CATIA V5 Surface-modeling

(Tutorial 2-Mouse)

GSD (Surface-modeling)

Part Design (Solid-modeling)

Assembly Design

A- 1

By Dickson Sham (ME dept, HKPU)

CATIA Surface-modeling

Tutorial 2A

- – Import 2D outline drawing into Catia
- – Build 3D curves based on the imported drawing
- – Build the upper surfaces of the mouse (by Generative Shape Design)

Tutorial 2B

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

Tutorial 2C

- – Build the parting surfaces based on the imported drawing
- – Create components from the finished model
- – Re-assemble the components into a product
- – Modify the outlook of the master model and then get all components updated automatically
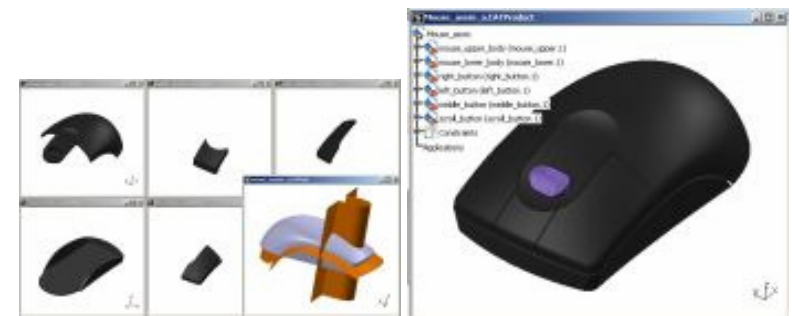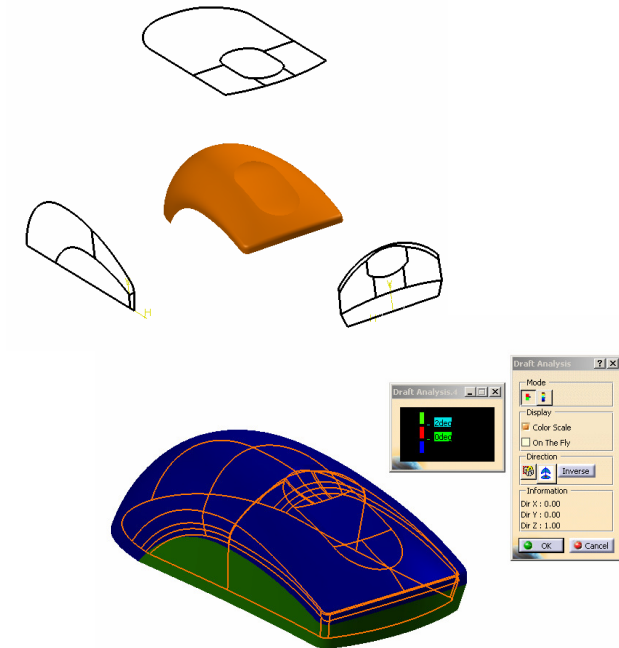
Please be reminded that this series of tutorials is designed to demonstrate a design approach with CATIA, rather than the command itself.

A- 2

Tutorial 2A

- Download the 2d outline drawing (mouse_outline.dxf) from the web:

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

- Create a new project folder and store the downloaded file into the folder

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

- (If the license menu pops up, select **ED2** and close CATIA. Then reopen again).

- By default, an empty "Product" file is created. But now, you don't need this, just select "**File/Close**" on the menu bar.

- Select "**File/Open**" on the menu bar and select the downloaded drawing (**mouse_outline.dxf**)
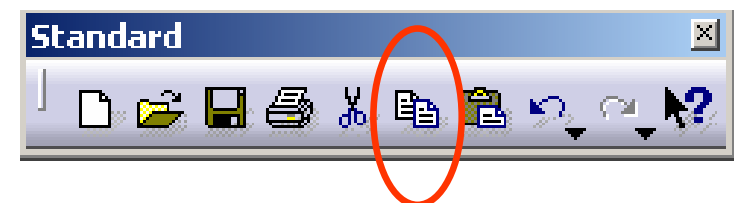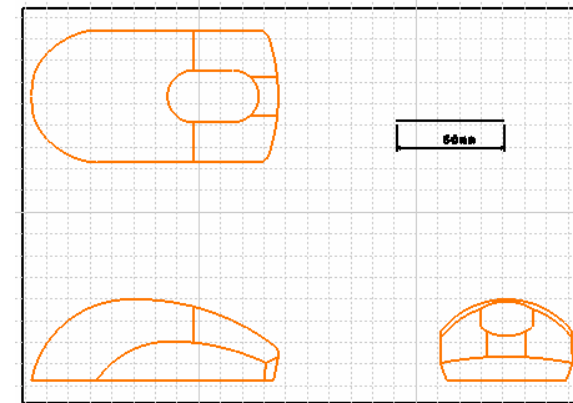
A- 3
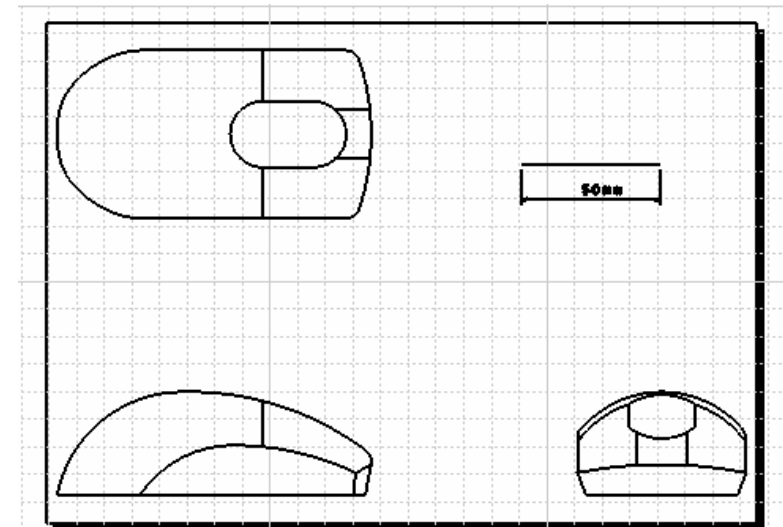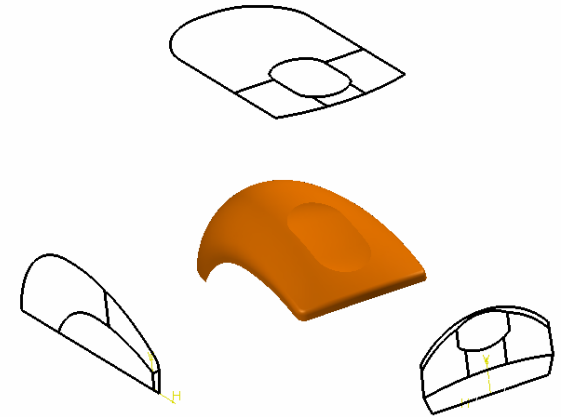
Tutorial 2A

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

- Click "**Dimensions**" icon;
- Click on the scale line of the drawing;
- Check if the displayed dimension is 50mm; If not, we need to enlarge or shrink the drawing into the correct size.
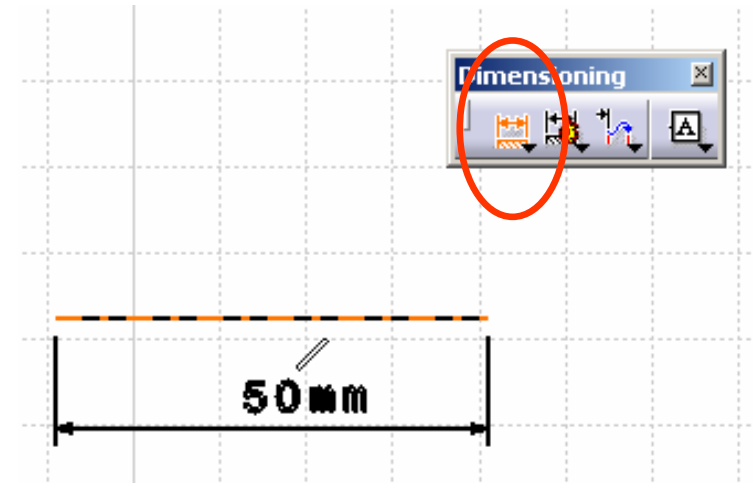
To copy and paste the drawing into 3D space:-

- Multi-select all entities on the drawing, except the scale bar;
- Click "**Copy**" icon

A- 4

Tutorial 2A

- Select "**File/New**" on the menu bar;

- Select "**Part**" as the Type;

- Enter "**Mouse_mastermodel**" as part name;

- Leave the two options "Enable hybrid design" & "Create a geometrical set" unchecked; now a new empty part is created;
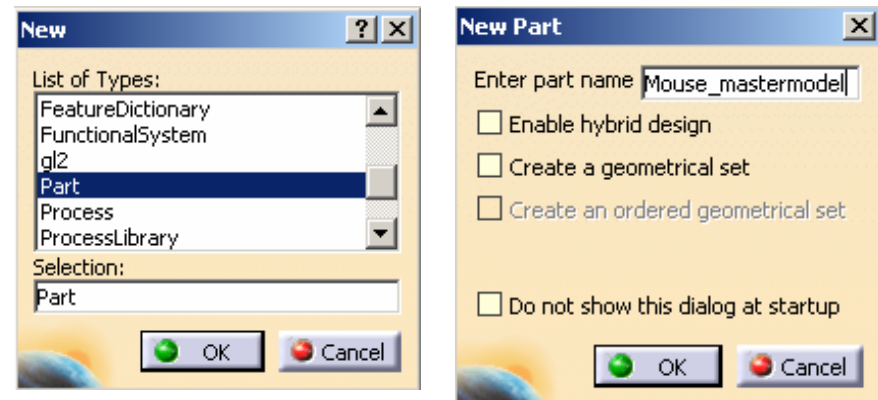
- Select Tools/Options/infrastructure/Part Infrastructure… then **deselect the option "Enable Hybrid Design inside part bodies and bodies"**
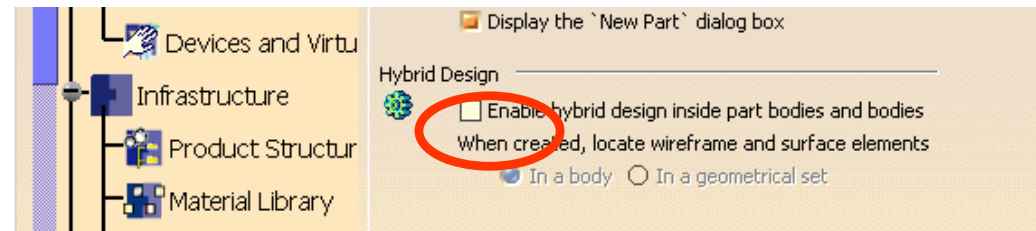
- Check if the current workbench has been "Generative Shape Design". You can see the workbench icon at the upper right-hand corner. If you cannot find the icon, reset the toolbar layout;
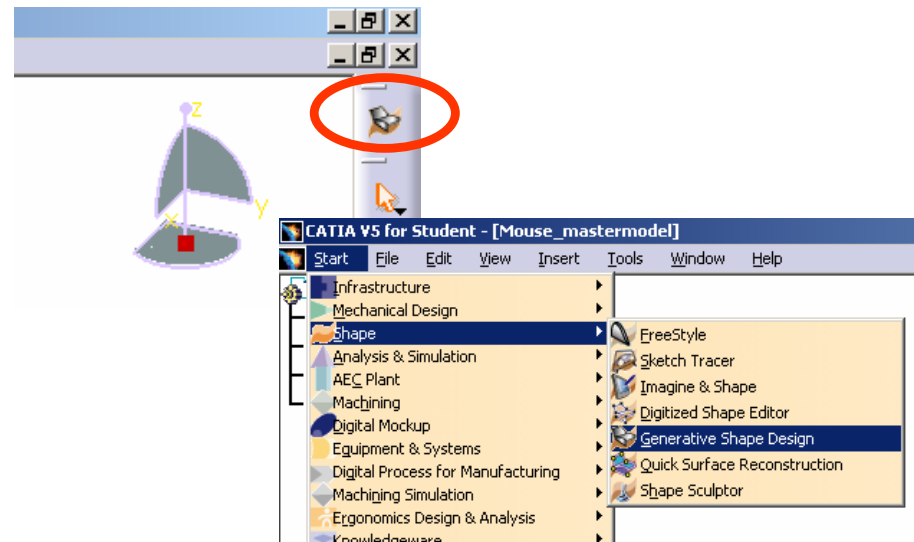
- If the current workbench is "Part Design" for example, select "**Start/Shape/Generative Shape Design**" on the menu bar;

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Tutorial 2A

- Select "**Insert/Geometrical Set…**" on the menu bar; then click ok to confirm; (This geometrical set is going to store all three reference views of the mouse)
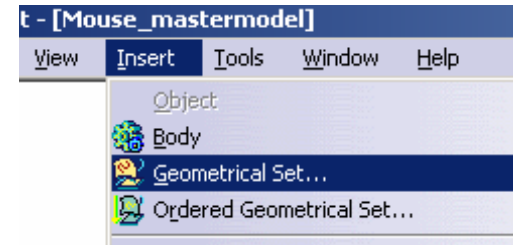

- Click "**Sketch**" icon and select **xy plane**;
- Click "**Paste**" icon to paste the drawing onto the xy plane;
- Click "Exit" icon to exit the sketcher mode. (Now "Sketch.1" is stored in "Geometrical Set.1")

A- 6
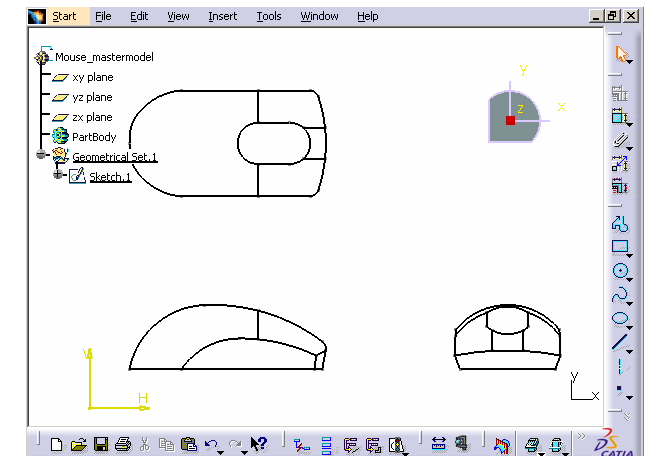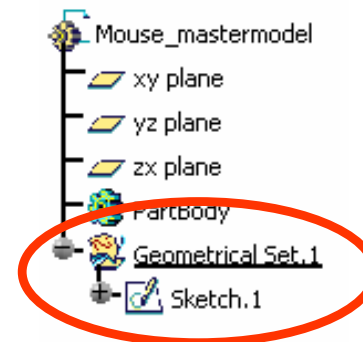
Tutorial 2A

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

- Duplicate two more "Sketch.1" by copy-and-paste function;
- Rename them as "Top View", "Side View" & "Front View" ;

- Click "**Plane**" icon;
- Select "offset from plane" as type;
- Pick xy plane as reference;
- Enter **150mm** as Offset value;
- Click ok to confirm;

- Create an offset plane, 150mm from **yz plane**;
- Create an offset plane, 150mm from **xz plane**;

A- 7
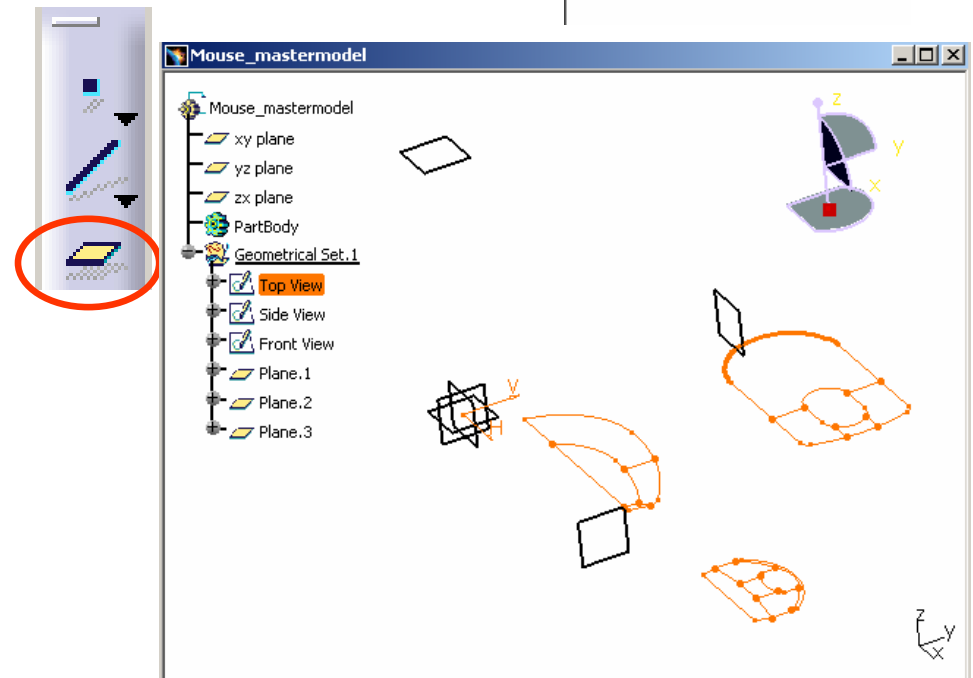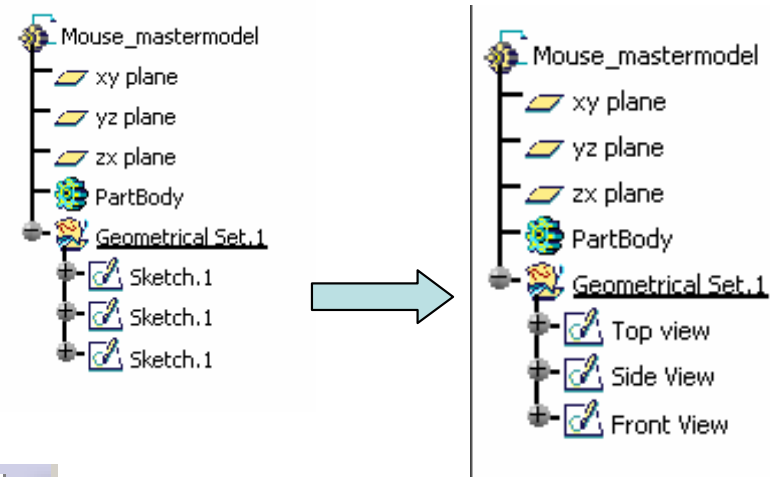
Tutorial 2A

- Right-click on "**Top View**" on the tree and select "Top View object/ **Change Sketch Support**";
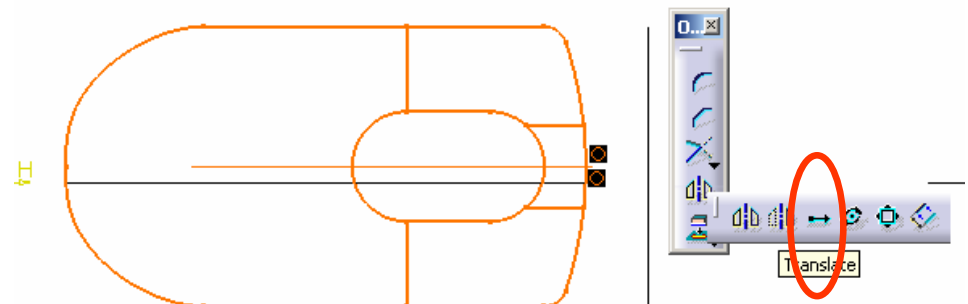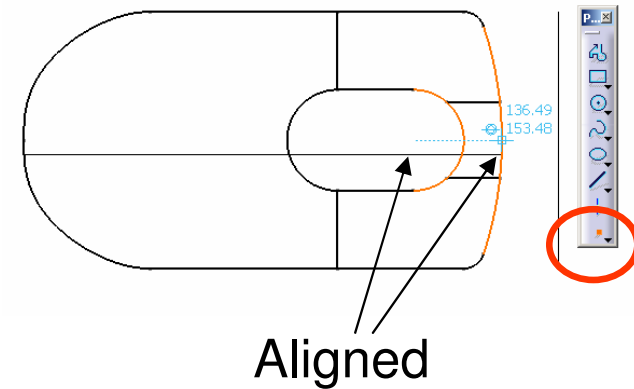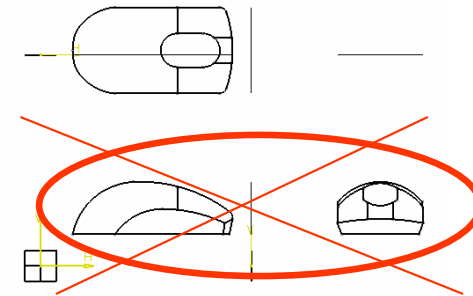
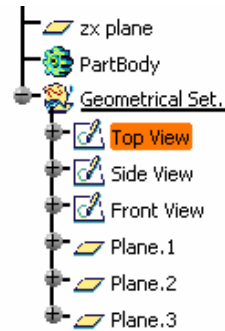- Select "**Plane.1**"

- Click ok to confirm

- Similarly, right-click "**Side View**" and select "**Change Sketch Support**";

- Select "**Plane.2**"

- Click ok to confirm

- Similarly, right-click "**Front View**" and select "**Change Sketch Support**";

- Select "**Plane.3**"

- Click ok to confirm

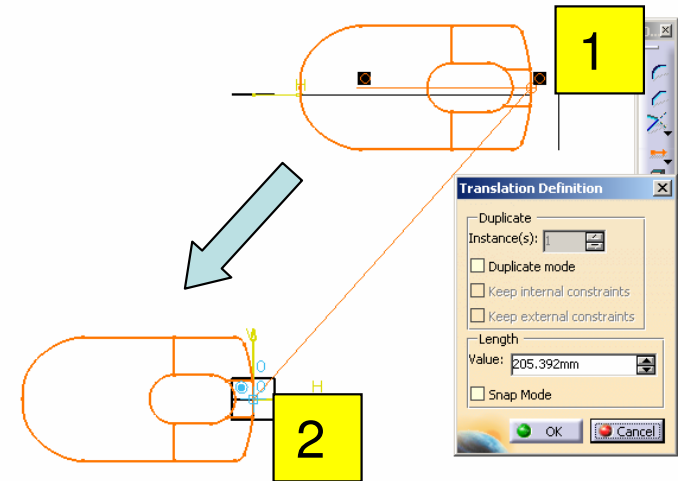A- 8

By Dickson Sham (ME dept, HKPU)
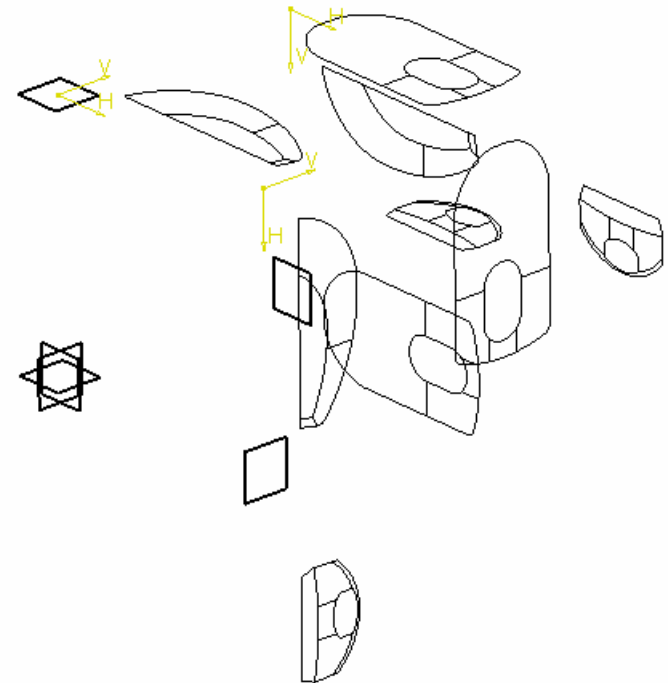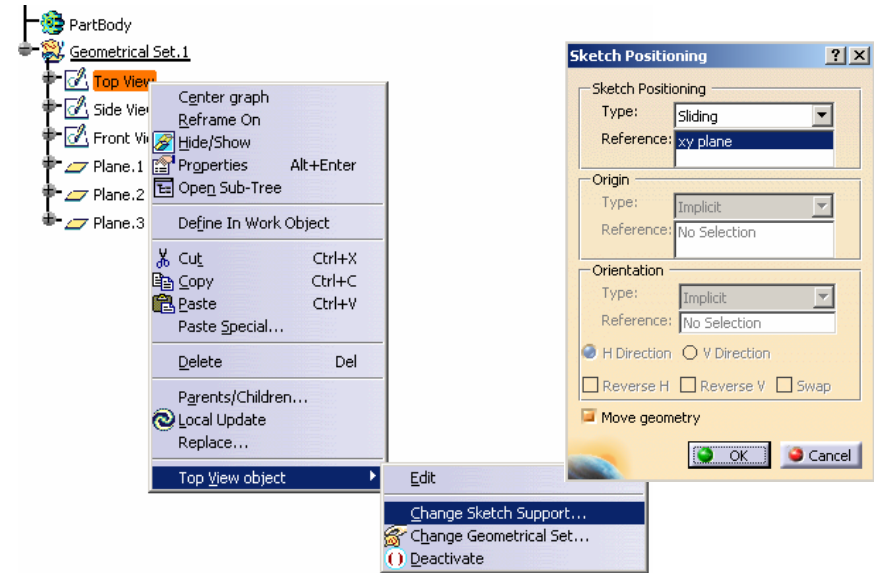
Tutorial 2A

- Double-click "**Top View**" sketch on the tree to edit;
- Select and delete the curves not related to the top view;

- Create a point at the rightmost of the shape (Click "**point**" icon, put the mouse cursor onto the rightmost arc, click to confirm its position when it is aligned on the same level as the center of the inner arc);

- Select all elements of the shape and click "**Translate**" icon;


Aligned

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Tutorial 2A

- Leave "Duplicate mode" unchecked;
- Click the point that we just created;
- Then Click the origin of the sketch. (Now the top view is relocated at the origin);
- Click "**Exit**" to complete.

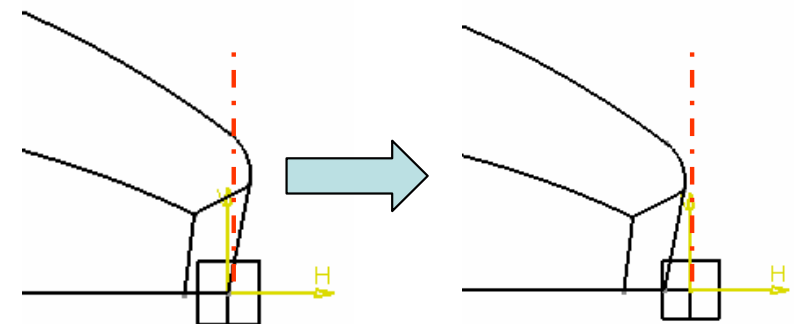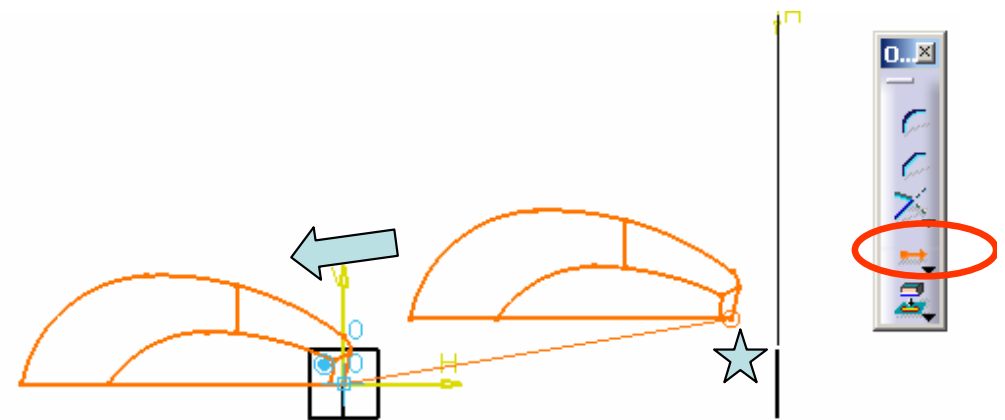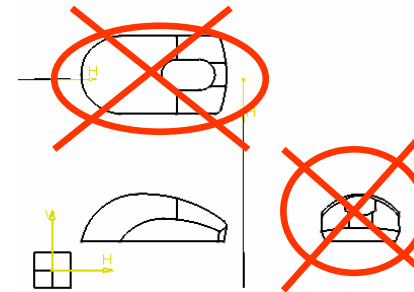Similarly, we can modify "**Side View**"…

- First we see the side view is upside down. To reverse it, right-click the "Side View" sketch on the tree and select "**Change Sketch Support**";
- Select "**Positioned**" as Type of sketch positioning;
- Select "**Implicit**" as both Origin Type & Orientation Type;
- Select "**Reverse H**" option;
- Click **ok** to confirm.

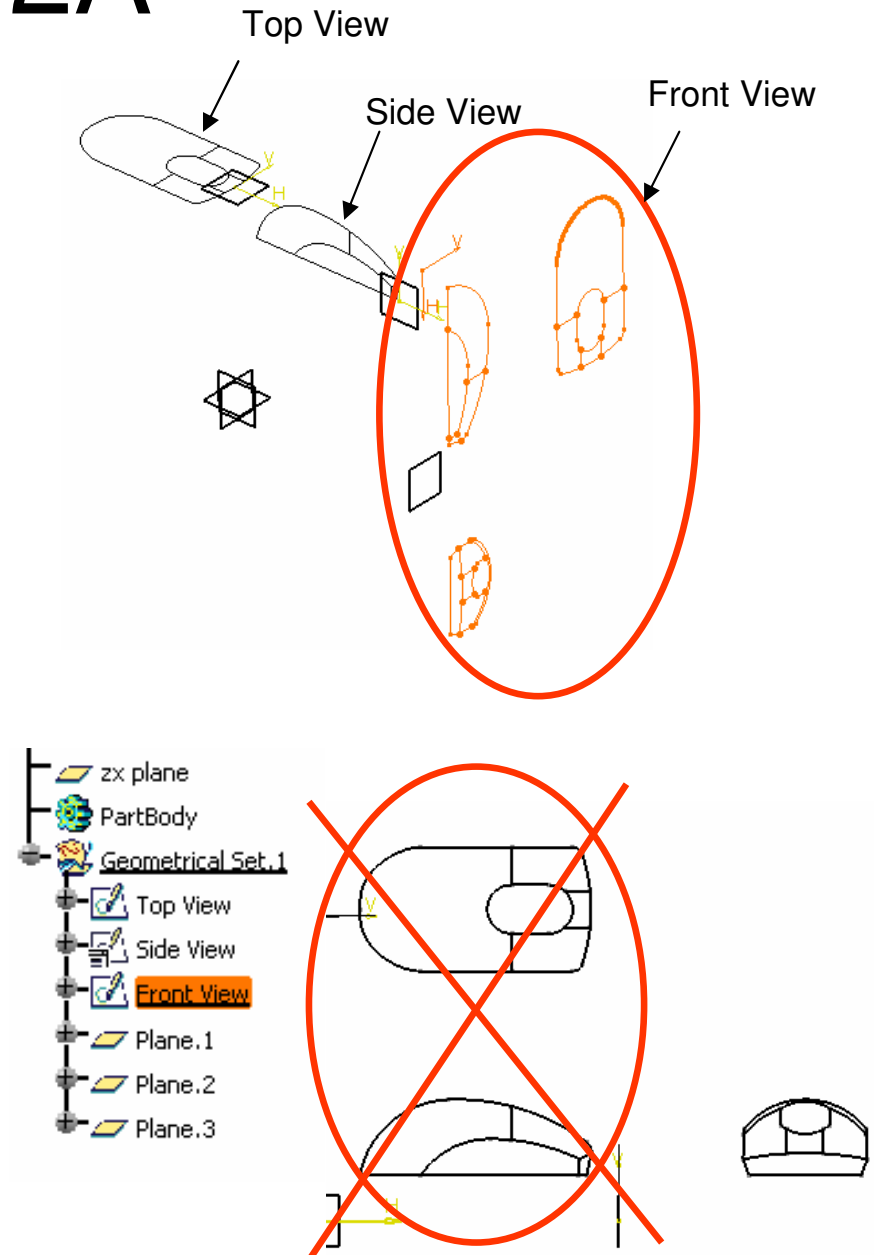A- 10

By Dickson Sham (ME dept, HKPU)
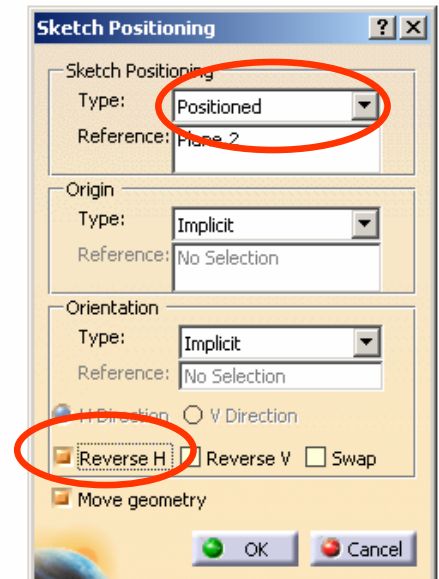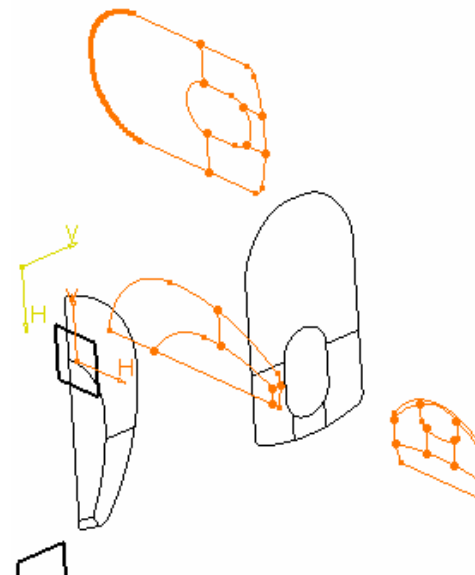
Tutorial 2A

- Double-click "**Side View**" sketch on the tree to edit;

- Select and delete the curves not related to the side view;

- Select all elements of the shape and click "**Translate**" icon;

- Leave "Duplicate mode" unchecked;

- Click the point ☆;

- Then Click the origin of the sketch. (Now we can see a portion of the sketch exceeds the y-axis, so we need to fine-tune it);

- Select all elements of the shape and click "**Translate**" icon again;

- Click the origin

- Enter **2.85mm** as length and press "**Enter**" key on the keyboard;

- Click another point on negative side of x-axis

- Click "**Exit**" icon to complete

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Tutorial 2A

Top View

Side View

Front View

Similarly, we can modify "**Front View**"…

- First we see the side view is orientated correctly. To adjust it, right-click the "Front View" sketch on the tree and select "**Change Sketch Support**";

- Select "**Positioned**" as Type of sketch positioning;

- Select "**Implicit**" as both Origin Type & Orientation Type;

- Click **ok** to confirm.

- Double-click "**Front View**" sketch on the tree to edit;

- Select and delete the curves not related to the front view;

```
zx plane
PartBody
Geometrical Set.1
    Top View
    Side View
    Front View
    Plane.1
    Plane.2
    Plane.3
```

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Tutorial 2A

- • Create a point at the middle of the lowest line (Click "**point**" icon, put the mouse cursor onto the lowest straight line, click to confirm its position when the auto-detect symbol is a solid blue circle);
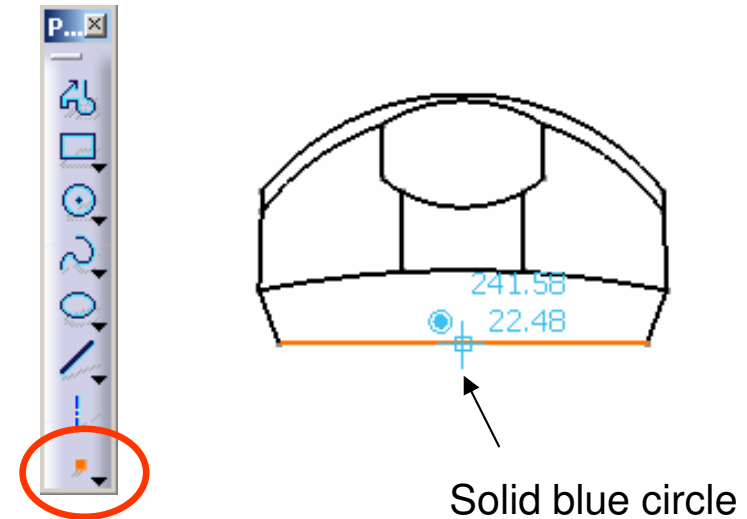

241.58

22.48

Solid blue circle

- • Select all elements of the shape and click "**Translate**" icon
- • Leave "Duplicate mode" unchecked;
- • Click the point that we just created;
- • Then Click the origin of the sketch. (Now the top view is relocated at the origin);
- • Click "**Exit**" to complete.

A point

Origin of sketch

A- 13
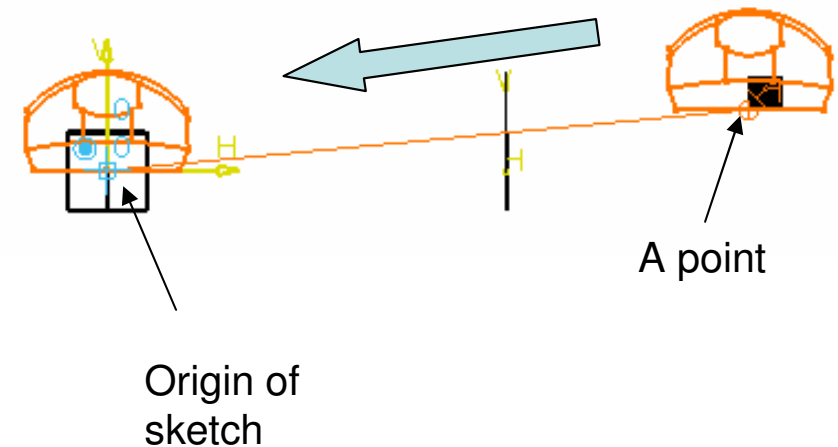
Tutorial 2A

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

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

- Right-click "Geometrical Set.1" on the tree and select "Properties";
- Enter "**Reference**" as Feature Name;
- **Deselect** "**Pickable**" option;
- Click ok to confirm.

  The reason why we choose the views are unpickable is that we don't want to use any curves from them or have any relations with them. We treat them as the background images only.

A- 14
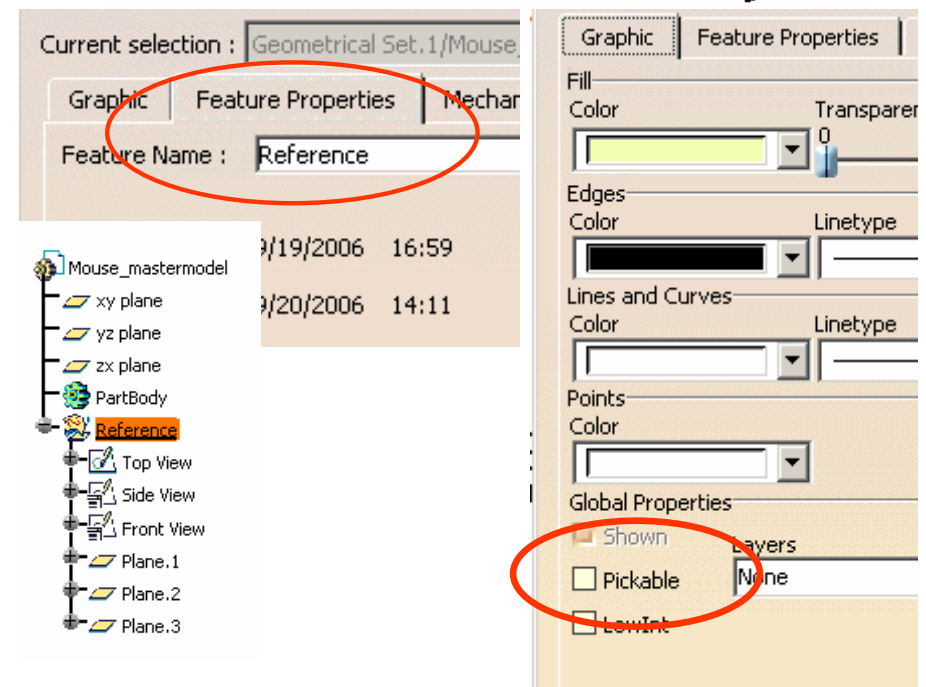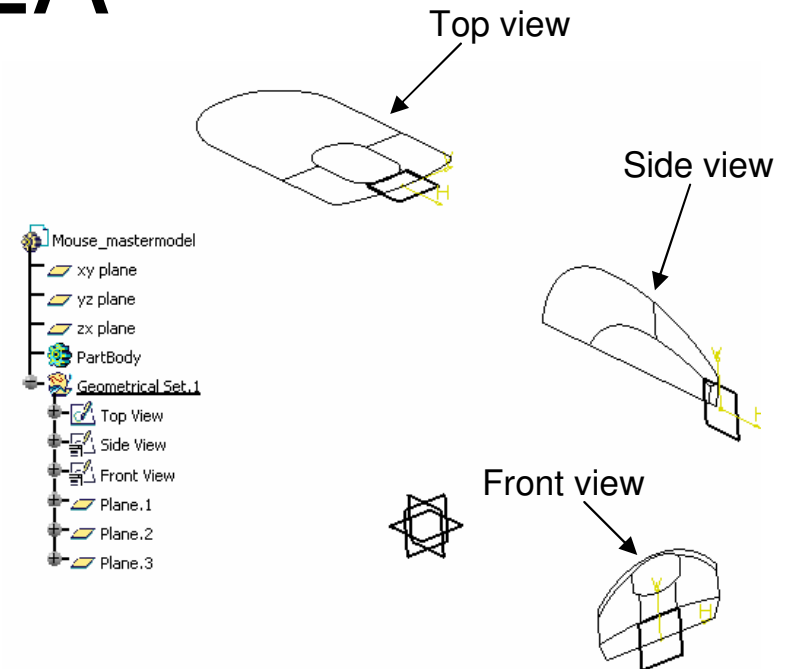
Tutorial 2A

To create 3D curves from the reference sketches:-

- Select "**Insert/Geometrical Set…**" on the menu bar (we are going to build a new folder to store new wireframe & surface elements);

- Click "**Plane**" icon and select "**Offset from Plane**";
- Select **xy plane** and enter **+50mm** as offset value;
- Click ok to confirm.

- Click "**Plane**" icon again and select "**Offset from Plane**";
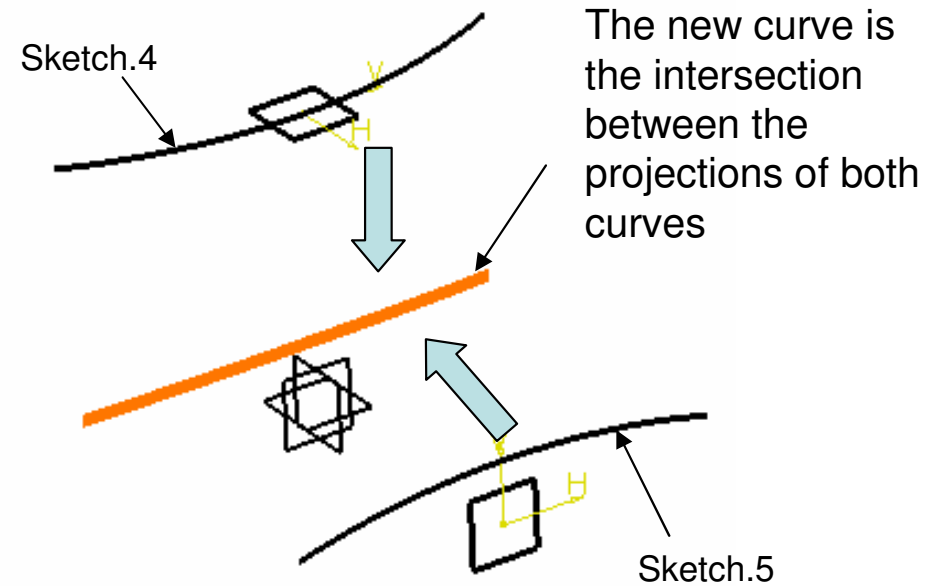- Select **yz plane** and enter **+50mm** as offset value;
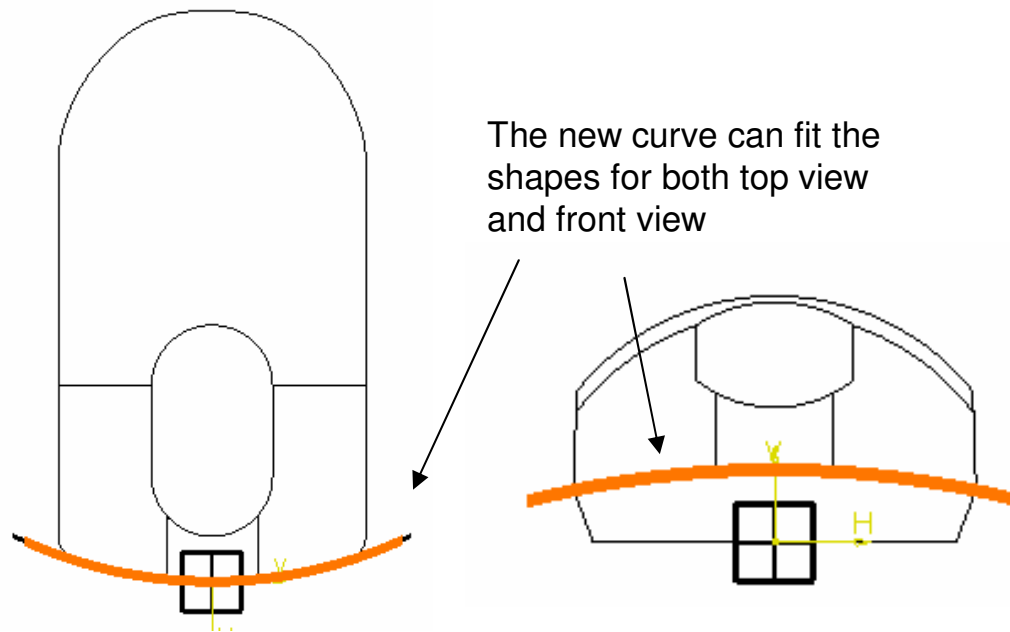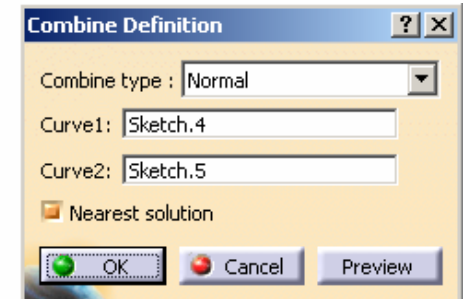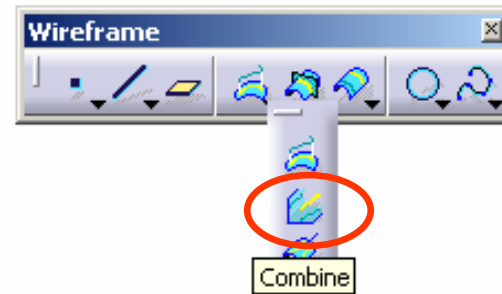- Click ok to confirm.

A- 15
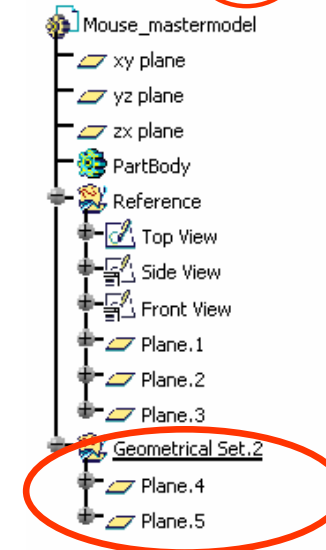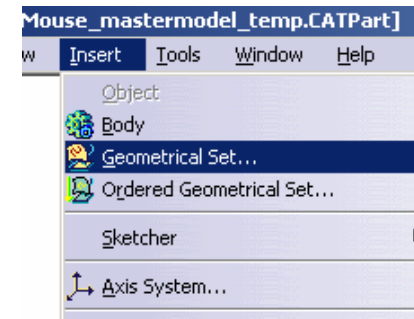
Tutorial 2A

- Click "**Sketch**" icon and select "Plane.4";
- Draw an **Arc** (R90mm), with two ends symmetric about the x-axis and the arc is tangent to the y-axis;
- Reminded that the arc is a little longer than the reference;
- Click "**Exit**" icon to complete.

- Click an open area to deselect the Sketch (Sketch.4);

- Click "**Sketch**" icon and select "Plane.5";
- Draw an **Arc** (R150mm), with two ends symmetric about the x-axis and the peak 11mm from the x-axis;
- Reminded that the arc should be a little bit longer than the reference;
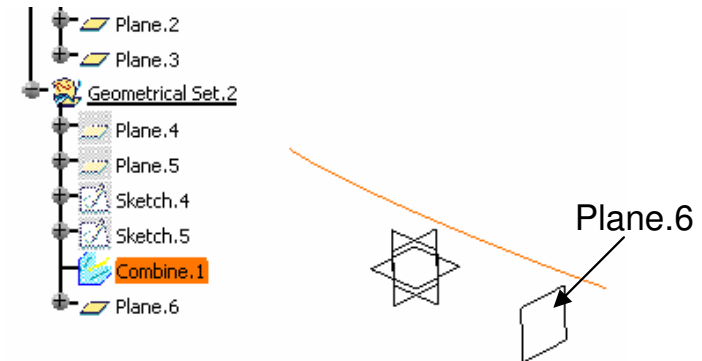- Click "**Exit**" icon to complete.

A- 16
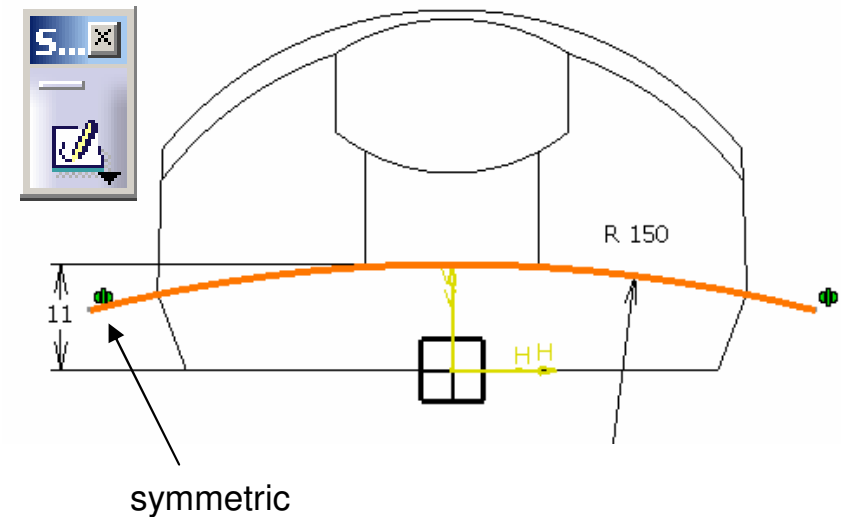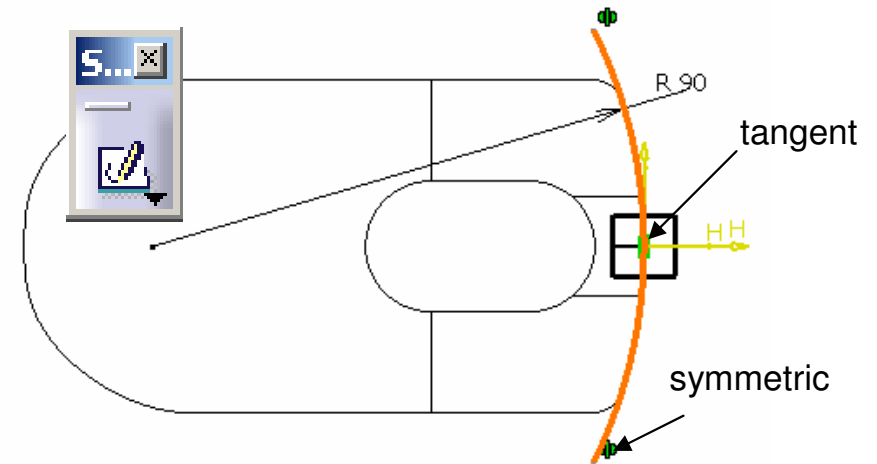
Tutorial 2A

- Click "**Combine**" icon;
- Select "**Sketch.4**" & "**Sketch.5**";
- Click ok to confirm.

- Then **Hide** "Plane.4", "Plane.5", "Sketch.4" and "Sketch.5"

Wireframe

Combine

Combine Definition

Combine type : Normal

Curve1: Sketch.4

Curve2: Sketch.5

☑ Nearest solution

● OK   ● Cancel   Preview

The new curve can fit the shapes for both top view and front view

Sketch.4

The new curve is the intersection between the projections of both curves

Sketch.5

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Tutorial 2A

- Click "**Plane**" icon and select "**Offset from Plane**";

- Select **zx plane** and enter **+30.5mm** as offset value;

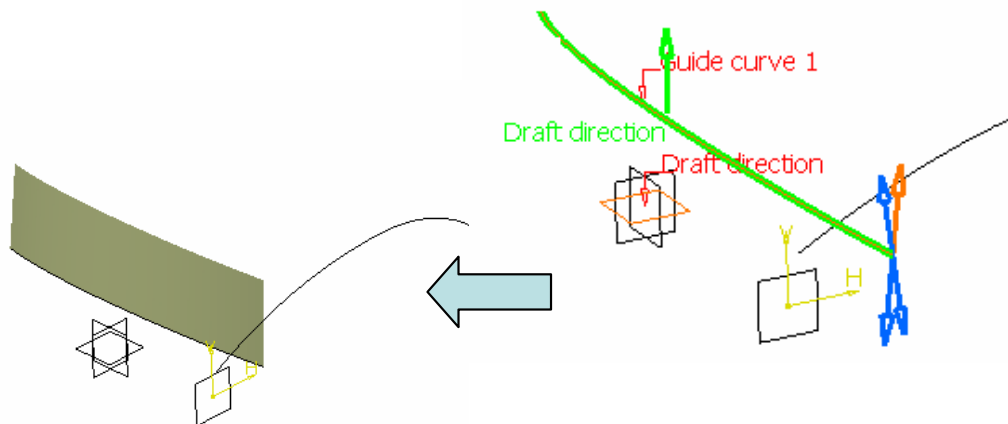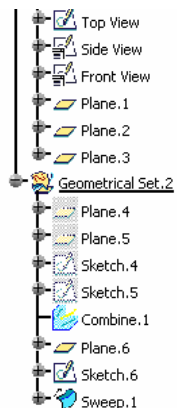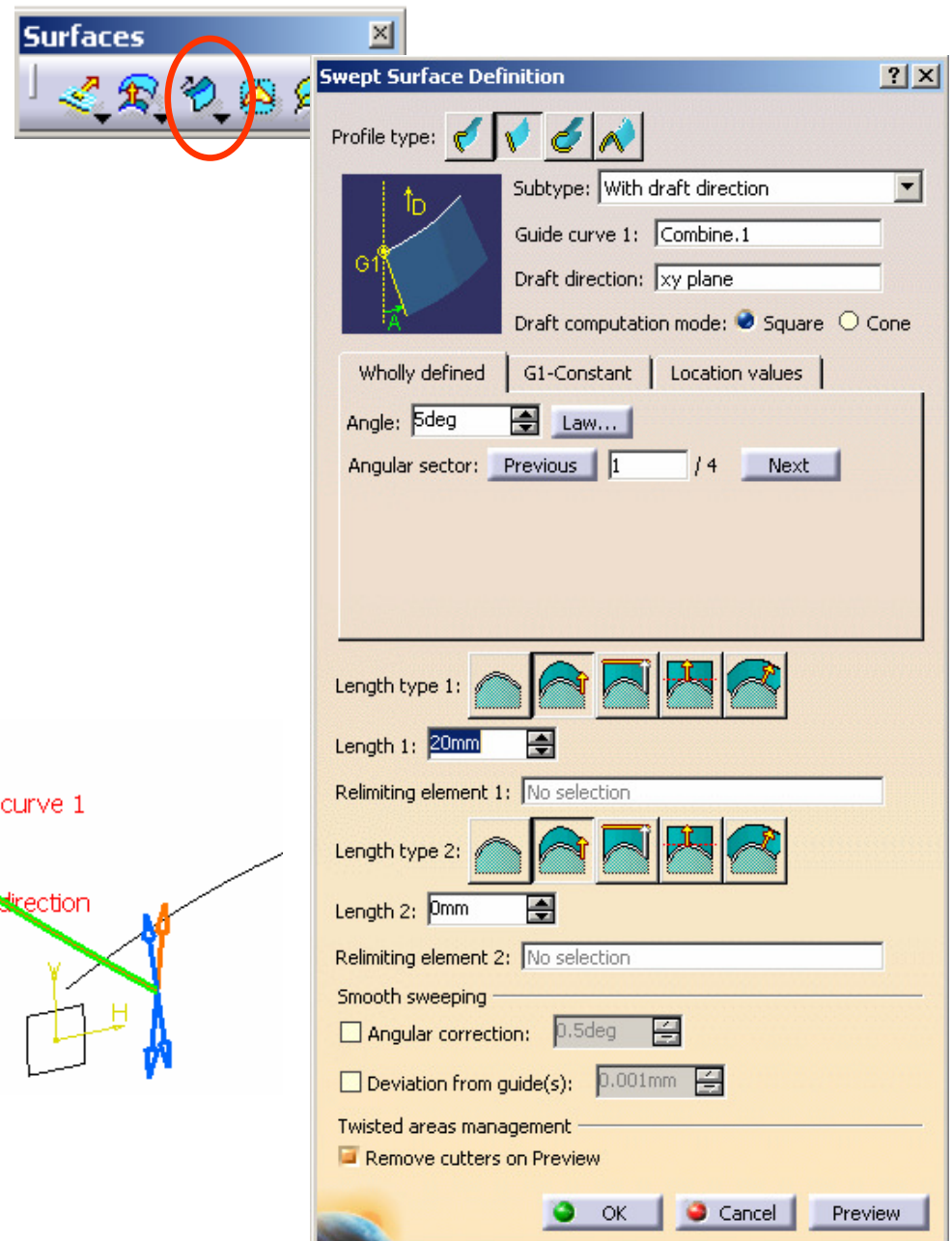- Click ok to confirm.

Plane.6

- Click "**Sketch**" icon and select the new plane "**Plane.6**";

- Draw **two arcs** as shown;

- Reminded that two arcs must be tangent to each other; one end of the small arc is touching the x-axis; one end of the bigger arc is just near y-axis;

- Click "**Exit**" icon to complete.
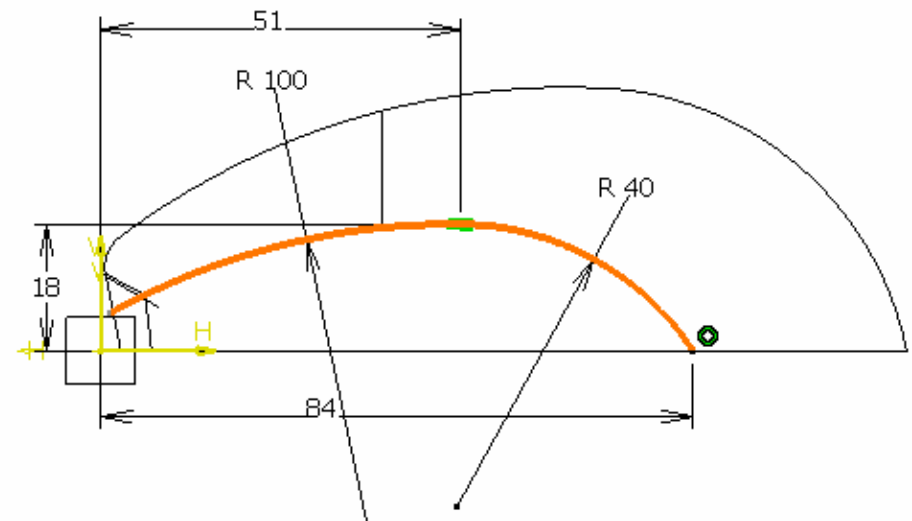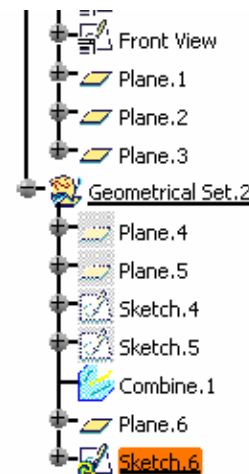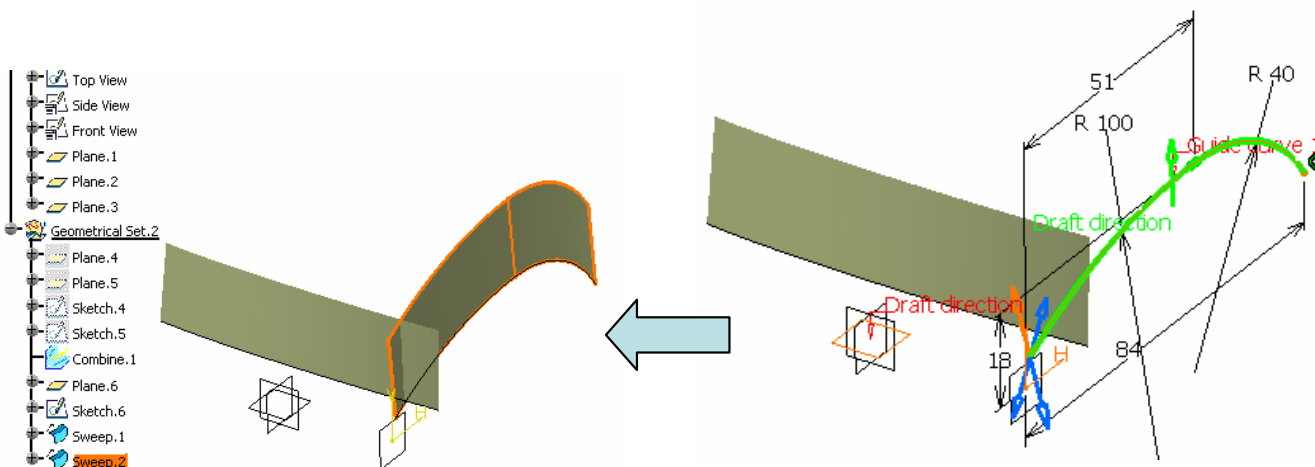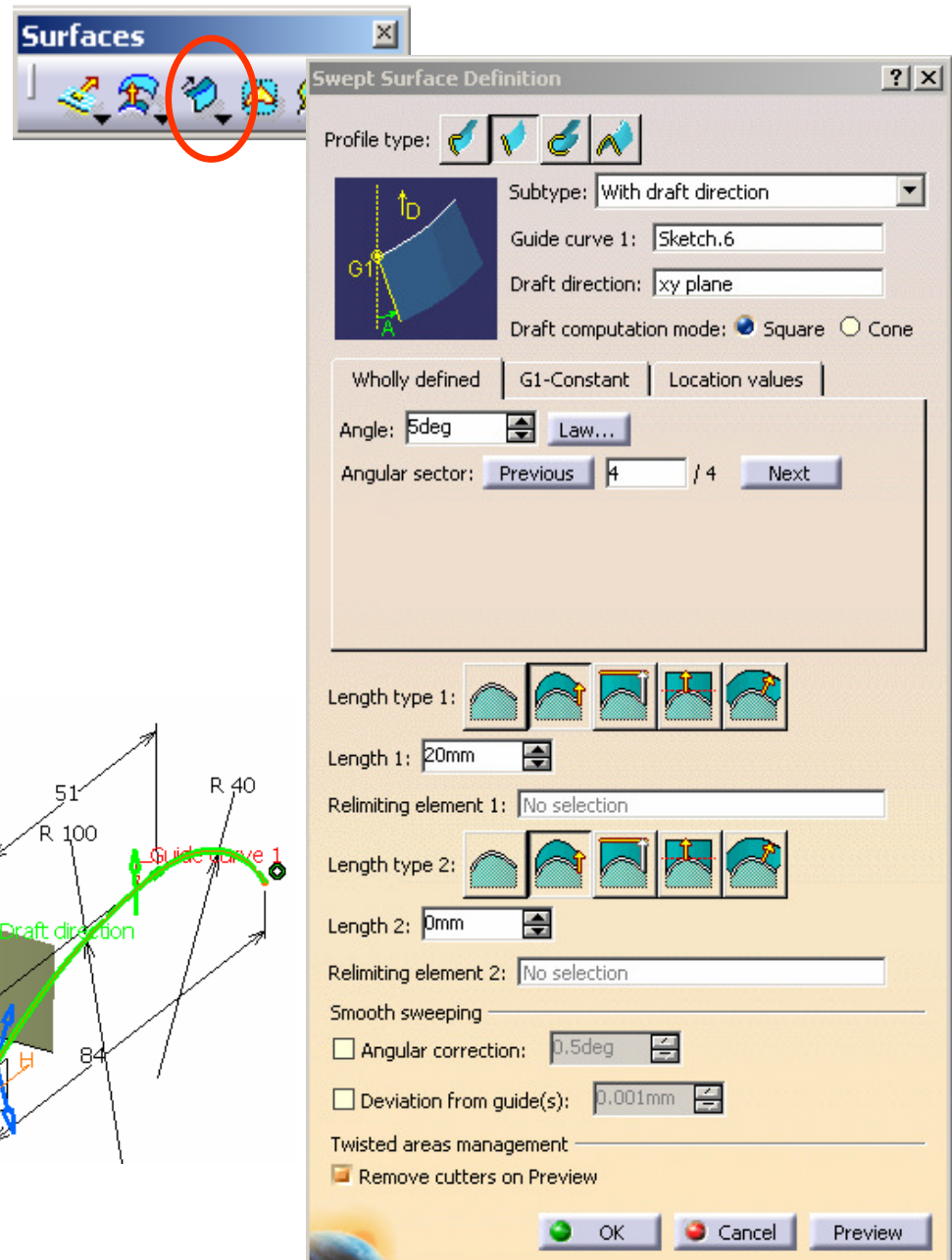
Tutorial 2A

To create surfaces from 3D curves:-

- Click "**Sweep**" icon;
- Select "**Line**" as Profile type;
- Select "**With Draft direction**" as Subtype;
- Select "**Combine.1**" as Guide curve1;
- Select **xy plane** as Draft Direction;
- Enter **5deg** as Angle;
- Enter **20mm** as Length1;
- There will be four arrows on the curve, 3 blue & 1 orange (highlighted);
- Select the arrow at **1st quarter** (+x,+y);
- Click ok to complete.

Surfaces

Swept Surface Definition

Profile type:

Subtype: With draft direction

Guide curve 1: Combine.1

Draft direction: xy plane

Draft computation mode: ● Square ○ Cone

| Wholly defined | G1-Constant | Location values |

Angle: 5deg Law...

Angular sector: Previous 1 / 4 Next

Length type 1:

Length 1: 20mm

Relimiting element 1: No selection

Length type 2:

Length 2: 0mm

Relimiting element 2: No selection

Smooth sweeping
☐ Angular correction: 0.5deg
☐ Deviation from guide(s): 0.001mm

Twisted areas management
☐ Remove cutters on Preview

● OK ● Cancel Preview

Top View
Side View
Front View
Plane.1
Plane.2
Plane.3
Geometrical Set.2
Plane.4
Plane.5
Sketch.4
Sketch.5
Combine.1
Plane.6
Sketch.6
Sweep.1

Guide curve 1

Draft direction

Draft direction

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Tutorial 2A

- Click "**Sweep**" icon again;
- Select "**Line**" as Profile type;
- Select "**With Draft direction**" as Subtype;
- Select "**Sketch.6**" as Guide curve1
- Select **xy plane** as Draft Direction
- Enter **5deg** as Angle
- Enter **20mm** as Length1
- There will be four arrows on the curve;
  3 blue & 1 orange (highlighted)
- Select the arrow at **4th quarter** (-x,+y)
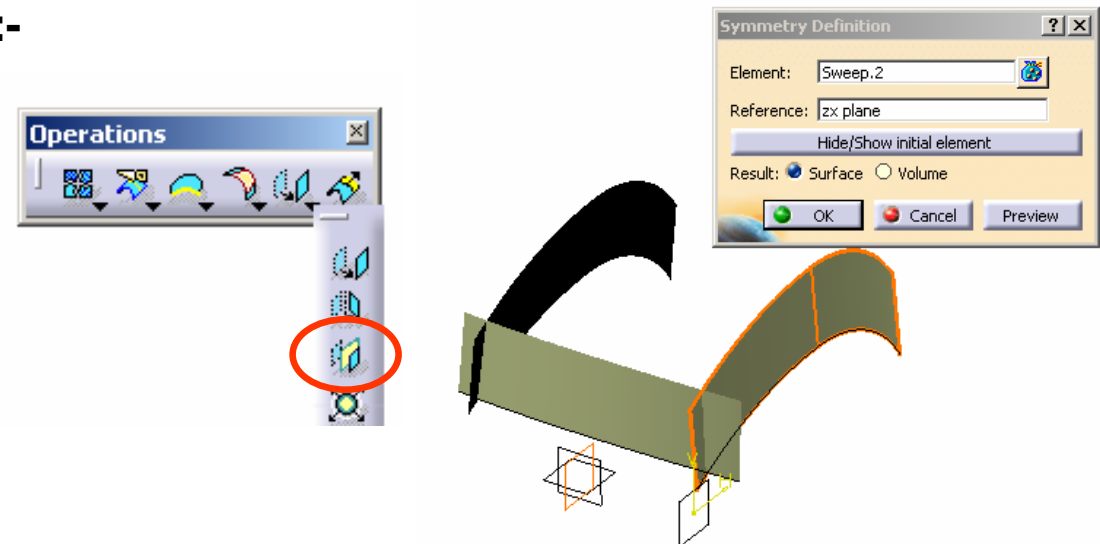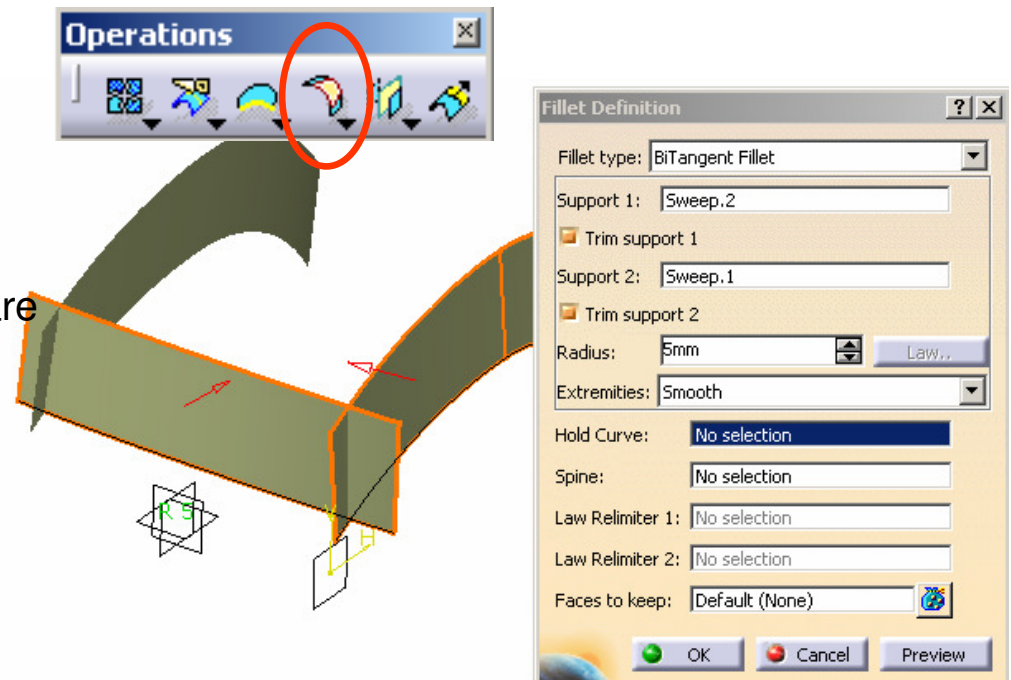- Click ok to complete

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Tutorial 2A

To duplicate a surface by mirroring:-

- Click "**Symmetry**" icon;
- Select "Sweep.2" as Element;
- Select "zx plane" as Reference;
- Click ok to complete.

To add a fillet between two surfaces:-

- Click "**Shape Fillet**" icon;
- Select surfaces "Sweep.1" & "Sweep.2";
- Click the red arrows on the surfaces if they are not pointing inwards;
- Enter **5mm** as Radius;
- Click ok to complete.

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Tutorial 2A

- Click "**Shape Fillet**" icon;
- Select surfaces "Fillet.1" & "Symmetry.1";
- Click the red arrows on the surfaces if they are not pointing inwards;
- Enter **5mm** as Radius;
- Click ok to complete
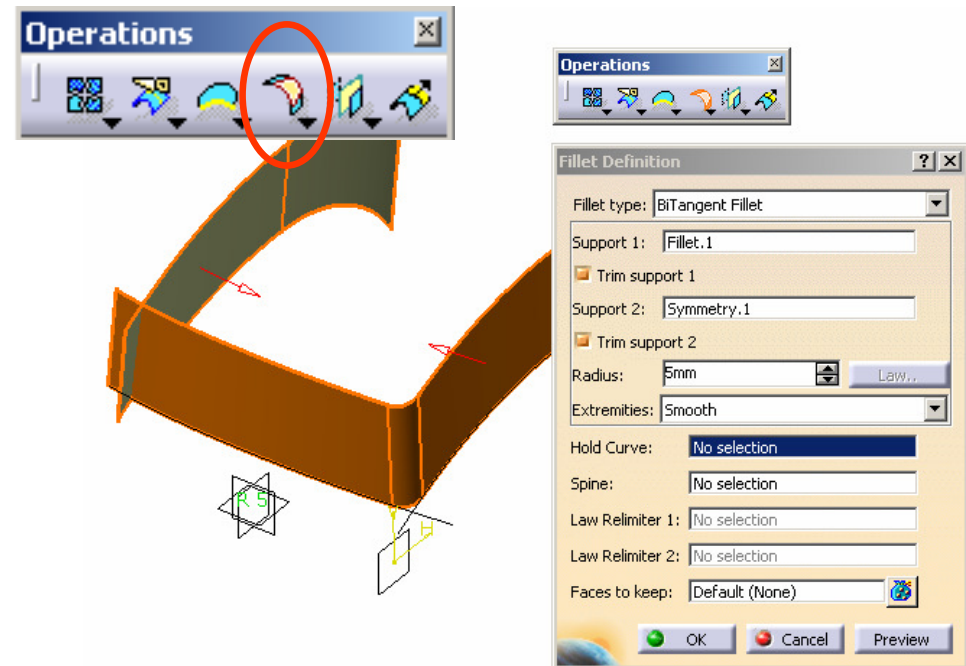
To duplicate a 3D curve by translation:-

- Click "**Translate**" icon;
- Select "**Combine.1**" as Element;
- Select "xy plane" as Direction;
- Enter **3.5mm** as Distance;
- Click ok to complete.

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By Dickson Sham (ME dept, HKPU)
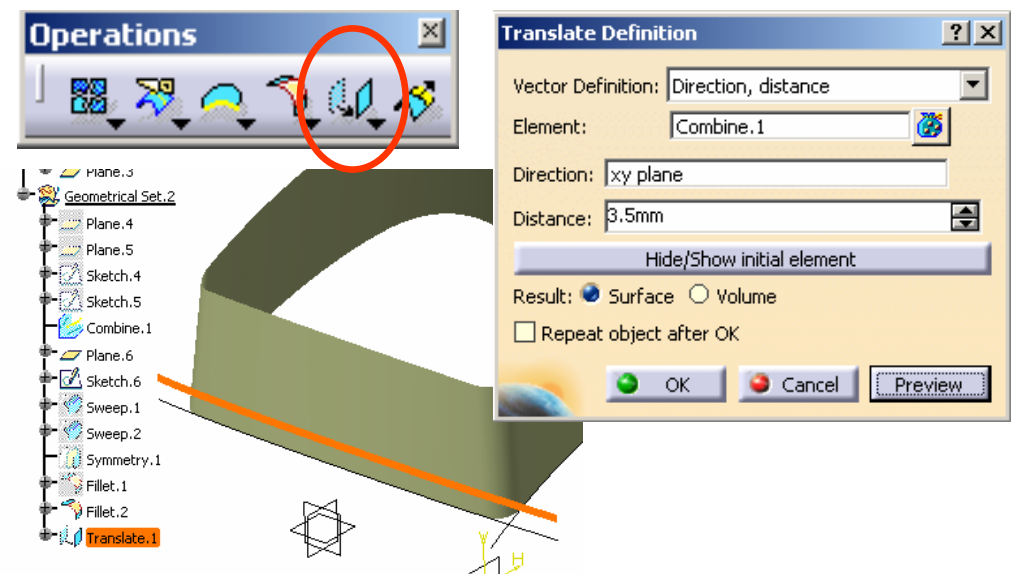
Tutorial 2A

Hide "Combine.1", "Plane.6", Sketch.6" & "Fillet.2"

- Click "Hide/Show" icon and select them on tree

To create a sketch mating with an external 3D curve:-

- Click "**Sketch**" icon;
- Select "zx plane";
- Draw two arcs as shown;
- Reminded that two arcs must be tangent to each other; one end of the small arc is touching the x-axis; one end of the bigger arc is just near y-axis

A- 23
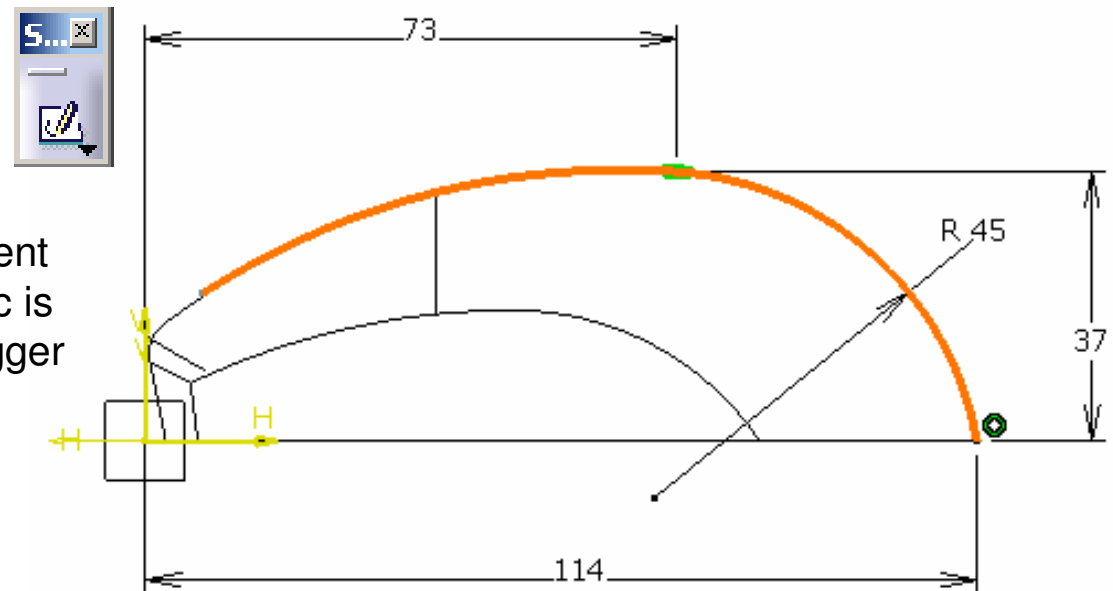
Tutorial 2A

- Click "**intersect 3D elements**" icon;
- Select "Translate.1" on tree or Direct-click the curve;
- Select the intersection point and click "**Construction/Standard element**" icon (The point shape will be changed from a cross to a point);
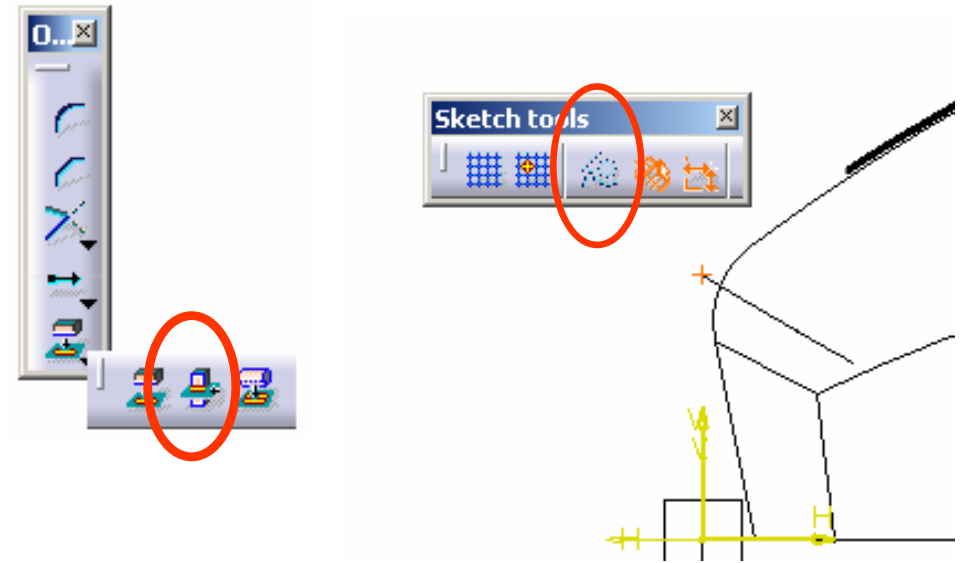
- **Add a coincidence constraint** between the endpoint of the bigger arc and the intersection point.

- Click ok to complete
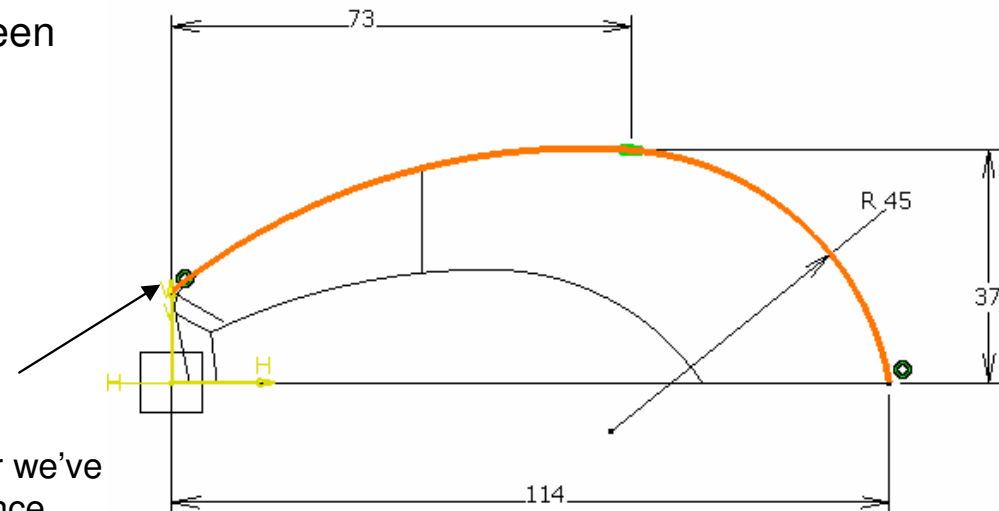
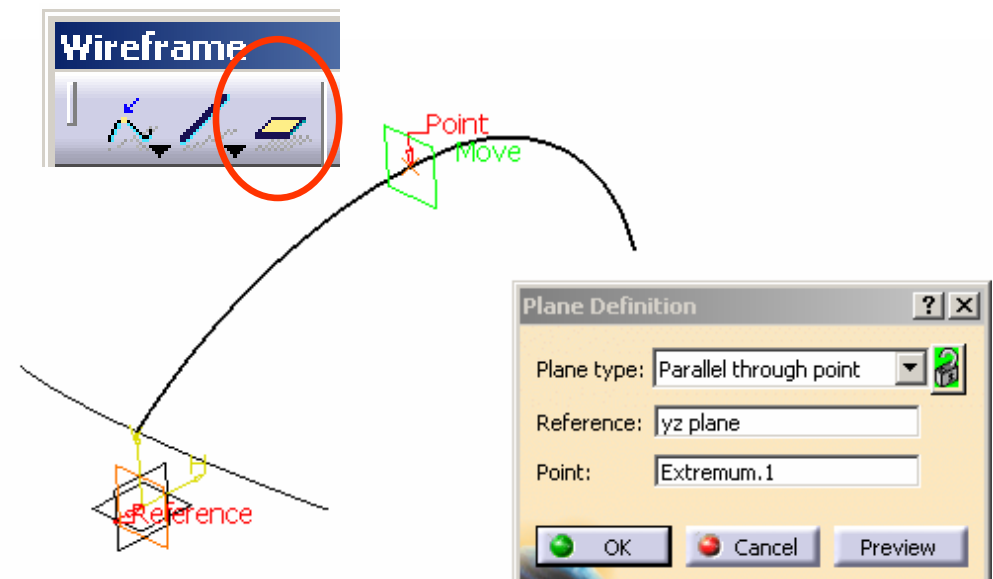The arc touches "Translate.1" after we've added a coincidence constraint

A- 24

Tutorial 2A

To create the highest point on a curve:-

- Click "**Extremum**" icon;
- Select "Sketch.7" as element;
- Select xy plane as direction;
- Click ok to complete.

Sketch.7

Direction

Make a point at the peak

Wireframe

Extremum Definition

Element: Sketch.7

Direction: xy plane   ● Max ○ Min

Optional directions

Direction 2: No selection   ● Max ○ Min

Direction 3: No selection   ● Max ○ Min

OK   Cancel   Preview
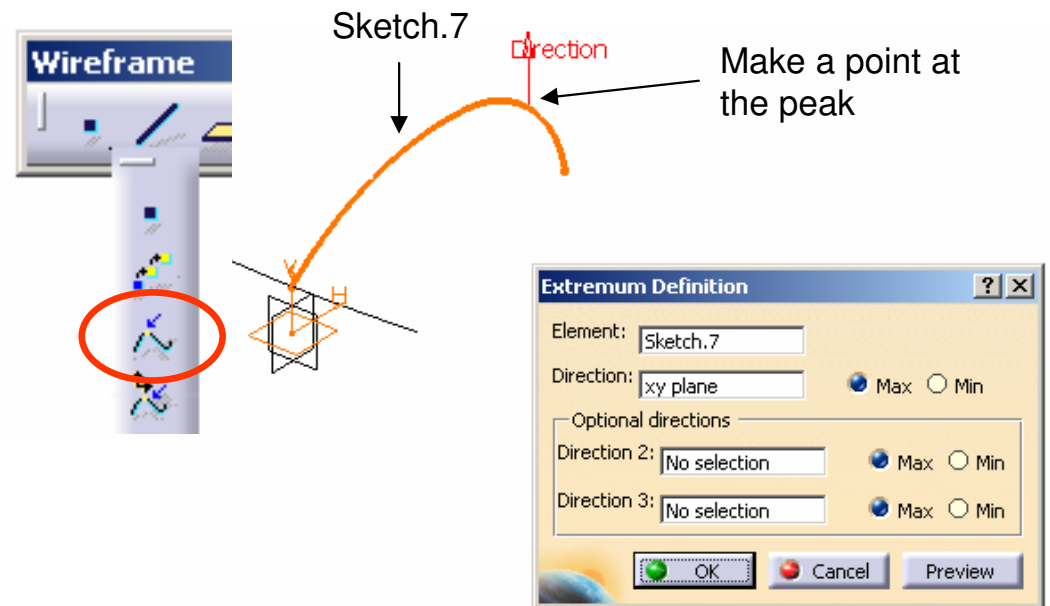
To create a plane at that maximum point:-

- Click "**Plane**" icon;
- Select "**yz plane**" and then "**Extremum.1**";
- "**Parallel through point**" will be automatically selected as plane type;
- Click ok to complete.

Wireframe

Point
Move

Reference

Plane Definition

Plane type: Parallel through point

Reference: yz plane

Point: Extremum.1

OK   Cancel   Preview

A- 25

Tutorial 2A

To create a sketch on the new plane:-

- Click "**Sketch**" icon;
- Select the new plane "**Plane.7**";
- Draw an **arc** as shown; add a symmetry constraint onto the endpoints;
- Then add a coincidence constraint between the arc and the maximum point "Extremum.1";

- Add a dimensional constraint **R38mm** onto the arc;

- Remark: the endpoints should be a little bit out of the background image

- Click ok to complete.

Add a coincidence constraint between the arc and "extremum.1"

symmetry

R 38

By Dickson Sham (ME dept, HKPU)
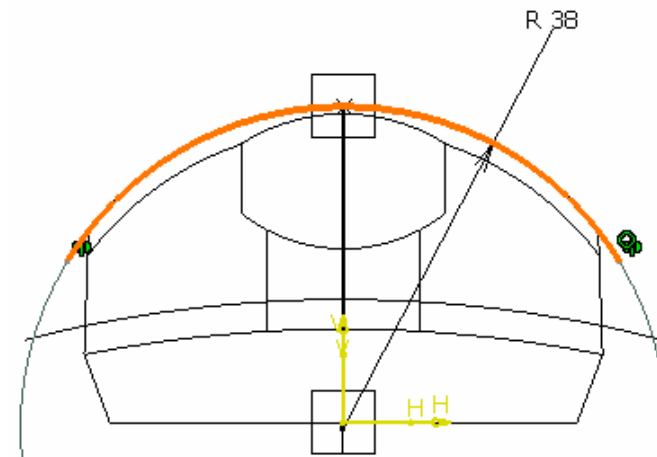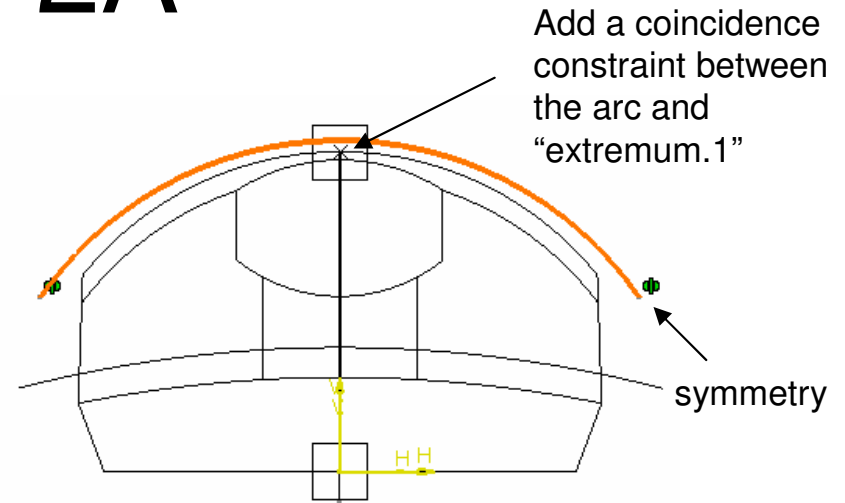
Tutorial 2A

To create a multi-sections surface:-

- Click "**Multi-sections surface**" icon;

- Select "Sketch.8" & "Translate.1" as **Section** (The red arrows should be pointing to the same direction; if not, click on the arrow to change)

- Select "Sketch.7" as **Guide**

- Click ok to complete
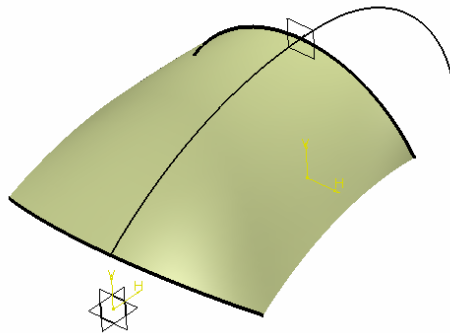
A- 27

Tutorial 2A

To Trim surfaces and form a joined surface:-

- **Show** "Fillet.2" on the tree;

- Click "**Trim**" icon

- Select surfaces "Fillet.2" & 'Multi-sections Surface.1"

- Click the option "Other side/next element" & "Other side/previous element" to obtain the result as shown on the right.
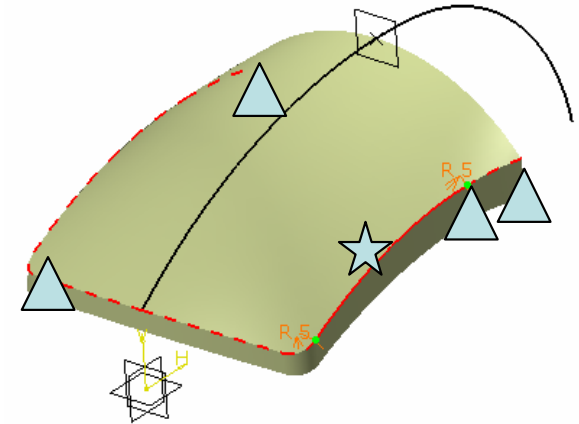
- Click ok to complete

Operations

Multi-sections Surface.1

Fillet.2

Trim Definition

Mode: Standard

Trimmed elements

Multi-sections Surface.1
Fillet.2

| Add after | Add before |
| Remove | Replace |
| Other side / next element |
| Other side / previous element |

Support: Default (None)

Elements to remove: Default (None)

Elements to keep: Default (None)

☐ Result simplification

☐ Intersection computation

☑ Automatic extrapolation

OK   Cancel   Preview

A- 28
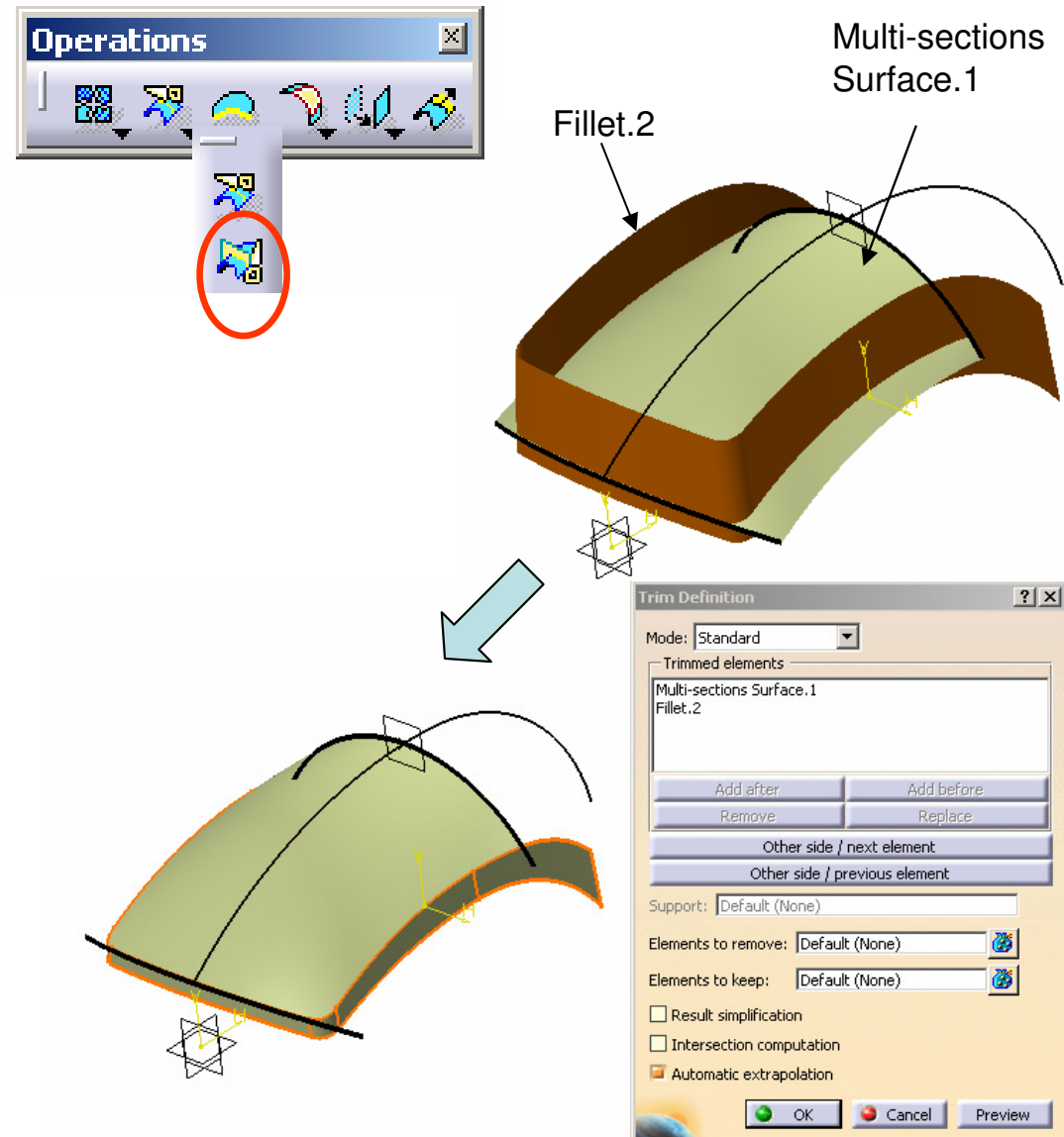
Tutorial 2A

To remove a portion from a surface:-

- Click "**Split**" icon;
- Select surface "Trim.1" as **element to cut**;
- Select "Plane.7" as **cutting element**;
- (It may be necessary to click the option "Other side" to obtain the result as shown on the right.)
- Click ok to complete
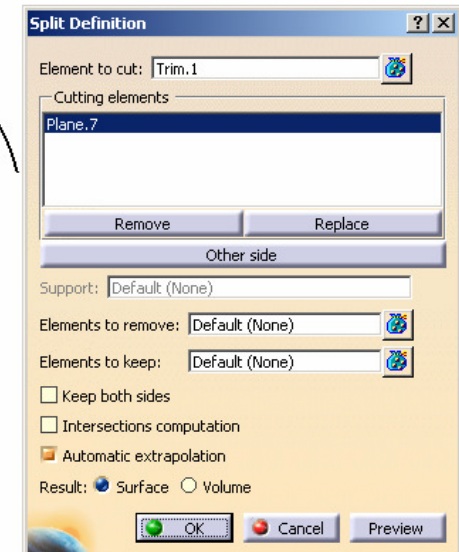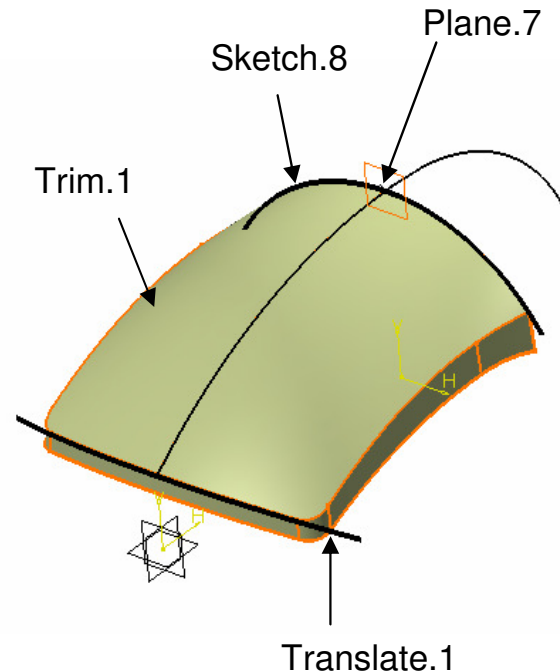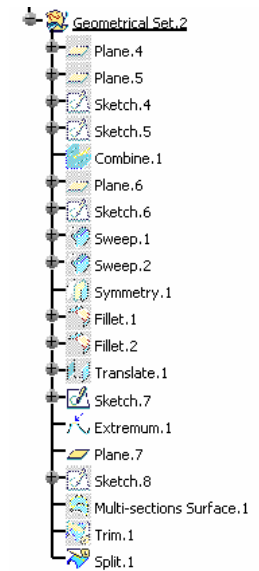
Hide "Sketch.8" & "Translate.1"

- Click "Hide/Show" icon and then select them on tree

Sketch.8

Plane.7

Trim.1

Translate.1

Tutorial 2A

To add a Variable Fillet along the edge:-

- Click "**Variable Fillet**" icon;

- Select the edge ☆ on the surface; (Then other portions along the edge will be selected automatically)

- Click the box "Points" on the command menu 💥

- Click the endpoints of the edges of the surface to add more control points; △

- Double-click the value on a control point to change. (R3mm front, R10mm back)

- Click ok to complete

Operations

R10

R10

R3

R3

Variable Radius Fillet Definition

Support:	Split.1
Extremities:	Smooth
Radius:	5mm
Edge(s) to fillet:	Split.1\Edge.1
Propagation:	Tangency
☐ Trim ribbons	
Points:	2 elements
Variation:	Cubic
☑ Trim support	

More>>

OK   Cancel   Preview

A- 30
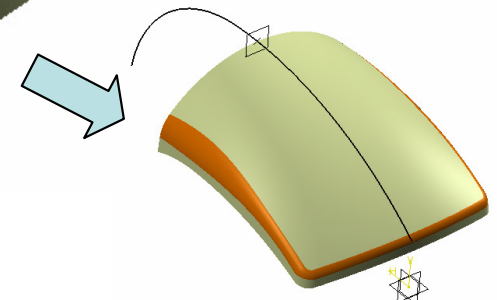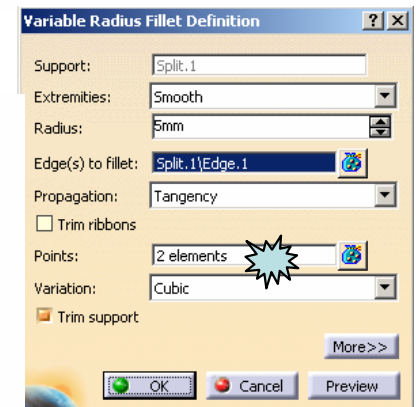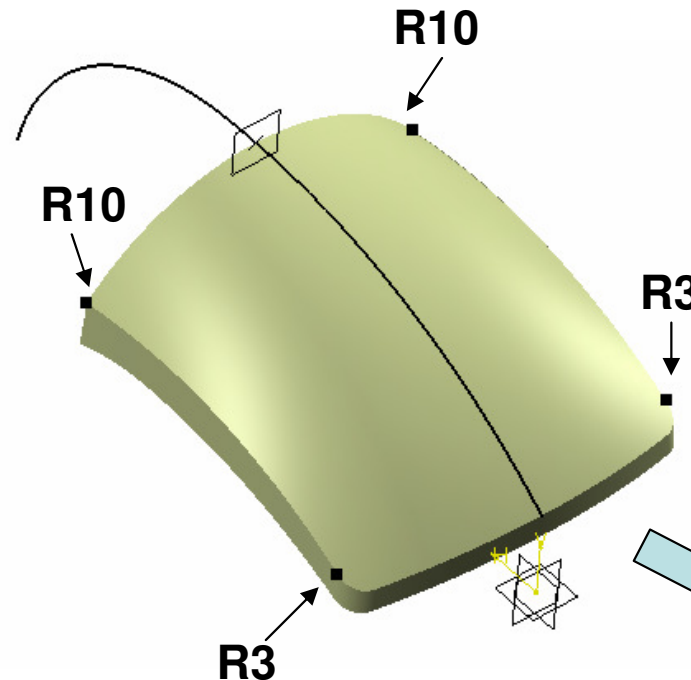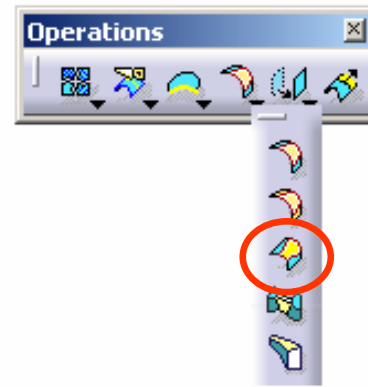
Tutorial 2A

Show "Sketch.6"

- Click "Hide/Show" icon and then select "sketch.6" on tree

To create two straight lines:-

- Click "**Line**" icon;
- Select the endpoint ☆ of Sketch.6;
- Select **yz** plane;
- Enter 20mm as End Length;
- Reverse Direction if needed;
- Click ok to complete

- Click "**Line**" icon again;
- Select the endpoint △ of Sketch.7;
- Select **zx** plane;
- Enter 20mm as End Length;
- Reverse Direction if needed;
- Click ok to complete
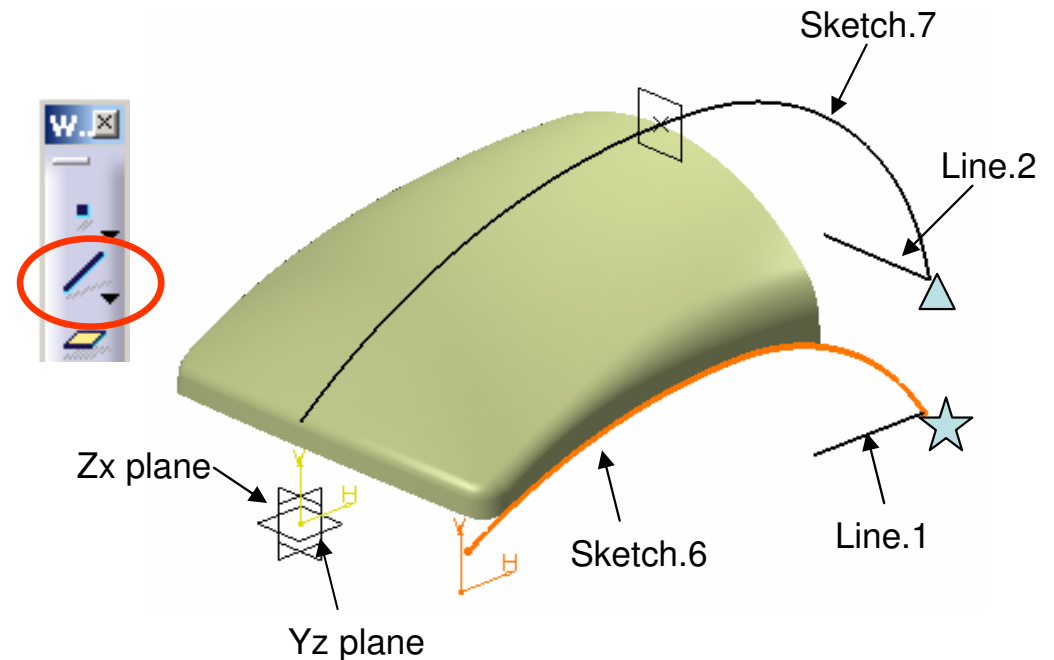
(Remark: "**Point-Direction**" will be automatically selected as the line type)

Tutorial 2A

To create a connecting curve linking two separate lines:-

- Click "**Connect Curve**" icon;

- (First Curve)
    - Select the endpoint of Sketch.7 as point;
    - Select "Line.2" as curve;
    - Select "Tangency" as Continuity;
    - Reverse Direction if needed;

- (Second Curve)
    - Select the endpoint of Sketch.6 as point;
    - Select "Line.1" as curve;
    - Select "Tangency" as Continuity;
    - Reverse Direction if needed;
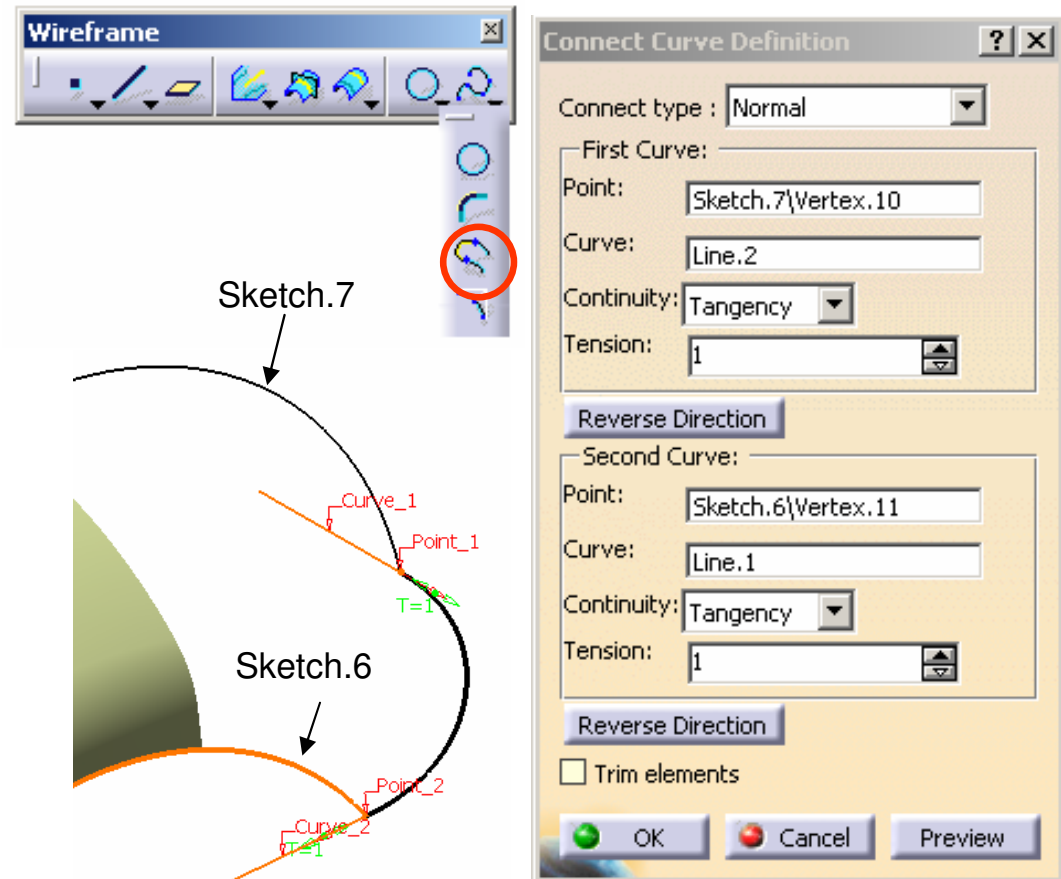
Then we are going to modify its curvature such that it is in the similar shape as the reference imported drawing.

A- 32

Tutorial 2A

- Click "**Top View**" icon and now we can see the reference top view;

Quick view

- Enter 1.7 as tension of 1st Curve
- Enter 0.4 as tension of 2nd Curve

- Click ok to confirm

Connect Curve Definition

Connect type : Normal

First Curve:
Point: Sketch.7\Vertex.10
Curve: Line.2
Continuity: Tangency
Tension: 1.7

Reverse Direction

Second Curve:
Point: Sketch.6\Vertex.11
Curve: Line.1
Continuity: Tangency
Tension: 0.4

Reverse Direction

☐ Trim elements

● OK   ● Cancel   Preview

A- 33
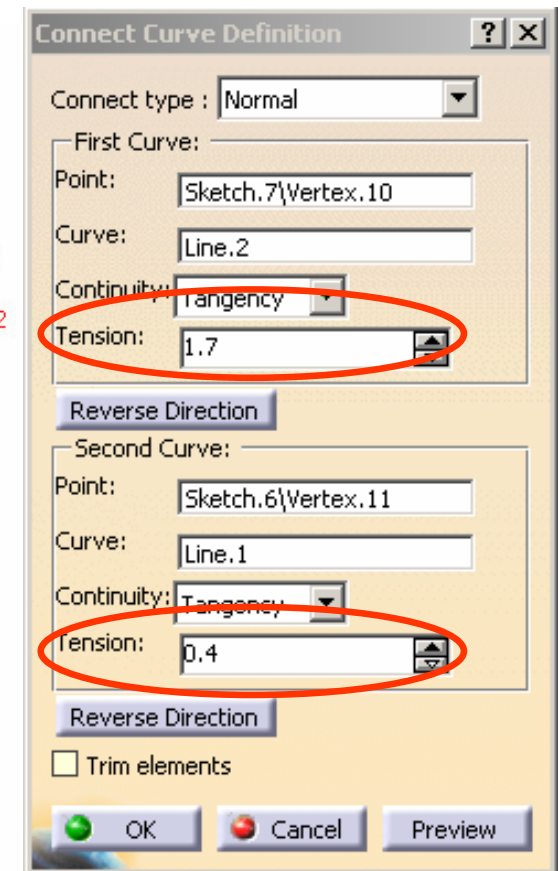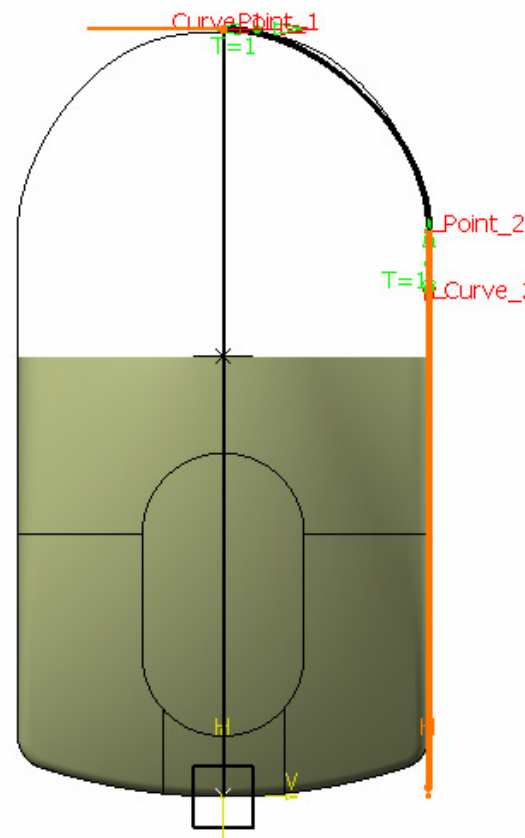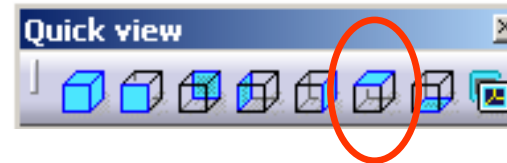
Tutorial 2A

To duplicate curves about a mirror plane:-

- Click "**Symmetry**" icon;
- Select "Sketch.6" as element;
- Select "zx plane" as reference;
- Click ok to complete.

- Similarly, Click "**Symmetry**" icon again;
- Select "Connect.1" as element;
- Select "zx plane" as reference;
- Click ok to complete.

Connect.1

Sketch.6

Zx plane

Fillet.2
Translate.1
Sketch.7
Extremum.1
Plane.7
Sketch.8
Multi-sections Surface.1
Trim.1
Split.1
EdgeFillet.2
Line.1
Line.2
Connect.1
Symmetry.2
Symmetry.3

A- 34
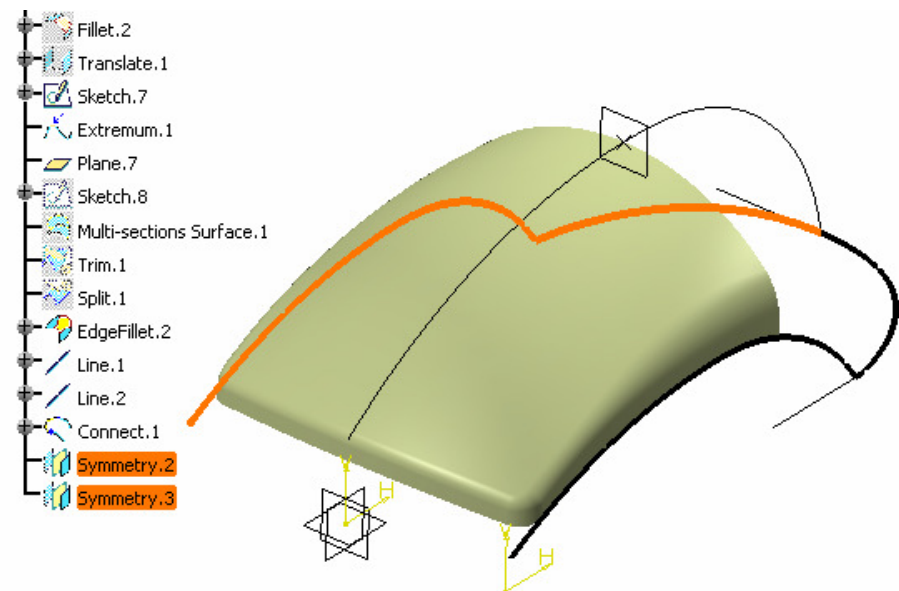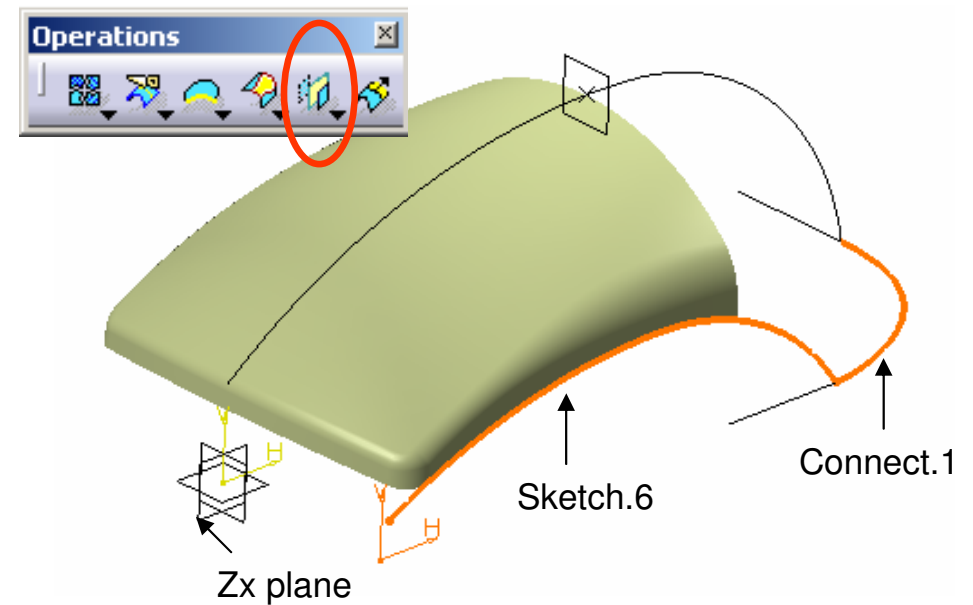
Tutorial 2A

To obtain a boundary of a combined surface:-

- Click "**boundary**" icon;
- Select "**Tangency continuity**" as propagation type;
- Select the edge as shown and the rest nearby will be selected automatically;
- Click ok to complete.

Remark: A combined curve (Boundary.1) will then be created with 5 segment curves embedded.

Click here

A- 35
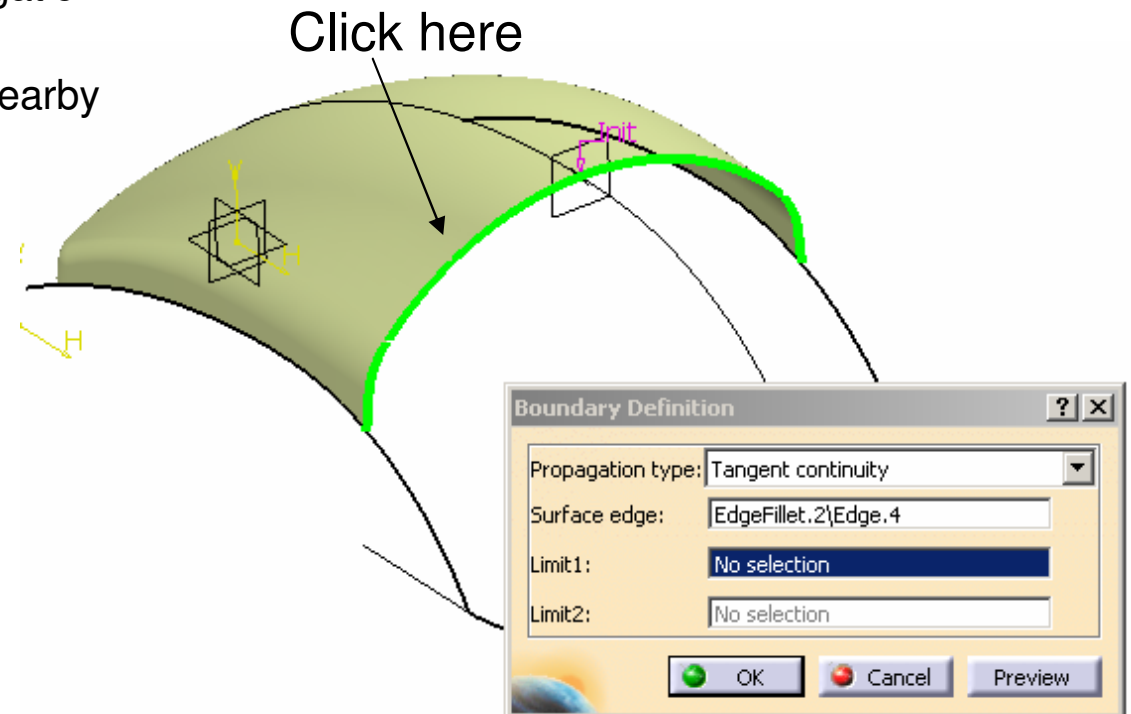
Tutorial 2A

To join two curves into one:-

- Click "**Join**" icon;
- Select "**Connect.1**" & " **Symmetry.3**";
- Click ok to complete.

Operations

Boundary.1

Symmetry.2

Symmetry.3

Sketch.6

Connect.1

Join Definition

Elements To Join

Symmetry.3
Connect.1

Add Mode   Remove Mode

Parameters | Federation | Sub-Elements To Remove

☐ Check tangency   ☑ Check connexity   ☑ Check manifold
☐ Simplify the result
☐ Ignore erroneous elements

Merging distance   0.001mm

☐ Angular Threshold   0.5deg

● OK   ● Cancel   Preview

A- 36
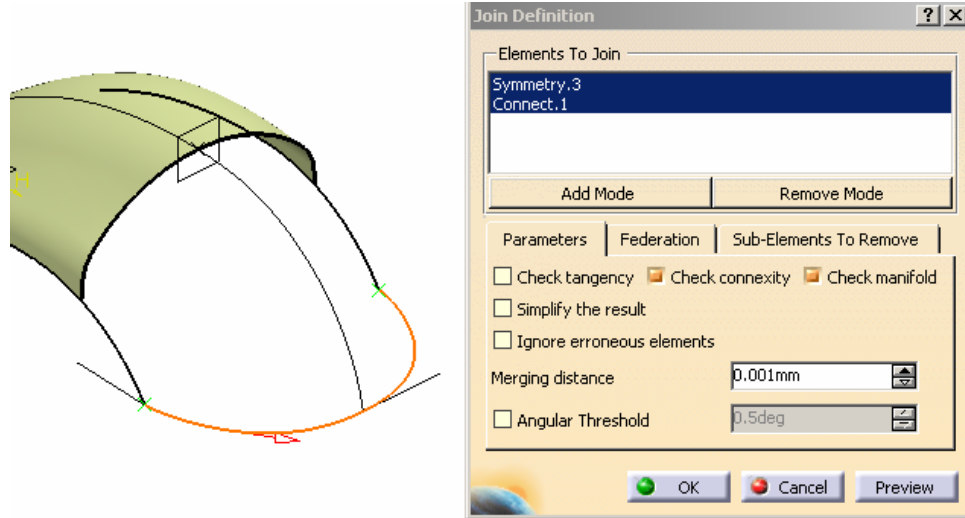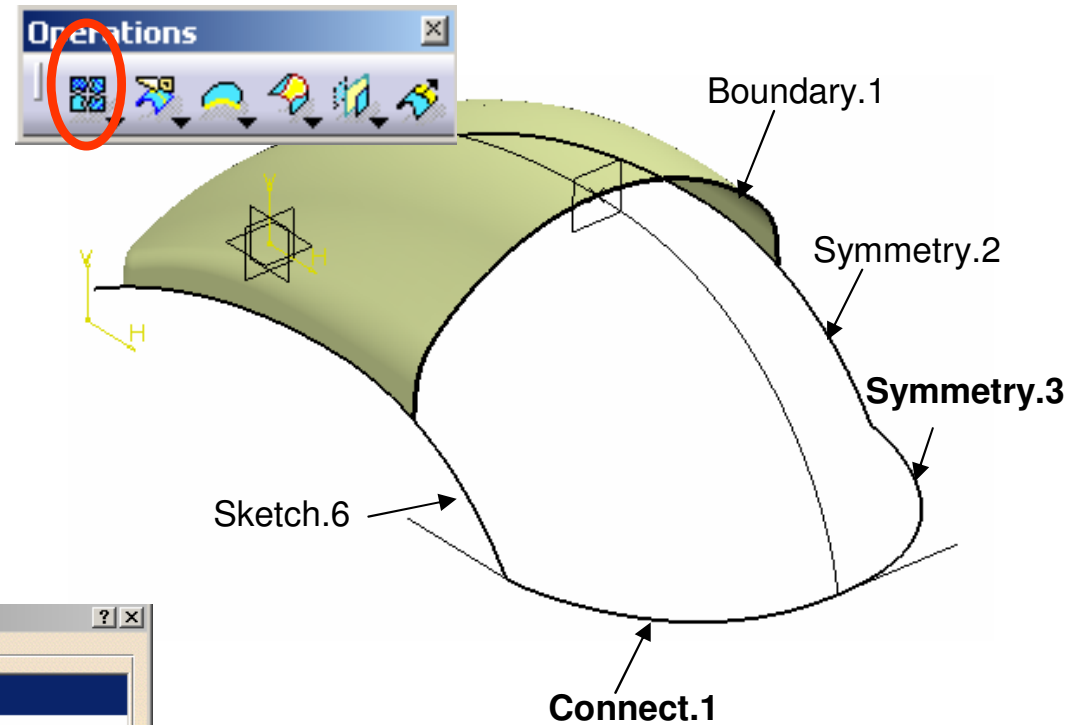
Tutorial 2A

To create a multi-sections surface:-

- Click "**Multi-sections surface**" icon;

- **Select "Boundary.1" , then select the surface** (ensuring that the new surface will be tangent to the existing surface);
- **Select "Join.1";**
- (Remark: Red arrows on the two sections should be pointing to the same direction. If not, change either one.)

- Click "**…**" **under Guides** in the menu;
- **Select "Sketch.6", "Sketch.7" & "Symmetry.2";**

- Click ok to complete.

Click here before defining guides

A- 37

Tutorial 2A

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

To improve the smoothness of a multi-section surface:-

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

We are going to increase the distance between two sections

A- 38

Tutorial 2A

To improve the smoothness of a multi-section surface:-

- Click "**Plane**" icon;
- Select "Offset from plane" as type;
- Select "Plane.7";
- Enter **10mm** as **Offset** value;
- Click ok to complete.
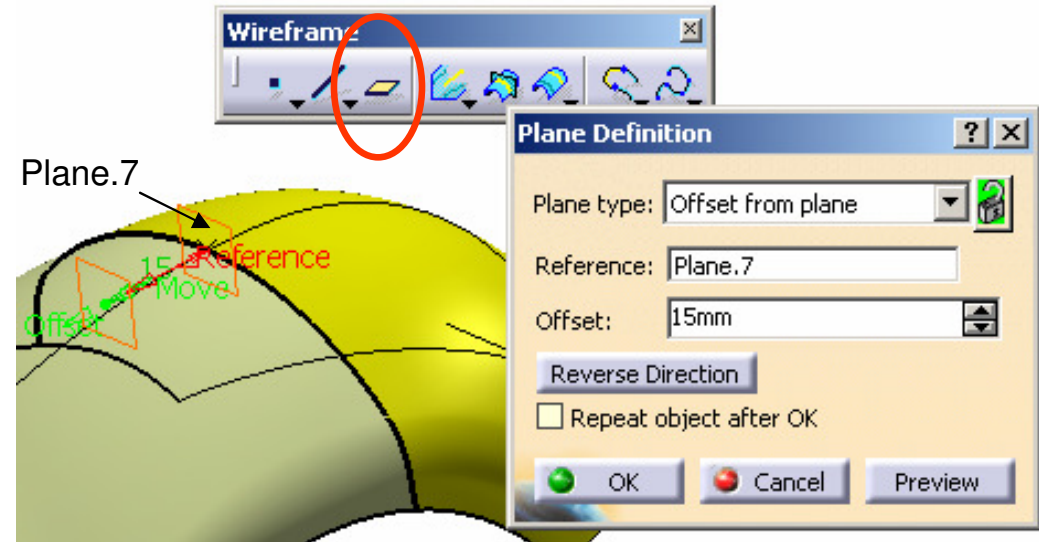
- Click **"Split"** icon;
- Select the front surface "Edge Fillet.2" as element to cut;
- Select the new plane "Plane.8" as cutting element;
- Click ok to complete.

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By Dickson Sham (ME dept, HKPU)
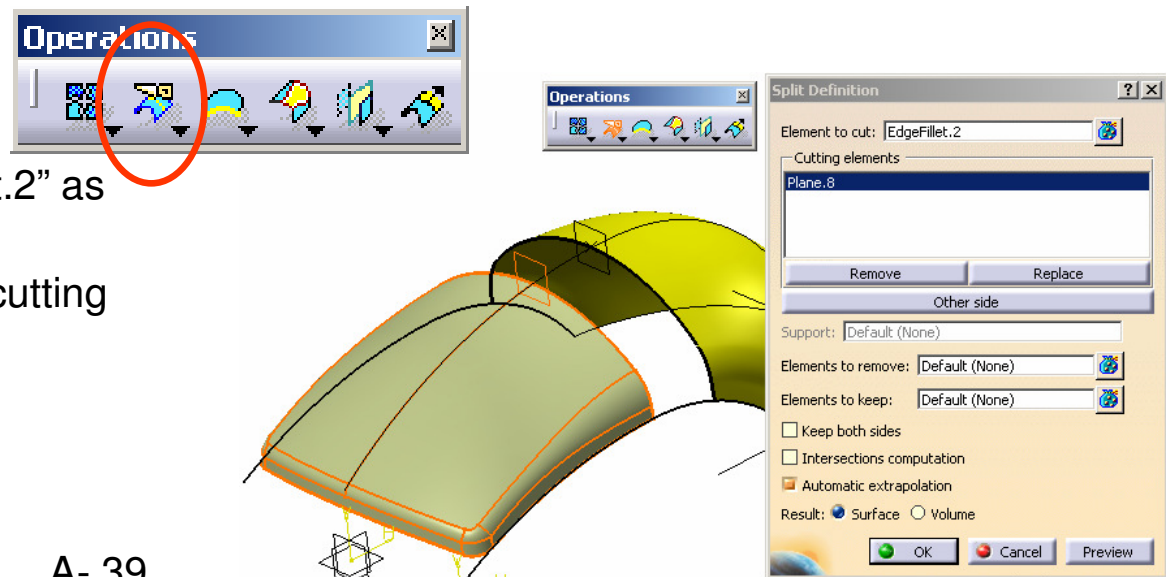
Tutorial 2A

- **Double-Click "Boundary.1"** on the tree to modify;
- Click the box ☆ "Surface Edge" once in the pop-up menu;
- Click the edge △ of the front surface;
- Click ok to confirm.

Remark: The back surface "Multi-sections surface.2" will be updated automatically.
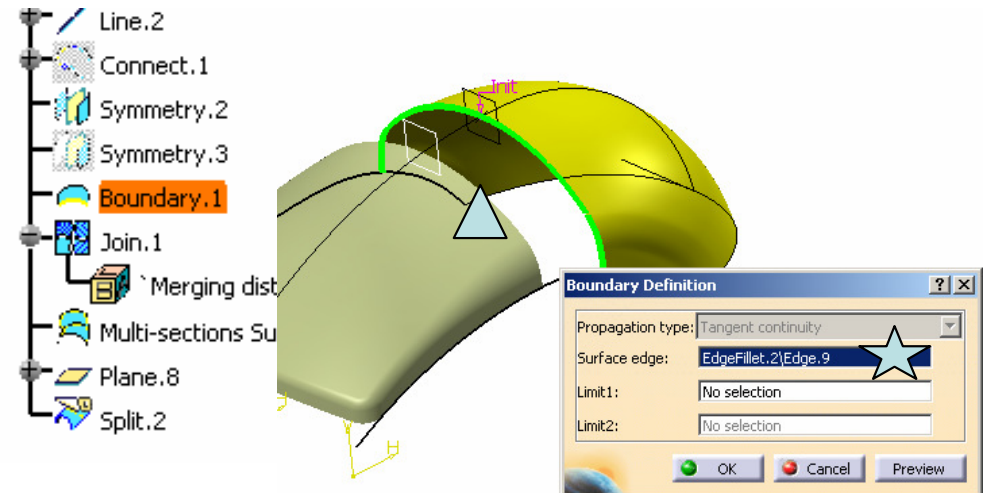
As you see, the back surface becomes smoother than before.

BEFORE
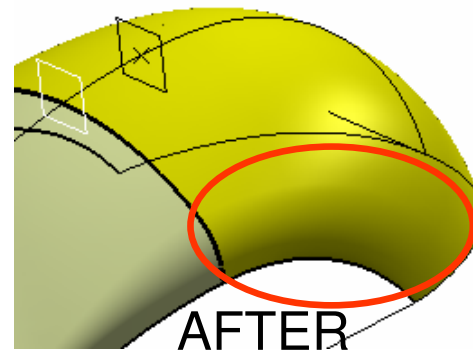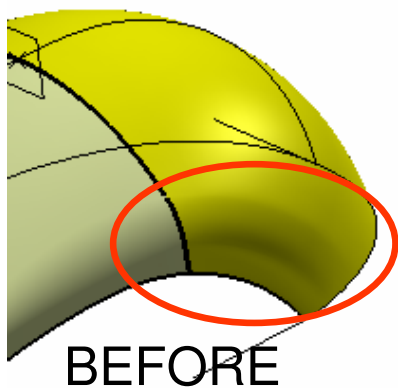
AFTER

A- 40
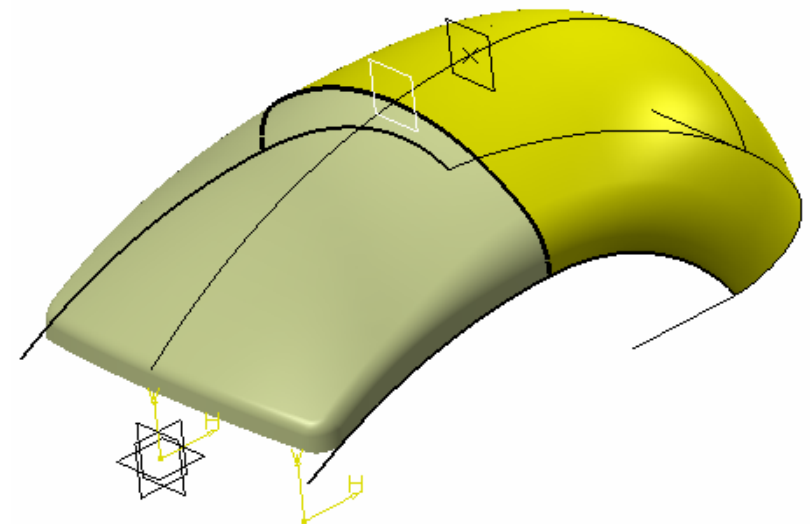
Tutorial 2A

To auto-sort the tree:-

After modifying the position of the front section, the back surface is updated but the tree is not in logical sequence. Although it is not a must and does not affect the final result, keeping the tree in a correct order can help us modify the model more easily in future.

e.g. "Split.2" should appear before "Boundary.1"

- Right-Click "Geometrical Set.2"
- Select "Geometrical Set.2 Object/ **Autosort**"
- Then the tree will be re-ordered in a logical way.

A- 41
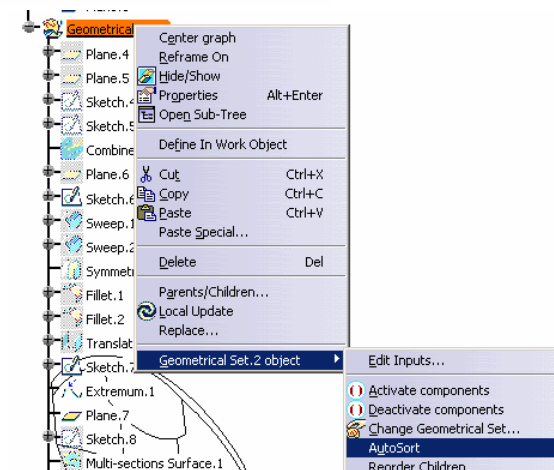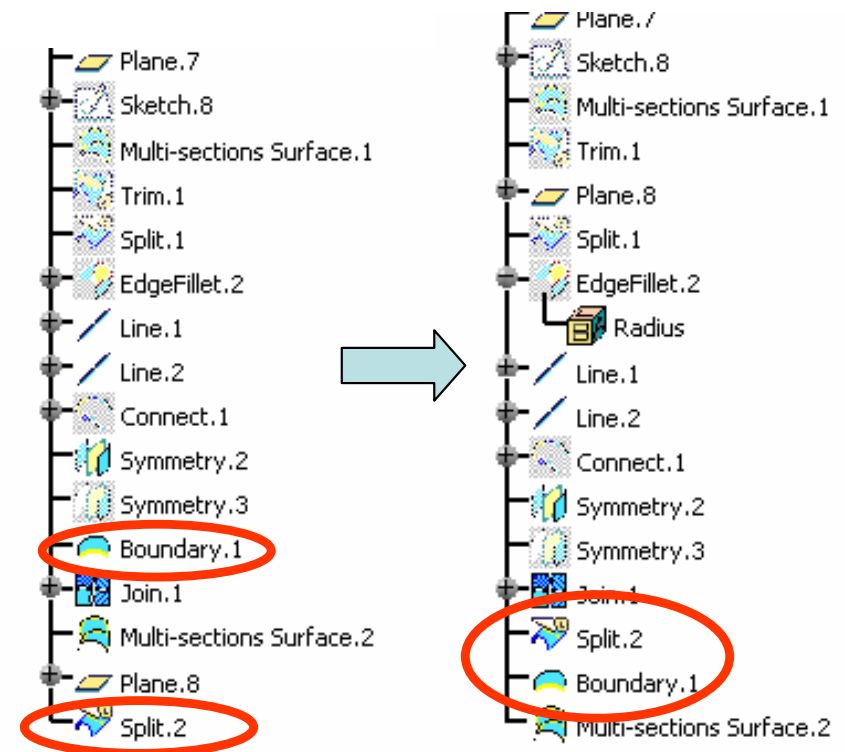
Tutorial 2A

To join two surfaces into one:-

- Click "**Join**" icon;

- Select Surfaces "Multi-sections surface.2" & "Split.2"

- Click ok to complete

Multi-sections surface.2

Split.2
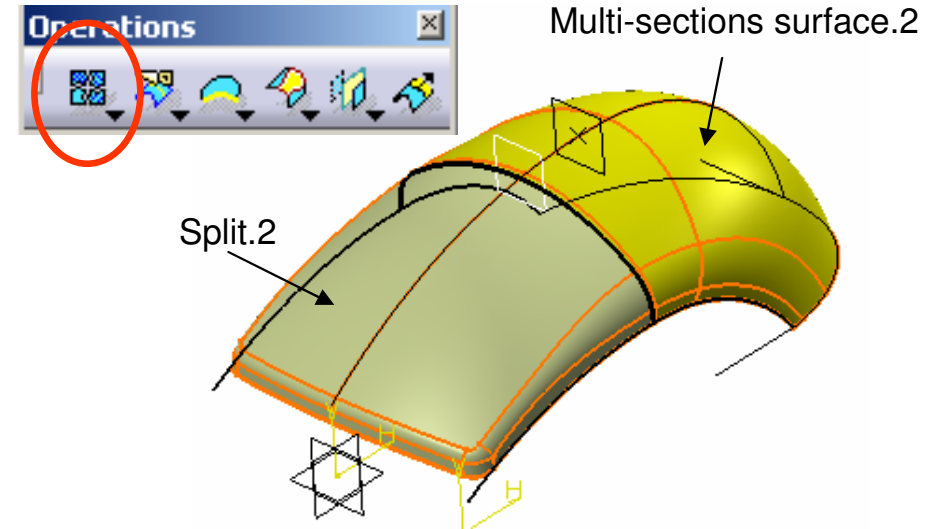
To save the file:-

- Select **File/Save** on the menu bar;

- Select your project folder;

- Enter "**Mouse_mastermodel_a.CATPART**" as the file name.

END of Tutorial 2A

By Dickson Sham (ME dept, HKPU)

Tutorial 2B

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

- *Reopen the file* "Mouse_mastermodel_a.CATPART" ;
- *Ensure that the current workbench is "Generative Shape Design";*
- Click **"Front View"** icon to check the front view;
- Click **"Right View"** icon to check the right view;
- Click **"Top View"** icon to check the top view;

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

Front

Right
Top
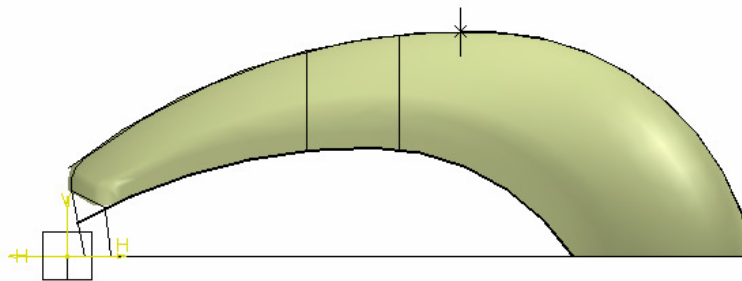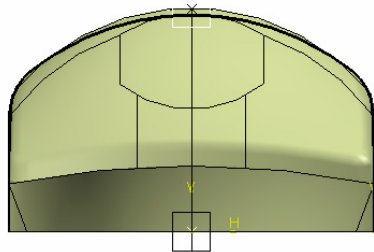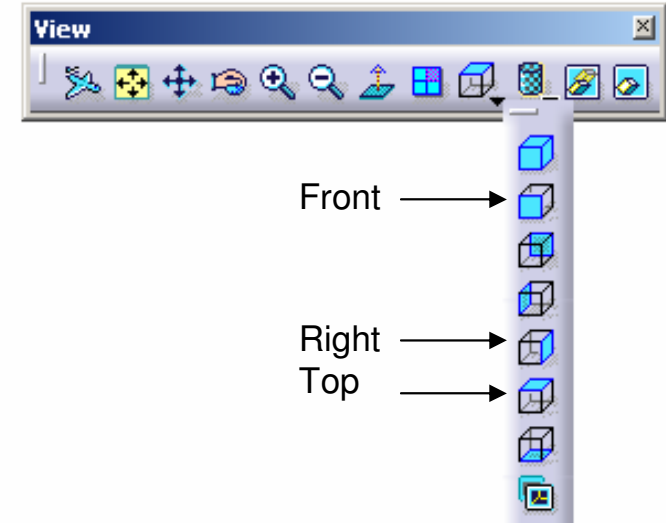
A- 43
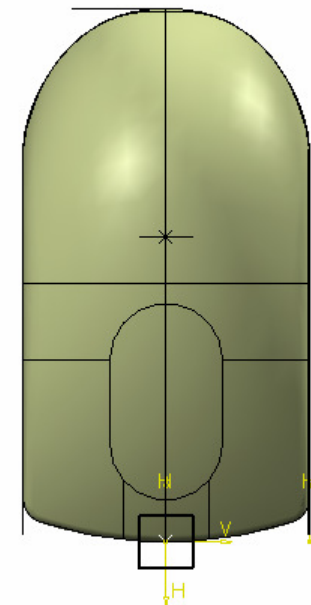
Tutorial 2B

To make a sketch:-

- Click "**Sketch**" icon;
- Select "Plane.4"
- Draw a profile as shown
- Click "Exit" icon to complete

Plane.4
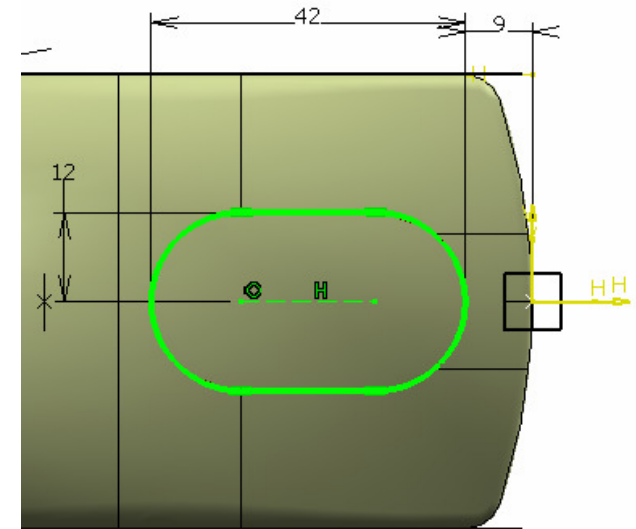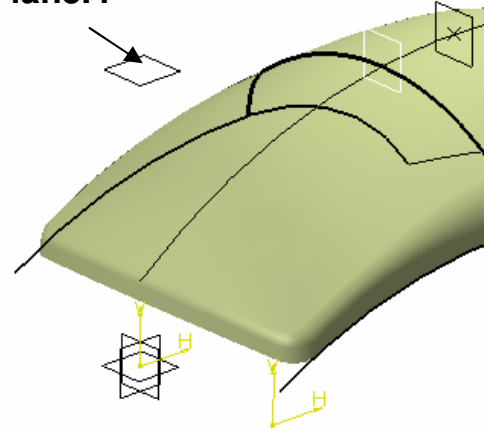
To project a curve onto a surface:-

- Click "**Projection**" icon;
- Select "Along a direction" as type
- Select "Sketch.9" as Projected
- Select "Join.2" as support
- Select "xy plane" as direction
- Click ok to complete

Wireframe

Sketch9

Xy plane

join2

A- 44
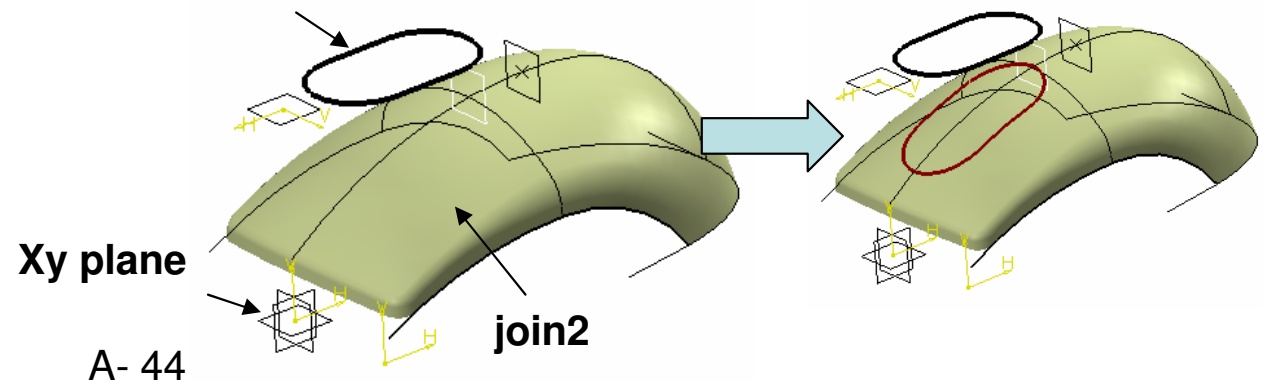
Tutorial 2B

To remove surface along the projected curve:-

- Click "split" icon;
- Select "Join2" as Element to cut
- Select "Project.1" as Cutting element
- Click ok to complete

Operations

Sketch.9

Project.1

Join.2
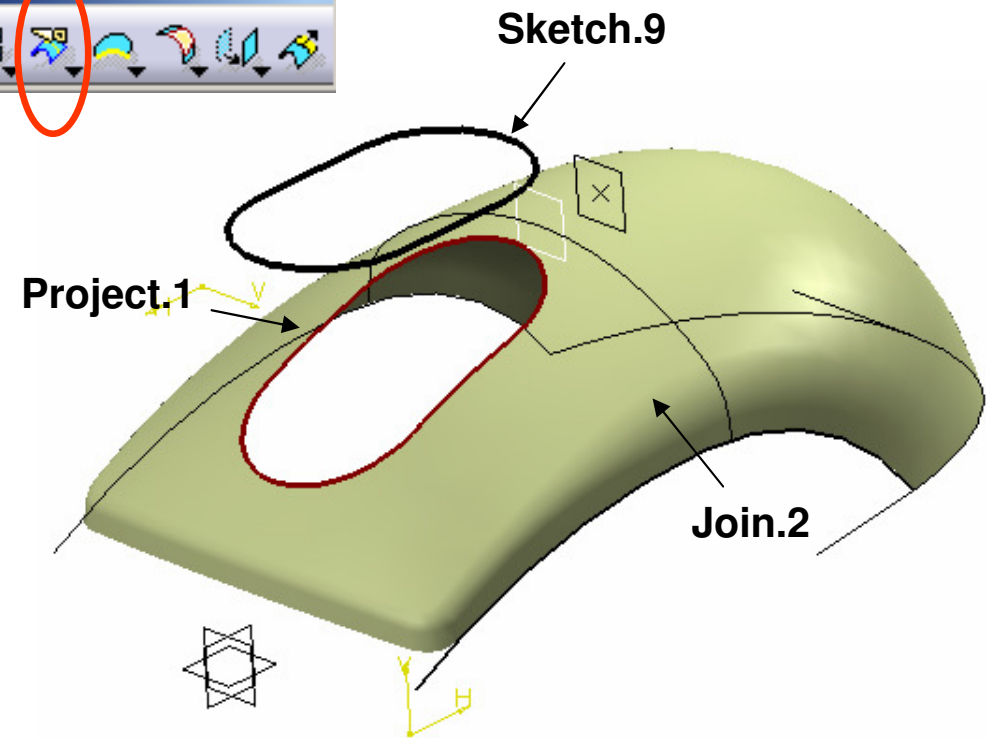
Then Hide "Sketch.9" & "Project.1"
(Do it yourself)

A- 45

Tutorial 2B

To create a swept surface:-

- Click "**sweep**" icon;
- Select "Circle" as Profile type;
- Select "Two guides and radius" as subtype;
- Click the two edges of the hole as guide curve 1 & 2.
- Enter 80mm as radius;
- Click "preview" icon;
- Click "Next "icon until the solution is the inner smaller arc;
- Click ok to complete.
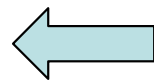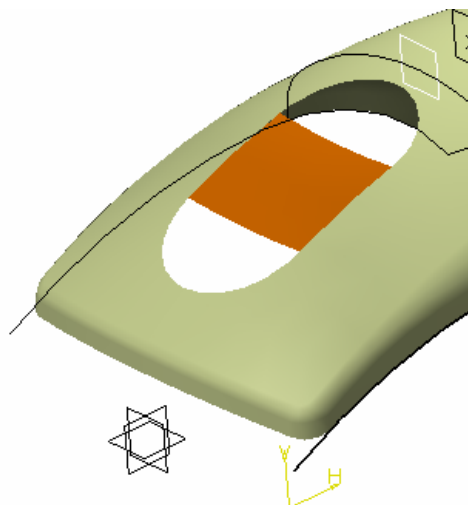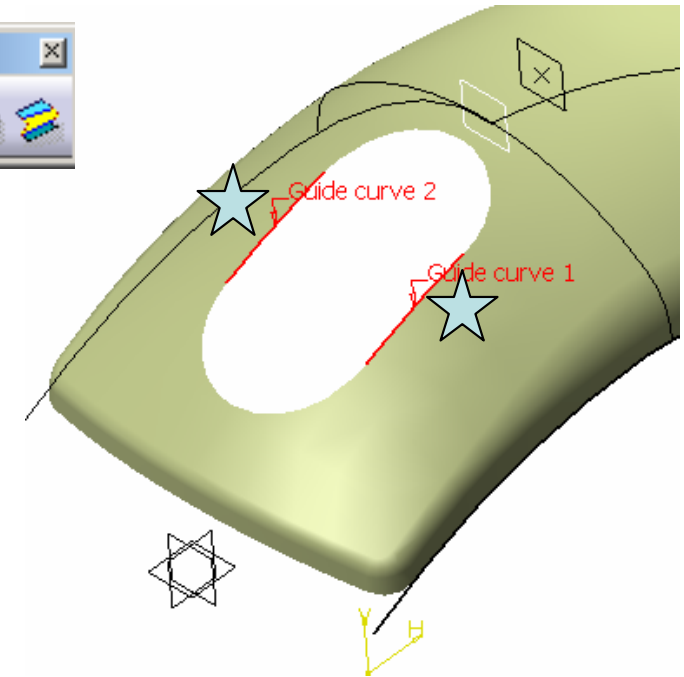
A- 46
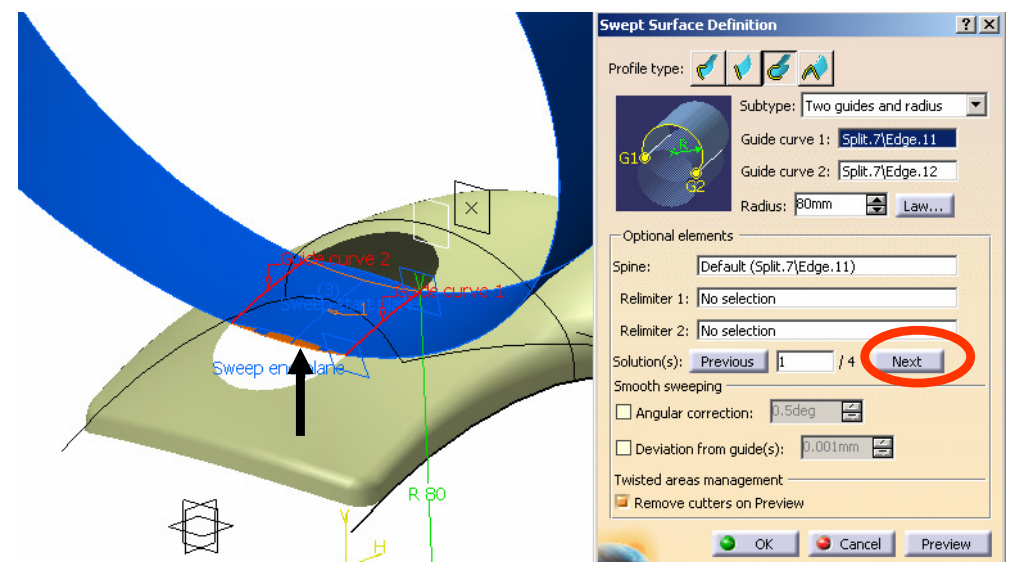
Tutorial 2B

To create a blend surface:-

- Click "**blend**" icon;
- Select the edge △ of Sweep.3 as First Curve;
- Select "Sweep.3" as First Support;
- Select the edge ☆ of Join.2 as Second Curve;
- Select "tangency" as first continuity;
- Click ok to complete

To create another blend surface:-

- Click "**blend**" icon again;
- Select the edge ✦ of Sweep.3 as First Curve;
- Select "Sweep.3" as First Support;
- Select the edge ◇ of Join.2 as Second Curve;
- Select "tangency" as first continuity;'
- Click ok to complete

Surfaces

Sweep.3

Join.2

A- 47
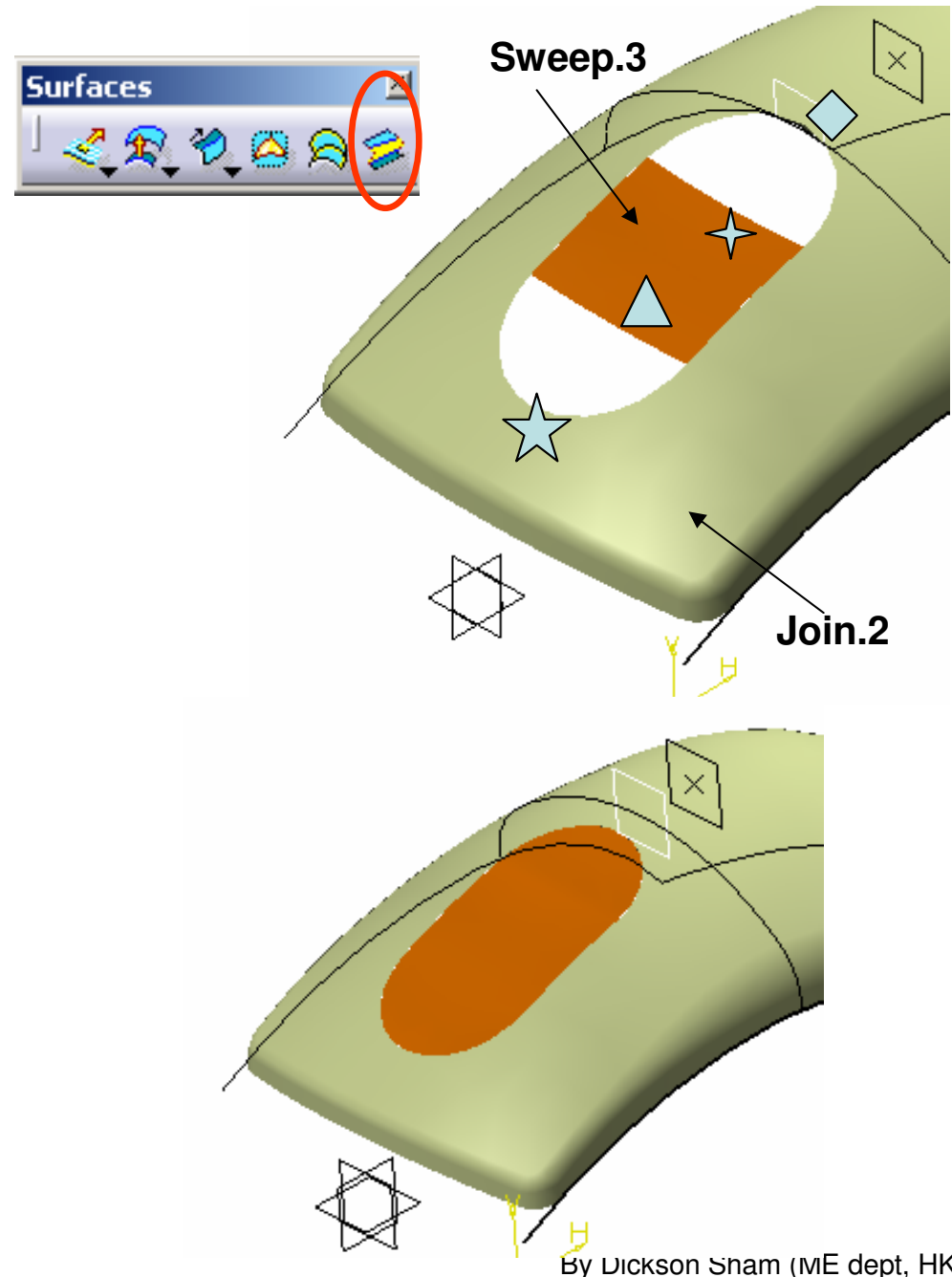
Tutorial 2B

To join surfaces as one:-

- Click "**Join**" icon;

- Select "Blend.1", "Blend.2", "Sweep.3" & "Split.3"

- Click ok to complete

Blend.2

Sweep.3

Blend.1

Split.3

A- 48
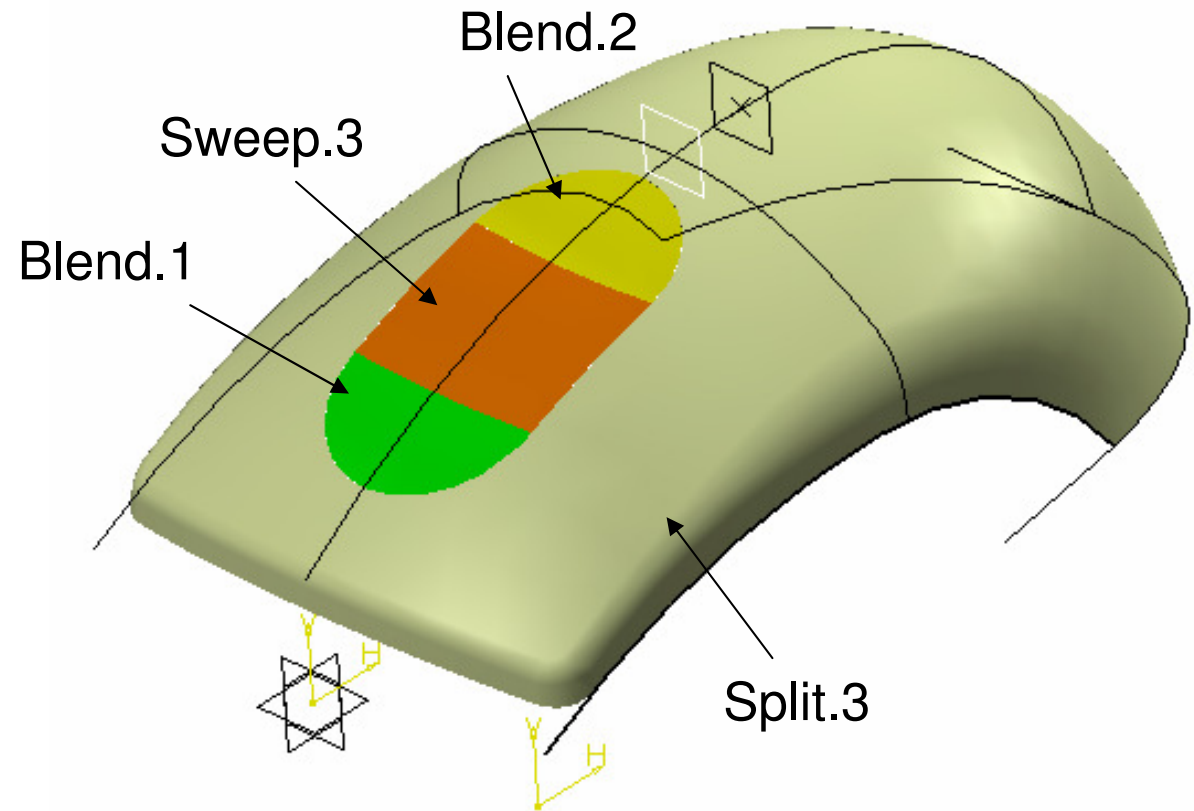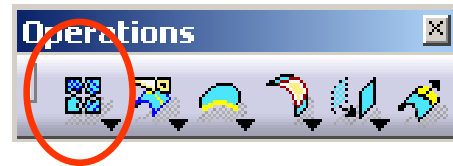
Tutorial 2B

To do the draft analysis on the surface:-

- Click " **Feature Draft Analysis**" icon;

- Select 'Quick Analysis Mode";

- Select "Color Scale" (Now a 3-color scale pops up: Green, Red & Blue);

- Double-click the upper value on the scale and modify it as 1deg

- Double-click the lower value on the scale and modify it as -1 deg

    (i.e. Green, draft > 1deg;

    Red, draft = 0deg;

    Blue, draft < -1 deg)

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

A- 49
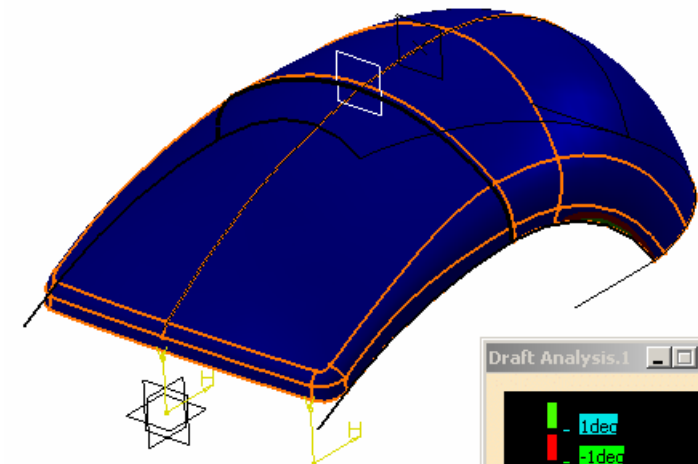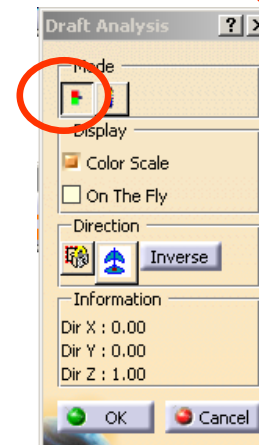
Tutorial 2B

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

- We find a portion ⭐ with Zero Draft (Red) along the parting line.

- Click **OK** to activate the draft analysis so that we can always get the draft distribution updated after every modification on the surface.

Draft Analysis

- Mode
- Display
  - ☑ Color Scale
  - ☐ On The Fly
- Direction
  - Inverse
- Information
  - Dir X : 0.00
  - Dir Y : 0.00
  - Dir Z : 1.00
- OK   Cancel
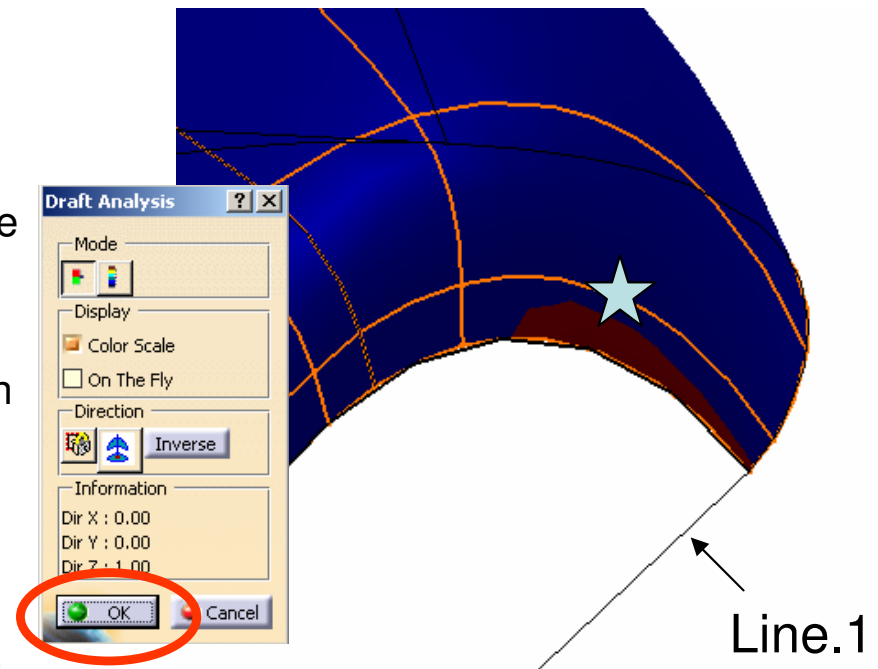
Line.1

To modify the curvature of the problem surface:-
- Right-click on **"Line.1"**
- Select "Line.1 object/**Isolate**"
- Drag the red dot of the compass onto the endpoint of **"Line.1"**

- Center graph
- Reframe On
- Hide/Show
- Properties   Alt+Enter
- Other Selection...
- Define In Work Object
- Cut   Ctrl+X
- Copy   Ctrl+C
- Paste   Ctrl+V
- Paste Special...
- Delete   Del
- Parents/Children...
- Local Update
- Replace...
- Line.1 object   ▶
  - Definition...
  - Deactivate
  - Isolate
  - Change Geometrical Set...
  - Reset Properties
  - Edit Parameters
  - Upgrade

Drag the compass onto the endpoint of Line1

A- 50
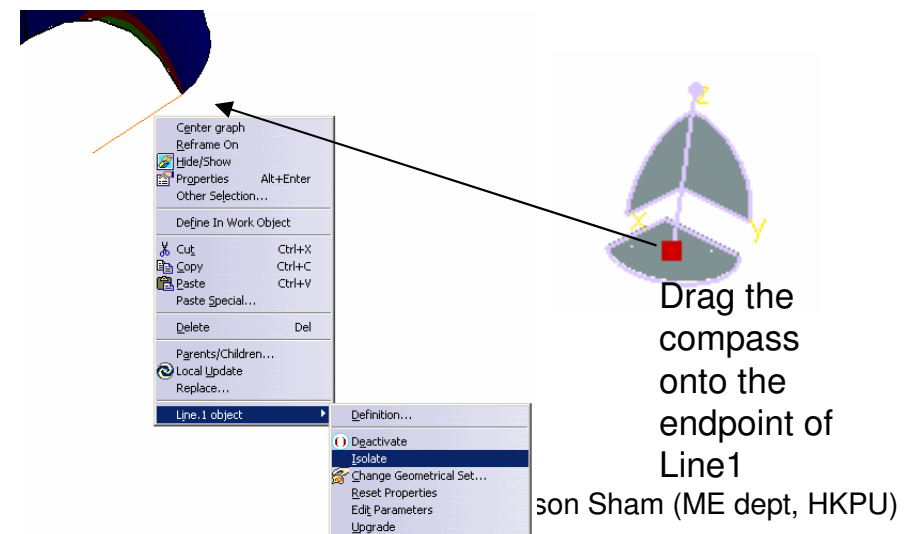
Tutorial 2B

Be sure that the W-axis of the compass should be
pointing upwards and the red-dot should be at the
endpoint of Line.1

- **Click Line.1 once**. The line will be orange and the
  compass will be green.
- Right-click the red-dot of the compass and select
  "**Edit**";
- Enter "10deg" as Rotation Increment;
- Click "positive rotation along W" icon;
- Click "Close"
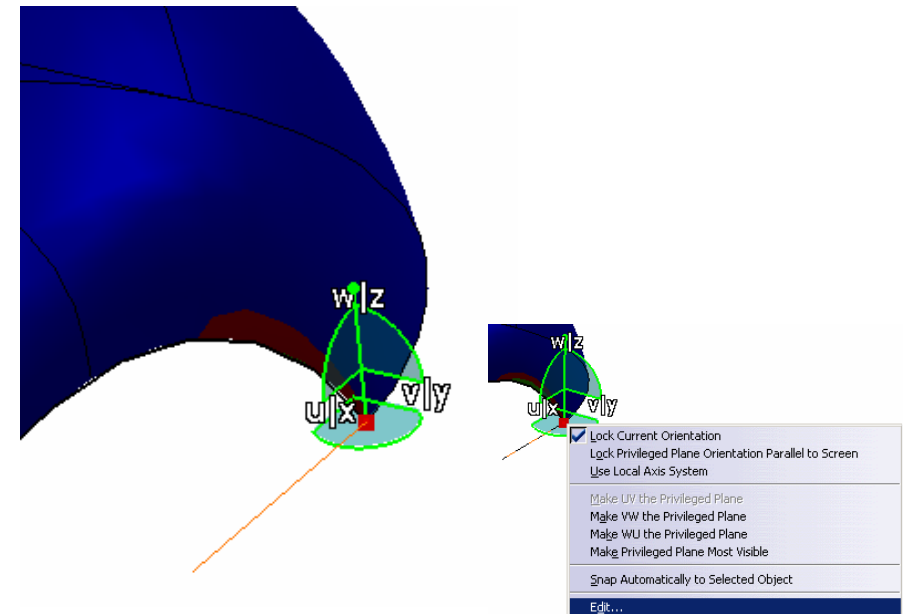
Now the portion with zero-draft ⭐ disappears

after we've modified the curvature of the problem surface.

A- 51
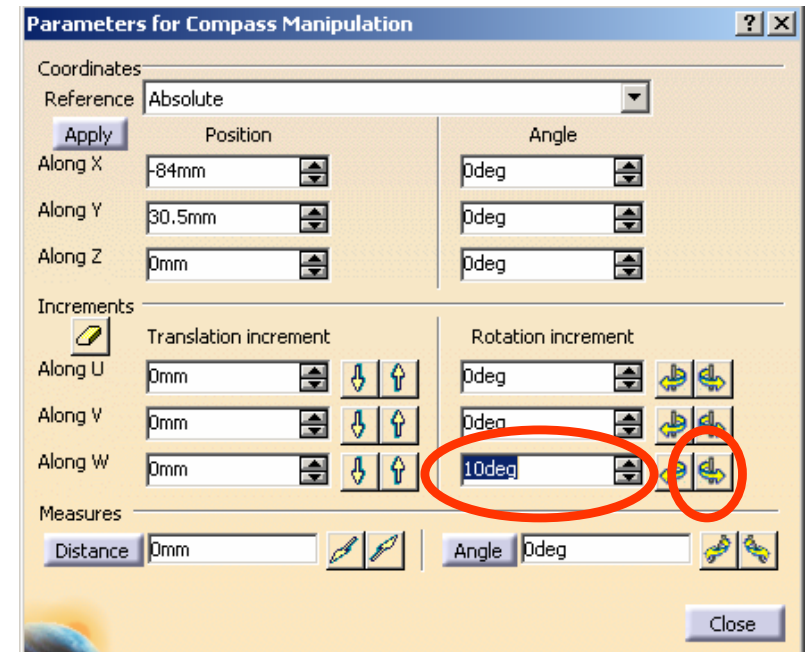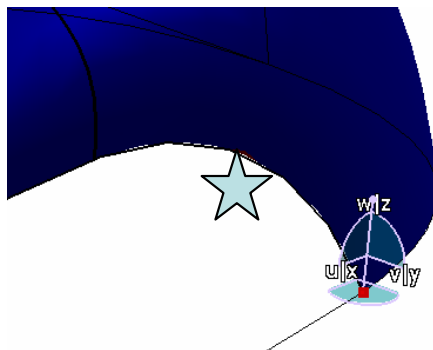
Tutorial 2B

- Delete "**Draft Analysis.1**" from the tree
- Drag the red-dot of the compass onto the global coordinate system at the lower right-hand corner and then release. (It will return to its original stage)

Now, the upper skin has been completed.
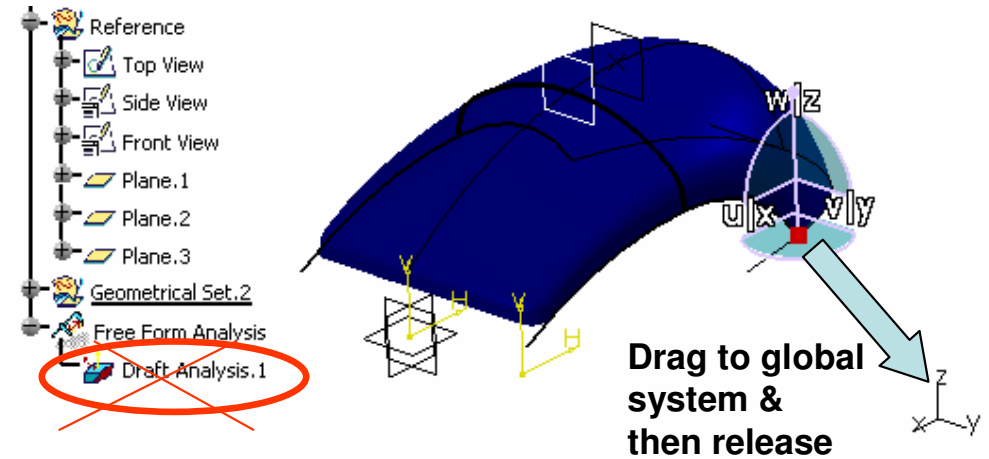
Hide all elements except "Join.3" ,

"Sketch.6" & Symmetry.2"

(Do it yourself)

Show "Combine.1"

(Do it yourself)

Drag to global system & then release

Combine.1

Sketch.6

Join.3

A- 52

By Dickson Sham (ME dept, HKPU)
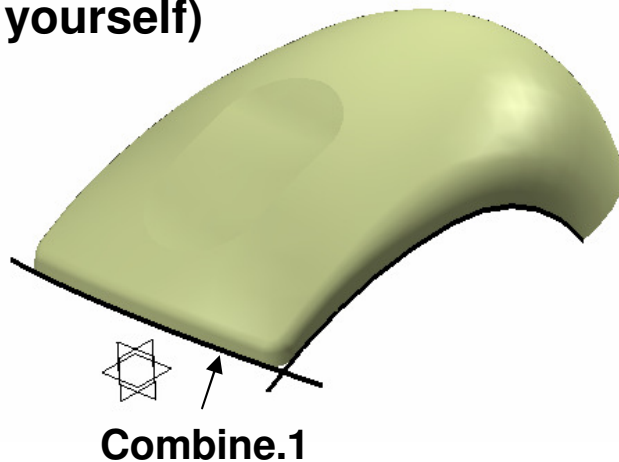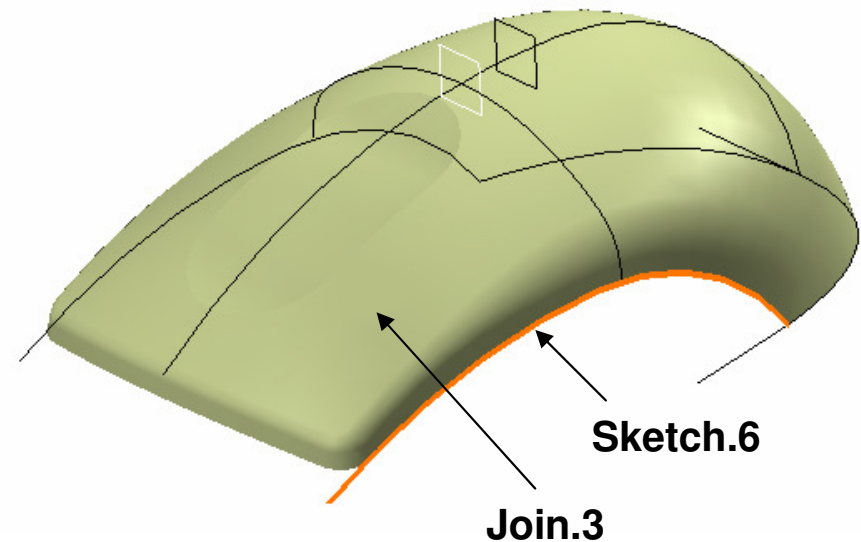
Tutorial 2B

We are going to build a lower skin…

To create a swept surface:-

- Click "**Sweep**" icon
- Select "Line" as Profile-type
- Select "with draft direction" as Subtype
- Select "Combine.1" as Guide Curve1
- Select "xy plane" as direction
- Enter **10deg** as Angle
- Select inward downward arrow
- Enter **20mm** as Length1
- Click ok to complete
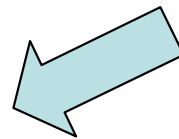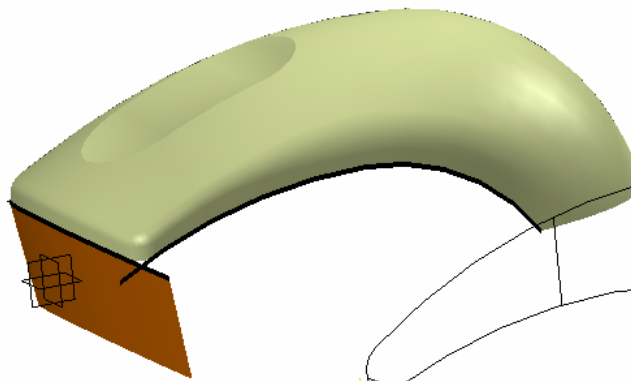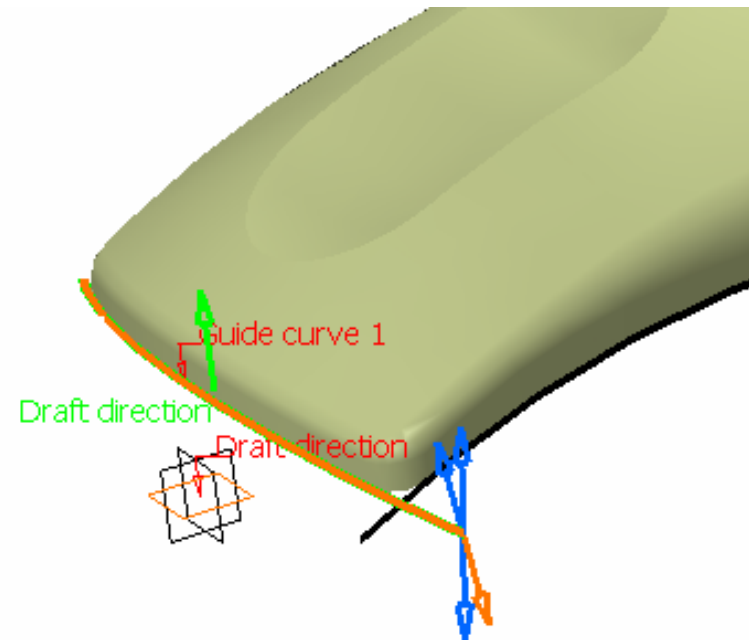
A- 53

Tutorial 2B

To create another swept surface:-

- Click "**Sweep**" icon
- Select "Line" as Profile-type
- Select "with draft direction" as Subtype
- Select "Sketch.6" as Guide Curve1
- Select "xy plane" as direction
- Select Tab-page" Location values"
- Click two endpoints of "Sketch.6 △☆
- Enter 10deg as Angle of △
- Enter 2deg as Angle of ☆
- Select inward downward arrow
- Enter **20mm** as Length1
- Click ok to complete

A- 54

By Dickson Sham (ME dept, HKPU)
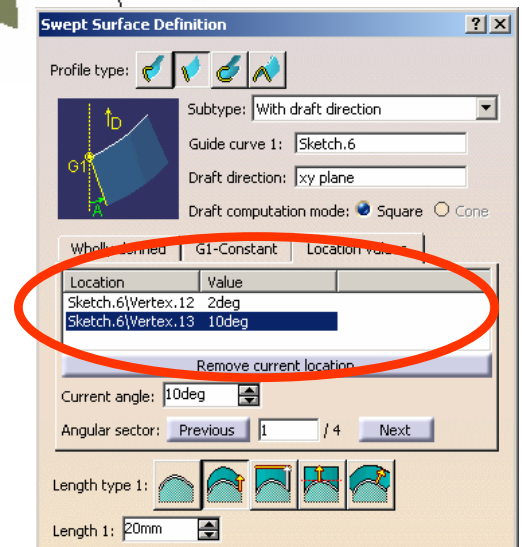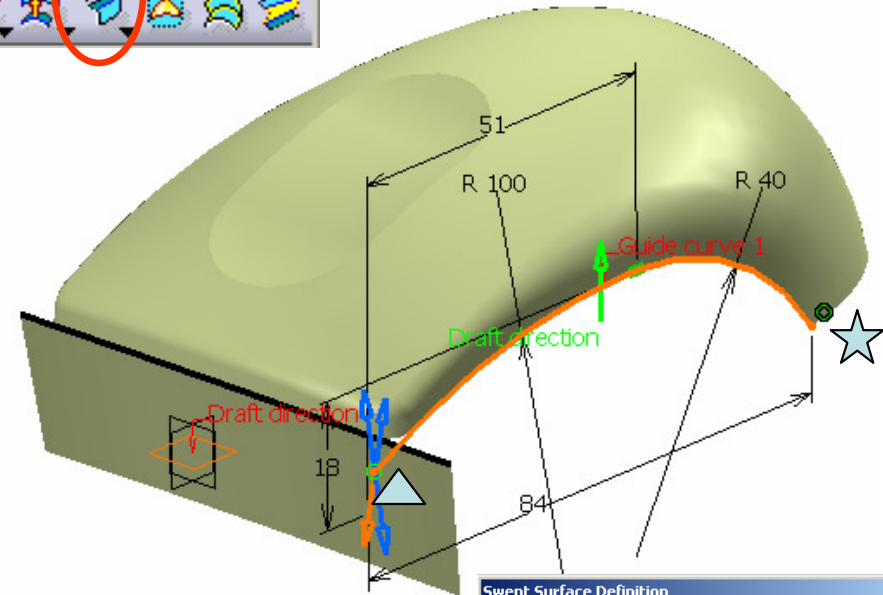
Tutorial 2B

To create a surface by mirroring:-

- Click "**Symmetry**" icon;
- Select "Sweep.5" as element;
- Select "zx plane" as reference;
- Click ok to complete.

Zx plane
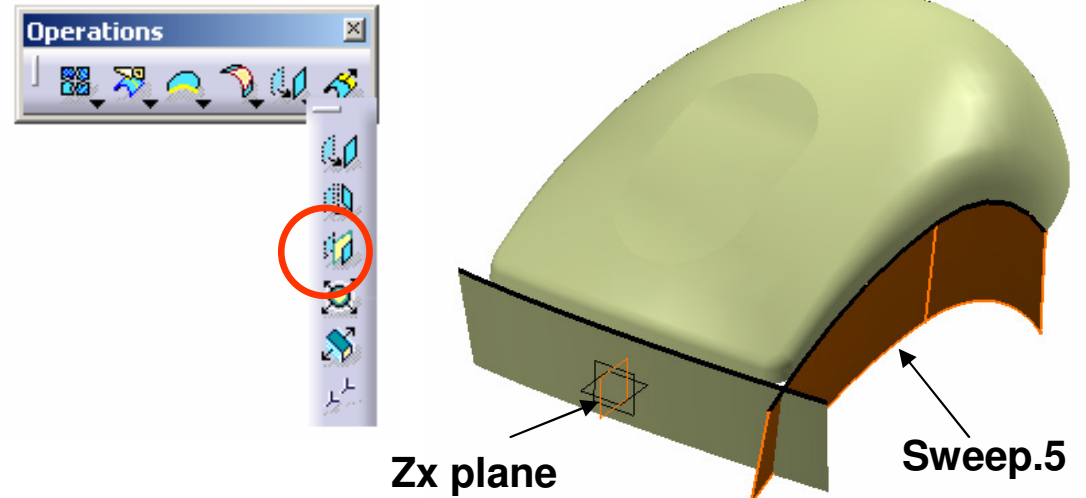
Sweep.5

To create a Fillet between 2 surfaces:-

- Click "**Shape Fillet**" icon;
- Select "Sweep.4" & "Sweep.5";
- Enter **4.5mm** as Radius;
- (Red arrows should point inward. If not, click it once)
- Click ok to complete.

Sweep.5

Sweep.4

A- 55
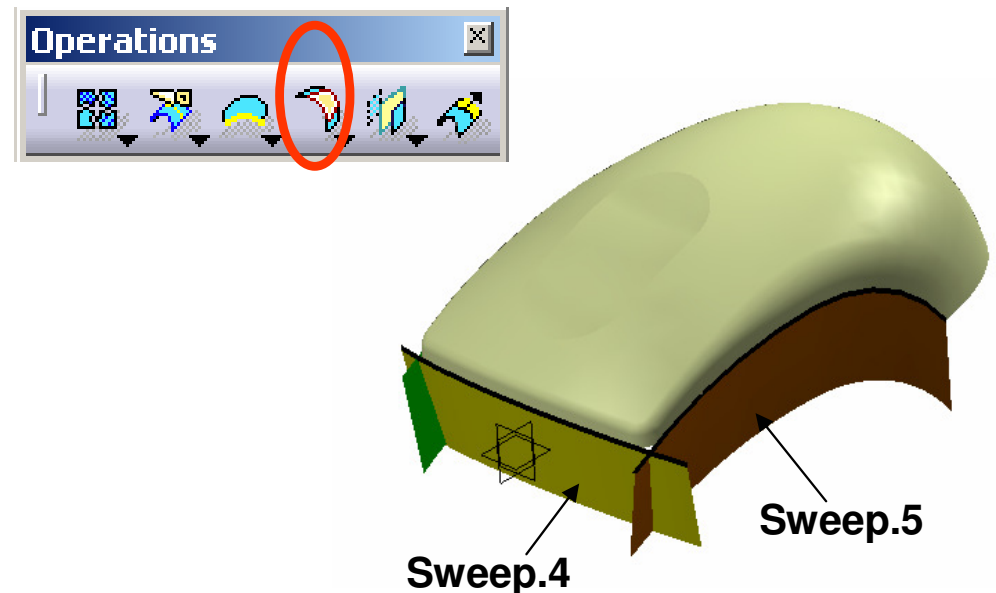
Tutorial 2B

To create another Fillet between 2 surfaces:-

- Click "**Shape Fillet**" icon
- Select "Fillet.3" & "Symmetry.4"
- Enter **4.5mm** as Radius
- (Red arrows should point inward. If not, click it once)
- Click ok to complete

Symmetry.4

Fillet.3
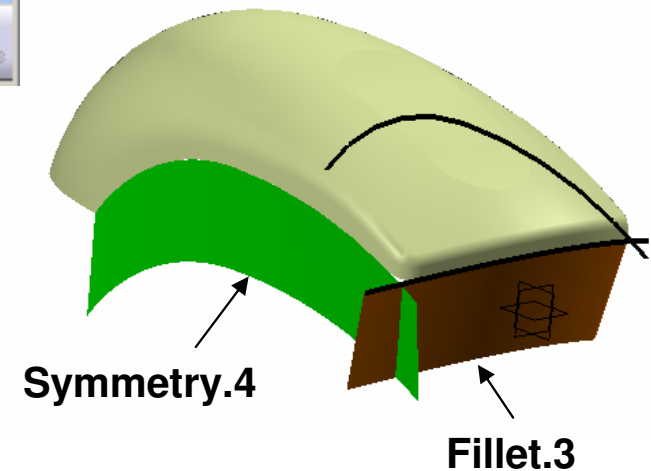
To create a bottom surface:-

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

Draw a line on x-axis

A- 56
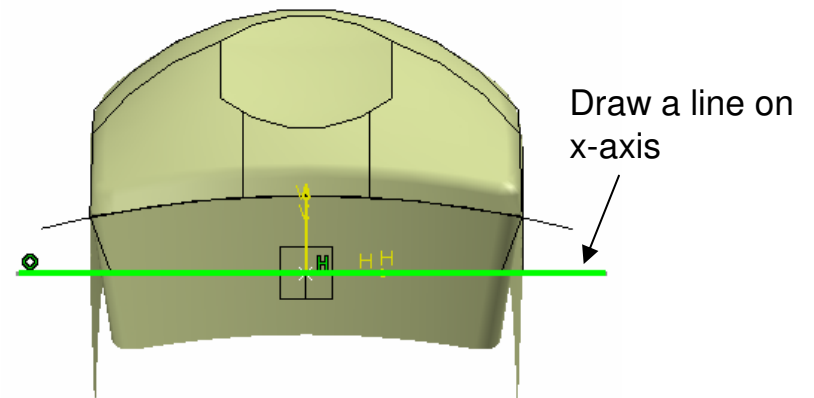
Tutorial 2B

- Click "**Extrude**" icon;
- Enter 120mm as Limit 1;
- (You may need to click "reverse direction" icon to change the extrusion direction)
- Click ok to complete.
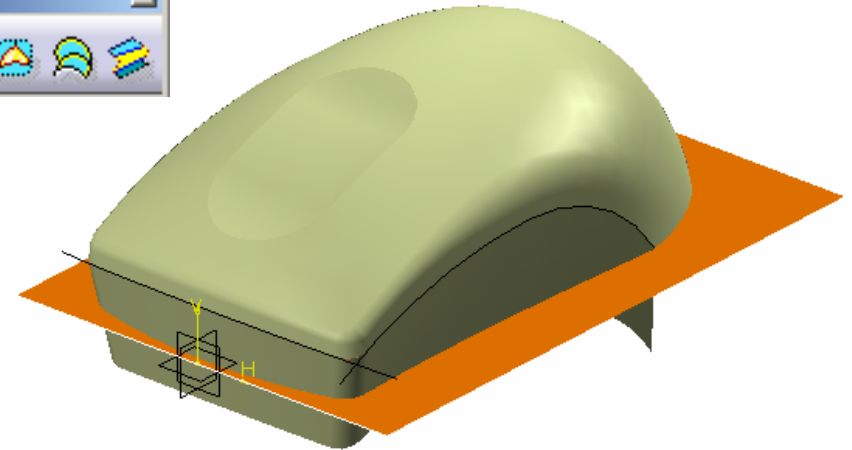
To join surfaces:-

- Click "**Join**" icon
- Select 'Join.3" & "Fillet.4"
- Enter **0.1mm** as Merging direction to correct the discrepancy between two fillet surfaces
- Click "ok" icon to exit

Join.3

A small gap

Fillet.4

A- 57
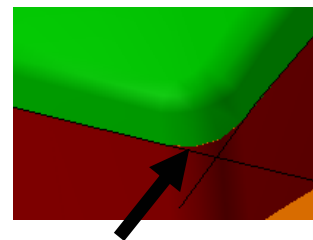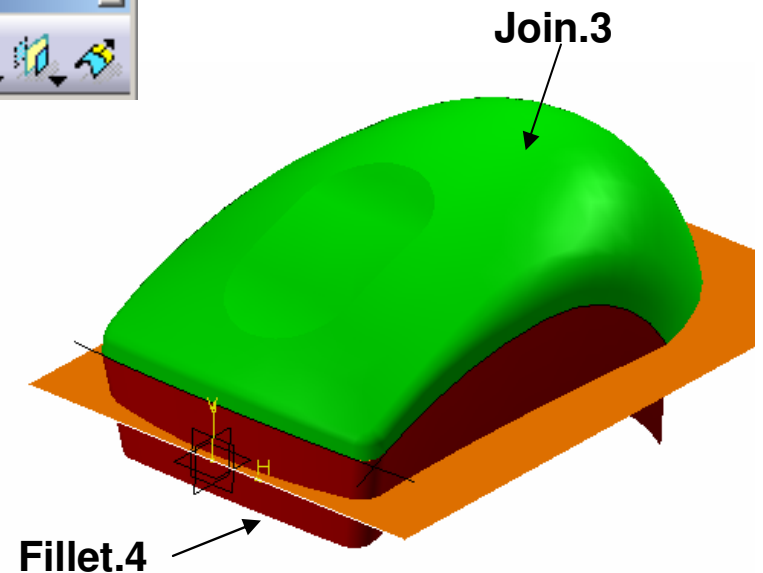
Tutorial 2B

To trim and join surfaces:-

- Click "Trim" icon
- Select 'Extrude.1" & "Join.3"
- Click "Other side" icon until you get the surface as shown
- Click ok to complete
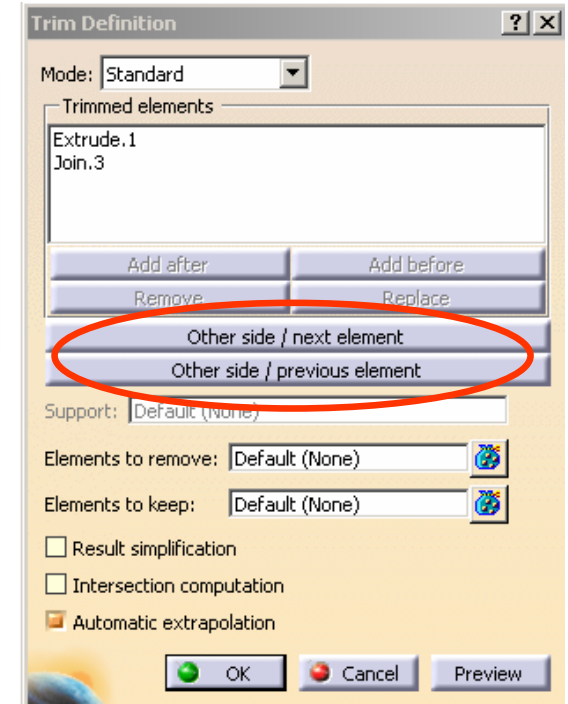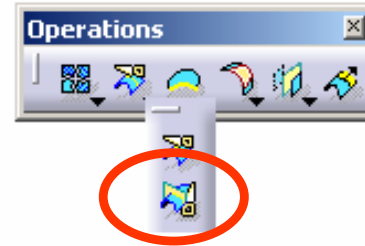
File/Save

 Mouse_mastermodel_a.CATPART

Join.3

Extrude.1

A- 58

END of Tutorial 2B

By Dickson Sham (ME dept, HKPU)
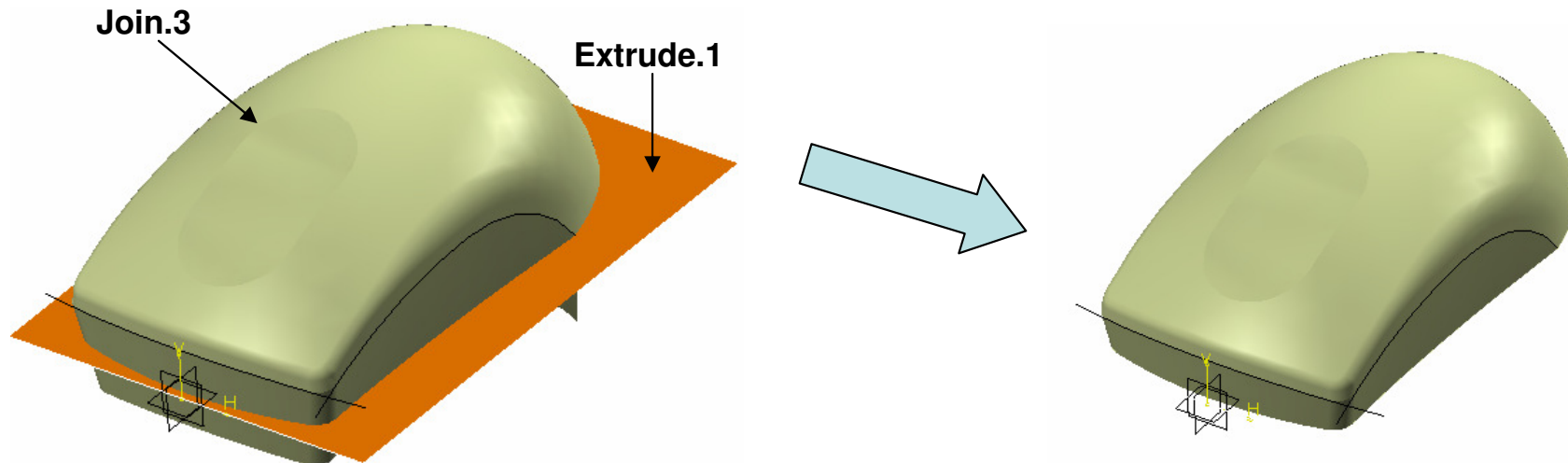
Tutorial 2C

We've built the upper skin & the lower skin of the mouse. Now, we are going to create parting surfaces, transform the skin surface into a solid, and then split it into separate components.
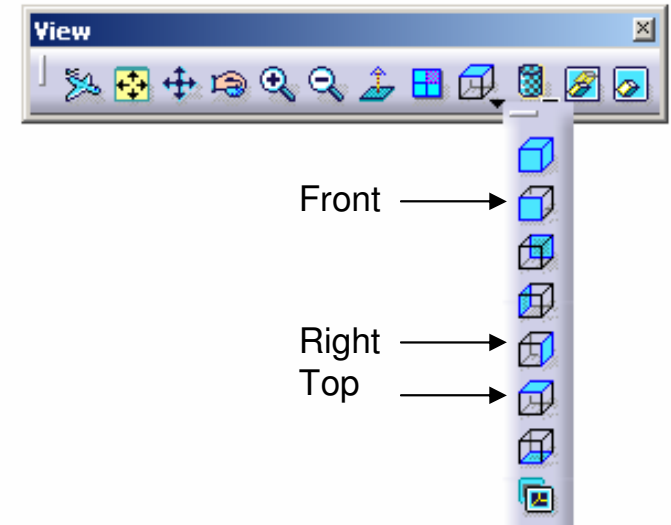
- *Reopen the file* "Mouse_mastermodel_a.CATPART" ;
- *Ensure that the current workbench is "Generative Shape Design";*
- Click **"Front View"** icon to check the front view;
- Click **"Right View"** icon to check the right view;
- Click **"Top View"** icon to check the top view;

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

View

Front

Right
Top
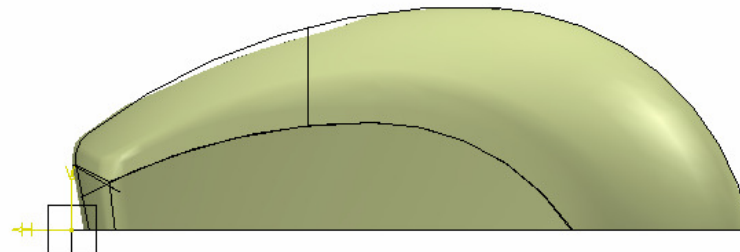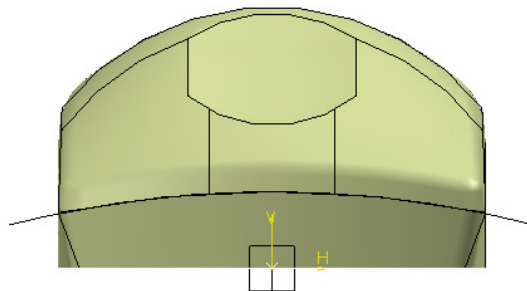
A- 59
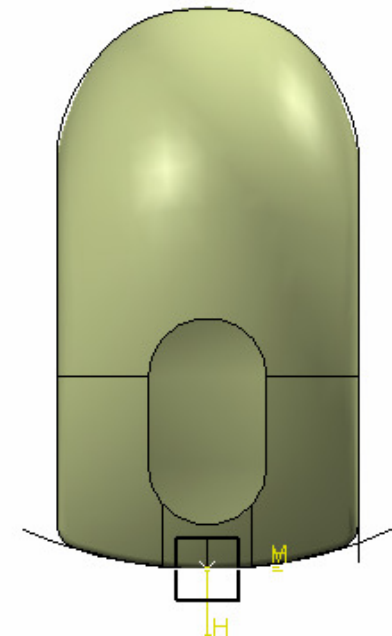
Tutorial 2C

To Insert a Geometrical Set:-

- Select "Insert/Geometrical Set…" from the menu bar;

- Enter "Parting Surfaces" as Name;

- Click ok to confirm.

- (Now a new set is created. It is underlined, and so all the coming elements will be stored under this set.)
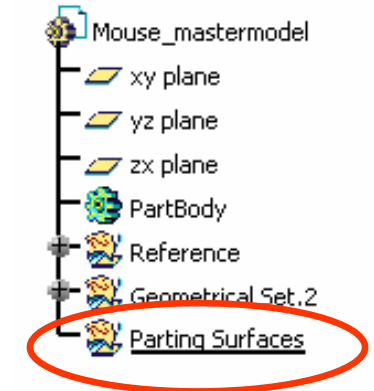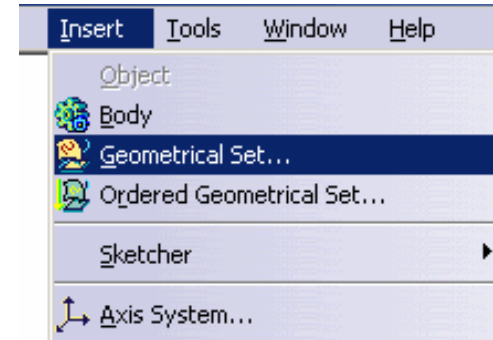
To create a swept surface:-

- Click "**Sweep**" icon;

- Select "Explicit" as profile type;

- Select "With reference surface" as subtype;

- Select "Combine.1" as profile;

- Select "Sketch.6" as Guide curve;

- Click ok to complete.

A- 60
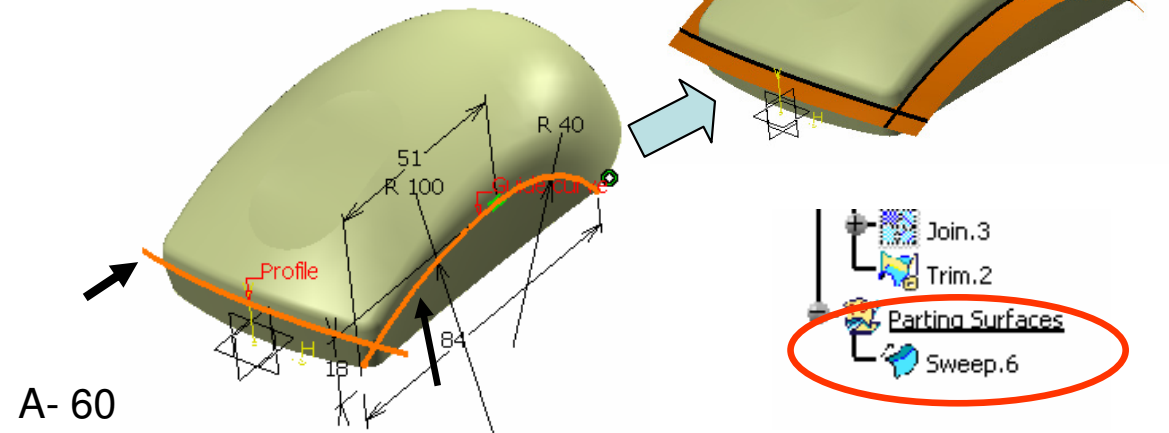
Tutorial 2C

To extract a curve from the surface:-

- Click **" Extract"** icon
- Select "Tangency Propagation" as continuity propagation
- Select the edge as shown
- Click ok to complete
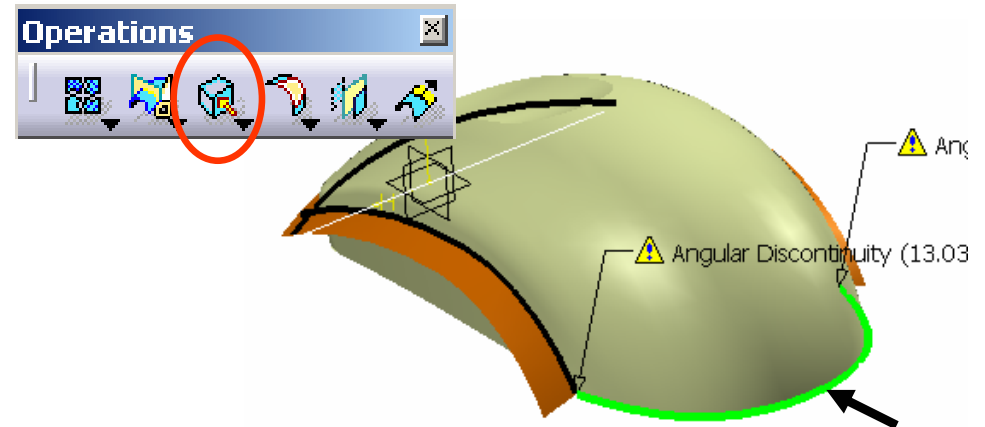
To create an extruded surface:-

- Click **"Extrude"** icon;
- Select the curve"Extract.1" as profile
- Select "xy plane" as direction
- Enter **3.0mm** as Limit1
- Enter **10.0mm** as Limit2
- Click ok to complete

A- 61
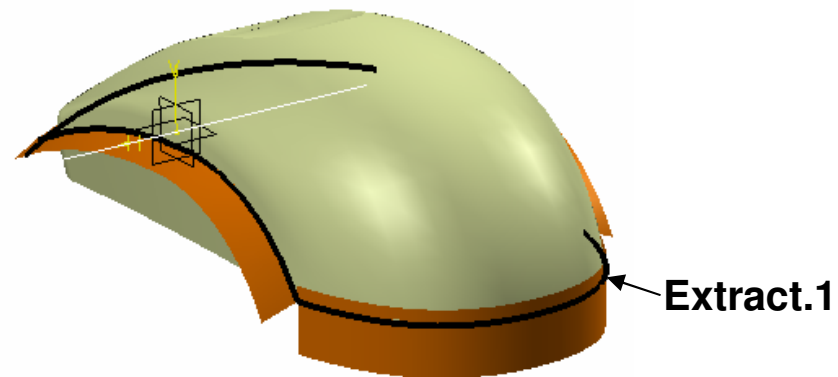
Tutorial 2C

Hide "Geometrical set.2"

(Do it yourself)
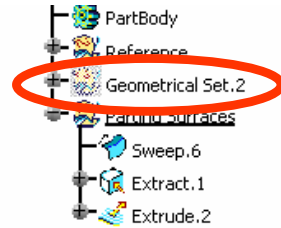
To create an offset surface:-

- Click **"Offset"** icon
- Select "Extrude.1"
- Enter **2.5mm** as Offset value
- Click " Reverse Direction" if the red arrow is NOT pointing inward
- Click ok to complete.

Offset Surface Definition

Surface: Extrude.2
Offset: 2.5mm

Parameters | Sub-Elements to remove
Smoothing: None
Maximum Deviation: 0.1mm
Reverse Direction
☐ Both sides
☐ Repeat object after OK

OK   Cancel   Preview
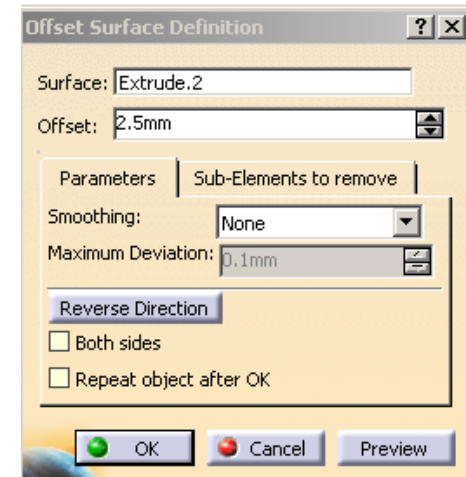
Hide Extrude.2 & Extract.1

(Do it yourself)

HIDE
Extract.1 &
Extrude.2

A- 62
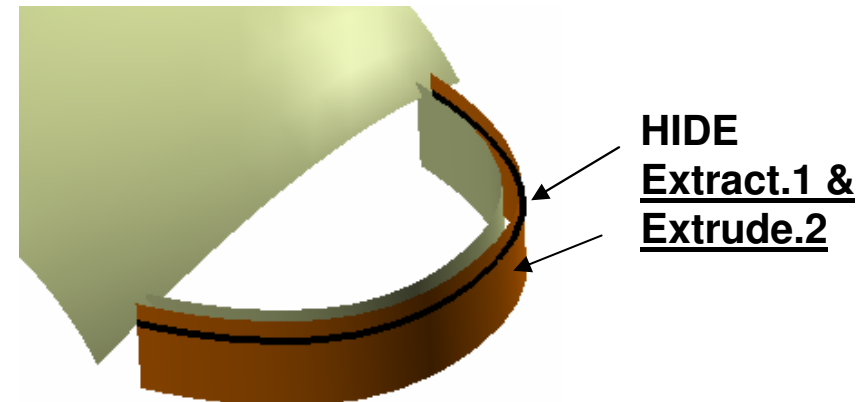
Tutorial 2C

To extend a surface:-

- Click **" Extrapolate"** icon
- Select the edge ☆ of Sweep.6 as boundary;
- Select "Sweep.6" as Extrapolated
- Enter **20mm** as Length
- Select "Assemble result" option
- Click ok to complete
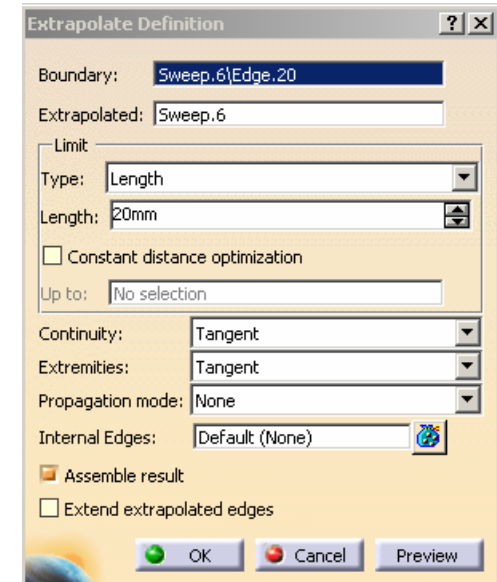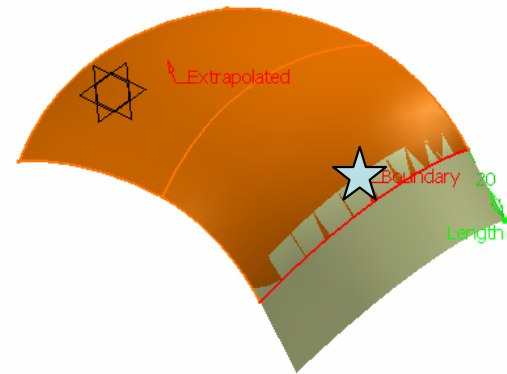
Similarly,

Extend the both edges of the Offset.1

by 10mm (Do it yourself)

Offset.1
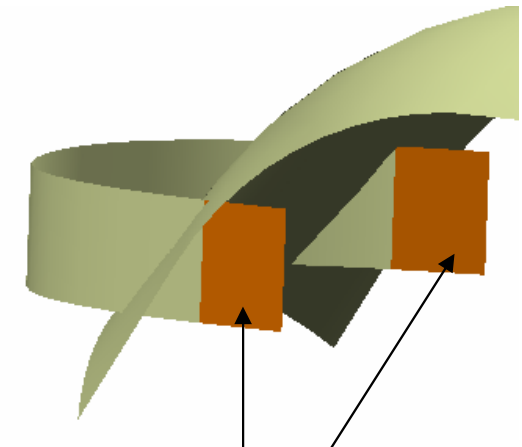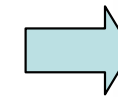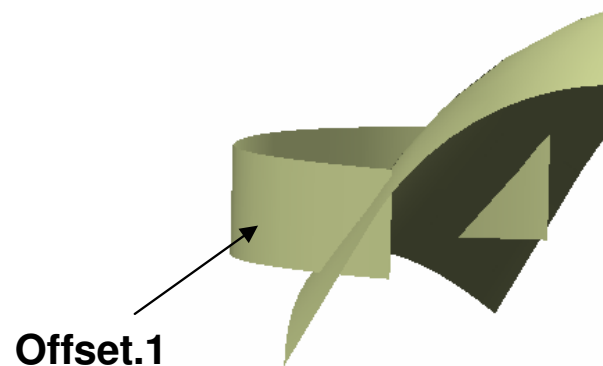
Extended by 10mm

A- 63

Tutorial 2C

To create a line:-

- Click **" line"** icon
- Select the two end points of the surface ☆
- Click ok to complete
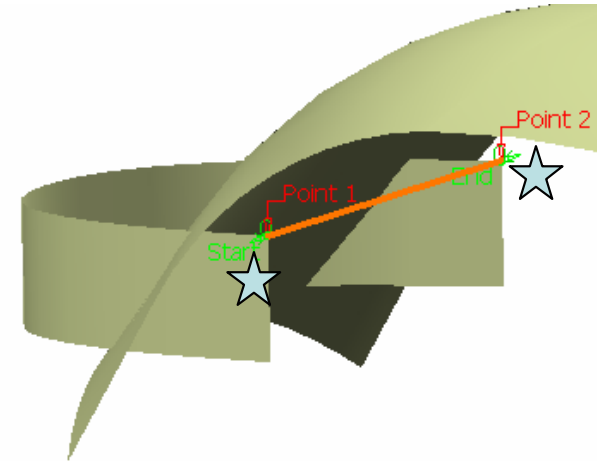
To create an extruded surface:-

- Click "**Extrude**" icon
- Select the line Line.3 as profile
- Select yz plane as direction
- Enter **50mm** as Limit1
- Enter **0 mm** as Limit2
- Click " Reverse Direction" to get the surface as shown
- Click ok to complete

Line.3

A- 64
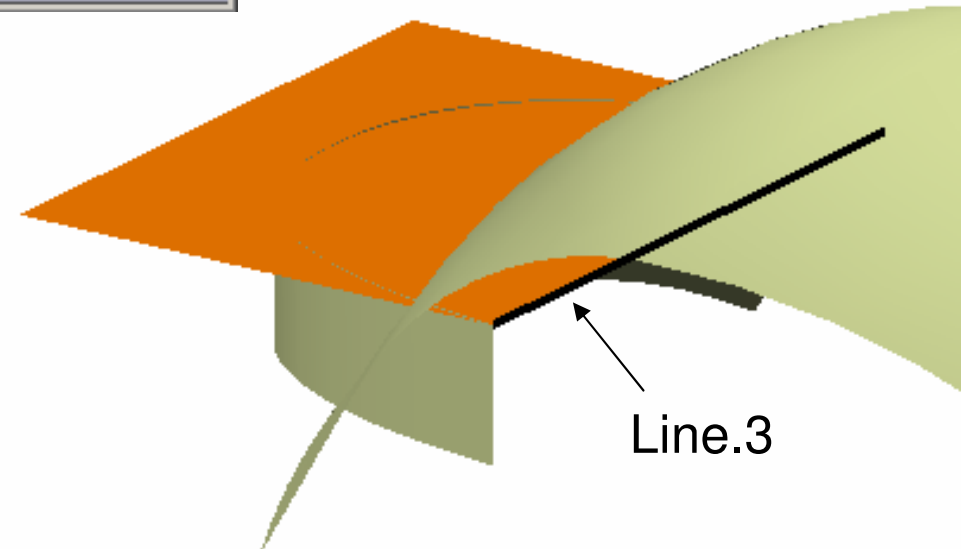
Tutorial 2C

To Trim & Join 2 surfaces:-

- Click **" Trim"** icon;
- Select "Extrude.3" & "Extrapolate.3";
- Click "Other side" option until the resultant is the same as shown;
- Click ok to complete.
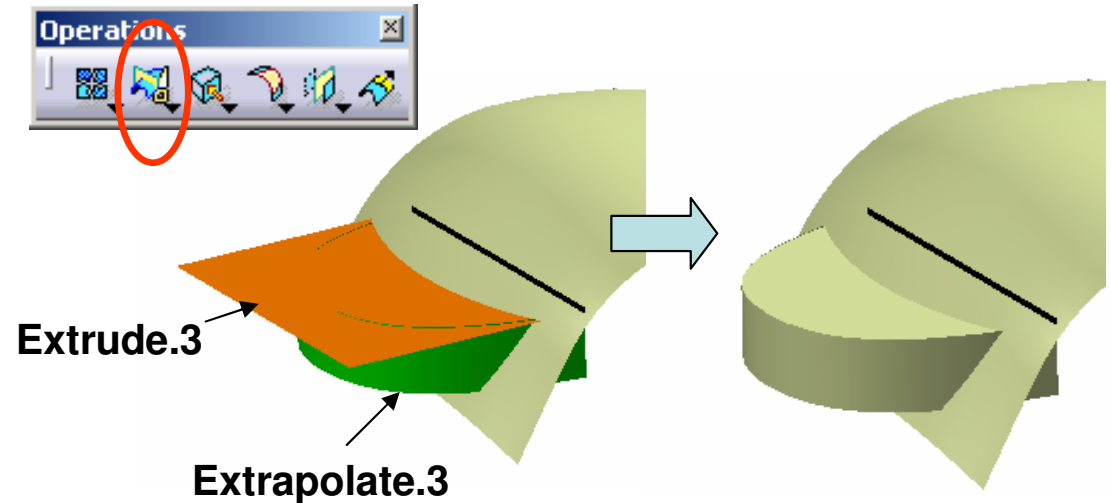
Similarly

- Click **" Trim"** icon;
- Select "Trim.3" & "Extrapolate.1";
- Click "Other side" option until the resultant is the same as shown;
- Click ok to complete.

Extrude.3

Extrapolate.3

Trim.3

Extrapolate.1

A- 65
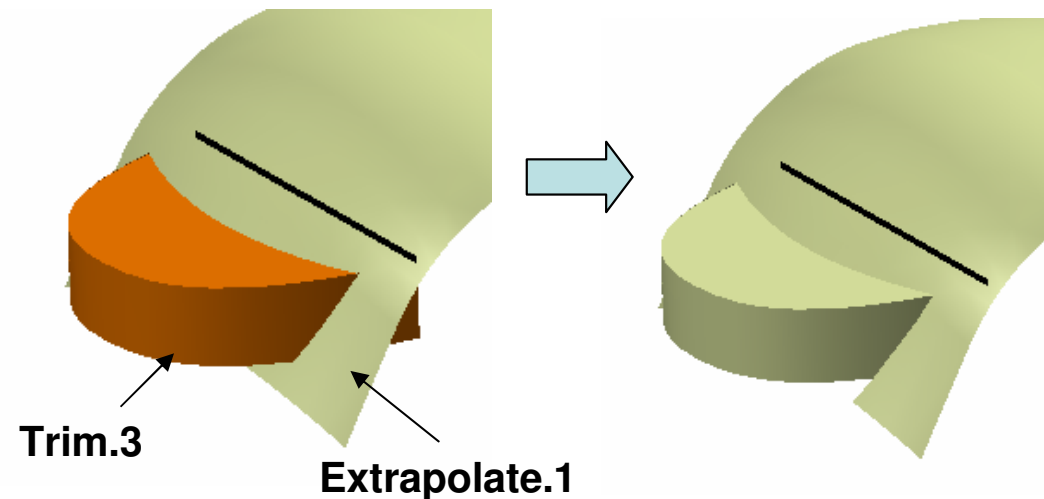
Tutorial 2C

Show Geometrical set.2 again
(Do it yourself)
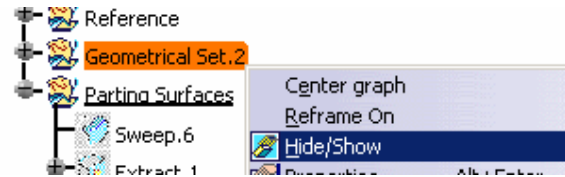
To show previous element:-

•   Click "**Swap visible space**" icon

•   Select "Sketch.9" & "Plane.4"

•   Click "Hide/Show" icon (Now the two elements are moved to the visible space)

•   Click "Swap visible space" icon

Sketch.9

Plane.4

A- 66
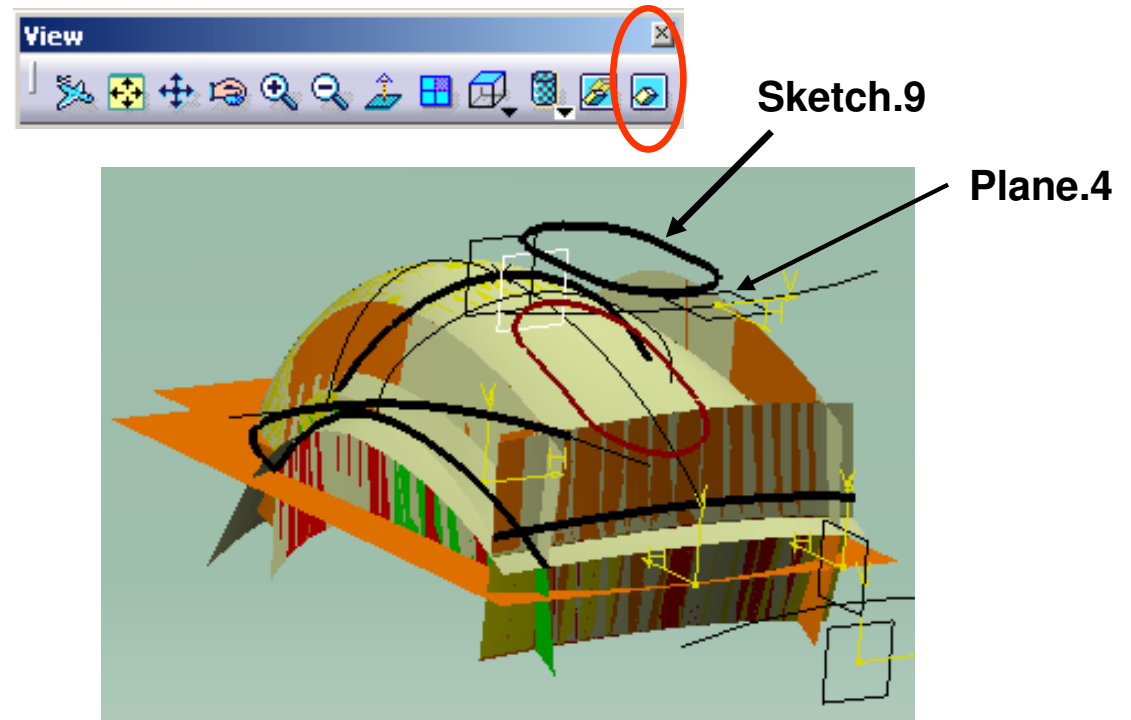
Tutorial 2C

To create 2nd parting surface:-

- Click "**Sketch**" icon;

- Select xy plane;

- Multi-Select the three edges of Sketch.9 ☆

- Click **"Project 3D elements"** icon (We now have three projected curves in yellow on the sketch)

- Draw 2 vertical lines to go across the model;

- Click Exit icon to exit.

Project 3d elements

Operation

Project 3 curves onto sketch
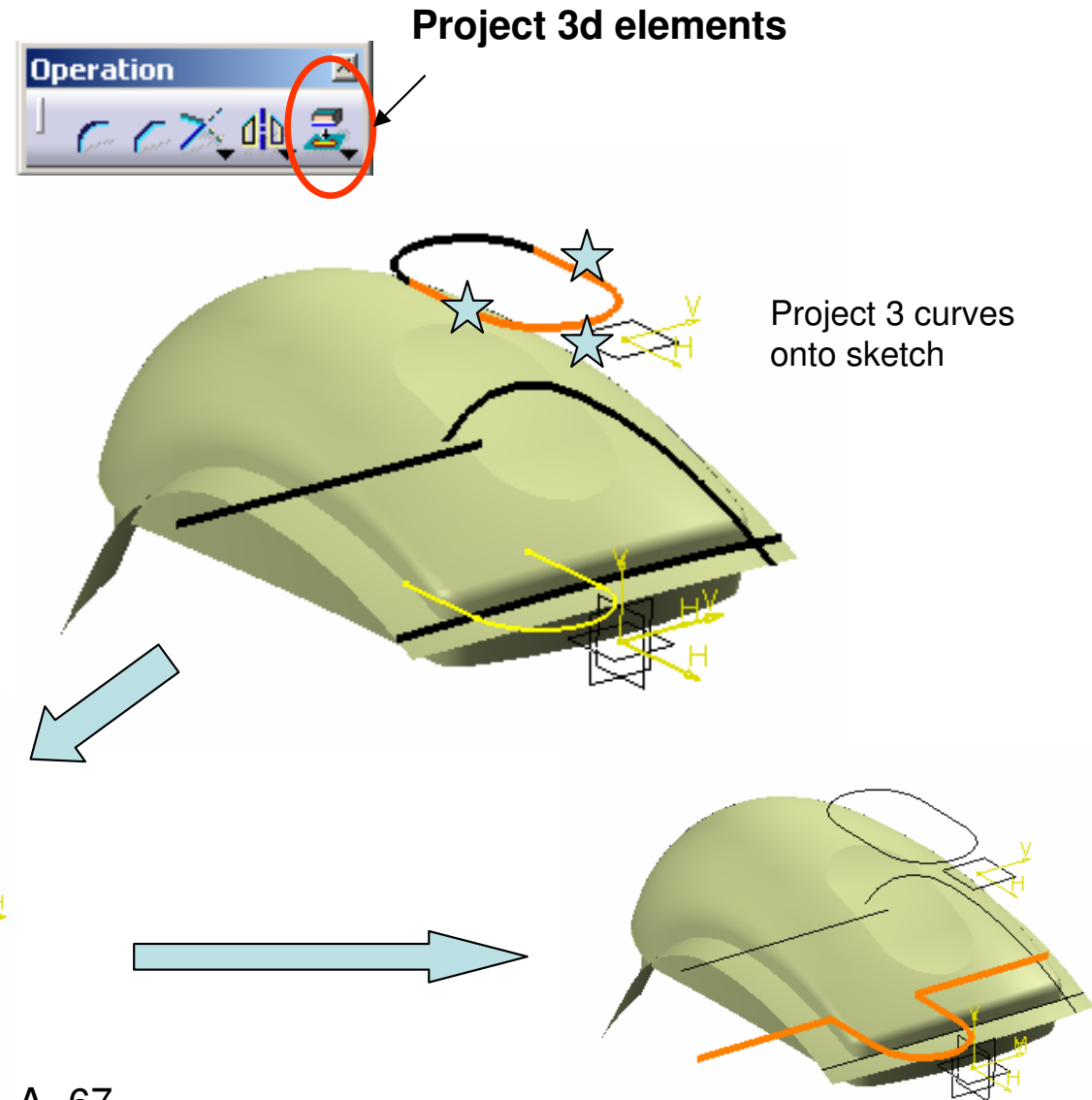
Draw 2 lines

A- 67

Tutorial 2C

- Click **"Extrude"** icon;
- Select "Sketch.11" as profile;
- Select "xy plane" as direction (automatically)
- Enter **50mm** as Limit1;
- Click ok to complete.

Sketch.11
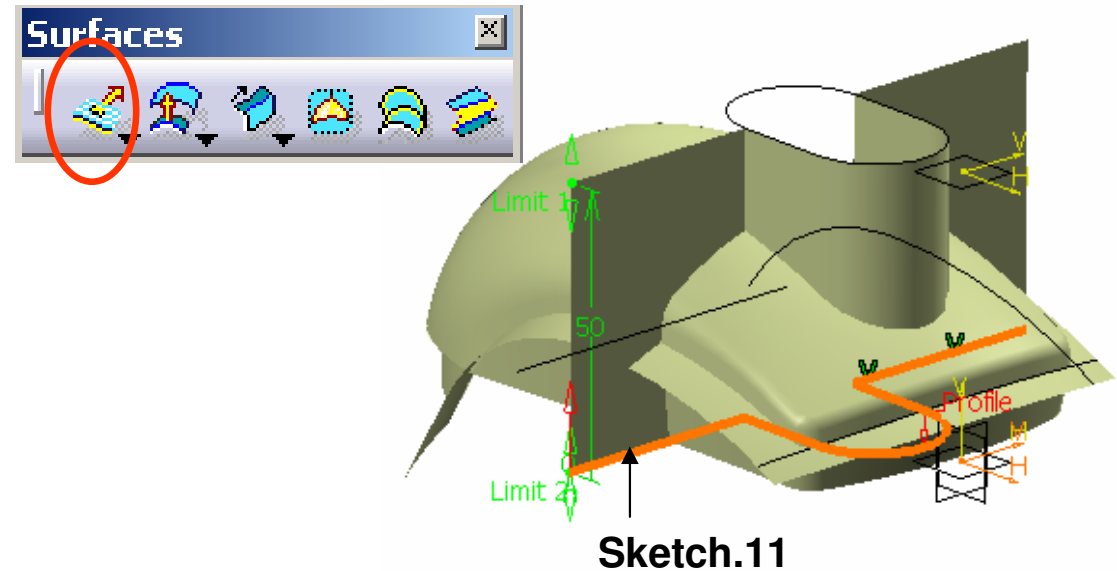
Hide all elements except

Xy plane

Yx plane

Zx plane

Trim.2 (last element under Geometrical set.2)

Trim.6 (under Parting Surfaces)

Extrude.4 (under Parting Surfaces)

(Do it yourself)

Trim.2

Extrude.4

Trim.6

A- 68
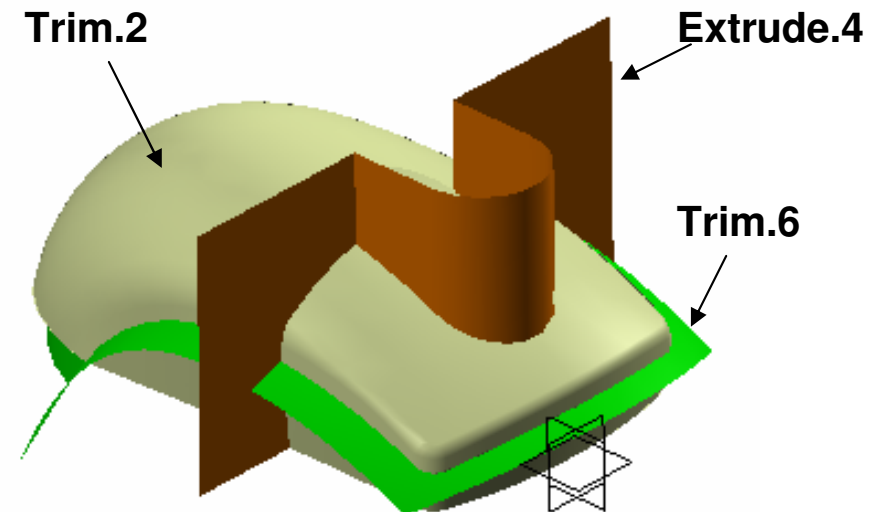
Tutorial 2C

To convert a surface into a solid:-

- Select **"Start/Mechanical Design/Part Design"**;

- Click **"close surface"** icon;

- Click ok on the warning message;

- Select "Trim.2" as object to close;

- Click ok to complete (Now a solid is created under "Partbody" on the tree)
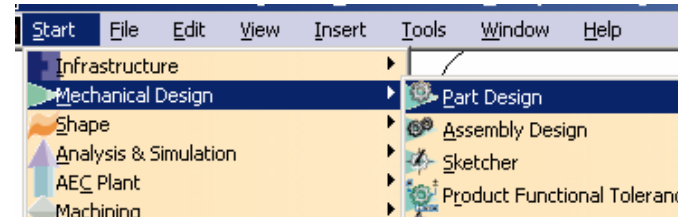
Close surface
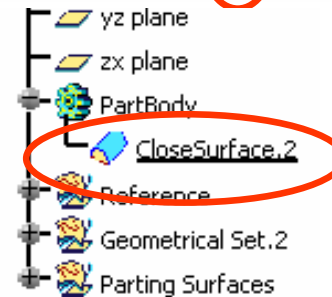
Hide "Geometrical Set.2" and we will see the solid

(Do it yourself)
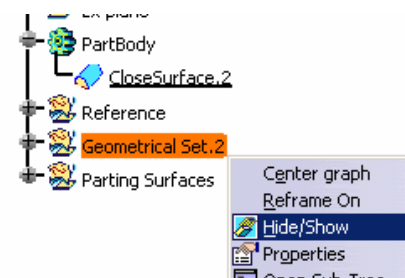
Default color of solid is blue

A- 69
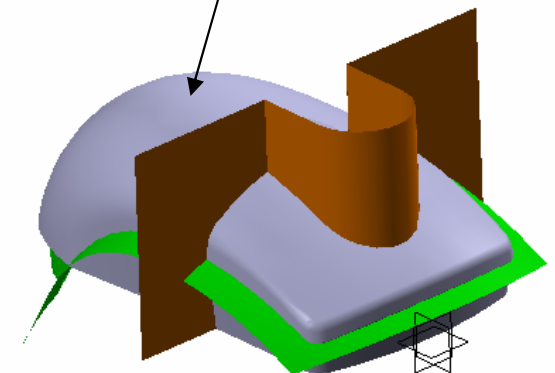
Tutorial 2C

To add a fillet on the solid:-

- Click **"edge fillet"** icon;
- Select "Tangency" as Propagation;
- Select the edge ☆
- Enter **1mm** as radius;
- Click ok to confirm
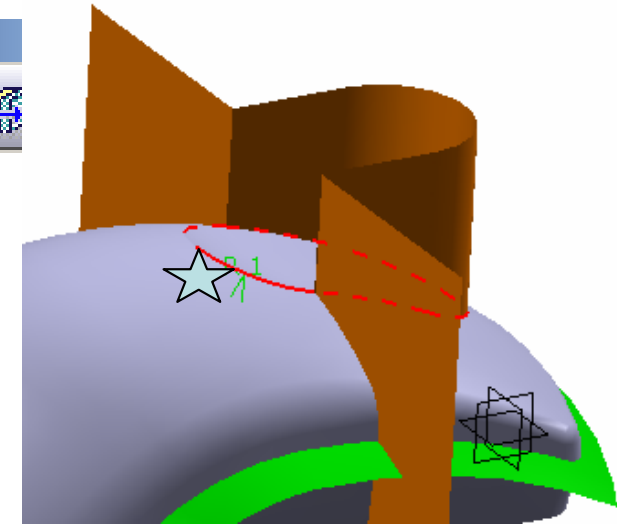
To hollow the solid:-

- Click **"Shell"** icon;
- Enter **2.5mm** as inside thickness;
- Enter **0mm** as outside thickness;
- (Do not pick any face of the solid)
- Click ok to confirm
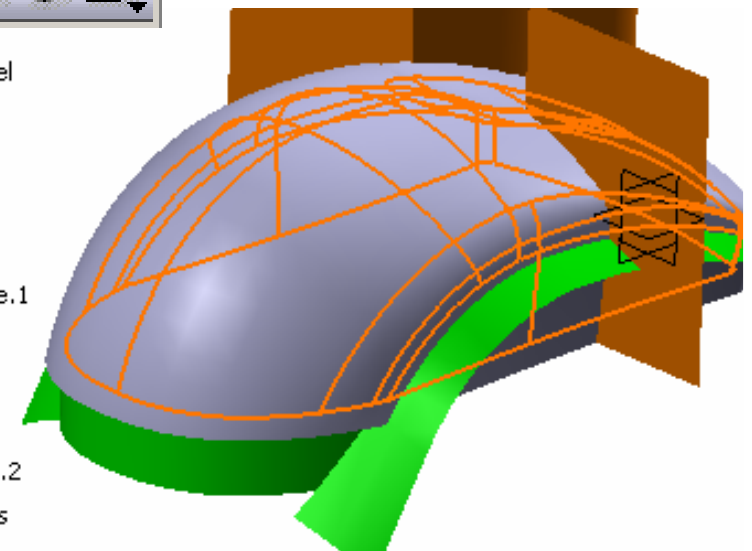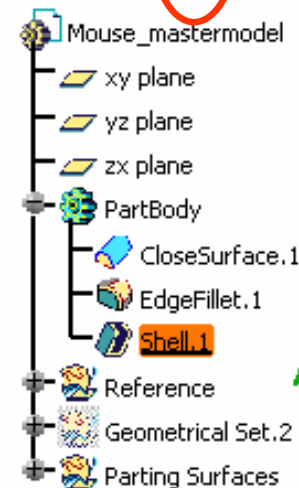
File/Save

(The MASTER model is finished, and we are going to split it into separate parts)

A- 70

Tutorial 2C
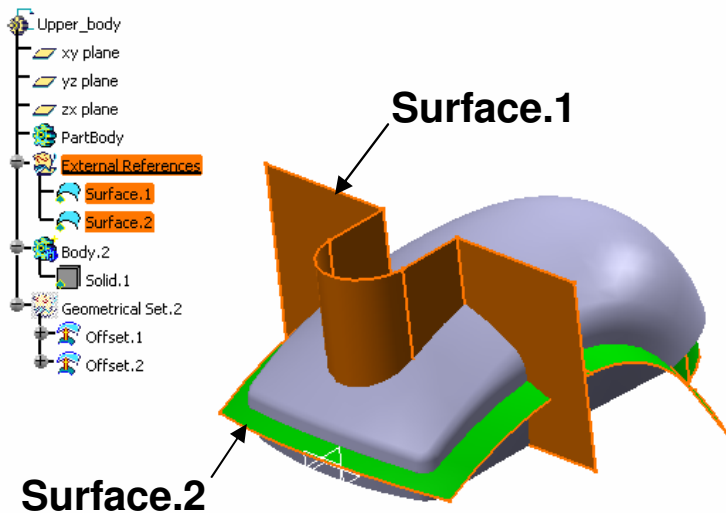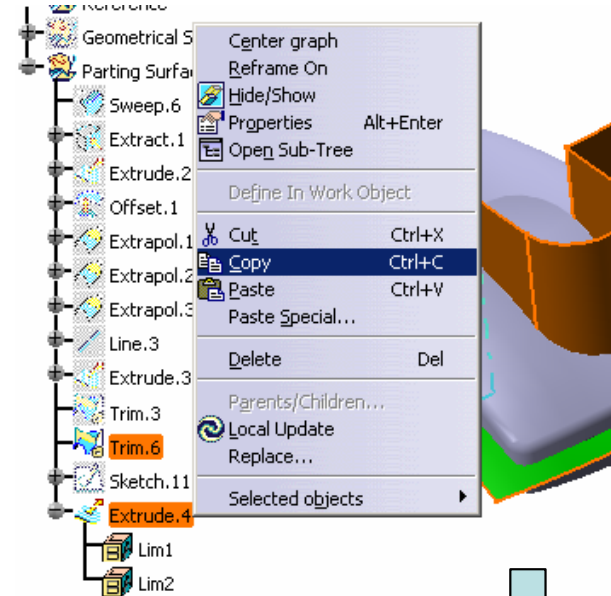
To create the upper body:-

- Select File/New;
- Select Part as type;
- Enter Upper_body as part name;
- Click ok to complete.

- Select Window/Tile Vertically (we can see Mouse_Master & Upper Body at the same time)
- Right-click "PartBody" of Mouse_master_a.CatPart;
- and then select **"Copy";**

- Right-click "Upper_body" of the tree of Upper_body and then select **"Paste Special…"**
- Select "**As Result with link**";
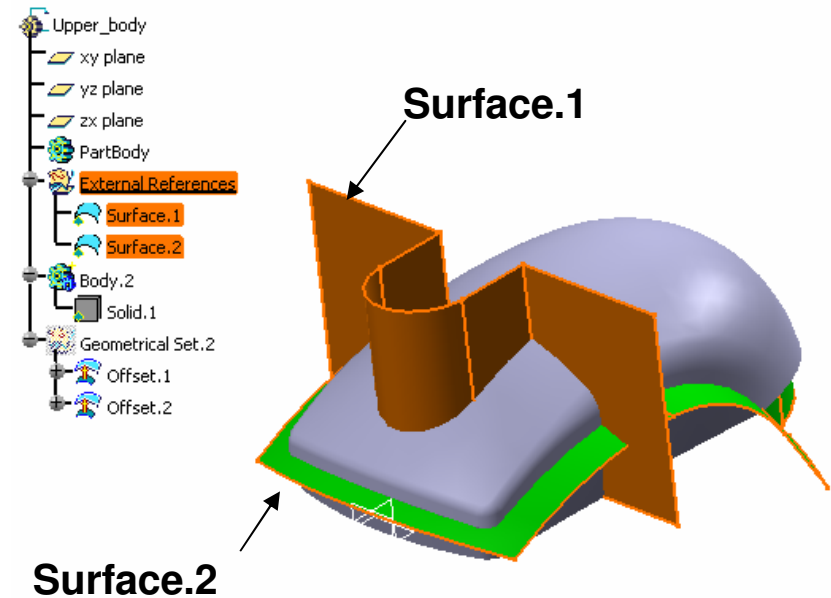- Click ok to complete.
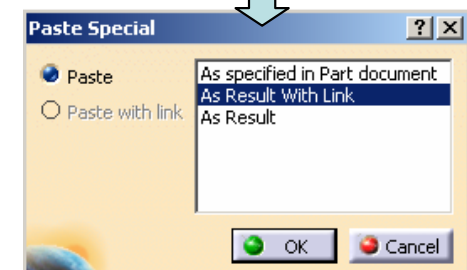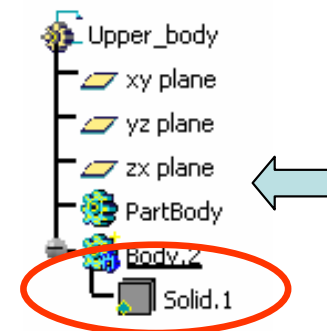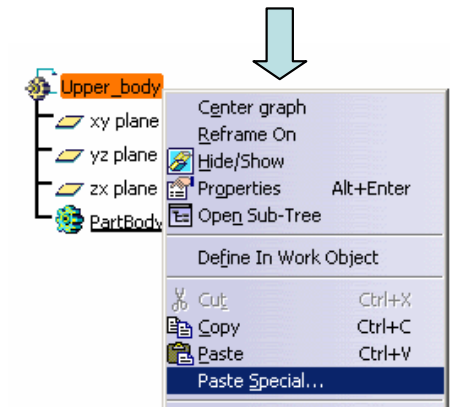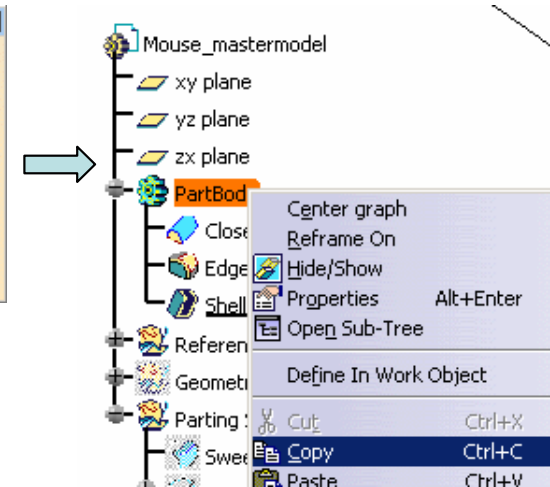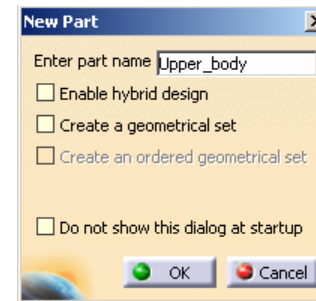
A- 71

Tutorial 2C

- Multi-Select the two parting surfaces "Trim.4" & Extrude.4"

- Right-click on either one and then select **"Copy"**

- Right-click "Upper_body" of the tree of Upper_body and then select **"Paste Special…"**

- Select **"As Result with link"**

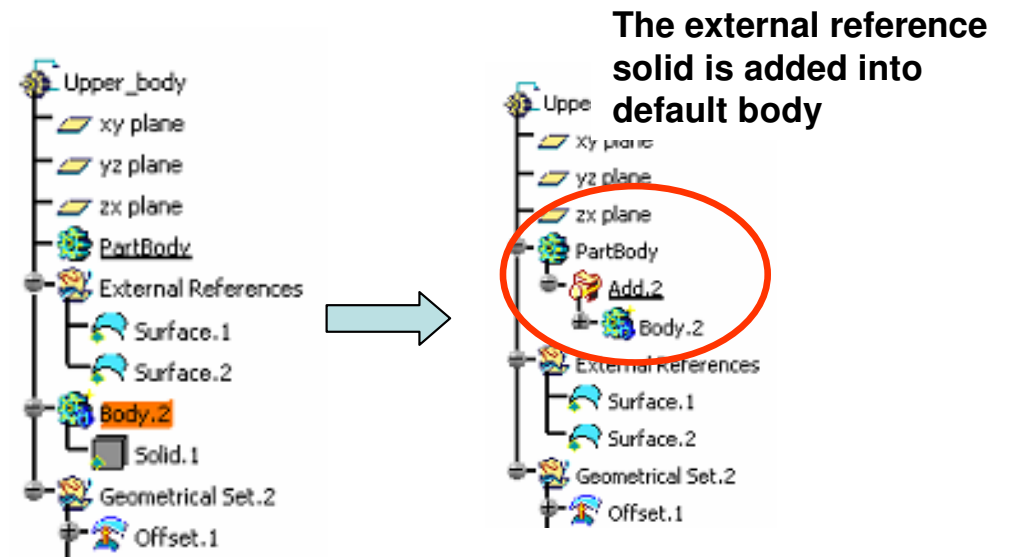- Click ok to complete

Surface.1

Surface.2

A- 72

Tutorial 2C

- Select Start/Shape/Generative Shape Design
- Click **"Offset"** icon
- Select "Surface.1"
- Enter **0.5mm** as value
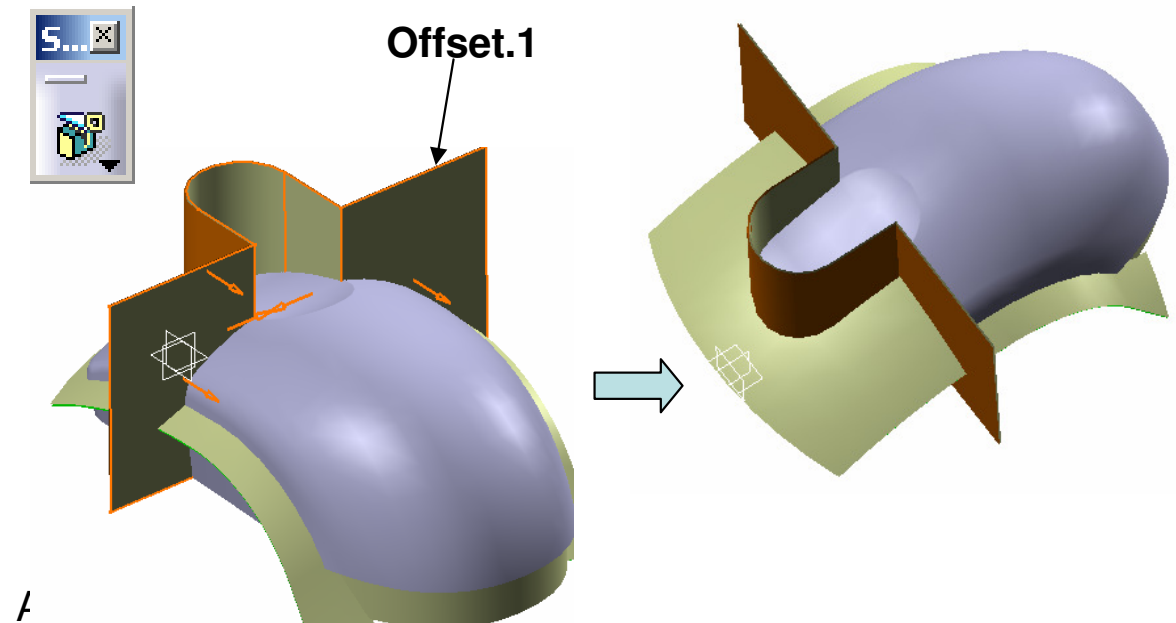- Click "Reverse Direction" icon if the red arrow is not pointing inward
- Click ok to complete

A- 73

Tutorial 2C

- Select Start/Mechanical Design/Part Design
- Right-Click "PartBody" and then select **"Define in work object"**
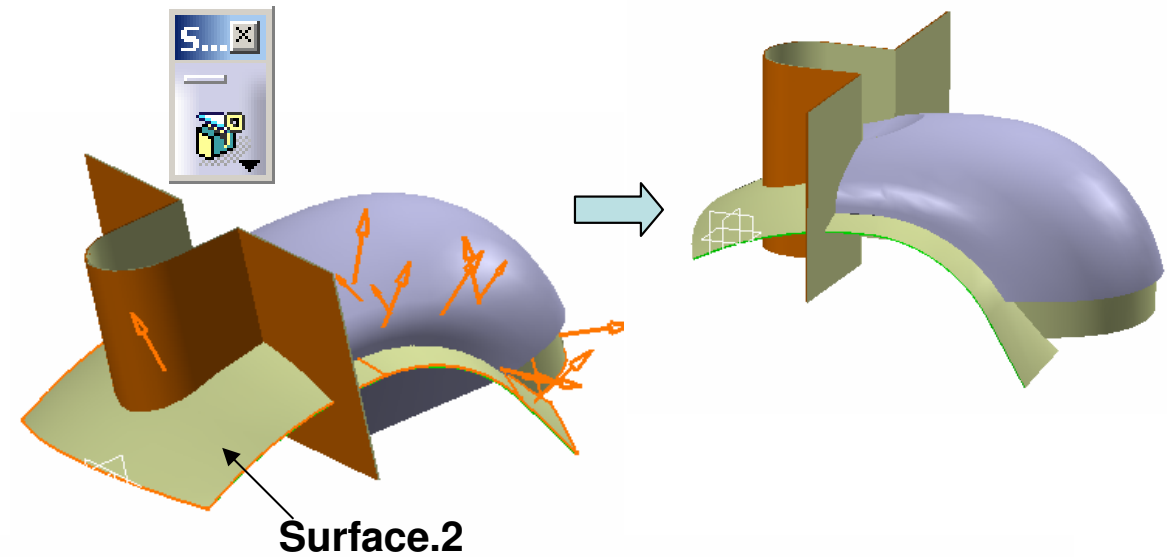- Right-click "Body.2" and select **"Add"**

The external reference solid is added into default body

- Click **"Split"** icon
- Select Offset.1
- Click on the arrow once if it is not pointing inward
- Click ok to complete

Offset.1

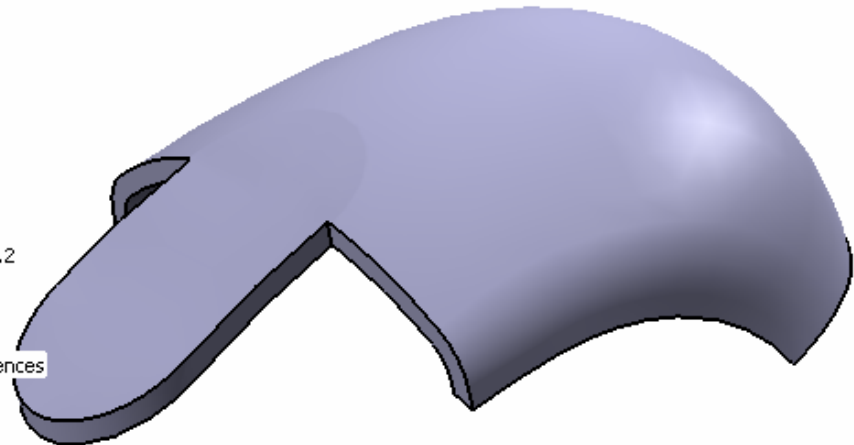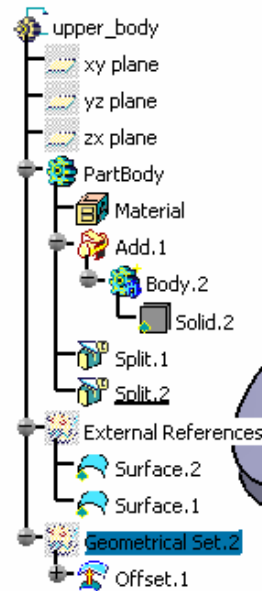# Tutorial 2C

- Click **"Split"** icon again
- Select Surface.2
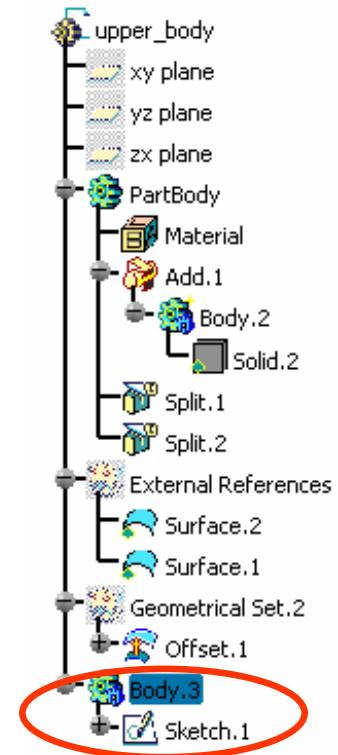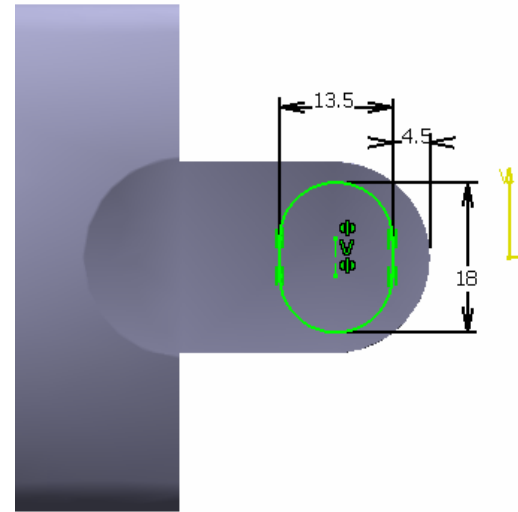- Click on the arrow once if it is not pointing upward
- Click ok to complete

Surface.2

**Hide External References &
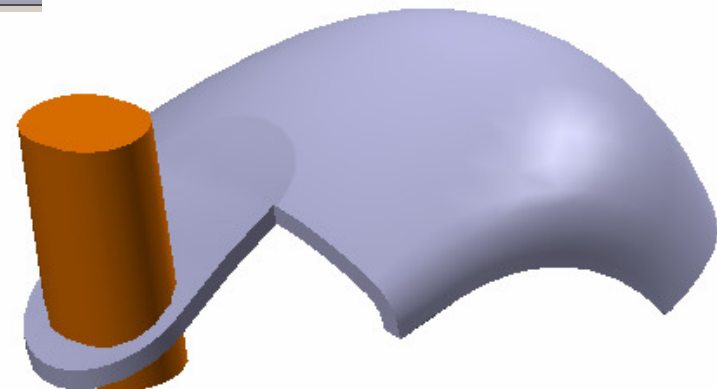Geometrical set.2 on the tree
(Do it yourself)**

A- 75

Tutorial 2C

- Select **"Insert/Body"** from the menu bar;
- Click **"sketch"** icon;
- Select xy plane;
- Draw a profile as shown;
- Click exit icon to exit.

- Click **" Pad"** icon;
- Enter **50mm** as First Limit;
- Enter **-16.5mm** as Second Limit;
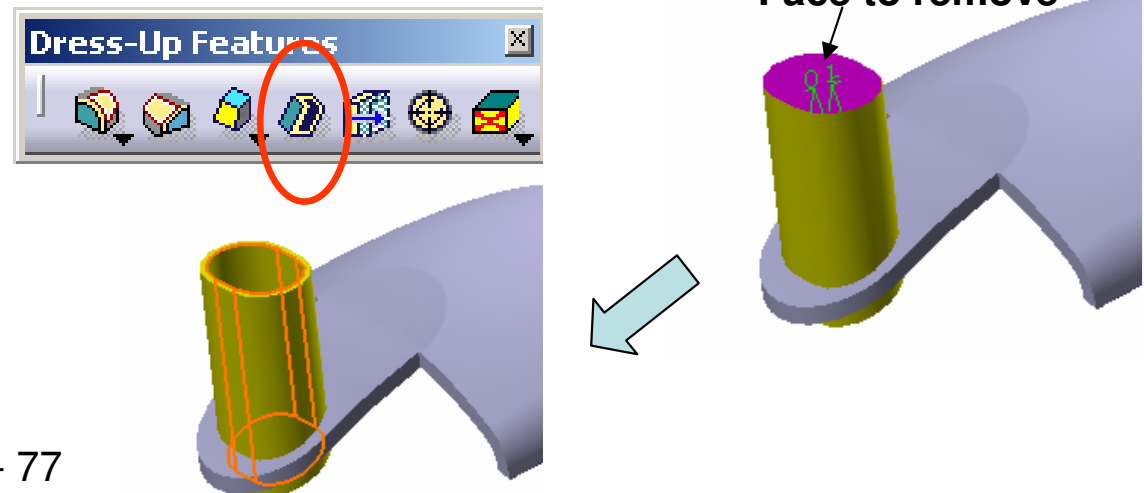- Click ok to complete.

A- 76

Tutorial 2C

- Select "**Draft Angle**" icon
- Select the bottom face as **Face to draft**
- Select the side planar face as **Neutral element**
- Click the red-arrow once if it is not pointing backward
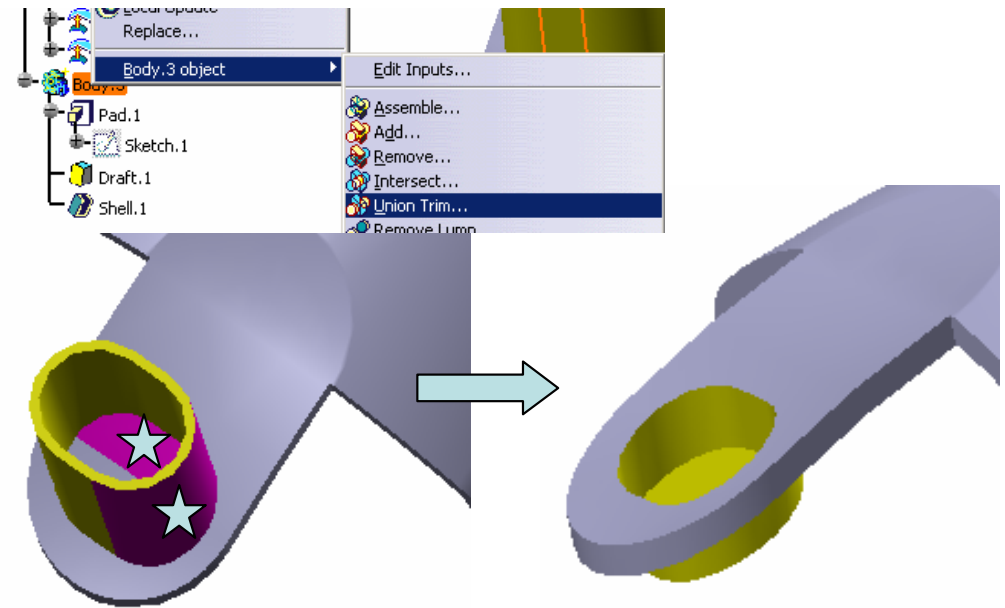- Enter **20deg** as Angle
- Click ok to complete

Neutral element

Face to draft
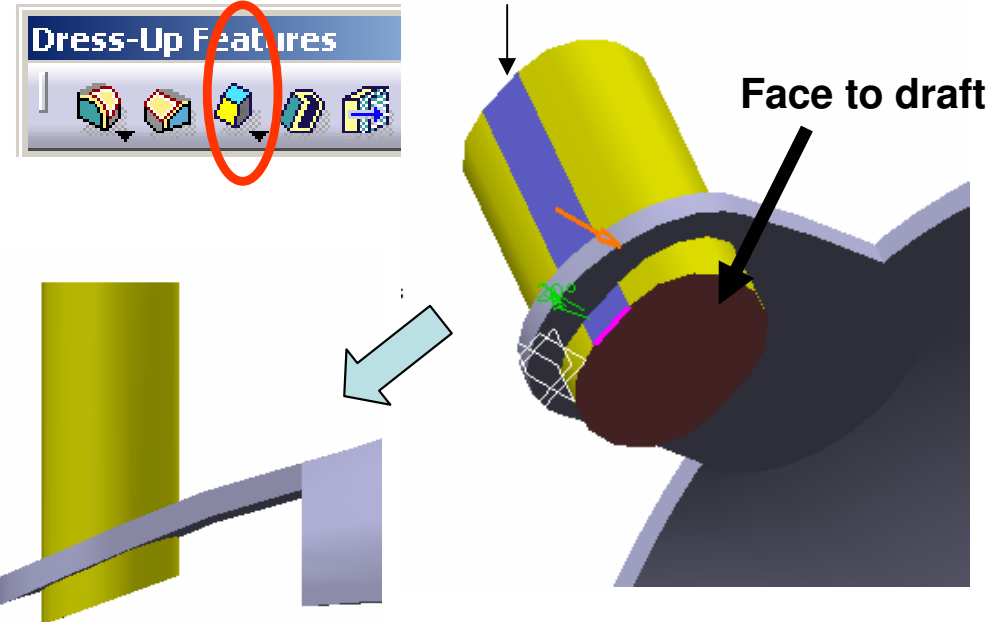
- Click "**Shell**" icon
- Enter **1.5mm** as inside thickness
- Select the top face as **face to remove**
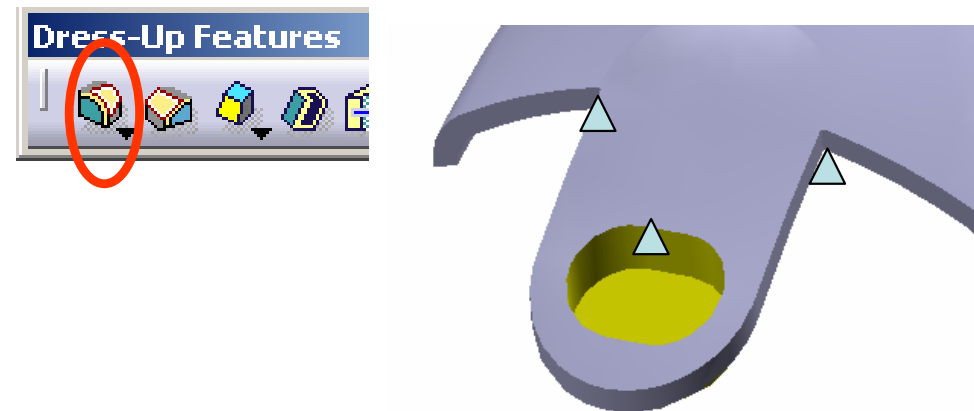- Click ok to complete

Face to remove

A- 77

Tutorial 2C

- Right-click "Body.3";
- Select "**Body3. object/Union-Trim**";
- Click the box "**Faces to remove**";
- Select the faces ☆
- Click ok to complete.

- Click "**Edge Fillet**" icon;
- Select the three sharp edges △
- Enter **1mm** as Radius;
- Click ok to complete.

**File/Save as Upper_body_a.Catpart
then File/Close**

By Dickson Sham (ME dept, HKPU)
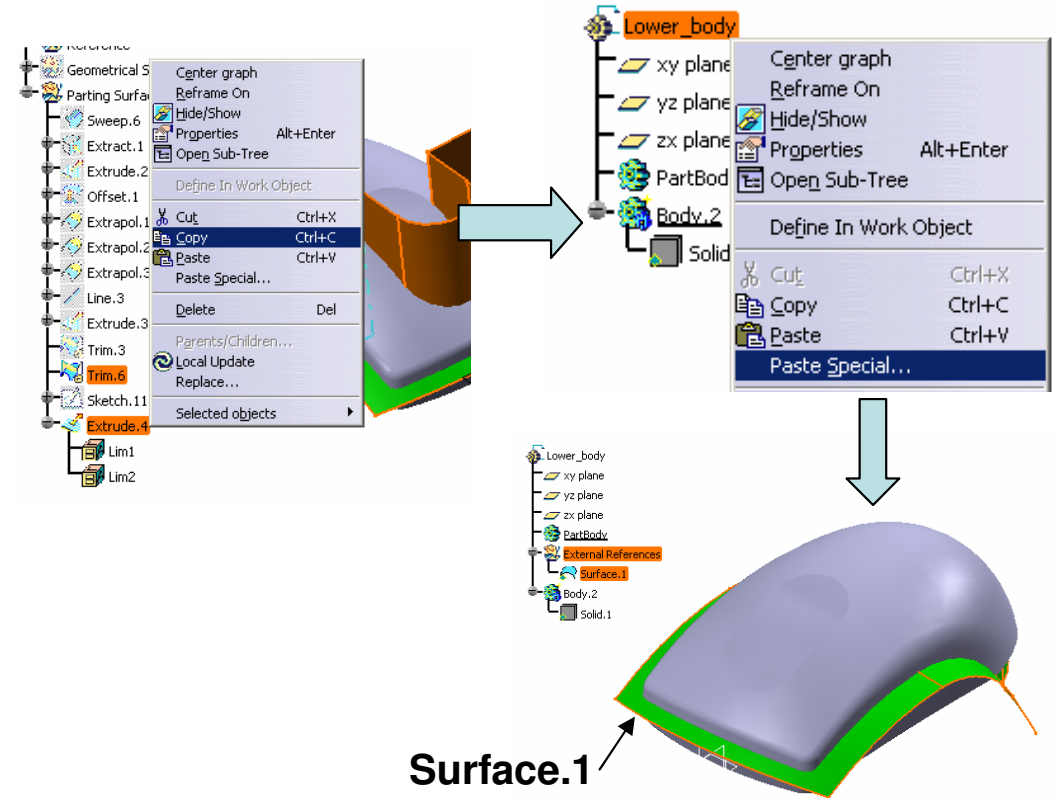
Tutorial 2C

To create the lower body:-

- Select File/New;

- Select Part as type;

- Enter Lower_body as part name;

- Click ok to complete.

- Select Window/Tile Vertically (we can see Mouse_Master & Lower Body at the same time)

- Right-click "PartBody" of Mouse_master_a.CatPart;

- and then select **"Copy"**;

- Right-click "Lower_body" of the tree of Lower_body and then select "**Paste Special…**"

- Select **"As Result with link";**

- Click ok to complete.

A- 79
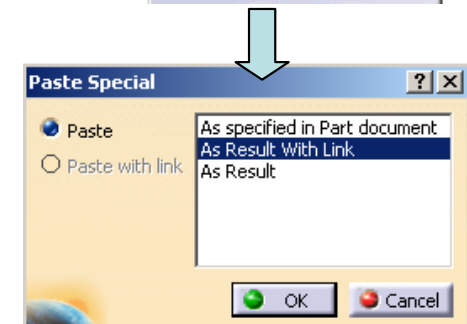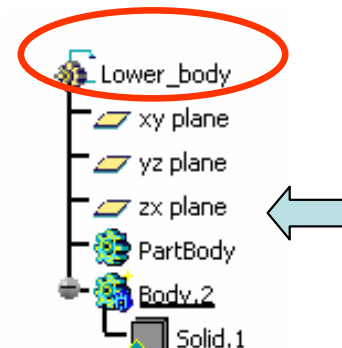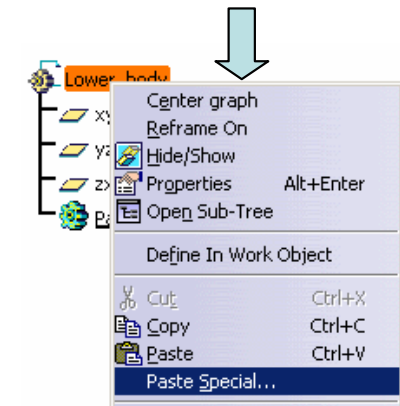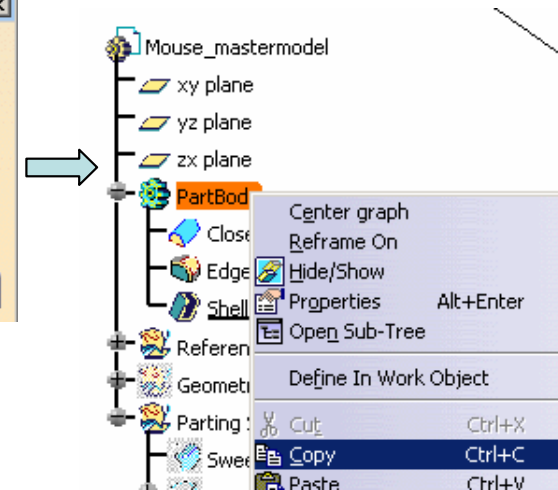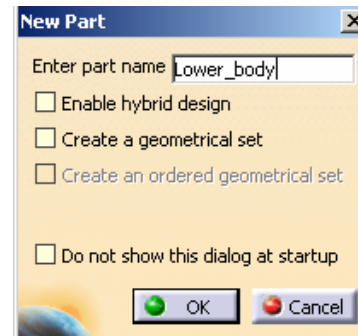
Tutorial 2C

- Select the parting surface "Trim.4""
- Right-click on either one and then select **"Copy"**

- Right-click "Lower_body" of the tree of Lower_body and then select **"Paste Special…"**
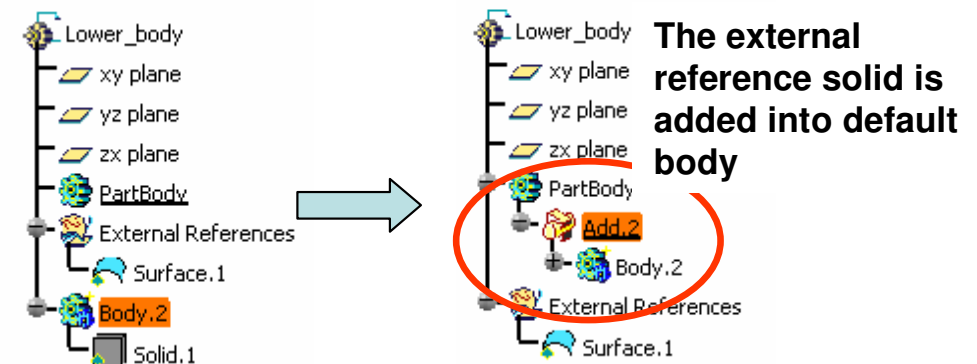- Select **"As Result with link"**
- Click ok to complete

Surface.1

- Select Start/Mechanical Design/Part Design
- Right-Click "PartBody" and then select "define in work object"
- Right-click "Body.2" and select **"Add"**

**The external reference solid is added into default body**

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Tutorial 2C

- Select Start/Shape/Generative Shape Design
- Click **"Offset"** icon
- Select "Surface.1"
- Enter **0.5mm** as value
- Click "Reverse Direction" icon if the red arrow is not pointing downward
- Click ok to complete

- Click **"Split"** icon
- Select Offset.1
- Click on the arrow once if it is not pointing download
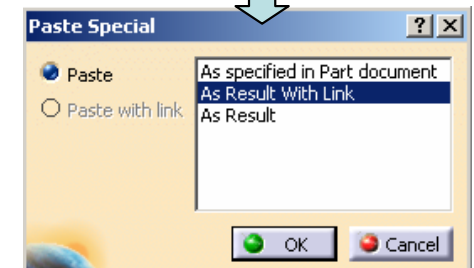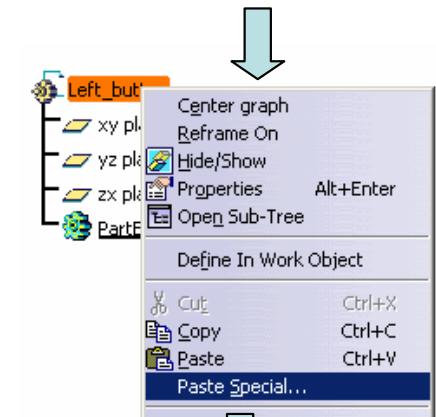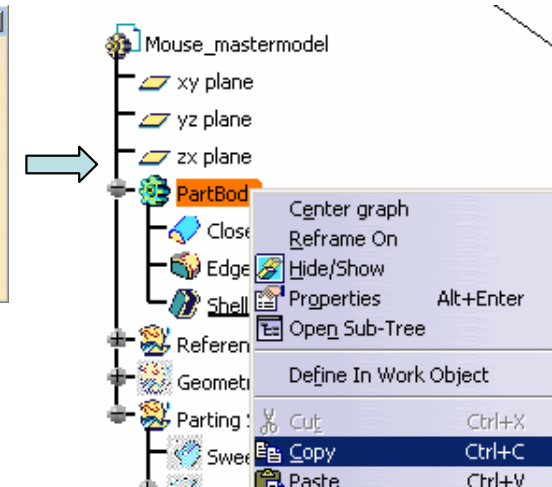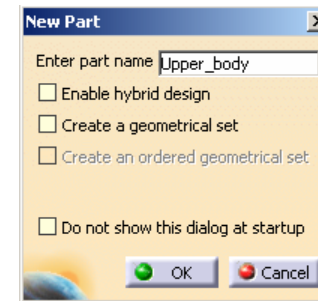- Click ok to complete
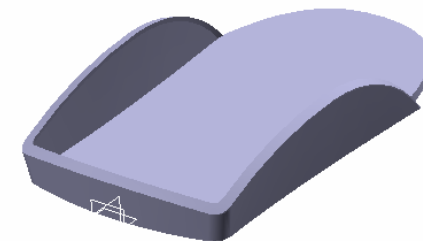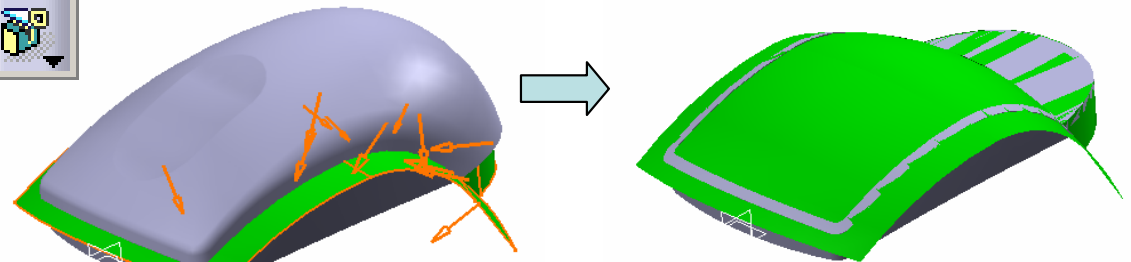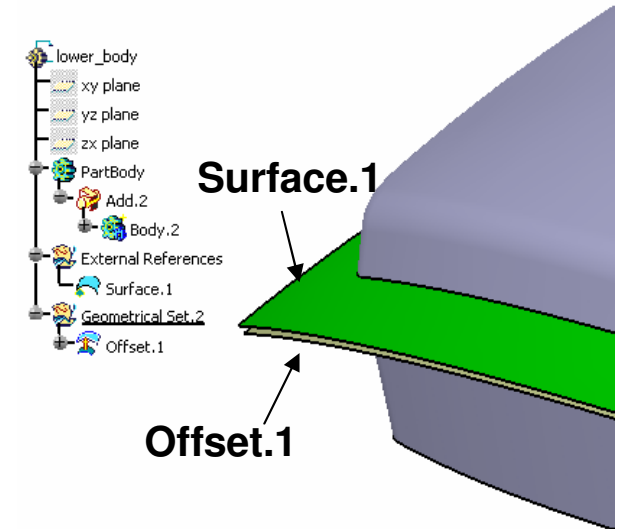
Hide External References & Geometrical set.2 on the tree

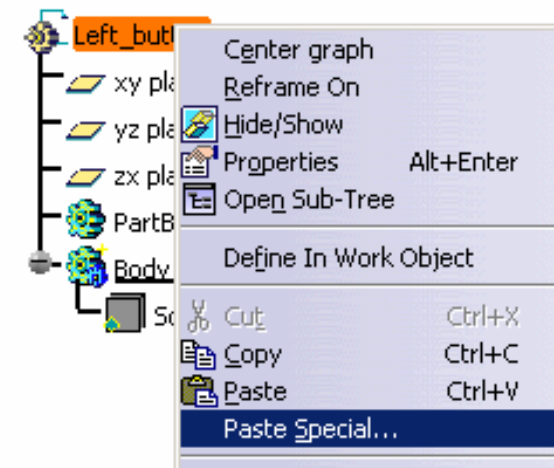**File/Save as Lower_body_a.Catpart Then File/Close**

A- 81

Tutorial 2C

To create the left button:-

- Select File/New;

- Select Part as type;

- Enter Left_Button as part name;

- Click ok to complete;

- Select Window/Tile Vertically (we can see Mouse_Master & Left_Button at the same time)

- Right-click "PartBody" of Mouse_master_a.CatPart;

- and then select **"Copy"**;

- Right-click "Left Button" of the tree of Left _Button and then select **"Paste Special…"**

- Select **"As Result with link"**;

- Click ok to complete.

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Tutorial 2C

- Multi-Select the two parting surfaces "Trim.4" & Extrude.4"

- Right-click on either one and then select **"Copy"**

- Right-click "Left_button" of the tree of Left Button and then select **"Paste Special…"**

- Select **"As Result with link"**

- Click ok to complete
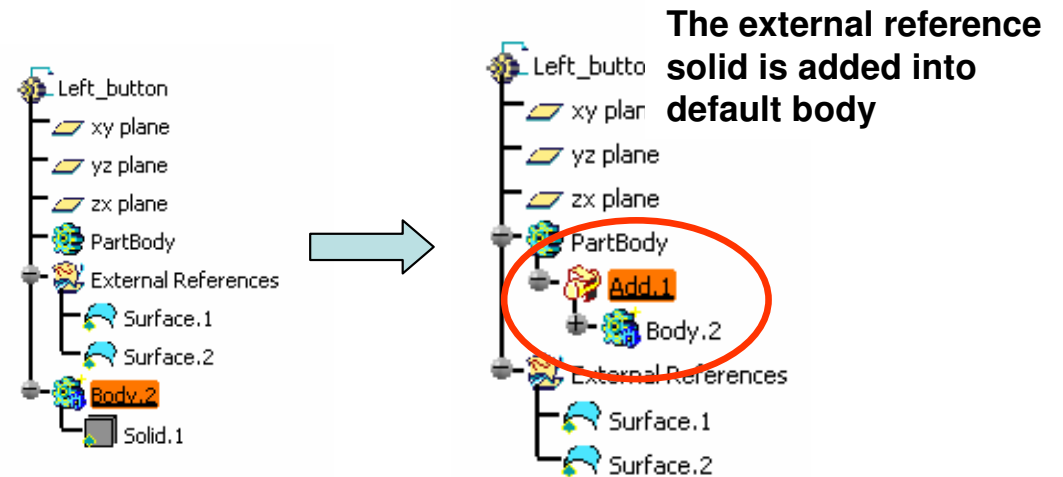
Surface.1

Surface.2

A- 83

Tutorial 2C

The external reference solid is added into default body

- Select Start/Mechanical Design/Part Design
- Right-Click "PartBody" and then select "define in work object"
- Right-click "Body.2" and select **"Add"**

- Click **"Split"** icon;
- Select Surface.1;
- Click on the arrow once if it is not pointing forward;
- Click ok to complete

- Similarly, Click **"Split"** icon again;
- Select Surface.2;
- Click on the arrow once if it is not pointing upward;
- Click ok to complete

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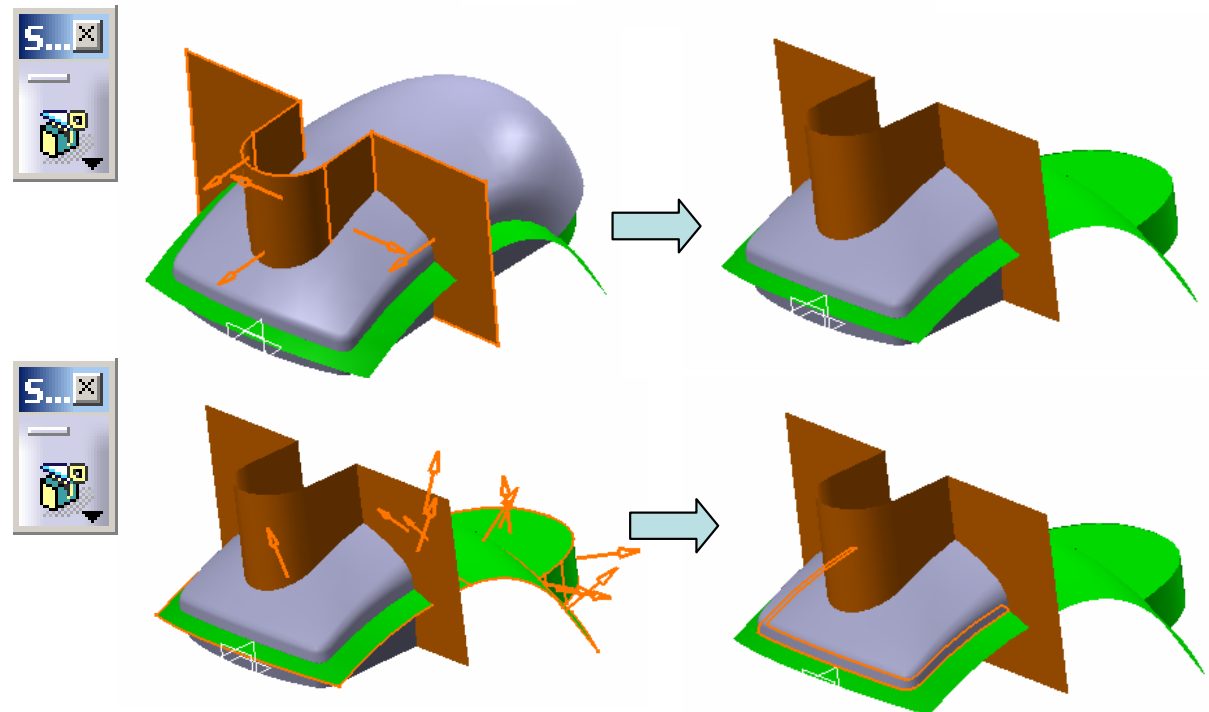
Tutorial 2C

- Click **"Plane"** icon
- Select yz plane
- Select Offset from plane as type
- Enter **9mm** as value
- Click ok to complete

- Click **"Split"** icon
- Select Plane.1
- Click on the arrow once if the direction is incorrect
- Click ok to complete
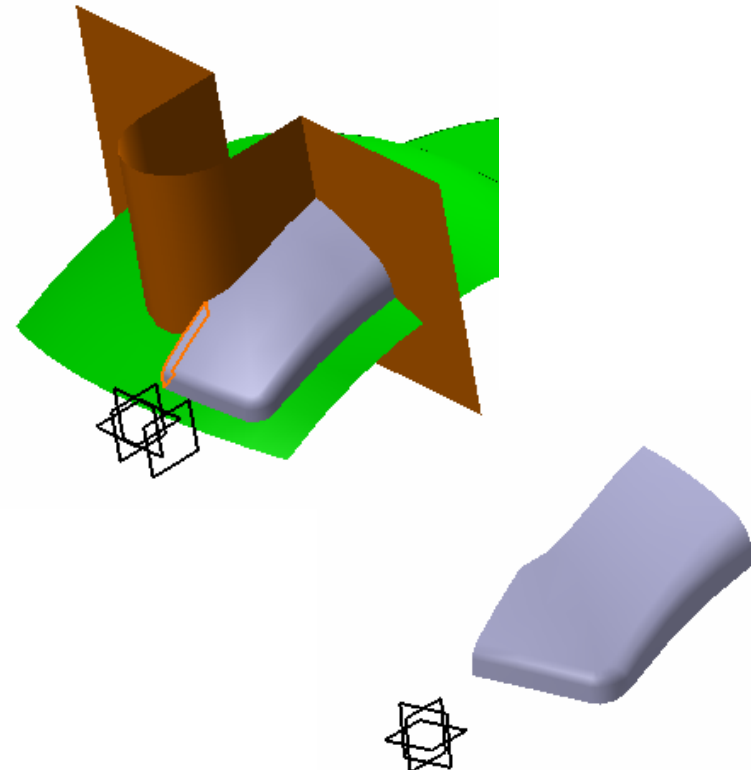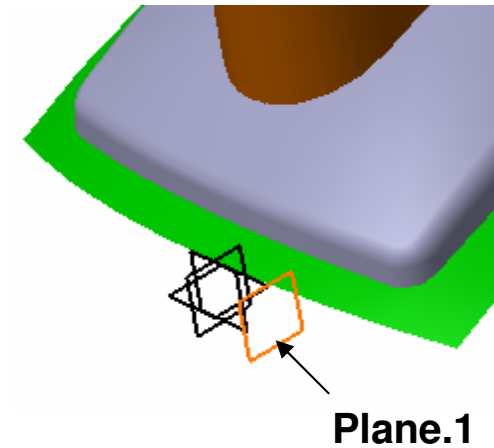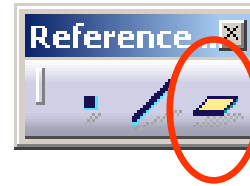
Plane.1

Hide External References & Geometrical set.2 on the tree

File/Save as Left_Button_a.Catpart Then File/Close

A- 85

Tutorial 2C

Similarly, create the Right button…

- Select File/New
- Select Part as type
- Enter Right_Button as part name
- :
- :
- Click **"Plane"** icon
- Select yz plane
- Select Offset from plane as type
- Enter **9mm** as value
- Click "Reverse Direction"
- Click ok to complete
- :

Hide External References &
Geometrical set.2 on the tree

File/Save as Right_Button_a.Catpart
Then File/Close

A- 86
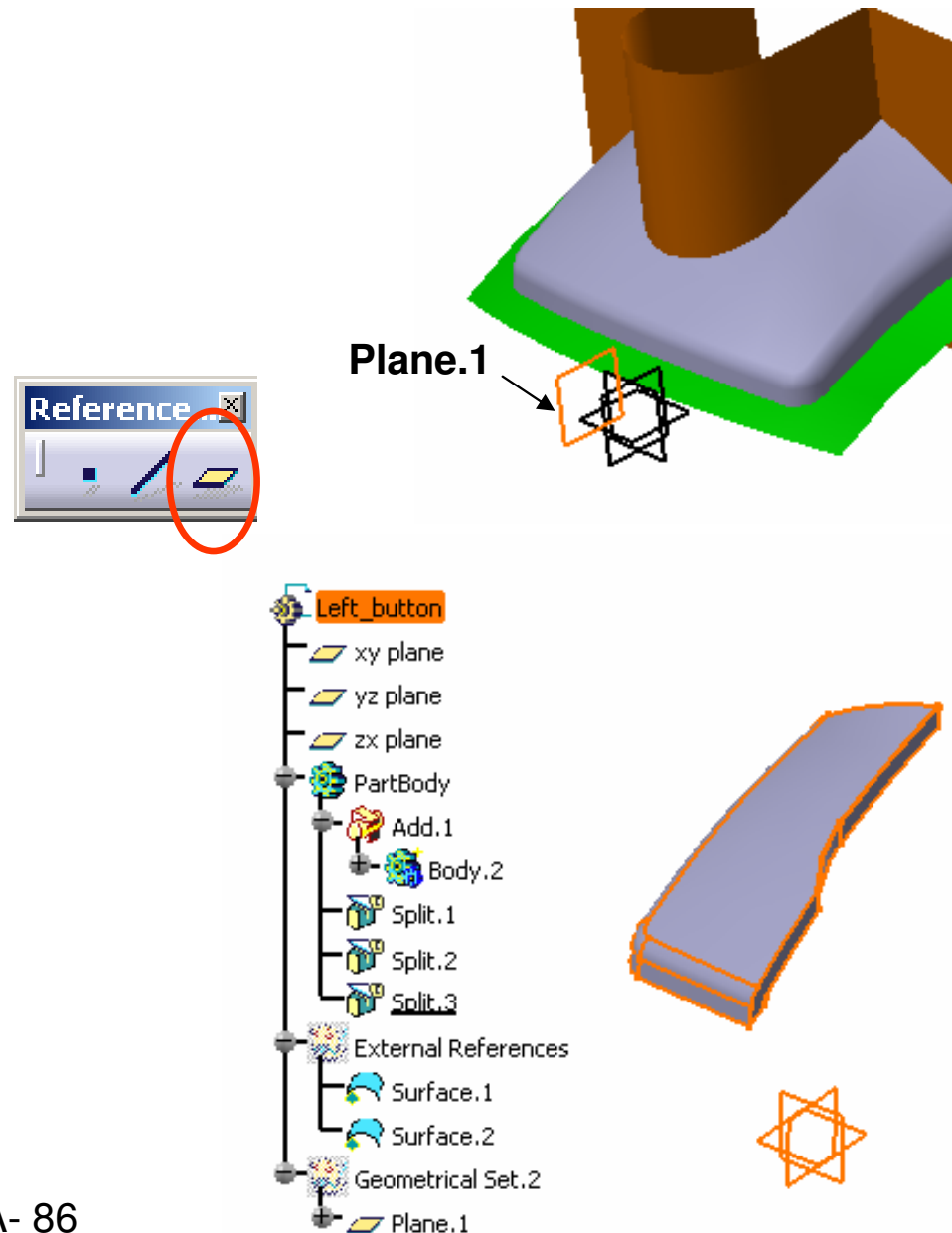
Tutorial 2C

Similarly, create the Middle button…

- Select File/New
- Select Part as type
- Enter Middle_Button as part name
- :
- :
- Click **"Plane"** icon;
- Select yz plane;
- Select Offset from plane as type;
- Enter **8.5mm** as value;
- Click ok to complete

- Click **"Plane"** icon again;
- Select yz plane;
- Select Offset from plane as type;
- Enter **8.5mm** as value;
- Click **"Reverse Direction"**;
- Click ok to complete

Reference

Plane.2

Plane1

Middle_button
- xy plane
- yz plane
- zx plane
- PartBody
  - Add.1
  - Body.2
  - Split.1
  - Split.2
- External References
  - Surface.1
  - Surface.2
- Geometrical Set.2
  - Plane.1
  - Plane.2

A- 87

Tutorial 2C

- Click **"Split"** icon;
- Select Plane.1;
- Click on the arrow once if the direction is incorrect;
- Click ok to complete

- Click **"Split"** icon again;
- Select Plane.2;
- Click on the arrow once if the direction is incorrect;
- Click ok to complete

Hide External References & Geometrical set.2 on the tree
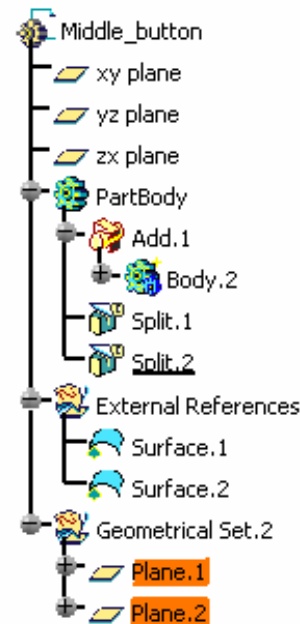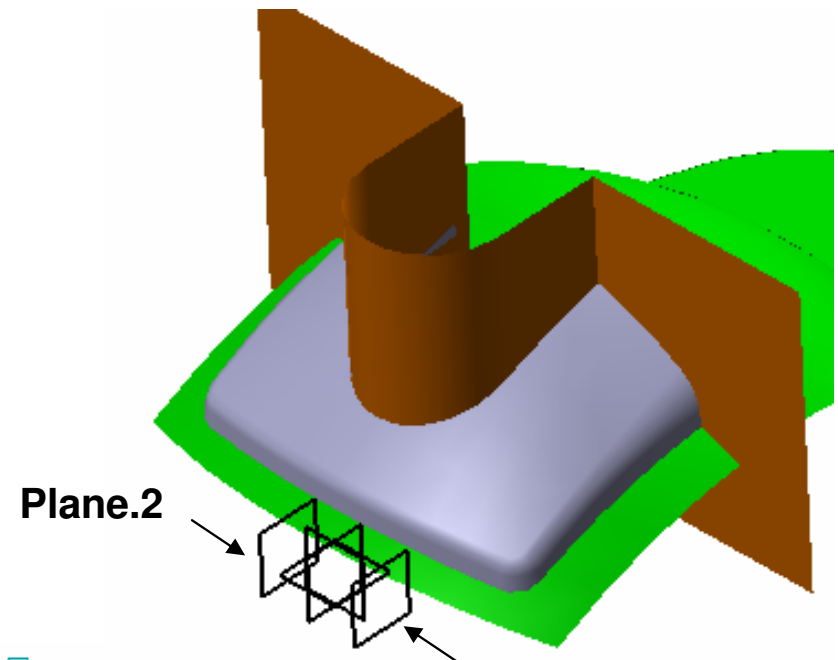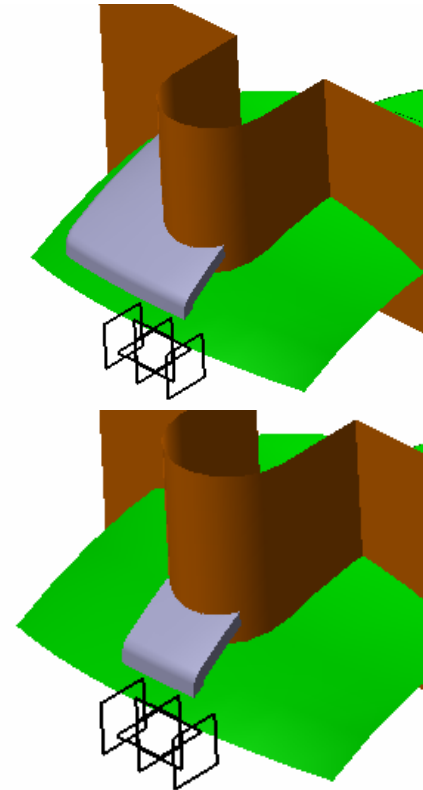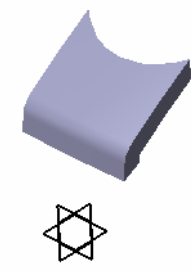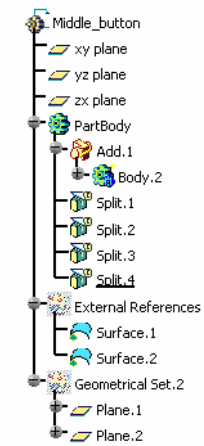
File/Save as Middle_Button_a.Catpart Then File/Close

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By Dickson Sham (ME dept, HKPU)

Tutorial 2C

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

Now we are going to assemble the parts into a product with other components, e.g. a scroll button.

- Close all files;
- Select **"Start/Mechanical Design/Assembly Design"** (A new Product will then be created);
- Right-Click "Product1" on the tree;
- Select "properties" and Select the tab page "Product";
- Enter **Mouse_assm** as part name;
- Click ok to confirm

A- 89

Tutorial 2C

CATproduct
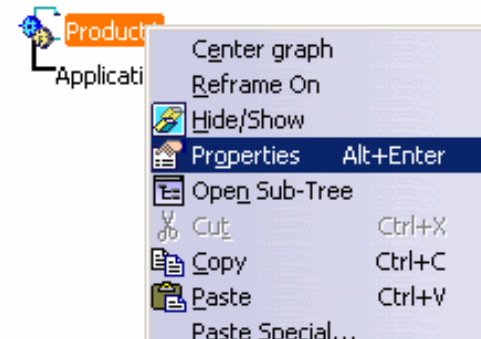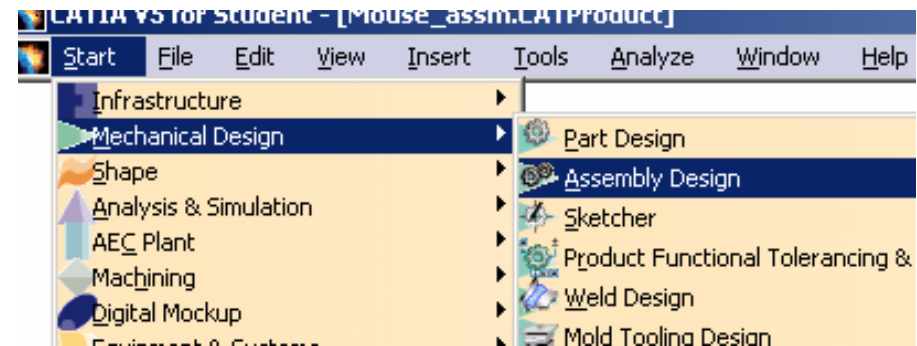
CATparts

- Right-Click "Mouse_assm" on the tree
- Select "Components/existing component"
- Multi-select the files:
  - Upper_body_a.CATpart
  - Lower_body_a.CATpart
  - Left_button_a.CATpart
  - Middle_button_a.CATpart
  - Right_button_a.CATpart
- Click Open
- (As the parts are created from the same master model at the same origin, they are located at the right places when inserted into a product.)
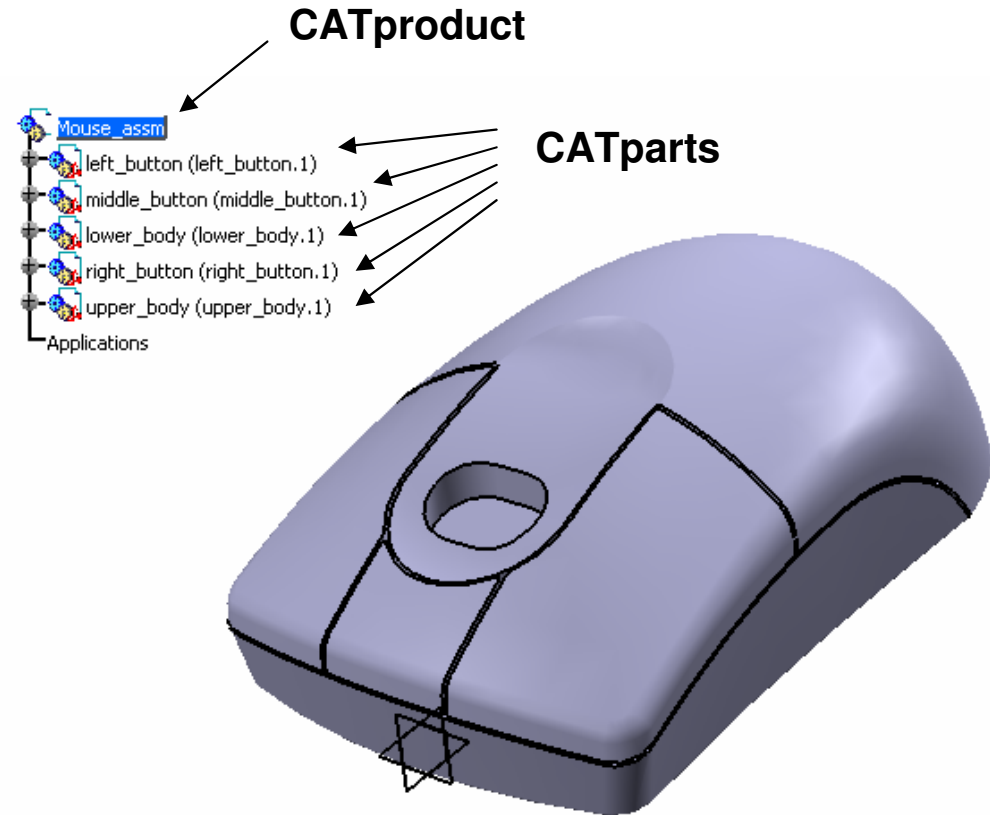
Download the part

"Scroll_button_a.CATpart" from the web:

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

A- 90
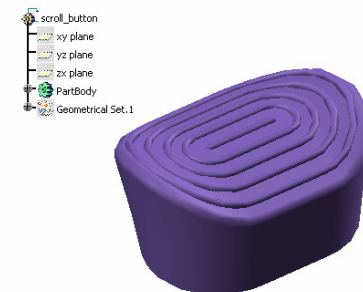
Tutorial 2C

- Right-Click "Mouse_assm" on the tree
- Select "Components/existing component"
- Select the files:
  - Scroll_button_a.CATpart

- Drag the red dot of the compass and drop it onto the Scroll_button (The compass will turn into green and the scroll_button will be highlighted on the tree)

- Use the compass to move the Scroll button on the top of the pocket of the upper_body (Remark: the scroll button is not symmetric, the side with a hole should be closer to the center of the mouse)

compass

A small hole

pocket

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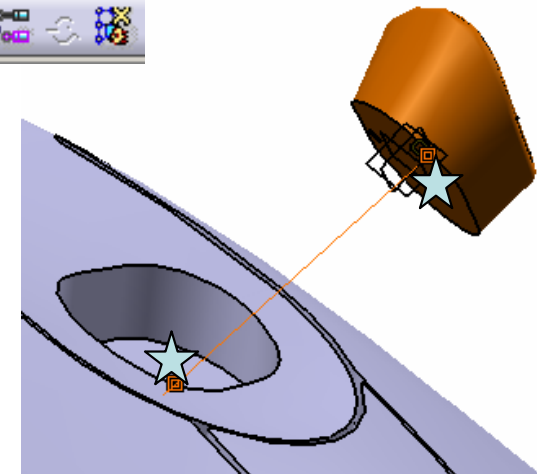By Dickson Sham (ME dept, HKPU)
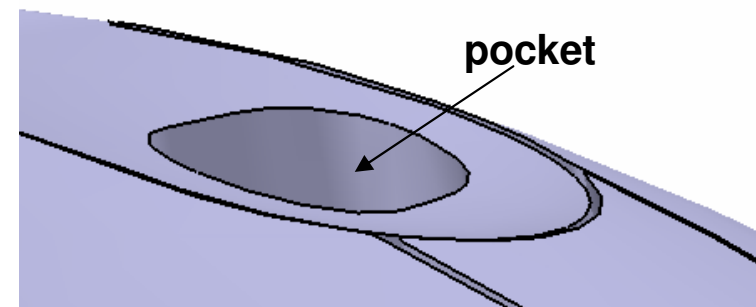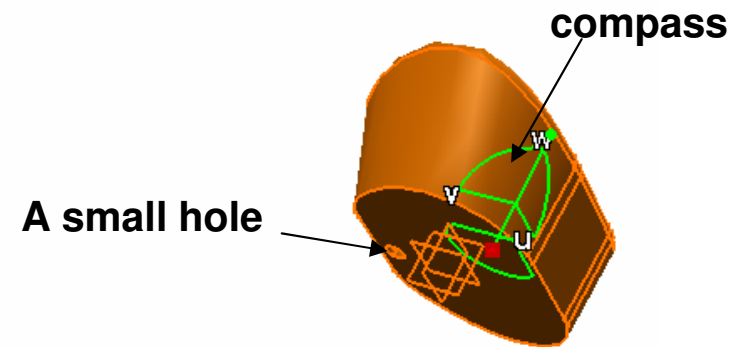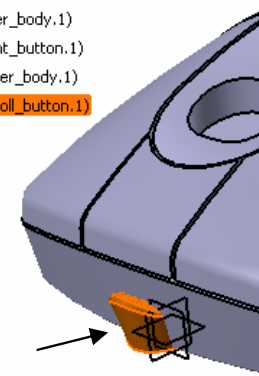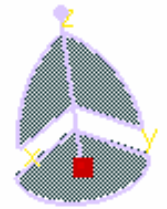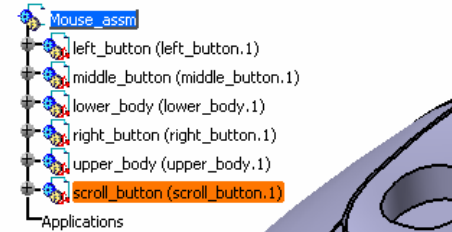
Tutorial 2C

- Click **"Coincidence"** icon;
- Select yz plane of Scroll Button;
- Select zx plane of Upper_body;
- Click ok to complete

- Click **"Contact"** icon;
- Select bottom face ☆ of Scroll Button
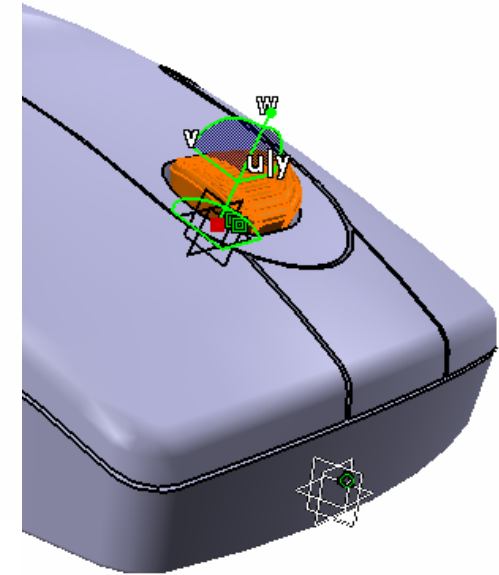- Select bottom face ☆ of the pocket of Upper_body
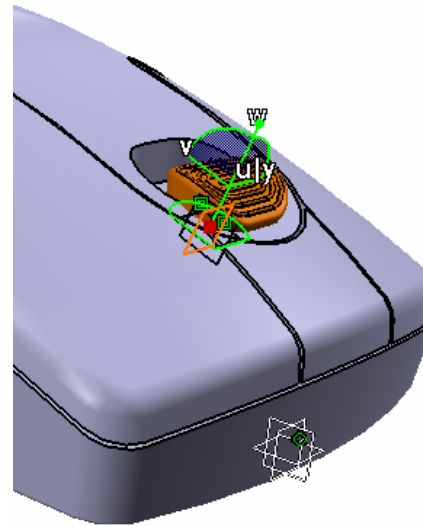- Click ok to complete

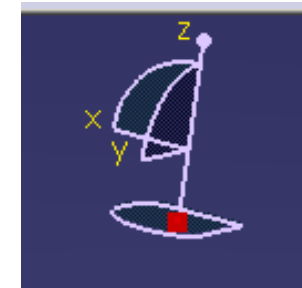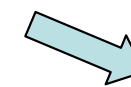- Click "**Update**" icon to update its position

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Tutorial 2C

- Drag the scroll_button to the center of the pocket by using the Compass;
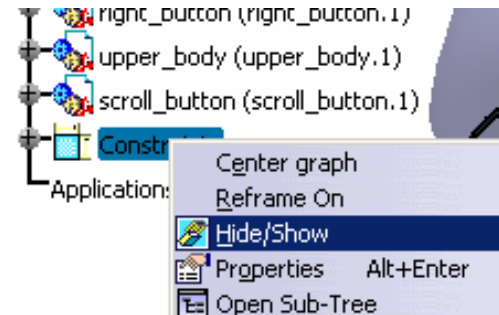
- Drag the red dot of the compass onto the global coordinate system at the lower righthand corner of the screen and then release;

- (Now, the compass is reset to original)

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Tutorial 2C

Hide the assembly constraints from the screen. (Right-click "Constraints" on the tree and then select "Hide/Show")

Select all datum planes and Click "Hide/Show" icon to hide them all

Hide all planes
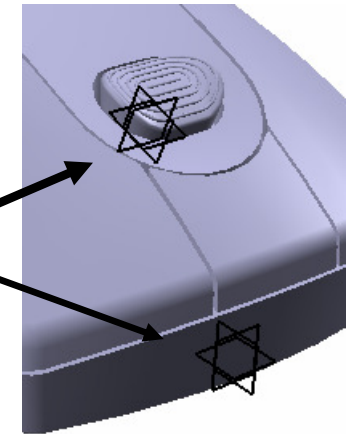
To add material properties onto the parts:-

- Double_click Left_button tree (now the workbench should be switched to Part Design)
- Click "**Apply Material**" icon;
- Select "Plastic" from the catalog;
- Select "Left_button" on the tree again;
- Click ok to confirm

Double –click the left_button tree to activate this part

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Tutorial 2C

- Right-click "Plastic" on the tree and select Properties;
- Select the tab page "Rendering";
- Change the setting as:
  - Ambient 0.9 (Color R80, G80, B80)
  - Diffuse 0.4 (Color R80, G80, B80)
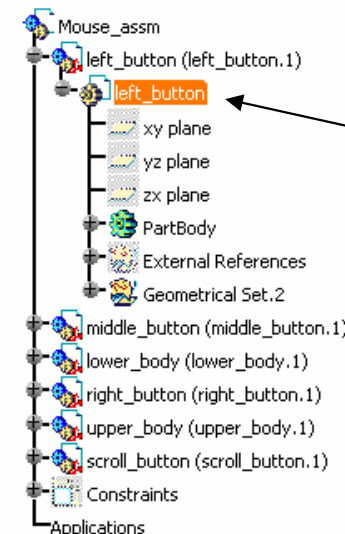 - Specular 0.6
  - Roughness 0.6
  - Transparency 0.0
  - Refraction 1.0
  - Reflectivity 0.0
- Click ok to complete

- Right-click "Plastic" on tree and select **"copy"**;
- Right-click "Middle-button" tree and select **"paste";**

- **Repeat the copy-and-paste steps onto other components except the scroll button.**

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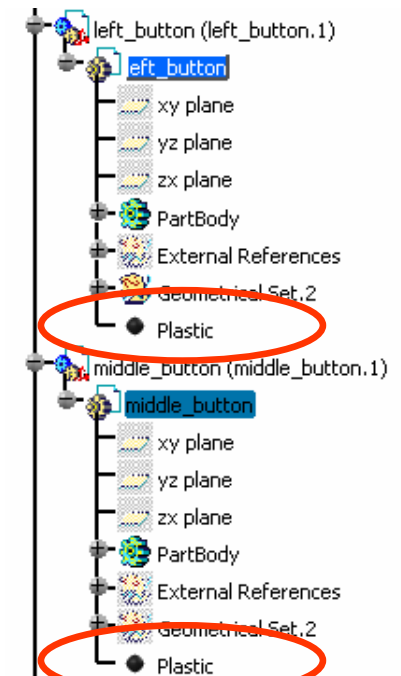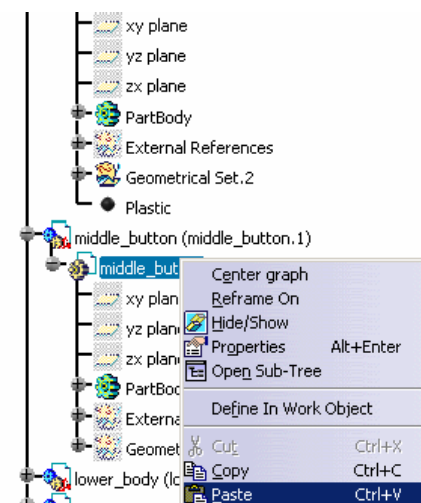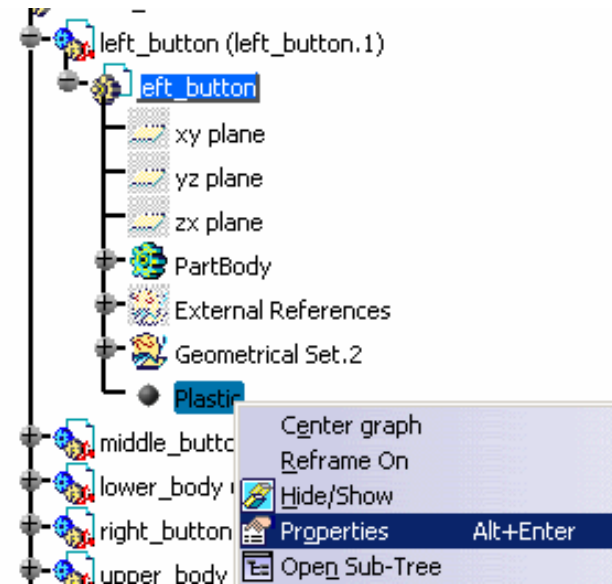
Tutorial 2C

- Click **"Shading with Material"** icon;

- (We should see the mouse with material rendering as shown)

- File/Save all

- Select Ok on the pop-up message box

- Select "Save as…"

- Enter "**Mouse_assm_a.CATproduct**" as file name

- Select Save and then click ok to complete

View

Save All

Following document(s) cannot be automatically saved:
\Mouse_assm.CATProduct
Do you want to proceed ?

OK   Cancel

Mouse_assm
left_button (left_button.1)
middle_button (middle_button.1)
lower_body (lower_body.1)
right_button (right_button.1)
upper_body (upper_body.1)
scroll_button (scroll_button.1)
Constraints
Applications

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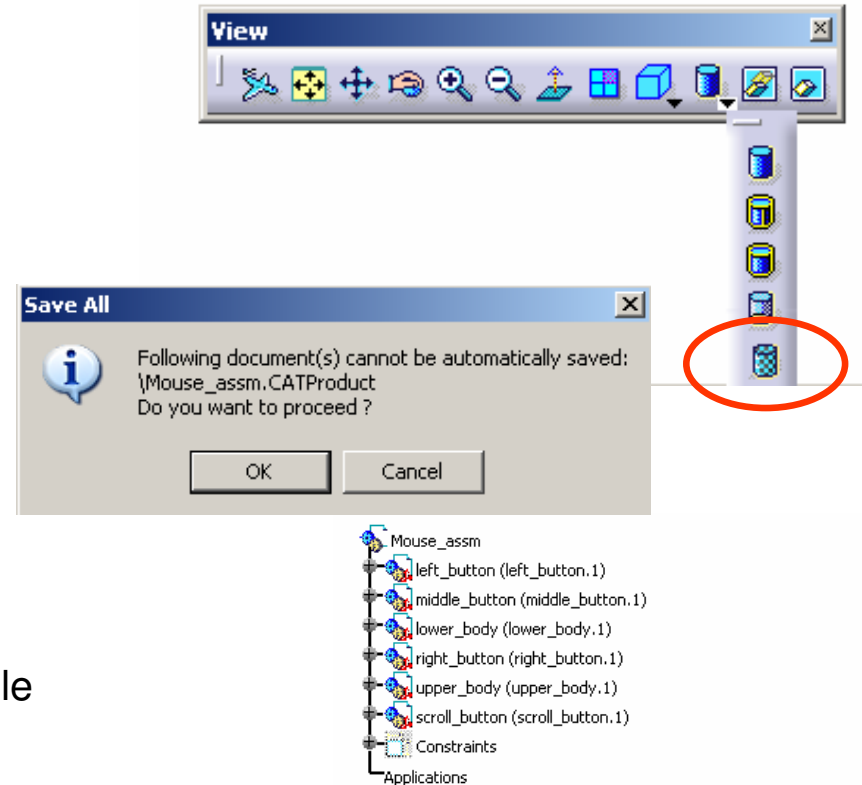
Tutorial 2C

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

MasterModel

Paste with link

Paste with link

Paste with link

Paste with link

Paste with link

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

Scroll Button

PCB

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Signal Cord

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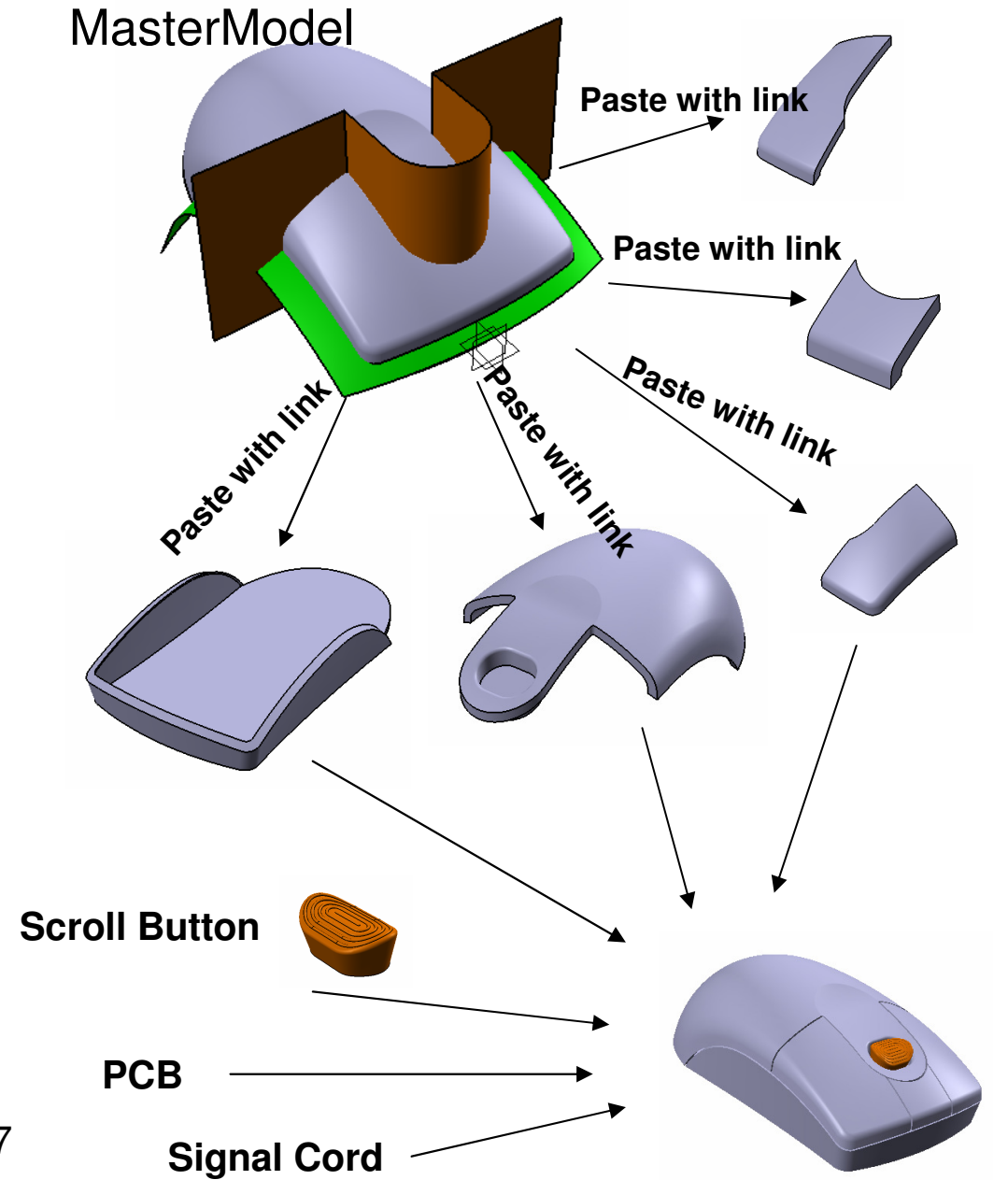
Tutorial 2C

- • File/Open... Mouse_mastermodel_a.CATpart

- • Show Geometrical Set.2
- • Double-Click "Sketch.7" to modify
- • **Change "37mm" to "40mm"**
- • Click 'Exit" icon to exit
- • (Now the mastermodel will be updated)
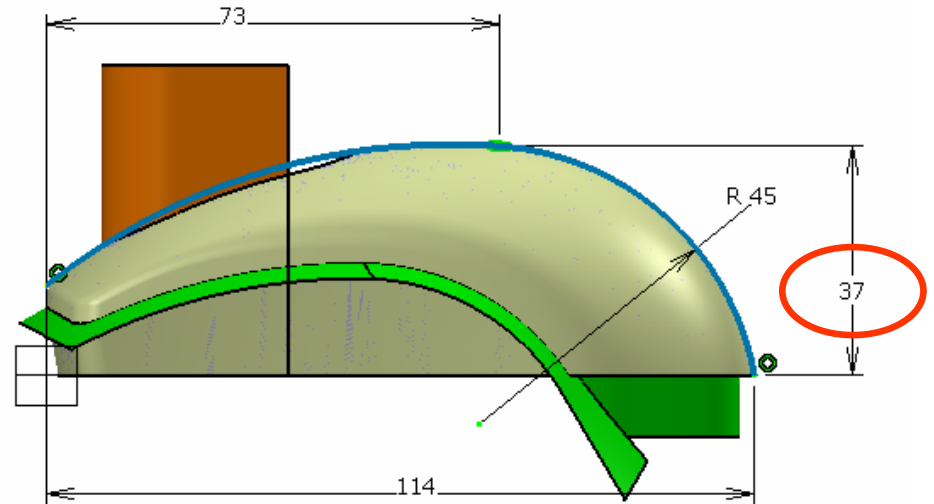
- • Select Window/Mouse_assm_a.CATProduct
- • (All parts are now turned in RED, except the scroll button; only linked parts are RED)
- • Click "Update" icon to update all

OLD
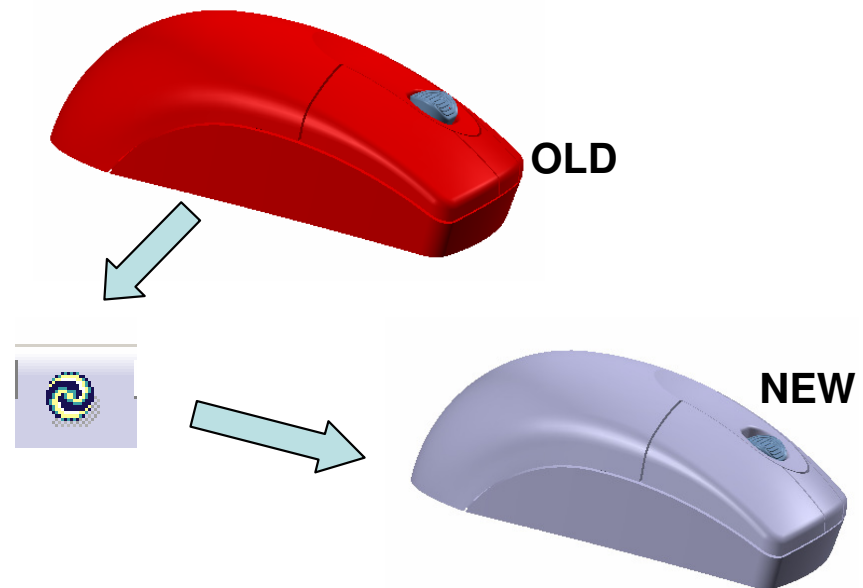
- • **(Wait for a few seconds, all parts will be updated automatically!)**

NEW

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END of Tutorial 2C

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