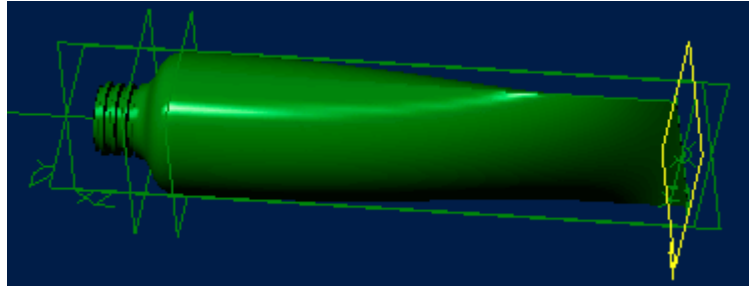


Alibre Design Tutorial: Loft, Extrude, & Revolve Cut Loft-Tube-1



Part Tutorial

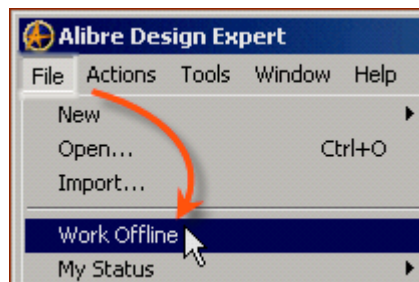
Exercise 5: Loft-Tube-1 [Complete]

In this Exercise, We will set System Parameters first, then part options. Then, in sketch mode, we will set the first three sketches of the tube for the loft, and loft them. Then we will use the Extrude Feature to create the Neck, or - threaded top, as a blank. Then we will use revolve cut to show simplified threads and hollow out the top neck. Setting the Color Properties and Saving will follow this.

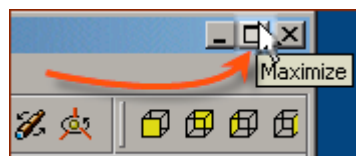
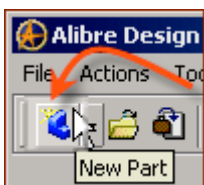
Launch **Alibre Design**.

From the Home window - select **File, Work Offline**. (You may switch to work online later.)

(Also - Even though these pictures show 'Alibre Design Expert' - these are basic instructions that can be done without Professional or Expert Versions.)

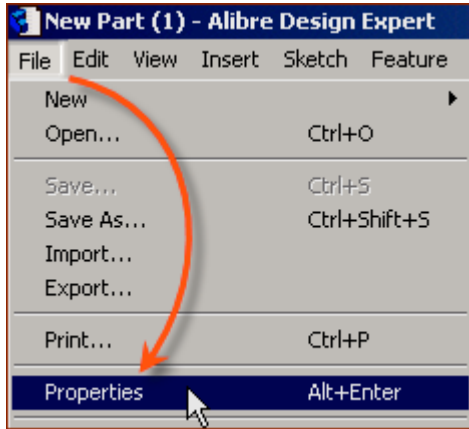


Open a **New Part** workspace. From the Home window - click on the **New Part** icon. Maximise the Part Workspace Window.



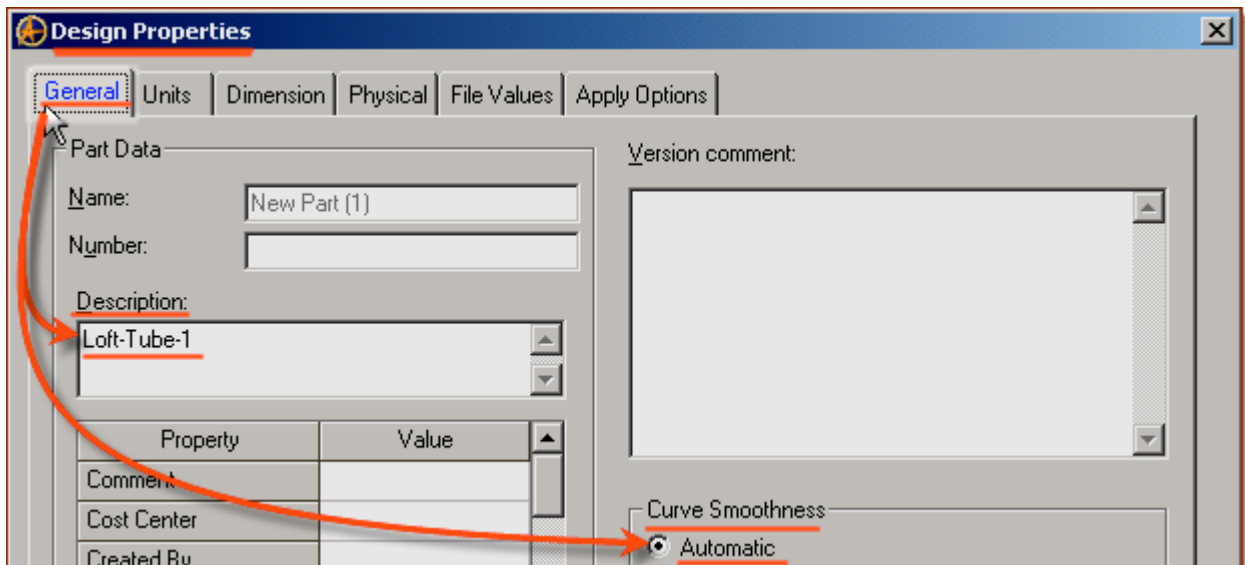
If at any time you are not sure of a keyboard item, see: [Keyboard Basics](#)
Set Design Properties.

Select **File, Properties**.



A) Select the Tab Marked "**General**"

- In **Description**: insert the information: 'Loft-Tube-1'.
- In **Curve Smoothness**, Select the Radio Button marked 'Automatic'.

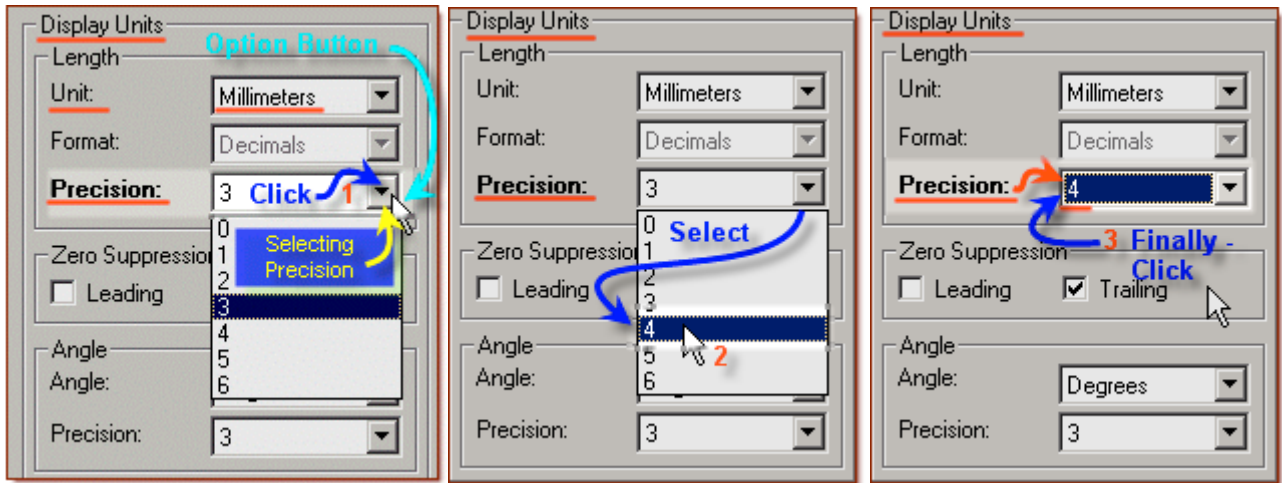


B) Select the Tab Marked "**Units**"

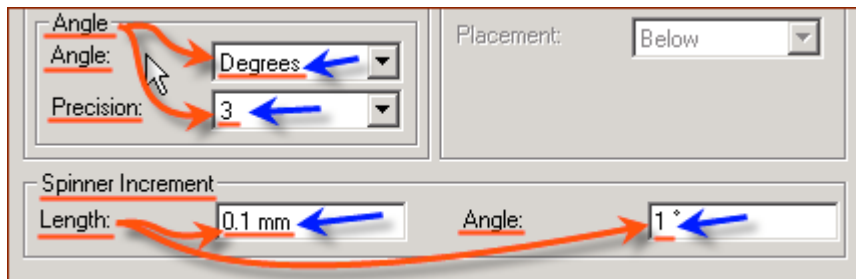
- Check (check mark in box is showing) the Box **Show Units for Dimensions**.



- On **Display Units** - confirm **Unit: Millimeters**, and **Precision: 4**.
If Not, Click option button, Select '4', Click to accept.

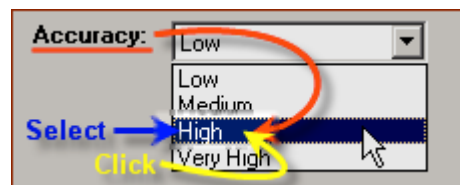
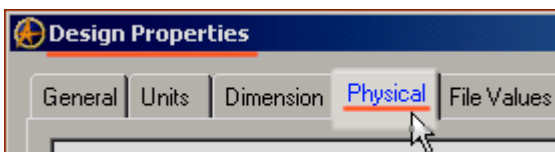


- On **Angle** - Confirm **Angle: Degrees** and **Precision: 3**. Adjust if not.
- On **Spinner Increment** - for **Length:** enter 0.1mm and for **Angle:** enter 1 degrees. Adjust to these parameters if not. Press 'Tab' to move after editing them.



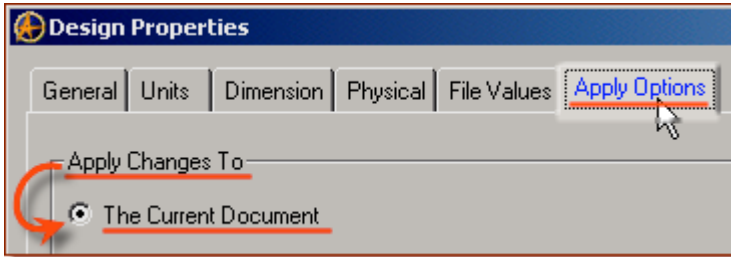
C) Select the Tab Marked "Physical"

- Change **Accuracy:** to **High**. Click on the **Calculate** Button. Notice the results.



D) Select the Tab Marked "Apply Options"

- On **Apply Changes to** - Click the Radio Button beside **The Current Document**.
- Click **Apply**, then Click **Close**.

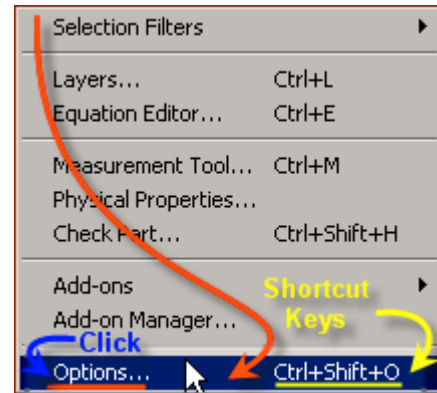
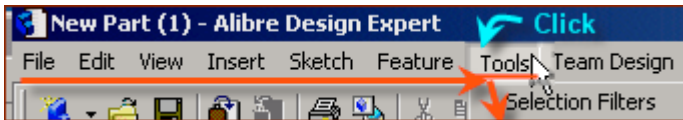


1) Click Apply 2) Click Close



Set Part Options.

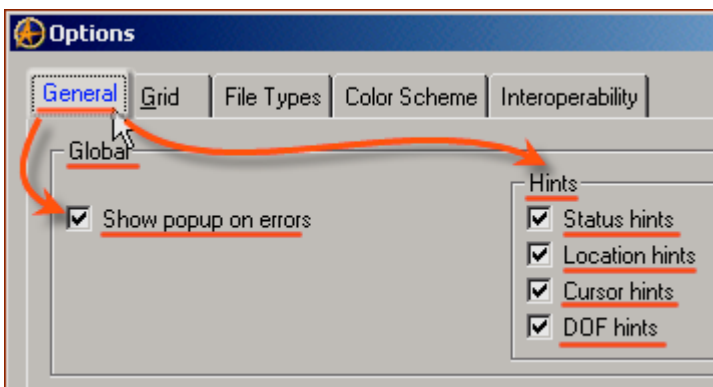
From Top Text Menu - Select - **Tools > Options > (Ctrl-Shift-O)**



General Tab > Selected or checked off (*check off = check mark in box*) are:

Global - Show popup on errors,

Hints - All

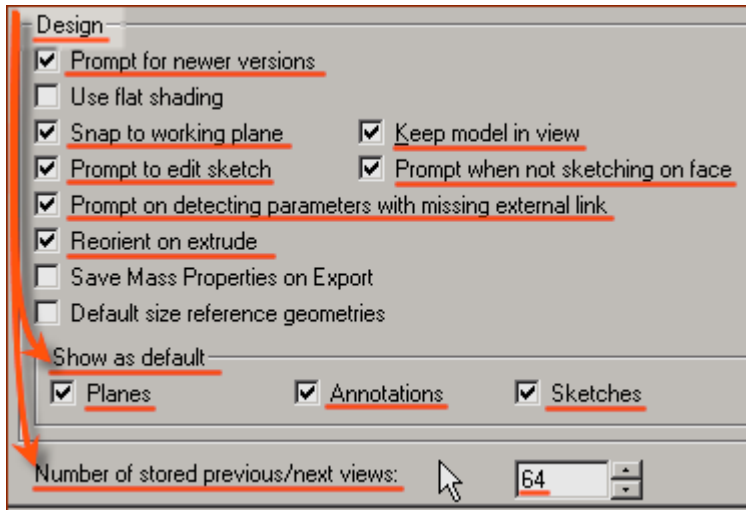


Design

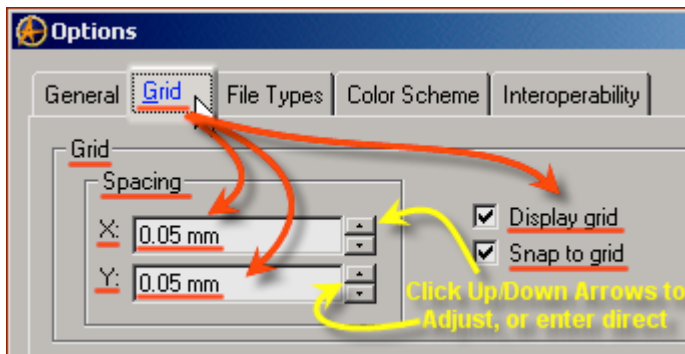
- Prompt for newer versions, - Snap to working plane, - Prompt to edit sketch,
- Prompt on detecting parameters with missing external link, - Reorient on extrude,
- Keep model in View, - Prompt when not sketching on face.

Show as Default - Planes, Annotations, and Sketches.

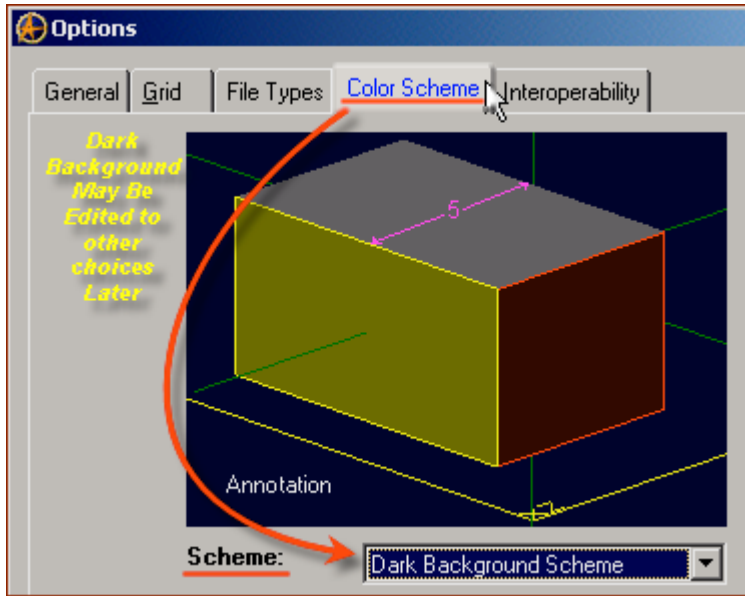
Number of stored previous/next views: 64



Grid Tab > Spacing: Make X: 0.05 mm and Y: 0.05 mm, check off **Display grid**, and **Snap to grid**.



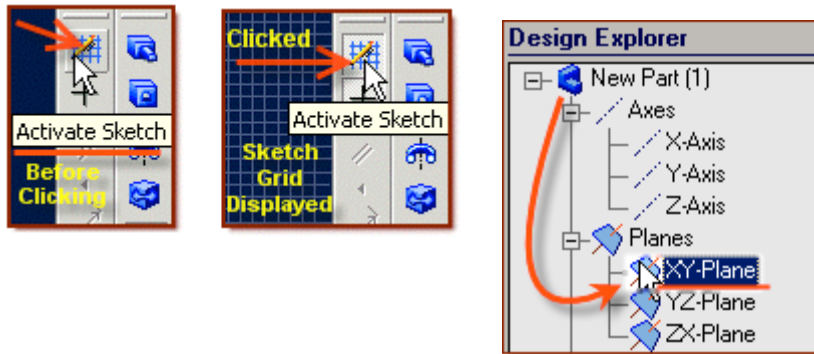
Color Scheme Tab> For **Scheme**: - confirm - Dark Background Scheme. Click '**OK**'.
(You may come back to this at any time later - and edit - or change - any element.)



Proceed to Starting the Sketches, Next Page.

Start the First Loft Sketch.

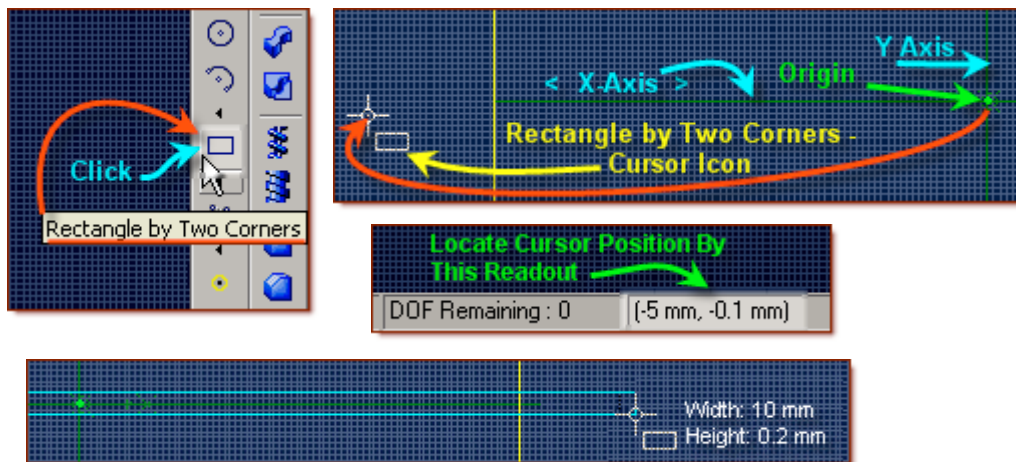
From the Sketch Icons > Click **Activate Sketch**. (*The XY-Plane should be selected.*)



From the Sketch Icons > Click **Rectangle by Two Corners**

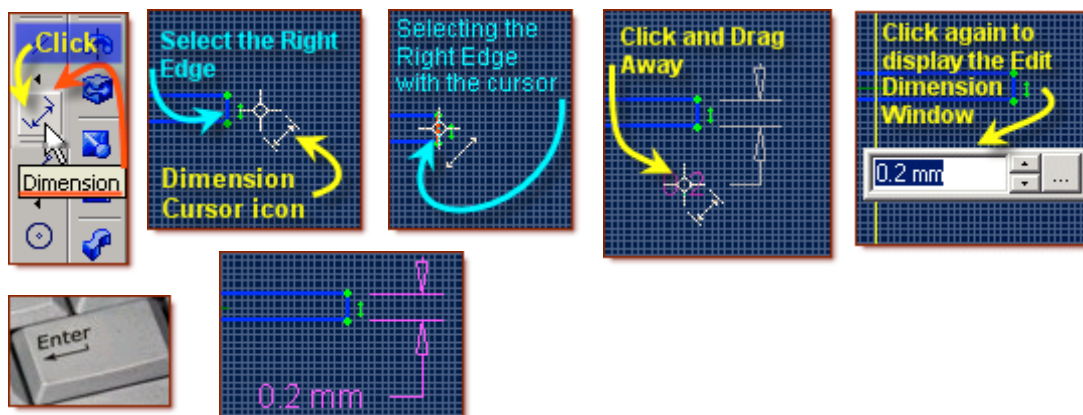
- Balanced above and below the Horizontal axis (X-Axis), draw a rectangle, approximately centered on the Vertical axis (Y-Axis) 10 mm wide by 0.2 mm tall.

Locate Cursor by the Readout on the Status Bar.

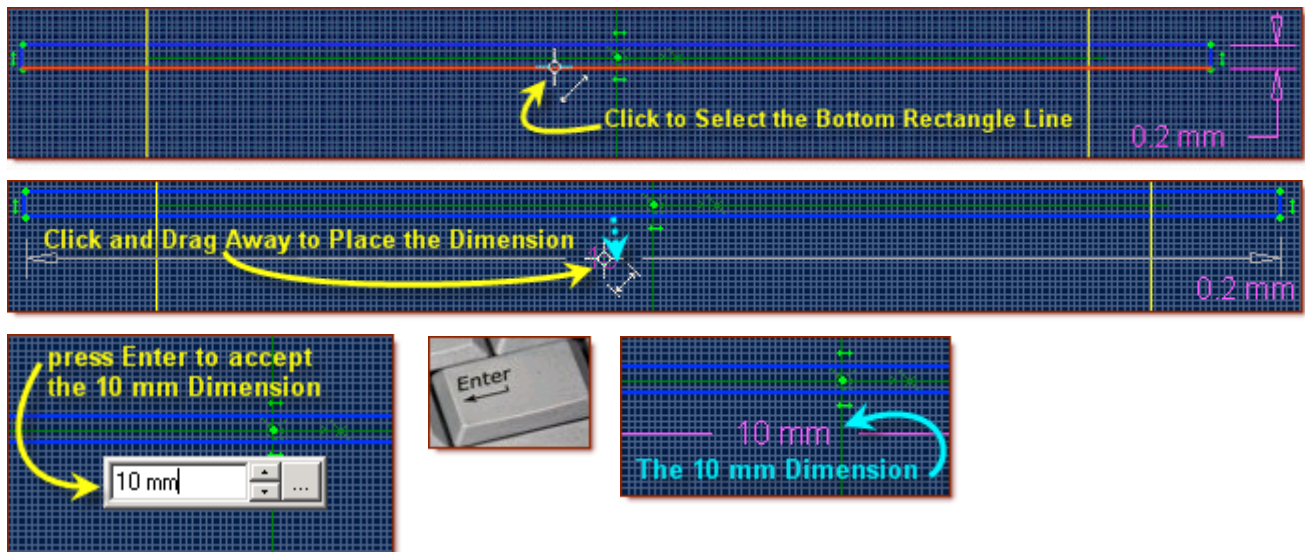


From the Sketch Icons > Click **Dimension** (*Add the dimensions to the rectangle*)

- Select the right side edge line, (Click, and drag away to place the 0.2 mm dimension.)
- Click to display the Edit Dimension window, press Enter to accept the 0.2 mm Dimension.



- Select the bottom rectangle line, Click, and drag away to place the 10 mm dimension.
- Click to display the Edit Dimension window, press Enter to accept the 10 mm Dimension.

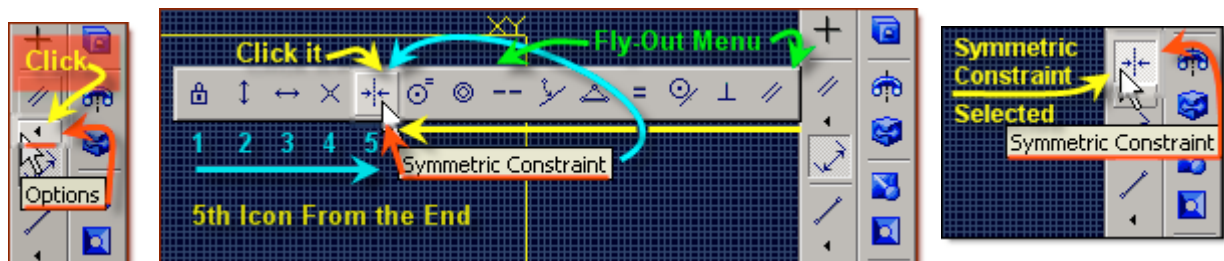


If the Dimension is not these numbers, simply enter these values in the Edit Dimension Box before pressing Enter.

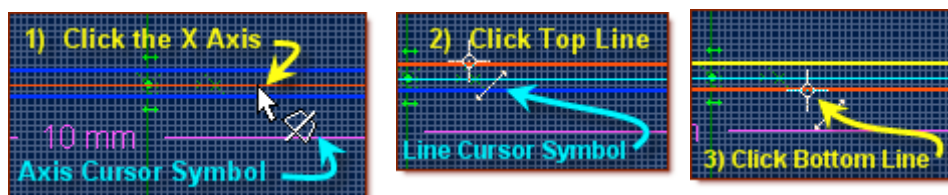
From the Sketch Icons > Click Options (Under the Constraint Icons)

Select the **Symmetric Constraint** by Clicking it.

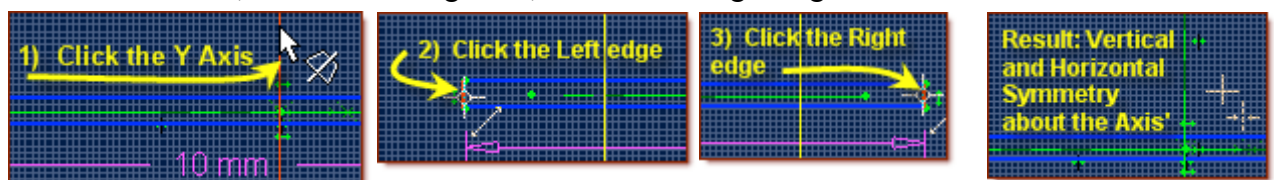
(Pass the mouse slowly over the constraints and a name tip will appear, Symmetric Constraint is 5th icon from the end of the flyout menu)



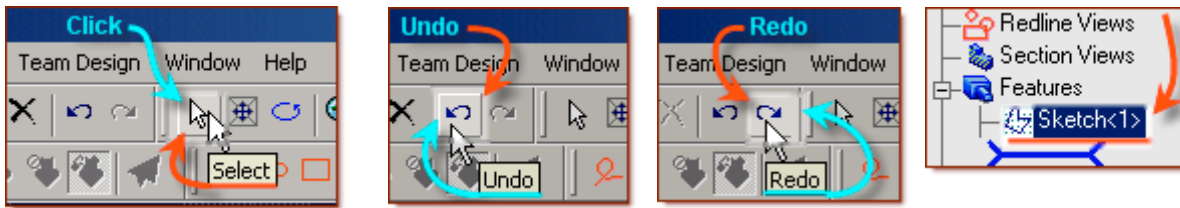
- Use the Symmetric constraint to make the top and bottom lines Symmetric about the X-Axis, and the left and right lines Symmetric about the Y-Axis.
- Click the X-Axis, then the top line, and then the bottom line.



- Click the Y-Axis, then the left edge line, and then the right edge line.



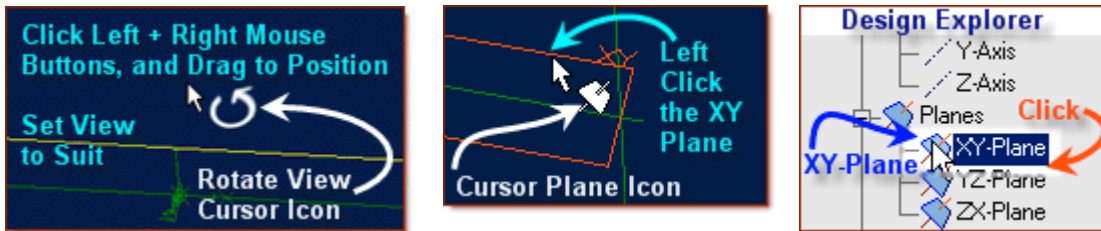
From the top Icons, click on **Select** (Beside Undo and Redo Icons) to exit sketch mode. This creates a new Sketch<1> Displayed in the Design Explorer.



Start the Second Loft Sketch.

First - Position the Viewing area to see the plane, Insert a new plane to place the next sketch on.

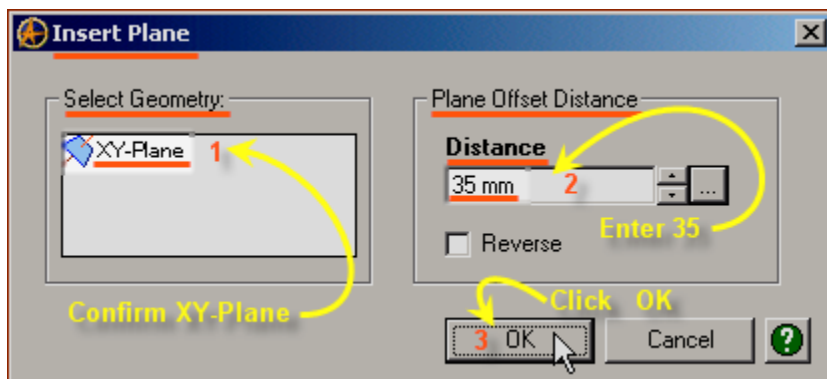
- Using a combination of left and right mouse buttons depressed together, rotate the viewing area.
- Left Click the **XY-Plane** in the workspace or in the Design Explorer.

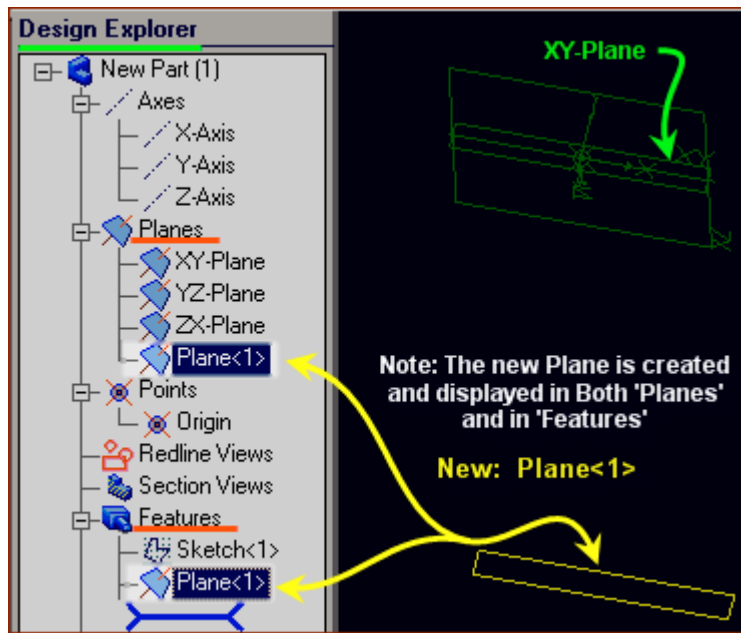


- Then, Select from the Main Menu - **Insert Click > Plane. Click.**



- In the **Insert Plane** popup, confirm the **XY-Plane** is displayed in the **Select Geometry** window, or else re-click on it now.
- For the **Plane Offset Distance**, Set **Distance** to **35 mm** (Highlight '0', enter 35, Press Tab)
- Click **OK**. **Plane<1>** is created, and displayed in the Design Explorer as well as the Workspace.



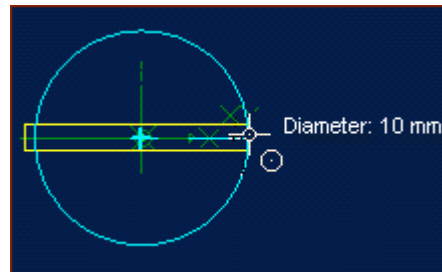
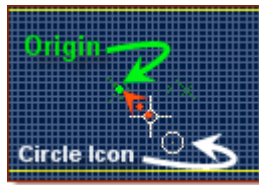


Next - Create the Sketch for the upper part of the Lofted Tube. (*Select Plane<1>*)

- Click the new Plane (*Plane<1>*) and from the *Sketch Icons*, Click *Select*.



- After the Sketch Workspace opens, from the *Sketch Icons*, click on *Circle*.
- Click on the *Origin*, and create a *10 mm* Diameter Circle. Click.

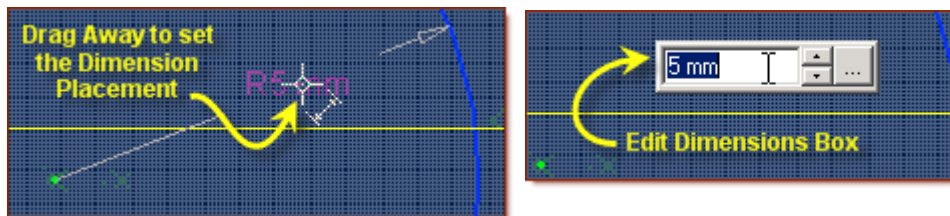


From the Sketch Icons > Click **Dimension** (*Add the dimensions to the Circle*)

- Click on the **Origin**, and **Shift-Click** on the **Circle**

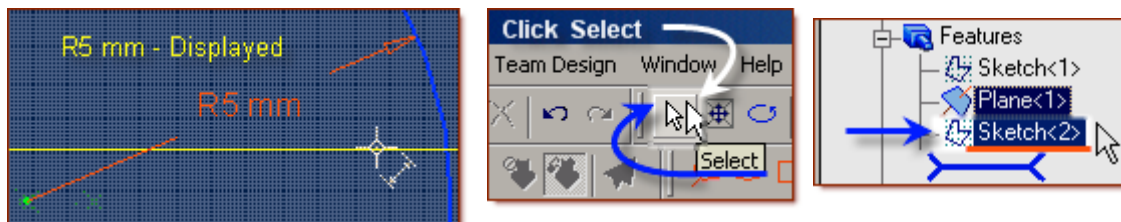


- Drag away to set the Dimension Place, **Click** to open the Dimension Edit Box.
- Press **Enter** to accept the dimension (5 mm).



- R5 mm is displayed with an arrow pointing to the circle.

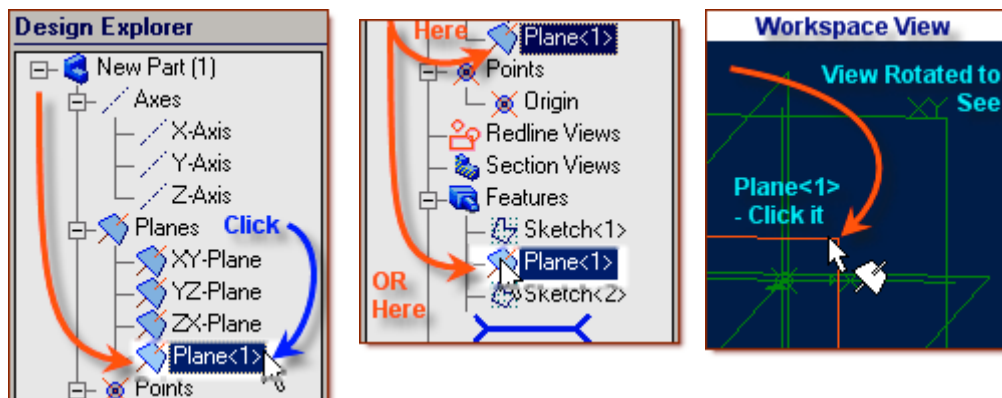
From the top Icons, click on **Select** (*Beside Undo and Redo Icons*) to exit sketch mode. Sketch<2> is now Displayed under Features in Design Explorer.



Start the Third Loft Sketch.

First - Insert a new plane to place the next sketch on.

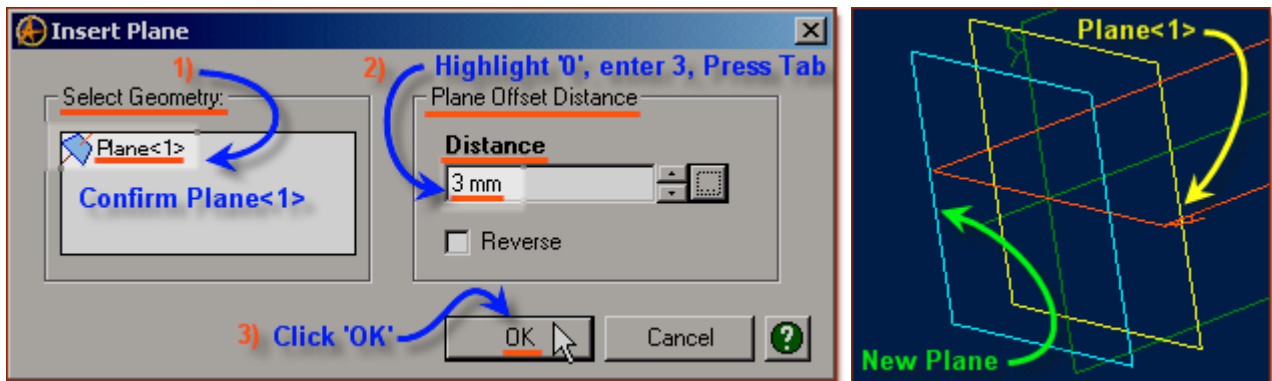
- Left Click **Plane<1>** in the workspace or in the Design Explorer. (*Rotate View To See It*)



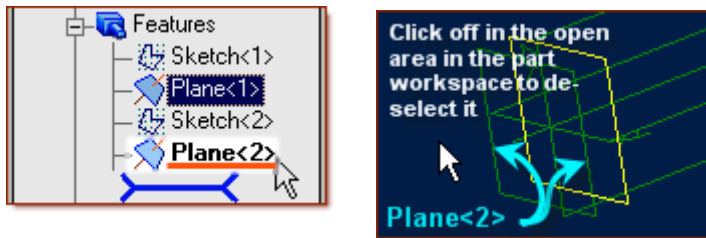
- Then, Select from the Main Menu - **Insert** **Click** > **Plane**. **Click**.



- In the **Insert Plane** popup, confirm **Plane<1>** is displayed in the **Select Geometry** window; or else re-click on it now.
- For the **Plane Offset Distance**, Set **Distance** to **3 mm** (*press Tab*)
- The new plane is displayed - showing in light blue where it will be if created.

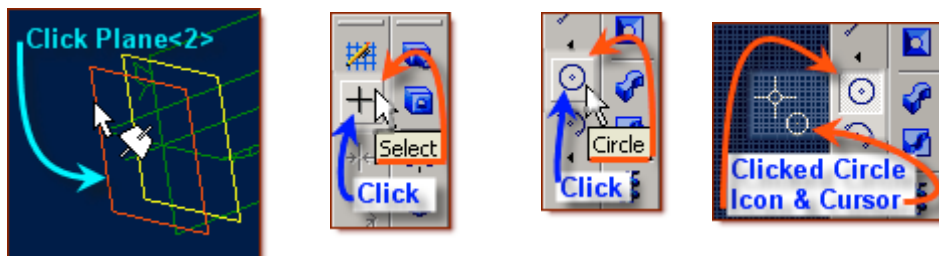


- Click **OK**. **Plane<2>** is created. Click off in the open area in the part workspace to de-select it.

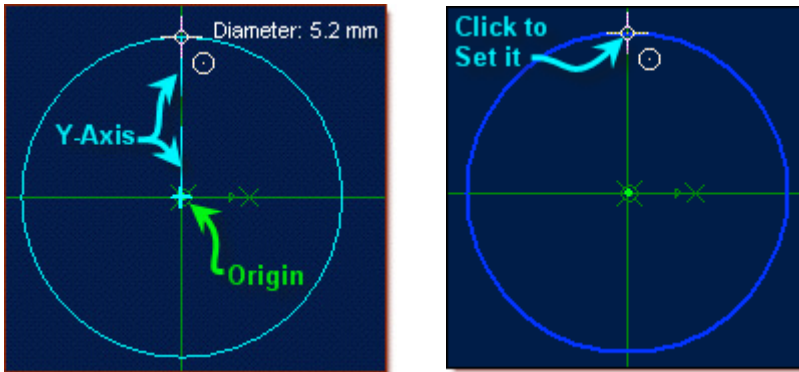


Next - Create the Sketch for the upper part of the Lofted Tube. (*Select Plane<2>*)

- Click the new Plane (**Plane<2>**) and from the **Sketch Icons**, Click **Select**.
- After the Sketch Workspace opens, from the **Sketch Icons**, click on **Circle**.

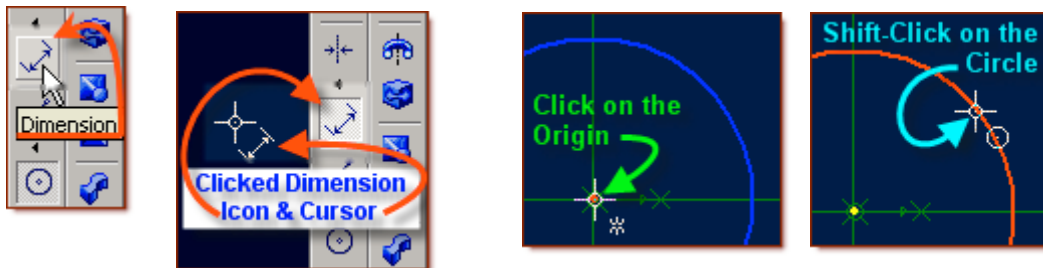


- Click on the **Origin**, and drag the cursor out along the **Y-Axis** to create a **5.2 mm** Diameter Circle.
- Click to set it.



From the Sketch Icons > Click **Dimension** (*Add the dimensions to the Circle*)

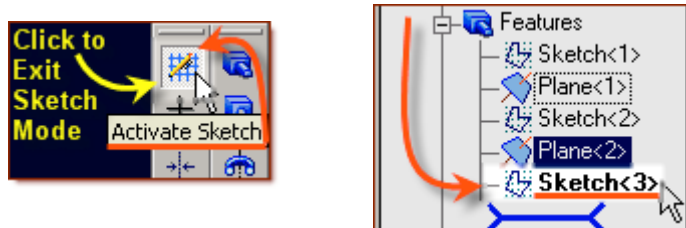
- Click on the **Origin**, and Shift-Click on the **circle**,



- Drag away and click to set the Dimension Place,
- Press **Enter** to accept the dimension (2.6 mm),
- R2.6 mm is displayed with an arrow pointing to the circle.

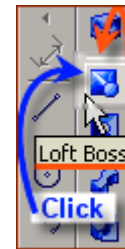
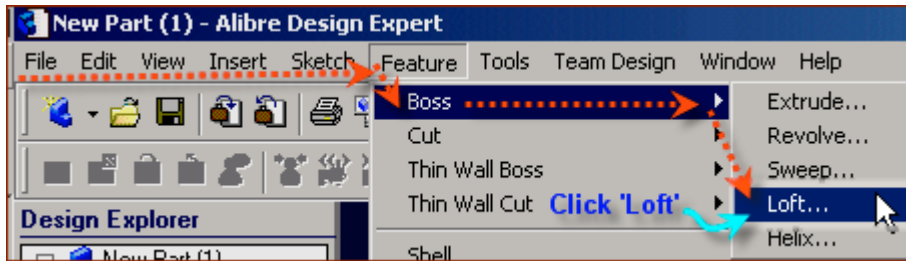


From the Sketch Icons, click on **Activate Sketch** (*Top Icon*) to exit sketch mode. **Sketch<3>** is created.

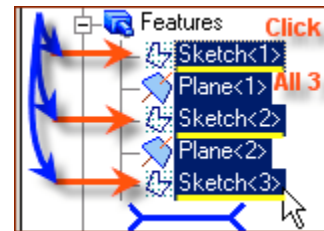
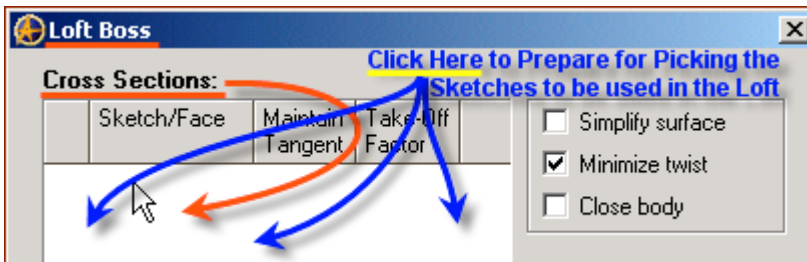


Create the Loft

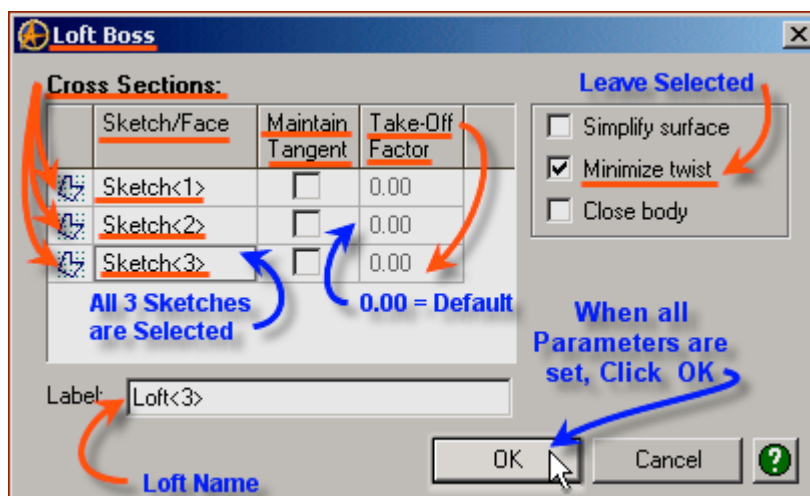
From the main menu, Click **Feature** > select **Boss** > **Loft**.
(Or Click **Loft Boss** from the **Feature Icons**)



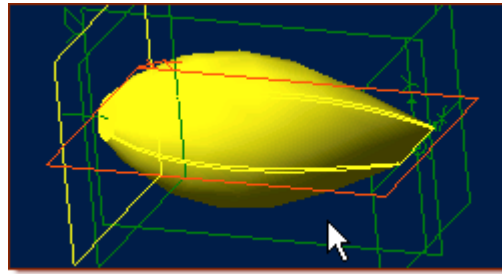
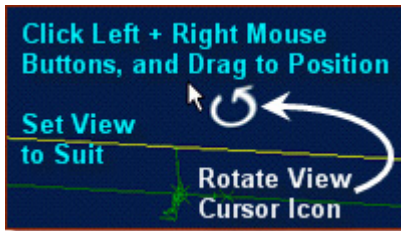
- From the **Loft Boss** popup, click in the White area under **Cross Sections** to prepare it for picking
- In the **Design Explorer**, click on **Sketch<1>**, then Shift-Click on **Sketch<2>**, and **Sketch<3>**



- You will see the three sketch's you have selected appear under **Sketch/Face**
- A selection box for each will appear under **Maintain Tangent**
- Under **Take-Off Factor** it will now show 0.00.
- On the right, **Minimize twist** is currently selected. Leave it that way.
- In the selection area beside **Label:** it is defaulting to **Loft<3>**. This is fine for now.
- Click **OK**. Your new lofted tube will be displayed (highlighted in yellow, initially)



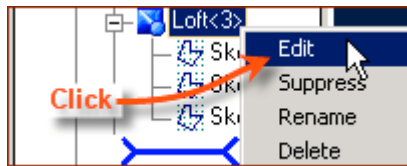
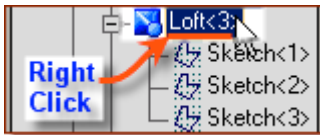
- Using a combination of left and right mouse buttons depressed together, rotate the viewing area. **You can see the result looks nothing like a Toothpaste or Make-up Squeeze Tube as it is!**



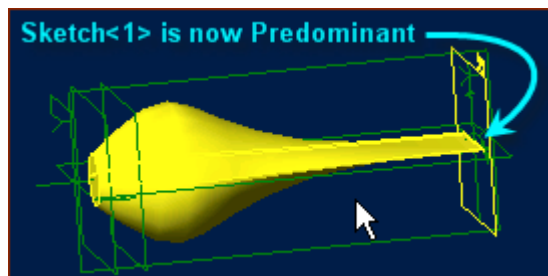
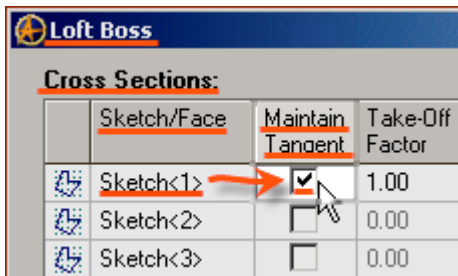
So - we will now edit the **Tangent** and **Take-Off** factors in the loft boss selection popup. First, we will just select the **Tangent**, and look at what the default value of 1.00 gives us. To see what is happening, we will select Tangent one at a time, for sketches 1 - 3.

Edit the Loft: Right-Click on Loft<3> in the Design Explorer

- Click on **Edit** in the drop down menu



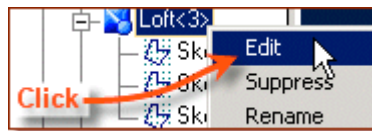
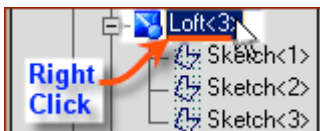
- From the **Loft Boss** popup, click in the first selection box under **Maintain Tangent** beside **Sketch<1>**
- Click **OK**. You can see the base **sketch<1>** (the skinny rectangle) is now predominant.
- Its control extends quite far up the feature.



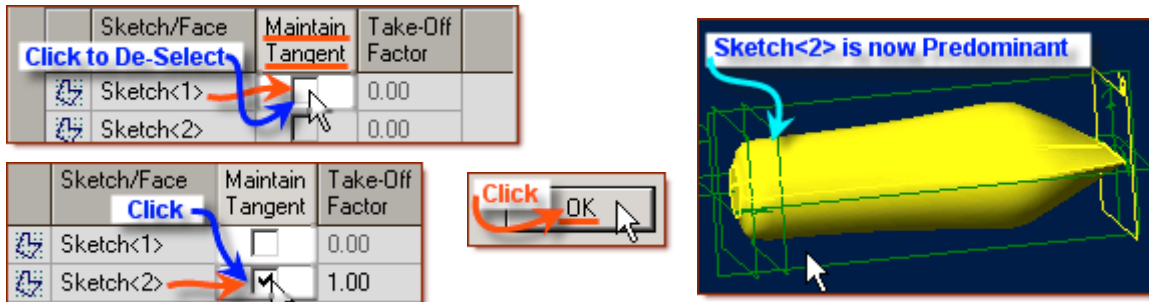
Let's now switch this to **Sketch<2>** and see the effects there.

Edit the Loft: Right-Click on Loft<3> in the Design Explorer

- Click on **Edit** in the drop down menu



- From the **Loft Boss** popup, click in the first selection box under **Maintain Tangent** beside **Sketch<1>** to de-select it.
- Now click on the selection box beside **Sketch<2>**
- Click **OK**. You can see the **sketch<2>** (the circle) is now predominant.
- Its control extends quite far down the feature.



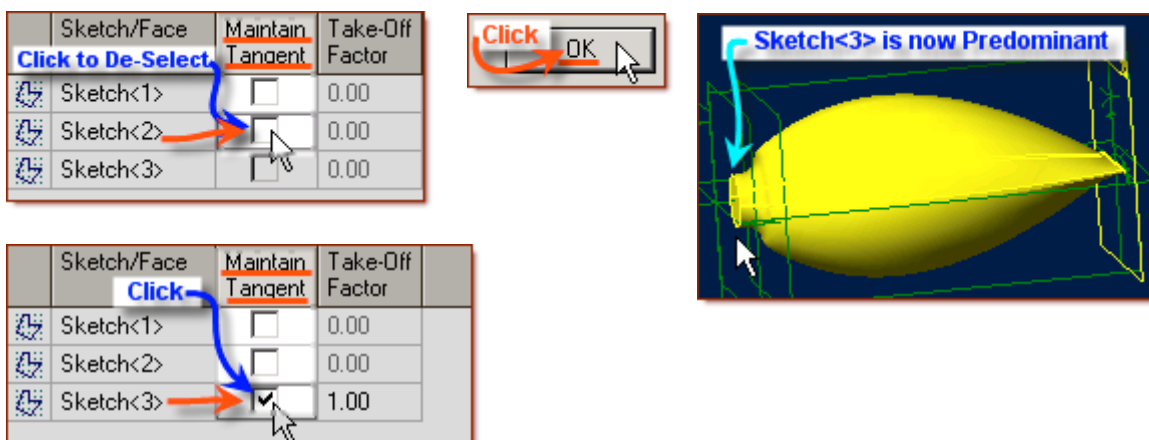
Let's now switch this to **Sketch<3>** and see.

Edit the Loft: Right-Click on **Loft<3>** in the Design Explorer

- Click on **Edit** in the drop down menu



- From the **Loft Boss** popup, click in the first selection box under **Maintain Tangent** beside **Sketch<2>** to de-select it.
- Now click on the selection box beside **Sketch<3>**
- Click **OK**. You can see the **sketch<3>** (the small circle) is now predominant.
- Its control also forces a new shape quite far down the feature.



So - you can see that simply selecting Tangent, causes quite a bit of change in the features you are creating, depending on where you force the tangent priority.

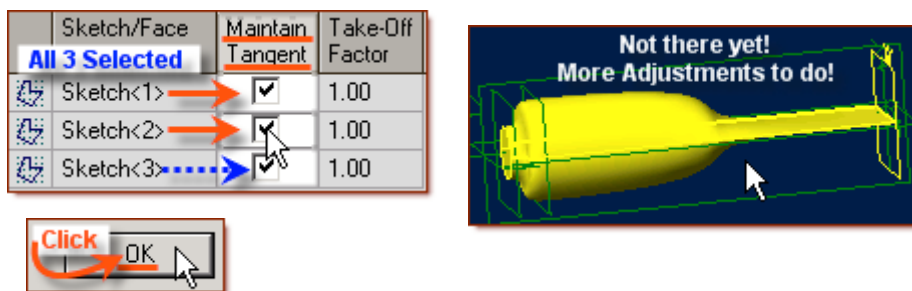
Now let's make them all Tangent.

Edit the Loft: Right-Click on Loft<3> in the Design Explorer

- Click on Edit in the drop down menu



- From the Loft Boss popup, click in the first selection box under Maintain Tangent beside Sketch<1>.
- Now click on the selection box beside Sketch<2>. (Sketch<3> should still be selected)
- Click OK.



Well! This doesn't get us all the way there either!

So - we will adjust the Take-Off Values to get the results we want.

Edit the Loft: Right-Click on Loft<3> in the Design Explorer

- Click on Edit in the drop down menu.

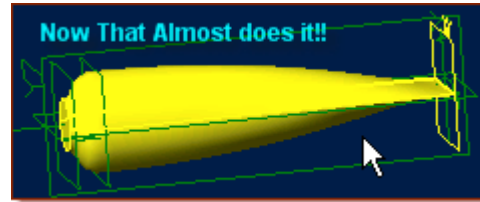


- In the Loft Boss popup, **double-click** in the Take-Off Factor beside Sketch<1>
- Click & Drag the mouse over the figure (1.00) to Highlight it,
- Enter 0.1. Press **Tab**.



- Now, **double-click** in the Take-Off Factor beside Sketch<2>.
- Click & Drag the mouse over the figure (1.00) to Highlight it,
- Enter 0.7 Press **Tab**.
- Finally, Now, **double-click** in the Take-Off Factor beside Sketch<3>.
- Click & Drag the mouse over the figure (1.00) to Highlight it,
- Enter 0.4 Press **Tab**.

Sketch/Face	Maintain Tangent	Take-Off Factor
Sketch<1>	<input checked="" type="checkbox"/>	0.10
Sketch<2>	<input checked="" type="checkbox"/>	0.70
Sketch<3>	<input checked="" type="checkbox"/>	0.4

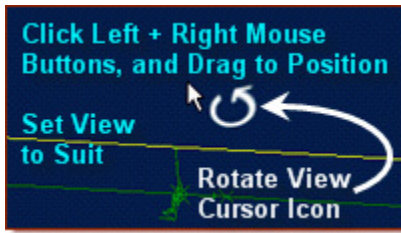


- Click **OK**. Now we have a reasonable approximation of your typical squeeze tube!

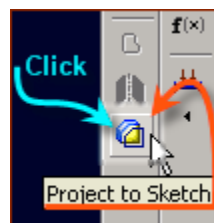
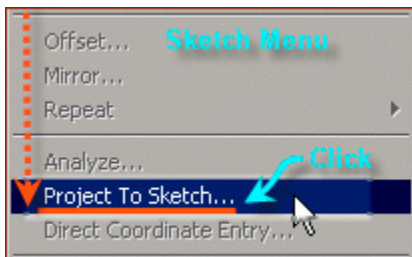
But Wait - there is more to add! The threaded top (Spout) of the tube!

To make the Spout, or threaded top, diameter match the size of the small circle at the upper end of the loft, we will use **Project to Sketch** from the loft itself.

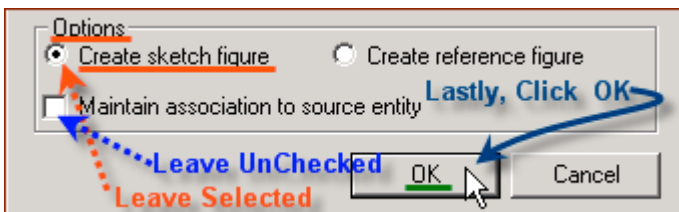
- Using a combination of left and right mouse buttons depressed together, rotate the viewing area.
- Click on the Circular end of the Lofted Tube



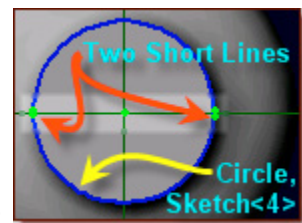
- Select **Project to Sketch** from the Main **Sketch Menu**, or **Project to Sketch** in the **Sketch Icons**.



- In the **Project to Sketch** popup, in the **Entities to project** selection area, **Face<1>** now shows.
- **Face<1>** should now show under **Entities to project**.
- Under **Options**, the Radio Button for **Create sketch Figure** is selected. Leave it like this.
- For now - we will leave **Maintain association to source entity** unchecked.
- Click **OK**. This has created **Sketch<4>**, a circle with two short lines dividing it in half. (They are from the sketch<1> edges.)



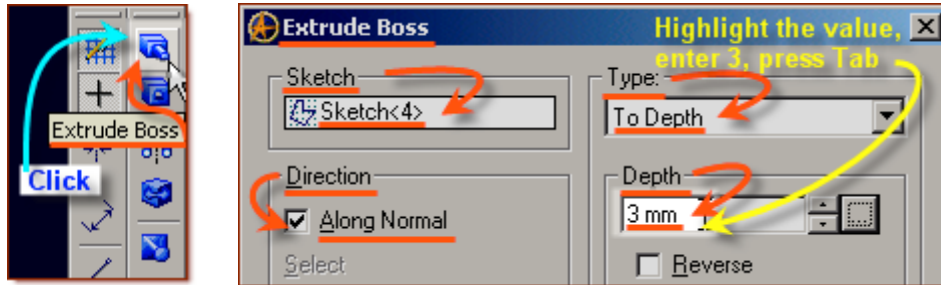
f 28



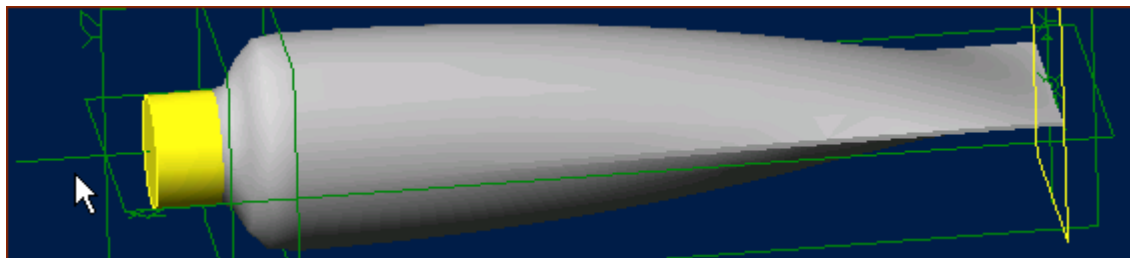
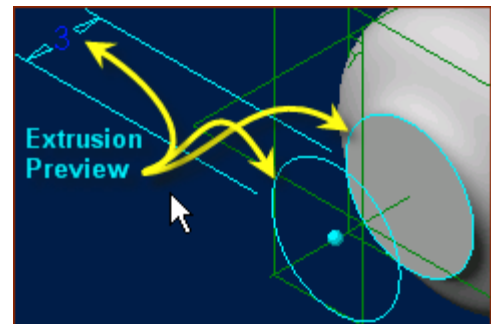
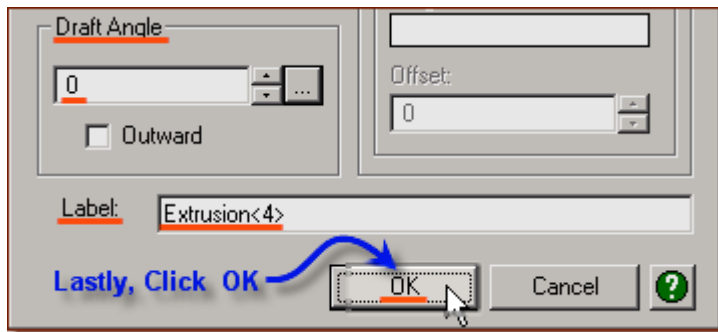
Now we will extrude this to make the spout.

While still in the sketch mode, click on **Extrude Boss** from the Feature Icons.

- In the **Extrude Boss** popup menu, **Sketch<4>** should be selected under **Sketch**,
- Leave **Type**: set at **To Depth**, and **Direction** Checked for **Along Normal**.
- Set **Depth** at 3 mm. (Highlight the value, and enter 3, press Tab.)



- Leave **Draft Angle** at 0,
- **Label**: is **Extrusion<4>** - may be left as is, or named to suit you.
- The preview shows the location of the spout extension.
- Click **OK**. This has now created the feature we will customize next.

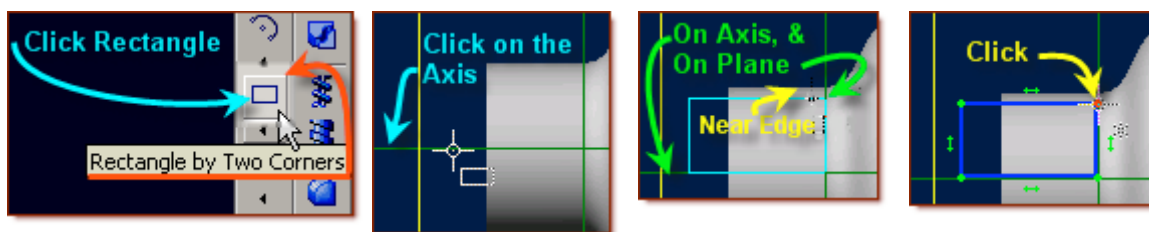


Create the Hollow Spout and Simple Threads.

Click on the YZ-Plane in Design Explorer, then Click on Activate Sketch, from the top of the Sketch Icons.

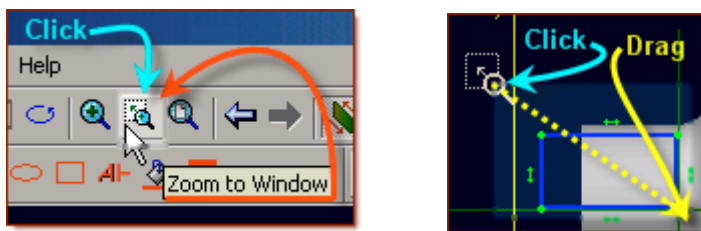


- In the Sketch Workspace, First click on the Rectangle by Two corners Icon,
- Then click on the axis (Green Line), outside of the existing extruded spout,
- drag over and up until you are near the edge of the extruded circle, and on the line for Plane<2>,
- Click.

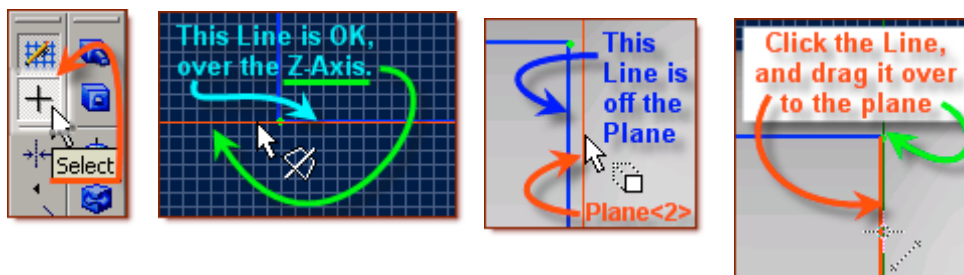


Confirm the Rectangle is on the lines representing the **Z-axis** and **Plane<3>**

- Click on the **Zoom to Window** Icon in the top row of Icons, and view close-up at the Rectangle and the **Z-Axis**. (Click above and left of the lower left corner of the rectangle, and drag down and to the right of this corner, creating a small selection rectangle, Release)



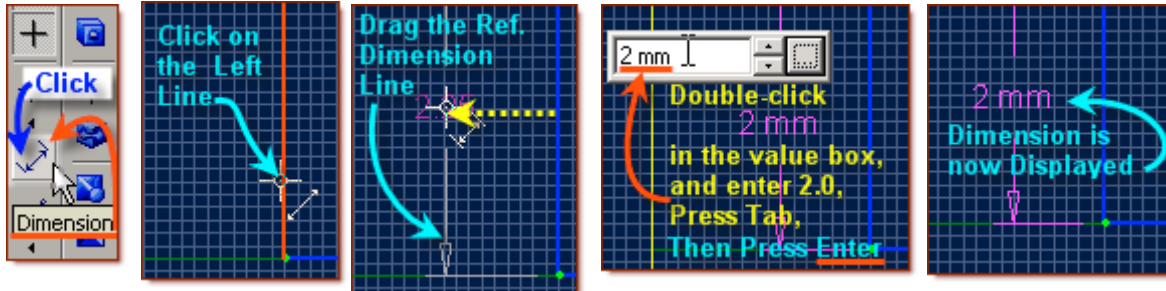
- If all is well, Click on **Previous View** (A bold Left Arrow) in the Top Icons, and repeat with the upper right corner of the rectangle, making sure it is on the line of **Plane<2>**.
- If either line is not over the green line representing the axis or plane, make sure you have selected the **Select** Icon in the **Sketch Icons**, and click the line, and drag it over the green line and release. *(It should snap in place automatically).*



- Click on the **Zoom to Fit** Icon in the top row of Icons, to see the whole sketch again.

From the Sketch Icons > Click **Dimension (*Add dimensions to the Rectangle*)**

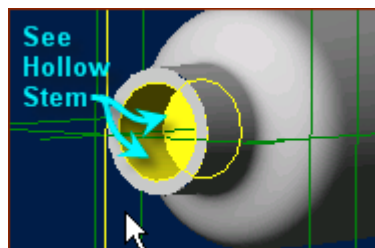
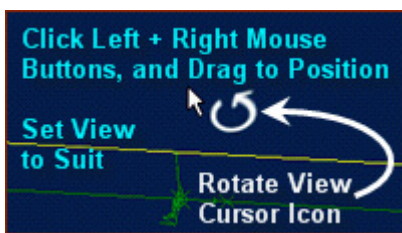
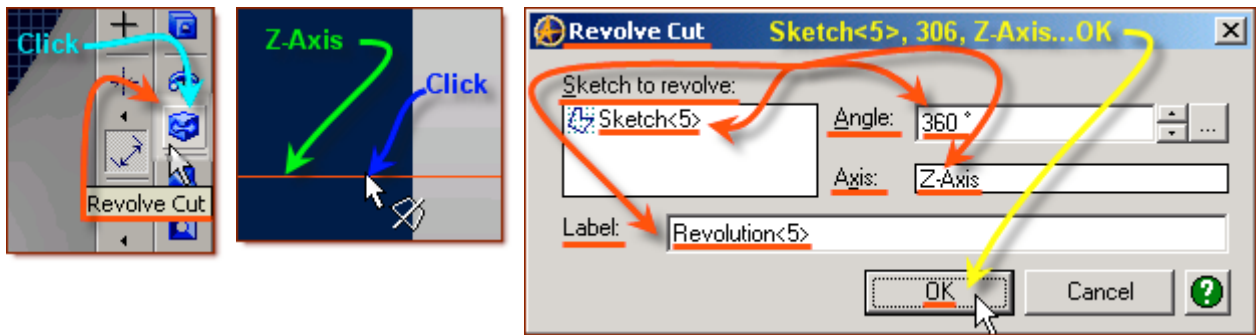
- Click on the left line of the Rectangle, and drag away and click to set the Dimension Place,
- Click, Edit the dimension to be 2 mm (double-click in the value box, and enter 2.0, press Tab)
- Press **Enter** to accept the dimension (2 mm).
- 2mm is now displayed beside the left edge of the rectangle.



Create the Hollow Spout

From the Feature Icons, Click on **Revolve Cut (4th Icon down)**

- In the **Revolve Cut** popup, the Sketch to Revolve should be defaulted as **Sketch<5>**,
- If not - select **Sketch<5>** from the Design Explorer.
- **Angle** should be defaulted at **360 degrees**, (*If not, double-click the value and enter 360, Press Tab.*)
- The **Axis** selection should be blank, click in the selection box, and Click the **Z-Axis** (Either in the workspace or in the **design explorer**.)
- **Label** should say **Revolution<5>**. That is fine, or you may name it to your liking.
- Click **OK**.



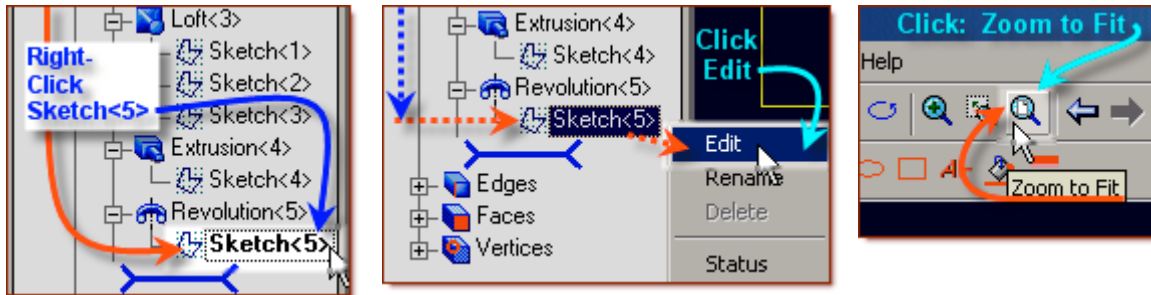
Holding both Mouse buttons down (Left and right) drag until you can see inside the top of the tube. You can see the now hollow stem at the top of the squeeze tube.

Now we will add the simple teeth, by editing the last sketch (Sketch<5>)

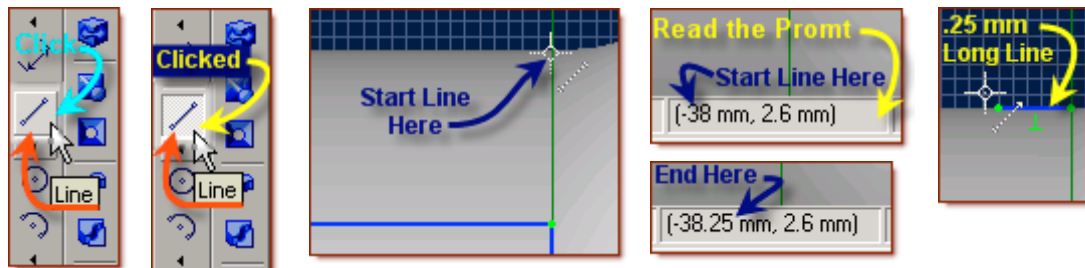
Create the Simple Threads.

Edit the Revolve Sketch:

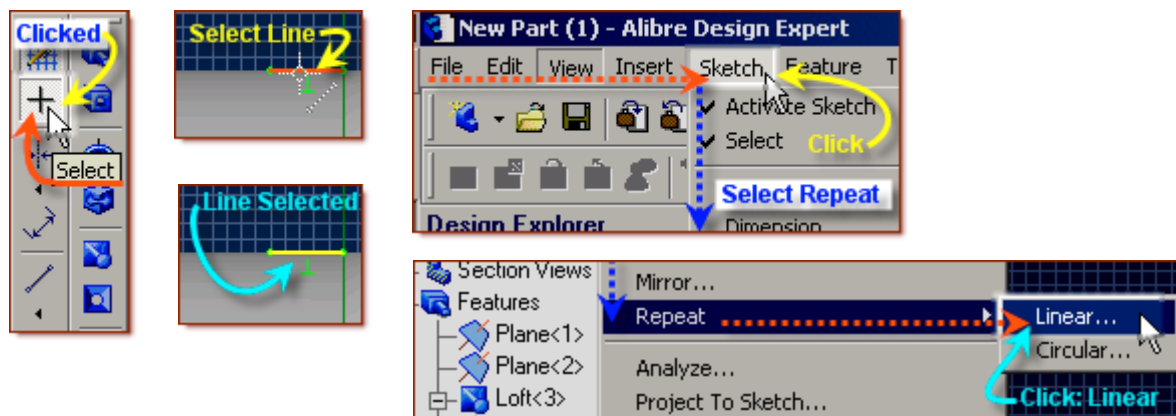
- Right-Click on Sketch<5> in the Design Explorer,
- Click on Edit in the drop down menu
- Click on Zoom to Fit in the top row Icons.



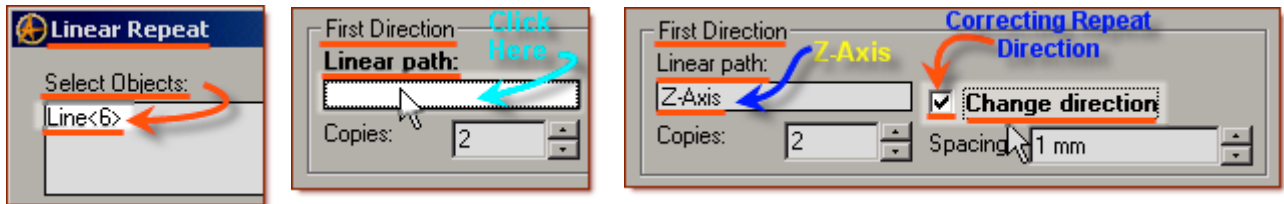
- Click on Line in the Sketch Icons,
- Start the line at the intersection of the extruded small circle and the plane<3>, Click and draw a line 0.25 mm long along the edge of the extruded circle. (*moving to the left from plane<3>*) Double-click to complete and exit the line.



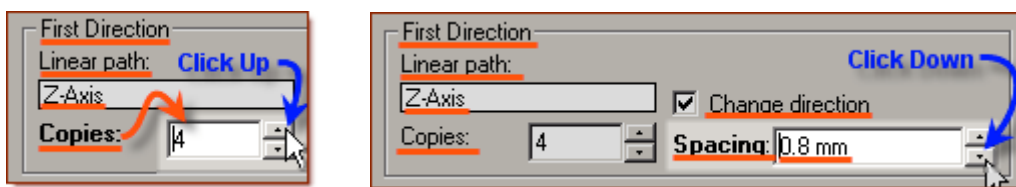
- Click on Select in the Sketch Icons, Then Click the line just created,
- From the Main Menu, Click Sketch > Select Repeat > Linear,



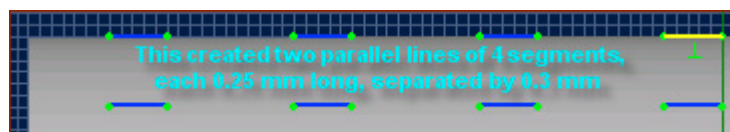
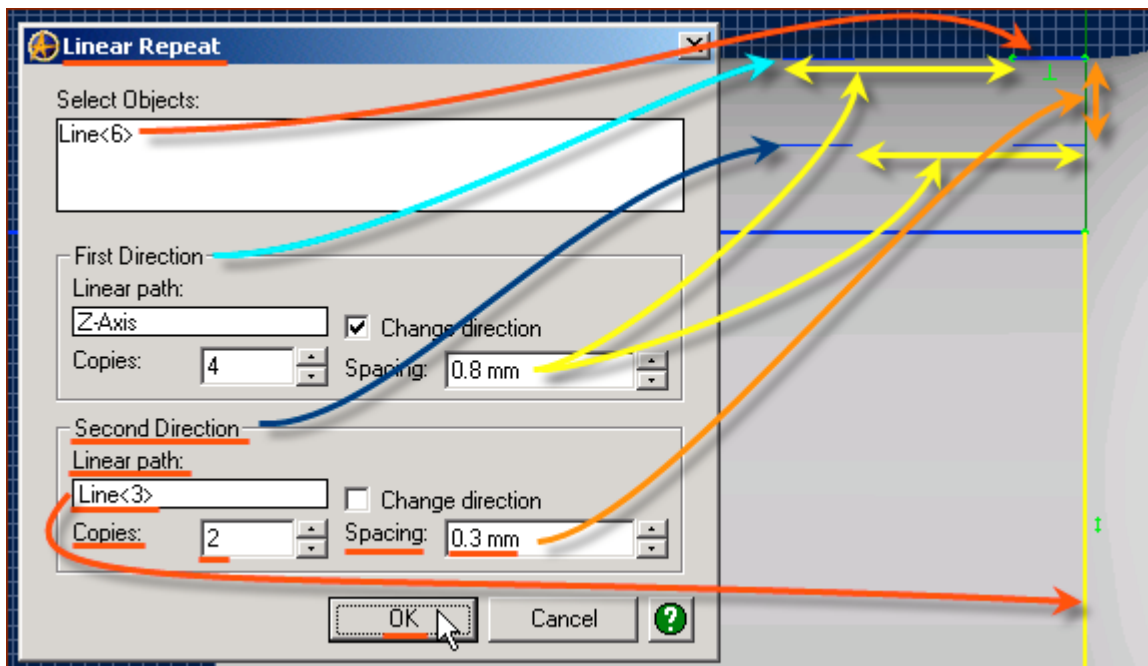
- In the **Linear Repeat** popup, **Select Objects** should show the selected **Line<6>**,
- Click in **First Direction - Linear path** selection box, and then click the **Z-Axis** (*If the line does not show to the left, click in the selection for Change direction.*)



- Increase the **Copies**: from 2 to 4 (*click the up arrow, or highlight the 2 and Enter 4, press Tab*)
- Set **Spacing** at 0.8 mm (*click the down arrow, or highlight the 1.0 and Enter 0.8, press Tab*)

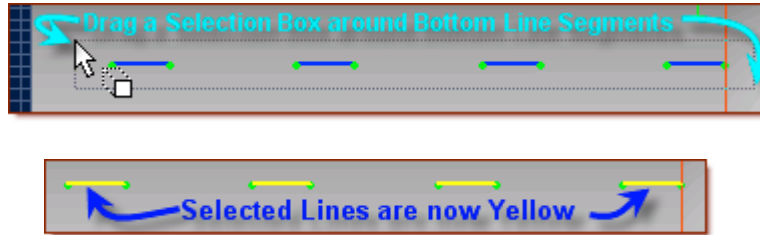


- Now in the **Second Direction - Linear path**: Click the selection box, and then click the right side line on the Rectangle (**Line<3>**)
- Leave the **Copies**: at 2, and set the **Spacing** at 0.3 mm. (*As Above*) Click **OK**.
- This will create two parallel lines of 4 segments, each 0.25 mm long, separated by 0.3 mm.

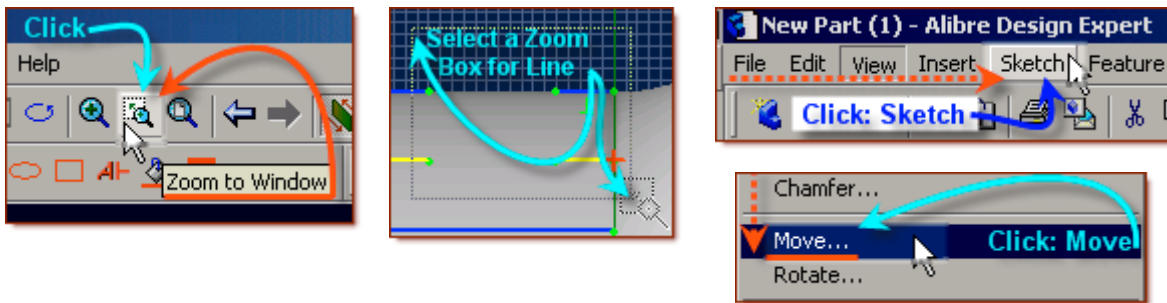


Now to re-locate the bottom line of segments to the mid-point in the gap of the segments in the upper row.

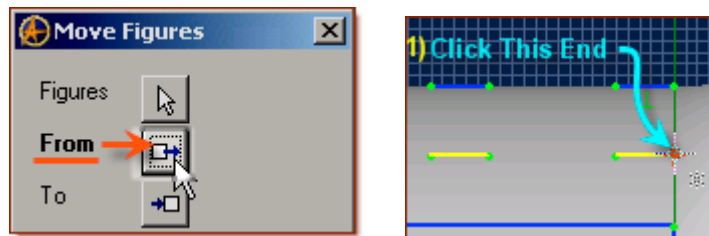
While still having Select as the active Icon from the Sketch icons, click and drag a selection box around all the segments in the lower line. The all should turn yellow.



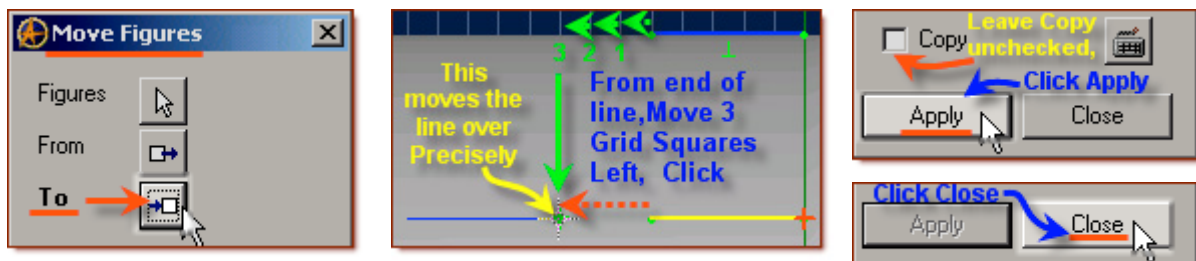
- Click on the **Zoom to Window** Icon in the top row of Icons, and Select a Zoom Box for Line
- From the Main Menu, Click **Sketch** > Select **Move**.



- In the **Move Figures** Popup, click the **From** button, then click the right end of the lower line (*Highlighted Yellow*) - at the edge of **plane<3>**.

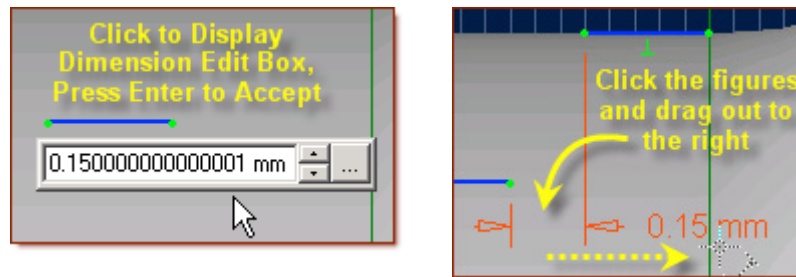
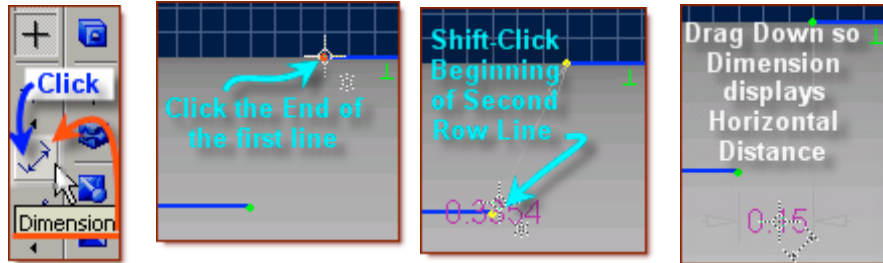


- Now, in the **Move Figures** Popup, click the **To** button, place the cursor at the other end of the same line (to the left) and move further 3 grid squares, and click.
- Leave **Copy** unchecked. Then Click **Apply** in the **Move Figures** Popup. Then Click **Close**.

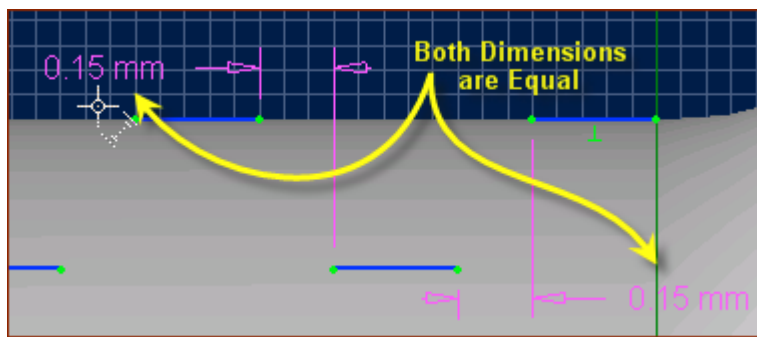


Now Show the Dimension the Move offset created between the two lines.

- Click **Dimension** in the **Sketch Icons**, and Click the End of the first line in the top row, and Shift-Click the beginning of the first line in the bottom row.
- Carefully drag down, so the cursor is about mid-point between the end of the first line, and the start of the second line, and below it, - click. Press Enter. Now - Click the figures and drag out to the right for better visibility. (*The dimension should be 0.15 mm*)

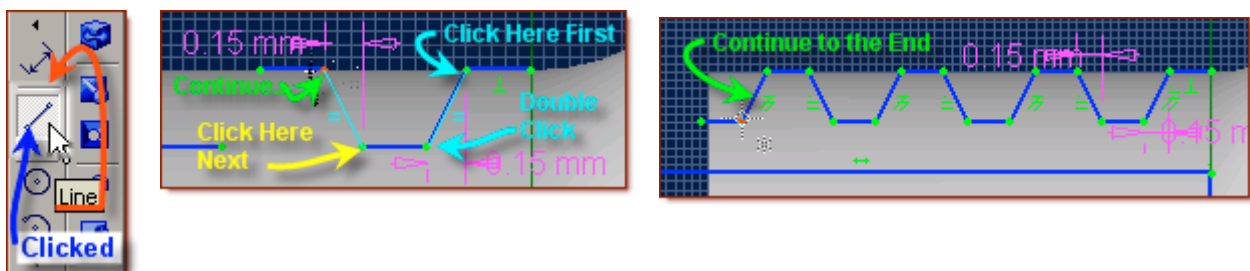


- Do the same between the end of the first line in the second row and the beginning of the second line in the first row. (*The dimension should also be 0.15 mm.*) (**Both Dimensions are Equal**)



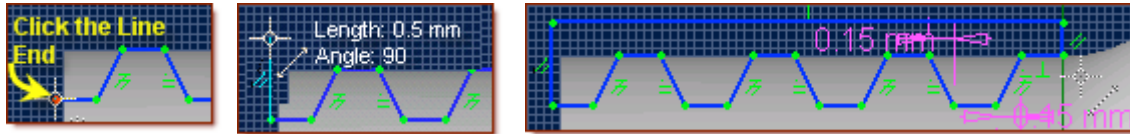
Now connect the offset lines, with the line tool

- Click **Line** in the **Sketch Icons**, Click on the end of the first segment in the upper line, and double-click the beginning of the first segment in the lower line.
- Click on the end of the first segment in the lower line, and the beginning of the second segment in the upper line. **Repeat to the end. (Manually)**

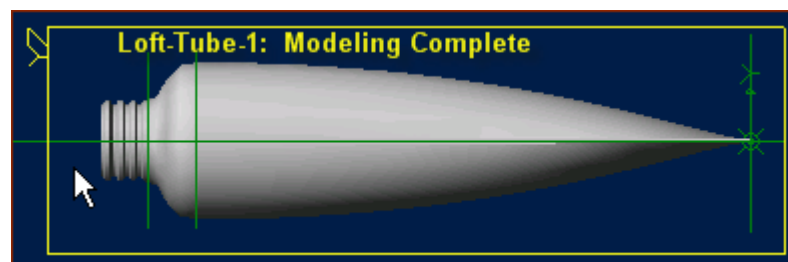
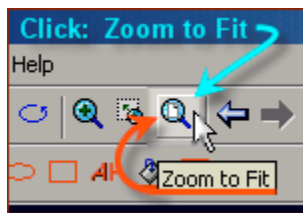
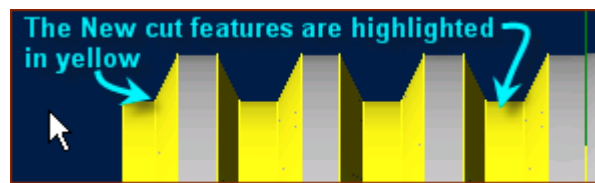
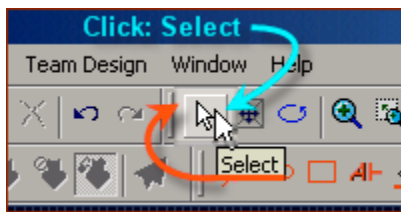


Now close these lines outside of the extruded circle by adding 3 lines to make a closed box with them.

- With **Line** still selected, click on the end of the last segment, on the lower line, and drag up 0.5 mm. Then to the right until over the start of the first segment in the upper line (3.05 mm), and then down to the start of the first segment, double-click to close the line.



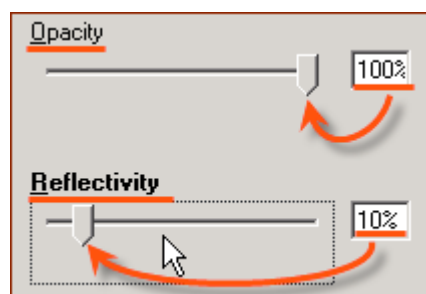
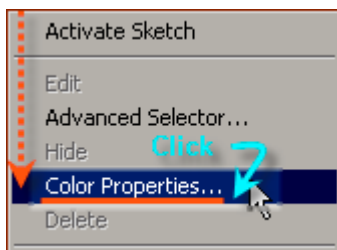
- Now, click on the **Select** Icon in the top row of icons to exit the sketch mode, and to process the new sketch features into the revolution, effectively creating the simple threads.
- You can now see the cut features highlighted in yellow.
- Click the **Zoom to Fit** Icon at the top. Click in the free area of the workspace to de-select the (Yellow) new features.



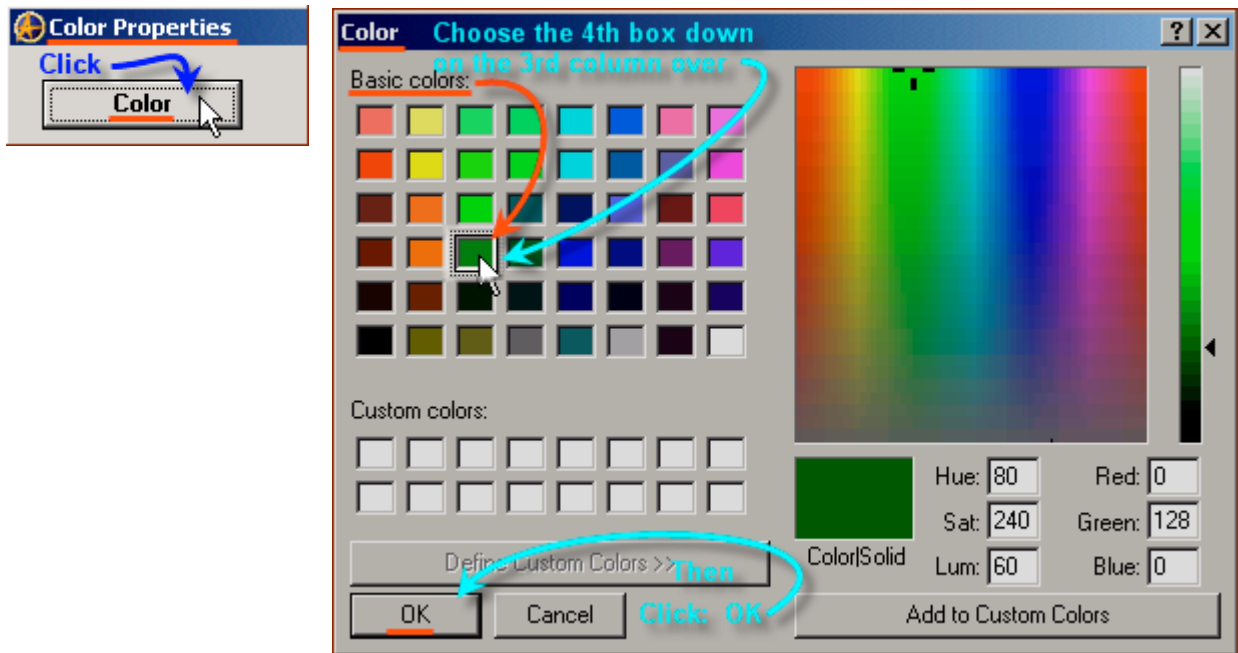
Congratulations! You have now completed the modelling of the Loft-Tube-1

Now you can edit the loft tube's Color properties and save!

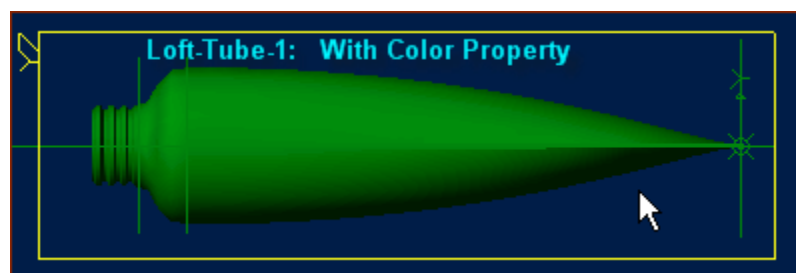
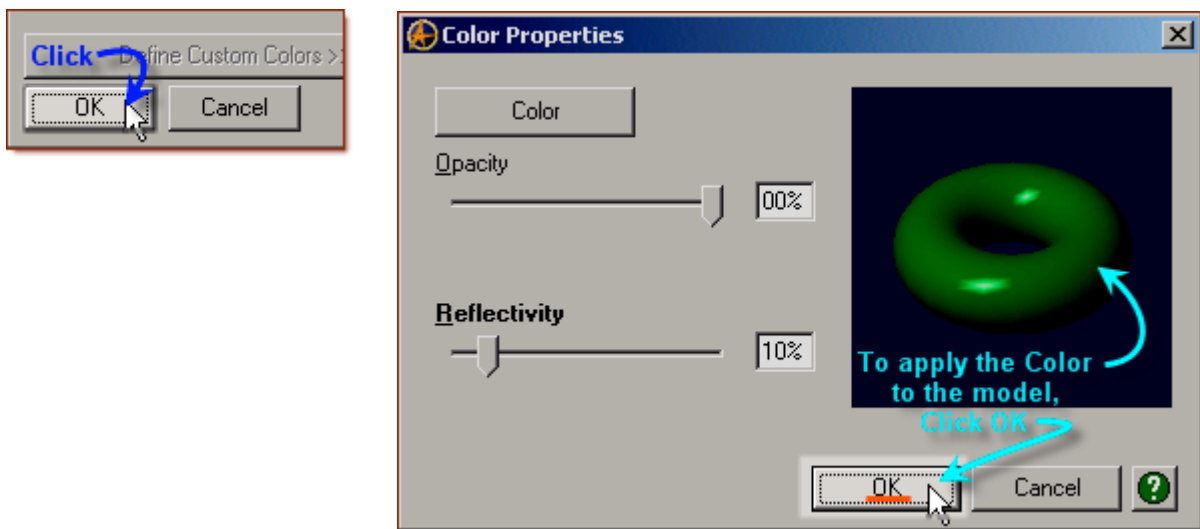
- Right Click in the Part Work Area Free Space, Select **Color Properties** from the Popup Menu.
- From the **Color Properties** Popup Window - Set the **Reflectivity** Slider to 10%,
- Leave **Opacity** at 100%,



- Click on **Color** to select a new Color:
[i.e. Choose the 4th box down on the 3rd column over under **Basic Colors** - a Dark Green]

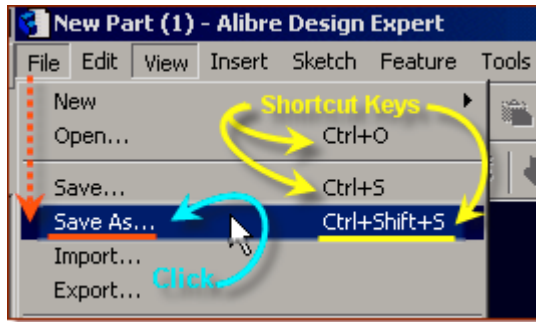


- Click **OK** to close the Color selection box and accept the new Color,
- Click **OK** to apply the Color to the model.

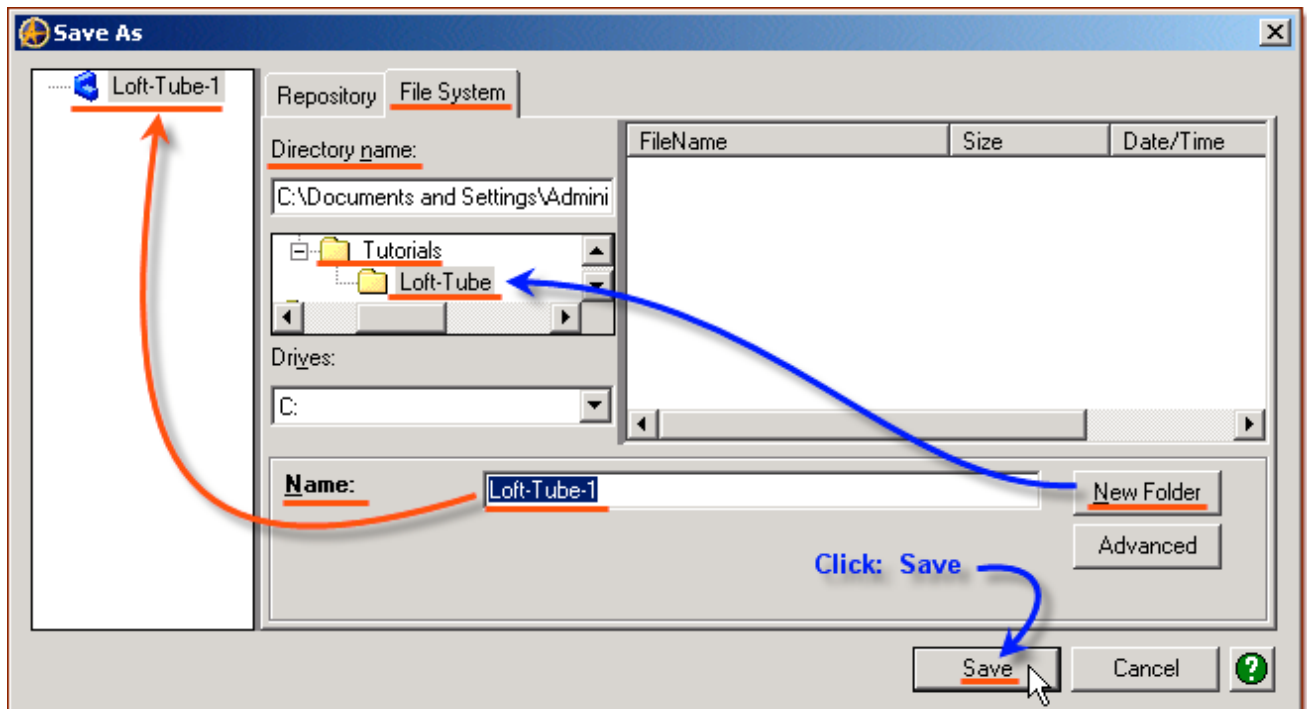


Now, to save this file in it's own folder:

- From the main text menu, select **File, Save As** (Also Press *Ctrl+Shift+S*, the *Shortcut Keys*)



- From the **Save As** Popup, under File System: Explorer Directory name view - select Tutorials folder.
 - Click on **New Folder** - create one with the name **Loft-Tube** under Tutorials.
 - Select this new folder named: **Loft-Tube**
- Edit the name of the part - to Loft-Tube-1, and click - **Save**.



Congratulations!

You have completed the Loft-Tube-1 Tutorial!

(Complete Version)

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