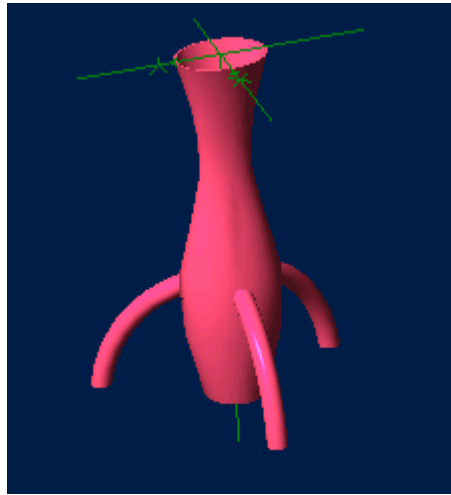


Alibre Design Tutorial: Loft, Sweep, Pattern, & Shell Loft-Vase-1



Part Tutorial

Exercise 6: Loft-Vase-1 [text version]

In this Exercise, We will set System Parameters first, then part options. Then, in sketch mode, we will set the first four sketches of the vase for the loft, and loft them. Then the Sweep Feature to create the Legs, and Feature Pattern to create the three of them. Then Shell to hollow out the vase and the legs. Setting the Color Properties will follow this. Finally, we will insert and view a 3D Section, and Save the Loft-Vase-1 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: 'Loft-Vase-1'.
- In **Curve Smoothness**, Select the Radio Button marked 'Manual'.
- Set **Minimal Circular Facets**: to 36.

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.
- 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**.

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: 0.5 mm and Y: 0.5 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.)

Create the First Loft Sketch.

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

From the Main Menu, Select Sketch> **Figures**> **Ellipse**.

- As per the hint line at the bottom of the workspace "*Pick a Center point for the ellipse*" - Click on the Origin.
- As per the hint line "*Pick next point for the major axis of the ellipse*" - drag up the Y-Axis to a distance of 4.5 mm, Click,
- As per the hint line "*Pick next point for the minor axis of the ellipse*" - drag right along the X-Axis to a distance of 3.5 mm, Click. Press Control+A (Ctrl-A) to select the Ellipse created.
- From the **Sketch Icons**, Click **Offset**. In the Offset Popup, **Figures to offset:** should be defaulted at **B-spline<1>**
- Set **Distance:** to 2 mm, and if new line is not inside selected ellipse, select **Flip direction**.
- Leave **Gap Type** as **Natural**, and Click **OK**.
- Click the new ellipse, to select it, and press Ctrl-X to delete it and place it on the clipboard.

From the top Icons, click on **Select** *(Beside Undo and Redo Icons)* to exit sketch mode.

This Completes Sketch<1>

Create the Second Loft Sketch.

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

- Left Click the **XY-Plane** in the **workspace** or in the **Design Explorer**.
 - Then, Right-Click in the workspace open area (away from any selectable planes, axis, etc.)
 - Select **Insert Plane** from the menu.
 - 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 10 mm (*press Tab*)
- Click **OK**. Then Click off in the workspace again, to de-select it. **Plane<1>** is created.

Next - Create the Sketch for the second part of the Lofted Vase. (Select Plane<1>)

- Click the new Plane (Plane<1>) and from the Sketch Icons, Click Select.
- When the Sketch Workspace Opens, press Ctrl-V to paste the previously deleted offset ellipse
- This will create the content for the next sketch (Sketch<2>).

From the top Icons, click on Select to exit sketch mode. This Completes Sketch<2>

Create the Third Loft Sketch.

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

- Click the new Plane (Plane<1>) in the workspace or in the Design Explorer.
- Then, Right-Click in the workspace open area (away from any selectable planes, axis, etc.)
- Select Insert Plane from the menu.
- In the Insert Plane popup, confirm the Plane<1> is displayed in the Select Geometry window; or else re-click on it now.
- For the Plane Offset Distance, Set Distance to 15 mm (press Tab)

Click **OK**. Then Click off in the workspace again, to de-select it. Plane<2> is created.

Next - Create the Sketch for the third part of the Lofted Vase. (Select Plane<2>)

- Click the new Plane (Plane<2>) and from the Sketch Icons, Click Select.
- From the Sketch Icons, select Line.
- Click on the Origin, and move right along the X-Axis to 4.0, Double-Click to end the line.
If you failed to correctly double click - when you move away - the line will show a light blue line between the last point - and the cursor, simple press the Esc key. To exit this line.
- Click again on the Origin, and move up along the Y-Axis to 5.0, Double Click to end the line.
These two lines will be used to make it easy to know you are placing the ellipse major and minor axis at the correct distance.
- From the Main Menu, Select Sketch> Figures> Ellipse.
- As per the hint line "*Pick a Center point for the ellipse*" - Click on the Origin.
- As per the hint line "*Pick next point for the major axis of the ellipse*" - drag up the Y-Axis to the end of the vertical line, and when the cursor symbol changes to indicate it is over the end of the line, Click. (*The cursor changes from an ellipse symbol - to a figure like an Asterisk [*]*)
- As per the hint line "*Pick next point for the minor axis of the ellipse*" - drag right along the X-Axis to the end of the Horizontal line, and when the cursor symbol changes to indicate it is over the end of the line, Click. (*This completes the 5 x 4 ellipse*)
- From the Sketch Icons, Click Select.
- Now - click on the vertical line, and shift-click on the horizontal line. (*They are both yellow*)
- Press the Delete Key on the keyboard to delete them both.
- This will create the content for the next sketch (Sketch<3>)

From the top Icons, click on Select to exit sketch mode. This Completes Sketch<3>

Create the Fourth Loft Sketch.

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

- Click the new Plane (Plane<2>) in the [workspace](#) or in the [Design Explorer](#).
 - Then, Right-Click in the workspace open area (away from any selectable planes, axis, etc.)
 - Select [Insert Plane](#) from the menu.
 - In the [Insert Plane](#) popup, confirm the Plane<2> is displayed in the [Select Geometry](#) window; or else re-click on it now.
 - For the [Plane Offset Distance](#), Set [Distance](#) to 10 mm (*press Tab*)
- Click **OK**. Then Click off in the workspace again, to de-select it. Plane<3> is created.

Next - Create the Sketch for the fourth part of the Lofted Vase. (*Select Plane<3>*)

- Click the new Plane (Plane<3>) in the [workspace](#) or in the [Design Explorer](#).
- From the [Sketch Icons](#), select [Circle](#)
- Click on the [Origin](#), and move right along the [X-Axis](#) until the Diameter is 6, Click.

From the top Icons, click on [Select](#) to exit sketch mode. **This Completes Sketch<4>**

Create the Loft

From the main menu, select [Feature](#)> [Boss](#)> [Loft](#).

(Or Select [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>](#), [Sketch<3>](#), and [Sketch<4>](#)
 - You will see the four 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<4>](#). This is fine for now.
- Click **OK**. Your new lofted vase will be displayed (highlighted in yellow, initially)

Create the First Leg

For this we will first create a new plane, and then create the two sketches for a sweep.

First - create a new plane.

- Click the Plane (Plane<3>) in the workspace or in the Design Explorer.
- Then, Right-Click in the workspace open area (away from any selectable planes, axis, etc.)
- Select Insert Plane from the menu.
- 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 0 mm (*press Tab*)
(*This plane is co-located with Plane<3>*)
- Click **OK**. Then Click off in the workspace again, to de-select it. Plane<7> is created.

Second - create the Sweep Path in a crossing plane, between Plane<7> and the lofted Vase.

Click on the ZX-Plane in the Design Explorer.

- From the Sketch Icons, click Select. (*Highlight Plane<7> and the YZ-Plane in the Design Explorer to make it easier to identify them*)
- From the Sketch Icons, select Line.
- Create a line 1.8 mm long, from Plane<7> down along the YZ-Plane.
- Create another line approximately 12-14 mm long from near the start of the first line, out to the right, an axis. (*So Sketch Constraints are automatically added*)
- From the Sketch Icons, select Dimension.
- Click the long line, and drag down slightly below the line, click to place the dimension, the edit the dimension to 15.0, Press Enter.
- From the Sketch Icons, select Options - Coincident Constraint.
- Click the upper end of the first line right at it's end point, then shift-click the nearest end of the long line. (They will automatically lock together)
- From the Sketch Icons, select Circular Arc - Center - Start - End. (*If this is not in the arc selection currently - click on the options below arc, and select it from the fly-out menu.*)
- Start the Arc at the bottom of the 1.8 mm line, drag the mouse to the right - along a Horizontal direction, and click when the radius Displays 14, then drag up and left until the Angle displays 77 - 80 degrees and you are over, or in front of, the Vase Body. Click.
- From the Sketch Icons, select Dimension.
- Click on the Arc, and drag inward to the Center, Click, enter 14.0, Press Enter.
- From the Sketch Icons, select Trim Figure.
- Trim the tail of the arc below the 15 mm long line.
- From the Sketch Icons, click Select. Click the long line and shift-click the short line. Press the Delete Key to delete them.
- *To make it easier to find the edge of the bottom of the Arc we just made; we will close the feature temporarily, and extrude it!*
- From the Sketch Icons, select Line.
- Click on one end of the Arc, and then double-click on the other end.
- From the Feature Icons, select Extrude Boss.
- In the Extrude Boss Popup, Sketch should be Sketch<5>, Type = To Depth, Direction = Along Normal, and Depth = / Make 5 mm. Click OK.

This Completes Sketch<5> initially, and creates Extrusion<6>.

We will return to Delete the Extrusion, and edit the Sketch.

Next - Create the sketch to be Swept - Sketch 6.

- Click the new Plane (Plane<7>) in the [workspace](#) or in the [Design Explorer](#).
- From the Top Row of icons, Select - [Zoom To Window](#), Drag a selection box around the outer edge of the extrusion just created, at the edge that is center to the vase. This gives us a close-up of this corner. This should be along the ZX-Plane, at the outer edge of the Extrusion.
(Highlight the ZX-Plane in the Design Explorer to make it easier to identify)
- From the [Sketch Icons](#), select [Circle](#).
- Click along the line of the ZX-Plane / The lower edge of the Extrusion, and Drag to the right to make the circle Diameter 2, Click.

Next - Create a reference line to lock the center of the circle to, and constrain it there.

- From the [Sketch Icons](#), select [Project to Sketch](#).
- In the [Project to Sketch](#) popup, first change the [Options](#) to [Create Reference Figure](#) by clicking in the other Radio Button.
- Then Click inside the selection area below [Entities to project](#), the click on the outer edge of the extrusion (To the right)
- Click [OK](#).
- From the [Sketch Icons](#), select [Options](#), and select [Coincident Constraint](#) from the fly-out menu.
- Click on the end of the line at the ZX-Plane, and shift-click on the point at the center of the circle. *The Circle now jumps to the end of the line, and is perfectly aligned.*
- From the [Sketch Icons](#), click [Select](#),
- Click on the reference Line, and press the **Delete** key.

From the top Icons, click on [Select](#) to exit sketch mode. **This completes Sketch<6>**

Now - we will go back, Delete Extrusion<6> and edit Sketch<5>

- In the [Design Explorer](#), Right Click on [Extrusion<6>](#) (above [Sketch<5>](#)), and select [Delete](#) from the drop-down menu.
- In the [Design Explorer](#), Right Click on [Sketch<5>](#), and select [Edit](#) from the drop-down menu.
- In the Sketch Workspace, (Select is default Selected when editing), Click the line between the Arc points, press the **Delete** key.

From the top Icons, click on [Select](#) to exit sketch mode. **This completes the edit of Sketch<5>**

From the [Feature Icons](#) on the right, Click on [Sweep Boss](#)

- In the [Sweep Boss](#) popup, Click under [Sketch to Sweep](#), and then click on [Sketch<6>](#) in the [Design Explorer](#),
- In the [Sweep Boss](#) popup, Click under [Path Object](#), and click on [Sketch<5>](#) in the [Design Explorer](#),
- Leave [Sweep Type](#) as [Entire Path](#), and [Draft Angle](#) as 0
- [Label](#): is set for [Sweep<6>](#), which is fine. Click [OK](#).
(The New Sweep Leg is now Visible.)

This completes the creation of Sweep<6> (Leg 1 of 3)

Create the Other Legs

From the Main Menu, Select **Feature> **Pattern**> **Circular**.**

- **Features to Pattern:** Confirm **Pattern Geometry** is checked.
- Click in the Selection Area under **Features to Pattern**, then click **Sweep<6>** in the Design Explorer,
- Click in the **Center:** Selection area, then click on the **Z-Axis** in the Design Explorer,
- Make **Copies:** 3 (Click up/down triangles to change),
- Set **Angle** at 120 Degrees (Highlight the value, enter 120, press Tab)
- **Label:** is currently **Pattern<7>**, which is fine. Click **OK**.
(*The new Legs are Highlighted Yellow.*)

This completes the creation of Pattern<7> (Legs 3 of 3)

Create the Shelled Walls

Click on the Ellipse Surface End of the Vase, Click on the Shell Icon from the Feature Icons

- In the **Shell** popup, **Faces to remove:** should show **Face<9>**, if not - click to select it.
- **Standard thickness:** change to **0.2 mm** either using the up/down arrows to select, or highlight the value, enter 0.2, and press Tab.
- Leave **Shell outward** unchecked. **Label:** is showing **Shell<8>**, this is fine.
- Leave **Oyerrides** alone, Click **OK**.
You can see the successful shell highlighted yellow inside the vase!

Create the Feature Colors

Now you can edit the loft-vase's Color properties!

- From the main menu, click **Edit, Color Properties**, (*Or - Right Click in an open space in the Part Workspace, select Color properties from the drop-down 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: Fuschia. [i.e. Choose the 3rd box down on the Last column over: Red 255, Green 0, Blue 128]
- Click **OK** to close the Color selection box and accept the new Color,
- Click **OK** to apply the Color to the model.

Create the 3D Section

From the Main Menu, Select **Insert> **3D Section View****

- From the **3D Section View** popup, **Parts to Section** should be showing **loft-vase-1** listed.
- Click in **Slicing Plane:** Selection area, then click on **ZX-Plane** in the **Design Explorer**.
- Click to select **Reverse**. This will split the vase across one leg and the whole body. Click **OK**.

View as normal, and Save.

In the Design Explorer, Under Section Views, Right Click on Section View<1> click on View.

- This will revert the view back to the normal view.

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

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

Congratulations!

You have completed the Loft-Vase-1 Tutorial!

(Text Version)

Give me feedback on this tutorial!

[Send email](#)

[If you found this and you don't yet have a copy of Alibre Design - Click this Link for a 30-Day Free Trial Copy!](#)