Chapter 16: Boat Hull - Loft and Sweep

This tutorial demonstrates classic boat hull lofting techniques using typical plan and profile curves. The classic hull shape is based on a design from an old Boat Builder’s Handbook magazine. Many designs similar to this are available over the Internet.

You will learn how to:
- Create 3-D curves from a 2-D lines drawing.
- Rebuild and simplify the curves.
- Use analytical techniques to ensure fairness.
- Loft surfaces from the curves.

Rhino is used by marine designers in many segments of the industry. For more tutorials and information about marine design, see the Rhino website at www.rhino3d.com.

Note: The images in this tutorial use a display setting to change the color of the back of the surfaces.

Front face (1), backface (2). Yellow arrows indicate surface normal direction, and green color indicates surface backface. This lets you see which way the normal direction of the object faces. Search the Rhino Help for Backface settings.
Marine Terms Used in this Tutorial

**Sheer**
The fore-and-aft curvature from the bow to the stern of a ship’s deck as shown in side elevation.

**Chine**
The intersection of the bottom and the sides of a flat or v-bottomed boat.

**Transom**
The planking forming the stern of a square-ended boat.

**Fair**
The meaning of “fair” is much debated in the marine industry. No one can define it, but they know when they see it. Although fairing a surface is traditionally associated with hull surfaces, all visible surfaces on any object can benefit from this process. In Rhino, the first cue for fairness in a surface is the spacing of the surface display isocurves.

There are other characteristics of fair curves and surfaces. Although a curve or surface may be fair without exhibiting all of the characteristics, they tend to have these characteristics. If you keep these in mind while modeling, you will end up with a better final product.

The guidelines for creating a fair surface include:
- Use the fewest possible control points to get the curve shape.
- Use the fewest possible curves to get the surface shape.

### Lay out the hull curves

The hull lines were created by tracing the original plans using a background bitmap. The first step is to check the lines for fairness before creating surfaces from them.

The designer’s lines are illustrated. The sheer and chine have been extended at the forward and aft ends to accommodate the lofting process.
### Start the Model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User’s Guide*.
3. Open the model file **Victory.3dm**.

The lines are laid out on the **Plan** layer and the **Profile** layer.

### Check the curves for fairness

Select each of the designer’s curve pairs in plan and profile and use the **CurvatureGraph** command to determine if the curves are "fair." In this case, the file has the original curves that were traced from the background bitmap. They are not "fair." In other words, the curves do not smoothly transition from one end of the sheer to the other. If any curve is not fair, adjust points to make it fair. Start with the sheer (the curve at the top of the hull shape). It has the biggest impact on the appearance of the vessel.

#### Check curvature

1. **Select** the curves you want to check.
2. Use the **CurvatureGraph** command to display its curvature graph.

   The illustration shows the curvature graph applied to the two-dimensional sheer in profile.

The curvature graph should be continuous and exhibit the characteristics desired for the curve. When the curve is concave downward, the graph will be above the curve. Conversely, concave upward curves will have their graphs below them. The point of inflection (where the curve is neither concave upward nor downward) is indicated where the graph crosses the curve.

#### Fix the curvature

Before doing any point editing to make the curves fair, rebuild the curves to remove excess control points. Select each curve and use the **Rebuild** command to reduce the number of points and set the degree. Do not use more points than you absolutely need.

Use the **CurvatureGraph** command to check the curves again for fairness. If the curvature graph is still not satisfactory, move the control points until you have a smooth graph. Proceed with the rest of the curves in the model to be certain they are fair before beginning to surface the model.

#### Rebuild the curves

1. **Select** the sheer curve.
2. Start the **Rebuild** command.
3. In the **Rebuild Curve** dialog box, change the **Point count** to 6 and the **Degree** to 5.

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**Create the 3-D curves**

So far, you have been working with two-dimensional curves. In order to loft the surfaces, these planar curves will be used to create three-dimensional curves and the planar curves can be discarded.

With the **3D Lines** layer current, select the profile and plan view representations of each curve. Use the **Crv2View** command to create the three-dimensional curve that combines the x-, y-, and z-coordinates of the two-dimensional curves. The two-dimensional curves must be planar for this command to work.

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**Create the three-dimensional curves**

1. Set the **3D Lines** layer current.
2. **Select** the plan and profile representations of the sheer curve.
3. Start the **Crv2View** command.  
   The three-dimensional representation of that curve will be created.

4. When you are satisfied that the proper curve was created, delete or **Hide** the two-dimensional representations.

5. Repeat the **Crv2View** command for the chine curve.
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Revise the curves

For the loft process to work on the bottom panel, it cannot come to a point. The lofted shape must be rectangular. This is why the curves are extended beyond the centerline. The curves can be lofted into a rectangular surface that can then be trimmed back. The curves in the Victory model are already extended for you except for the bottom centerline curve.

Copy the centerline curve

We are going to use a copy of the centerline to create a new extended curve for lofting the hull bottom.

1. **Copy** the centerline using the **InPlace** option.
2. **Hide** the centerline.

Shorten the centerline

1. Select the centerline.
2. Start the **SubCrv** command.
3. At the **Start of curve** prompt, using the **End** object snap, click the aft end of the centerline.

4. At the **End of curve**... prompt, using the **Mid** object snap, click the middle of the curve.

Extend the centerline

1. Start the **Extend** command, and at the **Select boundary objects**... prompt, press **Enter** for dynamic extend.
2. At the **Select curve to extend**... prompt, set the **Type** to **Smooth** and select the centerline near the forward end.
3. Draw the curve so it aligns nicely with the chine and sheer curves in the plan view as illustrated.
This creates a new bottom curve to use for the surface loft.

4. Turn on the control points (F10) to check the curve.
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Loft the hull surfaces

Now that you have created a set of edge curves for the side and bottom, create lofted surfaces from these curves. Start by lofting the bottom surface. Once you have finished it, use its upper edge as the curve from which to loft the side panel.

To loft the bottom panel, select the two edges (chine and centerline) and use the Loft command. In this case, be sure to select the new centerline you created in the previous step.

Loft chine and centerline

1. Select the chine and centerline.
2. Start the Loft command.
3. In the Loft Options dialog box, under Cross-section curve options, select Rebuild with..., and set the control point count to 15, click OK.

Loft the side and bottom

1. Select the surface edge and the sheer curve.

Tip: Hold Ctrl + Shift to select the upper edge of the loft surface.

2. Repeat the Loft for the side panel.
3. In the **Loft Options** dialog box, under **Cross-section curve options**, select **Rebuild with...**, and set the control point count to 15, click OK.

### Trim the bow and bottom

When you have successfully created both the side and bottom surfaces, construct a buttock one-half inch off the centerline and trim both surfaces to this buttock. To do this, in the **Top** viewport, draw a line longer than the hull and one-half inch to the right of centerline.
**Draw a trim line**

1. In the Top viewport, draw a Line along the centerline (x-axis) that is longer than the hull.

![Diagram](image1)

2. In the Top viewport, offset the line 1/2 inch toward the hull surfaces.

![Diagram](image2)

This creates a curve that will be used in the next step. We want a small gap down the center of the boat between the two halves for a keel.

**Trim the side and bottom to the trim line**

- Using the offset curve, Trim the bottom (1) and side (2) as illustrated.

![Diagram](image3)
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**Build the transom**

Like all surfaces in this tutorial, the transom will be built with a surface larger than the finished surface and then trimmed to the hull.

To make sure there is enough surface area to trim, **Extend** the transom centerline by a foot or two both above the sheer and below the centerline. **Trim** the hull surfaces with the transom centerline.

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**Extend the centerline**

1. With the **Profile** layer on, start the **Extend** command.
2. At the **Select boundary objects or enter extension length** prompt, press **Enter**.
3. At the **Select curve to extend...** prompt, in the **Front** viewport, set the **Type** to **Natural**, and select near the top of the transom centerline.
4. At the **End of extension** prompt, select a point above the current top of the transom centerline.

5. At the next **Select curve to extend...** prompt, select near the bottom of the transom centerline.

6. At the **End of extension** prompt, select a point below the current bottom of the transom centerline, press **Enter**.
Trim and Join the hull surfaces

1. Select the transom centerline.

2. Start the Trim command.
3. Set UseApparentIntersections=Yes.
4. In the Front viewport, at the Select object to trim... prompt, select the hull side and bottom surfaces aft of the transom centerline.

5. Join the hull bottom and side.

Mirror the hull surfaces

In the Right or Top viewport, Mirror the two hull surfaces about the centerline. Use the EdgeSrf command to create surfaces between the two hull halves.

1. Select the two hull surfaces.
2. Start the Mirror command.
3. In the Top viewport, at the Start of mirror plane... prompt, type 0, press Enter.
4. At the End of mirror plane prompt, with Ortho on, drag the mirror plane along the x-axis, and click.
Create the keel surface

1. Start the `Edgesrf` command.

2. At the **Select 2, 3, or 4 curves** prompt, select the two inner edges of the hull bottom along the keel.

3. Repeat the `Edgesrf` command.

4. At the **Select 2, 3, or 4 curves** prompt, select the two inner edges of the hull sides along the keel at the bow.

Tip: The order you select the surface edges determines what direction the keel surface faces.

Extrude the surface

To create the transom surface, **Extrude** the transom centerline.
1. In the **Front** viewport, select the extended transom centerline.

2. Start the **ExtrudeCrv** command.

3. At the **Extrusion distance** prompt, set the command-line option **BothSides=Yes**.

4. In the **Perspective**, **Top**, or **Right** viewport, drag the extension beyond the hull surface.

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**Trim the transom**

Trim the transom surface with the hull and a line from the hull edges.
1. Draw a line between the two aft hull top edges.
2. Start the **Trim** command.

3. At the **Select cutting objects** prompt, select all of the hull surfaces, including the keel surface and the line at the top of the hull, press **Enter**.

4. At the **Select object to trim...** prompt, select the transom surface outside of the hull lines and surfaces, press **Enter**.

The transom is now complete.

**Check for errors**

1. **Join** all of the surfaces.
2. Use the **ShowEdges** command to check that the join was successful.
Display the *naked* edges. Naked edges are surface edges that are not joined to other surfaces. In this case, the only naked edges should be the ones you expect around the outside of the hull surfaces – not those between the surfaces.

When you have your surfaces built and joined, and have no unjoined edges, look at the surface with the curvature analysis tools.

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**Add the deck**

The last step is to create the deck surface. In the profile lines, two curves describe the silhouette of the deck curve. You will use this curve to create the deck.

Use the **Project** command to project the vertical line to the side of the hull. This line will act as a marker for the end of the curve. In the **Front** viewpoint, draw a curve from the end of the deck centerline curve to the end of the projected curve on one side of the hull. Use **Planar** mode to keep the curve planar. Place the first three control points using **Ortho** to keep them lined up at the center.
Project the vertical deck edge to the hull

1. Select the hull and the vertical line.

2. In the Front viewport, use the Project command to project the curve to the hull.
   The curve will project to both sides of the hull, so you can draw your cross-section curve on either side.

Draw the cross-section curve

1. Click the Planar pane in the status bar to turn on Planar mode.

2. In the Right viewport, use the Curve command to draw a control point curve from the top end of the deck centerline curve to the top of the curve projected to the hull.
   Use Ortho to place the first three control points in a straight line.

Use the End object snap to place the last point at the top of the projected curve on the hull.
3. Use the **CurvatureGraph** command to check the curve.

![CurvatureGraph](image)

**Create the deck surface**

1. Use the **Sweep2** command to create the deck surface.
2. At the **Select rail curves** prompts, select the centerline curve and the hull edge.
3. At the **Select cross section curves...** prompt, select the cross-section curve you created from the deck centerline curve to the projected curve on the hull, press **Enter**.

![Diagram showing cross-section curves]

**Mirror the deck**

1. Use the **Mirror** command to copy the deck surface to the other side.
   At the **Start of mirror plane...** prompt, in the **Top** viewport, type **0**, press **Enter**.

2. At the **End of mirror plane...** prompt, in the **Top** viewport, drag the mirror plane with **Ortho** on.

![Diagram showing mirrored deck]
**Fill in the missing surface**

- Use the `Edgesrf` command to create the small triangular surface at the tip of the bow.

**Join the parts**

- Select all of the surfaces and use the `Join` command to create a single polysurface.