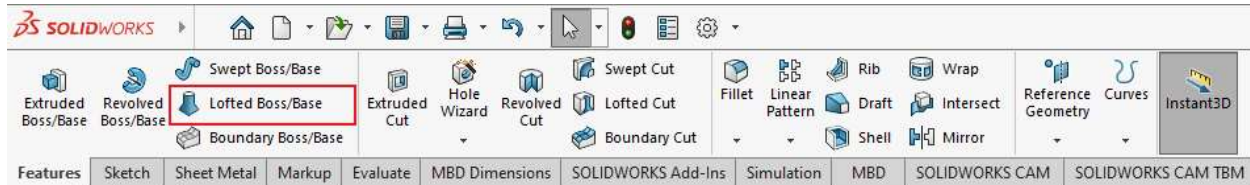# Show and Tell – Lofts and Guide Curves
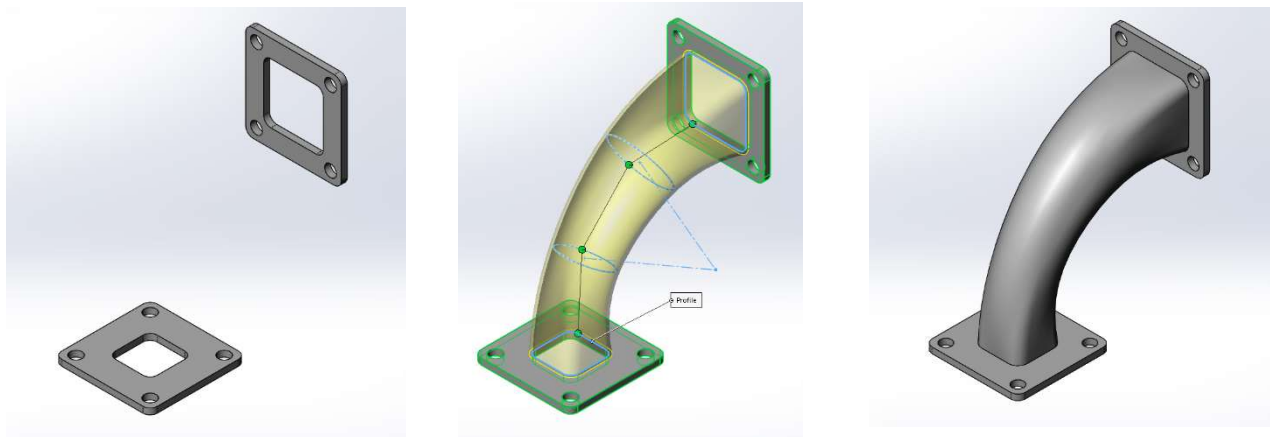
Topics covered: Reference Geometry, Sketches and Construction Lines, Loft Profiles, Guide Curves, Thin Feature

Why use a loft?

Lofts allow you to create arbitrary 3D shapes defined by two or more 2-D "profiles", and for which the path between them can be defined by one or more sketches called "guide curves".

The figure below, shows an example model, this model uses sketch profiles and guide curves to create a part that would otherwise not be easily modeled. Pre-loft (left) final model (right).

Download the "pre-loft "starting model: A "pre-loft" starting model can be downloaded from the in-class activities column on class day 9 or 10. The base feature (in this case) was created by two separate sketches and extrusions. The top flange was modeled on the front plane with an offset from the origin, while the bottom flange was modeled on the top plane with an equal offset from the origin.

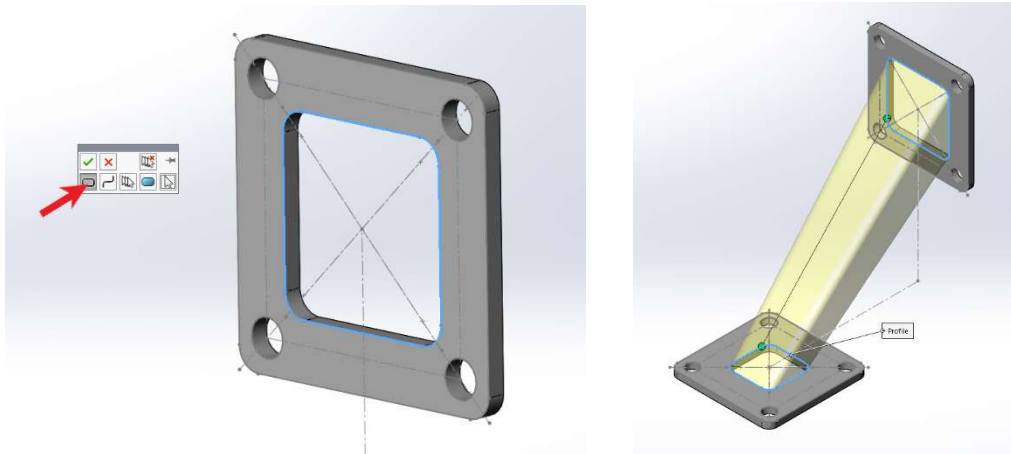In this demo we will cover lofting using three different common methods.

   A. Lofting through profiles.

   B. Lofting with guide curves.

   C. Lofting with guide curves and profiles.

Show and Tell – Lofts and Guide Curves

Lofting through Profiles.

Create lofted feature:

- From the feature toolbar, click the Loft icon.
- Make sure the Profiles dialog box is highlighted.
- Click the eye from the top center of your screen to toggle the sketch views on.
- Select the top inner sketch and click "Select closed loop".
- Select the bottom inner sketch and click "Select closed loop".
- Click and drag the locations of the green points as needed, making sure that the transparent yellow preview displays the shape you want (with or without twisting).
- When finished, select the green check mark to create the loft.

The shape of this loft can be improved with the addition of more sketch profiles!

Reference planes will be created to give the sketch profiles the optimal orientation. Reference geometry can be accessed in a couple ways.

a) from the feature toolbar by selecting Reference Geometry → Planes,
b) from the dropdown menus by selecting Insert → Reference Geometry → Plane,
c) holding CTRL, depressing the left mouse click over an existing plane, dragging the mouse, and releasing (this creates an offset reference plane, parallel to the plane used).

Note Additional Reference Geometry can be accessed through either option (a.) or (b.).
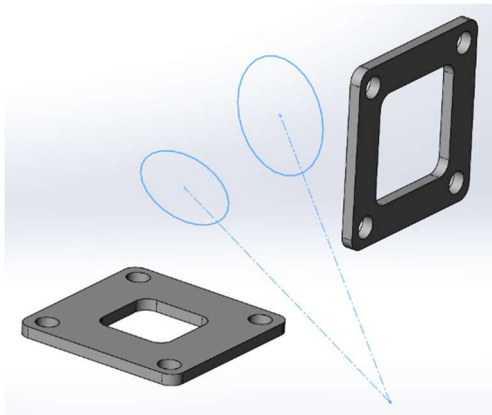
Create Reference Geometry:

- Select Loft1 and delete it. The sketches will appear at the main level of the Feature Tree and be shown. This will allow you to recreate the loft later using additional profiles or guide curves.
- Select Reference Geometry → Axis from the feature toolbar.
- In the Axis Property Manager, make sure *Two Planes* is selected, then click on the selection box.

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- o Expand the feature tree out and select the front plane.
- o Select the top plane and click the green check mark.
- Select Reference Geometry → Planes from the feature toolbar.
- In the Planes Property Manager, make sure the *First Reference* box is highlighted and select the Reference Axis you just created.
- In the Planes Property Manager, make sure the *Second Reference* box is highlighted and select the Top plane from the feature tree. Click on the angle definition and input 30°.
- Enter 2 in the *instances* box.
- Use the *Flip Normal* and *Flip offset* to set the planes between the existing geometry, 120° and 150° when looking from the right view. Click the green check mark to exit the plane manager.
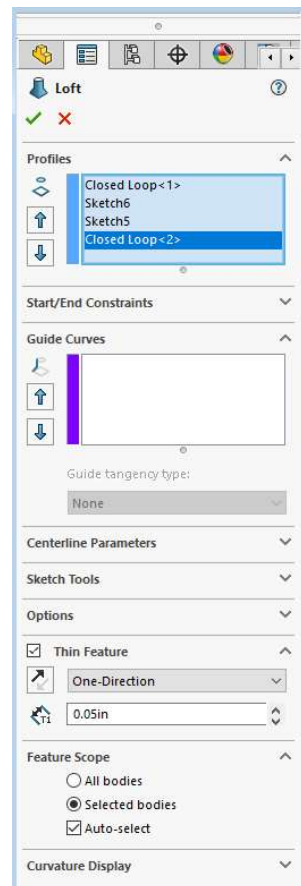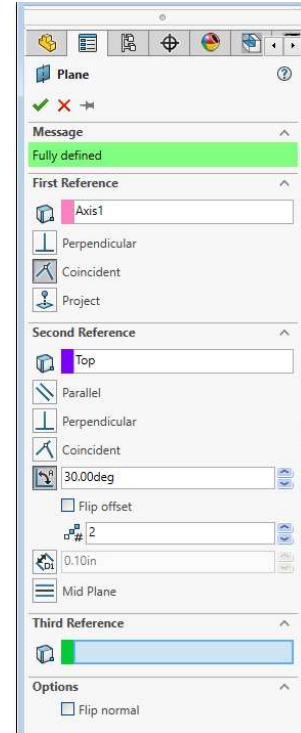
Create Additional Sketch Profiles:

- Select *Plane1* and click the sketch icon. Use Ctr+8 to bring the sketch normal to the view. Define a circle 3.125" out from the origin and 1.0" in diameter. Use a construction line from the origin to the center of the circle with a vertical relation to ensure symmetry. Click on the green checkmark to exit the sketch. Label sketch "Profile-1".
- Select *Plane2* and click the sketch icon. Use Ctr+8 to bring the sketch normal to the view. Define a circle 3.125" out from the origin and 1.25" in diameter. Use a construction line from the origin to the center of the circle with a vertical relation to ensure symmetry. Click on the green checkmark to exit the sketch. Label sketch "Profile-2".
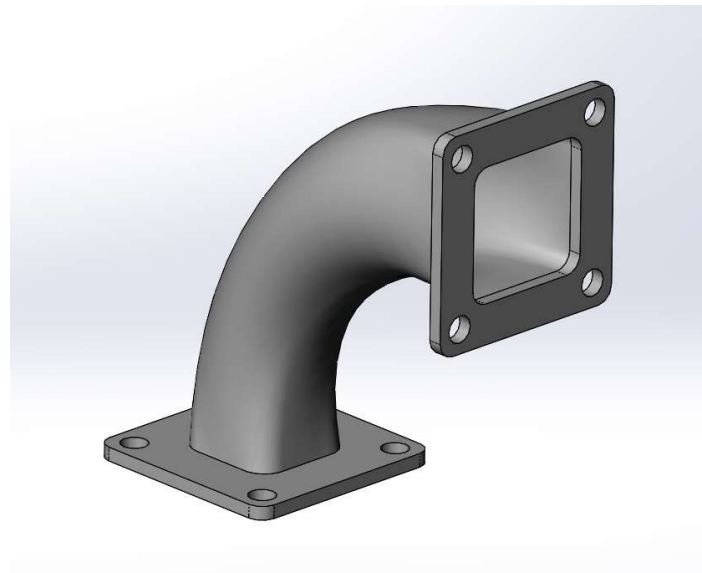
Create the Lofted Feature:

- From the feature toolbar, click the Loft icon.
- Create the lofted feature as done before by selecting one profile after another.
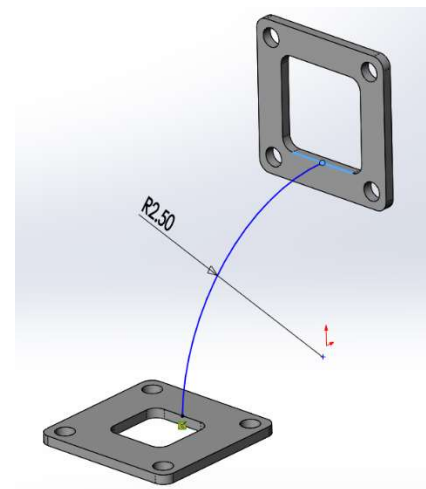
Thin Feature:

- Before clicking the green check mark expand the *Thin Feature* window in the Loft Property Manager.
- Check the *Thin Feature* option and enter .050". Select the drop down and click *One-Direction*. Click the green check mark to close the loft feature.

Lofting with Guide Curves

Creating the Guide Curve:

- Select *Loft2* and delete it.
- Select the right plane in the feature tree and click on the sketch icon.
- Select a 3-Point Arc from the sketch tools and place it on the plane.
  - Ctr + Select the bottom edge of the hole in the top flange and the point of the Arc, click on the Pierce relation.
  - Repeat the previous step for the bottom flange.
  - Add a radius dimension of R2.5 to the Arc. Click the green check mark to exit the sketch. Label sketch "Guide-1".

Show and Tell – Lofts and Guide Curves

Creating the Loft:

- Select the Loft icon to open a new loft feature.
- Select the top and bottom flange profiles as done before. *(Do not select Profile-1 or Profile-2)*
- Click on the Guide curve box to highlight it. Select the Arc that was just created. Click the green checkmark to exit the loft.

Creating Additional Guide Curves

- Select Loft3 and delete it.
- Select the right plane in the feature tree and click on the sketch icon.
- Add an Arc to the top side of the flange in the same way the bottom Arc was done, add a radius dimension of R4.0. Click the green checkmark to exit sketch. Label sketch "Guide-2".
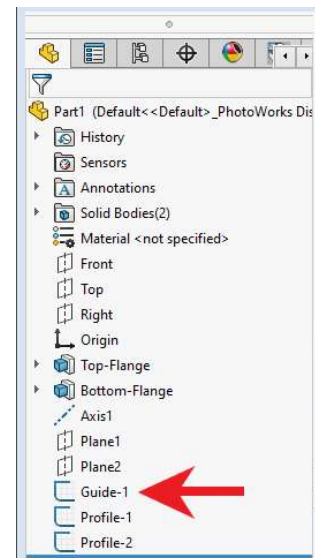
Creating the Loft with 2 Guide curves:

- Select the Loft icon to open a new loft feature.
- Select the top and bottom flange profiles as done before. *(Do not select Profile-1 or Profile-2)*
- Click on the Guide curve box to highlight it, select both Guide-1 and Guide-2. Click the green checkmark to exit the loft feature.
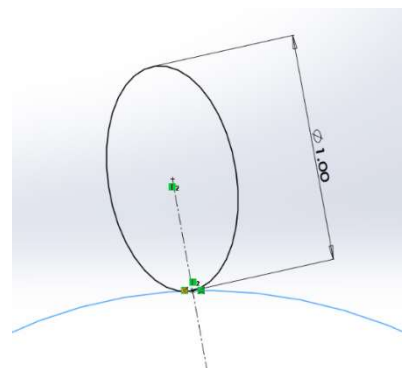
Lofting with Guide Curves and Profiles

Connecting the Guide Curve to the profiles:

- Select Loft4 and delete it.
- Select Guide-2 and delete it.
- Move Guide-1 to above Profile-1 and Profile-2.
- Right click on Profile-1 and click edit sketch.
  - Delete the distance from origin dimension.
  - Select a sketch "Point" and place it at the bottom center of the circle.
  - Select the sketch "Point" and Guide-1, click on the Pierce relation.
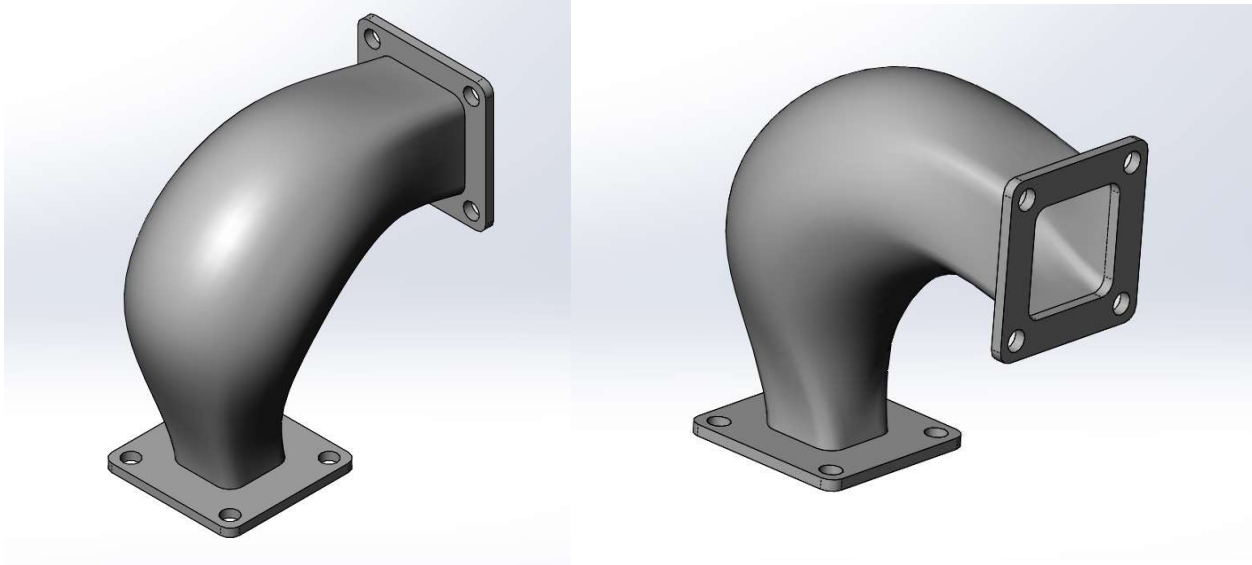 - Repeat the above steps for Profile-2.

Creating the Loft with Profiles and Guide Curve(s):

- Select the Loft icon to open a new loft feature.
- As done before select the top and bottom flange as well as Profile-1 and Profile-2.
- Click on the guide curve box so that it is highlighted and select Guide-1.

- Click the green checkmark to exit the loft feature.
- If you would like to change the dimension of the sketches after the loft has been made you can simply edit the sketch dimension and press Ctr+Q to rebuild the model.

Notes on Lofting

- When using guide curves and profiles the curve(s) must intersect each of the profiles for the loft to rebuild correctly.
- With more complex lofts it is sometimes hard to keep the model from twisting, in these cases you can use the extra connections accessed by right clicking on an open loft feature and clicking "show all connections". This allows you to define more than one connection line between profiles.
- If you are having trouble telling whether your loft is twisted or not you can use the "Curvature Display" located at the bottom of the Loft Property Manger window.
- "Start and End Constraints" within the Loft Property Manager window will control the loft direction at the first and last profile.