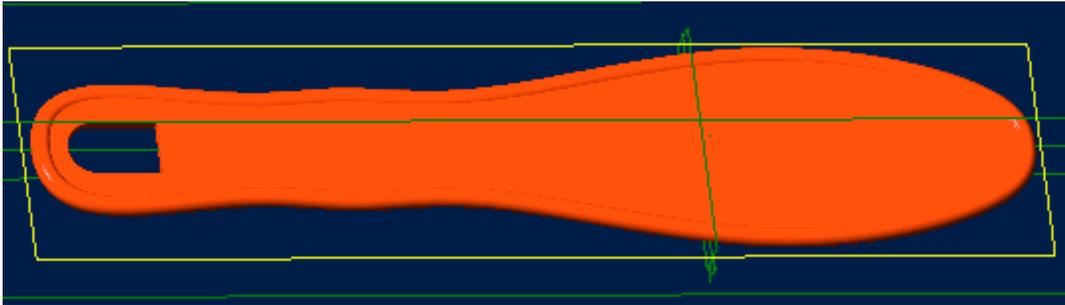


## Alibre Design Tutorial: Sweep, Extrude (Boss & Cut), & Fillet Sweep\_Foot-Scrubber-1



### Part Tutorial

#### Exercise 8: Sweep Foot-Scrubber-1 [text 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*)

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

Maximise the Part Workspace Window.

#### Set Design Properties.

##### Select **File, Properties**.

##### 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 0.10" and for **Angle**: enter 1 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 >**

**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.)*

## 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 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**.

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

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

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

- 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 2<sup>nd</sup> short line. 70 " Long, .45 " (about 4 grid squares) below the first line,
- The 3<sup>rd</sup> short line .40" Long, 1.60" (about 12 grid squares) below the second line,
- The 4<sup>th</sup> short line .60 " Long, 0.90 " (about 6-7 grid squares) below the third line,
- The 5<sup>th</sup> short line .375" Long, 1.175" (about 9 grid squares) below the fourth line,
- The 6<sup>th</sup> short line 1.1250" Long, 2.750" (22 grid squares) below the fifth line,
- The 7<sup>th</sup> short line .625" Long, 2.25" (18 grid squares) below the 6<sup>th</sup> line  
(2 square up from the bottom end of the bottom line)
- The 8<sup>th</sup> short line .25" Long running from the bottom end of the bottom line out to the right.

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

- 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.

**This has complete 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 top Icons, 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: Click the small horizontal reference line at the top,
- 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 on down clicking the end points and dragging the lines to one side until finishing at the 8<sup>th</sup> 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 " (~.450 ") and edit it to .4500 ", Click Enter
- Double click the next down Ordinate Dimension (~ 2.00 ") 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 (~ 4.00 ") and edit it to be 4.1250 ", Click Enter,
- Double click the next down Ordinate Dimension (~ 7.00 ") and edit it to be 6.8750 ", Click Enter,
- Double click the next down Ordinate Dimension (~ 9.100") and edit it to be 9.1250 ", Click Enter,
- The last Ordinate Dimension should be in Brackets and should read (9.3750 ") - meaning it is fixed.

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

**From the Sketch Icons > Click B-Spline by Control Points (9<sup>th</sup> Icon down)**

- Use Zoom to Fit, and Zoom to Window as required to see the line end points clearly.
- First, Click the Top end of the vertical Reference Lines,
- 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 " (2<sup>nd</sup> Line)
- Moving Further down, Click the end of the line at .4000 " , 2.0500 " (3<sup>rd</sup> Line down)
- Continuing down, Click the end of the line at .6000 " , 2.9500 " (4<sup>th</sup> Line down)
- Next, Click the End of the line at .3750 " , 4.1250 " (5<sup>th</sup> Line down)
- Next, Click the end of the line at 1.1250 " , 6.8750 " ( 6<sup>th</sup> Line down)
- Then Click the end of the line at .6250 " , 9.1250 " (7<sup>th</sup> Line down)
- Now Click the end of the line at .2500 " , (9.3750 ") (8<sup>th</sup> 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>

**This completes the Sketch for the Sweep Path Object.  
Now we will create the Sketch to Sweep Object.**

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

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

- 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**

- 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 (3<sup>rd</sup> Icon from the Bottom)**

- Click in the white area under Figures to Offset, in the Offset popup,
- Click the line just created to Select it, - Line<1> is displayed in the figures to offset,
- Set the Distance to be 1/16 " (enter 1, enter '/' enter 16, press tab, it changes to become .0625),
- Click OK,
- Click the Offset Icon again, Line<1> is still selected.
- This time Click - Flip direction, then click OK.

**Use Zoom to Window to get a close up view of the two new lines.**

- 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 5<sup>th</sup> 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 6<sup>th</sup> Icon down) Connect these two lines.**

- 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 (4<sup>th</sup> Icon down)**

- Click the center of the circle, and then the edge, drag out the place to locate the dimension and Click,
- 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 (2<sup>nd</sup> Icon down)**

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

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

- Click on the excess tails of the parallel lines inside the Circle,
- Click the inside segment of the circle between the two parallel lines, over the reference line.

**From the Sketch Icons, Select Activate Sketch**

*(We are in the Sketch Mode - this is a toggle - to Exit Sketch Mode)*

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

**From the Feature Menu Select Boss -> Sweep, or from the Feature Icons Select Sweep Boss**

- In the Sweep Boss popup, 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 Top Icons, and select Isometric from the list in the Orientations popup, Click Set, and then Close it.**

**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 popup selector, click in the white space under Features to mirror:
- In the Design Explorer, Click Sweep<1> - it will be displayed in the Feature to mirror segment,
- Then in the Feature Mirror popup selector, 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 extrude fill the middle portion left by creating the sweep first.**

**Start by Rotating the view to a suitable angle and plane to allow an easy click to one of the flat inside surfaces on the original Sweep.**

- 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](#) selection popup, 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.
- 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,
- Click the inside of the Sweep and Mirror Feature Edges, then Click the [Project to Sketch](#) Icon,
- [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](#) (*3<sup>rd</sup> icon from the bottom*)**

- From the [Offset](#) selection popup, 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](#).

**From the [Sketch Menu](#) > Select [Analyse](#)**

- The [Analyse Sketch](#) popup has a number of selections to check - marked with boxes beside them - for this exercise - they all should be checked,
- Click the [Analyse](#) 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..." (*That is why you want them all checked in the boxes!*) ([Email me if you get stuck on this one](#))
- A Common issue after a mirror - is [Disjoint Ends](#). Click on this listed item, then click the [Heal](#) Button to correct it. Then you should get 'No Potential Problems detected....'.

**While in the [Sketch mode](#) - 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 this 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 simply 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.

Click in the all Flat area between the Circular Edges of the Foot Scrubber Model to Select it, then click Select from the 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 Sketch Icons > Click [Line Options \(Below Line Icon\)](#) -> Select [Reference Line](#)

- 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.

From the Sketch Icons, Click - [Offset \(3<sup>rd</sup> icon from the bottom\)](#)

- In the Offset selection popup, Click in the white area under [Figures to offset:](#) and click the new reference line.
- [Reference Line<2>](#) 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 Sketch Icons, Click - [Arc \(7<sup>th</sup> Icon down\)](#)

- 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 Sketch Icons > Click [Line \(The 5<sup>th</sup> Icon down\)](#)

If Required, Click [Line Options \(Below Line Icon\)](#) -> 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 Sketch Icons, Click - [Select \(2<sup>nd</sup> Icon\)](#)

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

From the Sketch Menu > Select [Analyse](#)

- The Analyse Sketch popup has a number of selections to check - marked with boxes beside them - for this exercise - they all should be checked,
- Click the [Analyse](#) 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..." (*That is why you want them all checked in the boxes!*) ([Email me if you get stuck on this one](#))

While still in Sketch mode, Select [Extrude Cut](#) from the Feature Icons, or [Cut, Extrude](#) from the Feature Menu.

- From the [Extrude Cut](#) selection popup, [Sketch<4>](#) is pre-selected,
- Change [Type:](#) to [Through All](#), Leave [Direction](#) as [Along Normal](#), [Draft Angle](#) at [0](#) degrees,
- [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 selection popup, Leave Fillet Type set to Constant Radius, and leave 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, In the Fillet Edges selection popup, Leave Fillet Type set to Constant Radius, and leave 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!**

**From the Edit Menu - Select Color Properties.**

- 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, 2<sup>nd</sup> Row - Red Color,
- Click OK to accept the Color and close the Color selector,
- Then Click OK to Close the Color Properties popup and accept the settings of Color, Reflectivity, and Opacity.
- Click on the Save Icon (Floppy Disk) or select File, Save (Or Ctrl+S) to update our save of the file.

**Congratulations!**

**You have completed the Sweep\_Foot-Scrubber-1 Tutorial!**

*(Text Version)*

Give me feedback on this tutorial!

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