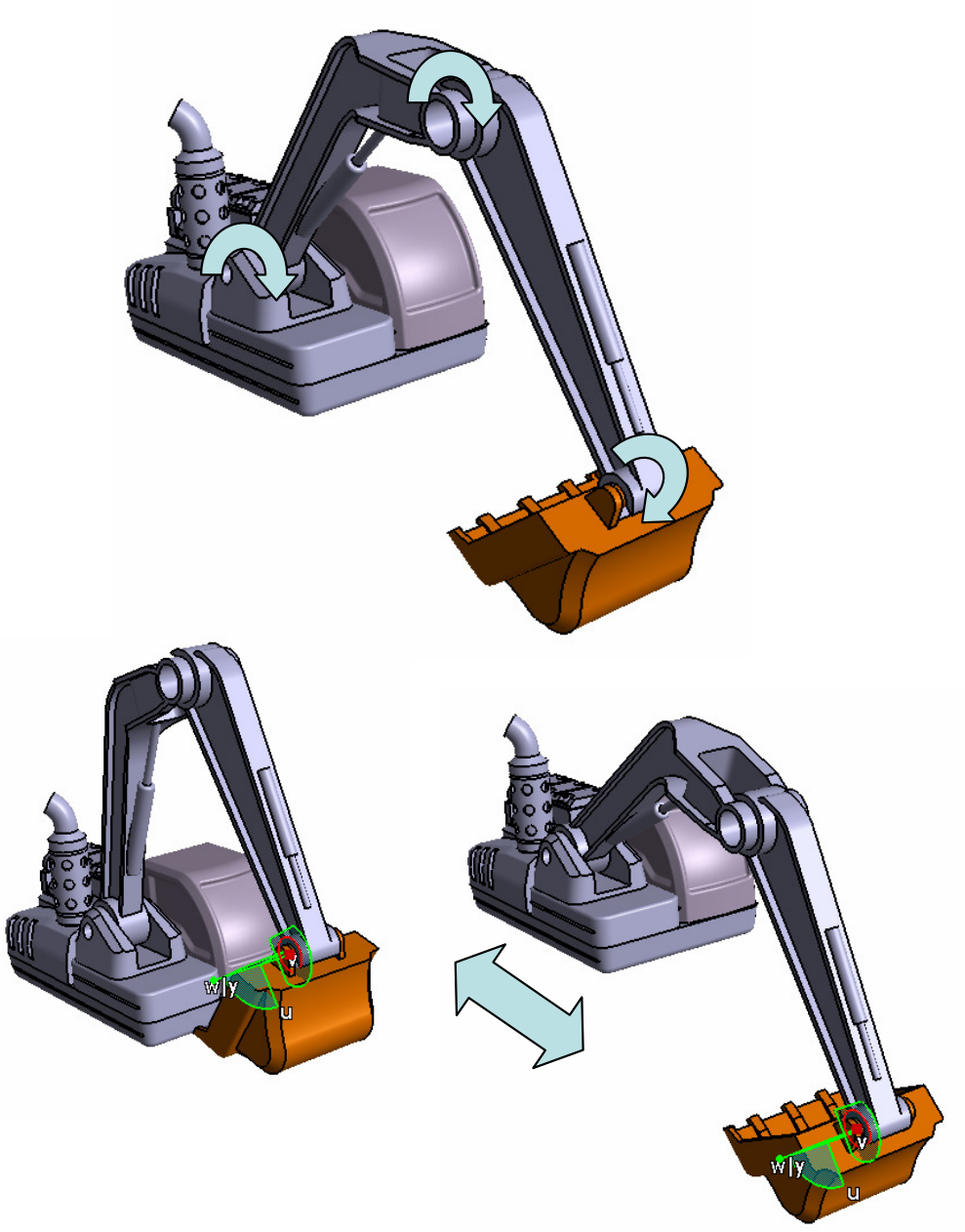


Search & select

Tutorial 1C

To simulate the motion of the machine arm:-

- Because the angular orientation at the joints is not constrained, we can use the Compass to change the angular positions of the arm and the bucket.
- Drag the compass onto the **axis** of the bucket and then release. Click the bucket once to ensure that it is activated.
- Press & Hold **"Shift" key** on the keyboard.
- Drag the bucket with the compass and you will see that front_arm & back_arm are moved correspondingly.



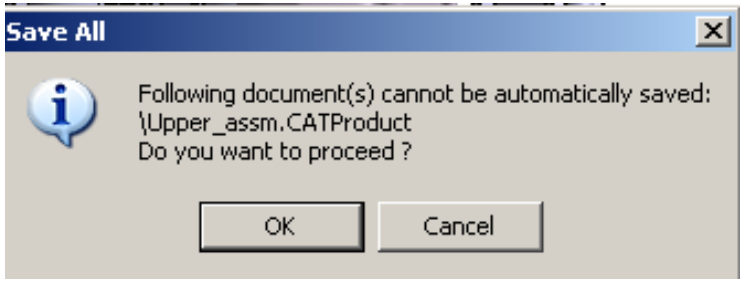
Tutorial 1C

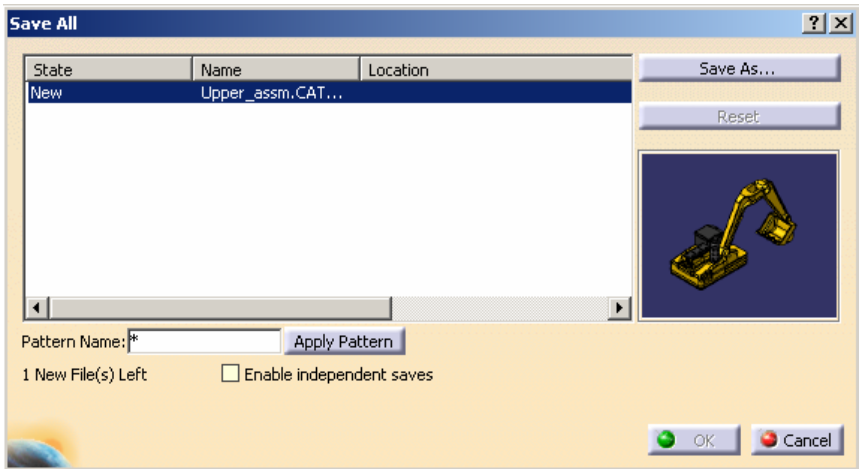
To save the file:-

- As there is a modification (hiding all planes) in each part file, we should save all documents again.

- Select "**File/Save all**"
- Click OK to close this message box (because you have to define the file location of the new Product file, "Upper_assm"
- Click "Save As…" icon
- Enter "Upper_assm_a.CATProduct" as filename and save it in your project folder.

- **Close All files.**

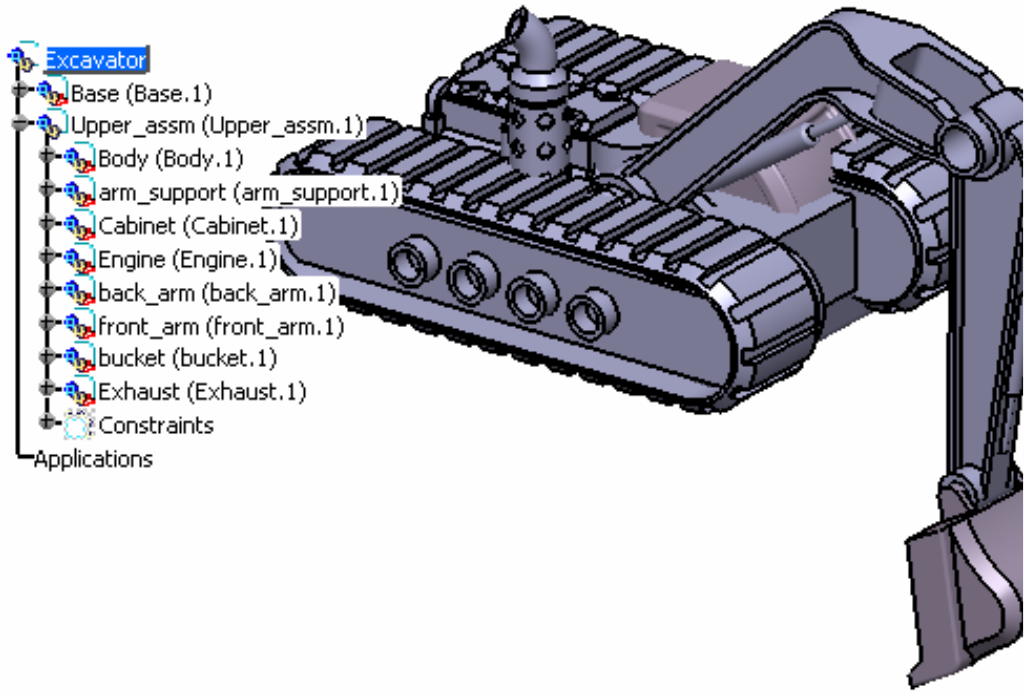




Tutorial 1C

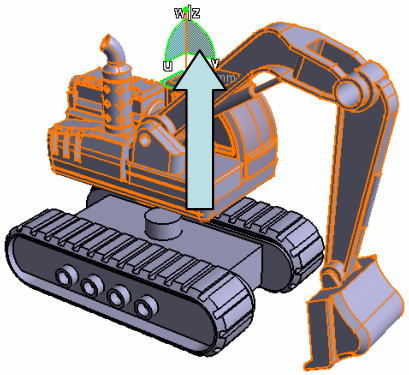
To assemble the upper assembly to the base:-

- Select "**Start/Mechanical Design/Assembly Design**" on the menu bar; A new product is then created.
- Rename the product tree as "**Excavator**"
- Insert an existing part… "Base_a.CATpart" in your folder
- Insert an existing product… "Upper_assm_a.CATproduct" in your folder

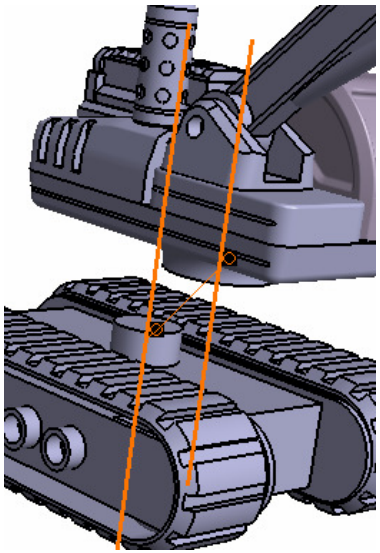


Tutorial 1C

Link "Upper_assm" to "Base"

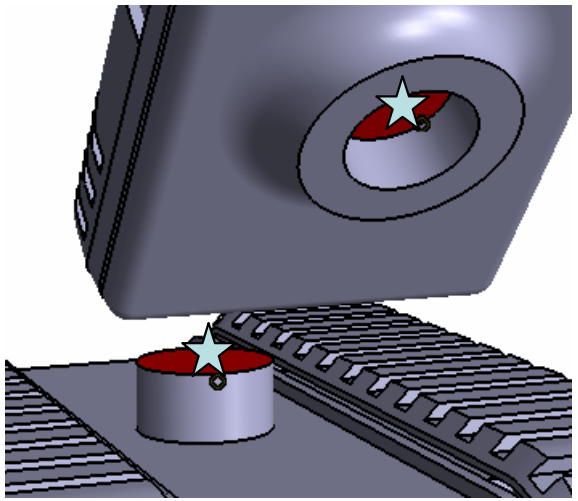


- Drag the upper_assm upward with the compass so that we can see the whole model



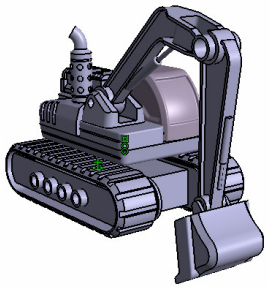
- Add a "**Fix Component**" constraint on the base

- Add a "**Coincidence**" constraint to align them



- Add a "**Contact**" constraint between the faces with ☆

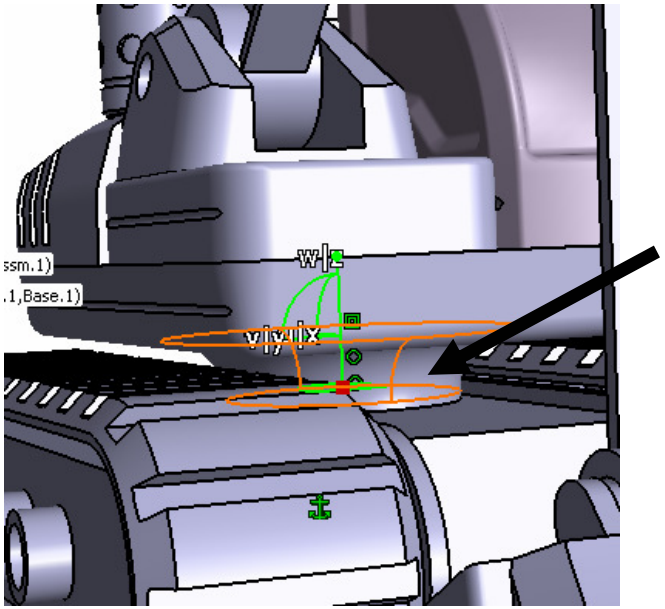
- Click "**Update**" icon 



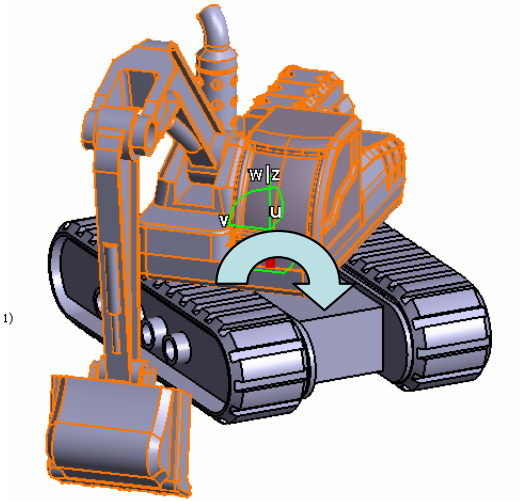
Tutorial 1C

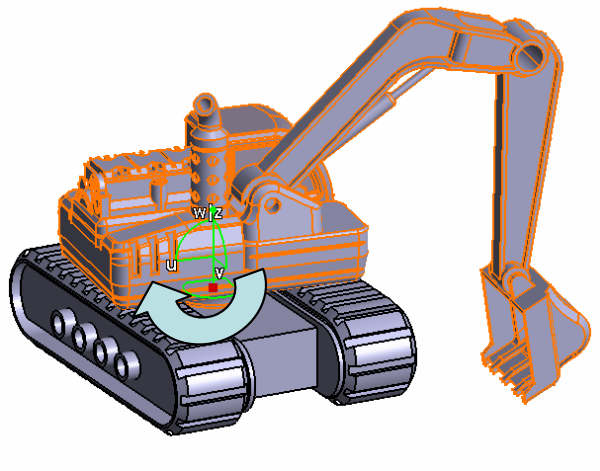
To simulate the angular motion of the upper assembly

- Drag the compass onto the circular surface under the upper assembly; The compass should snap onto the axis of the rotation.



- Rotate the upper assembly with compass





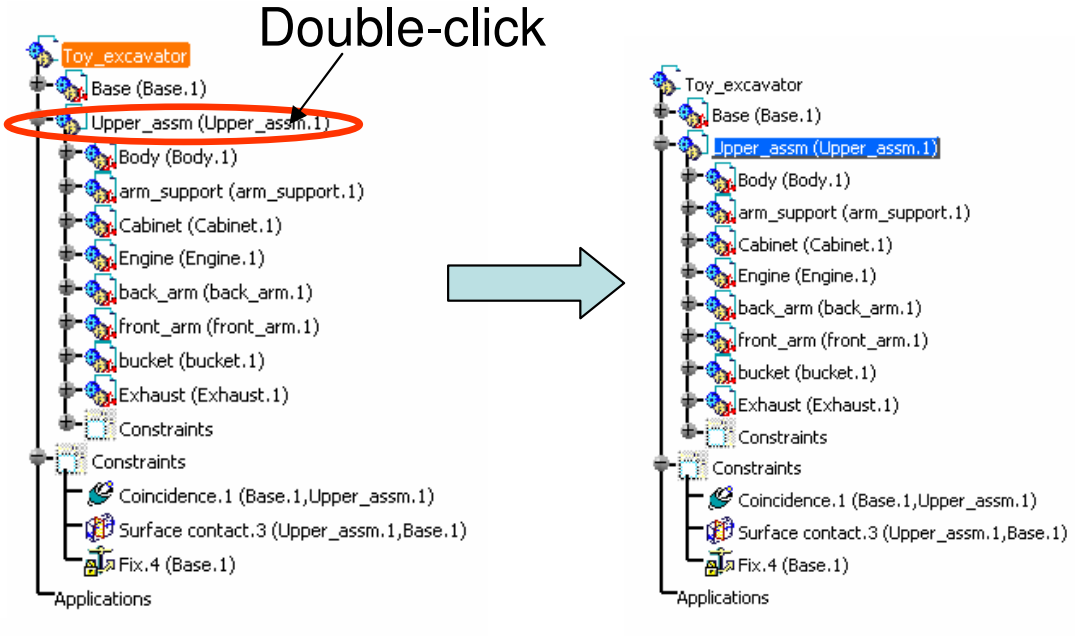
By Dickson Sham (ME dept, HKPU)

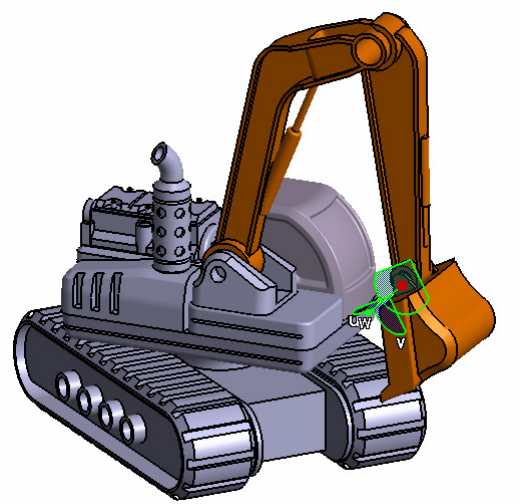
Tutorial 1C

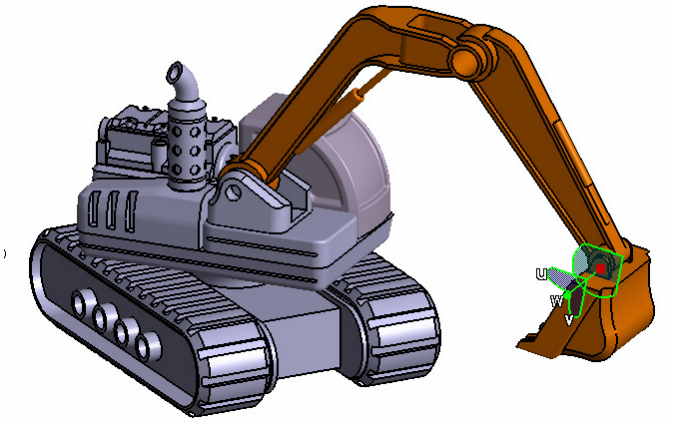
To simulate the angular motion of the machine arm:-

- Double-click “Upper_assm” on the tree to activate that level.

- Now, you can move the arm with the compass individually as before.



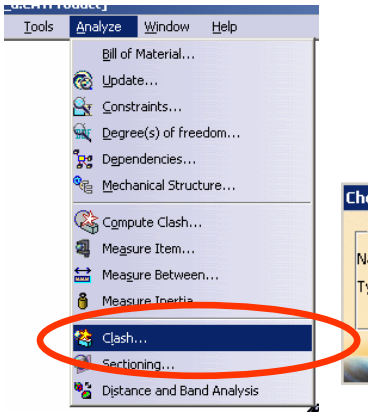


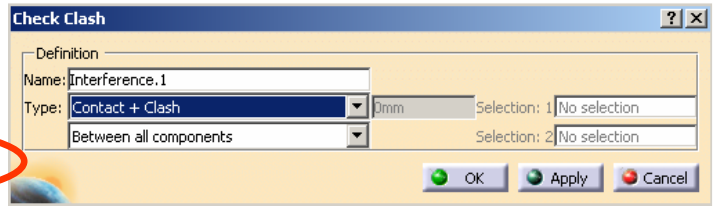


Tutorial 1C

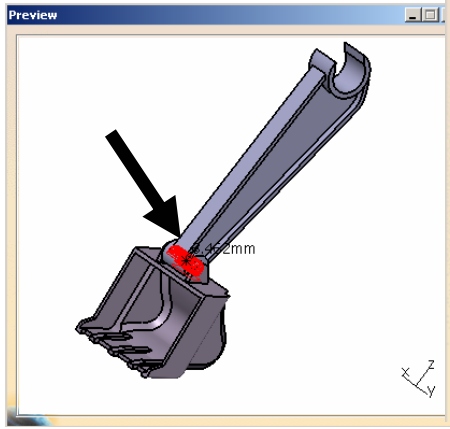
To check any collision among parts:-

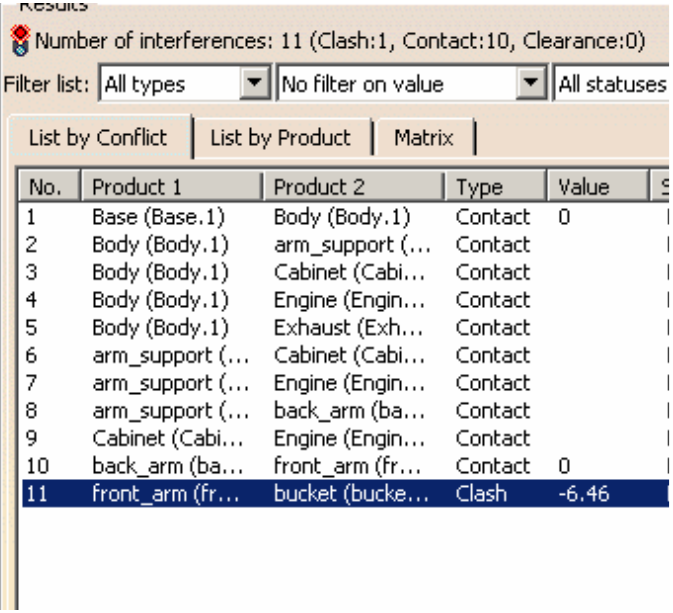
- Select “**Analyze/Clash…**”
- Select “**Contact + Clash**” as Type
- Select “**Between all components**”
- Click “**apply**” to view the result





- On the list of Conflict, we find a Clash, which happens between “front_arm” and “bucket”





No.	Product 1	Product 2	Type	Value
1	Base (Base.1)	Body (Body.1)	Contact	0
2	Body (Body.1)	arm_support (...	Contact	
3	Body (Body.1)	Cabinet (Cabi...	Contact	
4	Body (Body.1)	Engine (Engin...	Contact	
5	Body (Body.1)	Exhaust (Exh...	Contact	
6	arm_support (...	Cabinet (Cabi...	Contact	
7	arm_support (...	Engine (Engin...	Contact	
8	arm_support (...	back_arm (ba...	Contact	
9	Cabinet (Cabi...	Engine (Engin...	Contact	
10	back_arm (ba...	front_arm (fr...	Contact	0
11	front_arm (fr...	bucket (bucke...	Clash	-6.46

- The interference area is highlighted in **RED** in the PREVIEW window

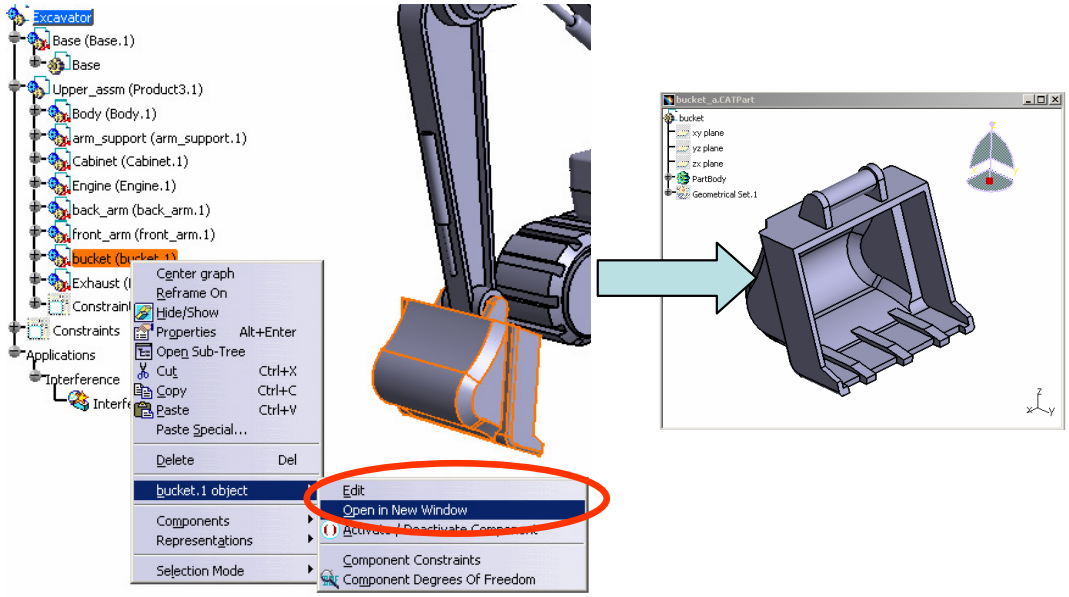
- Now we know where the problem is and we are going to correct the bucket design…

By Dickson Sham (ME dept, HKPU)

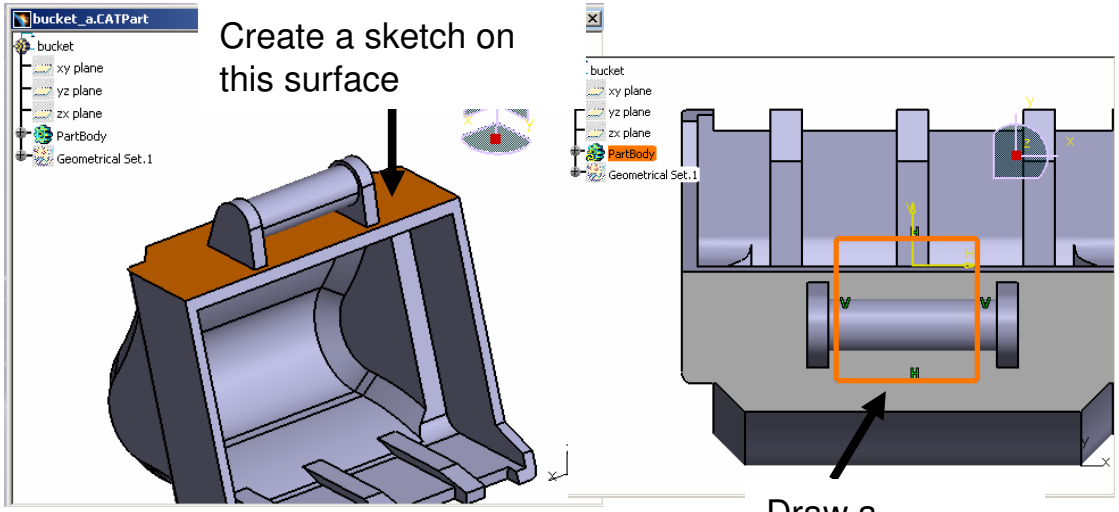
Tutorial 1C

To modify the error part:-

- Extend the Product Tree and locate the error part “Bucket”
- Single-click on it and right-click to show the contextual menu; then select “bucket.1.object/ **Open in new window**”



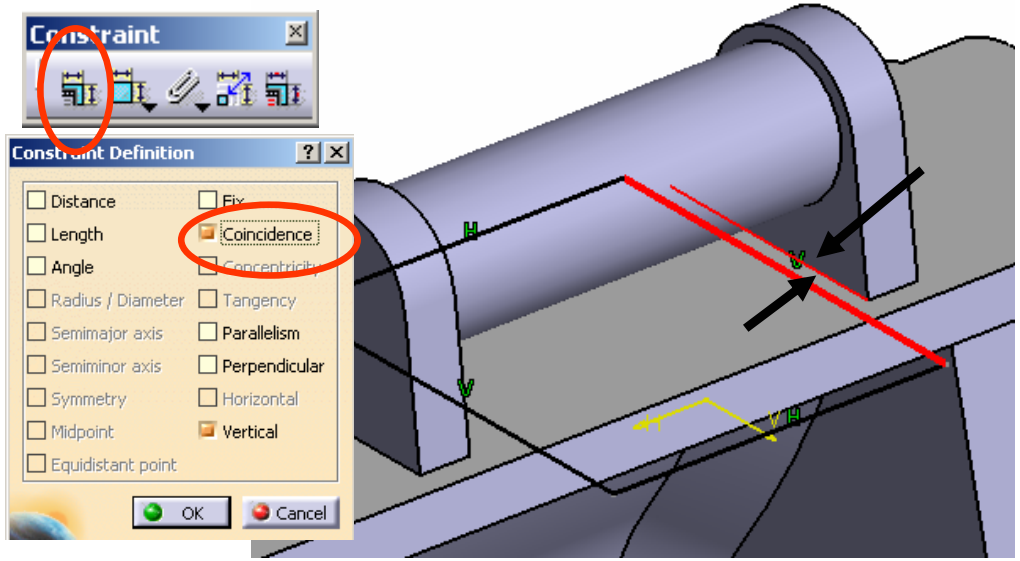
- Select “**Sketch**” icon and select the planar surface under the joint
- Draw a rectangle in the middle



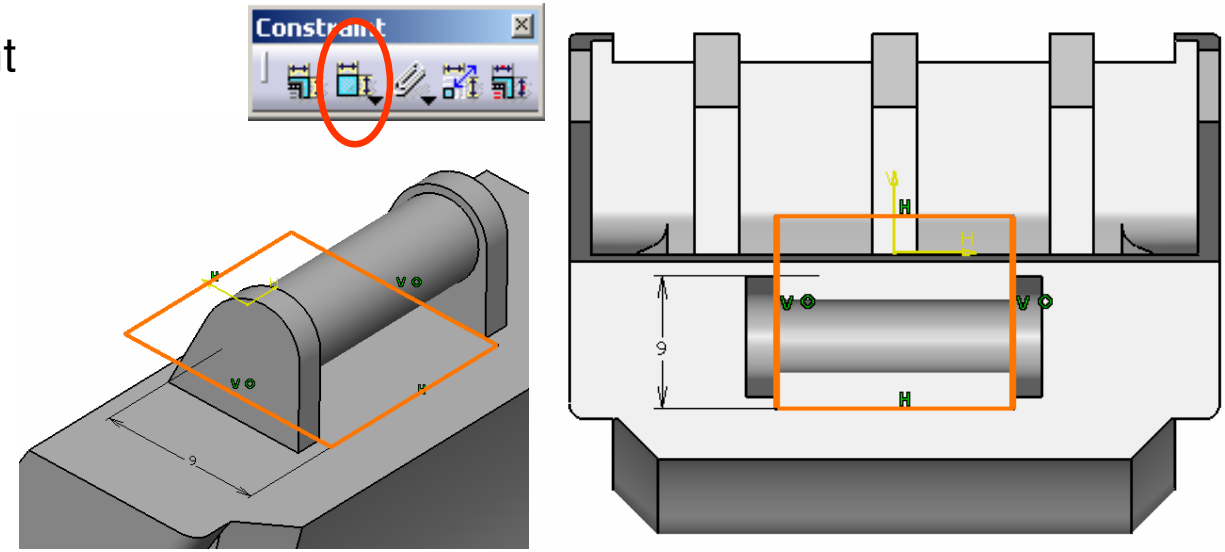
By Dickson Sham (ME dept, HKPU)

Tutorial 1C

- Add a "**coincidence**" constraint to align an edge of the rectangle onto the solid edge nearby.
- Similarly, align the edge on the opposite side.

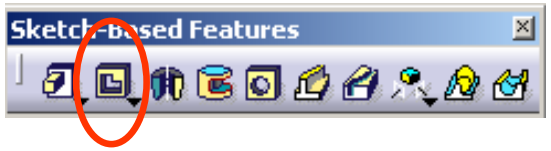


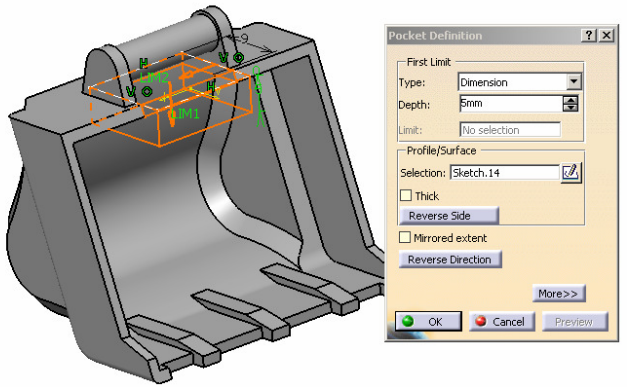
- Add a **dimensional** constraint 9mm as shown
- Exit to complete



Tutorial 1C

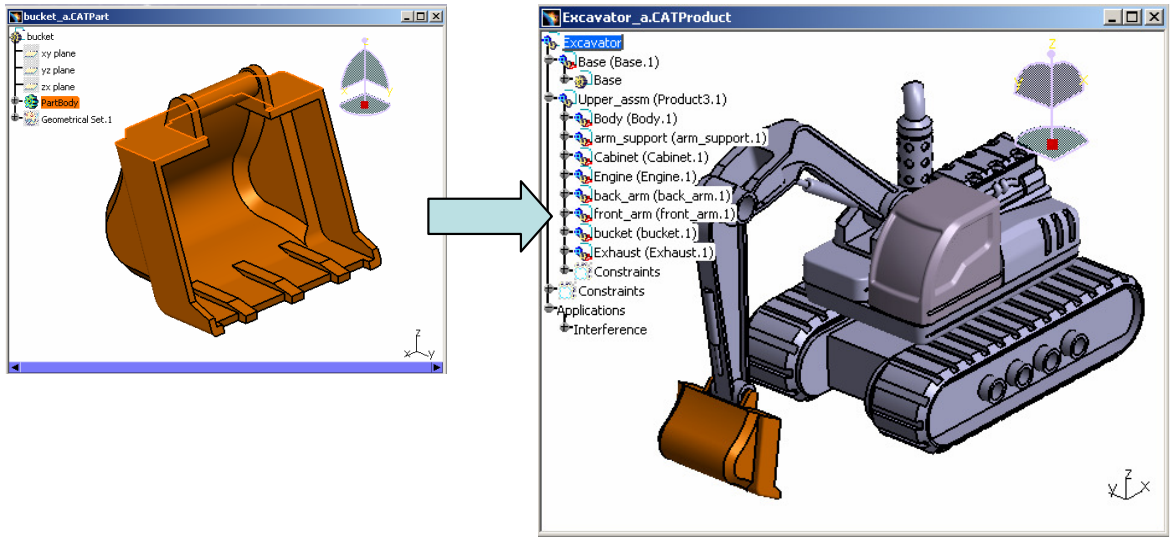
- Click “**Pocket**” icon
- Enter 5mm as First Limit
- Click Ok to complete





To re-do the clash analysis of the whole assembly:-

- Select “ Window/Excavator” to switch the display back to the whole assembly.
- You can see that the “Bucket” of the assembly has been updated with an opening.

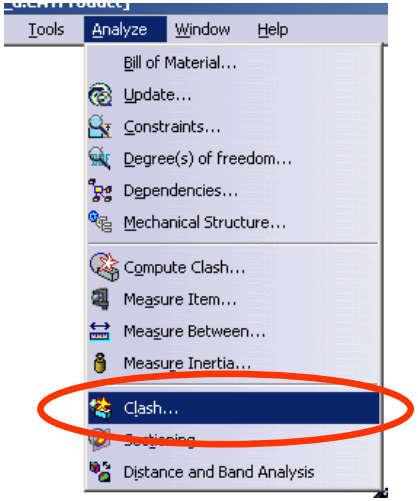


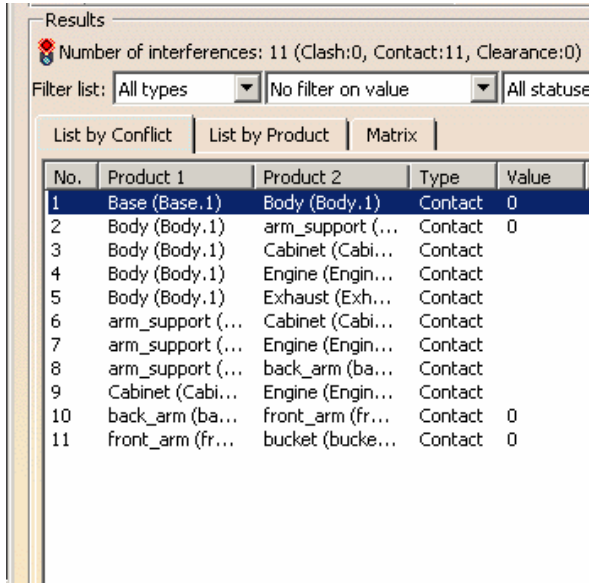
By Dickson Sham (ME dept, HKPU)

Tutorial 1C

- Select “**Analyze/Clash…**” on the menu bar.
- Select “**Contact + Clash**” as Type
- Select “**Between all components**”
- Click “**apply**” to view the result…

- There should be no clash error on the list now.



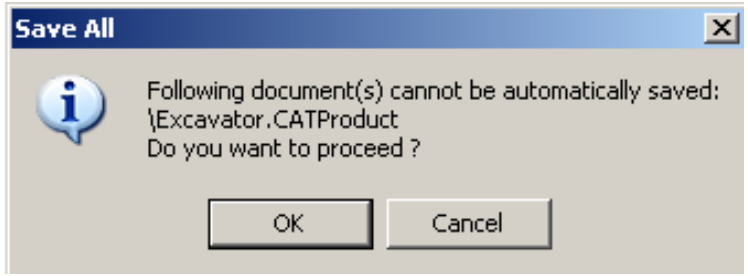


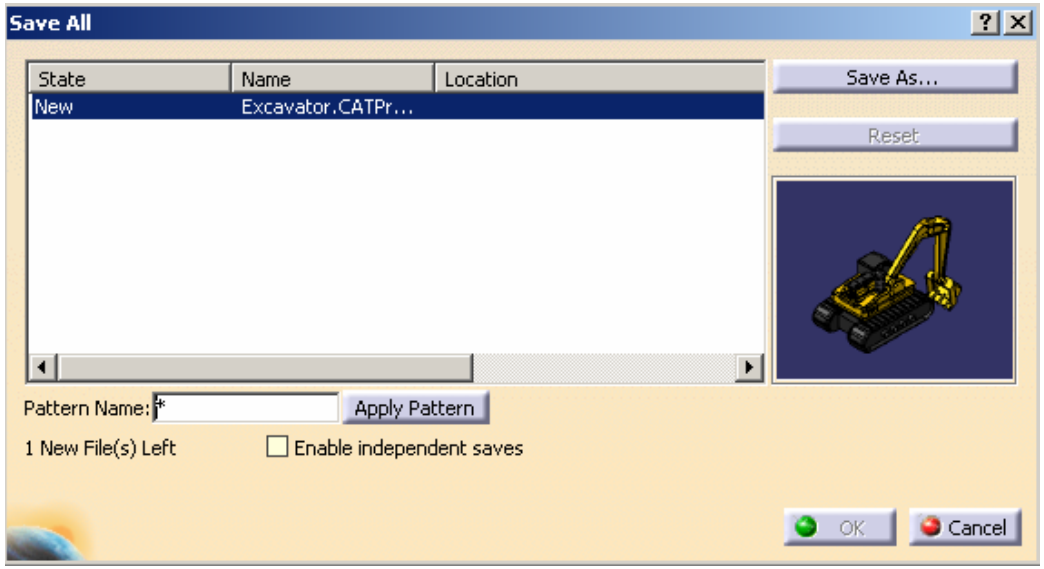
By Dickson Sham (ME dept, HKPU)

Tutorial 1C

To save the file:-

- (Hide all constraints)
- (Hide all datum planes)
- Select "**File/Save all**"
- Click OK to close this message box (because you have to define the file location of the new Product file)
- Click "Save As…" icon
- Enter "Excavator_a.CATProduct" as filename and save it in your project folder.
- **Close All files.**



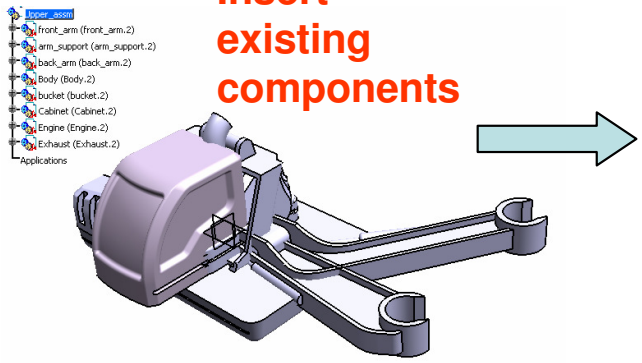


END of Tutorial 1C

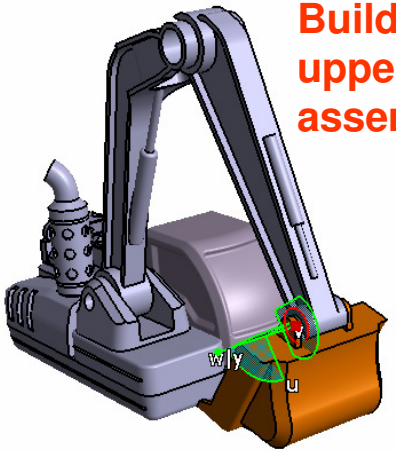
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Summary of Tut-1C

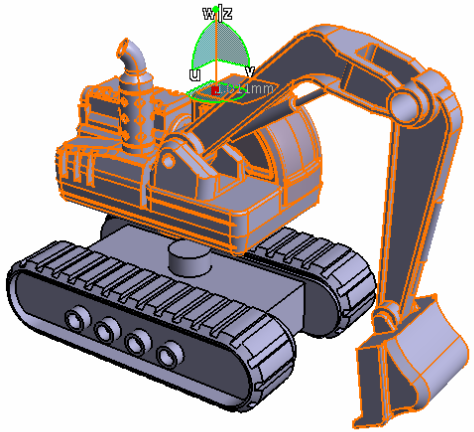
Assemble parts:-



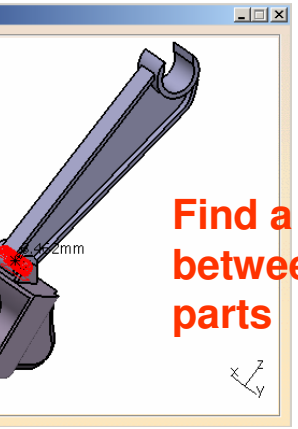
Insert existing components



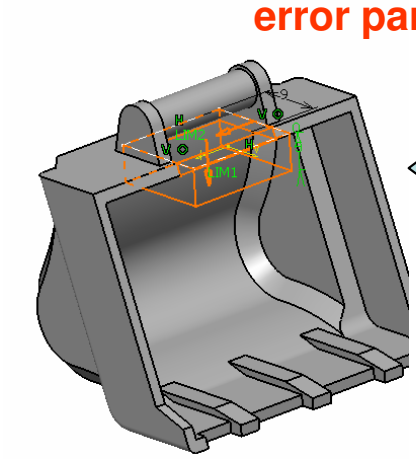
Build an upper assembly



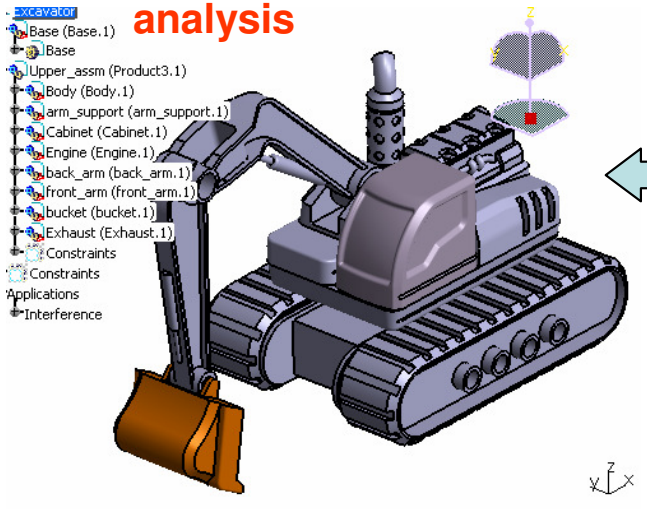
Put the upper assembly onto the base



Find a clash between parts



Modify the error part



Redo the clash analysis

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