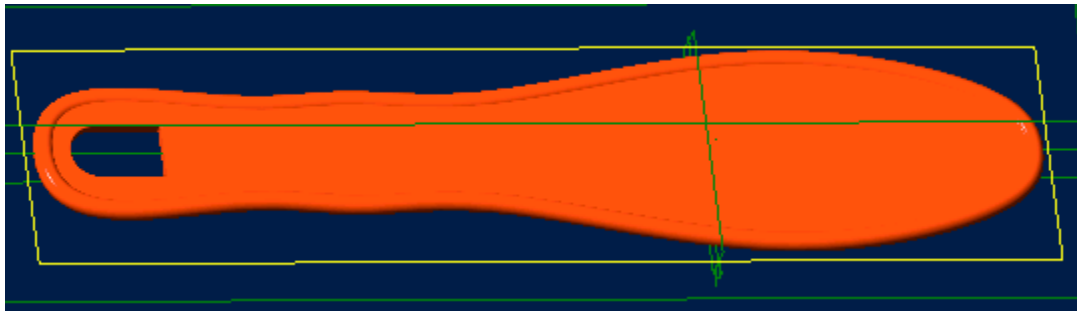


Alibre Design Tutorial: Sweep, Extrude (Boss & Cut), & Fillet Sweep_Foot-Scrubber-1



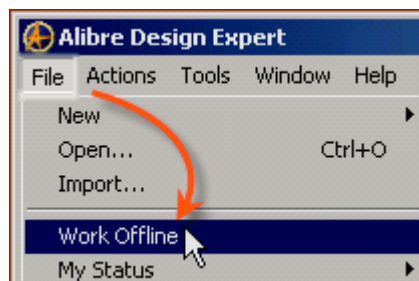
Part Tutorial

Exercise 8: Sweep Foot-Scrubber-1 [\[Complete Version\]](#)

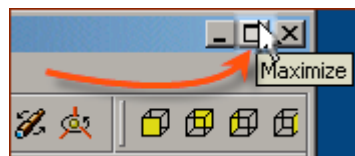
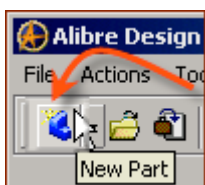
In this Exercise, We will set System Parameters first, then part options, Then setting up the save, at the beginning - so as to save easily as we perform each step. Then, in sketch mode, we will set the first sketch of the scrubber for the Sweep Feature. Then we will use the Sweep Feature to create the Primary Smooth edge. Next, We will use a Feature Mirror to replicate the Sweep to cover Both Sides, Then we will create a sketch and use Extrude Boss it to fill in the middle of the handle. Then we will create a sketch and use Extrude Cut to make the hole in the handle. We will fillet the edges of this hole for a smooth finish, Setting the Color Properties will follow this, and re-saving the file.

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.)



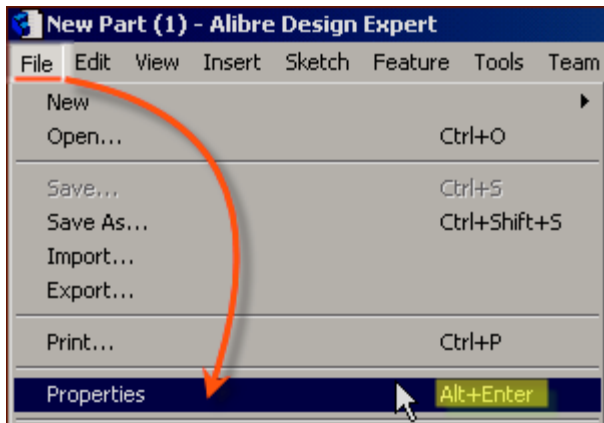
Open a **New Part** workspace. From the Home window - click on the **New Part** icon.
Maximise the Part Workspace Window.



*If at any time you are not sure of a keyboard item, see: [Keyboard Basics](#) or [Here](#).

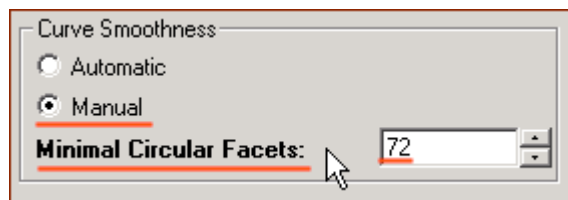
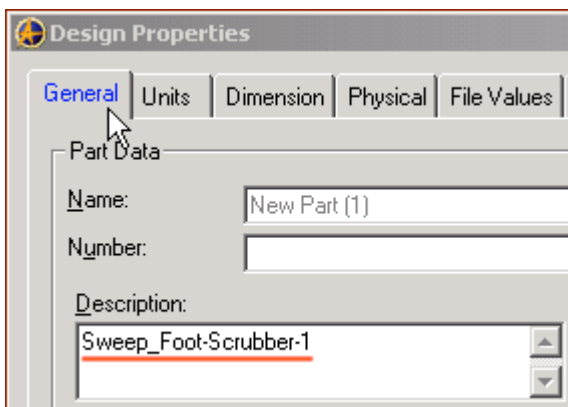
Set Design Properties.

Select **File, Properties.** **Keyboard shortcut = Alt+Enter**



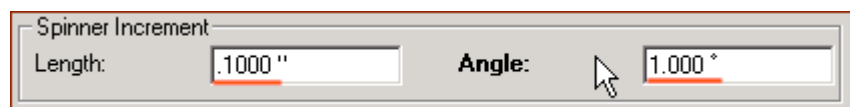
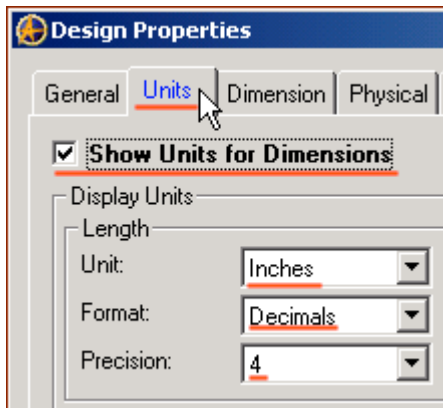
A) Select the Tab Marked "General"

- In **Description**: insert the information: 'Sweep_Foot-Scrubber-1'.
- In **Curve Smoothness**, Select the Radio Button marked 'Manual', and set the 'Minimal Circular Facets:' at 72.



B) Select the Tab Marked "Units"

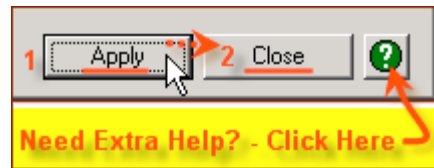
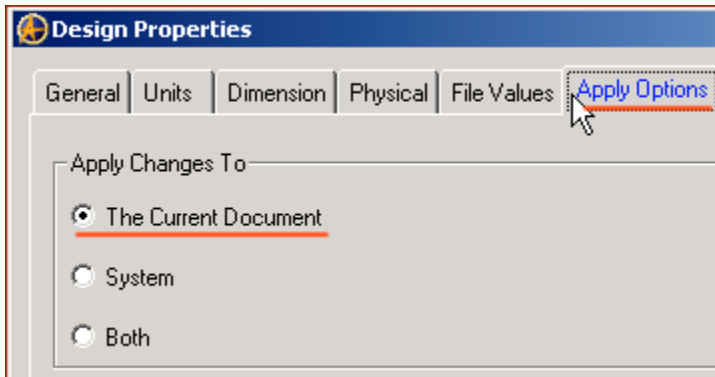
- Check (check mark in box is showing) the Box **Show Units for Dimensions**.
- On **Display Units** - confirm **Unit: Inches**, **Format: Decimals**, and **Precision: 4**.



- On **Angle** - Confirm **Angle: Degrees** and **Precision: 3**. Adjust if not.
- On **Spinner Increment** - for **Length**: enter 0.1000 " and for **Angle**: enter 1.000 ° (degrees). Adjust to these parameters if not. Press 'Tab' to move after editing them.

C) Select the Tab Marked "Apply Options"

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



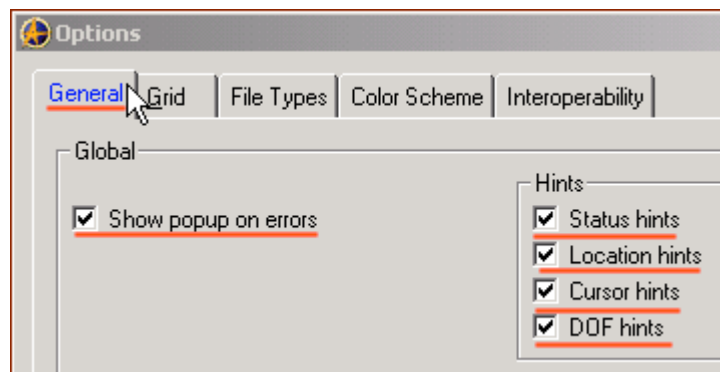
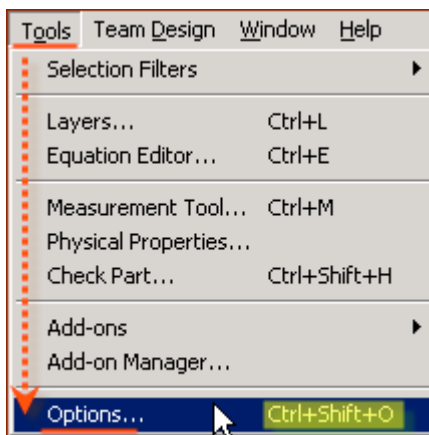
Set Part Options.

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,

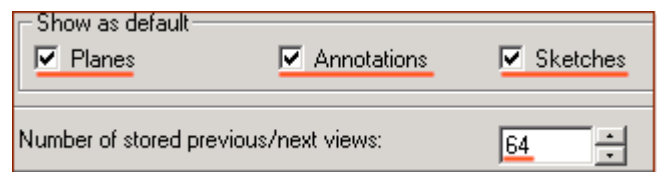
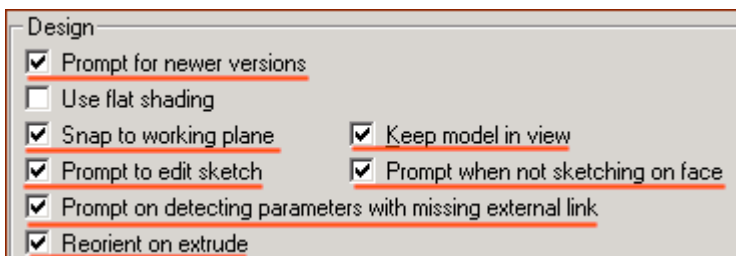
Hints - All



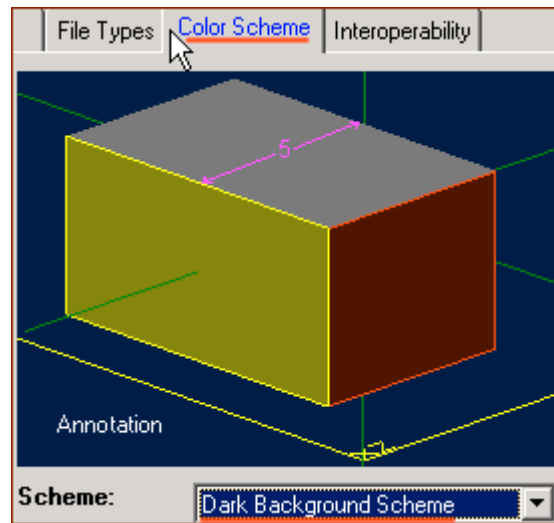
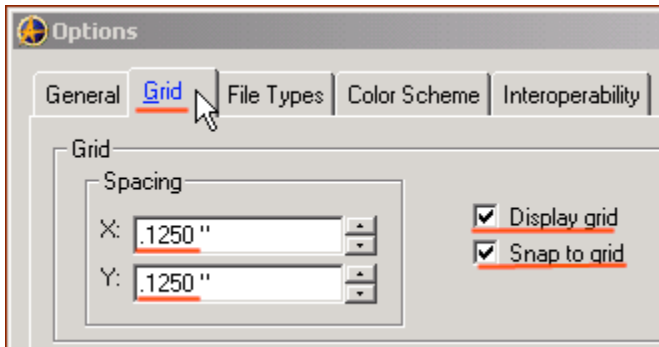
Design

- 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: .125 " and Y: .125 ", 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.)

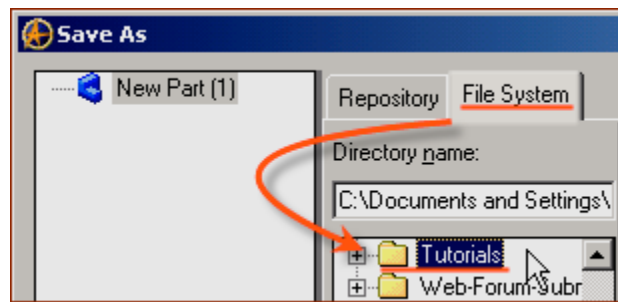
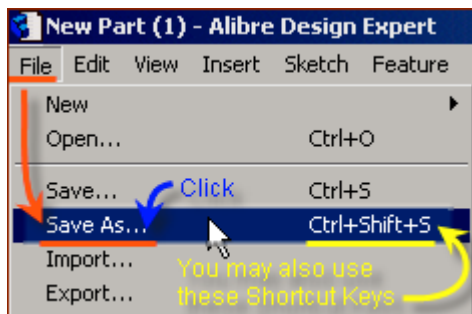


Now you have set up the workspace to easily create the Model, next you will set up the save. *(This is the recommended procedure to safely keep your file up to date, while you are working on your file.)*

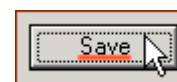
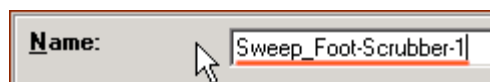
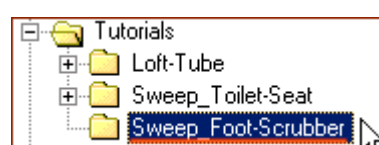
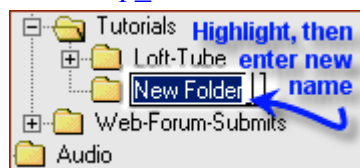
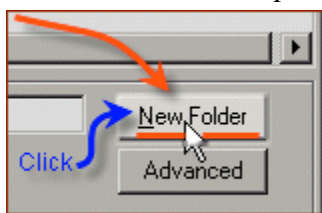
Set-up Save.

Now, to save this in it's own folder, making it easy to simply save as we proceed:

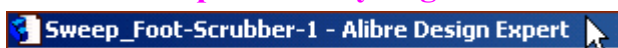
- From the main text menu, select **File, Save As**. (Also Press **Ctrl+Shift+S**, the **Shortcut Keys**)
- From the **Save As** popup, under the **File System Explorer** directory view - select **Tutorials** folder.



- Click on **New Folder** - create one with the name **Sweep_Foot-Scrubber** under **Tutorials**.
- Select this new folder named: **Sweep_Foot-Scrubber**
- Edit the name of the part - to: **Sweep_Foot-Scrubber-1**, and click - **Save**.



Ok - All set-up and ready to go!

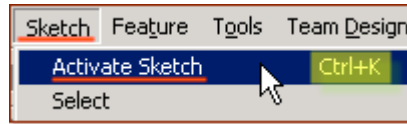
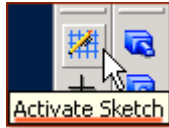
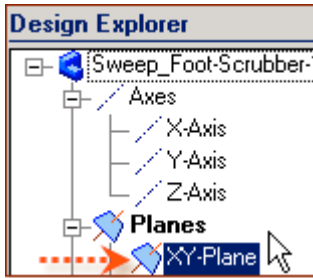


Start the First Sketch for the Sweep Path (This is a bit long!).

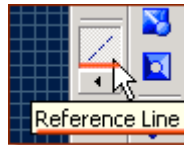
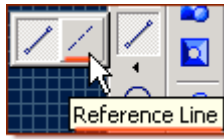
In the **Design Explorer**, confirm **The X-Y Plane** is selected under **Planes**.

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

Or, you can select **Activate Sketch** from the **Sketch** Menu, or press **Ctrl+K**.




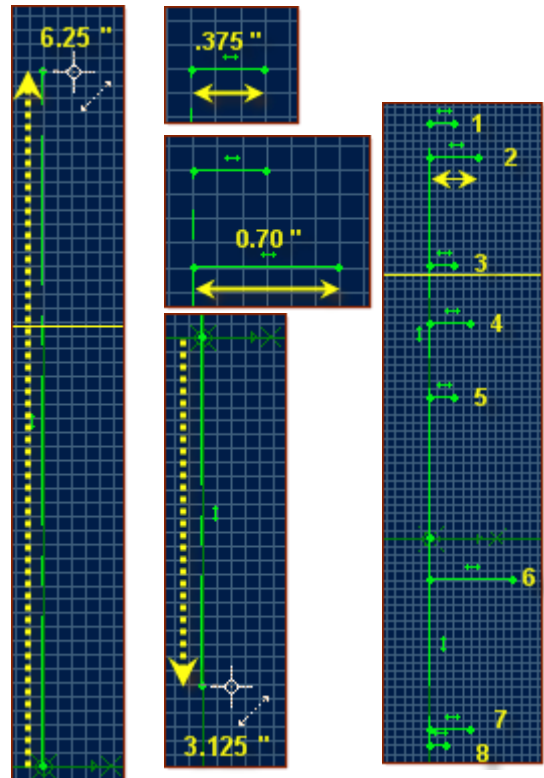
From the **Sketch Icons** > Click **Line Options** (Below Line Icon) -> Select **Reference Line**



(Note: the Cursor Icon for the Reference Line - is broken/dotted.)

- Click the Origin, and drag a vertical Reference Line up the 'Y' Axis 6.250 " - Double Click,
- Click the Origin, and drag a vertical Reference Line down the 'Y' Axis 3.125 " - Double Click,
- Still using the Reference Line, create 8 horizontal lines to the right of the two vertical lines,
- We will add dimensions and force the lines into place after placing them:
- Place a short line **.375 "** Long from the axis and from the top end of the top line,
- The **2nd** short line **.70 "** Long, **.45 "**
(about 4 grid squares) below the first line,
- The **3rd** short line **.40"** Long, **1.60"**
(about 12 grid squares) below the second line,
- The **4th** short line **.60 "** Long, **0.90 "**
(about 6-7 grid squares) below the third line,
- The **5th** short line **.375"** Long, **1.175"**
(about 9 grid squares) below the fourth line,
- The **6th** short line **1.1250"** Long, **2.750"**
(22 grid squares) below the fifth line,
- The **7th** short line **.625"** Long, **2.25"**
(18 grid squares) below the 6th line
(2 square up from the bottom end of the bottom line)
- The **8th** short line **.25"** Long running from the
bottom end of the bottom line out to the right.

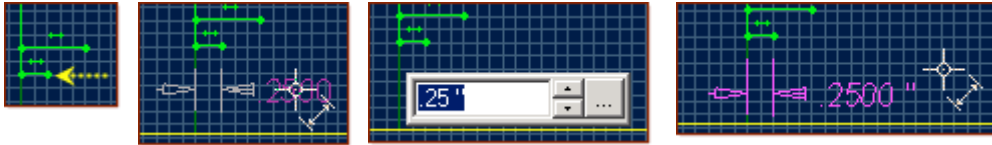
Click **Zoom to Fit**  - so everything is on screen.



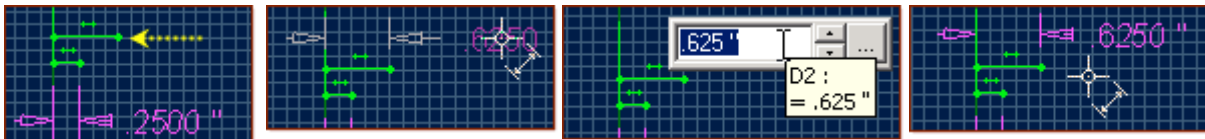
From the **Sketch Icons** > Click **Dimension** (4th Icon down, note the cursor icon)



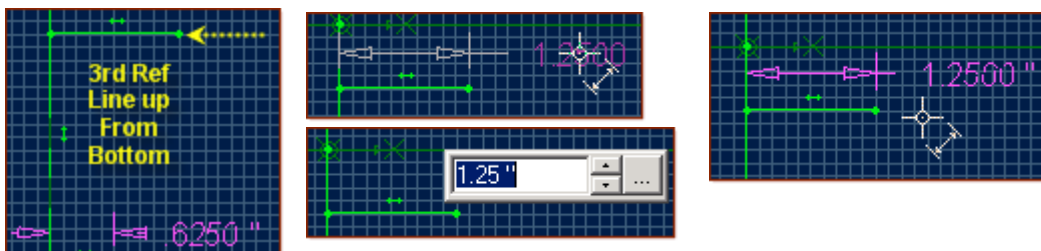
- Click the Bottom Reference Line, and drag the dimension below it - Click, Confirm the Length is .2500 " or edit to be .2500 " and press Enter.



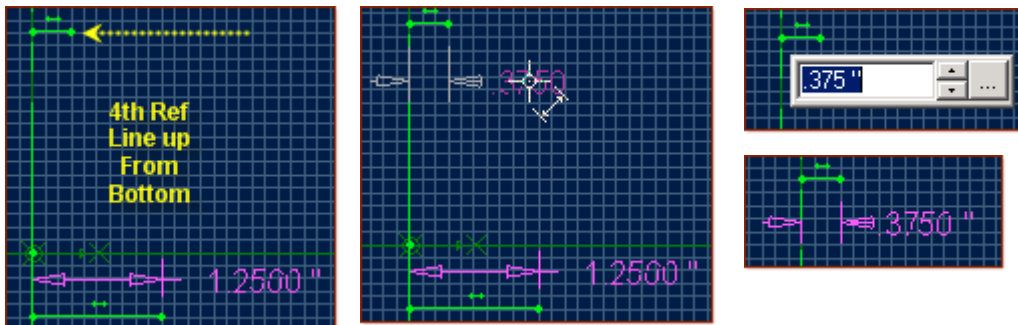
- Click the next reference line up from the bottom, and drag the dimension above it - Click and confirm the Length is .6250 " or edit it to that, and press Enter.



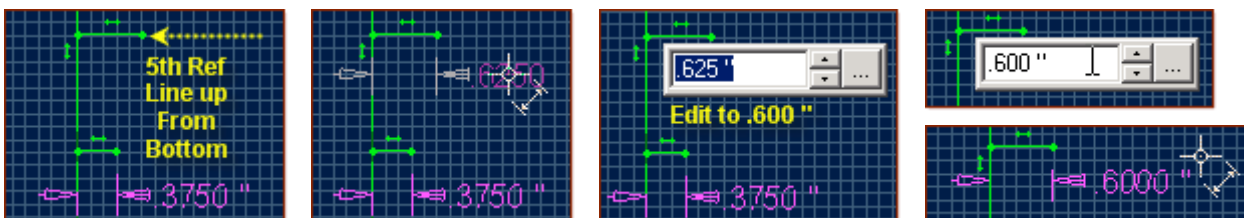
- Click the third reference line up from the bottom, drag the dimension above it, Click and confirm the Length is 1.1250 " or edit it to that, and press Enter.



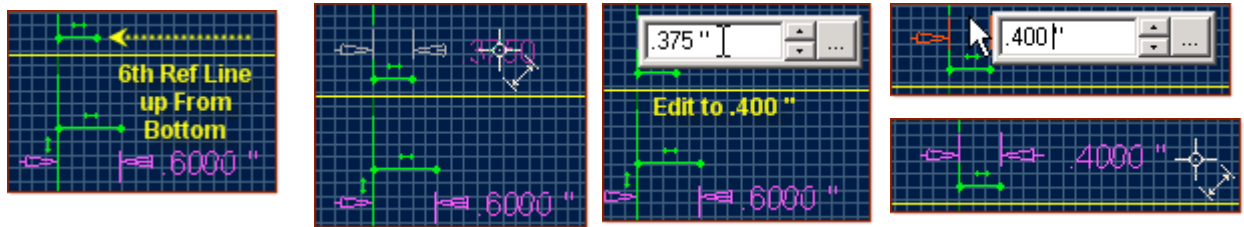
- Click the fourth reference line up from the bottom, drag the dimension down below it, click and confirm the Length is .3750 " or edit it to be that, and press Enter.



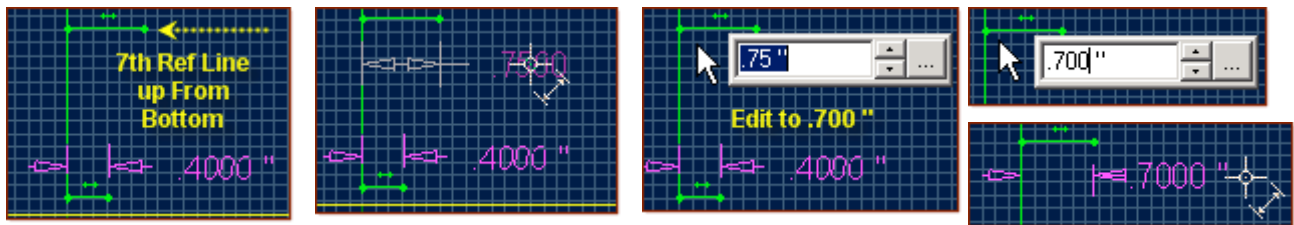
- Click the fifth reference line up from the bottom, drag the dimension below it, Click and confirm the Length is .6000 " or edit it to be that, and press Enter.



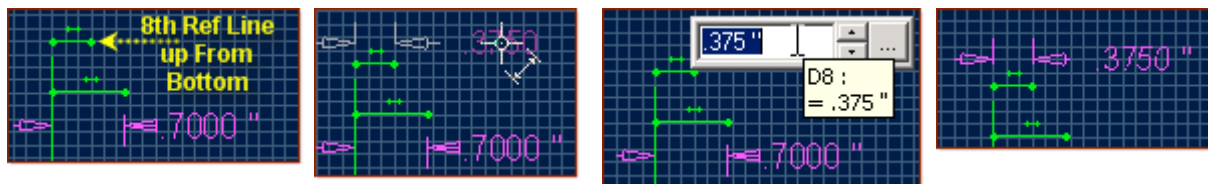
- Click the sixth reference line up from the bottom, drag the dimension above it, click and confirm the Length is .4000 " or edit it to be that, and press Enter.



- Click the seventh reference line up from the bottom, drag the dimension below it, Click and confirm the Length is .7000 " or edit it to be that, and press Enter.

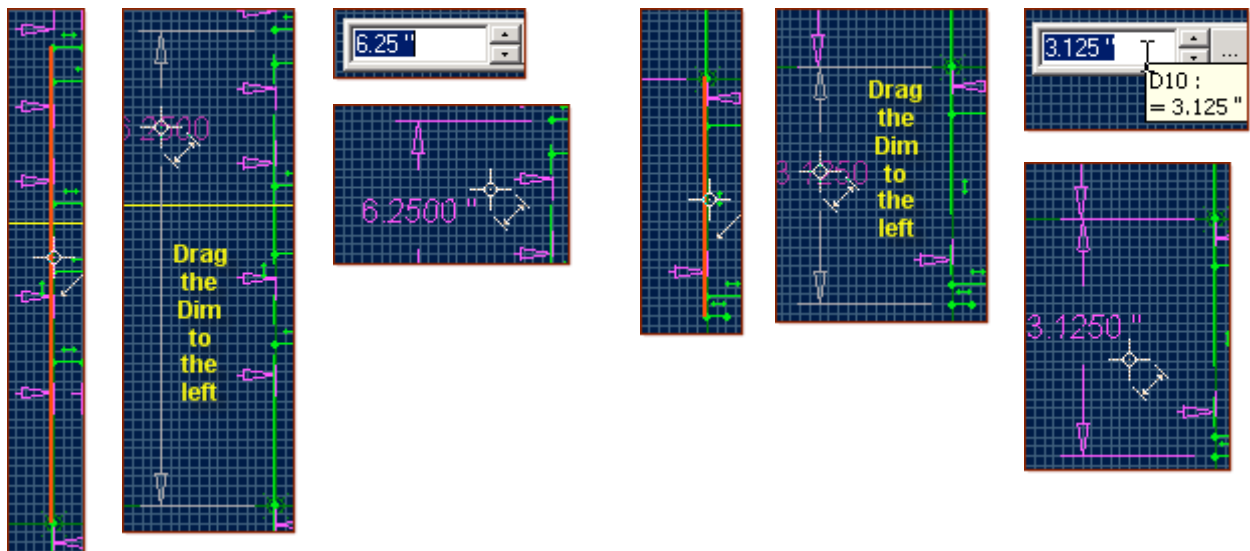


- Click the eighth reference line up from the bottom, drag the dimension up above it, Click and confirm the Length is .3750 " or edit it to be that, and press enter.



Now - Measure and confirm the length of the two Vertical Reference Lines.

- First, Click the upper Reference Line and drag out to the left to place the Dimension, Click, and confirm or edit the Value to be 6.25 ", Click Enter.
- Second, Click the Lower Reference Line and drag out to the left to place the Dimension, and confirm or edit the value to be 3.125", Click Enter.




Now would be a good time to save the work done.

Simply Click the **Save Icon (Disk)**. Click **Save** in the Dialog Box
 Saving exits the sketch mode, and creates **Sketch<1>** under **Features** in the **Design Explorer**.
 To return to the Same Sketch, Right-Click on the **Sketch<1>** Listing, and select **Edit**.

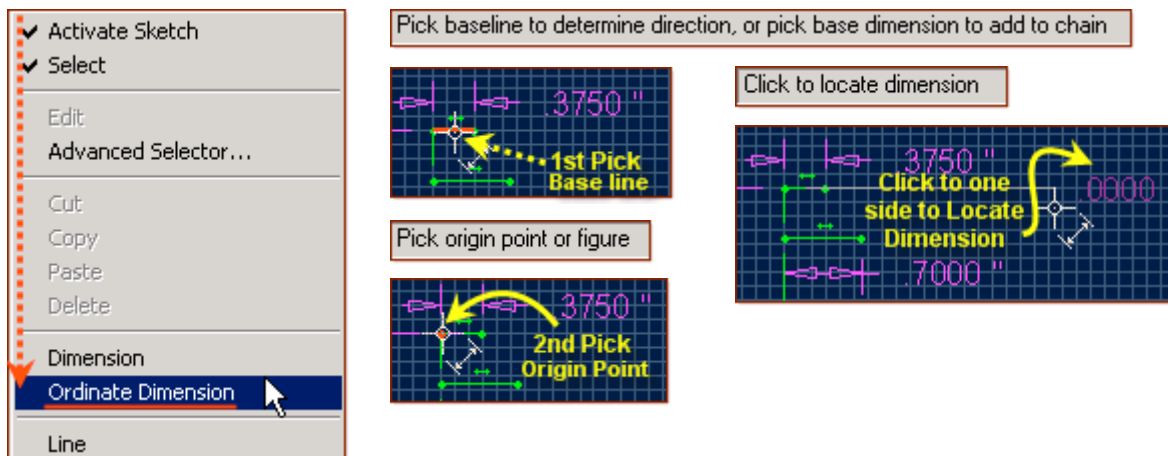


This has completed setting the lines and specifying their lengths.

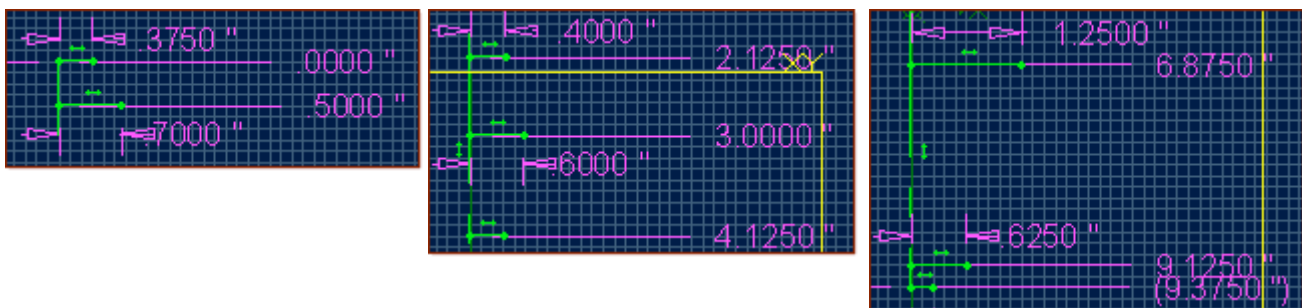
Now we will use **Ordinate Dimensions** to identify the lines with reverence to a base line and then edit these dimensions to lock the heights of the short lines where we want them.

First - Select **Zoom to Fit**  from the View Toolbar, to get the whole sketch into view, and make sure the workspace window is still maximised.

- Right Click in the Part Workspace area, and select **Ordinate Dimension** from the popup menu,
- Next step is to pick a baseline to determine the direction of the running dimensions:
(Note the Status Bar Hints) - Click the small horizontal reference line at the top, (*0.375 " Length*)
- Then - to pick an origin point or figure: we will pick the end point of the first short reference line at the top to set out ordinate 0" point,
- Then - we will click to one side - to locate the side the dimensions will be placed.



- Next - pick the next line down - end point, to measure it in reference to the 0.00 point, drag the same side again, click and the dimension is placed.
- Continue down clicking end points and dragging the lines to one side until finishing at the 8th line.



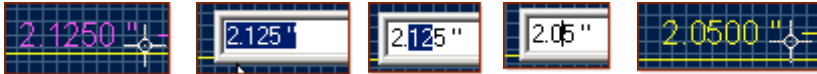
When done placing the dimensions - Click on **Select** in the **Sketch Icons** -  to exit **Ordinate Dimension**.

Now - we will edit these Dimensions to set the lines at the right heights:

- Double Click the first Ordinate dimension below .0000 " (~.5000") and edit it to **.4500 "**, Click Enter



- Double click the next down Ordinate Dimension (~ 2.125 ") and edit it to be **2.0500 "**, Click Enter,



- Double click the next down Ordinate Dimension (~ 3.00 ") and edit it to be **2.9500 "**, Click Enter,



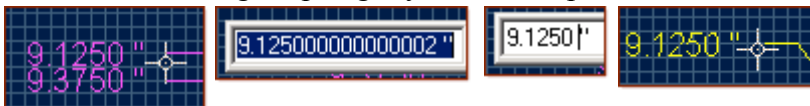
- Double click the next down Ordinate Dimension (4.125") and confirm it to be **4.1250 "**, Click Enter,



- Double click the next down Ordinate Dimension (6.875") and confirm it to be **6.8750 "**, Click Enter,



- Double click the next down Ordinate Dimension (9.1250") and confirm it to be **9.1250 "**, Click Enter, Then Click & Drag it up slightly to a clear spot.



- The last Ordinate Dimension should be in Brackets and should read **(9.3750 ")** - meaning it is fixed and can not be edited. Click on it and drag the Dimension down to a clear space. Note the offset line.



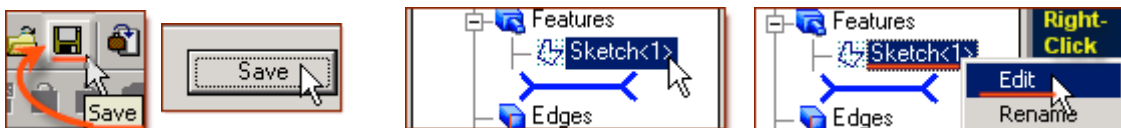
Now we have all the lines with end points located as required we can connect the B-Spline to them.


This would be another good time to save the work done.

Simply Click the **Save** Icon (Disk). Click **Save** in the Dialog Box

Saving exits the sketch mode, and creates **Sketch<1>** under **Features** in the **Design Explorer**.

To return to the Same Sketch, Right-Click on the **Sketch<1>** Listing, and select **Edit**.

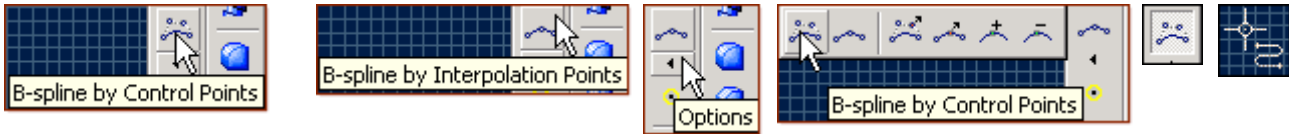


Click **Zoom to Fit**  - so everything is on screen.

Now to add the B-Spline

From the **Sketch Icons** > Click **B-Spline by Control Points** (9th Icon down, note the cursor icon)

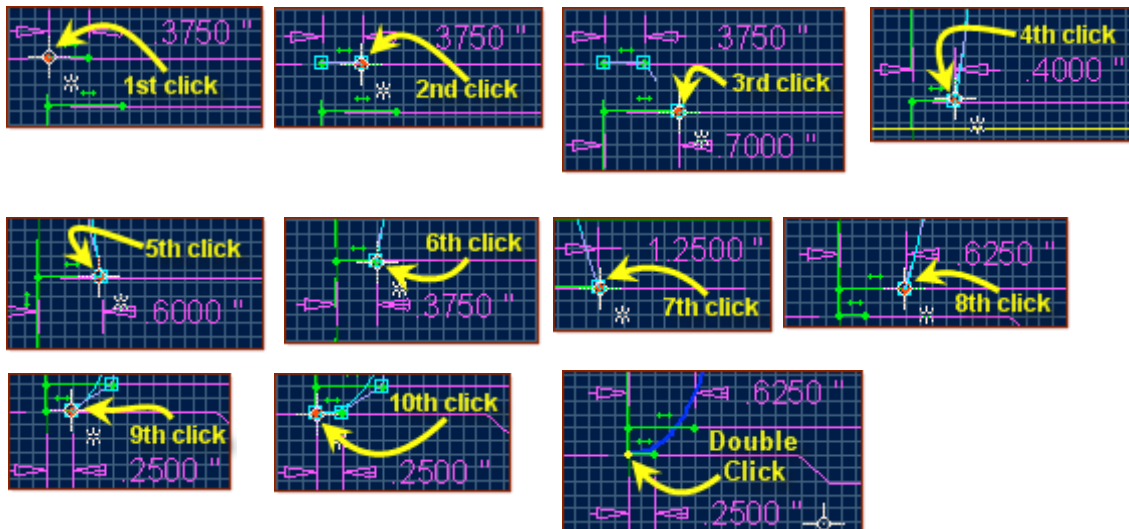
If **B-Spline by Control Points** is not currently selected, you may click the option button, and select the icon from the Flyout menu.



- Use Zoom to Fit, and Zoom to Window as required to see the line end points clearly.



- First, Click the Top end of the Top Vertical Reference Line,
- Then moving to the right, click the end of the line at .3750 " , .0000 " (The first short line)
- Then, moving down and to the Right, Click the end of the line at .7000 " , .4500 " (2nd Line)
- Moving Further down, Click the end of the line at .4000 " , 2.0500 " (3rd Line)
- Continuing down, Click the end of the line at .6000 " , 2.9500 " (4th Line)
- Next, Click the End of the line at .3750 " , 4.1250 " (5th Line down)
- Next, Click the end of the line at 1.1250 " , 6.8750 " (6th Line down)
- Then Click the end of the line at .6250 " , 9.1250 " (7th Line down)
- Now Click the end of the line at .2500 " , (9.3750 ") (8th Line down)
- Finally, Double Click the bottom end of the two vertical lines to complete the B-Spline.
- Now Click Select in the Top Icons to exit sketch and create Sketch<1>



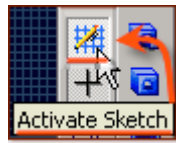
For a look at the whole sketch, [click here](#).

This completes the Sketch for the Sweep Path Object.

Now we will create the Sketch to Sweep Object.

If Still in **Sketch Mode**, Select **Activate Sketch**

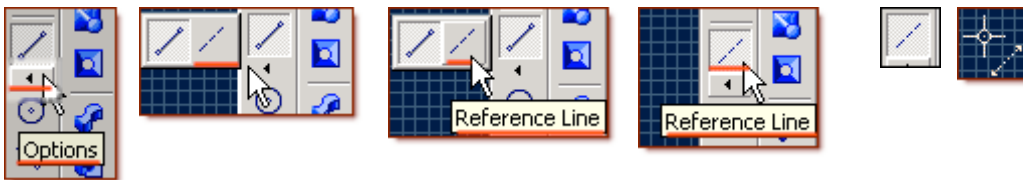
(While we are in the Sketch Mode - this is a toggle - to Exit Sketch Mode)



In the **Design Explorer**, Click **YZ-Plane**, then from the **Sketch Icons**, Click **Select** - to activate sketch and sketch on this plane.

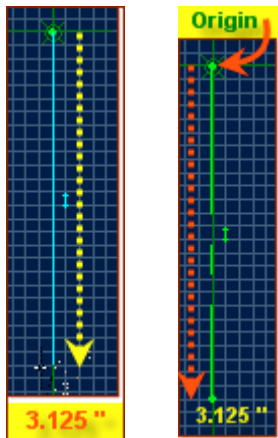


From the **Sketch Icons** > Click **Line Options** (*Below Line Icon*) -> Select **Reference Line**

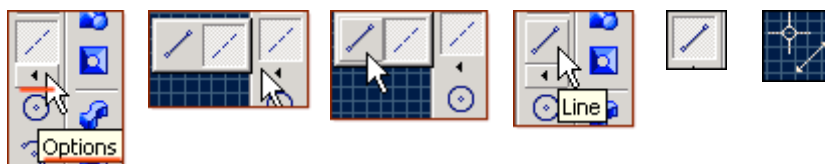


(Note: the Cursor Icon for the Reference Line - is broken/dotted.)

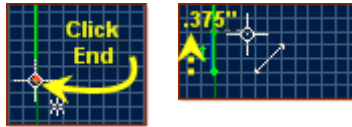
- Click the Origin, and drag a vertical Reference Line down 3.1250 " along the Y-Axis, Double Click.



From the **Sketch Icons** > Click **Line Options** (*Below Line Icon*) -> Select **Line** from the **Flyout**



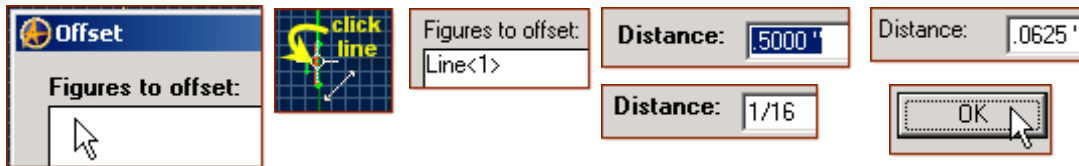
- Click the bottom end of the reference line; drag out a line .3750 " up the Y-axis, Double Click.



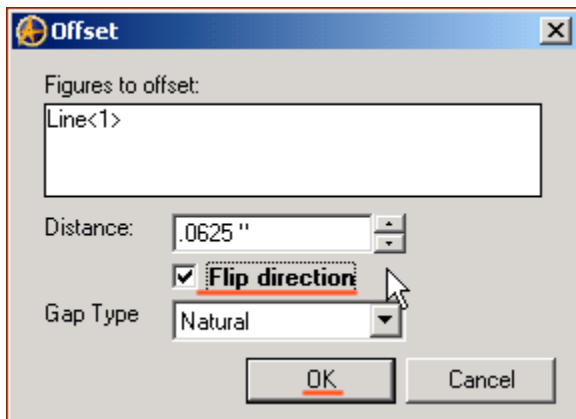
From the **Sketch Icons** > Click **Offset** (3rd Icon from the Bottom)



- Click in the white area under **Figures to Offset**, in the Offset dialog, the selection becomes **Bold**.
- Click the line just created to Select it, - Line<1> is displayed in the **figures to offset**,
- Click - Drag in the distance selection to highlight the value, the selection becomes **Bold**,
- Set the Distance to be 1/16 " (enter 1, enter '/' enter 16, press tab, value changes to become .0625),
- Click OK,

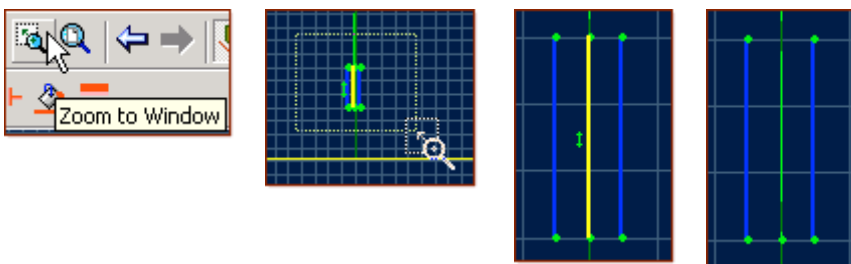


- Click the **Offset** Icon again, Line<1> is still selected, and distance is still .0625.
- This time Click - **Flip direction**, then click OK.



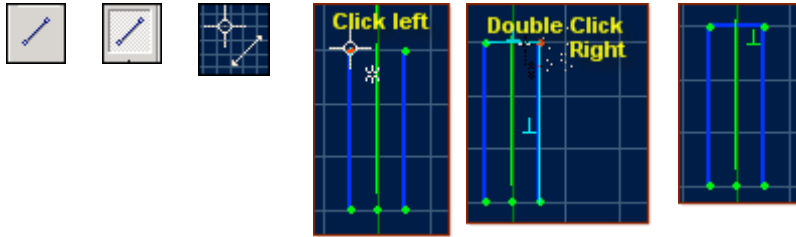
Use **Zoom to Window** to drag a selection around the two new lines, to get a close up view.

- Delete the center Line - if it is still selected (Yellow) or re-select it first, press the **Delete Key**.



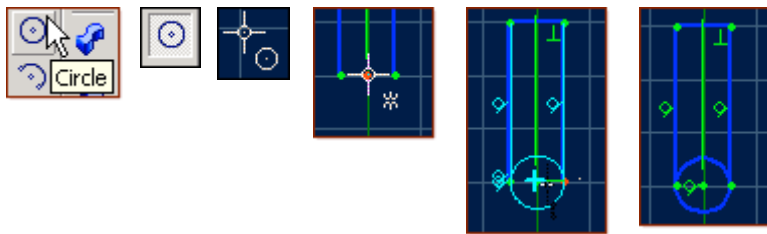
From the **Sketch Icons** > Click **Line** (The 5th Icon down) Connect these two lines.

- Click the upper end of the left line and the upper end of the right line, **Double Click**.



From the **Sketch Icons** > Click **Circle** (The 6th Icon down)

- Click the lower end of the reference line between the connected lines,
- Drag out a circle to the first grid intersection to the right, Click.



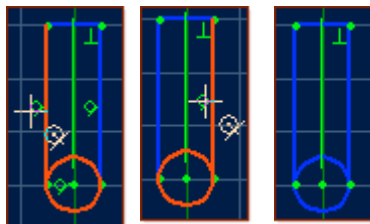
From the **Sketch Icons** > Click **Dimension** (4th Icon down)



- Click the center of the circle, and then the edge, you will notice **Tangent Constraints** are inserted, drag out the cursor to locate the dimension and Click.

From the **Sketch Icons** > Click **Select** (2nd Icon down)

- Mouse over the Tangent constraints, click and press **Delete** to remove them.



Again from the **Sketch Icons** > Click **Dimension** (4th Icon down)

- Edit the dimension to 3/32 and press Enter. The Displayed Dimension will be **R.0938**.



- Click the short line connecting the two parallel lines, drag the dimension location up and to one side, Click, Press **Enter** to accept the dimension (.1250 ")



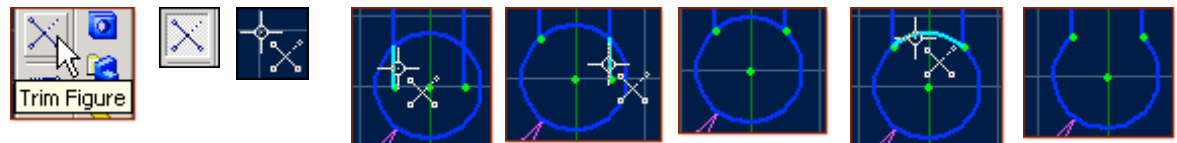
From the Sketch Icons > Click Select (2nd Icon down)

- Click the Reference Line to select it, and press **Delete**.



From the Sketch Icons > Click Trim Figure (6th Icon up from Bottom)

- Click on the excess tails of the parallel lines inside the Circle to trim them,
- Click the inside segment of the circle between the two parallel lines, to trim it.



From the Sketch Icons, Select Activate Sketch

(We are in the Sketch Mode - this is a toggle - to Exit Sketch Mode. It becomes unselected)

We now see Both Sketch<1> and Sketch<2> under Features in the Design Explorer.



This would be another good time to save the work done.

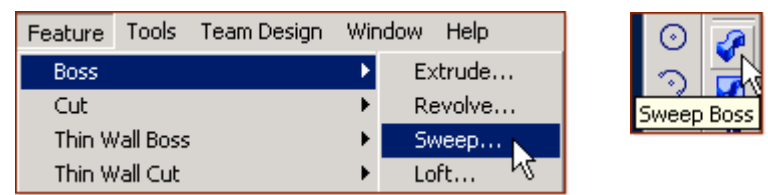
Simply Click the Save Icon (Disk). Click Save in the Dialog Box



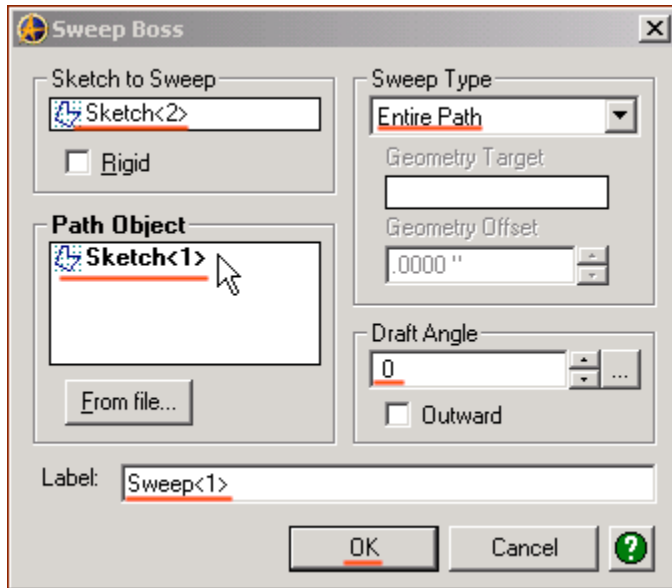
Next you will create the Models' first Solid Feature - a Sweep

From the Feature Menu Select Boss -> Sweep, or

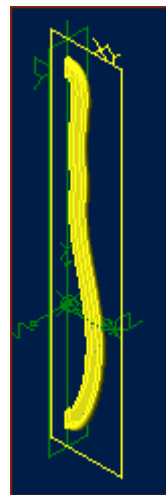
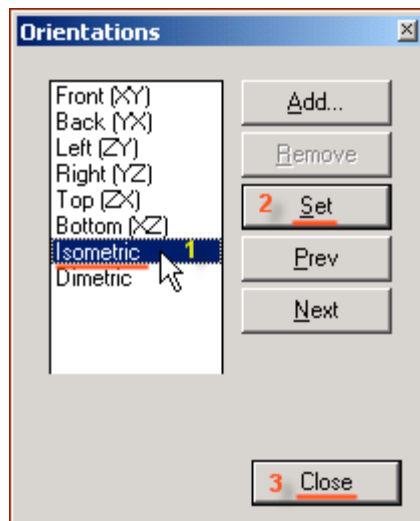
From the Feature Icons Select Sweep Boss. (7th Icon down)



- In the **Sweep Boss** dialog, Click inside the white area below **Sketch to Sweep** to select it if it is not already bold,
- Click **Sketch<2>** in the **Design Explorer**,
- Click in the white Space under **Path Object**, Click **Sketch<1>** in the **Design Explorer**,
- Leave Sweep Type set for **Entire Path**, Draft Angle at 0, and Label as **Sweep<1>**,
- Go ahead and Click **OK** to Complete the Sweep.



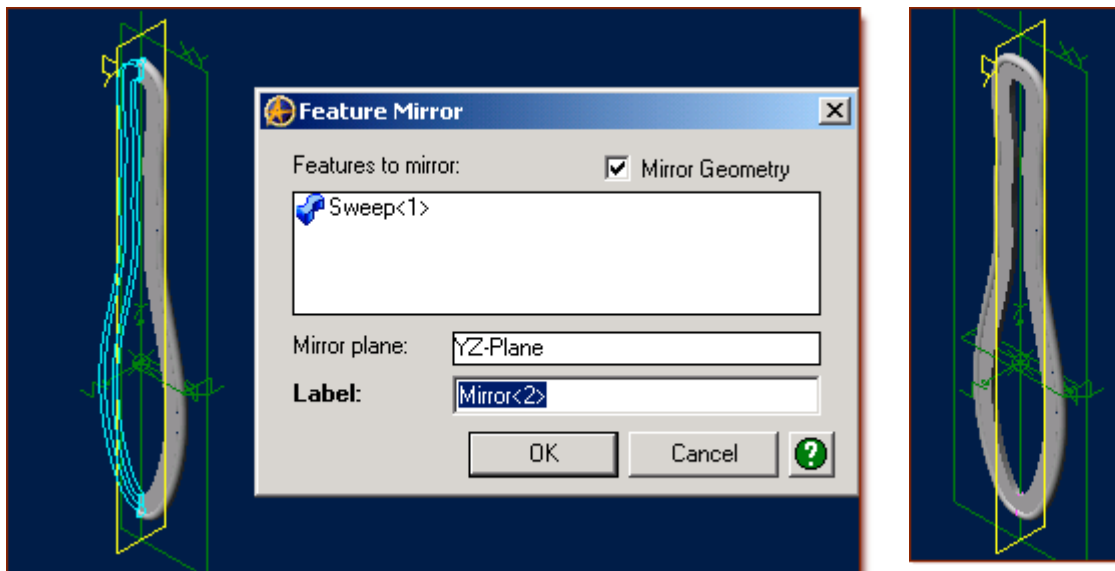
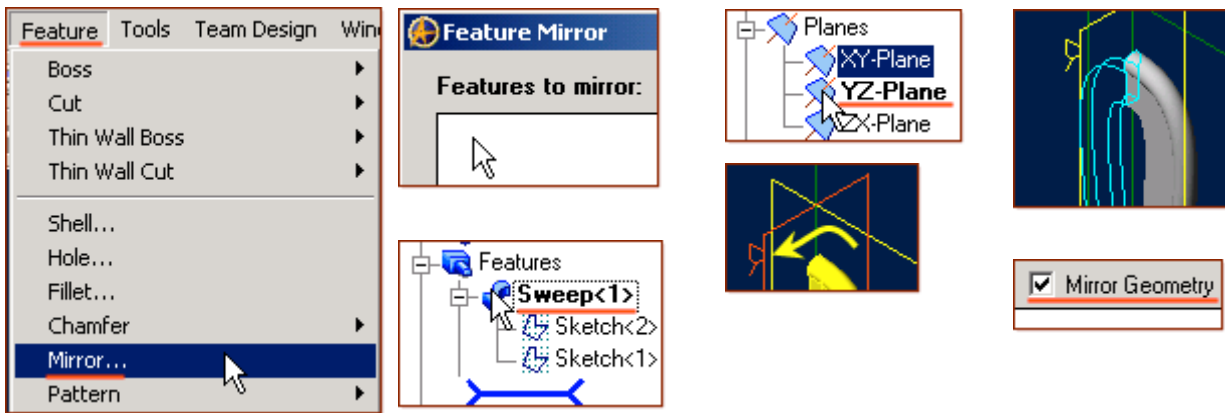
To see a better view of it, Click the **View Orientations** Icon, from the **View Toolbar**, and select **Isometric** from the list in the **Orientations** popup, Click **Set**, and then **Close** it. You can also Click **Orient to Isometric** from the **Orient View** Toolbar.



Now, We will create the other half of the sweep by using a Feature Mirror to close the shape.

From the Feature Menu, Select Mirror

- In the **Feature Mirror** dialog, click in the white space under **Features to mirror**:
- In the **Design Explorer**, Click feature **Sweep<1>** - it will be displayed in the **Feature to mirror** segment,
- Then click in the white space beside **Mirror plane**:
- Click the **YZ-Plane**, either in the **Design Explorer**, or in the **Part Workspace**,
- Quickly you will see the light blue wireframe preview of where the **Mirror Feature** will be,
- Make sure **Mirror Geometry** is checked, and Click **OK** to complete the Mirror operation.
- This will Create **Mirror<2>** (The Second Feature item in this Project, default numbering)



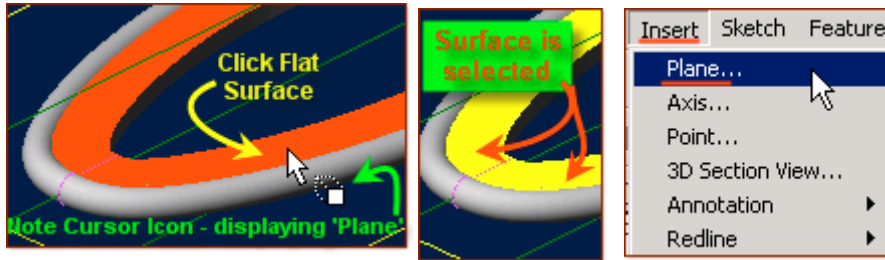
This Completes the Mirror Feature.

Next, we will use **Extrude Boss** to fill the middle portion left by creating the sweep first.

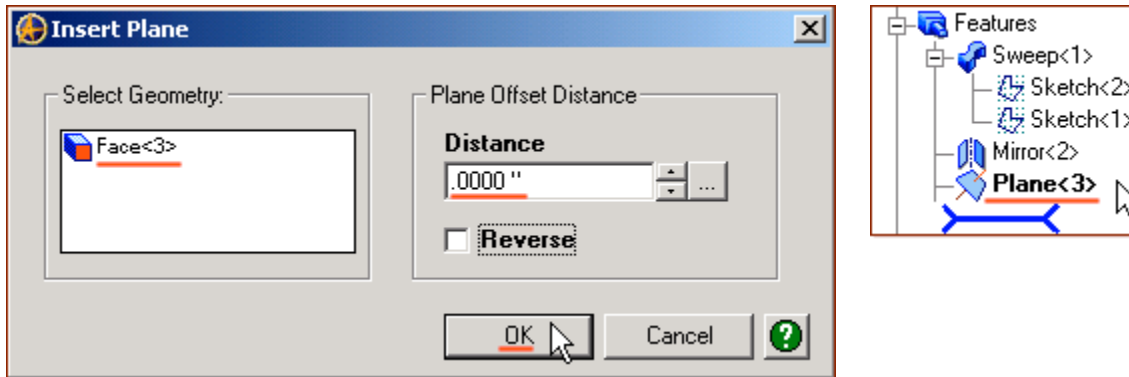
Start by using **Zoom to Fit**, or **Rotate** the view to a suitable angle and plane to allow an easy click to one of the flat inside surfaces on the original Sweep.



- Also, you can click left and right mouse buttons while the cursor is in the **Part Workspace** to rotate the view,
- Click the surface, then Select from the **Insert** menu, -> **Plane**,



- In the **Insert Plane** dialog, The **Select Geometry**: is pre-selected with **Face<3>**,
- Under **Plane Offset Distance**, Leave the **Distance** at **.0000 "**, and Click **OK**.
- **Plane<3>** is created and displayed in the **Design Explorer**, below **Mirror<2>**.
- Click in the open part workspace are to de-select planes and references.



Use this new Plane as the Sketch Plane now by selecting it in the **Design Explorer**, and clicking **Activate Sketch** in the **Sketch Icons**.

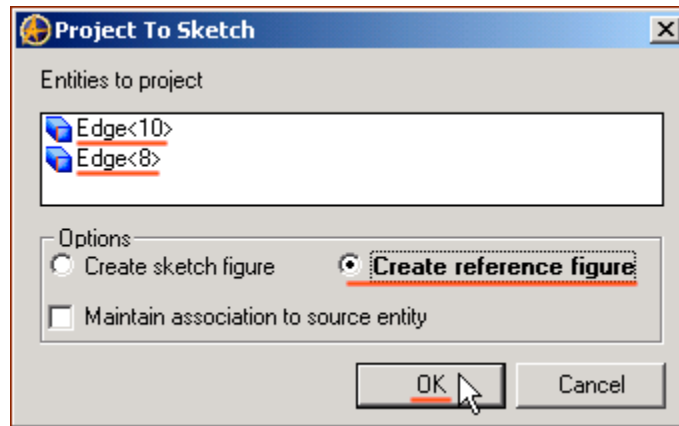
- When the Sketch is now active - you will see the grid over the **Sweep** and **Mirror** Features,



- Shift-Click the inside of the **Sweep** and the **Mirror** Feature Edges, (*Note the Cursor Icon for Edge Selection*). Then Click **Project to Sketch** Icon, (*Bottom of the Sketch Icons, it may also be selected from the Sketch Menu.*)

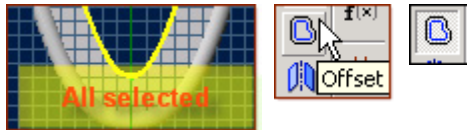


- Edge<8> and Edge<10> become displayed in the **Project to Sketch** selection popup under **Entities to project**,
- Default is **Create sketch figure** - change that to the selection beside **Create reference figure**,
- Click **OK**.

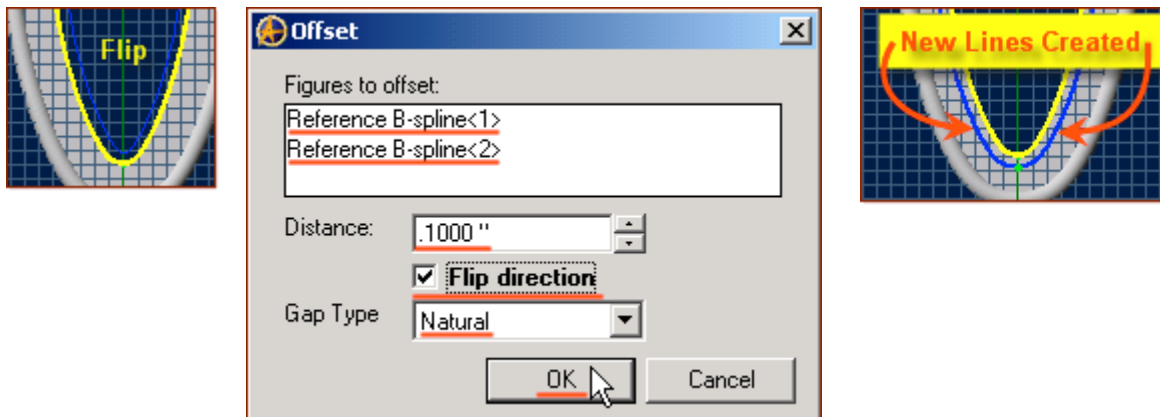


Now what we will do - is offset the edges into the sweep and mirror areas using **Offset**.

Press **Ctrl+A** (*Control key and the 'A' key at the same time*) to **Select All** the sketch items
From the Sketch Icons, Click - Offset (*3rd icon from the bottom*)



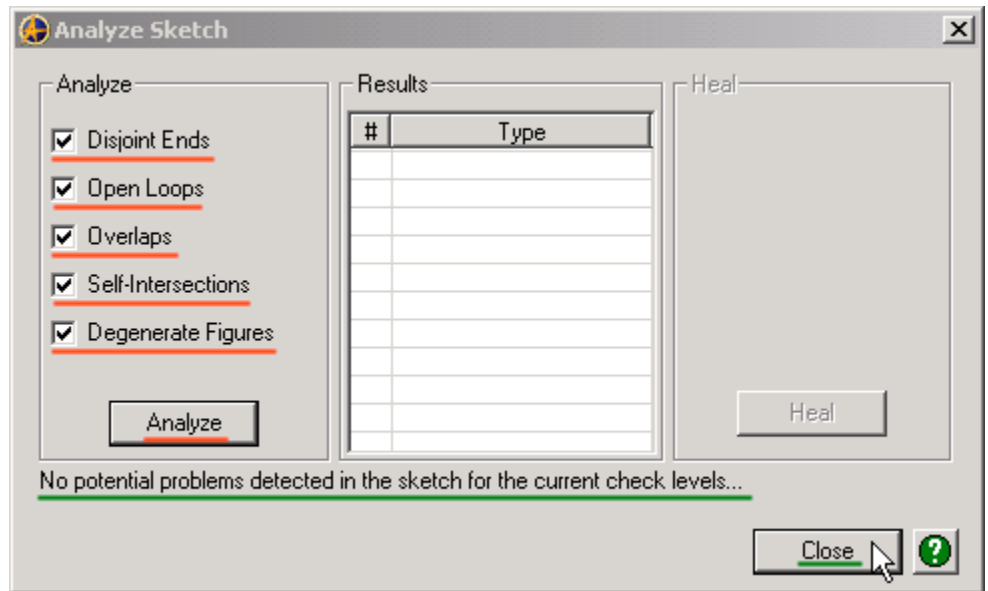
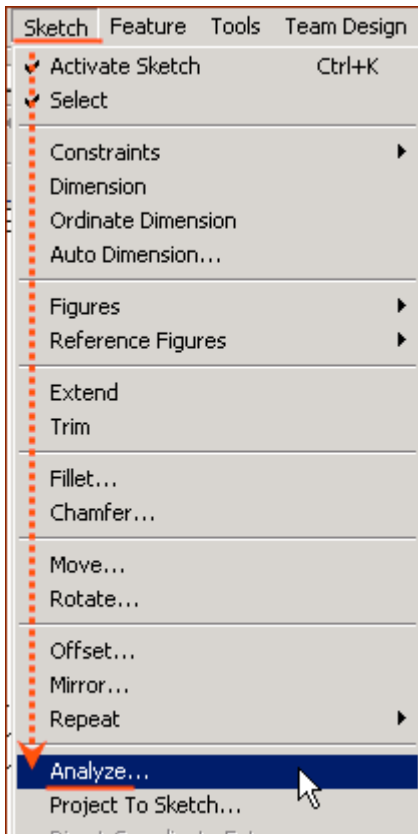
- From the **Offset** selection dialog, you should see in **Figures to offset** - two items already selected: **Reference B-Spline<1>**, and **Reference B-Spline<2>**
- Edit the **Distance** to be **.1000 "** and make sure the blue line is over the existing **Sweep** and **Mirror**,
- If not - Click the selection beside **Flip Direction**. Leave **Gap Type** at **Natural**, and Click **OK**.



Now you can check for any issues with the lines, using the analyse feature.

From the Sketch Menu > Select Analyze

- The Analyze Sketch popup has a number of selections to check - marked with boxes beside them - for this exercise - they all should be checked,
- Click the Analyze Button - Ideally you should see a message below this button that is now displayed stating "No Potential problems detected in the sketch for the current check levels..."
- If not - (IE: Disjoint Ends), Click the error, and it will be displayed in the sketch, and if it can be Healed automatically, the Heal button will no longer be Greyed out. Click it to fix the error.
(That is why you want them all checked in the boxes!) ([Email me if you get stuck on this one](#))
- Click Close.



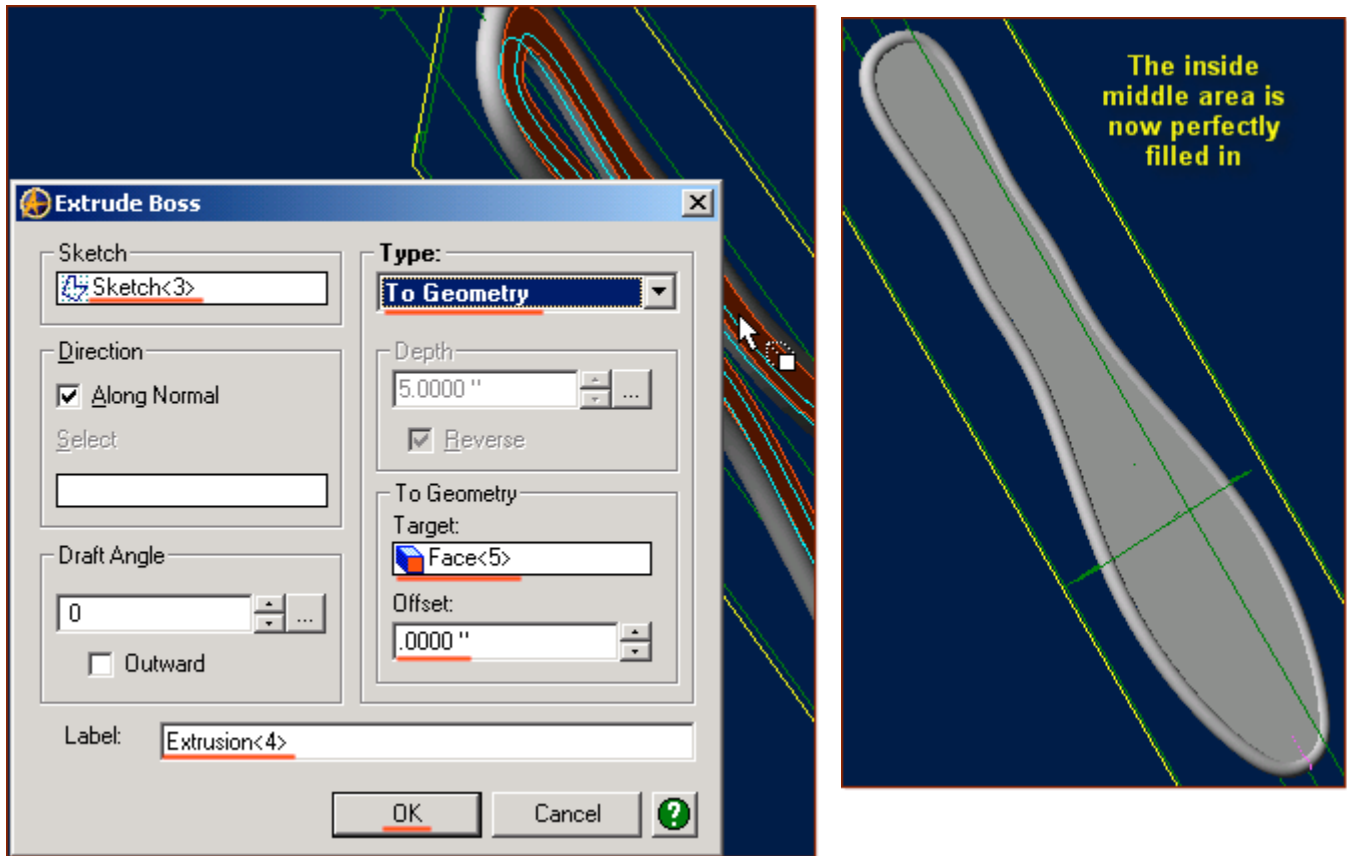
Now you can make the Extrusion to fill the center area of the handle.

While in the Sketch mode - Click Boss, Extrude, under the Feature menu, or Click Extrude Boss from the Features Icons




- Unless used recently in this project with another setting or distance, the Extrude Boss selection popup will usually open with a default distance of 5" (5.0000 " with the current properties setting)
- Pre-selected - under Sketch will be Sketch<3>, the sketch we just made,
- Set the Type: To Geometry, Leave Direction set to Along Normal, and Draft Angle set to 0,


- Click left and right mouse buttons while the cursor is in the Part Workspace to rotate the view of the sweep/mirror item until you see the opposite side, or back face from where we began,
- Click in the **To Geometry Target:** selection area, then Click this back face for the 'To Geometry' setting,
- **Face<5>** is now displayed in this selection. Leave **Offset** at **.0000 "**, and **Label:** as **Extrusion<4>**,
- Click **OK** -when all things are set up and confirmed to complete the extrusion fill of the Middle.



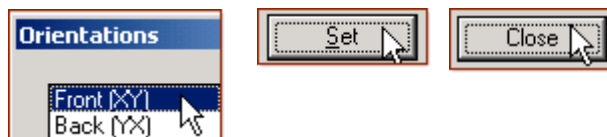
One thing left to Model, the shaped hole in the handle.

While we might be able to edit the last extrusion and simple modify the sketch there, for this exercise - we will create a new sketch on the new surface, and use **Extrude Cut** to create the hole.

First - Click **Orient to Front** from the **Orient View** toolbar,  or

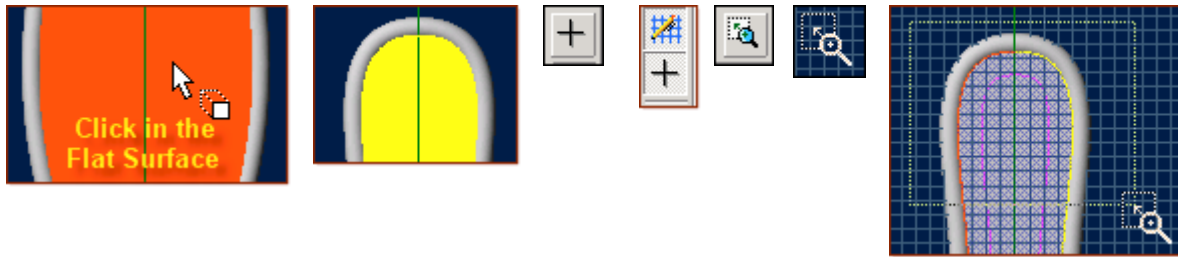
Click **View Orientation**  from the View toolbar, and

Select: **Front(XY)**, **Set**, **Close**.

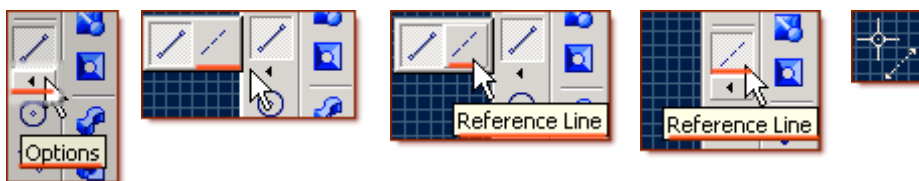


Click in the all Flat area between the Circular Edges of the Foot Scrubber Model to Select it, then click **Select** from the **Sketching** toolbar (**Sketch Icons**).

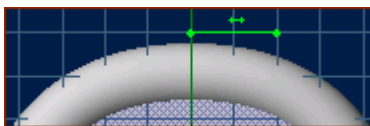
- Now in **Sketch** mode - use the **Zoom to Window** to zoom in on the slim handle area (Top) of the view,



From the **Sketching** toolbar > Click **Line Options** (Below **Line Icon**) -> Select **Reference Line** (Note: the Cursor Icon for the Reference Line - is broken/dotted.)

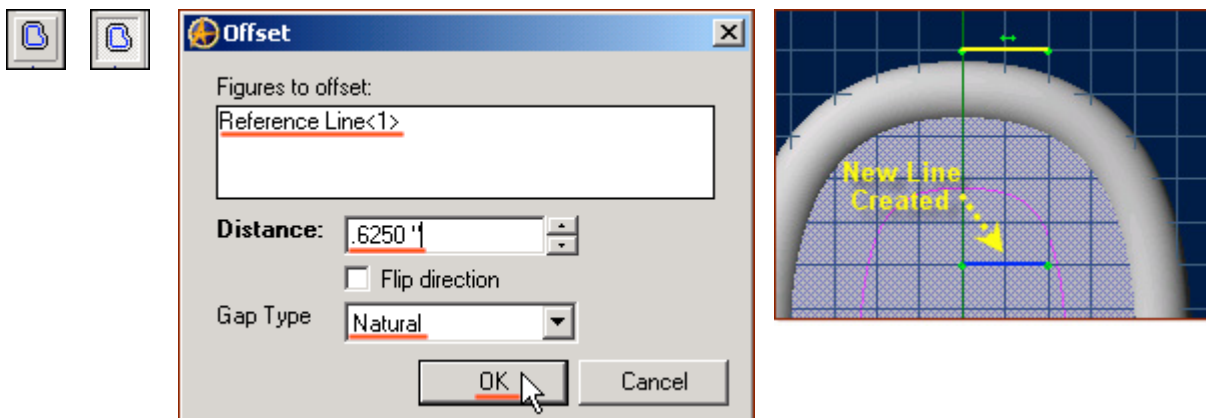


- Click the **Y-Axis** to set the start of the reference line - just above the edge of the **Sweep**, and place a horizontal line along the grid, about 0.25" Long. **Double Click**.



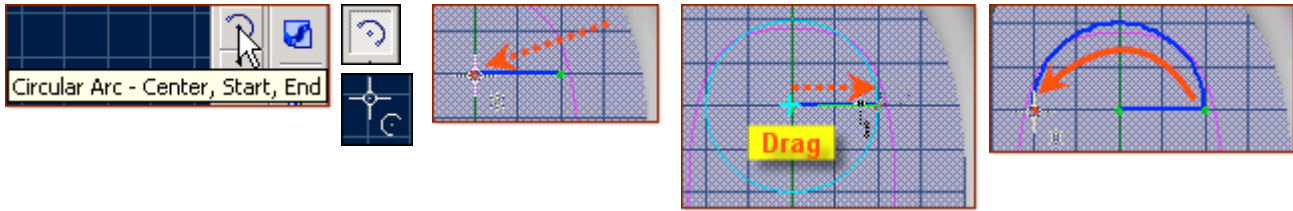
From the **Sketching** toolbar, Click - **Offset** (**3rd icon from the bottom**)

- In the **Offset** selection popup, Click in the white area under **Figures to offset**: and click the new reference line.
- Reference Line<1>** is displayed now in **Figures to offset**,
- Set the **Distance**: to be **.6250 "**, Click **Tab**, Leave **Gap Type** at **Natural**, and Click **OK**.



From the **Sketching** toolbar, Click - **Circular Arc, Center, Start, End** (*7th Icon down, default*)

- Click the end of the newly offset line that is on the **Y-Axis**,
- Drag to the right 2 grid squares (.2500") and click on the grid to start the **arc**,
- Drag up and over the center and down to the other side for a 180-degree arc, Click to complete it.

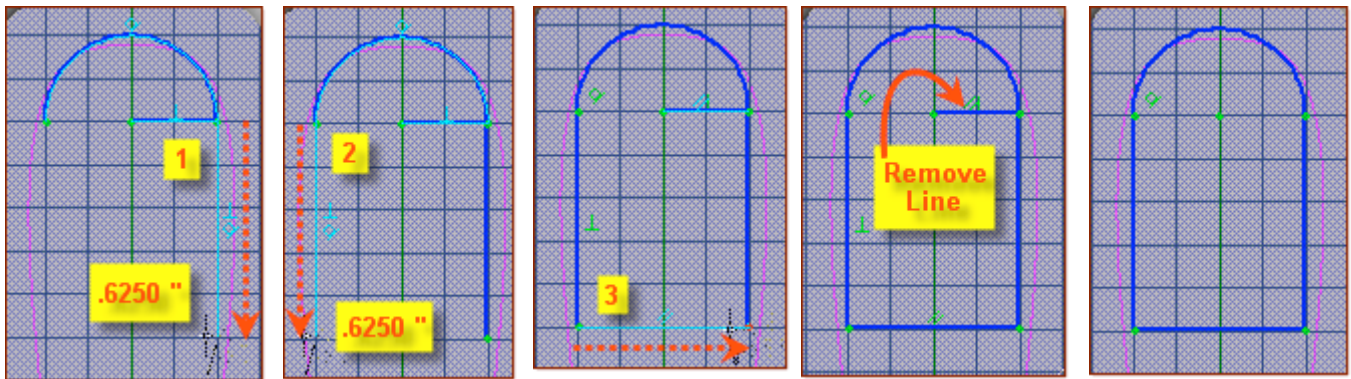


From the **Sketching** toolbar, Click **Line** (*The 5th Icon down*)

If Required, Click **Line Options** (*Below Reference Line Icon*) -> **Select Line**

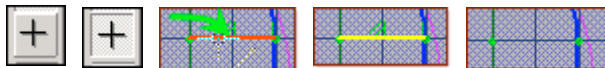


- Click the beginning point of the arc and drag down for a line length of .6250 ", Double Click,
- Click the ending point of the arc and drag down for a line length of .6250 ", Double Click,
- Click the Bottom end of this line, and drag to the right and Double Click on the Bottom end of the previous Line.



From the **Sketching** toolbar, Click - **Select** (*2nd Icon*)

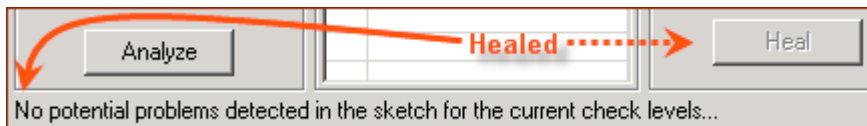
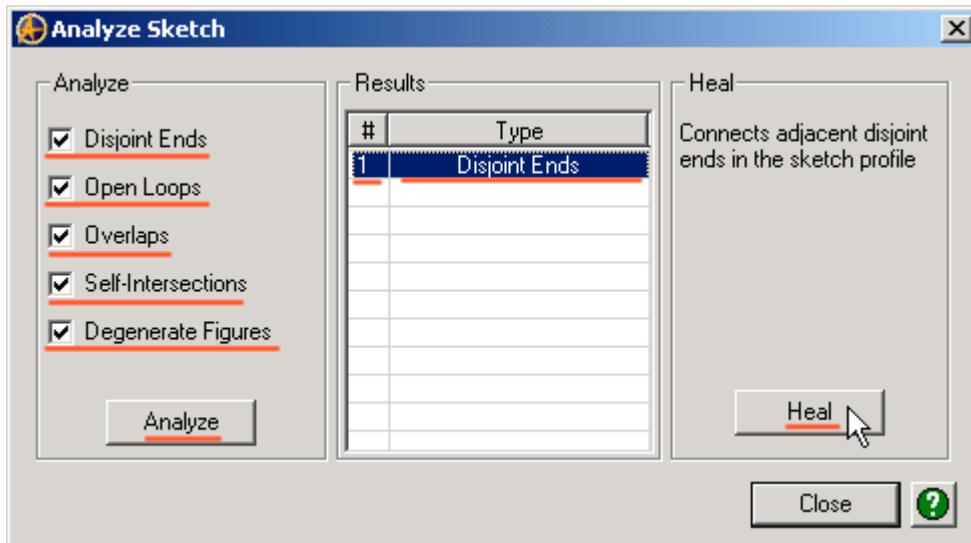
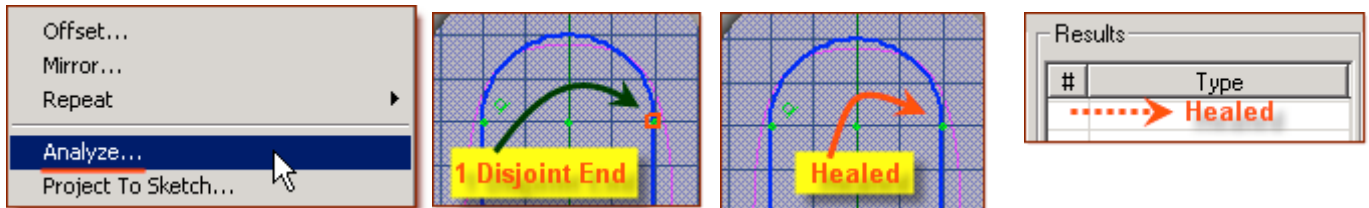
- Click the Line that intersects with the arc's Radius and goes out to the right to select it,
- Press **Delete**, to remove it.



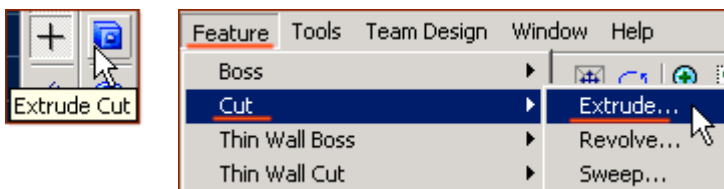
Next, you will check the sketch with **Analyse**.

From the Sketch Menu > Select Analyze

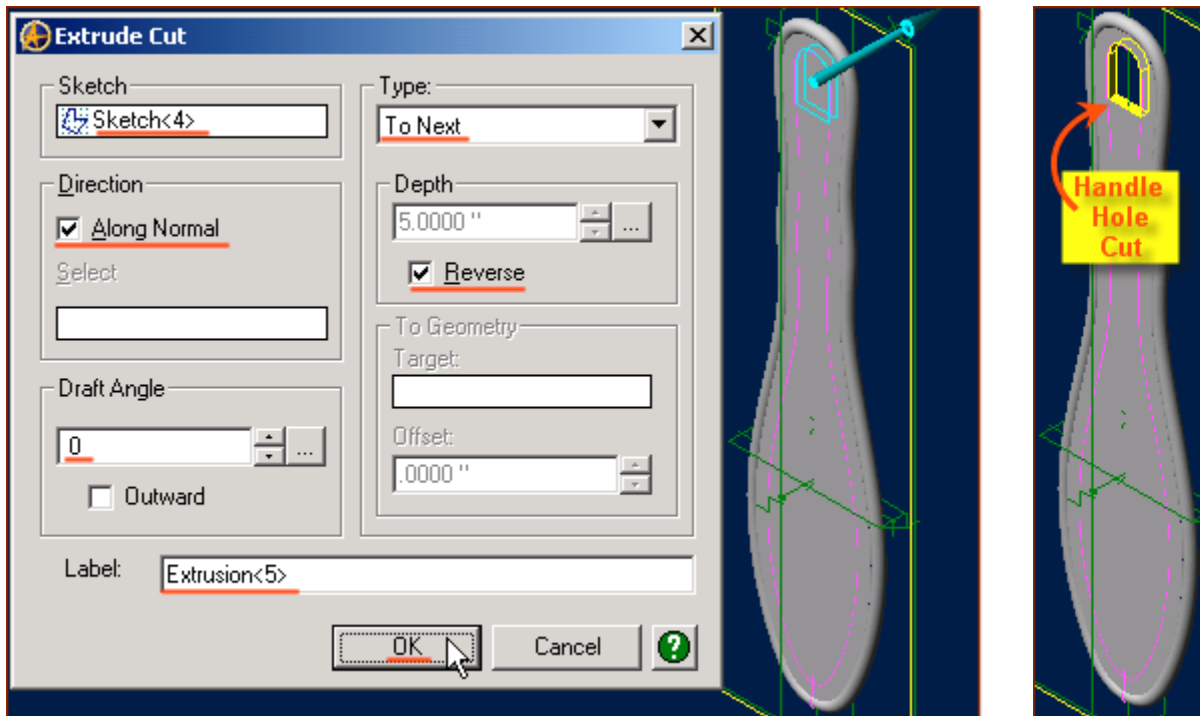
- The **Analyze Sketch** dialog has a number of selections to check - marked with boxes beside them - for this exercise - they all should be checked,
- Click the **Analyze** Button - Ideally you should see a message below this button that is now displayed stating "No Potential problems detected in the sketch for the current check levels..."
- If not - (IE: Disjoint Ends), Click the error, and it will be displayed in the sketch, and if it can be Healed automatically, the Heal button will no longer be Greyed out. Click it to fix the error.
(That is why you want them all checked in the boxes!) ([Email me if you get stuck on this one](#))
- Click **Close**.



While still in Sketch Mode, Select Extrude Cut from the Feature Icons, or Cut, Extrude from the Feature Menu.



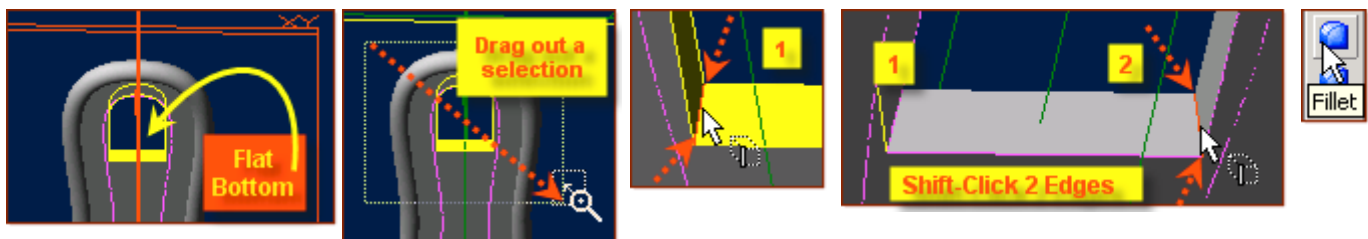
- From the **Extrude Cut** dialog, **Sketch<4>** is pre-selected,
- Change **Type:** to **To Next**, Leave **Direction** as **Along Normal**, **Draft Angle** at **0** degrees,
- Select **Reverse** if required, under **Depth**,
- **Label:** is **Extrusion<5>**, which is fine. Click **OK** to complete the Cut.



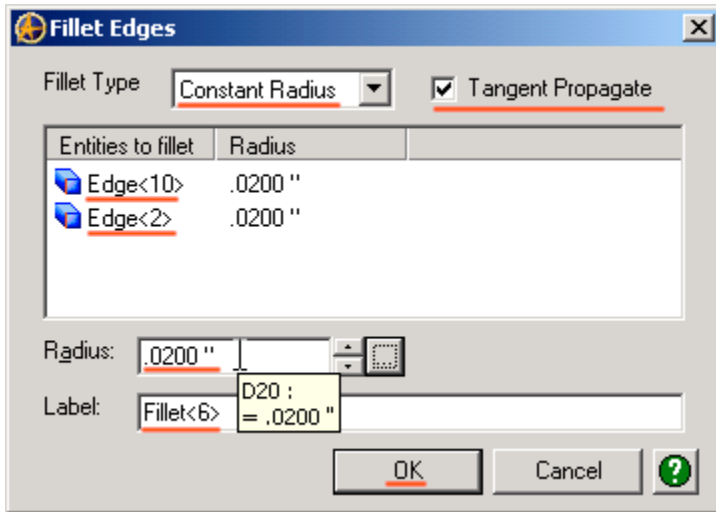
Great! You have cut the Hole - now lets clean it up and smooth the edges!

Left + Right Mouse Click and drag to Rotate the handle around to see the flat bottom inside of the cut we just made. Select Zoom to window and do a close-up view of this area.

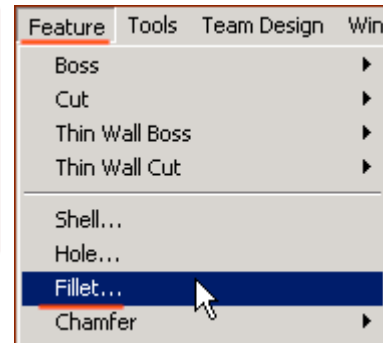
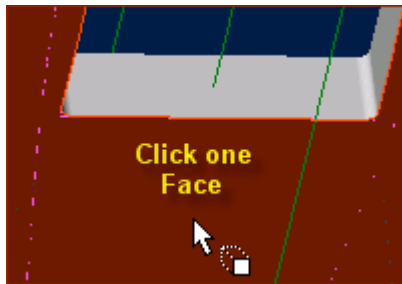
While in feature mode (not in sketch) - Shift-Click the lines at the ends of the Flat at the bottom of the cut, then select **Fillet. (Slightly move/rotate the handle to see both ends of the flat, so as to click both lines)**



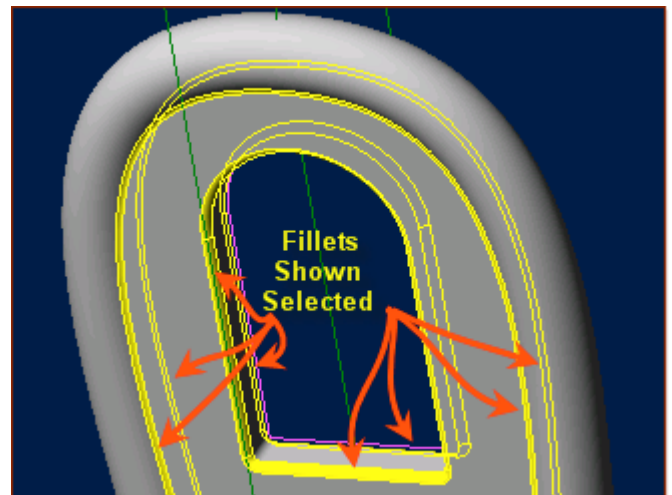
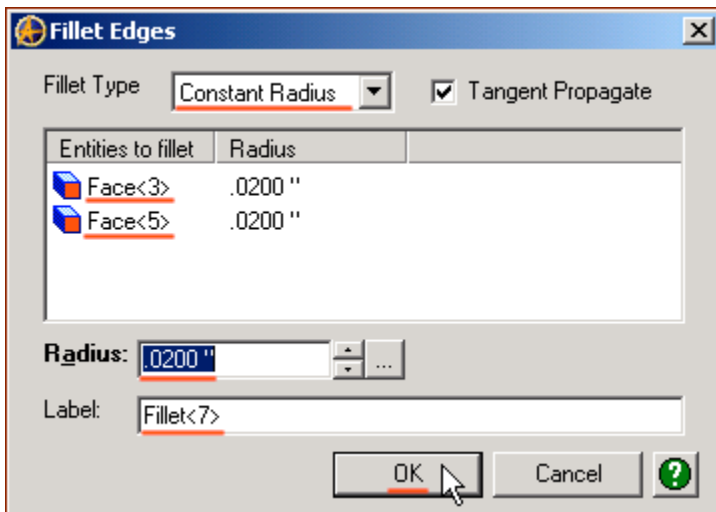
- In the **Fillet Edges** dialog, Leave **Fillet Type** set to **Constant Radius**, and **Tangent Propagate** Checked,
- In the **Entities to Fillet**, should be showing **Edge<2>** and **Edge<10>**,
- Set the **Radius:** to **.0200 "** - Click Tab,
- The **Label:** is **Fillet<6>**, which is fine, Click **OK** to Complete the Base Fillet Radius.



- Now Click the two Flat surfaces upper and lower (Rotate the part around to get both sides selected, remember to shift-click the second side)
- Again, Select Feature Fillet.

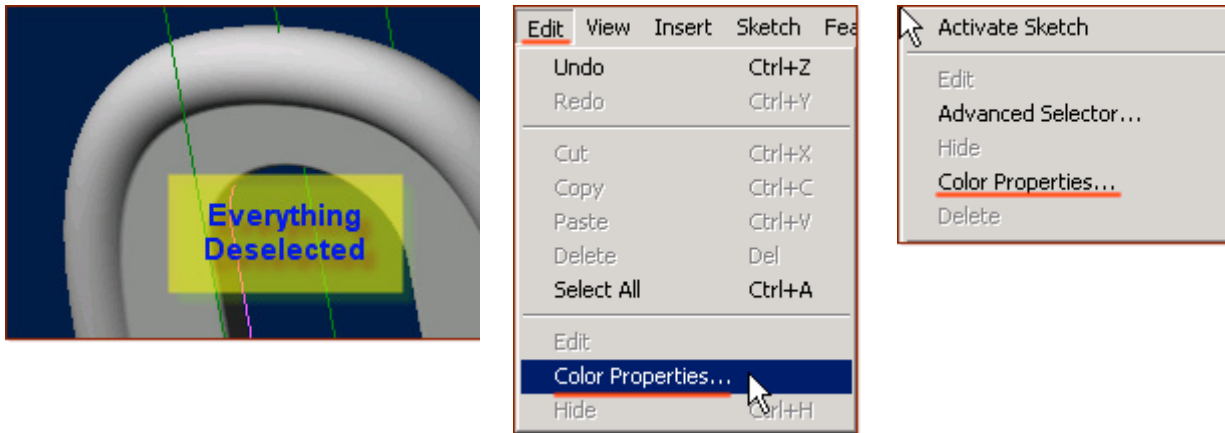


- In the Fillet Edges dialog, Leave Fillet Type set to Constant Radius, and Tangent Propagate Checked,
- In the Entities to Fillet, should be showing Face<5> and Face<3>,
- Leave the Radius: set at .0200 ", and Label: defaulted at Fillet<7>. Click OK to complete.

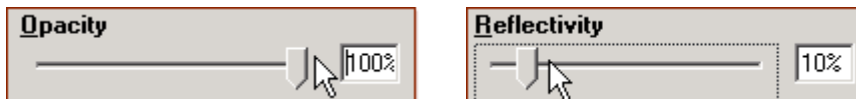


Super! Much Nicer! Now let's give it some Color, and re-save it!

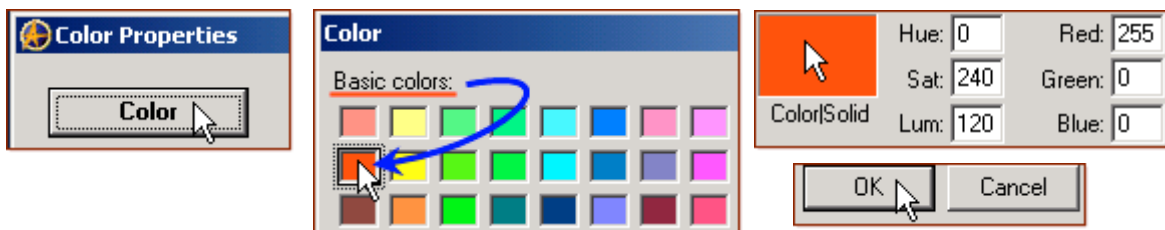
First - Click in the Workspace in an open area to deselect the part and any features.
 From the **Edit** Menu - Select **Color Properties**, or
 Right-Click in the part workspace, and Select **Color Properties** from the menu.



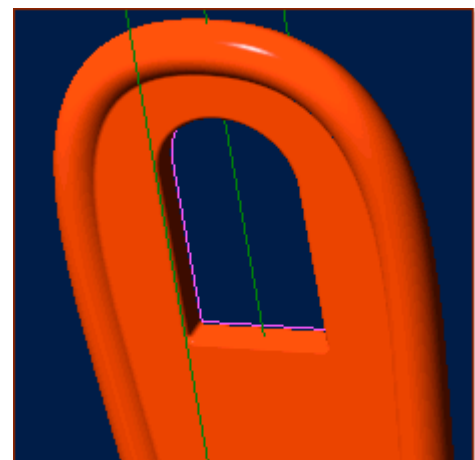
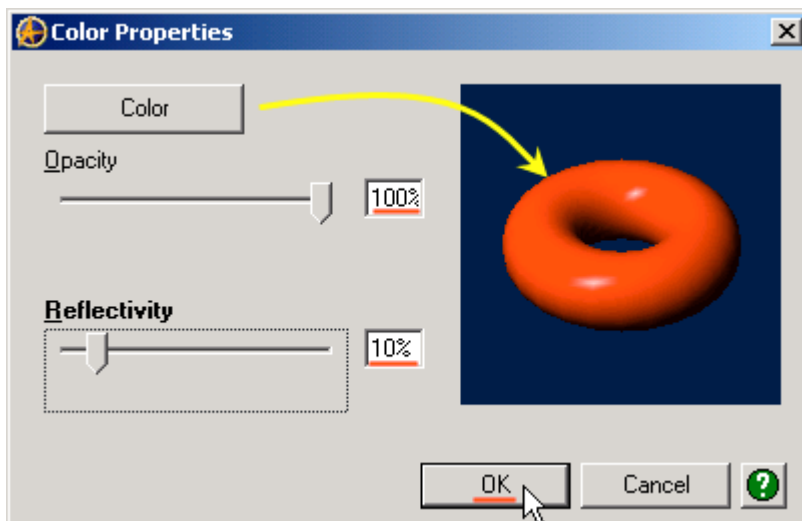
- Leave **Opacity** at 100%, Set **Reflectivity** to 10% by Clicking to the right of the slider once,



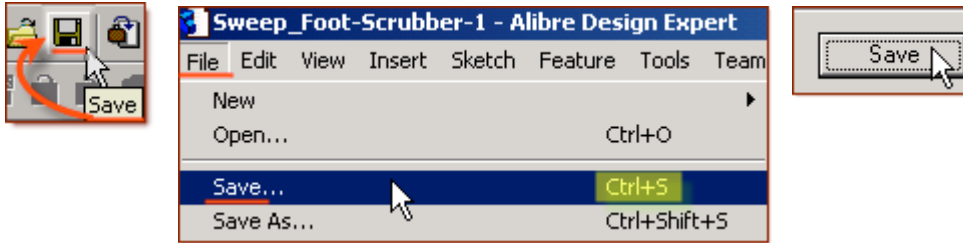
- Select the **Color** button - and from the **Basic Colors** - click the First Column, 2nd Row - **Red** Color,
- Confirm the color values, and Click **OK** to accept the Color and close the **Color** selector,



- Then Click **OK** to Close the **Color Properties** dialog and accept the settings of **Color**, **Reflectivity**, and **Opacity**.



Finally, Click on the **Save Icon** (Floppy Disk) or select **File, Save (Or Ctrl+S)** and click **Save** in the **Save** dialog to update our save of the file..



Congratulations!

You have completed the Sweep_Foot-Scrubber-1 Tutorial!
(Complete Version)

Give me feedback on this tutorial!
[Send email](#)

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