

Rhino**ceros** Advanced Training Series

Marine Design



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Introduction

Rhino is used in many phases in the marine industry, because with Rhino it is possible to integrate the design and building processes.

Rhino is used for:

- Designing
- Visualizing
- Engineering
- Lofting
- Tooling

Design

With Rhino you can develop shapes for the hull, tunnels, superstructure, interiors, and cabinets, then extract outlines and parts for manufacturing information.



Alan Andrews J Bird III.

Rhino is:

- Flexible enough to model detailed superstructures.
- Accurate enough to check clearances.
- Not limited to certain vessels.

With Rhino you can:

- Blend and match adjacent surfaces.
- Create clean geometry for later use.
- Model interior areas.

Visualize

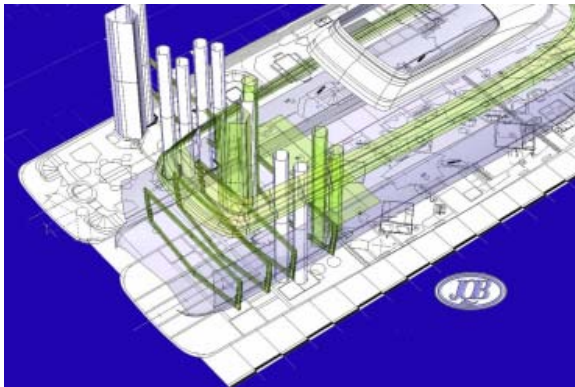
Rhino can be used for concept validation and visualization. These images can be used for client presentations or meetings and to seek funding.



Kvaerner Masa 85 meter offshore patrol vessel.

Engineer

Rhino is also used for engineering structure, ductworks, powering.



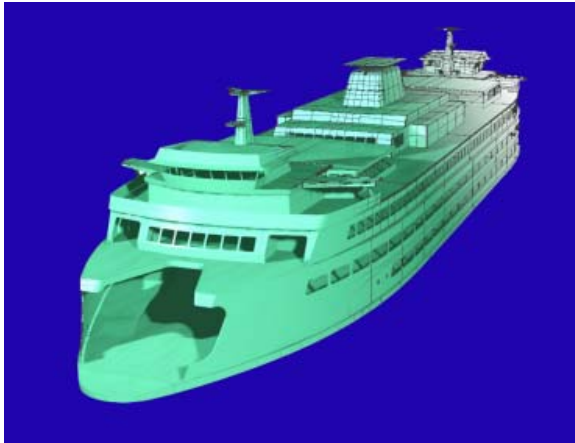
124' motor yacht designed by JQB Ltd. Built by Delta Marine.

With Rhino you can:

- Model structure and all systems.
- Check clearances and tolerances.
- Work out difficult areas of the boat.
- Translate information into other software for analysis.
- Metalwork detailing.
- Lay out equipment.
- Detail railings, stairs, and equipment.
- Detail joinery.

Loft

When working on lofting the shapes to build, Rhino can assist in developing the shapes needed for construction.



*Washington State Ferry Jumbo Mark II. Lofted by Eric Jolley Marine Design.
Built by Todd Shipyards, Seattle, WA.*

With Rhino you can:

- Loft accurately and with less time.
- Analyze lofted surfaces.
- Model tooling and jigs.
- Take sections at any angle.
- Create and unroll developed surfaces.
- Use data to water jet or plasma cutting.
- Use data for CNC and composite lay-up.

Tool



Hull plug machined with [Janicki's](#) 5-axis milling machine from Rhino files.

For tooling with Rhino you can:

- Create accurate geometry.
- Manipulate the model to get parts.

- Use the model directly for CNC.
- Use the model for plate work.
- Work out difficult areas.

In addition Rhino provides:

- Many formats to export.
- Quick rendering to replace builder's model.

Tutorial Summaries

The tutorials in this set focuses on fairing and creating surfaces in the marine industry. Although most of these tutorials focus on various hull forms, they are meant to serve as examples of surfaces generally used in the industry. The techniques demonstrated in the tutorials apply to all aspects of boat design, whether it is a bow details, dashboard, swim step, galley details, intake ducts.

About the tutorials

The tutorials in this set focuses on fairing and creating surfaces in the marine industry. Although most of these tutorials focus on various hull forms, they are meant to serve as examples of surfaces generally used in the industry. The techniques demonstrated in the tutorials apply to all aspects of boat design, whether it is a bow detail, dashboard, swim step, galley details, intake ducts.

This CD contains six tutorials. Each focuses on a different modeling method and each model is progressively more complicated. The later tutorials expect that you are familiar with the procedures covered in the earlier tutorials.

In addition, each tutorial starts with a different type of information. Some start with existing 2-D drawings and start some drawn from your own design.

The tutorials on this CD cover:

- a general tutorial about fairing techniques for curves and surfaces.
- a 17-foot seine **skiff**. This tutorial takes a simple skiff design and takes you step by step through developing its form. It focuses mainly on plate development and expansion.
- a Transpac-type **sailboat** hull and deck. This tutorial shows you how to model a 70' ULDB (ultra-light displacement boat) sailboat. It focuses on starting a hull from scratch.
- an **offshore** racing power boat. This tutorial is an offshore powerboat that starts from 2-D AutoCAD drawings and takes you through the steps of modeling a chined hull.
- a motor yacht **hull**. This tutorial focuses on a more complicated powerboat hull. In this tutorial you use many steps you learned in the earlier tutorials.
- the **superstructure** for the motor yacht. The final tutorial takes you through the process of modeling the powerboat superstructure.

While creating these models you will learn:

- Techniques for ensuring a fair hull with a limited number of control points.
- Analytical techniques to determine whether the hull is actually fair.
- How to use a set of two-dimensional lines to create three dimensional lines.

- How to surface the three-dimensional lines to create developable surfaces.
- How to unroll developable surfaces to produce flat plate templates.

Each process will be described briefly. If this description does not give you enough information, it is followed by step-by-step instructions. The step-by-step instructions are only included if the process has not been covered in a previous tutorial.

In the online version, click the **Step-by-Step** button at the top of the window to see more detailed instructions.

Requirements

In order to complete this tutorial, you should have at least the following:

- Rhino 2.0.
- Rhino proficiency to Level 1 Training (see Rhino website <http://www.rhino3d.com/> for details).

You also need an understanding of:

- Lofting surfaces.
- Curvature graphs for surfaces and curves.
- Editing surface and curve control points.
- Editing curve edit points.
- Developable surfaces.
- Marine design principles.

Using the online tutorial file

Table of contents

The table of contents lets you move around in the online tutorial and also is a road map of where you are in the process. The current topic is always displayed in the table of contents.

Back and Forward buttons

The online tutorial has navigation buttons at the top of the screen that let you move forward and back in the tutorial.

Command location pop-ups

Throughout the tutorial Rhino command names are used. In the online tutorial, you can display the menu and toolbar location of the command by clicking the command name displayed in blue underlined text.

Annotate

The online tutorial contains an annotation feature that lets you add your own comments and reminders to a topic.

To add a comment

From the tutorial **Edit** menu click **Annotate**.

Topics with annotation, display a paperclip icon next to the topic title.

Bookmarks

You can place bookmarks in the tutorial to help you keep track of where you are.

To place a bookmark

- From the tutorial **Bookmark** menu click **Define**.
A list of bookmarks appears in the Bookmark menu.

How to print

You can print individual topics from the online tutorial.

To print a topic

- From the tutorial **File** menu, click **Print Topic**, or click the **Print** button on the button bar.

To print the entire tutorial

- Print this PDF file.

Conventions used in this document

Where reference is made to a Rhino command, in the online tutorial, click the command name to see the menu location and toolbar button image.

Example: Start the **Trim** command.

Where a prompt requires you to type a response on the command line (for example, entering coordinates or options), the required response will be shown in **bold**.

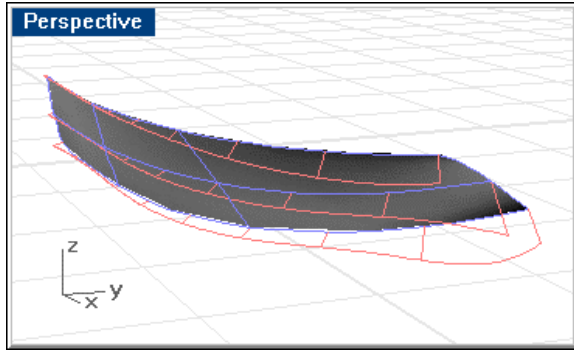
Example: At the **Point to copy to** prompt, type **1.5**.

We assume you know how to use Rhino commands and do not need to be told to press the Enter key to complete commands and selections or click OK to close dialog boxes.

Any reports of progress made by Rhino during the execution of a command will be shown in italics.

Example: *Seven curves found.*

Part I: Fair Curves and Surfaces



Introduction

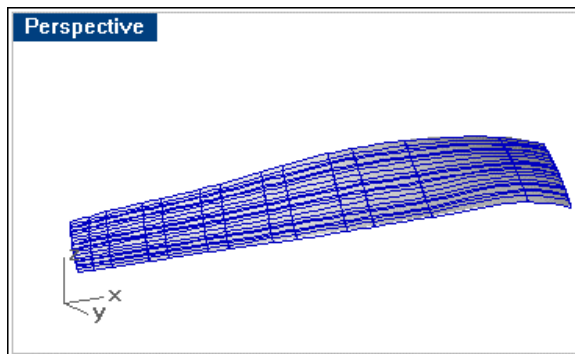
The meaning of “fair” is much debated in the marine industry. No one can define it, but everyone knows when they see it. Although fairing a surface is traditionally associated with hull surfaces, all visible surfaces on a vessel can benefit from this process.

This tutorial will help you understand the general characteristics of fair curves and surfaces in Rhino and the steps it takes to create a surface that is both fair and accurate. We will start with a set of hull lines and create the surface from these.

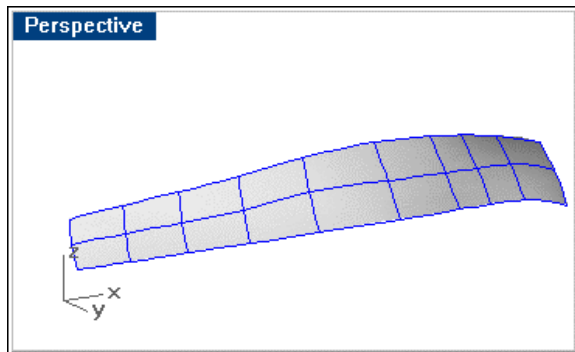
Characteristics of fairness

As in the real world, fairness on the computer can be quickly assessed visually. In addition to standard visual characteristics, Rhino has a few tools that assist in analyzing the fairness of a curve or surface.

In Rhino, the first cue for fairness in a surface is the spacing of the surface display isocurves. Comparing the two hull side surfaces illustrated below, the first surface has very little chance of being fair. There are many unevenly spaced isocurves. They bunch up in areas and spread out in others. These are signs that the surface is not fair. The second surface has far fewer isocurves, and they are evenly spaced across the surface. Although this in itself does not assure fairness, it is one characteristic of a fair surface.



Surface not fair.



Fair surface.

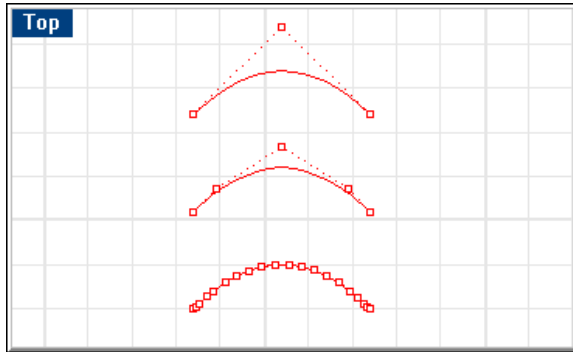
There are other characteristics of fair curves and surfaces. If you keep these in mind while modeling, you will end up with a better final product.

Guidelines for creating a fair surface include:

- Use the fewest possible control points to get the curve shapes you want.
- Use the fewest possible curves to get the surface shape you want.
- Curvature tendencies in one station line are similar in neighboring station lines.

Use few points

Using the fewest possible control points to get the curve shape, sets the stage for all processes that follow. Changes to hull shapes are often subtle. They should not require many points to define them. When importing lines drawings or curves created from digitized points or scanned data, the curves contain a very large number of control points. In this case, having more data is not better. For example, the three curves illustrated below have the same shape.



Simple vs. complex curves.

The top curve will be fair no matter where the points are located. The second curve can be faired with a little work. The bottom curve has too many points to make fair or to use for making a fair surface.

Use few curves

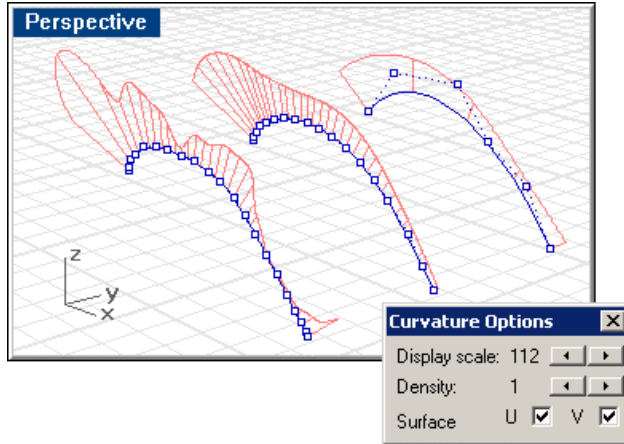
Like points in curves, using the fewest number of curves will help create a fair surface. This does not mean you have to sacrifice the shape of the surface. Simply do not use more data than you need. Use only the curves that describe new features in the surface.

Maintain station relationships

Station lines should transition in a consistent way from one to the next. This means that if a station at the bow has a lot of flare, and the stations amidships have no flare, each station in between should have progressively less flare. This last characteristic of a fair curve or surface seems obvious, but it is often overlooked.

Using the curvature graph for fairing

The **CurvatureGraph** command displays the changes in curvature of a curve. If the graph changes smoothly, the curve is smooth or fair. Jumps in the curvature graph indicate kinks. In the image below, the blue lines are the curves. The orange "hairs" coming off the curves are the curvature graphs. This graph magnifies the bumps, wiggles, and inflections in the curve. It also shows you which way the curve is curving. The **Curvature Options** dialog box lets you change the size and density of the hairs.



Curvature graphs.

In this illustration, the shapes of three curves are very similar. The curvature graph tells us that the curves are convex.

The curvature graph for the curve on the left shows that this curve is not fair. The middle curve, which has had some fairing work done on it, has a much smoother curvature graph. The curve to the right is even better. The curvature graph is very smooth and much simpler than the other two curves. We would consider this curve fair.

Steps to achieving fairness

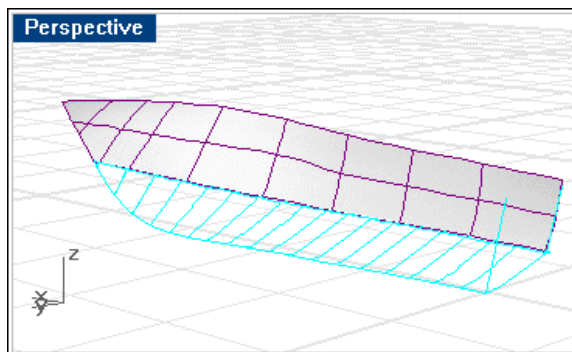
If a curve or surface is not fair, you will have to change the shape to make it fair.

You cannot change the number of control points on a surface without at least slightly changing the shape of the surface; but the fewer points the surface has, the better possibility you have of creating a fair surface.

You will see that we are able to get surfaces very close to the original curves, while at the same time making them very fair.

The tutorial starts with a set of designer's lines. Lines can come from many different sources including 2-D plan and profile drawings, traced scanned sketches, and curves created with 3-D digitizers or drawn from scratch in Rhino. Regardless of the origin of the curves, the fairing process is similar.

This tutorial we will be fairing the topside surface on this generic powerboat hull.



Completed powerboat hull.

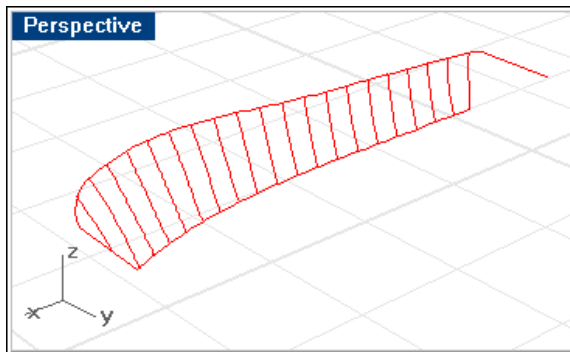
A good strategy on a surface that has subtle curvature like this is to use a few of the station lines, make them simple curves in Rhino and then loft a surface through them.

In this tutorial, the general steps we will follow are:

- 1 Copy the station lines we need to build the surface, saving the existing lines to check the final surface against.
- 2 Use Rhino to automatically fair and rebuild the curves to *simple curves*.
- 3 Manually fair stations and their relationship to other station lines.
- 4 Extend and copy lines to complete the lines for a simple surface.
- 5 Loft the hull surface.
- 6 Edit the surface for fairness and edge shape.
- 7 Trim the surface at the edges to get the desired edge.

Copy the lines needed to create the surface

- 1 Open the model **Fairing.3dm**.
- 2 On the **Designer curves** layer, use the **Copy** command with the **InPlace** option to duplicate the station lines of the topside and the sheer and chine line.
- 3 Change the layer for these curves to the **Surface curves** layer.

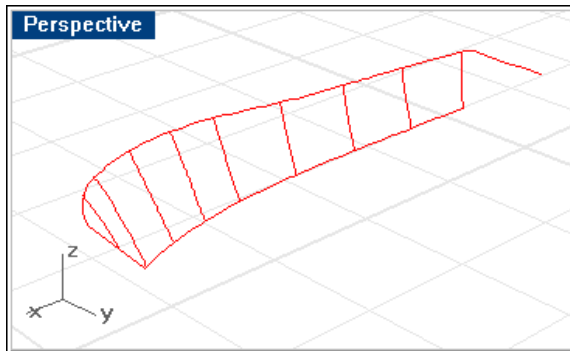


Curves from lines drawing.

- 4 Turn off the **Designer curves** layer.
- 5 There are many more curves here than we need, so delete about half the stations.

In areas where there is little change in shape from one to the next, you can delete more curves than in areas where there is more change. The following image illustrates the curves remaining.

You can delete two out of every three stations at the aft end of the surface and every other station toward the bow.



Curves needed to create surface.

Now you are ready to start fairing these curves.

Fairing to simple curves

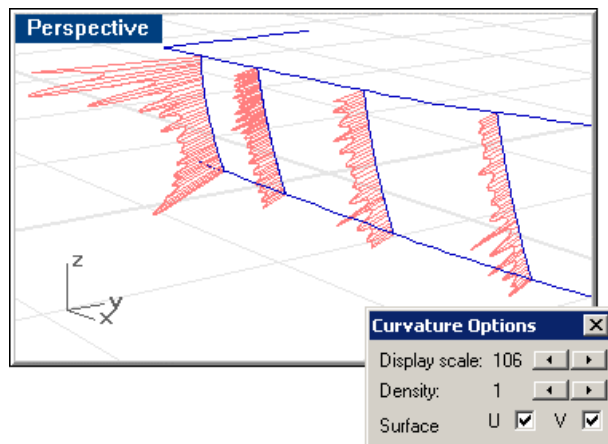
Curves have two characteristics in Rhino: the number of control points and degree. Degree determines how each control point affects the curve.

We are interested in *simple curves* made of a single polynomial span. Simple curves have the fairest curves possible and make very simple surfaces that are easy to edit.

To make simple curves in Rhino, create curves that have only one more control point than the curve's degree. Degree-1 curves (straight line) need only two control points; degree-2 curves need only three control points; degree-3 curves need only four control points, degree-4 curves need only five control points, and degree-5 curves need only six control points. If you find you need more than six control points, you are probably trying some extreme fairing, which makes fairing possible but much harder. Keep this idea of a simple curve in mind as we can look at the curves on the hull.

Analyzing the hull curves

Select a few stations and turn on the curvature graph. As you can see these curves are not fair. They have very dense curvature graphs that wiggle back and forth.



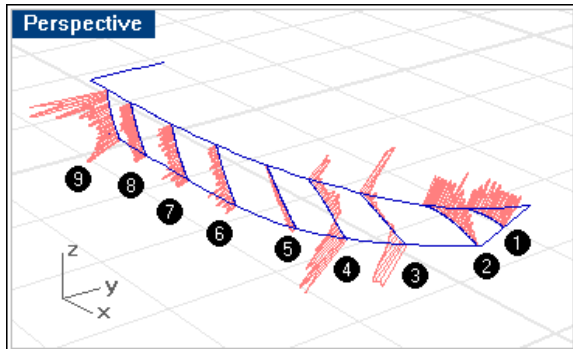
Hull curves curvature

Fairing the curves

To fair these curves, first determine how many points each curve will require. These are some general rules for determining how many points you need:

- If a curve should be a straight line, use two control points.
- If the curve is in only one direction (c-shaped), and you only need to control where and how much that curve changes, you will need a 3-degree curve with four-control points.
- If a curve is s-shaped, reversing curvature in the middle of the curve, you need a 5-degree curve with six control points.

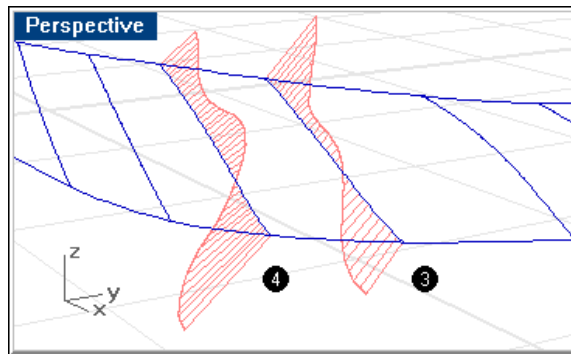
Based on the curvature graphs, curves 3 and 4 curve in two directions due to the transition from the bow flare, so we will rebuild them into 5-degree curves with six control points. Curves 1, 2, and 5 through 9 curve only one direction, so we will rebuild these into 3-degree curves with four control points.



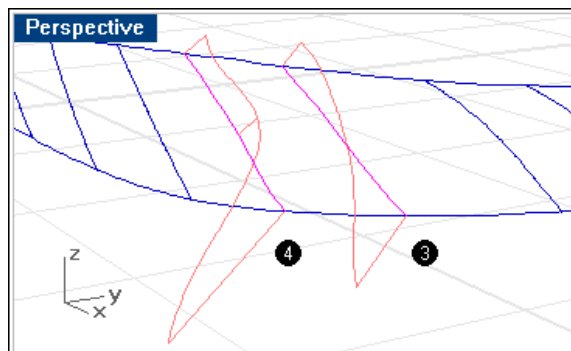
Curvature graphs on all hull curves.

To rebuild the curves:

- 1 Select curves 3 and 4.
- 2 Use the **Fair** command to automatically fair the curves to give you a start in the fairing process.

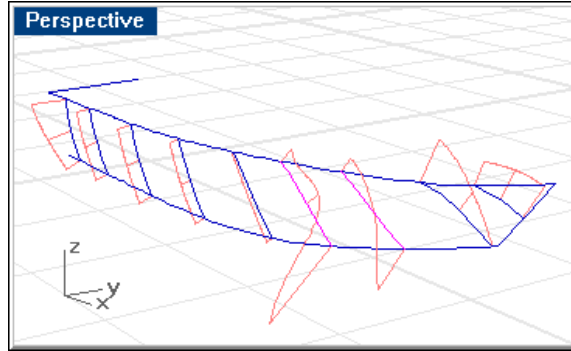


- 3 Use the **Rebuild** command to refine the curves further. Set the number of points to 6 and the degree to 5. Now the curves are simple curves.



- 4 Repeat the process the remaining curves rebuilding the curves with 4 control points and degree 3.

After this process of auto-fairing and rebuilding you can see that the curves are much more fair and the curvature graph is much less dense than before; both good signs that we are working toward fairness.



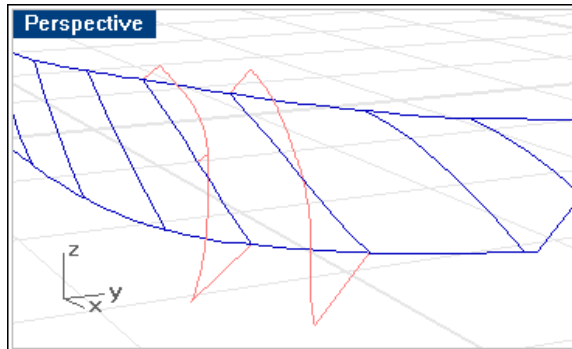
Rebuilt curves.

Refining the curve shapes

Now we will start moving points on the curves to continue fairing the curves and make sure they will create a fair surface.

We are looking for two things. First, we want to modify the curves so they have the shape we want. For instance, we must decide whether we want the curvature of the curve to be towards the ends of the curve or toward the middle. Second, how does one station line relate to the next? If you have a slight convex shape to one station and the next station has a little less curvature in it, the third station should have even a little less, so there is a smooth transition of curvature from one station to the next.

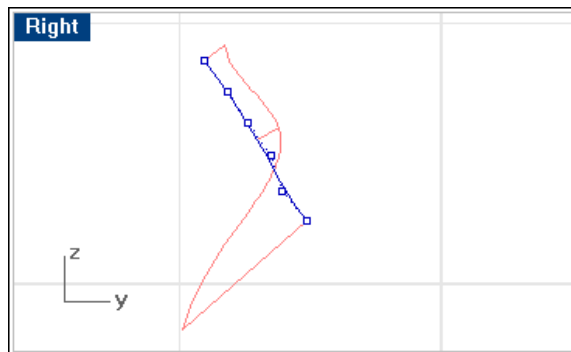
For this example, we are going to change the shape of stations 3 and 4 slightly.



Curves to refine.

To edit the curve:

- 1 Select curve **4** and use the **PointsOn** command turn on its control points.
- 2 Use the **CurvatureGraph** command to turn on its curvature graph.

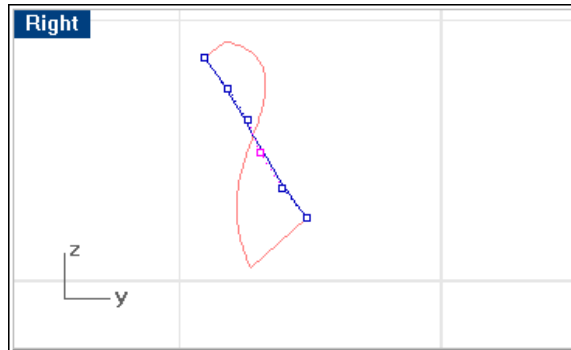


Editing an s-shaped curve.

Normally you should not edit the location of the points at the ends of the curve. This will keep the ends on the sheer and chine lines. That leaves four other points to edit.

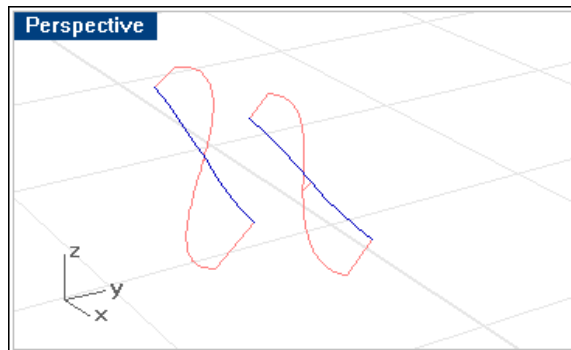
You can drag the points with the mouse to move them, but for the small editing of these points, you can use the **nudge** feature. By using nudge, you will get a lot finer control over the position of the points.

- 3 Select the point.
- 4 Hold down the **Alt** key and press the Down arrow key to move the point down to change the shape of the curvature graph so the lower part does not show such a dramatic change.



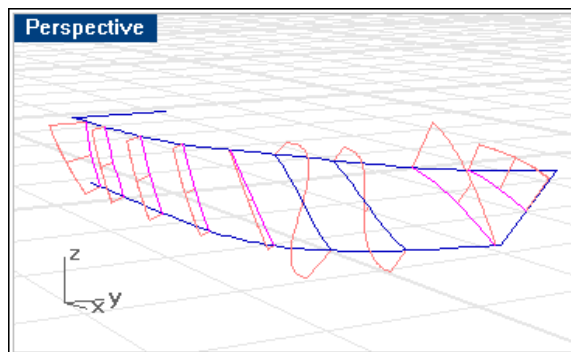
Control point moved to remove curve reverse.

- 5 Edit curve 3 so it is similar in shape.
In this case, we moved it down and slightly to the left.



Transitions shown by curvature graphs.

Nudging is sometimes easier if you set the option to use the construction plane axes rather than the world axes. Use the settings in the **Options** dialog box to change the nudge options.



Notice how the curvature of each station transitions from one curve to another.

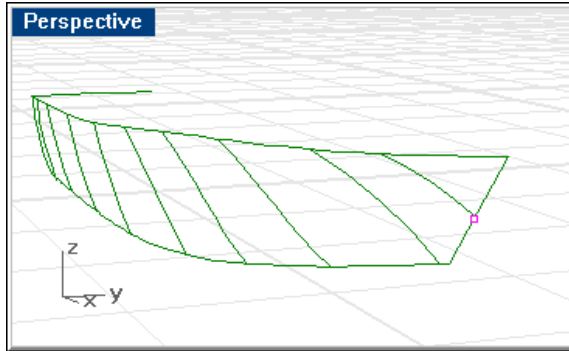
Bow profiles

When we use the **Loft** command to make this surface, if we include the straight bow stem as a section, the resulting surface will not be fair. So we will extend the bow station beyond the stem line, so the surface will pass through bow profile and continue on past centerline.

To create a bow profile:

- 1 Use the **Point** command to place a point at the endpoint of the bow station and the stem line.

This point will serve as a reference when we extend the curve.

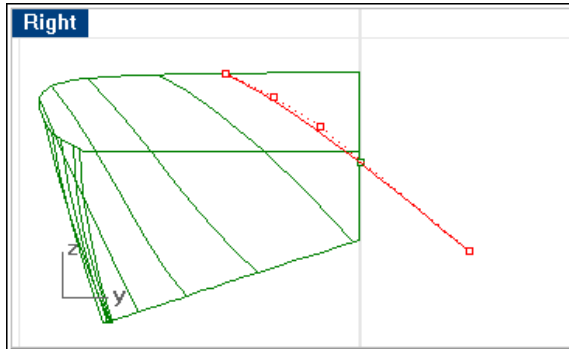


Bow reference point.

- 2 Use the **PointsOn** command to turn on the control points of the station line, and in the **Right** viewport, drag the station beyond centerline.

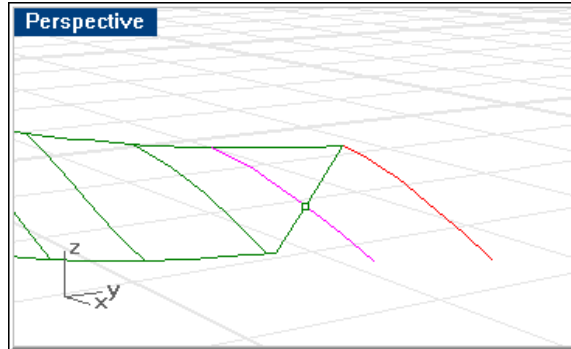
Make sure it stays intersecting the point on the stem.

You can check this by eye, or turn off the control points and use the **EditPtOn** command to turn on edit points, and move the closest edit point to the reference point on the stem.



The red curve is the extended station.

- 3 **Copy** this curve to the bow of the boat, so there will be two extended profiles at the bow.

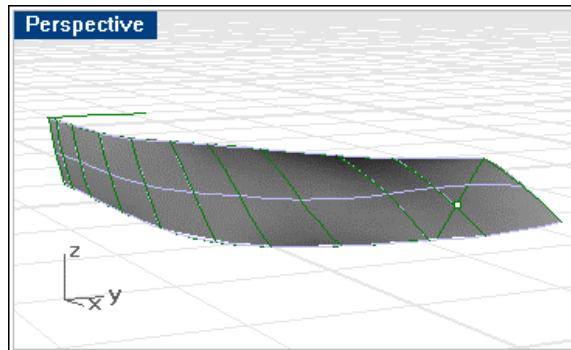


Copy the extended curve to end of sheer curve.

Create the Surface

To create the surface:

- 1 Select all the station curves.
Do not select the bow stem line.
- 2 Use the **Loft** command to create the surface.



Lofted surface.

The surface has very few isocurve lines. Because we used as few simple curves as possible, we have a simple surface that can be faired easily.

Assess the surface fairness

The next step is to see how fair this surface is. The three tools we use to do this are surface curvature color analysis, surface curve curvature graphs, and looking at the order of the control points on the surface.

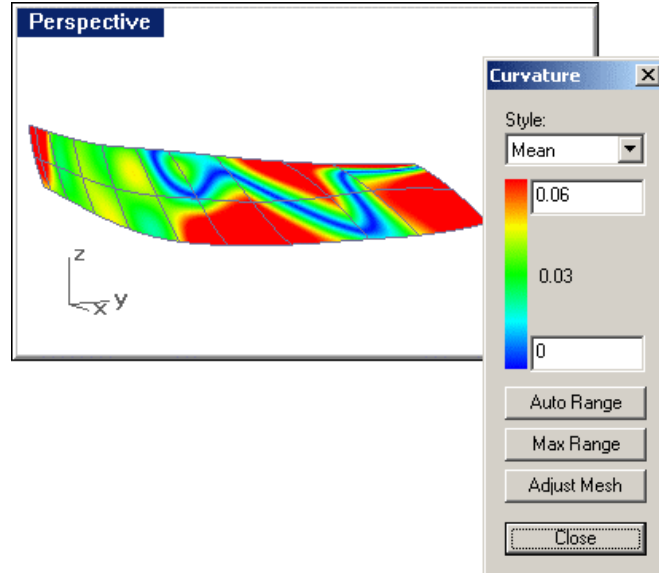
To use surface curvature analysis:

- 1 Select the surface.
- 2 Start the **CurvatureAnalysis** command.

- 3 In the **Curvature** dialog box, set the **Style** to **Mean**.

This displays colors on the surface based on the average curvature at each point.

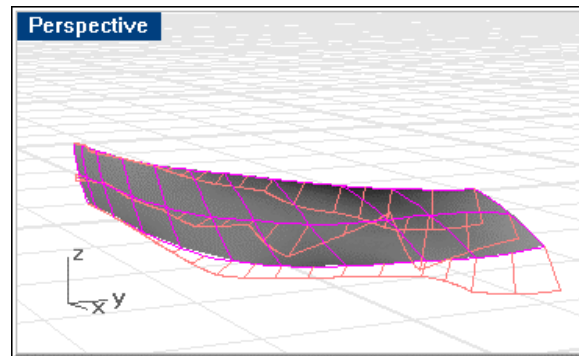
The color here is not as important as the pattern that the color creates. You want the pattern to smoothly transition from one color to another.



False color mean curvature analysis.

To show surface curvature:

- 1 Select the surface.
- 2 Start the **CurvatureGraph** command.
This is similar to the curvature graph that we used previously for curves. On a surface, the curvature graph displays on the surface isocurves and edges. There are two directions, U and V.
- 3 In the **Curvature Options** dialog box, turn off the V-direction.
The analysis should look something like this:

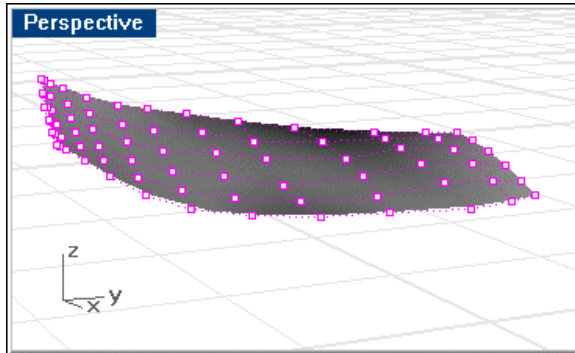


Surface analysis with curvature graph.

The curvature graphs here show that surface can be improved slightly.

To examine the visual order of the control points:

- 1 Select the surface.
- 2 Use the **PointsOn** command to turn the control points on.



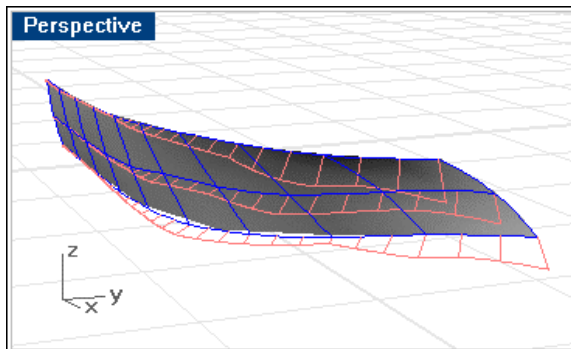
Surface analysis with control points.

Look for area that the control points take unusual turns or where they bunch up. You can improve these areas. In this case, the third and fourth stations show some uneven spacing.

Final fairing of the surface

When starting to fair the surface, you have a choice. The **Loft** command creates degree-3 surfaces. This will make the control point of the surface very close to the original points on the curves you used. You can leave this as it is, or you can use the **Rebuild** command to rebuild the surface to a degree-5 surface in the longitudinal (U) direction. Although the surface will still be on your original curves, the control points on the surface will be in a different place than the original curves. This may make the surface easier to fair.

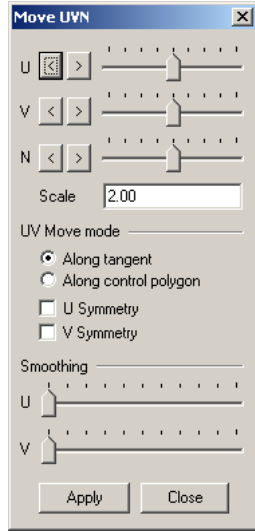
In this case, we chose to rebuild the surface, changing only the degree in the U direction to 5. This creates a surface that is slightly fairer as demonstrated by the smoother curvature graph curves.



Surface rebuilt in longitudinal direction.

To use control points to edit the surface

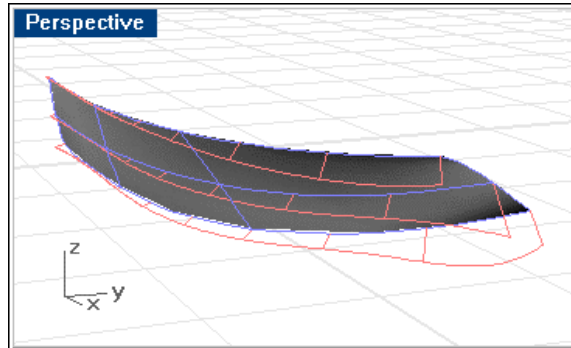
- 1 Use the **MoveUVN** command to edit the surface control points.



MoveUVN dialog box.

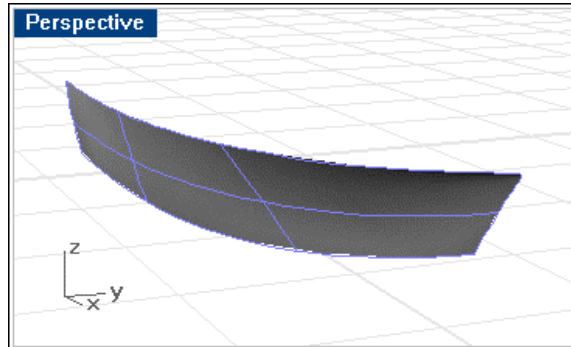
The **Move UVN** dialog box lets you drag the control points along the U and V directions of the surface and the normal direction. Use this to move points toward and away from the surface to make the surface fair.

As you move points around, you can get to a surface that looks like this:



Surface edited by moving control points.

- 2 Use the **Trim** command to trim the bow profile and the transom line from the surface to create your finished surface.



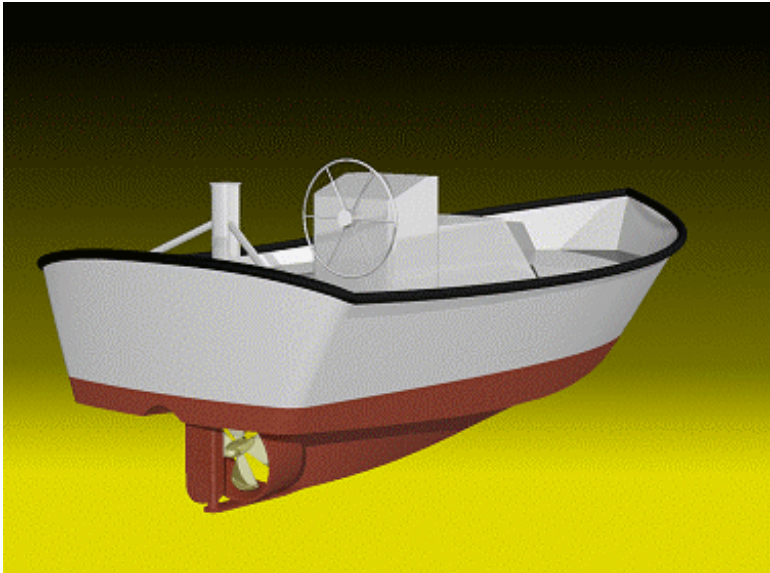
Trimmed surface.

To find out how much this surface deviates from your original curves

- 1 Turn on the layer with the original curves on it.
- 2 Use the **Pull** command to pull the original curves back to the surface.
- 3 Use the **CrvDeviation** command to determine how much the new pulled back curve deviates from the original curves.

The deviation should be very small: below the tolerance of your manufacturing process.

Part II: 17-Foot Seine Skiff



Introduction

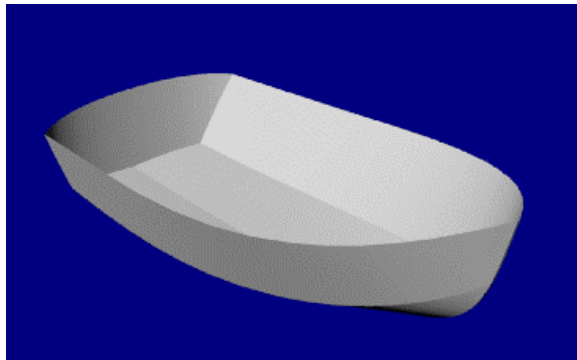
This tutorial will introduce you to the methods needed to create a fair hull of developable surfaces from a set of two-dimensional lines. You should be able to generate developable surfaces for any design that follows the rules of developability.

You will learn how to:

- Create 3-D curves from a 2-D lines drawing.
- Loft developable surfaces from the 3-D curves.
- Unroll developable surfaces to produce flat plate templates.
- Generate a fair hull with a limited number of control curves.
- Use analytical techniques to ensure a fair hull.

The vessel

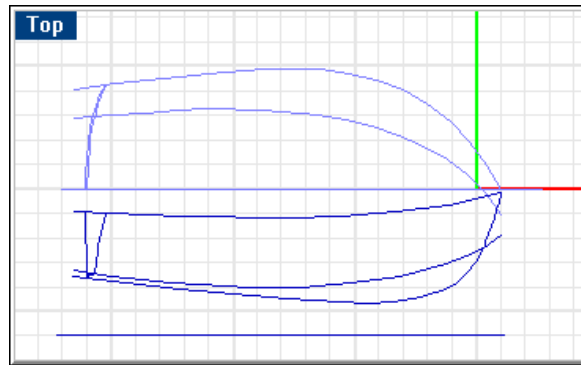
The vessel in this tutorial is a 17-foot aluminum seine skiff, similar to those used in the Alaska salmon fishery. This is not a tutorial on skiff design. The purpose of this tutorial is to instruct in the art of fairing developable surfaced hulls and unrolling the resulting plates.



The completed skiff hull.

The designer's lines are shown below. There are a couple of things to note about this lines drawing.

- There are no stations, waterlines, or buttocks. A typical set of lines would have such features, but since all surfaces are to be developable, these would serve no purpose in this drawing. The reasons why will become apparent as we go along.
- The chine and centerline have been extended at the forward and aft ends to accommodate the development process.



The designer's lines.

As we proceed through the tutorial, you will learn that the hull surfaces will be lofted larger than the required size and then trimmed back to the required profile. This is typical for all hull fairing in Rhino.

The files

This tutorial includes a lines drawing from which to work. The model is set up with the **Perspective** viewport set to shaded mode.

The files included with this tutorial include:

Skiff Lines.3dm – the lines drawing for the skiff.

Skiff Finished.3dm – The finished project.

Lay out the hull curves

The hull lines come from an unknown designer whose work you have no reason to suspect or respect. Therefore, as in any lofting project, check his lines for fairness before you begin laying surfaces over them. The first step in this process is to rebuild the lines you have been given, since CAD lines are notorious for containing many more points than necessary.

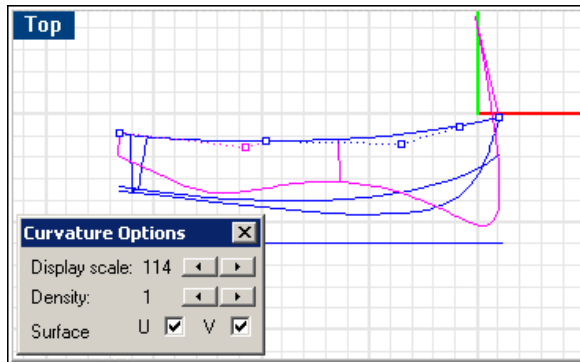
Other ways a curve can get a large number of points include:

- Curves projected to a surface.
- Curves resulting from using **Crv2Views**.
- Curves resulting from using **DupEdge** and **DupBorder**.

Check for fairness

Select each of the designer's curves in plan and profile and use the **CurvatureGraph** command to determine if you have been given fair curves to work with. If any curve is not fair, adjust points to make it fair. Start with the sheer. It is the curve with the least curvature, and it has the biggest impact on the appearance of the vessel.

The illustration below shows the curvature graph applied to the two-dimensional sheer in profile. You will note the wildness at the stemhead and the point further aft that is highlighted.



Check the sheer profile for fairness.

Step-by-Step: Check for fairness

Check the curves for fairness:

- 1 Select the curves you want to check.
- 2 Start the **CurvatureGraph** command.

Curvature graph

The curvature graph requires a little explanation. If you have ever laid down a curve on the drafting board using ducks and splines, you know how you can think it is fair. Then, when viewing it by squinting down the curve from an inch or two above the drafting board you notice lumps and bumps that were not visible when viewed from directly above.

This is a very mild example of the power of the curvature graph. Since you cannot squint your curves on the computer screen, the curvature graph is provided to do the squinting for you. It has an adjustable scale that lets you magnify the flaws in your curves. With this feature, a sheer can be faired to a much finer degree than that afforded by the squint method.

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.

All these conditions are evident in the sheer curve. The area of downward concavity is very slight and is right at the forward end of the sheer. The point of inflection is approximately 6 inches aft of the stem.

Rebuild the curves

Before doing any point editing to make the curves fair, we will 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.

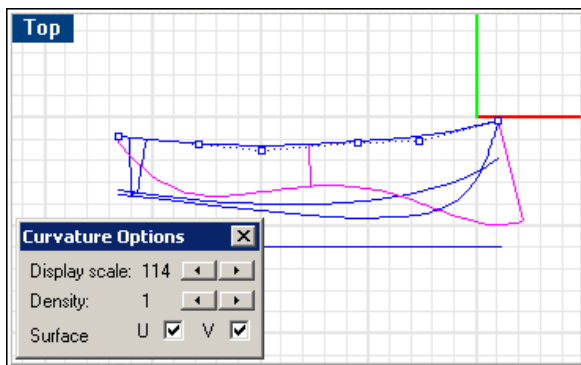
Use the values shown for curves that have extreme curvature. Otherwise, use lesser values as appropriate.

There are two important things to note:

- Do not use more points than you absolutely need. If you can get by with three points, do not use four.
- Make the degree one less than the number of points (for example, 3 points and degree 2; 6 points and degree 5).

Keep a copy of the original curves for checking the results. If the resulting curve is sufficiently close to the original, then do not increase the point count. The more you reduce the point count, the less trouble you will have with your surfaces later. Additionally, if the original curves are not fair, it will be much simpler to fair them if they have fewer points.

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.

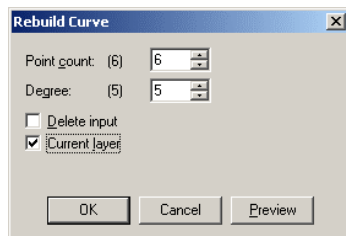


Curvature graph for sheer.

Step-by-Step: Rebuild the curves

Rebuild the curves:

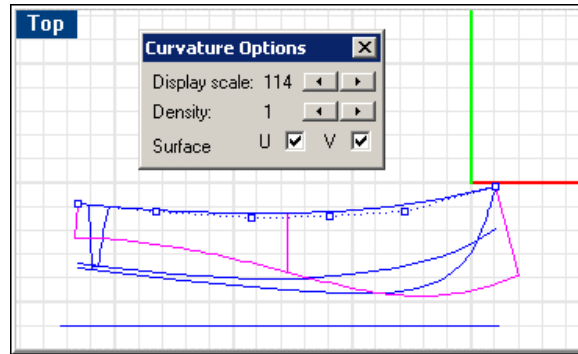
- 1 **Select** the sheer curve.
- 2 Start the **Rebuild** command.
- 3 In the **Rebuild Curve** dialog box, change the **Point count** to **6** and the **Degree** to **5**.



Rebuild Curve dialog box.

If the curve needs editing:

- 1 **Select** a curve.
- 2 Use the **PointsOn** command or press F10 to turn on the curve's control points. You can use the mouse cursor or the nudge keys to move the points around until the curvature graph is satisfactory.



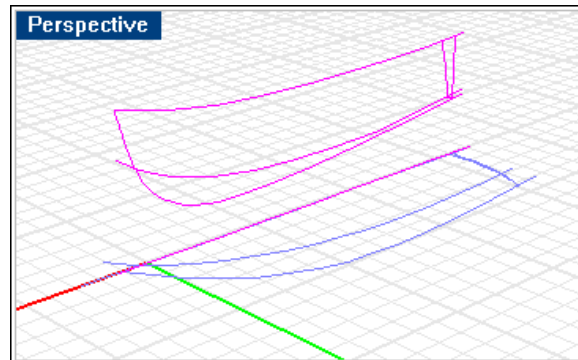
Edited curve and curvature graph.

Stand the profile up

The skiff lines drawing is a typical two-dimensional lines drawing with profile and plan all in the same plane. Unlike most lines drawings, the skiff has no body plan. Before you can use this lines plan for developing surfaces, it will be necessary to stand the profile up so that its y-dimensions become z-dimensions.

This will facilitate the next step—creating the three-dimensional curves that will be used in lofting the surfaces.

- 1 **Select** all the profile curves (sheer, chine, stem, forefoot, run and baseline).
- 2 **Move** these so the baseline goes through 0,0.
- 3 In the **Right** viewport, **Rotate** the curves **90** degrees about the 0,0 point.

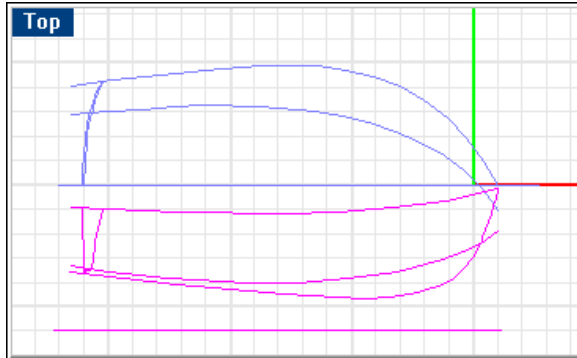


Stand the profile up.

Step-by-Step: Stand the profile up

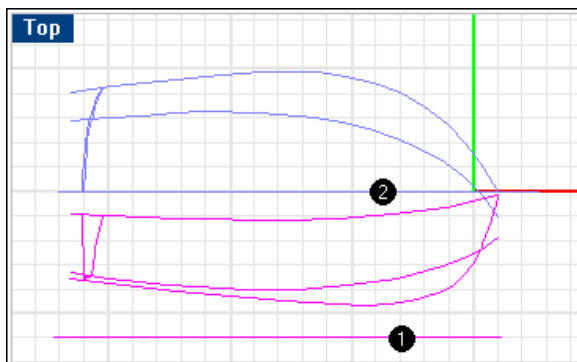
Move the profile curves to the centerline:

- 1 Use the **SelLayer** command to select all the curves on the **Profile** layer.



The profile curves.

- 2 Start the **Move** command.
- 3 At the **Point to move from ...** prompt, use the **Near** object snap to pick a point on the baseline (1).
- 4 At the **Point to move to** prompt, use the **Perp** object snap to pick a point on the plan view centerline (2).

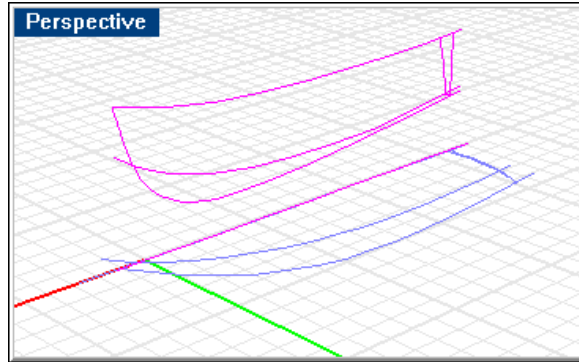


Move the profile curves to the baseline.

Rotate the curves so they are perpendicular to the plan curves:

- 1 With the profile curves still selected, start the **Rotate** command.
- 2 In the **Right** viewport, at the **Center of rotation ...** prompt, enter or pick **0,0**.

- 3 At the **Angle or first reference point ...** prompt, with **Ortho** on, drag the angle to **90** degrees.

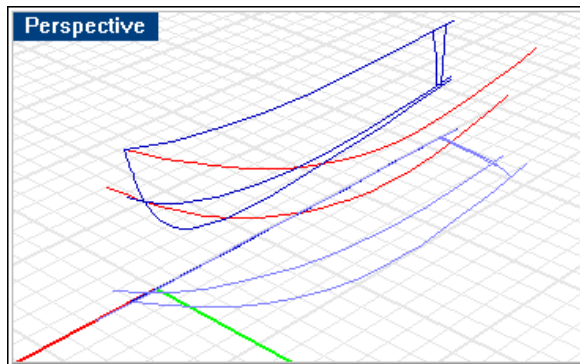


Rotate the profile curves around the baseline.

Create the three-dimensional 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 **3-D 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. In the illustration, the red curves in the image below have been created from the blue ones. These red curves will be used in developing the hull surfaces.



The three-dimensional chine and sheer curves.

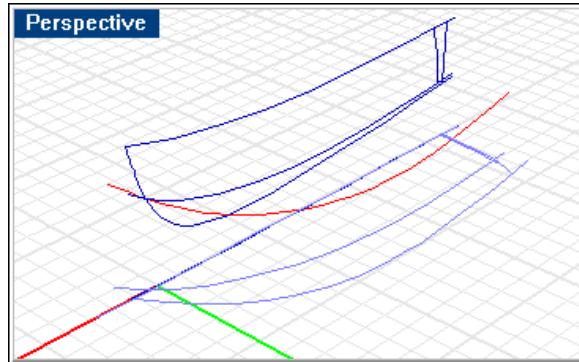
Step-by-Step: Create the three-dimensional curves

Create the three-dimensional curves:

- 1 Set the **3-D Lines** layer current.
Start the **Crv2Views** command.

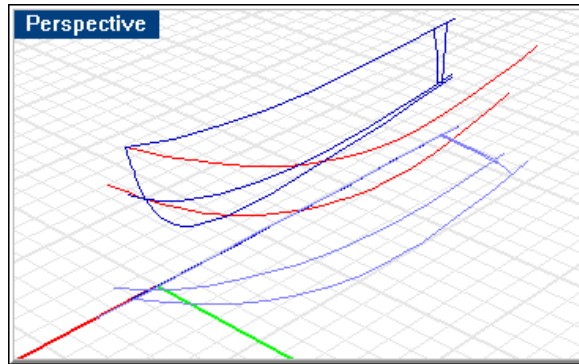
- 2 At the **Select planar curve** prompts, select the plan and profile representations of the chine curve.

The three-dimensional representation of that curve will be created.



Create a 3-D curve from the plan and profile chine.

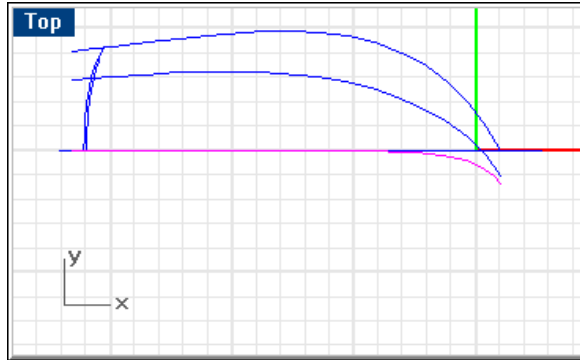
- 3 When you are satisfied that the proper curve was created, **delete** or **Hide** the two-dimensional representations.
- 4 Repeat the **Crv2Views** command for the sheer curve.



Create a 3-D curve from the plan and profile sheer.

About the curves

As previously discussed, the chine and centerline have some funny business at the bow. This is intentional. For the loft process to work on the bottom panel it is imperative that it not come to a point. The eventual bottom will come to a point after trimming; however, the lofted panel cannot. Therefore, the chine extends into negative space; the bottom surface is developed from it and trimmed to the centerline (the half-inch buttock in this case). In addition, since the chine and centerline come together at a point, it is necessary to construct a false centerline (as shown in the illustration below that also goes into negative space and in profile is below the actual centerline. This false centerline joins the real centerline tangentially at the aft end of the forefoot. The real centerline can be used from this point aft.



Extend curves into negative space.

To develop the side panel this is not strictly necessary; however, developing the side panel to just the forward end of the chine will mean that the forward edge (the one that must mate with the stem) will probably curve outboard along its length. The side panel is therefore extended into negative space and trimmed to the half-inch buttock as well.

If the profile that results from these changes is too far removed from the designer's intent, modify the curves in negative space and try again.

Loft the hull surfaces

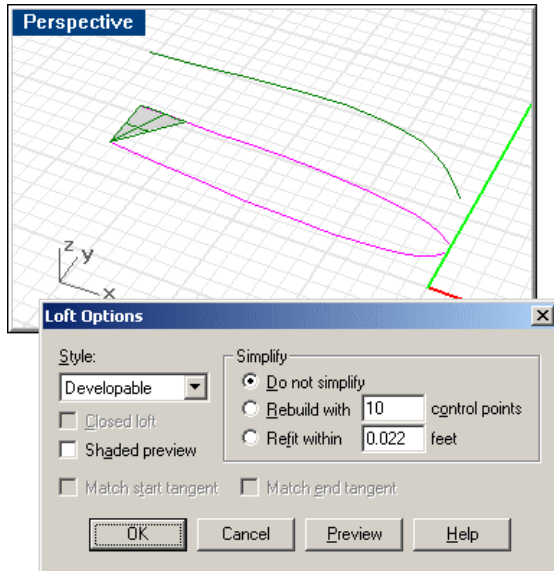
Now that you have created a set of edge curves for the side and bottom, create lofted developable surfaces from these curves. This assumes the curves actually define the edges of developable surfaces. In the case of this model, they do. However, the centerline curve has been modified slightly so that your first effort to develop the bottom surface will fail. Start by lofting the bottom surface, since it is the more shapely of the two. Once you have finished it, you can use its upper edge as the chine from which to develop the side panel. In this way, if it is necessary to modify the bottom panel by "tweaking" the chine, you will not have to discard and redo the side panel.

There are no stations to use for lofting, only the top and bottom edges of each panel.

Loft the chine and centerlines

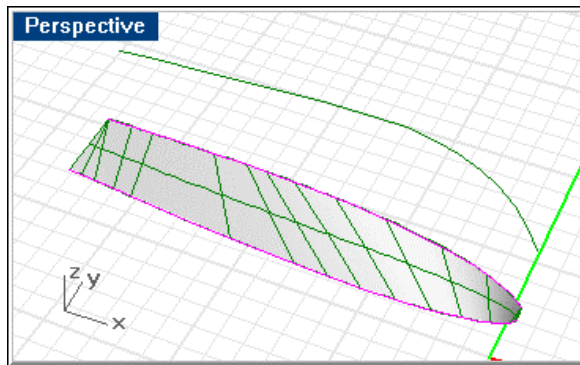
To loft the bottom panel, select the two edges (chine and centerline) and use the **Loft** command with the **Developable** style. In this case, be sure to select the 3-D centerline, since it extends into negative space unlike the profile centerline. If the edges describe a developable surface, the proposed surface and the Loft Options dialog box display.

If the edges do not represent a developable surface, the **Loft** command will display as much of the surface as can be developed. In this case, only part of the surface is developable.



The part of the surface that is developable.

Cancel the **Loft** command, modify one or the other of the edge curves, and try again. In this case moving the next to last control point on the chine a very small amount in the y-direction was enough.



A developable loft surface.

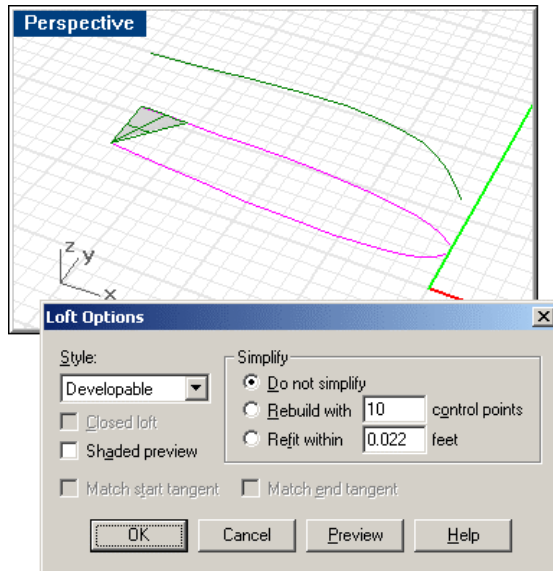
Step-by-Step: Loft the chine and centerlines

Loft the side and bottom and the chine and centerline:

- 1 Select the chine and 3-D centerline.
Start the **Loft** command.

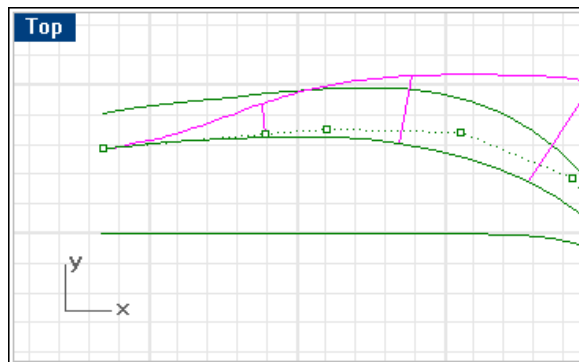
- 2 Complete the **Loft Options** dialog box as shown.

The proposed surface displays. In this case, only a small portion of the surface is developable.



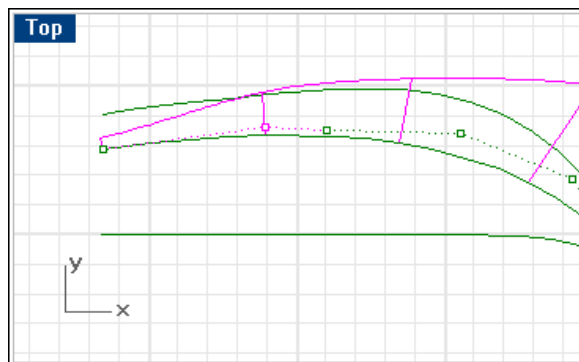
The part of the surface that is developable.

- 3 Click **Cancel**.
- 4 In the **Top** viewport, turn on the **control points** and **curvature graph** for the chine.



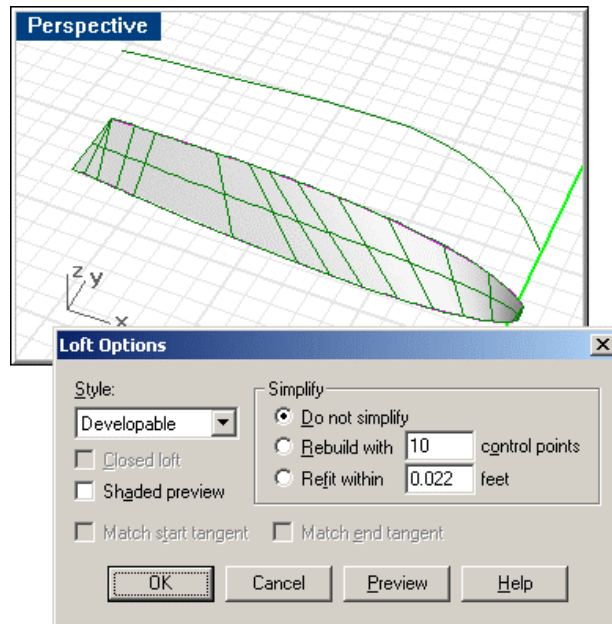
Turn on control points and curvature graph.

- 5 Drag the next-to-last control point in the y-direction a small amount.



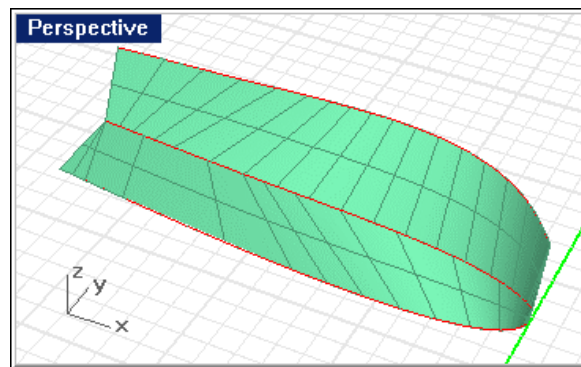
Drag the control point.

- 6 **Loft** the centerline and chine again.



A developable surface.

- 7 Repeat the **Loft** for the side panel, selecting the chine and sheer.



The lofted bottom and side panels.

Trim the bow to the buttock line

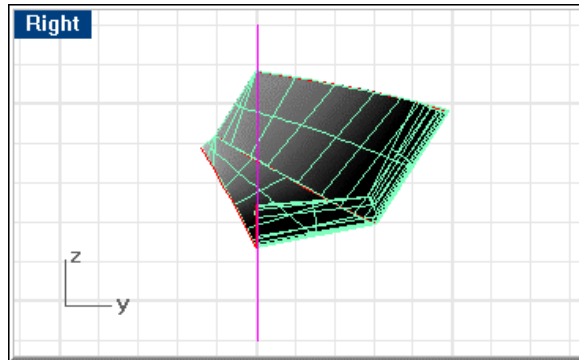
When you have successfully created both the side and bottom surfaces, construct a buttock 1/2 inch off the centerline and trim both surfaces to this buttock. To do this, in the **Right** viewport, draw a line from below the baseline to above the tip of the bow 1/2 inch to the right of centerline.

Step-by-Step: Trim the bow to the buttock line

Draw the trim line:

- 1 Start the **Line** command.
- 2 In the **Right** viewport, at the **Start of line ...** prompt, and type **.04,-1**.

- 3 At the **End of line ...** prompt, type **.04,7**.

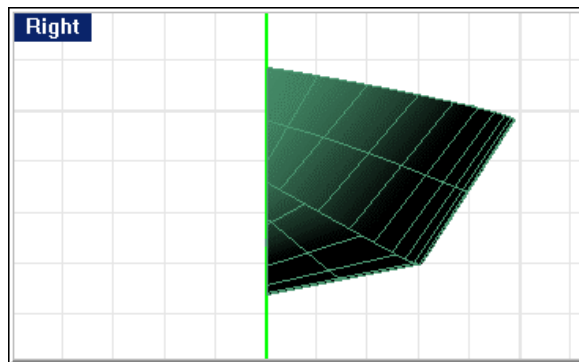


Trimming line in Right viewport.

Trim the bow to the buttock line:

Because of the way they were lofted, the hull side and bottom are polysurfaces. Polysurfaces cannot be trimmed with a line, so you have to **explode** them first.

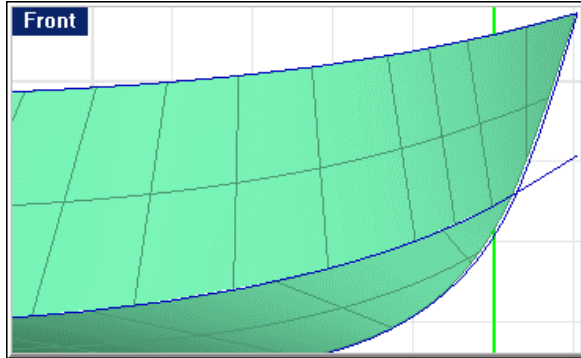
- 1 **Explode** the hull side and bottom.
- 2 Select the buttock trim line.
- 3 Start the **Trim** command.
- 4 At the **Select objects to trim ...** prompt, in the **Right** viewport, select the side panel and bottom panel to the left of the buttock line.
- 5 **Delete** the buttock trim line and any surfaces left over after trimming
- 6 Use the **SelSrf** command to select all the surfaces.
- 7 Use the **Join** command to join all the hull surfaces together.



The trimmed hull surfaces.

Check the results

Inspect the centerline in the **Front** viewport (profile) to see how much the developed surface differs from the designer's intent. There is very little difference. The comparison can be seen in the illustration below. The blue lines are the original forefoot and stem lines from the profile, while the trimmed surfaces are shown in green.



Compare the developed bow to the designer's bow.

Build the transom

Like all surfaces in this tutorial, the transom will be built with extra material and then trimmed to the hull. This facilitates joining to the hull later.

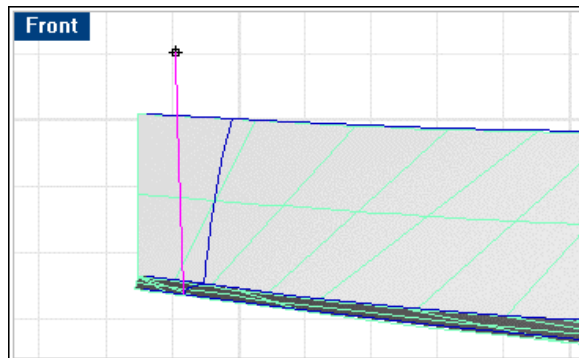
Extend the centerline

To make sure there is extra, extend the transom centerline by a foot or two both above the sheer and below the centerline.

Step-by-Step: Extend the centerline

Extend the centerline:

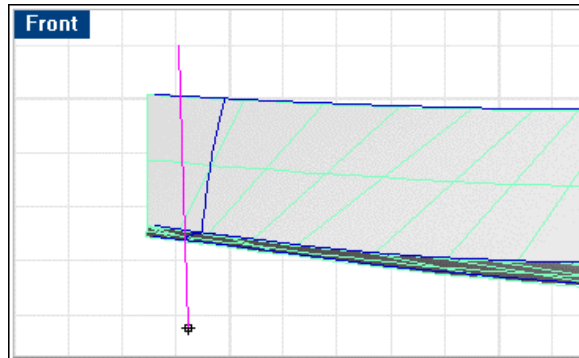
- 1 Start the **ExtendByLine** command.
- 2 At the **Select object to extend ...** prompt, in the **Front** viewport, select near the top of the transom centerline.
- 3 At the **End of extension** prompt, select a point approximately 12 inches above the current top of the transom centerline.



Extend the transom centerline up.

- 4 Repeat the **Extend** command.

- 5 At the **Select object to extend ...** prompt, select near the bottom of the transom centerline.
- 6 At the **End of extension** prompt, select a point approximately 12 inches below the current bottom of the transom centerline.

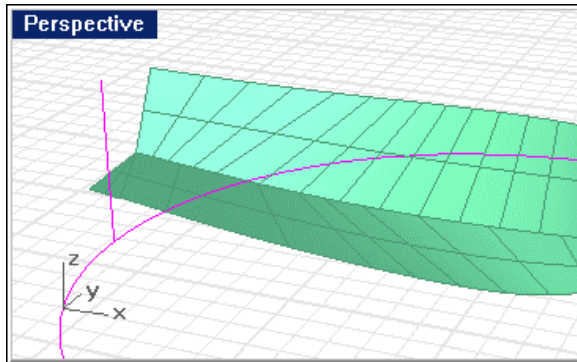


Extend the transom centerline down.

Draw the construction circle

The transom on this vessel, like on most steel and aluminum vessels of this size, is a portion of a tilted cylinder.

Draw a **Circle** with a radius of 11'-4" around the transom centerline using the **AroundCurve** option. **Move** the circle from its aft quadrant to the endpoint of the centerline as shown.



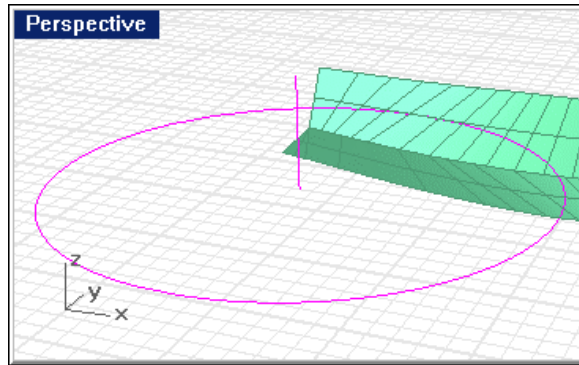
Transom construction circle.

Step-by-Step: Draw the construction circle

Draw the construction circle:

- 1 From the **Curve** menu, click **Circle** and then click **Center, Radius**.
- 2 At the **Center of circle (... AroundCurve)** prompt, in the **Top** viewport, type **A**.
- 3 At the **Point on curve for center of circle** prompt, use the **Near** object snap to pick near one end of the extended transom centerline.

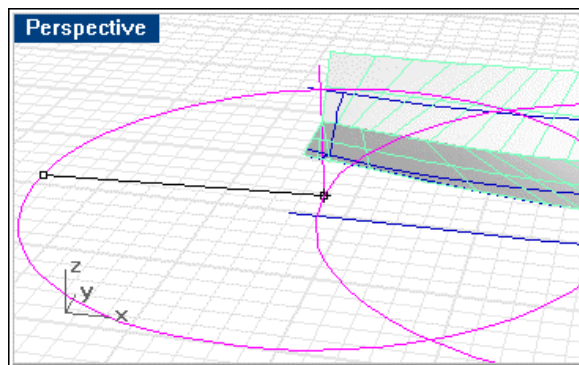
- 4 At the **Radius <1> ...** prompt, type **11'4"**.



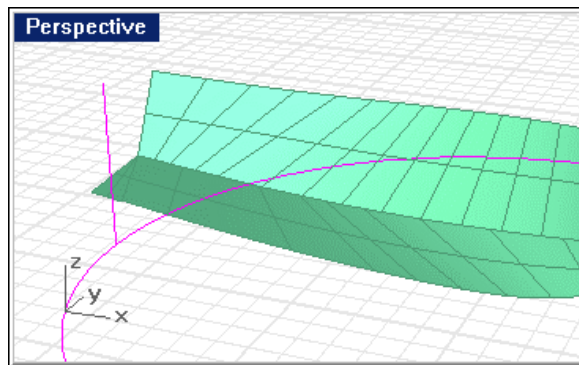
The transom construction circle.

Move the circle to the transom centerline:

- 1 Select the circle.
- 2 Start the **Move** command.
- 3 At the **Point to move from ...** prompt, use the **Quad** object snap to pick the circle at the aft quadrant.
- 4 At the **Point to move to** prompt, use the **End** object snap to pick the lower end of the transom centerline.



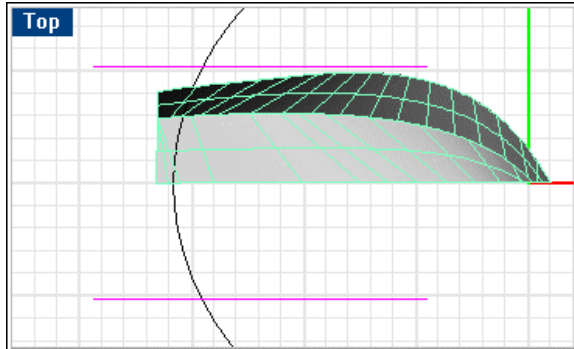
Move the circle quadrant to the endpoint of the transom centerline.



The circle in position.

Draw the trimming lines

In the **Top** viewport, draw a line from just outboard of the maximum beam, parallel to centerline to intersect this circle and mirror it across centerline.



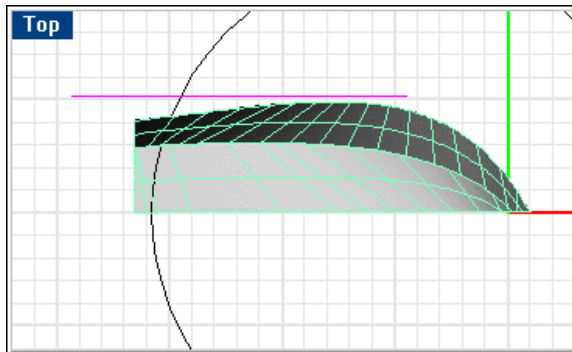
Trimming lines for the transom construction circle.

Note: The transom surface that will be extruded from this circle segment should be at least as wide as the sheer at midship so it can be trimmed to the side panel. This is the reason you are trimming the circle to a width slightly greater than the maximum beam.

Step-by-Step: Draw the trimming lines

Draw the trimming line:

- 1 Start the **Line** command.
- 2 In the **Top** viewport, draw a line just outboard of the maximum beam that extends from inside the circle to outside of it.

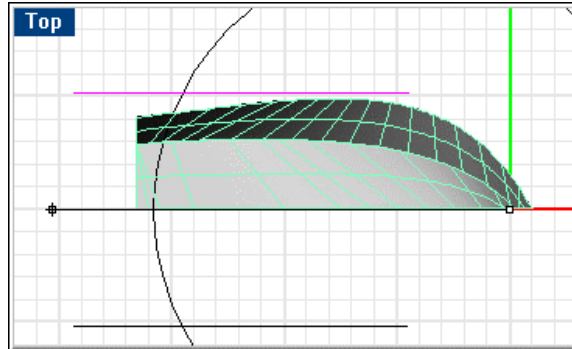


Transom circle trimming line.

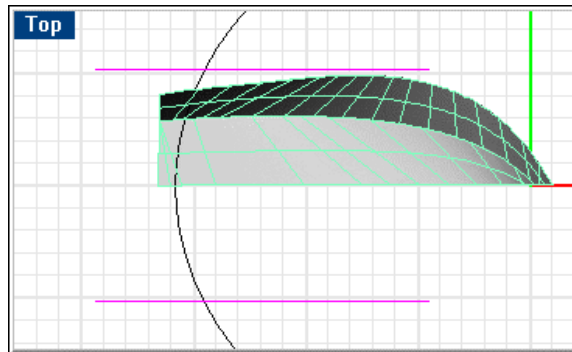
Mirror the trimming line:

- 1 Select the line.
- 2 Start the **Mirror** command.
- 3 In the **Top** viewport, at the **Start of mirror plane ...** prompt, type **0**.

- 4 At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line along the x-axis.



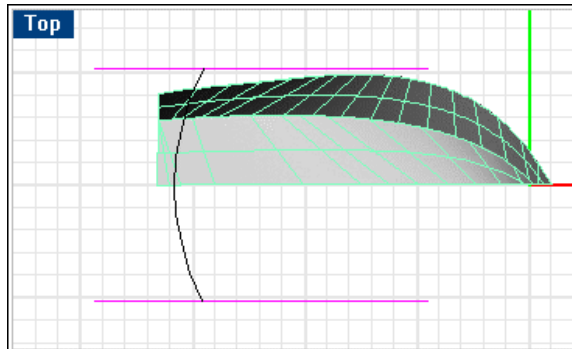
Mirror the transom circle trimming line.



The transom circle trimming lines.

Trim the construction circle

Trim the circle to the two lines.



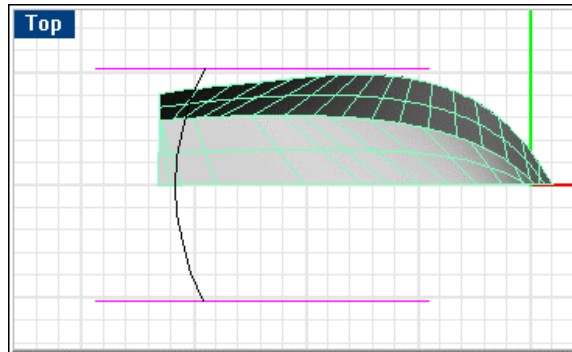
The trimmed transom circle.

Step-by-Step: Trim the construction circle

Trim the curve:

- 1 Select the two lines.
- 2 Start the **Trim** command.

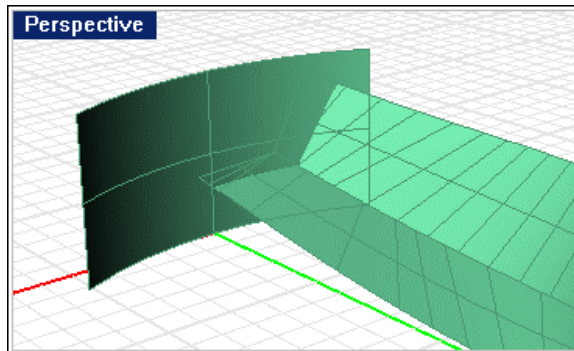
- At the **Select object to trim ...** prompt, select the circle outside the lines.



The trimmed transom circle.

Extrude the surface

To create the transom surface, **extrude** the trimmed circle along the transom centerline.

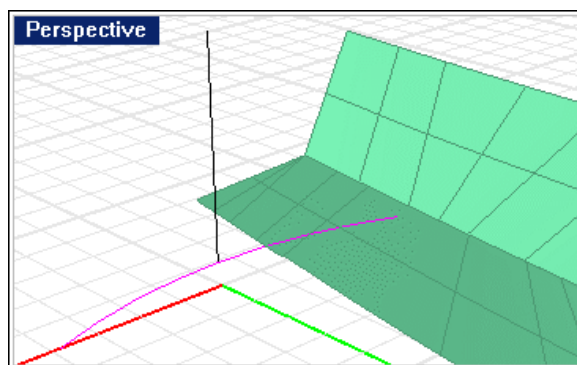


The constructed transom cylinder.

Step-by-Step: Extrude the surface

Extrude the surface:

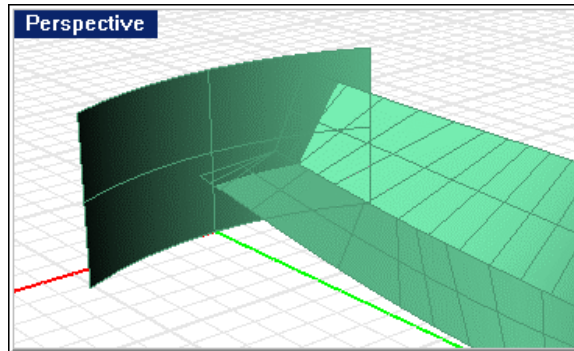
- Select the trimmed circle segment.



The trimmed circle segment.

- Start the **ExtrudeAlongCrv** command.

- At the **Select path curve** prompt, select the transom centerline near the lower end.



The constructed transom cylinder.

Build the camber curve

To build the camber curve, it will be necessary to know where the transom corner will intersect the sheer, even before you have trimmed the transom to the sheer.

Intersect the side and transom panels

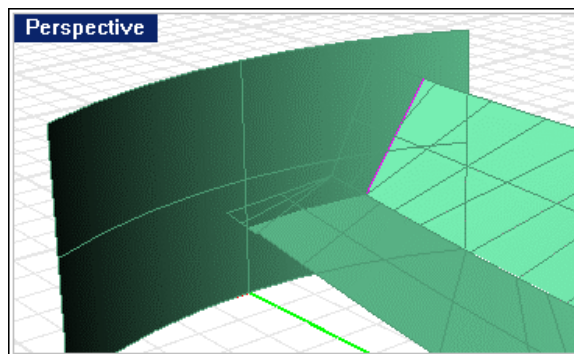
To find this intersection, select the side and transom panels and use the **Intersect** command. The result will be a curve where the two surfaces meet. **Mirror** this line across centerline.

Step-by-Step: Intersect the side and transom panels

Create the transom/side intersections:

- Select the transom surface and the side panel.
- Start the **Intersect** command.

The result will be a curve that represents the intersection of these two surfaces.



The intersection of the transom and side.

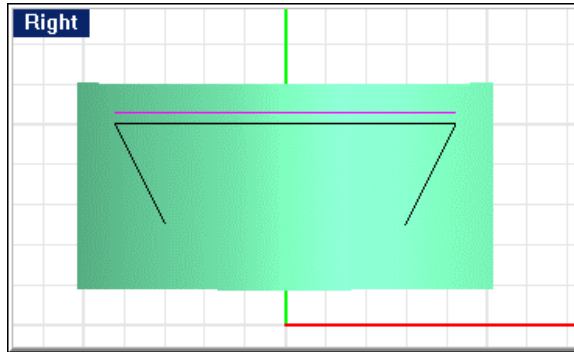
Mirror the intersection curve:

- With the intersection curve still selected start the **Mirror** command.
- In the **Top** viewport, at the **Start of mirror plane ...** prompt, type **0**.
- At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line along the x-axis.

Draw construction lines

To build the camber curve, determine how much camber you wish to have in the transom. A camber of slightly greater than three inches was used in the example.

In the **Right** viewport, draw a line from the starboard corner of the transom to the port corner and copy it upward an amount equal to the desired camber.

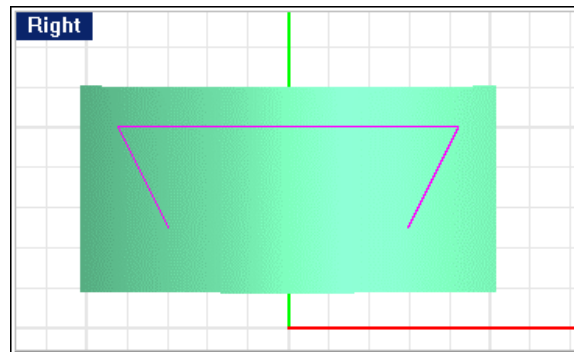


Camber curve construction lines.

Step-by-Step: Draw construction lines

Draw construction lines:

- 1 **Hide** the side and bottom surfaces.
- 2 Start the **Line** command.
- 3 At the **Start of line ...** prompt, in the **Right** viewport, use the **End** object snap to pick the top of the port intersection curve.
- 4 At the **End of line ...** prompt, use the **End** object snap to pick the top of the starboard intersection.

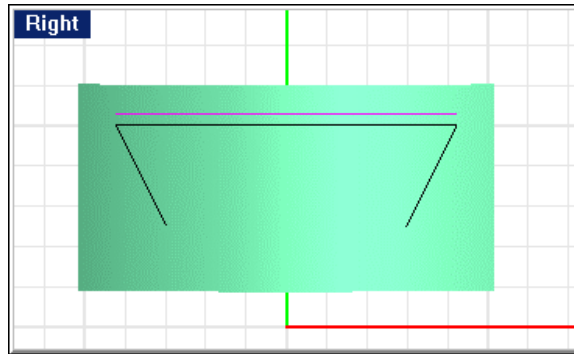


Line between endpoints of the intersection curve.

Offset the construction line:

- 1 Select the horizontal line.
- 2 Start the **Offset** command.

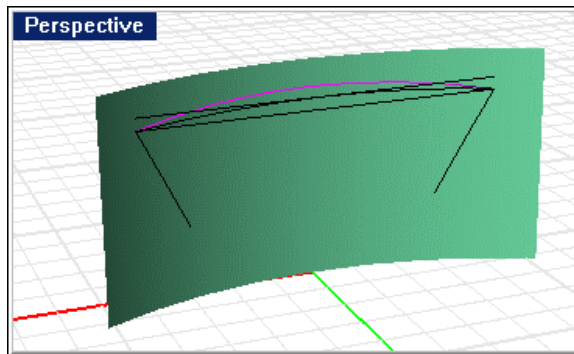
- 3 At the **Side to offset (Distance=3.5 ...)** prompt, type **3.5"**.
- 4 Click above the horizontal line.



Offset the horizontal line.

Draw the camber arc

Draw an arc from the starboard end of the lower of these two lines to the middle of the upper of these two lines and to the port end of the lower line. This arc will be your camber curve. **Project** the arc onto the transom cylinder in the Right viewport. The projected arc will terminate at the upper end of the intersection between the side and transom.



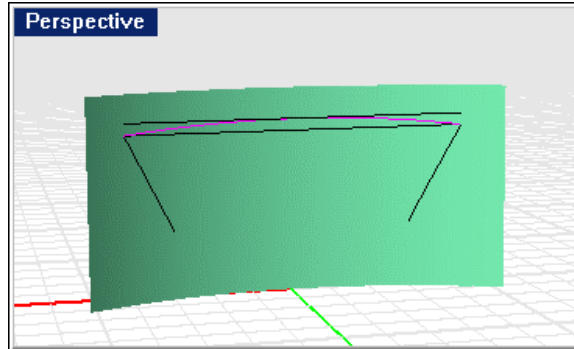
Project the offset line to the transom surface.

Step-by-Step: Draw the camber arc

Draw the camber arc:

- 1 Start the **Arc3Pt** command.
- 2 At the **Start of arc** prompt, in the **Right** viewport, use the **End** object snap to pick the top end of the port intersection line.
- 3 At the **End of arc** prompt, use the **End** object snap to pick the top end of the starboard intersection curve.

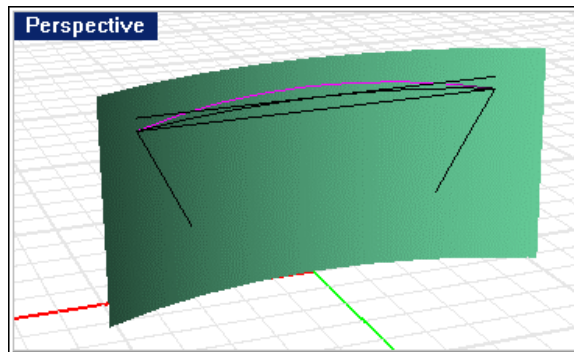
- At the **Point on arc** prompt, use the **Mid** object snap to pick the midpoint of the upper horizontal line.



Draw the arc.

Project the camber arc:

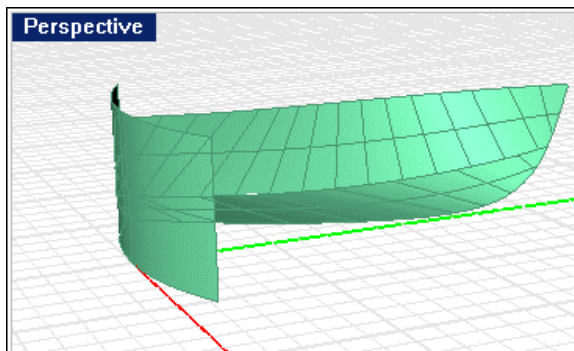
- Select the transom surface and the arc.
- In the **Right** viewport, start the **Project** command.
The projected arc will terminate at the upper end of the intersection between the side and transom.



Project the arc to the transom surface.

Trim the transom

Trim the bottom and side panels to the completed transom cylinder. This will complete the bottom and side. To do this, use the **Trim** command and select the cylinder as the cutting edge. Trim both panels.

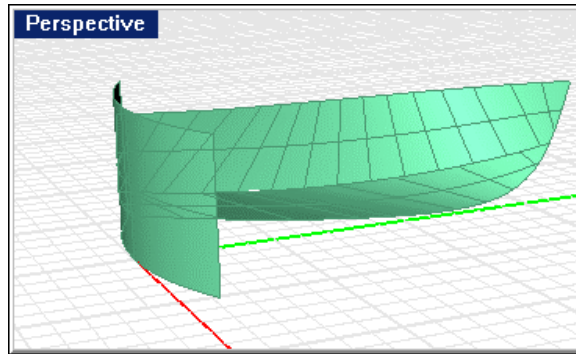


The side and bottom panels trimmed to the transom.

Step-by-Step: Trim the transom

Trim the transom:

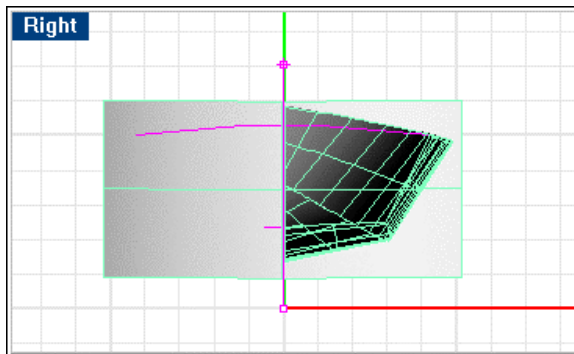
- 1 Select the transom surface.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, select the side panel then the bottom panel outboard of the transom.



The side and bottom panels trimmed to the transom.

Create the centerline and deadflat

At this point, we have almost all the curves we need to trim the transom. Remember that you trimmed the bottom panel one-half inch off centerline to accommodate a 1-inch keel. You need a straight horizontal line from the inboard end of the bottom panel at the transom to cross centerline and a vertical line at centerline in the **Right** viewport.



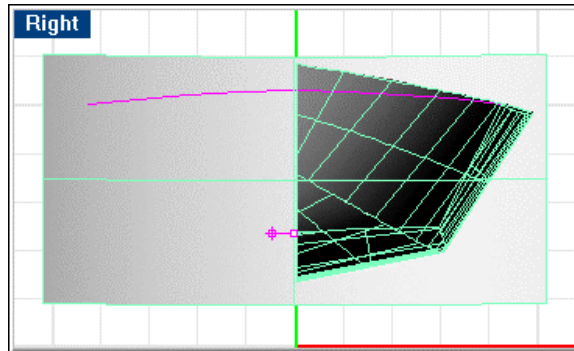
Trimming line.

Step-by-Step: Create the centerline and deadflat

Create the centerline and deadflat:

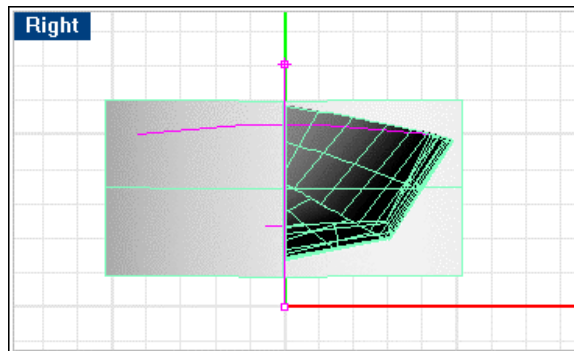
- 1 Start the **Line** command.
- 2 At the **Start of line ...** prompt, in the **Right** viewport, use **End** object snap to pick the aft lower corner of the bottom panel.

- 3 At the **End of line ...** prompt, pick a point just to starboard of centerline, using **Ortho**, but no object snap.



Trimming line across the centerline.

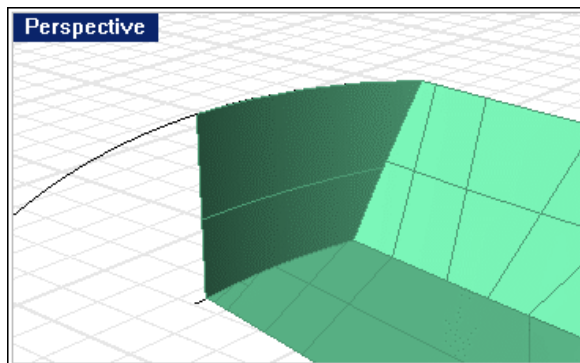
- 4 Repeat the **Line** command.
- 5 At the **Start of line ...** prompt, type **0,-1**.
- 6 At the **End of line ...** prompt, type **0,7**.



Centerline trimming line.

Trim the transom surface

Trim the transom cylinder to the centerline you just created and then to the bottom and side using the bottom and side surfaces, the transom camber curve, and the little half-inch line we just constructed.

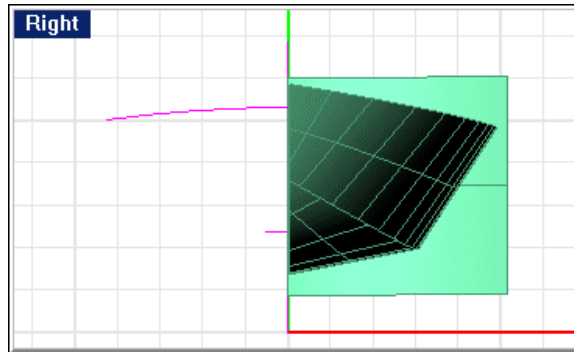


Trimmed transom surface.

Step-by-Step: Trim the transom surface

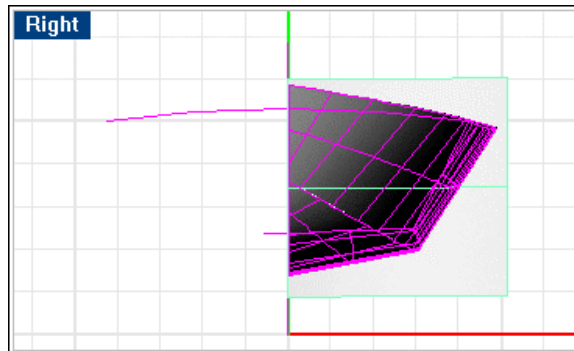
Trim the transom cylinder:

- 1 Select the vertical centerline you created in the last step.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, in the Right viewport, select the starboard side of the transom cylinder.



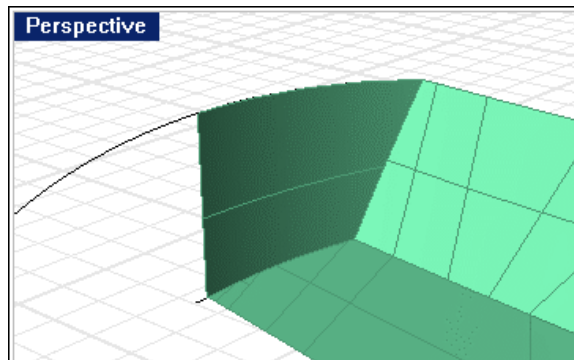
Trim the starboard side of the transom surface.

- 4 Select the projected arc, the side surface, bottom surface, and the short line.



Trim the other parts of the transom surface.

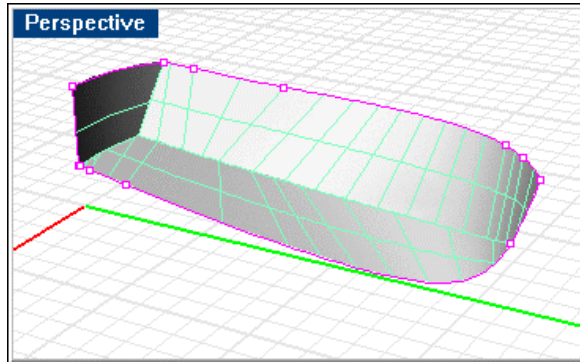
- 5 Start the **Trim** command.
- 6 Select the transom surface above the projected arc.



The trimmed transom.

Complete the transom and check for unjoined edges

The transom is now complete. **Join** the transom to the bottom and side. Use the **ShowEdges** command to check that the join was successful. 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 three surfaces.



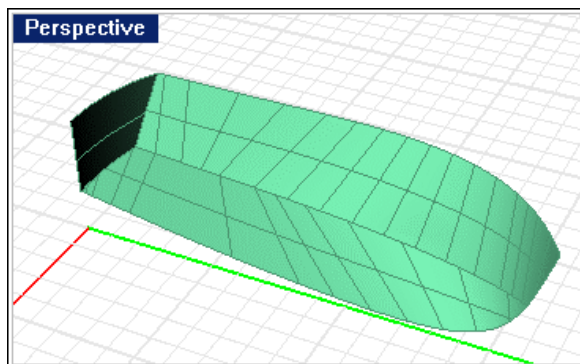
Transom, side, and bottom joined, showing naked edges only at outside edge.

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

Analyze the surfaces

You have finished creating the hull surfaces, and you can now look at their fairness in detail. For this process, use the shaded model as well as the **EMap**, **Zebra**, and **CurvatureAnalysis** commands. With these features, small problems with the hull that do not ordinarily show up when shading or rendering become more pronounced. It is like turning off all the lights and standing beside the hull with a lamp on at the far end of the room. Small ripples and flaws show more in this light.

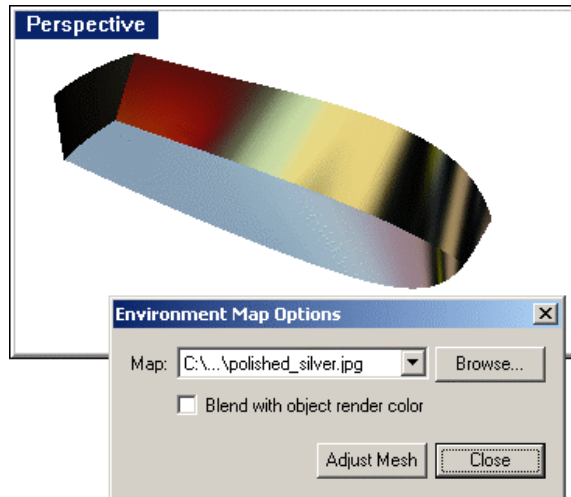
Note: Because this model was created with developable surfaces (including the transom, which is a cylinder), there should be no unfairness at all. However, it is good practice to check.



Model shaded with wireframe display. No unfairness evident.

Environment map analysis

In the illustration below, you see the result of applying an environment map (**EMap** command) to the model. This is like chrome plating your model. As you rotate the view, you can see flaws because of the undulation of the image reflected in the geometry. The map "material" used in the example was polished silver. Rhino comes with a variety of material images that you can select for this purpose.



Environment map showing fairness of the surfaces.

Step-by-Step: Environment map analysis

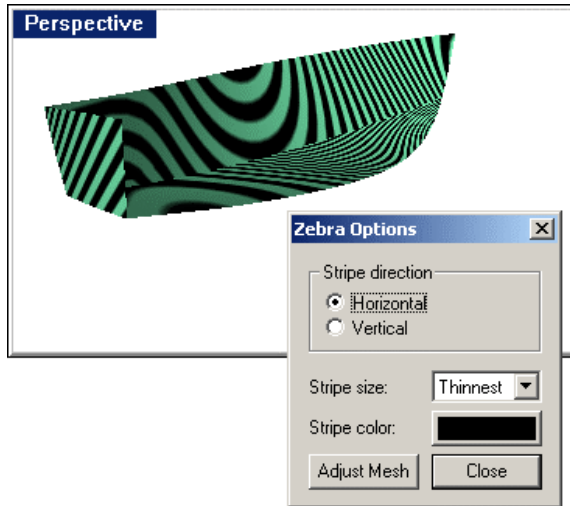
View the environment map:

- 1 Select the hull.
- 2 Start the **EMap** command.
- 3 When finished viewing, close the **Environment Map Options** dialog box.

In the Environment Map Options dialog box, you can adjust the analysis mesh, change the map image, and blend the surface render color with the image colors to produce other effects.

Zebra stripe analysis

The illustration below shows the results of the **Zebra** command. This command maps black stripes onto the surface of the hull. It is particularly helpful where you have surfaces that you have joined that are supposed to be tangent (G1) or curvature (G2) continuous. The **Zebra** command magnifies problems with these **continuities**. We do not have any such surfaces here, but the method still allows us to see otherwise indistinguishable flaws in our surfaces. Also noteworthy is that at any G0 continuity, there is a decided break between the stripes at either side of the seam. You can see this at the chine.



Model with zebra stripes applied.

Step-by-Step: Zebra stripe analysis

To view the zebra stripe map:

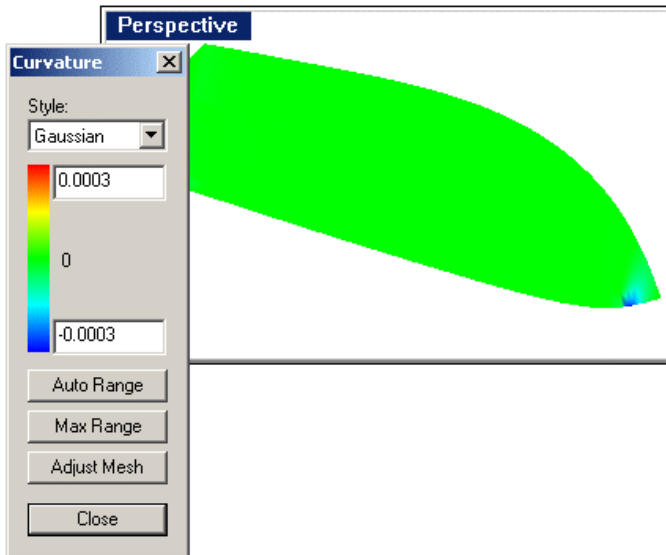
- 1 Select the hull.
- 2 Start the **Zebra** command.
- 3 When finished viewing, close the **Zebra Map Options** dialog box.

In the Zebra Map Options dialog box, you can change the size, color and orientation of the stripes, and adjust the analysis mesh density.

Surface curvature analysis (Gaussian)

The image below shows the results of the **CurvatureAnalysis** command using the **Gaussian** style. The Curvature dialog box displays the values of Min and Max that were applied to get the display shown. These values are automatically selected for you when you start the command or click the Auto Range button, but you can change them to see the effects.

Here you can see there are very small deviations from developability at the stemhead (shown in blue shading). These differences are so slight as to be safely ignored. If the values in the two edit boxes were changed to a larger number, say 0.1 (1/10 of a foot), and the displayed curvature was anything other than zero, there might be cause for concern. With the values set at 0.0005 and -0.0005 , the hull is entirely green.



Model with Gaussian surface curvature applied.

Step-by-Step: Surface curvature analysis (Gaussian)

To view the curvature analysis map:

- 1 From the **Analyze** menu, click **Surface**, and then click **Curvature Analysis**.
 - 2 At the **Select objects for Select objects for curvature analysis** prompt, select the hull and press **Enter**.
 - 3 When finished viewing, click the **Close** button on the **Curvature** dialog box.
- In the **Curvature** dialog box, you can change the limits of the analysis and adjust the analysis mesh density.

Complete the hull

When you are satisfied with the fairness of your hull, apply the finishing touches.

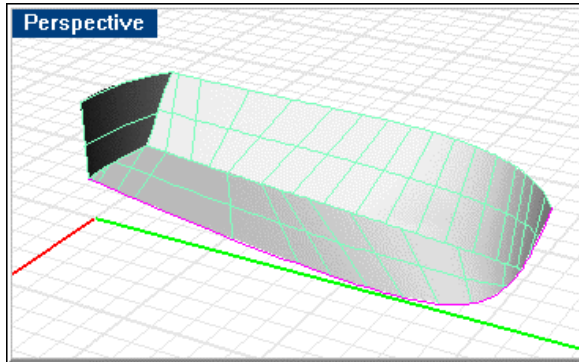
You now have one side of the hull, which stops one-half inch off centerline. This hull surface extends back to the very tip of the centerline aft in two continuous surfaces. It is joined there by a transom that extends from the side to centerline and from the bottom panel to the sheer.

Create the centerline deadflat

You need to define the deadflat on centerline, mirror the hull, and join everything to ensure that the centerline edges are "clean."

To create the deadflat that extends across centerline use the **DupEdge** command to create a curve that matches the inboard edges of the hull surfaces. Select all the edge segments from stem to stern (except the centerline edge of the transom).

After completing the command, **join** the duplicated edge curves together to form one continuous curve.



Centerline deadflat.

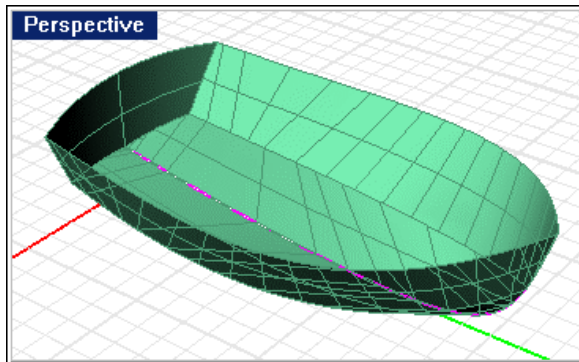
Step-by-Step: Duplicate and bottom and side inboard edges

To duplicate the bottom and side inboard edges:

- 1 Start the **DupEdge** command.
- 2 At the **Select edges to duplicate** prompt, select all the edges along the inboard sides of bottom and side, and press Enter.
The surface edges are duplicated as curves.
- 3 While the new curves are still selected, use the **Join** command to join all the curve segments into a single curve.

Mirror the hull and transom

Mirror the hull and transom polysurface and the curve just created across centerline.



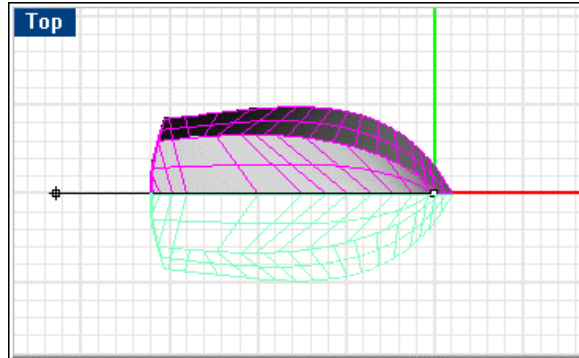
Mirrored hull and transom.

Step-by-Step: Mirror the hull and transom

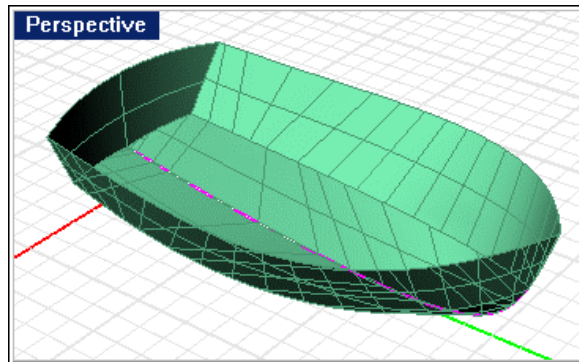
Mirror the hull and transom:

- 1 Select the hull, transom, and joined curve.
- 2 Start the **Mirror** command.
- 3 At the **Start of mirror plane ...** prompt, in the **Top** view, type **0**.

- At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line along the x-axis.



Mirror the hull, transom, and edge curve.

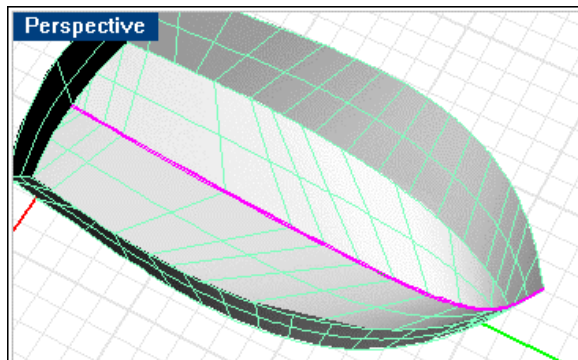


Result of mirror.

Create the deadflat surface

Use the **EdgeSrf** command to create the deadflat surface. Select the two joined curves. This command will create a 1-inch wide ribbon from stem to stern that shares edges with the two halves of the hull. If we were to complete this project, we would model the keel and insert it in place of the deadflat.

Note: The surface's edges cannot be used for this command, since these edges are each made of multiple curves.

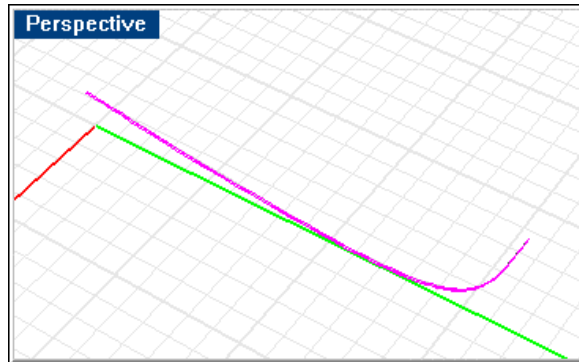


Deadflat surface.

Step-by-Step: Create the deadflat surface

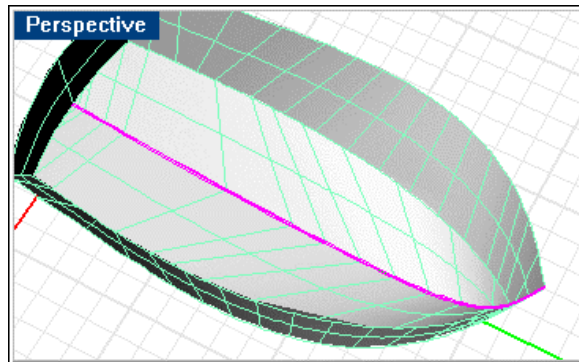
Create the deadflat surface:

- 1 **Hide** the hull polysurfaces.
- 2 Select the two curves.



Hull edge curves.

- 3 Start the **EdgeSrf** command.



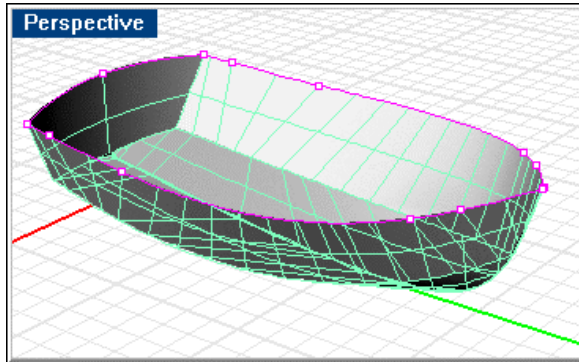
Deadflat surface.

Join the two hull halves and deadflat surface

Join the two halves and the centerline deadflat to create one continuous polysurface. This polysurface should have no **unjoined edges** except the sheer and transom.

To check for naked edges, use the **ShowEdges** command. The result should look like the illustration below.

In this illustration, the magenta curve shows the only naked edges in the model. If there are other naked edges, then work needs to be done to **correct** this.



Joined hull halves and deadflat.

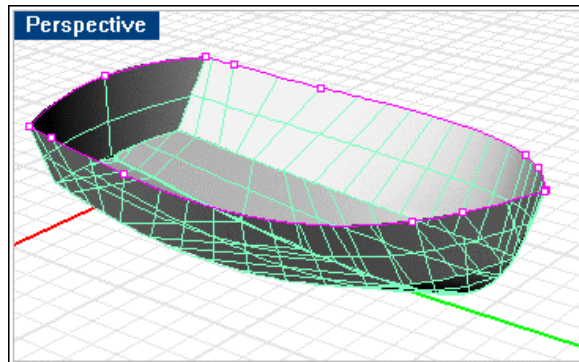
Step-by-Step: Join the two hull halves and deadflat surface

Join the hull surfaces:

- 1 Select all three hull polysurfaces including the deadflat surface.
- 2 Use the **Join** command to join the three polysurfaces into one.

View naked edges:

- 1 Select the joined hull.
- 2 Start the **ShowEdges** command.
- 3 When finished viewing the naked edges, press **Enter**.



Naked edges on after joining two halves and centerline deadflat.

Unroll the plates

When you have lofted the surfaces and joined them all together into a continuous hull with no gaps or naked edges, you can explore the **UnrollSrf** command. In this section, you will learn how to make a flat pattern from one of these surfaces, complete with marking lines.

The primary reason for using developable surfaces is that the builder will be able to place a prescribed plate against the frames and have it fit without any trimming or excessive bending. The plate should be fair, should meet the edges exactly where it is supposed to, and should require no deformation to do so. With Rhino, you can unroll the transom, side, and bottom of the skiff with reference marks (frames, waterlines, stiffeners, etc.) and have the plate cut "neat." Then the builder should be able to apply the finished plate to the hull with a minimum of effort.

Prepare the hull for unrolling

To unroll the plate, make a copy of the model from which you can extract surfaces. These copied surfaces will be what you unroll. This is done because you will need to explode the joined hull and rejoin the side and bottom panels into polysurfaces after deleting the starboard side of the boat. Copy the hull in place and change the copy to the Plates layer. Turn off all other layers except Plates.

Shortcut: To copy objects in place use the Windows copy and paste hot-keys (**Ctrl+C** and **Ctrl+V**).

Explode the hull; **delete** the starboard side, bottom, and the centerline deadflat surface. You will be left with the port side, bottom, and transom surfaces.

The transom surface extends from side to side. Since the side and bottom were lofted with the **Developable** option, they were created as polysurfaces.

Since you just exploded the entire model, these polysurfaces reverted to their constituent surfaces.

Join these into a single bottom panel and a single side panel.

Step-by-Step: Prepare the hull for unrolling

Copy the hull:

- 1 Select the hull.
- 2 Press **Ctrl+C** to copy the hull to the Clipboard.
- 3 Press **Ctrl+V** to paste the hull from the Clipboard.

Change the layer of the copy:

- 1 With the hull still selected, use the **ChangeLayer** command to change the layer of the hull to the **Plates** layer.

Turn off all layers except the Plates layer:

- 1 Start the **OneLayerOn** command.
- 2 In the **Layer to Leave On** dialog box, select the **Plates** layer.

Explode the hull and delete the unwanted surfaces:

- 1 Select the hull.
- 2 Use the **Explode** command to explode the hull polysurface into individual surfaces.
- 3 **Delete** the starboard side and bottom, and centerline deadflat surface.

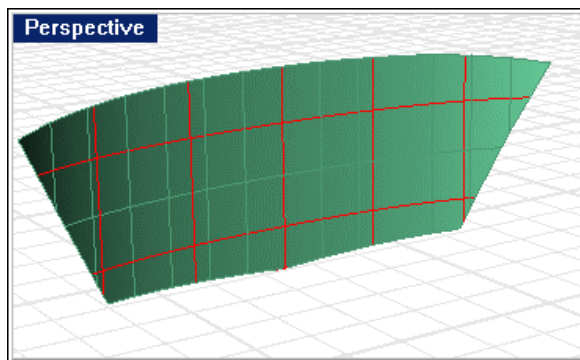
Join the bottom and side surfaces:

- 1 Select the bottom panel surfaces.
- 2 Use the **Join** command to rejoin these surfaces into one polysurface.
Rhino will report how many surfaces were joined into how many polysurfaces.
- 3 Repeat the **Join** for the side panel.

Place marking lines

Start with the transom. Select the side and bottom and **Hide** them.

Place marks on centerline and every 18 inches off centerline out to the 4.5-foot buttock to represent transom stiffeners. Place waterlines at 3 and 4.5 feet. To do this, draw these lines in the **Right** viewport and **Project** them onto the transom surface.



Marking lines on the transom.

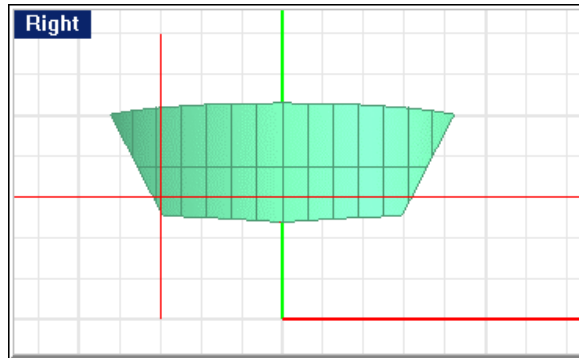
Step-by-Step: Place marking lines on the transom**Hide the unwanted surfaces:**

- 1 From the **Edit** menu, click **Visibility**, and then click **Hide**.
- 2 At the **Select objects to hide** prompt, select the side and bottom panels, and press **Enter**.

Draw and array construction lines:

- 1 From the **Curve** menu, click **Line**, and then click **Single Line**.
- 2 At the **Start of line ...** prompt, in the **Right** viewport, type **-3,0**.
- 3 At the **End of line ...** prompt, type **-3,7**.
- 4 Repeat the **Line** command.
- 5 At the **Start of line ...** prompt, type **-8,3**.

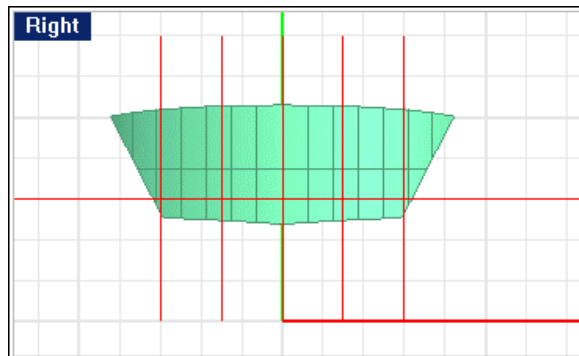
- 6 At the **End of line ...** prompt, type **8,3**.



Create first marking lines.

Array the vertical line:

- 1 Select the vertical line.
- 2 Start the **Array** command.
- 3 At the **Number in X direction** prompt, type **5**.
- 4 At the **Number in Y direction** prompt, type **1**.
- 5 At the **Number in Z direction** prompt, type **1**.
- 6 At the **X spacing** prompt, type **1.5**.

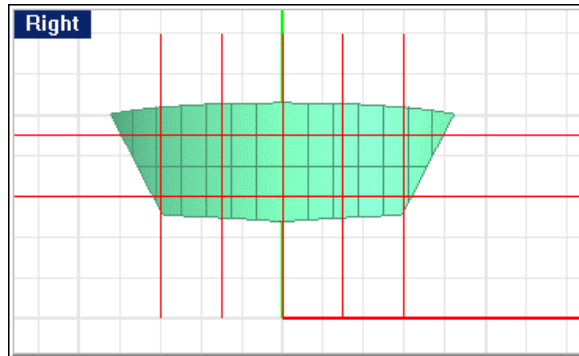


Array vertical marking line.

Copy the horizontal line:

- 1 Select the horizontal line.
- 2 Start the **Copy** command.
- 3 At the **Point to copy from ...** prompt, pick a point in the **Right** viewport (location is not important).
- 4 At the **Point to copy to** prompt, type **1.5**.

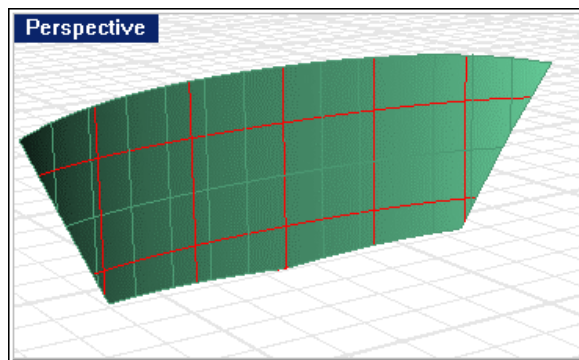
- 5 At the **Point to copy to** prompt, in the **Right** viewport use **Ortho** to pick a point above the "copy from" point.



Copy horizontal marking line.

Project the straight lines onto the transom:

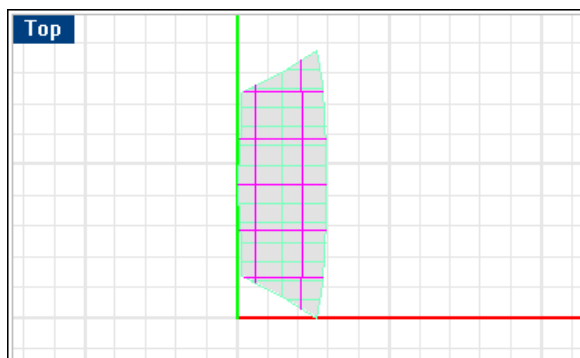
- 1 Select the horizontal and vertical lines and the transom surface.
- 2 Start the **Project** command.
- 3 **Delete** the original lines.



Marking lines on the transom.

Unroll the transom

Select the transom and use the **UnrollSrf** command. Select the two waterlines and five buttocks we previously created on the transom. Rhino will unroll the transom plate and the seven marking lines.



Unrolled transom.

Step-by-Step: Unroll the transom

Unroll the transom:

- 1 Select the transom surface and the seven marking lines.
- 2 Start the **UnrollSrf** command.

Rhino will report progress as follows:

Flattening curve 1 of 7... Press Esc to cancel

Flattening curve 2 of 7... Press Esc to cancel

...

Flattening curve 7 of 7... Press Esc to cancel

Calculating moments... Press Esc to cancel

...

Creating polygon meshes... Press Esc to cancel

The unrolled surface is also a surface and the marking lines are curves. These are highlighted in the image. This makes it easy to differentiate the isocurves from the marking lines.

If you are creating flat patterns for NC cutting and only need the edge of the part and not the surface, use the **DupBorder** command to draw the border around the part.

You can then **delete** the surface. If the surface has cutouts, you can create these and trim the surface to include them. Then, when you use the **DupBorder** command, the cutouts will be reproduced as curves.

That is how easy it is to unroll the plate. Try the other two surfaces on your own. Try sketching some longitudinals along the side and bottom surfaces and erecting a few frames and flats to make things interesting.

Part III: 68-Foot Sailing Yacht



Introduction

This tutorial will introduce you to the methods needed to create a fair hull from a set of design parameters. Unlike other tutorials on this CD, this tutorial will not present you with a lines drawing from which to work. You will create the lines as you go along and produce a finished lines drawing at the end of the tutorial.

You will see how to take a set of basic design parameters and fair a hull to meet these. Along the way, you will visit most of the techniques necessary to achieve a fair hull. You will see how easily you can generate a fair hull with a limited number of control curves. Additionally, you will use analytical techniques to determine whether the hull, which your eye says is fair, is actually fair to a microscopic degree.

This tutorial, along with the others on this CD gives you a methodology that will enable you to create any design you can imagine. Once complete, this model will be invaluable in the production of the boat.

You will learn to:

- Loft surfaces from a minimal set of curves.
- Fair lofted surfaces using curvature-mapping techniques.
- Analyze polysurfaces for geometric flaws.
- Calculate hydrostatics.
- Generate GHS geometry files.
- Create a finished lines drawing from the model.

The model created for this tutorial (Sail Finished.3dm) is included on the distribution CD. You can work from it or create your own while performing the tasks.

The vessel

The vessel you will design in this tutorial is a moderate displacement (DLR=47.7) sailing yacht similar in form to **ULDB's**. This is not a tutorial in sailing yacht design. In fact, most yacht designers would find fault with this design. The purpose of this tutorial is to instruct in the art of fairing hulls using Rhino techniques.

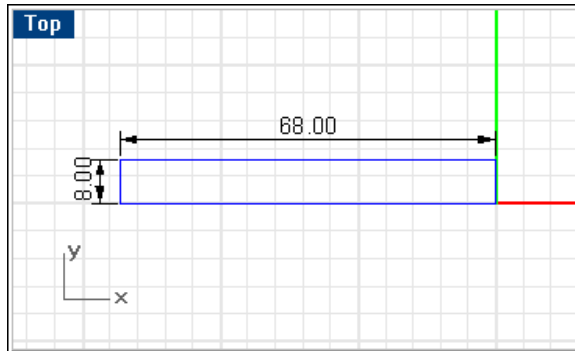
Design Parameters.

- Length, Overall 68'-0".
- Length, DLWL 64'-0".
- Beam, 16'-0".
- Displacement, 28,000 lb.

Lay out the hull

To begin designing your sailing yacht, proceed just as you have always done in yacht design: lay out the sheer in plan and profile, the rabbet line profile, and the midship station. Following these layouts, add a few stations forward and aft of midship to help fill in the shape of the hull.

Prior to sketching the two-dimensional curves, it is useful to draw bounding boxes so you can adjust the two-dimensional curves to meet the design parameters. Accordingly, on the **Bounding Boxes** layer, in the **Top** viewport, draw a 68-foot by 8-foot box on the base plane for the plan view of the sheer.



Top view bounding box.

Draw a 68-foot by 8-foot box in the **Front** viewport on the center plane for the sheer in profile.

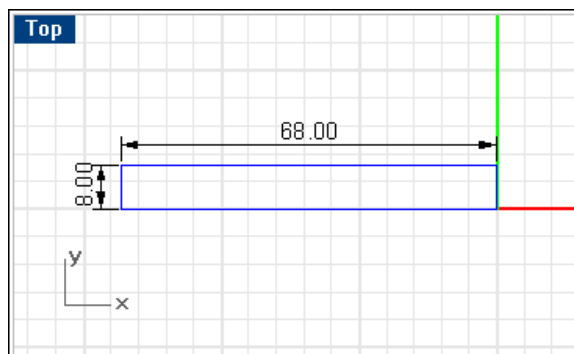
Use the **InterpCrv** command to sketch the rabbet line profile on the center plane. Since you will have a two-inch deadflat from stem to stern, immediately **move** this profile one inch off centerline. Attending to this sort of detail early in the design prevents having to go back later and make modifications to an otherwise finished model.

Step-by-Step: Lay out the hull bounding boxes

Draw bounding boxes

- 1 Make the **Bounding Boxes** layer current and turn on the **Centerline**, **Xsom Profile**, and **Sheer 2D** layers.
All other layers should be off.
- 2 Start the **Rectangle** command.
- 3 In the **Top** viewport, at the **First corner of rectangle ...** prompt, type **0**.
- 4 At the **Other corner or length** prompt type **-68,8**.

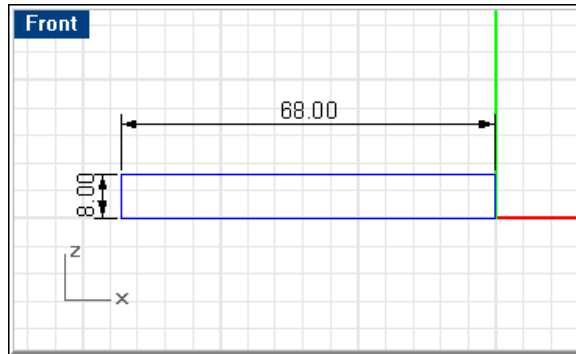
The x-coordinate for the transom is negative. Since boat designers like to have the bow at 0,0 and pointing to the right, this is necessary according to Cartesian coordinates. You need not feel bound by this convention.



Top view bounding box.

- 5 Repeat the **Rectangle** command.
- 6 In the **Front** viewport, at the **First corner of rectangle ...** prompt, type **0**.

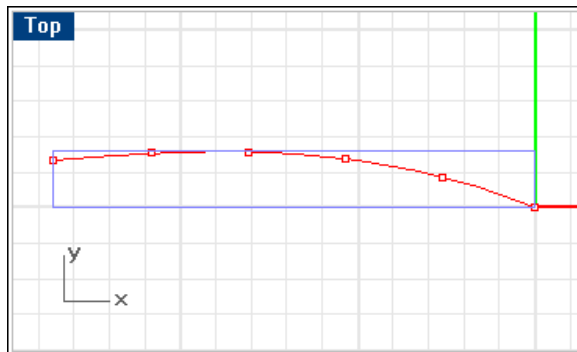
- 7 At the **Other corner or length** prompt, type **-68,8**.



Front view bounding box.

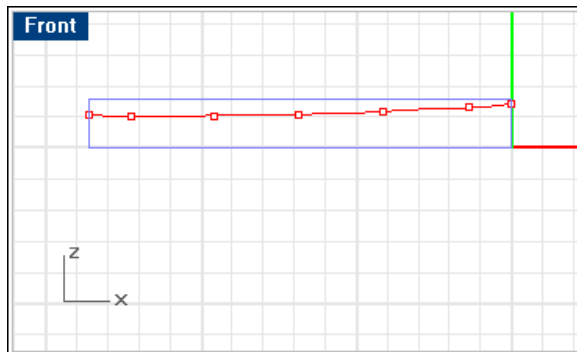
Lay out the sheer

The sheer profile is drawn in one continuous curve. In the **Top** viewport, use the **InterpCrv** command to sketch the sheer in plan. Start with the lower right corner of the plan-view bounding box; continue sketching the sheer until your last point snaps to the left edge of the bounding box. You should only need 3 or 4 points.



Sheer curve in plan.

In the **Front** viewport, use the **InterpCrv** command to sketch the sheer in profile. Start with a point along the right edge of the bounding box. Continue sketching the sheer in profile until your last point snaps to the left end of the bounding box.



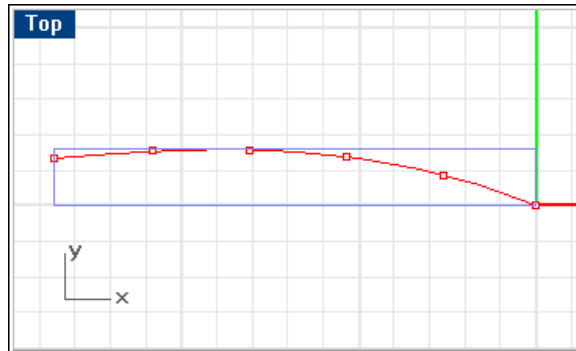
Sheer curve in profile.

Step-by-Step: Lay out the sheer

Draw sheer curves in plan and profile

- 1 Make the **Sheer 2D** layer current.
- 2 Start the **InterpCrv** command.
- 3 At the **Start of curve ...** prompt, in the **Top** viewport, use the **End** object snap to pick the lower right corner of the plan-view bounding box.
- 4 Continue sketching the sheer in plan until, using the **Near** object snap, your last point snaps to the left edge of the bounding box.

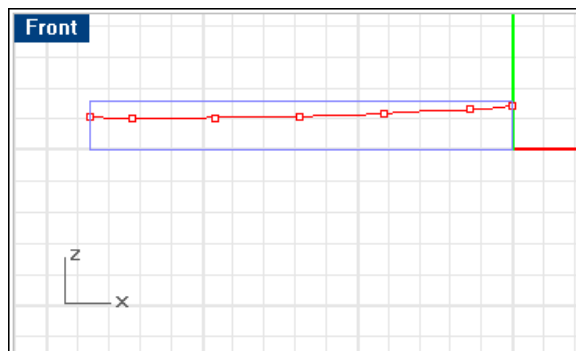
Because of the simplicity of this curve, you should only need a few points.



Sheer curve in plan.

- 5 Repeat the **InterpCrv** command.
- 6 At the **Start of curve ...** prompt, in the **Front** viewport, use the **Near** object snap to pick a point along the right edge of the bounding box.
- 7 Continue sketching the sheer in profile until, using the **Near** object snap, your last point snaps to the left end of the bounding box.

Because of the relative simplicity of this curve, you should only need a few points.

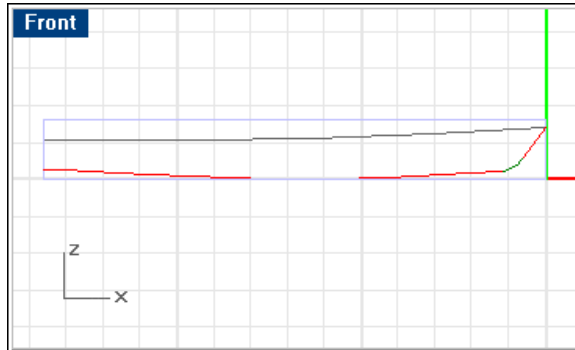


Sheer curve in profile.

Lay out the rabbet profile

Lay out the rabbet line profile using three curves: the stem, forefoot, and rabbet line from forefoot to transom. This ensures that the forefoot is continuous (G1) with the stem and rabbet profile aft. To accomplish this, draw the stem at the desired slope and draw and fair the rabbet using the Rhino curve fairing techniques. Then use the **Blend** command to create the forefoot.

If you have a preferred radius, you can use the **Fillet** command to achieve the forefoot blending.

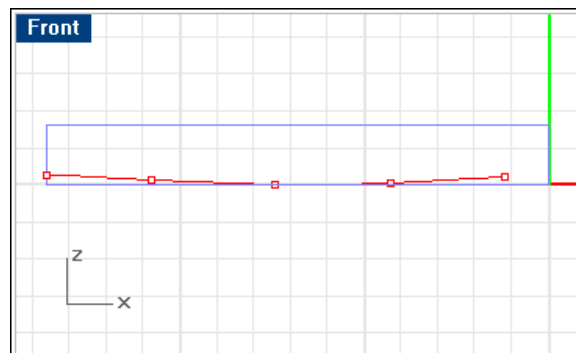


Rabbit curves in profile.

Step-by-Step: Lay out the rabbet profile

Lay out the rabbet in profile

- 1 Make the **Rabbet** layer current.
- 2 Start the **InterpCrv** command.
- 3 At the **Start of curve ...** prompt, in the **Front** viewport, use the **Near** object snap to pick the left edge of the profile-view bounding box.
- 4 Disable object snaps, and continue sketching the rabbet in profile.
Make your last point to the left of the right edge of the bounding box as shown.



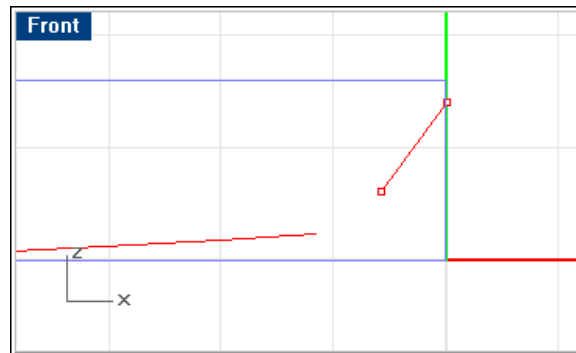
Rabbet curve in profile.

The low point of this curve should be tangent with the bottom of the bounding box at about midship. Because of the simplicity of this curve, you should only need 3 or 4 points.

Lay out the stem

- 1 Start the **Line** command.
- 2 At the **Start of line ...** prompt, use the **End** object snap to pick a point at the upper right corner of the profile-view bounding box.

- At the **End of line ...** prompt, click a point above and to the right of the forward end of the rabbet line you just sketched, approximately as shown.

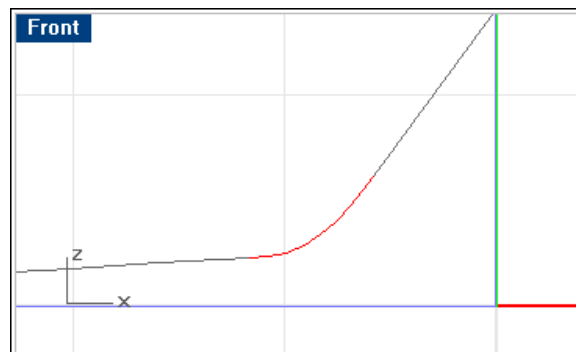


Stem line in profile.

Blend the stem and rabbet curves

- Start the **Blend** command.
- At the **Select first curve ...** prompt, select the stem you just created.
- At the **Select second curve ...** prompt, select the rabbet you created previously.

Rhino will create a curve that is curvature continuous (G2) between the stem and rabbet.



Blend curve between stem line and rabbet.

Move the curves 1 inch off the center plane

- Select the stem, forefoot, and rabbet curves.
- Start the **Move** command.
- At the **Point to move from ...** prompt, in the **Right** viewport, pick a point.
- At the **Point to move to** prompt, type **1"**, press Enter, and use **Ortho** to pick a point to the right.

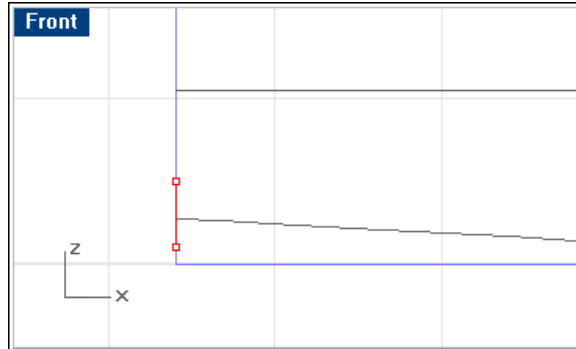
Lay out the transom profile

For the transom, you will draw a plumb line from below the rabbet to a point part way up the transom and then a forward-sloping line to a point above the sheer in profile. This is done because in Rhino most surfaces should be defined larger than the finished size and trimmed.

Step-by-Step: Lay out the transom profile

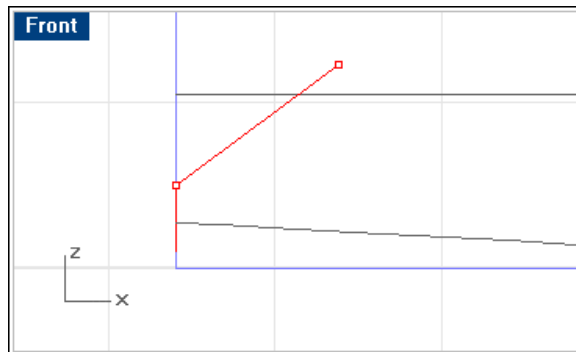
Lay out the transom profile

- 1 Make the **Xsom Profile** layer current.
- 2 Start the **Lines** command.
- 3 At the **Start of first line** prompt, in the **Front** viewport, use the **End** object snap to pick the lower left corner of the profile-view bounding box.
- 4 At the **End of line ...** prompt, use **Ortho** to pick a point as shown directly above the point you just picked and above the rabbet curve.



Lower transom line.

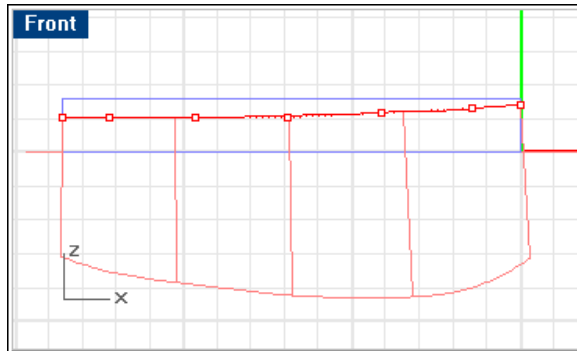
- 5 At the next **End of line...** prompt, pick a point as shown above and to the right of the point you just picked. This point should be above the sheer profile.



Upper transom line.

Fair the two-dimensional curves

Now that you have sketched the sheer in plan and profile, check them for fairness before proceeding. Select the profile sheer 2-D curve and use the **CurvatureGraph** command. The result should look similar to the image below.



Fair sheer in profile.

This graph indicates a very fair curve. This is so primarily because you constructed the curve with only a few points. Since you are using so few points, it would be quite hard to have anything but a fair curve. No fairing work is required.

If it were necessary to rework this curve, you would move the curve's control points to modify the shape until the curvature graph was similar to the image. You will see more on this valuable fairing tool when you begin fairing surfaces later on.

Step-by-Step: Fair the two-dimensional curves

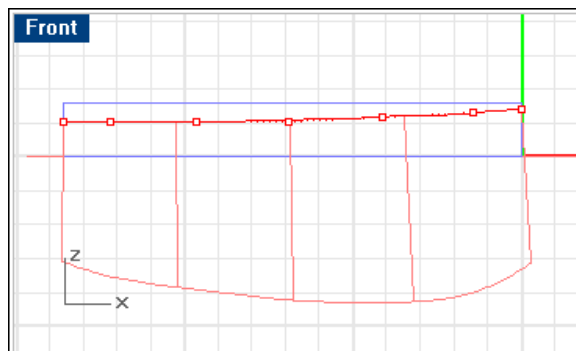
To display the curvature graph

- 1 Select the sheer curve.
- 2 Use the **CurvatureGraph** command to view the curvature graph.

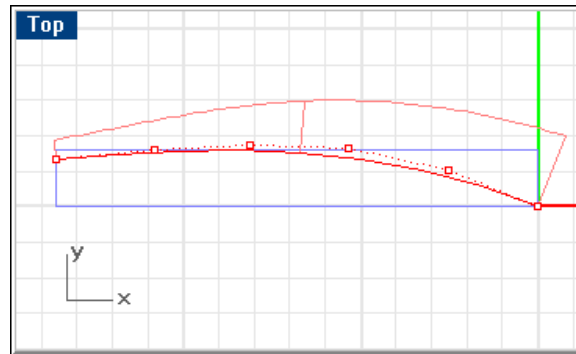
If the curve requires fairing

- 1 Select the sheer.
- 2 Use the **PointsOn** command to turn on the curve's control points.
- 3 In the **Front** and **Top** viewports, **rebuild** the curve and **nudge** control points until the curvature graph resembles the images below.

You may have to adjust the curvature graph scale.



Fair sheer in profile.

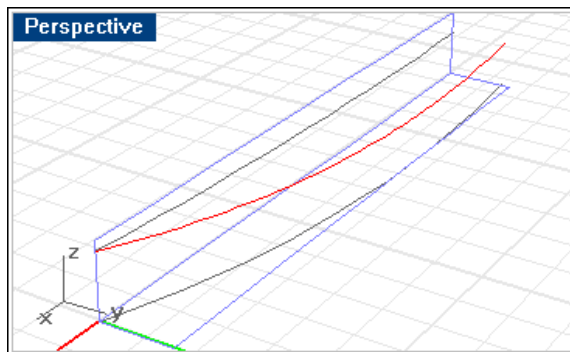


Fair shear in plan.

Create the three-dimensional shear

Rhino provides you with the functionality to create a three-dimensional curve from two planar two-dimensional curves. This lets you create the 3-D shear using curves from lines drawings provided by other software without laboriously typing the coordinates of each curve.

In the image below, you see the results of using the **Crv2Views** command. The two gray two-dimensional curves representing the shear in plan and profile were selected and Rhino created the red curve. This curve, together with the rabbet, forefoot, and stem, will be used to construct stations through the model that will be lofted to create the hull surface.



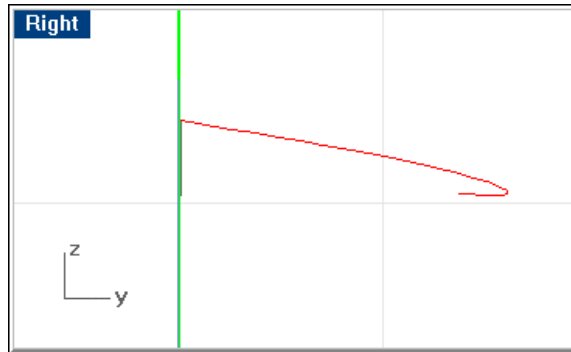
The three-dimensional shear.

Step-by Step: Create the three-dimensional shear

To create the three dimensional curve

- 1 Turn on the **Sheer 3D** layer and make it current.
- 2 Turn off the **Bounding Boxes** layer.
- 3 Select the shear plan and profile curves.

- 4 Use the **Crv2View** command to create the 3-D shear.



The finished shear in the right viewport.

If this curve looks the way you intended, you can proceed with construction of stations. If it requires additional work, you can simply erase it, adjust either the plan or profile of the shear, and run the **Crv2Views** command again.

- 5 **Save** your work.

Construct preliminary stations

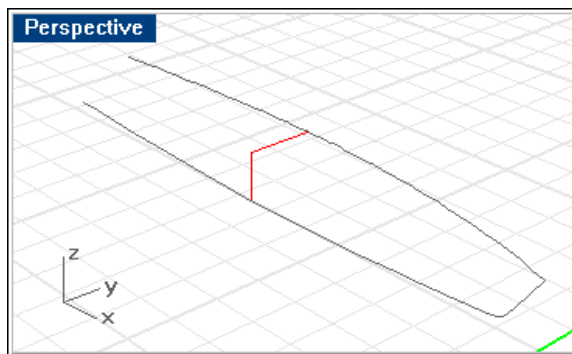
Construct several stations to let you loft the hull surface. Place these on the **Prelim Stations** layer.

To begin each station, construct a pair of guide **lines** between the shear and rabbet.

At the desired location, draw a straight horizontal **line** from the shear to a point beyond centerline. **Trim** this horizontal line so that its inboard end is flush with the rabbet in plan view.

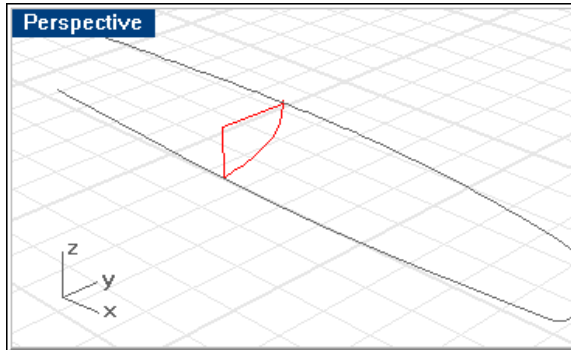
Draw a vertical **line** from the inboard end of this line to a point below the rabbet. **Trim** this line to the rabbet in profile.

You now have the two endpoints of your preliminary station. You can now begin sketching the preliminary station.



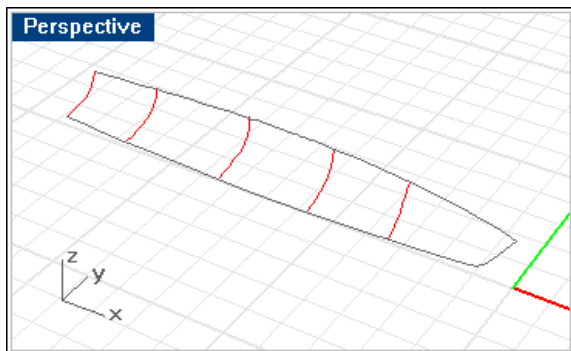
Draw straight guidelines between shear and rabbet to start preliminary station.

Turn on **Planar** mode and disable **Ortho**. Use the **InterpCrv** command to sketch the station using the two endpoints of the guidelines as endpoints of the station.



Preliminary station curve.

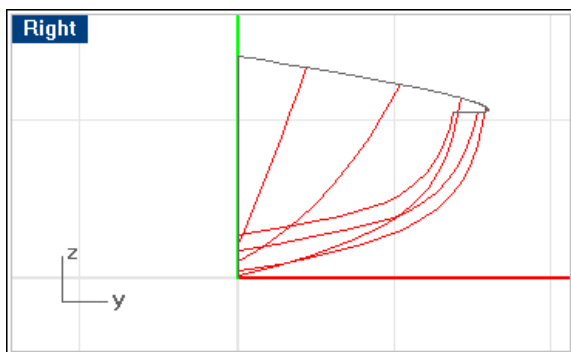
Repeat this procedure to create more stations as shown below.



Preliminary stations.

You will fair the hull surface to the very aft end of the boat and then trim in the actual transom. The locations of the stations in this exercise are arbitrary. If you require the stations at exact locations, take the necessary steps to locate them more accurately; however, Rhino does not require them at a specific location.

In the image below you can see these same stations head-on. This is where you do most of the fairing of these preliminary stations. Only a few stations will be necessary. You could probably have gotten by with even fewer if it were not for the tight forefoot.



Body plan preliminary stations.

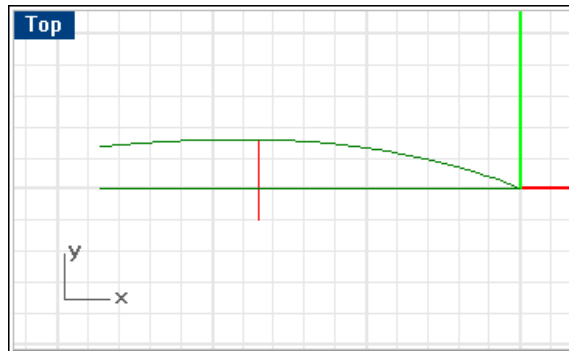
Step-by-Step: Construct preliminary stations

Set up viewports

- 1 Make the **Stations** layer current.
- 2 Turn off the **Sheer 2D** layer.
You should now have only the rabbet and sheer visible in all four viewports.
- 3 Use the **Zoom Extents All** command to bring everything into view.

Draw a horizontal guideline

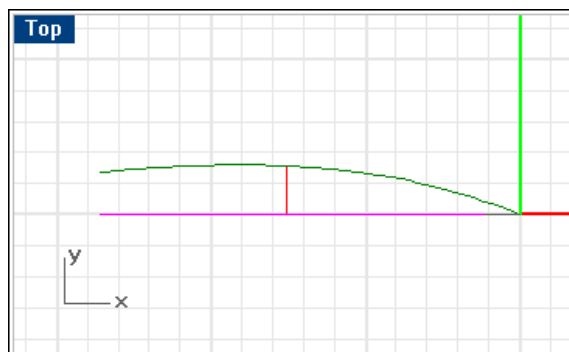
- 1 Enable **Planar** and **Ortho**.
- 2 Start the **Line** command.
- 3 At the **Start of line** prompt, in the **Top** viewport, use the **Near** object snap to pick the sheer approximately at midship.
- 4 At the **End of line** prompt, click at a point below the centerline.



Horizontal guideline.

Trim the horizontal guideline with the rabbet curve

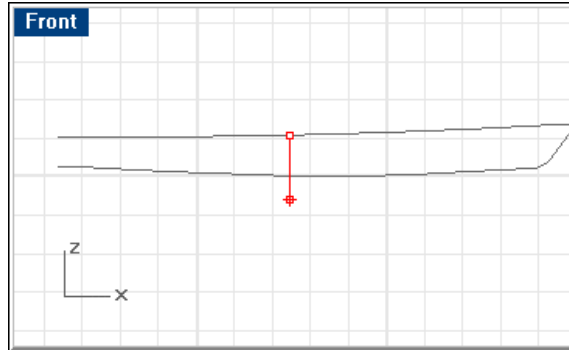
- 1 Select the rabbet curve.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim** prompt, in the **Top** viewport, select the horizontal guideline you just created.



Trimmed horizontal guideline.

Draw a vertical guideline

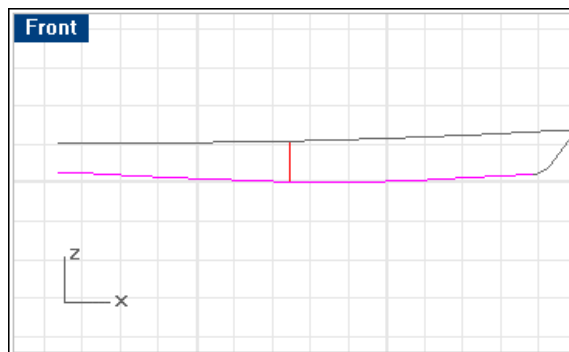
- 4 Start the **Line** command.
- 5 At the **Start of line** prompt, in the **Top** viewport, use the **End** object snap to click the inboard end of the horizontal guideline you just created.
- 6 At the **End of line** prompt, in the **Front** viewport click a point below the rabbet.



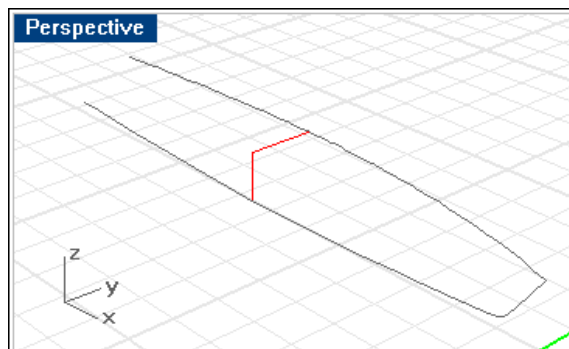
Vertical guideline.

Trim the vertical guideline with the rabbet curve

- 1 Select the rabbet curve.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim** prompt, in the **Front** viewport, select the lower end of the vertical guideline.



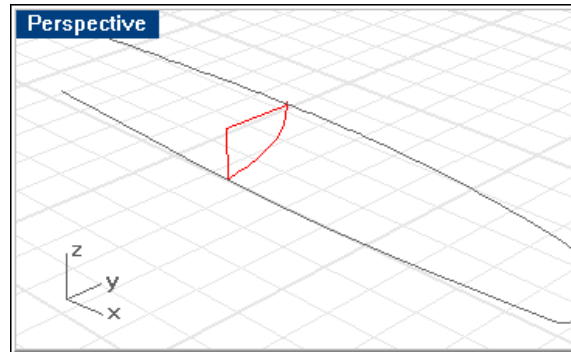
Trimmed vertical guideline.



Preliminary station guidelines.

Create the midship station

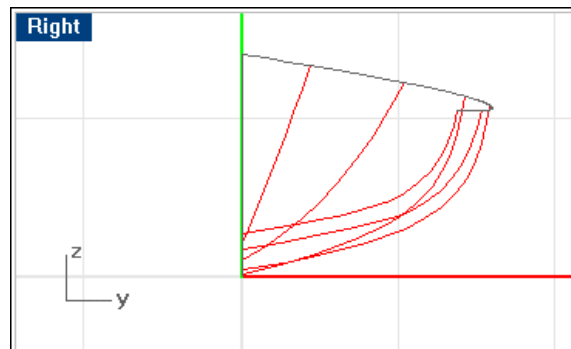
- 1 Start the **InterpCrv** command.
- 2 At the **Start of curve ...** prompt, use the **End** object snap to pick the outboard end of the horizontal guideline at the sheer in the **Right** viewport.
- 3 Continue clicking points to construct your station, snapping to the lower end of the vertical guideline as the last point.



Preliminary station.

Edit the midship station line

- 1 Select the new midship station you just constructed.
- 2 Use the **PointsOn** command to turn control points on.
- 3 Edit control points until your preliminary midship station resembles the one displayed below.

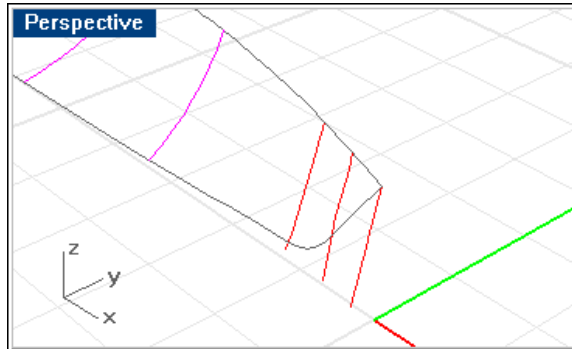


Body plan showing preliminary stations.

- 4 Repeat this process for three curves aft of midship (including one at the transom) and two forward of midship.

Create the bow stations

The first station back from the stemhead connects the sheer to the stem and projects into negative space. The same technique was used to create the hull in the **Skiff** tutorial. The station at the stemhead is a copy of this curve. This station, too, extends entirely into negative space.

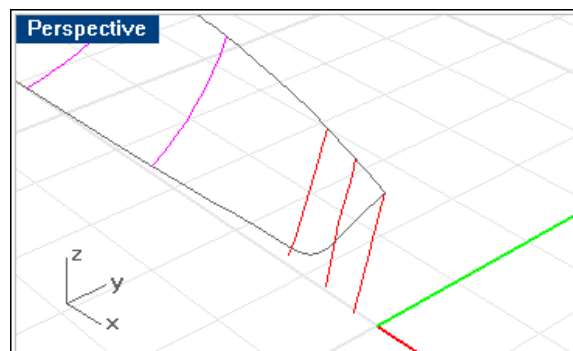


Bow stations.

Step-by-Step: Create the bow stations

Create the bow stations

- 1 Create an additional station at the forward end of the forefoot.
This station should extend into negative space. Do not trim this curve.
- 2 Select the last station created.
- 3 Start the **Copy** command.
- 4 At the **Point to copy from ...** prompt, use the **End** object snap to pick the upper end of this station.
- 5 At the **Point to copy to** prompt, use the **End** object snap to pick the end of the sheer at the stemhead.

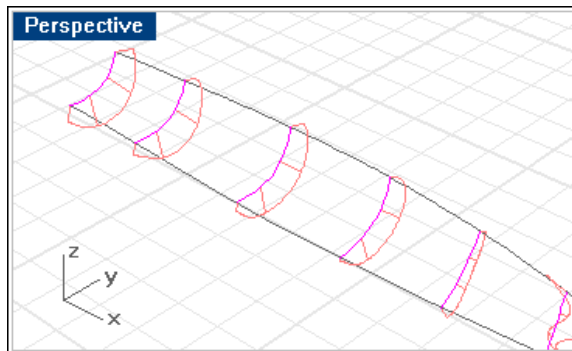


Bow stations.

Loft the hull surface

Once you've drawn the stations, **rebuild** them to be certain they all have the same number of points and are the same degree and that the degree is one less than the number of points. For curves of this nature, try to use degree 5 with 6 points. This should be sufficient for the complexity of these curves.

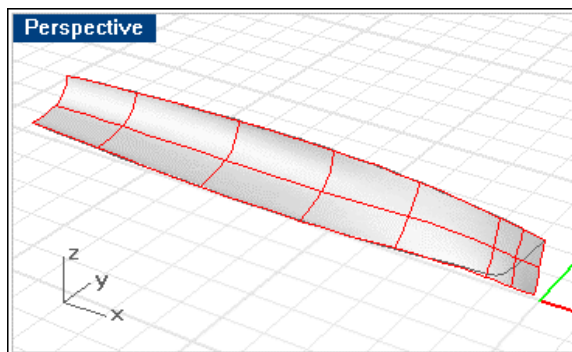
Now you can begin to add your own personal design. Push and pull the control points until the stations have the character you are looking for and follow a smooth progression one to the next from stem to stern. These curves will be all you use when you build the surface, so get them to your liking.



Faired preliminary stations.

The stations copied to the bow will give you the forward extremity of the hull and will give Rhino something to use in fairing the actual hull surface. Just like all the other curves, it should fit into the family of curves you have created for your preliminary stations. You will probably want to push and pull a little to get it the way you want it.

Now that you have created a set of stations, you can create the hull surface in a single operation. Use the **Loft** command with the **Normal** option to create a surface through the stations.



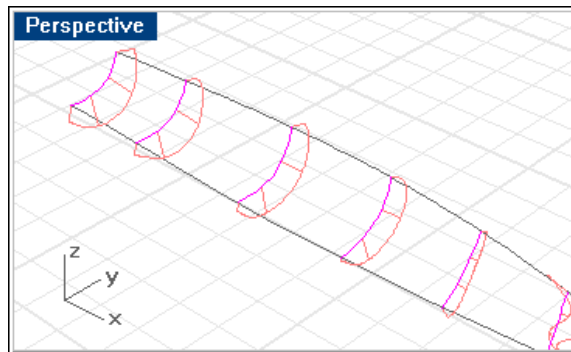
Lofted hull surface.

Step-by-Step: Loft the hull surface

Rebuild the station curves

- 1 Select all the stations.
- 2 Start the **Rebuild** command.
- 3 In the **Rebuild Curve** dialog box, set the **Point count** to **6** and the **Degree** to **5**.

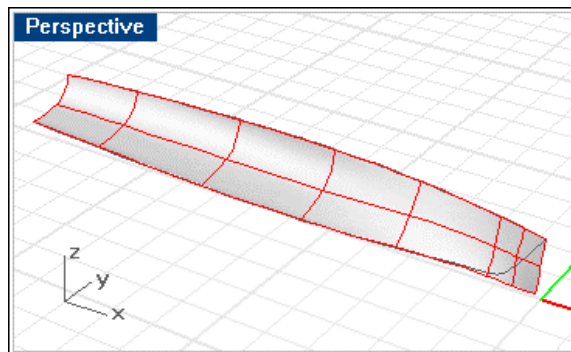
- 4 Use the curve fairing techniques to ensure the stations created are fair.



Faired preliminary stations.

Loft the hull

- 1 Make the **Hull and Deck** layer current.
- 2 Turn off the **Sheer 3D** and **Rabbit** layers.
- 3 Select all stations
- 4 Start the **Loft** command.
- 5 In the **Loft Options** dialog box, under **Style**, select **Normal**, and click **OK**.



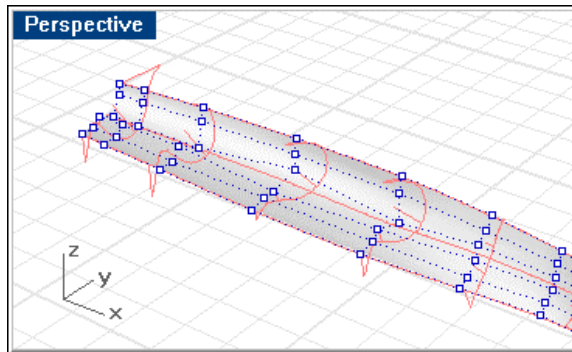
Lofted hull surface.

The simplicity of this surface is going to make it extremely easy to fair this hull. Each time you move a point, the isocurve corresponding to the nearest station will change. Since you have the preliminary stations “behind” the isocurves, you can see how much you are deviating from your preliminary stations.

Adjust the surface

At this point, you should look at how fair your surfaces are, in basic terms. Doing so means looking at a magnification of the curvature of each isocurve and analyzing how it looks relative to its neighbors. If the isocurves do what you expect and progress from one station to the next in a gradual transition, then you can assume, for the moment, that the surface is reasonably fair. You can then proceed and look further into the fairness of the surface later.

Select the surface and use the **PointsOn** command to turn on the control points. Use the **CurvatureGraph** command to display isocurve curvature graphs. The character of the curvature graphs will vary, depending on your preliminary stations.



Surface curvature graph and control points turned on.

If your curvature graphs are very large or too close to the surface, you can adjust these to more usable proportions by setting the **Display scale**.

The very pronounced hollow in the graphs of the aft four stations means there is a slight hollow at each of these stations near centerline. You would probably never see this by looking at the stations on the screen, but it is quite pronounced in the curvature graph display and the hollow would probably appear on the finished surface.

You will adjust control points in the neighborhood until this hollow disappears. For this operation, use the **MoveUVN** command. In the MoveUVN dialog box, use the sliders to adjust the U, V, or N (normal) position of the control point.

Moving the U slider will move the point in the fore and aft direction. Moving the V slider will move the point along the girth of the station in our example. Moving the N slider will move the point inboard and outboard, normal to the surface.

The **MoveUVN** command simplifies point editing by allowing you to select points in any view and move them in any direction relative to the surface without regard to which viewport is active. Trying to accomplish this by nudging can be challenging.

Step-by-Step: Adjust the surface

Display the curvature graph and control points

- 1 Select the lofted surface.
- 2 Use the **CurvatureGraph** command to display the surface curvature graph.
- 3 Use the **PointsOn** command to display the surface control points.

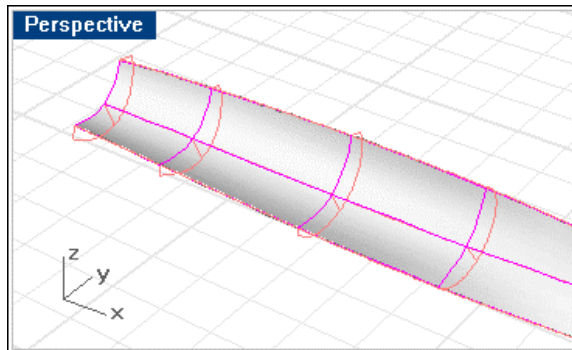
Edit the surface control points

- 1 Select the point to edit.
- 2 Start the **MoveUVN** command.
- 3 Move the sliders according to how you wish to adjust the location of the selected point.
- 4 Once you are satisfied with the new location, select other points to edit.
You can edit control points this way as long as the **Move UVN** dialog box is displayed.
- 5 Click **Apply** to set the changes, and **Close** when you are finished editing.

When moving the control points, remember that the curvature graphs are greatly exaggerated. Moving a control point a small amount will make a big difference in the graph. The idea is to get the graphs as smooth as possible and make each graph "agree" with its neighbors. A little practice will make a big difference in your ability to spot difficulties in the hull surface.

In the illustration below, the hollow aft has been removed.

You should spend some time becoming proficient with this tool, since it will allow you to adjust your surfaces quickly and early in the project when it is easiest to correct flaws.

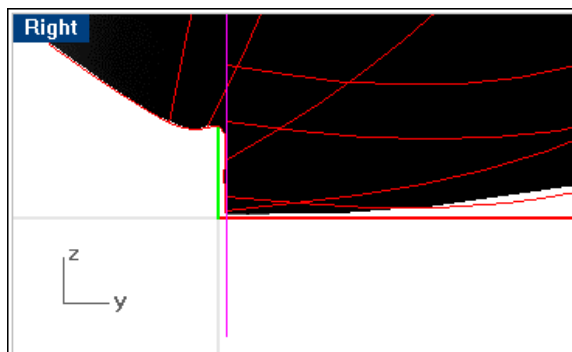


Hollow in aft curvature graphs repaired.

Modify the forefoot

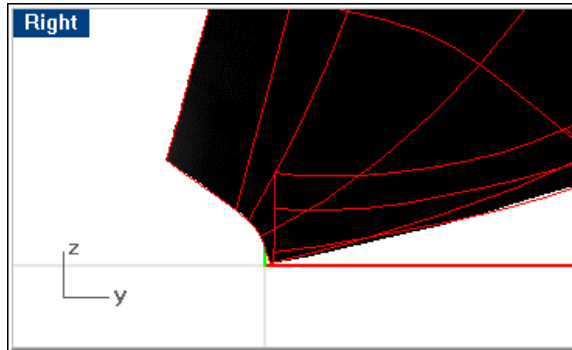
Since you took some liberty earlier in causing the hull to cross the centerline into negative space, you should inspect the results to see what damage was caused by doing this.

The inboard edge of the lofted surface may cross the rabbet line in its effort to satisfy our desires. To fix this, edit control points in this area to cause the entire surface to cross the rabbet line from the stemhead back to the point where it (the surface edge) finally meets the rabbet line tangentially.



Inboard edge possible problem.

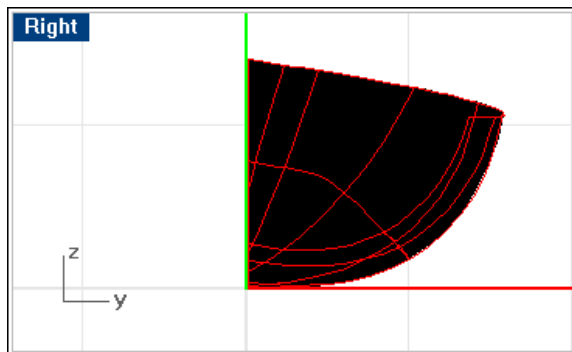
To accomplish this, turn on control points and curvature graph in the same manner used when correcting the hollow in the aft stations.



Problem solved.

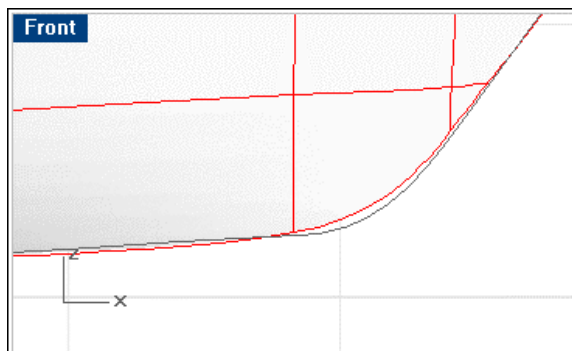
Try trimming the surface to the rabbet, causing the inboard edge of the hull surface to resemble your sketched forefoot and stem. The question is, since you did not use the stem and forefoot in your controlling geometry, how close will the resemblance be?

Once you are happy with the resulting surface, draw a vertical line 1 inch from the centerline in the body plan (**Right** viewport) and trim the surface back to that line.



Trimmed hull surface.

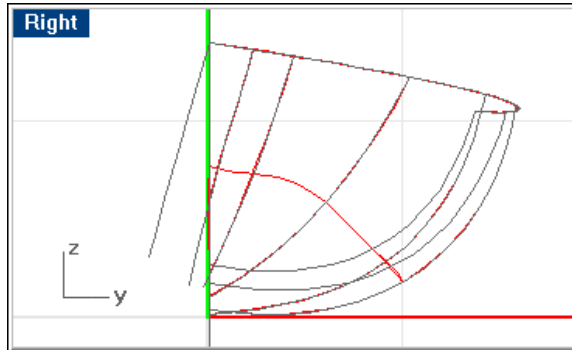
If the trimmed surface does not exactly match the forefoot and stem drawn at the beginning of this exercise, move the control points nearest the offending portion of the forefoot and stem. With a little practice, you will go right to the appropriate point(s) and be able to massage the surface in minutes.



Check hull surface against profile curve.

Note: Since you will be moving your control points in only one plane you can use the **nudge** method of point editing.

How closely does your lofted surface fit the stations body plan? Visually check your surface isocurves against the station lines.



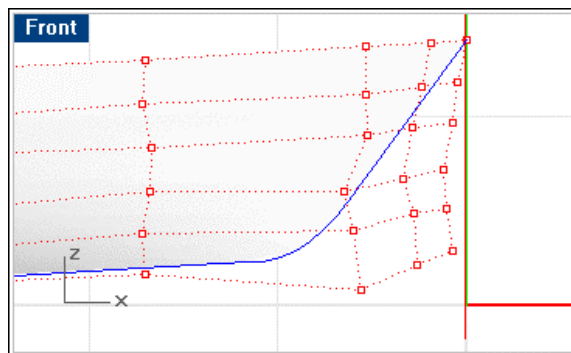
Lofted surface isocurves and stations.

The original intent of the stations is closely matched. If necessary, you can continue to manipulate the stations to get exactly what you want. Do not forget, though, Rhino is helping you find imperfections in your surface. If it means minor modifications to the stations to get rid of these flaws, it is a small tradeoff.

Step-By-Step: Modify the forefoot

Edit control points using nudging

- 1 Select a control point to edit.
- 2 In the **Front** viewport, press the **Alt** key with the **Up**, **Down**, **Left**, and **Right** arrows as appropriate until the control point is in the desired location, and the surface edge more closely matches the rabbit line.



Forefoot and stem shape repaired.

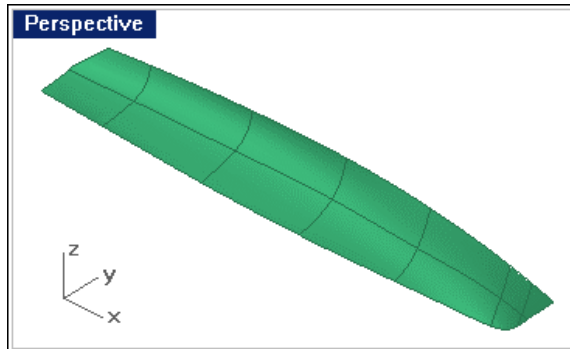
Move control points at the forefoot to make the stem and the rabbit closely resemble what you originally sketched.

- 3 Press **Esc** to turn off the control points when they are no longer needed.

Analyze the surface

Having finished creating the hull surface, you can now look at the surface's fairness in detail. For this process, use the **EMap** and **CurvatureAnalysis** commands. With these commands, small problems with the hull that will not ordinarily show up when shading or rendering become more visible.

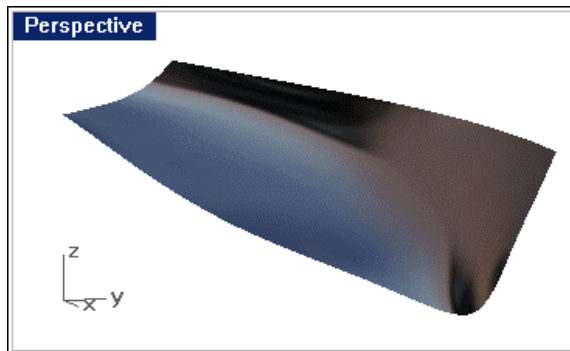
In the image below, you can get a pleasing look at the hull. If you want to see just the surfaces, simply shade the view. It can be advantageous to leave the shading and wireframe displayed all the time since you can tell where the nearest isocurve to a problem lies.



Model with wireframe shown and viewport shaded. No unfairness evident.

Environment map

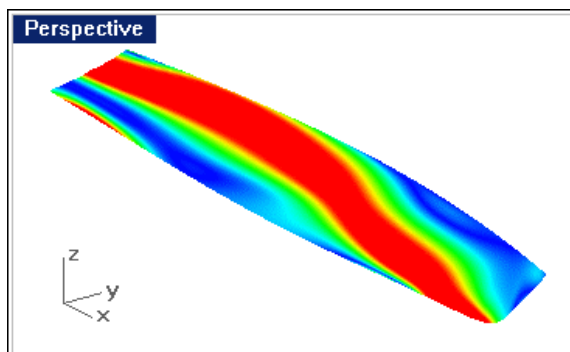
The shaded model does not always tell you about subtle problems in the surface. Flaws can be even more obvious when the model is rotated with the **EMap** displayed.



Environment map display.

Curvature analysis

Although the **EMap** display does a good job of showing the hollow in the aft sections, you still have a little of it left. You will use an even more powerful curvature analysis tool to display it.



Model with surface curvature applied.

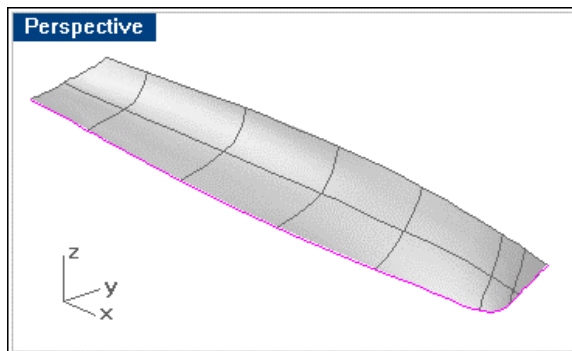
By alternatively displaying the curvature analysis and editing control points, you can remove any surface imperfections. As you do, it is a good idea to keep an eye on the **Right** viewport to be certain you are not introducing any unwanted characteristics to your body plan.

Inspect your model carefully. Any place where the curvature jumps quickly from one color to another or has "spots" of a particular color needs some attention.

Mirror and join the hull surface

Now that you have seen how easy the hull surface was to build and fair, try something even easier. You have one side of the hull that extends continuously from stem to stern and from sheer to rabbet. You need to cut in a transom and develop the deadflat on centerline. Then, to calculate hydrostatics and make the model suitable for export to third-party software, you need to close it up into a solid.

To create the deadflat that extends across centerline, use the **DupEdge** command to create a curve that matches the inboard edge of your hull surface (the rabbet). If there is more than one edge on the surface, duplicate all the edges from stem to stern. After completing the command, join the edges together to form one continuous curve.



Edge of surface duplicated.

Step-By-Step: Mirror and join the hull surface

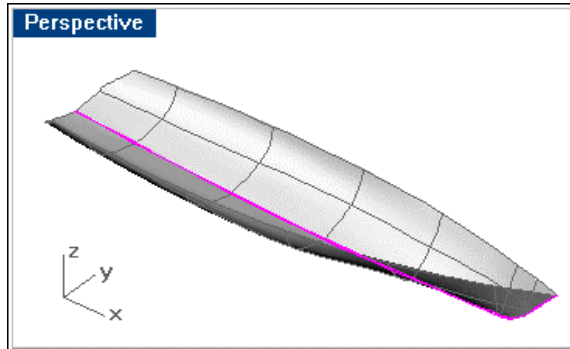
To duplicate the edge curves

- 1 Make the **Temp** layer current.
- 2 Start the **DupEdge** command.
- 3 Select all the edges along the rabbet.
Rhino will create curves that match the edges.
- 4 With these still selected, use the **Join** command to join all the duplicated edges into one curve.

If this does not happen, investigate why and correct the problem. Most likely, you missed picking one or more surface edges.

Create the deadflat

Mirror the surface and the rabbet edge curve you just created across centerline and use the **EdgeSrf** command to create a flat surface between the two rabbet curves. This will create a 2" wide ribbon from stem to stern that shares edges with the two halves of the hull.



Surface and surface edge curve mirrored.

Step-By-Step: Create the deadflat

Mirror the surface and duplicated curves

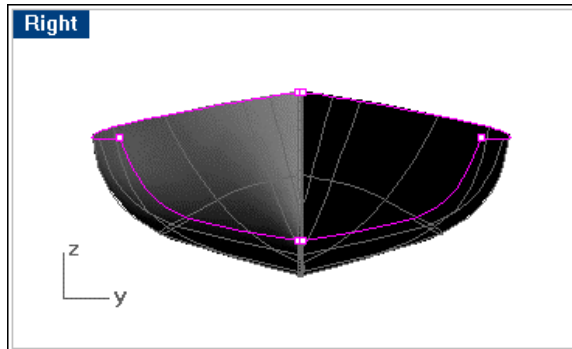
- 1 Select the joined rabbet curve and the hull surface.
- 2 Start the **Mirror** command.
- 3 In the **Right** viewport, at the **Start of mirror plane ...** prompt, type **0**.
- 4 At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line up along the y-axis.

Create the deadflat surface

- 1 Make the **Hull and Deck** layer current.
- 2 Select the port and starboard rabbet curves.
- 3 Use the **EdgeSrf** command to create the deadflat surface.
- 4 Delete the port and starboard rabbet curves.

Join the sides and deadflat

Join the two sides and the centerline deadflat to create one continuous polysurface. This polysurface should have no unjoined edges (except the sheer and top of the transom). To find out if this is true, use the **ShowEdges** command.



Joined hull halves and centerline deadflat with unjoined edges displayed.

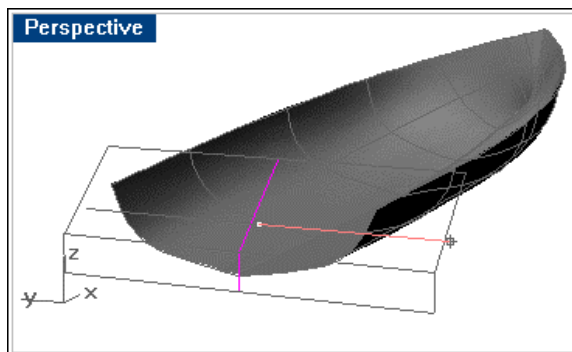
Step-By-Step: Join the sides and deadflat

Join the hull surfaces

- 1 Select the two sides and the centerline deadflat.
- 2 Use the **Join** command to join all the surfaces into one polysurface.
- 3 Use the **ShowEdges** command to display any unjoined edges in the hull.

Add the transom

You will construct the transom with two rectangular planar surfaces made from the two profile curves, and then trim the resulting surfaces to the hull and the hull to the resulting surfaces.



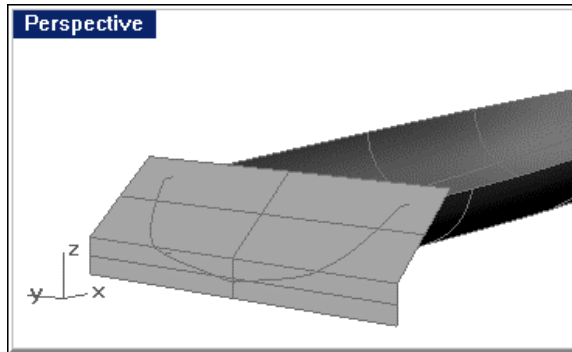
Extrude transom profile curves.

The selected curves are extruded to create the two planar surfaces that will become the transom. Use the **Extrude** command and specify the **BothSides** option to get continuous planar surfaces from outboard of the port side to outboard of the starboard sides as shown.

Before extruding, be certain that the curves touch at their endpoints and that the upper curve extends above the sheer and that the lower curve extends below the hull. If they do not, the resulting surfaces will not be good candidates for cutting objects for the **trim** command. Once the curves are extruded, the result is a polysurface.

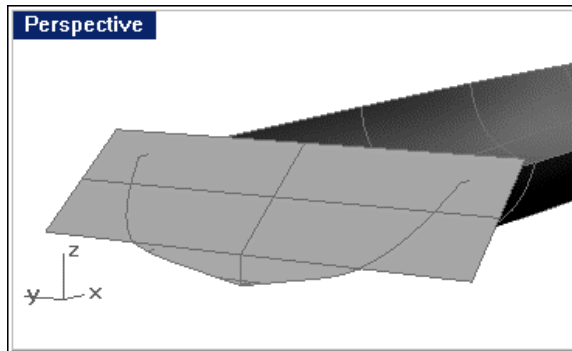
Explode it into its constituent surfaces, since the upper panel will be trimmed to a curve and polysurfaces cannot be trimmed to curves.

Using the resulting surfaces, **trim** the hull polysurface. This will give the hull its final shape.



Hull trimmed to transom surface.

Repeat the **Trim** command to trim the vertical portion of the transom outside the hull.



Trimmed vertical transom.

Step-by-Step: Build the transom

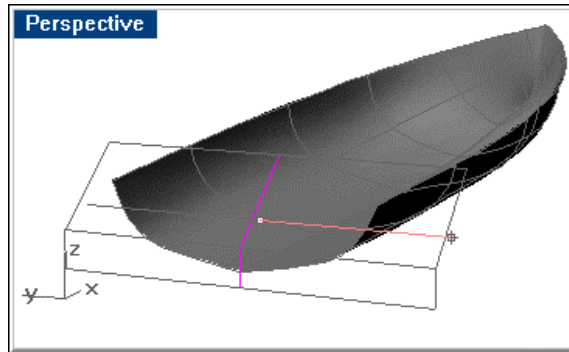
Extrude the transom profile

- 1 Turn on the **Xsom Centerline** layer.
- 2 Select the two transom curves created at the beginning of this tutorial.
- 3 Start the **Extrude** command.
- 4 At the **Extrusion distance (Direction BothSides Tapered)** prompt, type **D**.
- 5 At the **Extrusion distance ...** prompt, in the **Top** viewport, use **Ortho** to select two points to indicate the direction—one point near centerline and the second directly outboard of the first.

This is an indication of direction, so the distance between these two points is not important.

- 6 At the **Extrusion distance (Direction BothSides Tapered)** prompt, type **B**.

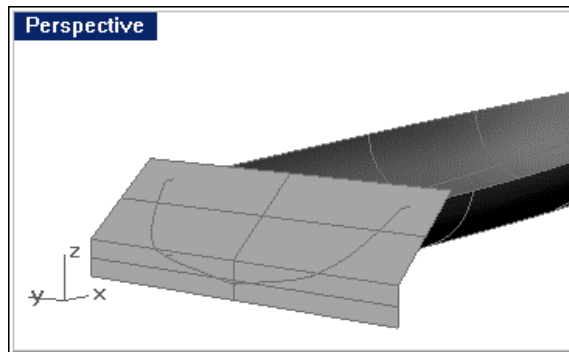
- 7 At the **Extrusion distance ...** prompt, click a point outboard of the hull in the **Top** viewport, such that the resulting extruded surface will be wider than the sheer in that view.



Extrude transom profile curves.

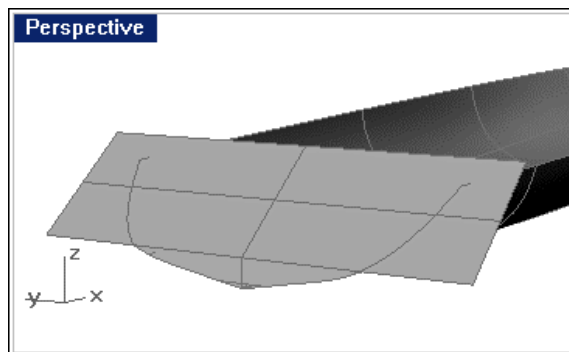
Trim the hull

- 1 Select the two extruded surfaces.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, select the hull polysurface.
- 4 Select the hull polysurface.



Trimmed hull.

- 5 Repeat the **Trim** command.
- 6 At the **Select object to trim ...** prompt, select the vertical portion of the transom outside the hull.



Hull and plumb transom trimmed.

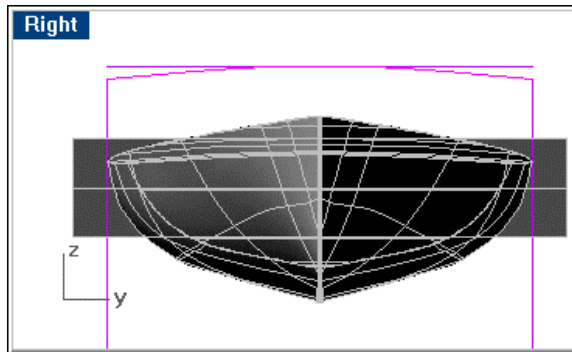
Create the camber curve

You must create a curve on the transom from one side to the other that represents the intersection between the deck and transom. This curve will become the camber curve when you loft the deck.

To create the transom curve, in the **Right** viewport, draw a straight **line** 16 feet long centered on the vessel centerline and above the body plan.

Move this line up in body plan an amount equal to the deck camber. A distance of six inches was used in the example.

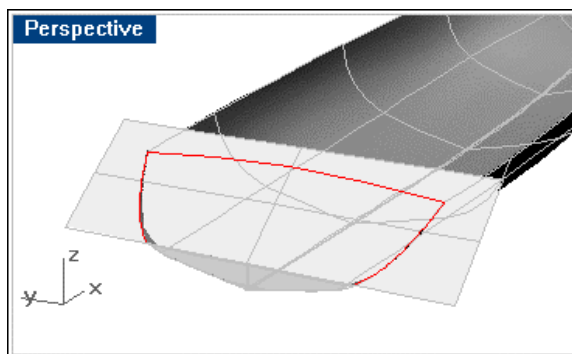
To complete the camber curve, draw an **arc** from the endpoint of the lower of the two lines to the midpoint of the upper line and to the opposite endpoint of the lower one. This will create a pleasing curve for our deck camber.



Camber arc.

The curve will not land on the sheer at the transom, however. To solve this, in body plan (Right viewport) draw a vertical **line** from each corner of the transom up to the newly created camber curve.

Working with a **copy** of the camber curve (save one untrimmed camber curve to use at other stations later) **trim** the copied curve to the two vertical lines you just created and **move** the result to the corner of the transom.



Move the camber arc to the corner.

If you prefer a different camber curve, that is a matter of choice. This method of constructing the camber curve is not mandatory. The transom should be trimmed to whatever camber curve you choose.

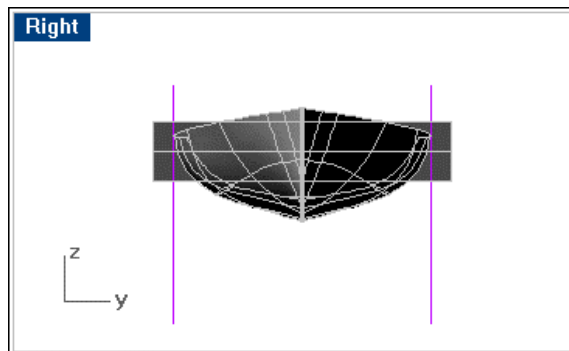
Step-by-Step: Construct the top of the transom

Create a vertical construction line

- 1 Make the **Temp** layer current and turn on the **Camber Crv** layer.
- 2 Start the **Line** command.
- 3 At the **Start of line ...** prompt, in the **Right** viewport, type **96",0**.
- 4 At the **End of line ...** prompt, type **96",175"**.

Mirror the construction line

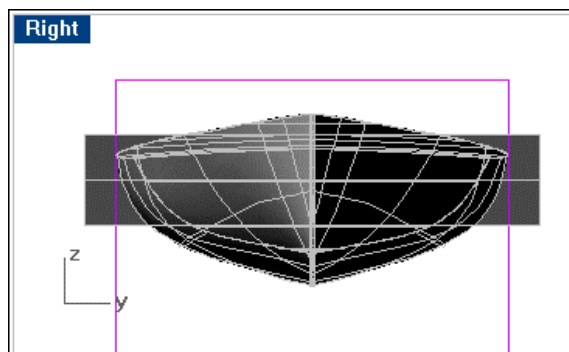
- 1 Select the line you just created.
- 2 Start the **Mirror** command.
- 3 In the **Right** viewport, at the **Start of mirror plane ...** prompt, type **0**.
- 4 At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line up along the y-axis.



Camber curve construction lines.

Draw a horizontal construction line

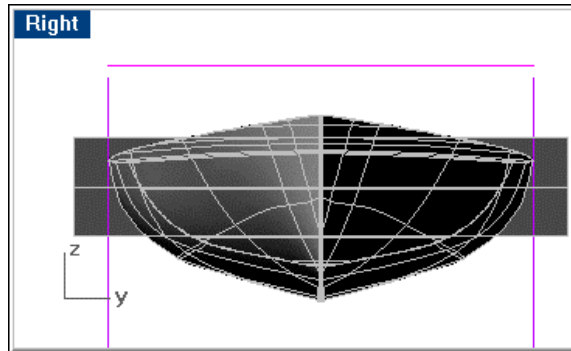
- 1 Start the **Line** command.
- 2 At the **Start of line ...** prompt, use the **End** object snap to pick the upper end of the vertical line on the port side.
- 3 At the **Start of line ...** prompt, use the **End** object snap to pick on the upper end of the vertical line on the starboard side.



Camber curve horizontal construction line.

Move the line up 6"

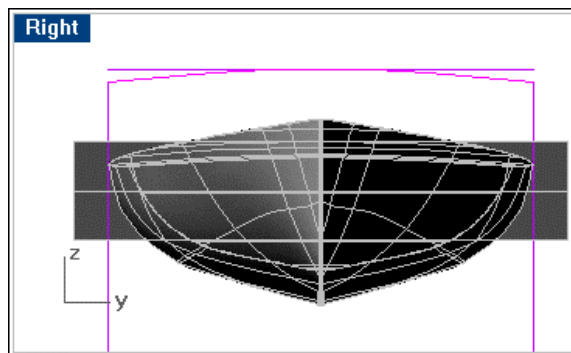
- 1 Select the line you just created.
- 2 Start the **Move** command.
- 3 In the **Right** viewport, at the **Point to move from ...** prompt, pick a point.
- 4 At the **Point to move to** prompt, type **6"** and pick a point above the previous point.



Horizontal construction line moved up.

Draw the camber arc

- 1 Make the **Camber Crv** layer current.
- 2 Start the **Arc3Pt** command.
- 3 At the **Start of arc** prompt, use the **End** object snap to pick the endpoint of the vertical line on the port side.
- 4 At the **End of arc** prompt, use the **End** object snap to pick the endpoint of the vertical line on the starboard side.
- 5 At the **Second point on arc** prompt, use the **Mid** object snap to pick the midpoint of the horizontal line.



The camber arc.

Make a copy of the camber curve

- 1 Select the arc you just created.
- 2 Start the **Copy** command.
- 3 At the **Point to copy from ...** prompt, in the **Right** viewport, click a point.
- 4 At the **Point to copy to** prompt, use **Ortho** to click a point directly above or below the point to copy from.

Create the transom/side intersection curves

- 1 Make the **Temp** layer current.
- 2 Select the sloped transom and both side surfaces.
- 3 Use the **Intersect** command to find the curve that defines the intersection of the two surfaces.

You should now have two intersection curves on the side surfaces.

Construct a line from the transom intersections

- 1 Start the **Line** command.
- 2 At the **Start of line ...** prompt, use the **End** object snap to pick the top of the intersection curve at the starboard sheer.
- 3 At the **End of line ...** prompt, use **Ortho** to pick a point above the copied arc you created previously.

Mirror the vertical line

- 1 Select the vertical line you just created.
- 2 Start the **Mirror** command.
- 3 At the **Start of mirror plane ...** prompt, type **0**.
- 4 At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line up along the y-axis.

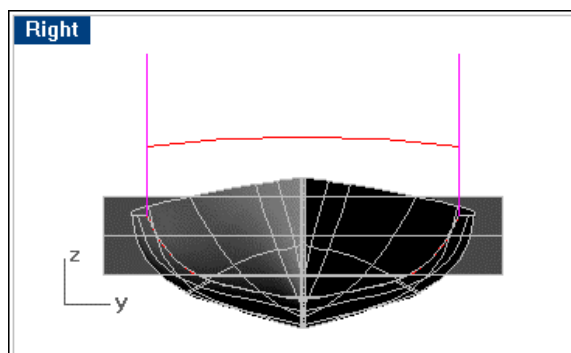
Trim the copy of the camber curve to the width of the transom at sheer

- 1 Select the two vertical lines from the transom intersection corners you just created.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, select the copy of the arc you previously created.

Make your picks at points outboard of the transom intersection corners on each side, and press **Enter**.

You should now have an arc whose length is the same as the width of the sloped transom at the sheer.

This arc is located on the construction plane of the **Right** viewport.

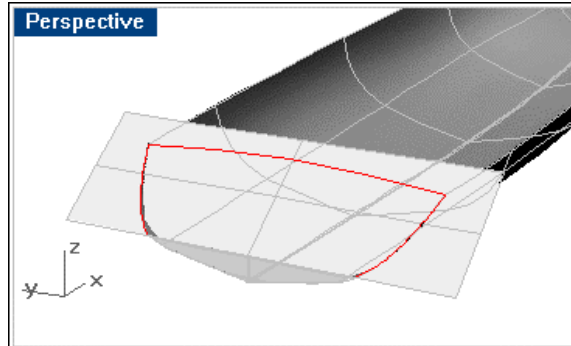


Trimmed camber curve.

Move the trimmed camber curve into place at the transom

- 1 Select the copy of the arc you just trimmed.
- 2 Start the **Move** command.
- 3 At the **Point to move from ...** prompt, use the **End** object snap to pick the starboard endpoint of the trimmed arc.
- 4 At the **Point to move to** prompt, use the **End** object snap to pick the end of the starboard transom intersection curve.

You should now have a curve located at the top of the intersection between the sloped transom and the side.

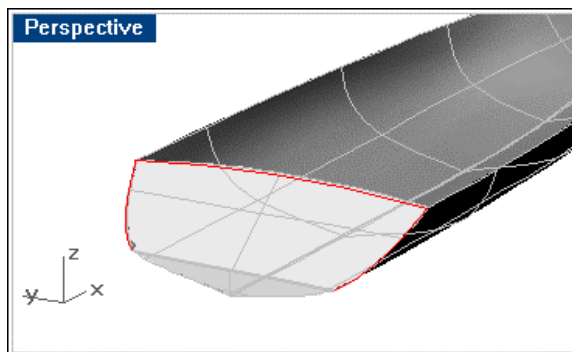


Camber curve moved to endpoint of intersection of hull and transom.

- 5 **Delete** the construction lines created in this phase of the project.
Keep one copy of the untrimmed camber curve.

Finish the transom

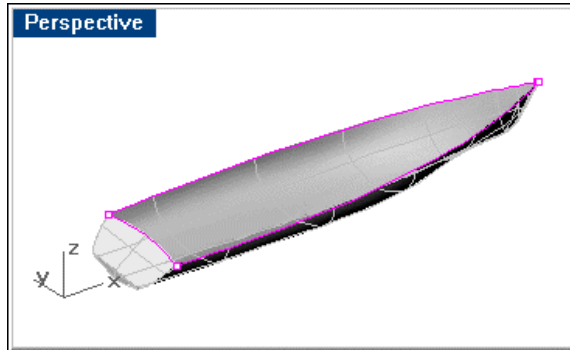
Trim the two transom surfaces to the hull and the camber curve.



Transom trimmed to hull and camber curve.

Join all the surfaces, and check for **unjoined edges**.

You should still have a copy of the untrimmed camber curve is standing by ready to be used in creating the deck.



Trimmed and joined hull and transom.

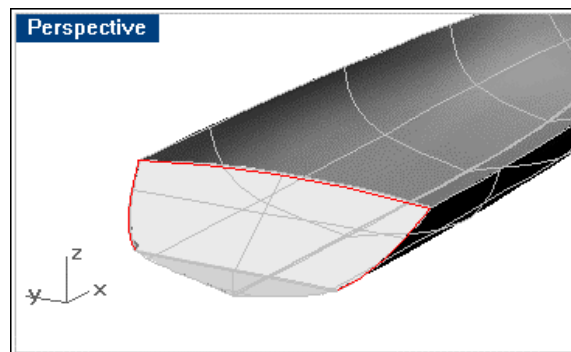
Step-by-Step: Finish the transom

Trim transom

- 1 Make the **Hull and Deck** layer current.
- 2 Select the transom camber curve and the hull on both sides.
- 3 Start the **Trim** command.
- 4 At the **Select object to trim ...** prompt, select the sloped transom outboard of the hull.

You should now have a sloped transom panel that matches the hull around its outer edges and matches the camber curve across the top.

Be certain all surfaces created to date are on the **Hull and Deck** layer.

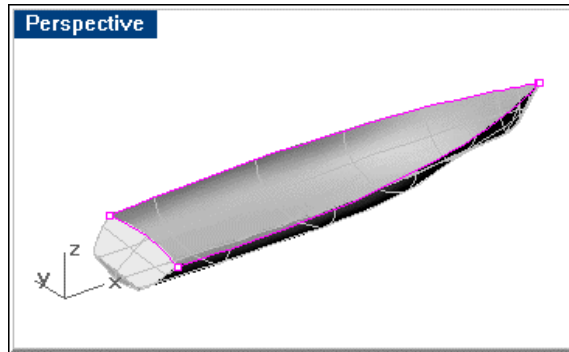


Trimmed transom.

Join the hull surfaces

- 1 Turn off all layers except **Hull and Deck**.
- 2 Select all surfaces.
- 3 Use the **Join** command to join the hull surfaces.

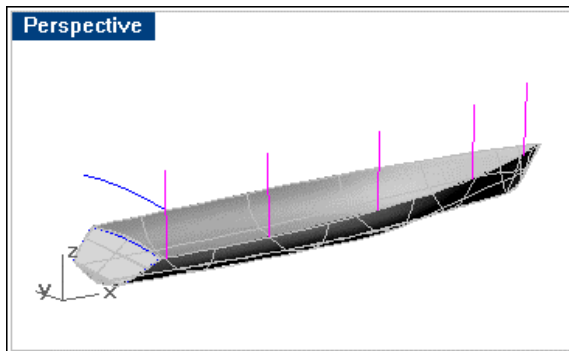
- 4 Use the **ShowEdges** command to check for unjoined surfaces.



Trimmed and joined hull and transom.

Add the deck

To complete the hull so you can calculate hydrostatics, you need to create a deck that will "close the box." This is where the spare camber curve comes in handy. At about 5-6 locations along the sheer, draw a vertical **line**. You can draw one and **copy** it however many times you need it (4 copies in the example) using the **End** and **Near** object snaps.

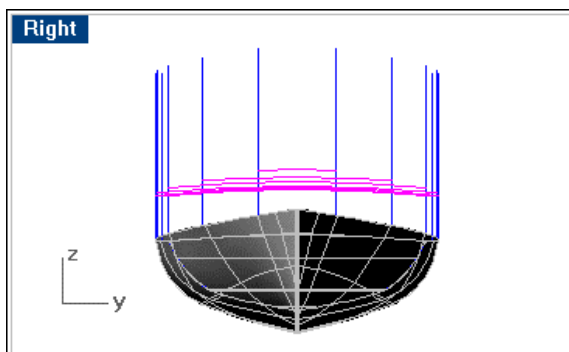


Deck construction lines.

Make one copy of the camber curve for each pair of "posts" plus one (never leave yourself without an untrimmed camber curve).

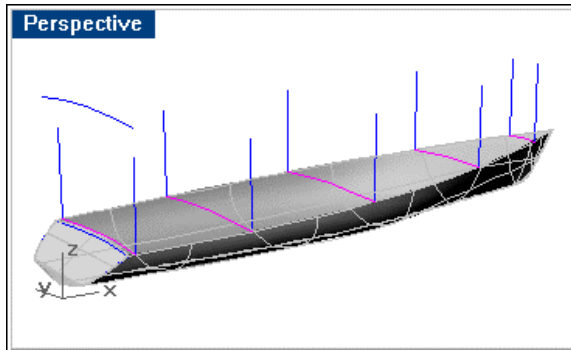
Mirror these posts across centerline.

Trim each camber curve to a pair of posts, so that the camber curve's chord length matches the full breadth of the boat at one of the pairs of posts.



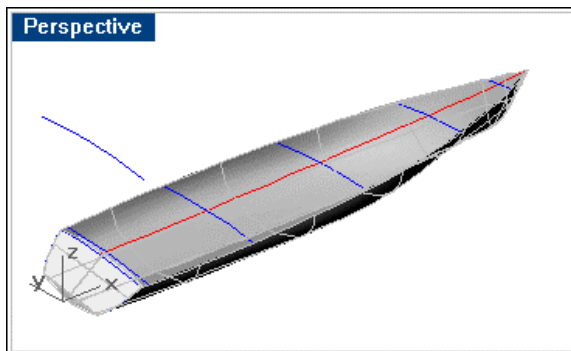
Create deck beams along the sheer.

Once you have all the camber curves trimmed to these posts, **move** each camber curve from its endpoint to the endpoint of its corresponding post. In this way, you will have created a deck beam at each location where you placed one of these pairs of posts. These deck beams will be used to create the centerline of the deck and then will be discarded. Once the temporary deck beams are in place, you can delete the posts.



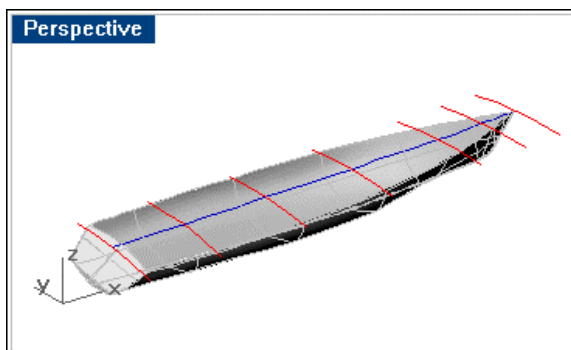
Temporary deck beams in place.

Use the **InterpCrv** command to create a deck centerline by connecting the midpoint of the transom to the midpoints of each of the deck beams and finally the midpoint of the 2" deadflat on center at the stemhead.



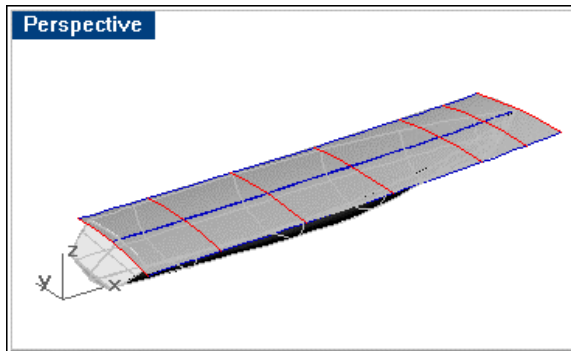
Temporary deck beams in place and deck centerline drawn through midpoints.

Once you have created the deck centerline, **extend** its ends beyond the ends of the hull, **delete** the trimmed deck beams created previously, and make a **copy** of the untrimmed camber curve from its midpoint to various points along the deck centerline as well as one to each end of the deck centerline.



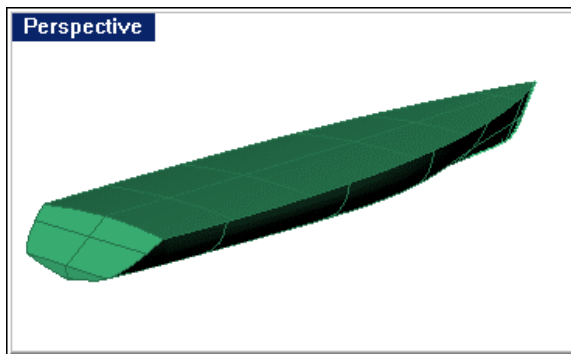
Untrimmed camber curve copied to centerline.

Loft these new curves.



Lofted deck.

Trim the lofted deck to the hull and **join** it to the hull. Use the **ShowEdges** command to check the model for unjoined edges. You should no longer have any unjoined edges on this model.



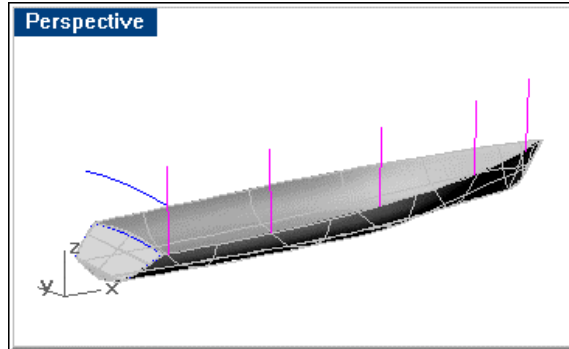
Deck joined to the hull. There should be no unjoined edges.

Step-by-Step: Add the deck

Create vertical trimming lines

- 1 Make the **Temp** layer current and turn on the **Camber Crv** layer.
- 2 Start the **Line** command.
- 3 At the **Start of line ...** prompt, use the **End** object snap to pick a point at the starboard corner of the transom at the sheer.
- 4 At the **End of line** prompt, in the **Right** viewport, pick a point approximately 6 feet above the transom corner.
- 5 Select the vertical line you just created.
- 6 Start the **Copy** command.
- 7 At the **Point to copy from ...** prompt, use the **End** object snap to pick the lower endpoint (at the sheer) of the vertical line you just created.

- 8 At the **Point to copy to** prompt, use the **Near** object snap to pick four evenly spaced points along the starboard sheer.



Deck construction lines.

Mirror the vertical lines across centerline

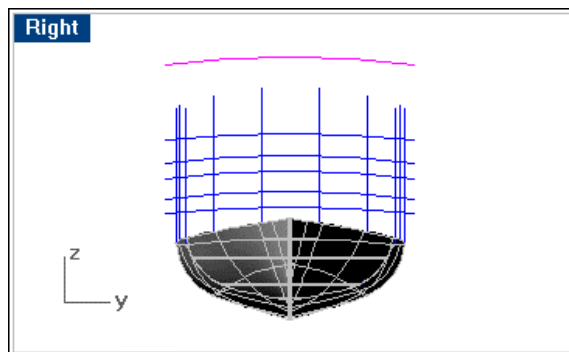
- 1 Select the vertical lines you just created.
- 2 Start the **Mirror** command.
- 3 In the **Right** viewport, at the **Start of mirror plane ...** prompt, type **0**.
- 4 At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line up along the z-axis.

Make copies of the camber curve

- 1 Select the camber curve.
- 2 Start the **Copy** command.
- 3 At the **Point to copy from ...** prompt, in the **Right** viewport, pick a point.
- 4 At the **Point to copy to** prompt, use **Ortho** to pick 5 points directly above or below the origin point you just picked.

The copies of the camber curve being created should cross the vertical lines previously created.

- 5 Select all of the temporary deck beams you just created and use the **ChangeLayer** command to change their layer **Dk Beams Temp**.



One camber curve copy for each construction line.

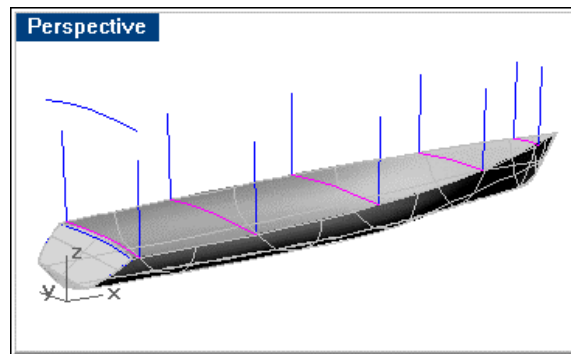
Trim copies of the camber curve to match the beam at each vertical line

- 1 In the **Right** viewport, select one of the vertical lines on the starboard side and the corresponding vertical line on the port side.
- 2 Start the **Trim** command.

- 3 At the **Select object to trim ...** prompt, pick the outboard end of one of the camber curves on the starboard side and the outboard end of the same camber curve on the port side.
- 4 Repeat these steps for each of the camber curves.
Each of the curves thus created will be a temporary deck beam to enable creation of the deck centerline.

Move the temporary deck beams into place on the sheer

- 1 Select one of the temporary deck beams.
- 2 Start the **Move** command.
- 3 At the **Point to move from ...** prompt, use the **End** object snap to pick a point on the end of the selected temporary deck beam.
Do not pick a point on the vertical line.
- 4 At the **Point to move to** prompt, use the **End** object snap to pick the endpoint of the corresponding vertical line where it meets the sheer.
Do not pick a point on the sheer.
- 5 **Delete** the two vertical lines associated with the temporary deck beam you just moved.
- 6 Repeat these steps for each of the temporary deck beams.

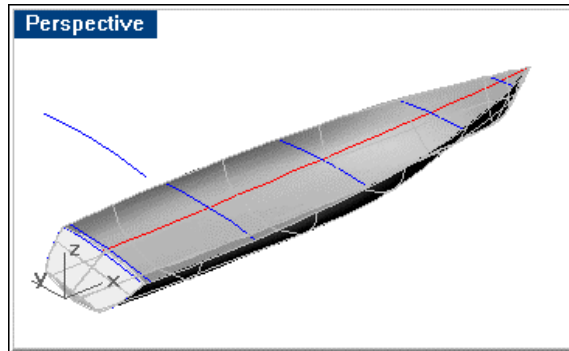


Temporary deck beams in place.

Draw the deck centerline

- 1 Start the **InterpCrv** command.
- 2 At the **Start of curve ...** prompt, use the **Mid** object snap to pick a point at the midpoint of the transom sheer.
- 3 At the **Next point of curve...** prompts, use the **Mid** object snap to pick a point at the midpoint of each of the temporary deck beams.

- 4 At the **Next point of curve...** prompt, use the **Mid** object snap to pick a point at the midpoint of the upper edge of the centerline deadflat at the stemhead.



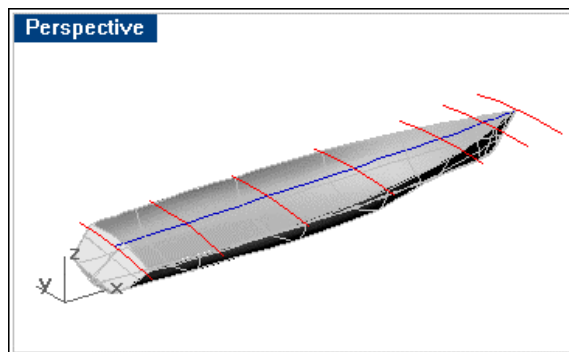
Temporary deck beams in place and deck centerline drawn through midpoints.

Extend the deck centerline

- 1 Turn on **Planar** mode.
- 2 Start the **Extend** command.
- 3 At the **Select boundary objects ... Press Enter for dynamic extend** prompt, press **Enter** to indicate dynamic extend.
- 4 At the **Select object to extend (Type=Line)** prompt, in the **Front** viewport, pick a point near the transom on the new deck centerline.
- 5 At the **End of extension** prompt, pick a point approximately 1 foot aft of the transom.
- 6 Repeat these steps for the forward end of the deck centerline curve.

Copy full-breadth camber curves to the deck centerline

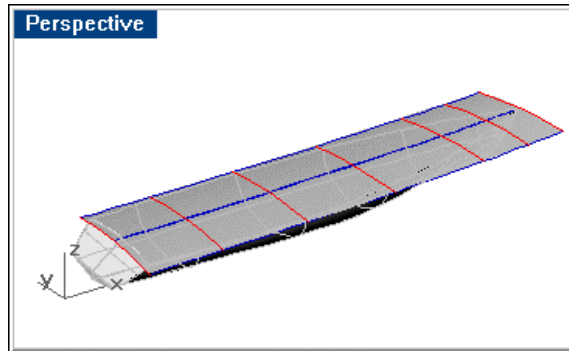
- 1 Select the temporary deck beams.
- 2 Select the camber curve that remains suspended above the hull.
- 3 Start the **Copy** command.
- 4 At the **Point to copy from ...** prompt, use the **Mid** object snap to pick the midpoint of the selected camber curve.
- 5 At the **Point to copy to** prompt, use the **End** object snap to pick the aft end of the deck centerline.
- 6 At the **Point to copy to** prompt, use the **End** object snap to pick the forward end of the deck centerline.
- 7 At the **Point to copy to** prompt, use the **Near** object snap to pick four evenly spaced points along the deck centerline.



Untrimmed camber curve copied to centerline.

Loft the full-breadth deck

- 1 Select the full-breadth camber curves.
- 2 Start the **Loft** command.
- 3 In the **Loft Options** dialog box, click **OK** to accept the defaults.



Lofted deck.

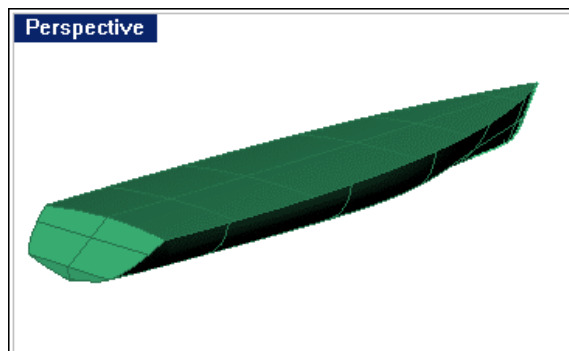
- 4 Select the full-breadth camber curves.
- 5 Use the **ChangeLayer** command to change them to layer **Dk Bms Full**.

Trim the full-breadth deck to the hull

- 1 Select the hull.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, select the deck outboard of the sheer. The deck will trim to match the sheer, transom and centerline deadflat.

Join the full-breadth deck to the hull

- 1 Select the hull and deck.
- 2 Use the **Join** command to join the hull and deck surfaces into one polysurface.
- 3 Select the hull/deck polysurface.
- 4 Use the **ShowEdges** command to display any unjoined edges.



Trimmed and joined hull, transom, and deck.

Diagnostics

One final step awaits you before you can move on to calculating hydrostatics, generating a lines drawing, and accepting your accolades for a job well done. You must determine if you have any geometric problems. Rhino has several tools to tell you about bad geometry. You will use the **SelBadObjects** and **ShowEdges** commands.

With these two commands, you can determine if Rhino's hydrostatics function as well as other third-party software will be able to work with your model. If you run these and get positive results, you can rest. Otherwise, you must seek out the culprits (usually broken edges), fix them, and rerun the commands until the results are positive.

In creating this model for the tutorial, a bad edge was encountered at the lower panel of the transom. **Untrimming** the panel and the hull, **retrimming** and **rejoining** everything solved the problem. As with most diagnostic procedures, there is no one answer. Each situation requires its own solution.

Step-by-Step: Diagnostics

Find objects containing errors

- ▶ Use the **SelBadObjects** command to check for objects with errors.
If the model contains bad objects, these will be selected. Measures should then be taken to correct the problems.

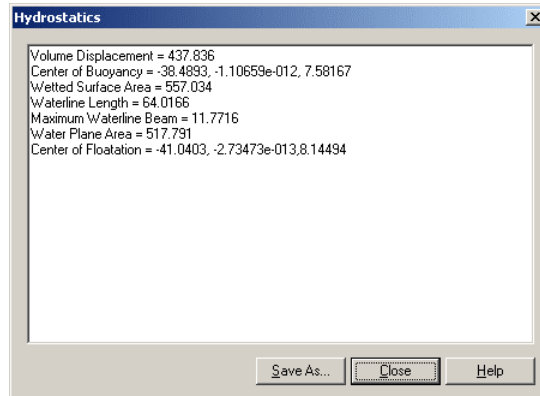
Find broken edges

- 1 Select the hull/deck polysurface.
- 2 Use the **ShowEdges** command to find unjoined edges.
If any unjoined edges are present, Rhino will highlight them.
If not, we can move on to calculating hydrostatics and generating a GHS file.

Calculate hydrostatics

Since the target displacement is 28,000 lb, the volume displaced will be 437.5 cu ft. Run the **Hydrostatics** command, and modify the **WaterlineElevation** until you achieve this displacement. For this model, it turns out to be 1.905 feet. Your results may vary.

The image below shows the other information available from the **Hydrostatics** command.



Rhino Hydrostatics calculation.

Step-by-Step: Calculate hydrostatics

Analyze Hydrostatics

- 1 Start the **Hydrostatics** command.
Do not select the hull first. The options we want to use do not display if an object is selected
- 2 At the **Select surfaces or polysurfaces (WaterlineElevation=0.00000 Symmetric=No Longitude=X)** prompt, type **W**.
- 3 At the **Waterline elevation < 0 >** prompt, type **1.905**.
- 4 At the **Select surfaces or polysurfaces ...** prompt, select the hull/deck polysurface.

The **Hydrostatics** dialog box displays.

If the **Volume Displacement** does not equal **437.5** within your tolerance, repeat the Hydrostatics command with a different **WaterlineElevation** value until you find the waterline elevation that gives you the displacement you are looking for.

If other values do not appear, there may be something wrong with your design, or you may need to set a looser tolerance for the **Hydrostatics** command to work.

- 5 Click **Save As** to save the display to a text file.
- 6 Click **Close**.

Generate GHS File

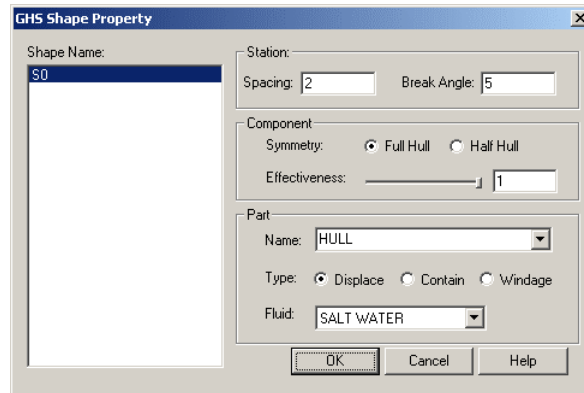
You can also export your model to a geometry file suitable for use with GHS© by Creative Systems, Inc. This software is the industry standard for hydrostatic analysis.

Step-by-Step: Generate GHS File

To create a GHS compatible file

- 1 Start the **Export** command.
- 2 In the **Export Selected Objects** dialog box, in the **File name** box, enter a name.

- 3 From the **Save as type** list, select **GHS Export File (.gf)**.
- 4 At the **Select the base point of the centerline** prompt, pick a point on the longitudinal axis of the boat.
This is normally a point on the bow at centerline.
- 5 At the **Select end point of center line** prompt, pick another point at the end of the longitudinal axis.
- 6 In the **GHS Shape Property** dialog box, accept the defaults.



GHS Shape Property dialog box.

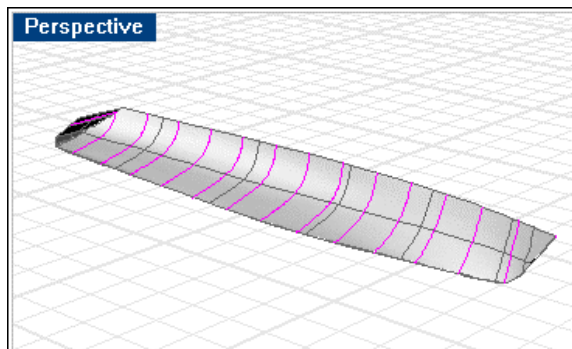
Create the lines drawing

To create the finished lines plan, you need only half of the boat. **Save** the model to a separate file, open the file and **trim** the model to centerline. **Delete** the deck from the model, since the waterlines and buttocks across the deck are rather distracting in a lines drawing. Select the hull polysurface. Hide everything else.

After performing these steps, use the **Contour** command to create the lines that go into the traditional lines plan: stations, waterlines, and buttocks. For this purpose, use layers to keep the various curves separate.

For the stations in the **Top** or **Front** viewport, specify the **Contour basepoint** at 0, **Contour Plane Direction** towards the stern, and the **Spacing** at 5. **Rhino** will then calculate the appropriate curves and display them on the hull.

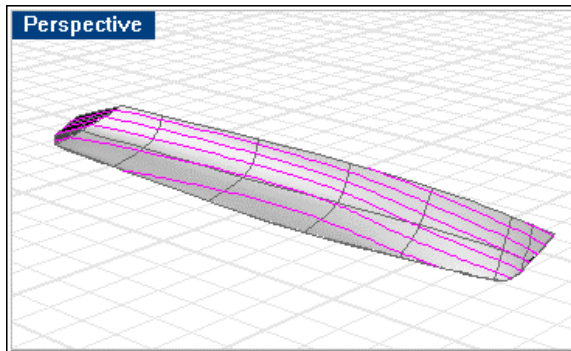
These curves remain selected after the command has executed. Use the **ChangeLayer** command to change the stations to the **Stations** layer.



Station lines.

For the waterlines in the **Front** viewport, specify the **Contour basepoint** at 0, **Contour Plane Direction** as up, and the **Spacing** at 1.

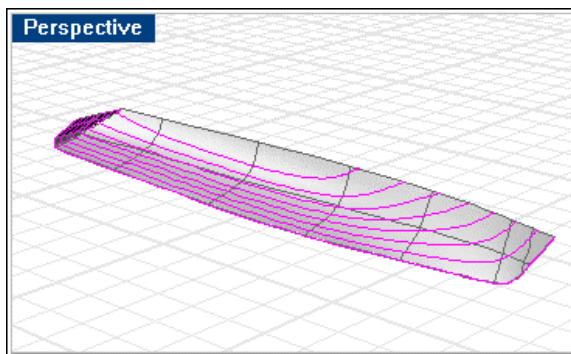
Use the **ChangeLayer** command to change the waterlines to the **Waterlines** layer.



Waterlines.

For the buttocks in the **Top** viewport, specify the **Contour basepoint** at 0, **Contour Plane Direction** as outboard, and the **Spacing** at 1. **Rhino** will then calculate the appropriate curves and display them on the hull.

Use the **ChangeLayer** command to change the waterlines to the **Buttocks** layer.



Buttock lines.

Make sure only the hull/transom polysurface and the curves you want included in the lines drawing (buttocks, waterlines, stations) are visible.

It is also a good idea to draw a baseline in the **Front** and **Right** viewports (for the profile and body plan) and a centerline in the **Right** and **Top** viewports for the body plan and plan.

Then, when all the curves are visible and selected, use the **Make2D** command. Rhino will create a lines drawing with four views (body plan, plan, profile and perspective with hidden lines removed) and place the views on the construction plane.

The resulting curves will still be selected, so immediately after the command finishes, use the **Export** command and write the curves out to another file. This will most likely be in the format supported by your CAD program, so you can label and dimension the lines drawing.

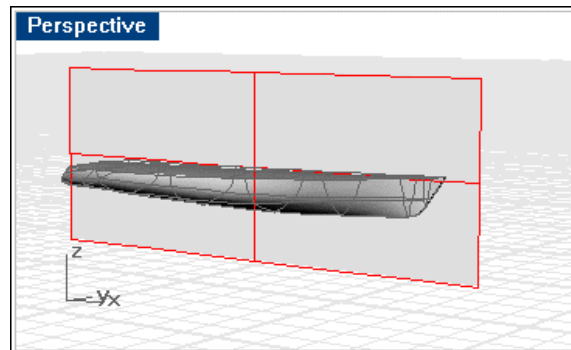
Step-by-Step: Create the lines drawing

Create a new model to generate a lines drawing from

- 1 From the **File** menu, click **Save As**.
- 2 In the **Save As** dialog box, type **Sail Lines**.
- 3 Make the **Hull and Deck** layer current and turn off all other layers.

Draw a trimming plane

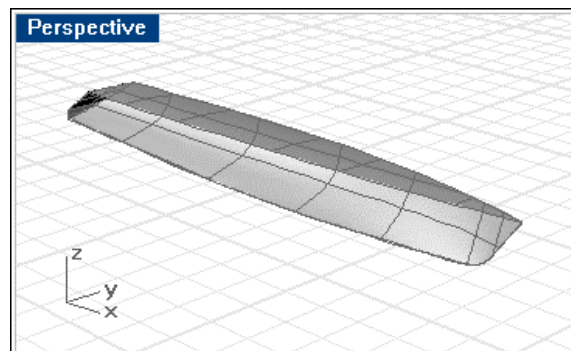
- 1 In the **Front** viewport, use the **Zoom Extents** command to bring all objects into the view.
- 2 Zoom out a little so that there is room all around the profile.
- 3 Start the **Plane** command.
- 4 At the **First corner of plane ...** prompt, in the **Front** viewport, pick a point below the model and aft of the transom.
- 5 At the **Other corner or length** prompt, pick a point above the sheer and forward of the stemhead.



Trimming plane.

Trim the hull with the plane

- 1 Select the rectangular surface you just created.
- 2 Start the **Trim** command.
- 3 In the **Right** viewport at the **Select object to trim ...** prompt, select the hull on the starboard side.
- 4 **Delete** the rectangular surface.

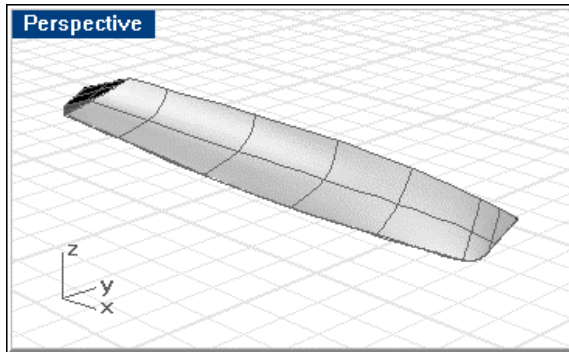


Trimmed hull.

Delete the deck

- 1 Start the **ExtractSrf** command.
- 2 At the **Select surfaces to extract ...** prompt, select the deck.

- 3 **Delete** the deck surface.

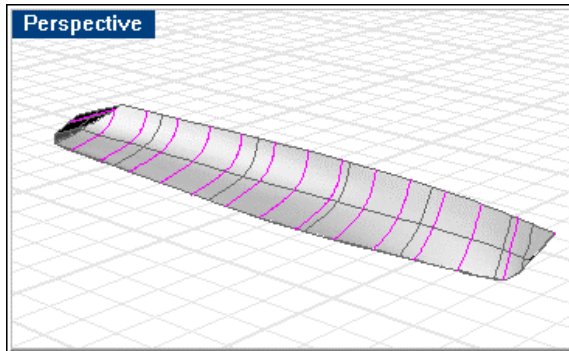


Deck deleted.

- 4 If anything other than the hull polysurface is visible, use the **Hide** command to remove it from your view.

Create the stations

- 1 Select the hull/transom polysurface.
- 2 Start the **Contour** command.
- 3 In the **Front** viewport, at the **Contour plane base point** prompt, type **0**.
- 4 At the **Contour plane direction** prompt, use **Ortho** to click a point aft of the transom.
- 5 At the **Distance between contours<1>** prompt, type **5**.



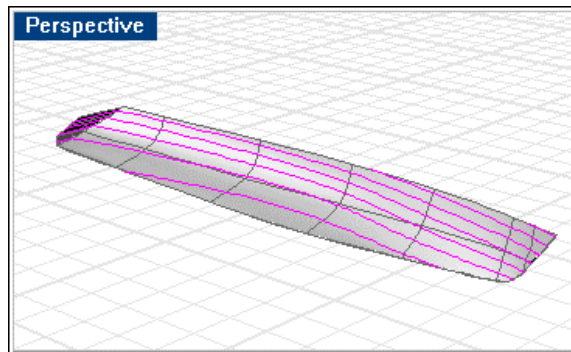
Station lines.

- 6 With the newly created station curves still selected, use the **ChangeLayer** command to change the curves to the **Stations** layer.

Create the waterlines

- 1 Select the hull/transom polysurface.
- 2 Start the **Contour** command.
- 3 In the **Front** viewport, at the **Contour plane base point** prompt, type **0,0**.
- 4 At the **Contour plane direction** prompt, use **Ortho** to click a point above the sheer.

- 5 At the **Distance between contours <1>** prompt, type **1**.

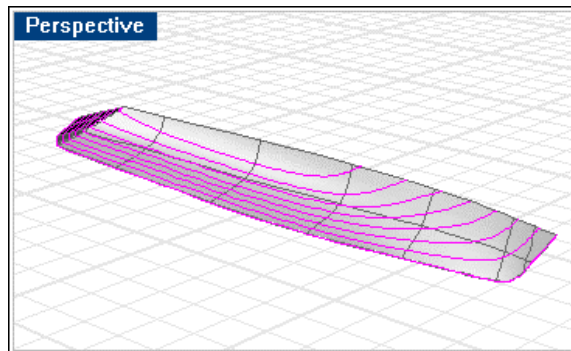


Waterlines.

- 6 With the newly created waterline curves still selected, use the **ChangeLayer** command to change the curves to the **Waterlines** layer.

Create the buttocks

- 1 Select the hull/transom polysurface.
- 2 Start the **Contour** command.
- 3 In the **Top** viewport, at the **Contour plane base point** prompt, type **0**.
- 4 At the **Contour plane direction** prompt, use **Ortho** to click a point outboard of the sheer.
- 5 At the **Distance between contours <5>** prompt, type **1**.



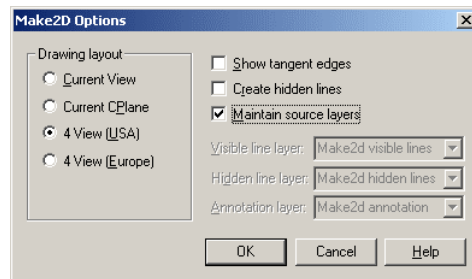
Buttock lines.

- 6 With the newly created buttock curves still selected, use the **ChangeLayer** command to change the curves to the **Buttocks** layer.

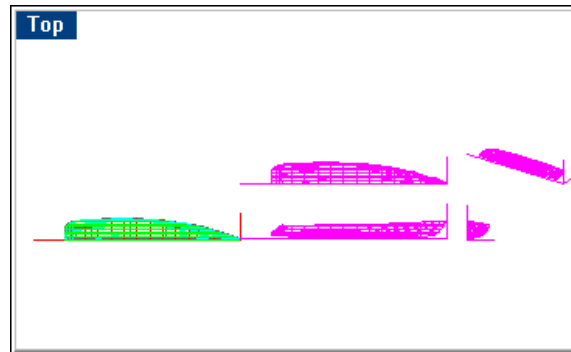
Create the two-dimensional lines drawing

- 1 Draw a baseline in the **Front** and **Right** viewports (for the profile and body plan) and a centerline in the **Right** and **Top** viewports for the body plan and plan.
- 2 Select the hull/transom polysurface and the station, buttock, and waterline curves you just created.
- 3 Start the **Make2D** command.
- 4 In the **Make2D Options** dialog box, under **Drawing layout**, click **4-view (USA)**.

5 Click **Maintain source layers**.



Make 2D Options dialog box.

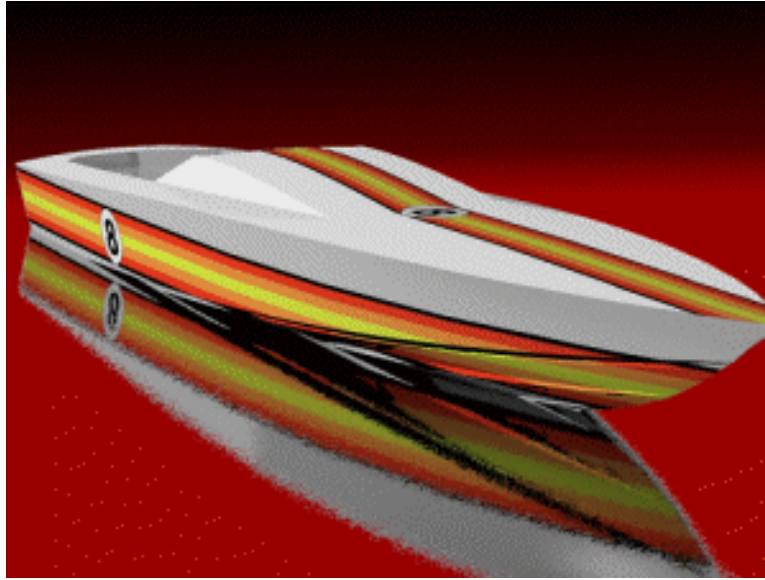


2-D drawing created.

- 6 With the newly created 2-D curves still selected, use the **Export** command to save the 2-D curves to a file.
- 7 In the **Export Selected Objects** dialog box, select the file type corresponding to your CAD program.
- 8 Specify a path and file name for your lines drawing.

You should now have a two-dimensional lines drawing on your hard drive in the location you specified. Use your CAD program to complete the dimensioning and annotation of the lines drawing.

Part IV: Offshore Racing Power Boat



Introduction

This tutorial will introduce you to the methods needed to create a fair hull from a designer's lines drawing. You will import the curves needed from the lines into an empty .3dm file, fair the hull, add running strakes, and use Rhino analysis tools to be sure the hull is fair. When the hull is finished, you will join the sides, add a deck, and check to be sure the completed assembly is suitable for export to other programs, as well as further analysis, should that be necessary.

You will learn to:

- Import curves from a designer's CAD drawing.
- Loft surfaces from a set of minimal curves.
- Fair lofted surfaces using curvature-mapping techniques.
- Use objects created in more than one Rhino model.
- Analyze polysurfaces for geometric flaws.

If you have not completed the Skiff tutorial, you should do this before attempting the Offshore tutorial.

The model created for this tutorial is installed with this tutorial to help you if you are stuck.

The vessel

The vessel to be modeled in this tutorial is similar in form to an offshore racing powerboat. This is not a tutorial in boat design, the purpose of this tutorial is to instruct in the art of fairing hulls and superstructures using Rhino techniques.

The files

The files for this model are in the **Offshore Tutorial** folder. The files included with this tutorial include:

Offshore.3dm – A blank Rhino model set up with layers, grid, units, and tolerances.

Hull Lines.3dm – 2-D lines drawing from which the vessel will be lofted.

Deck Lines.3dm – 2-D lines drawing from which the deck, cabin, and cockpit will be lofted.

Example files of finished parts:

Deck Finished.3dm – Finished deck/cockpit/cabin model.

Offshore Finished.3dm – The working model.

If you have not completed the Skiff tutorial, do that first.

Layers

An empty model file, **Offshore.3dm**, is provided for this project.

In the model file, layers beginning with lower case "c" will be used for storing curves. The layers beginning with lower case "s" will be for storing surfaces. In this way, you can hide or display all curves or surfaces quickly.

Model units are set to **feet**.

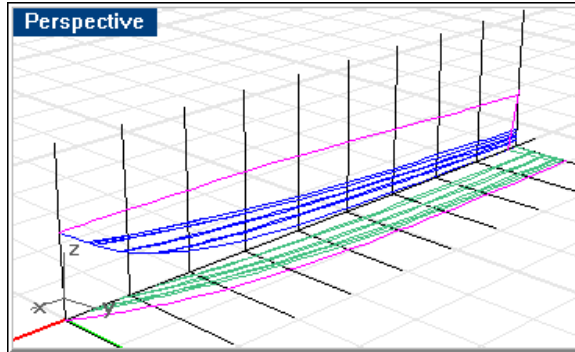
The absolute tolerance is set to **0.01**.

Lay out the hull curves

In the **Hull Lines** model, **rotate** the profile and body plan curves so they are perpendicular to the plan curves.

Move the curves so they line up with the centerline.

Copy the sheer in plan and profile and the transom centerline to the new model.

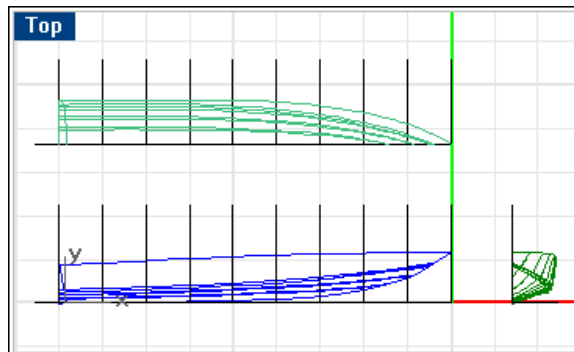


Lines oriented in 3-D.

Step-by-Step: Copy the two-dimensional curves

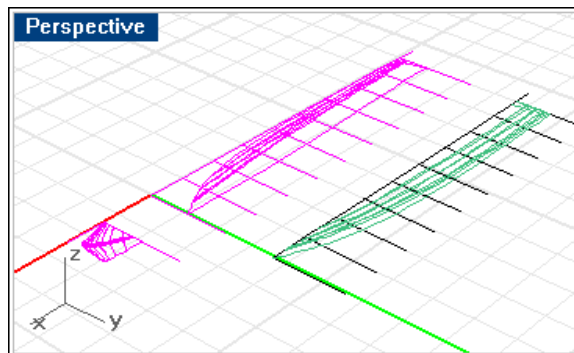
Rotate the profile and body curves

- 1 Open the model **Offshore.3dm**.
- 2 Start a second Rhino session and open the model **Hull Lines.3dm**.



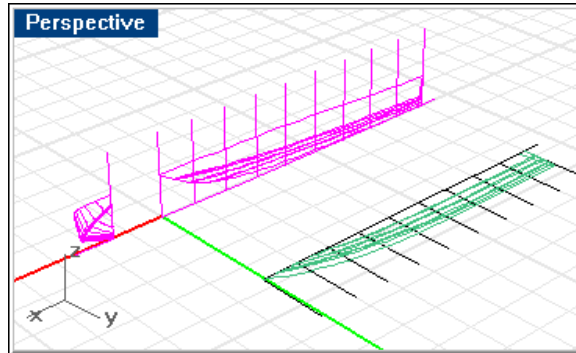
Lines drawing.

- 3 In **Hull Lines.3dm**, select all the curves in the profile and body plan including the grid lines.



Select profile and body plan lines.

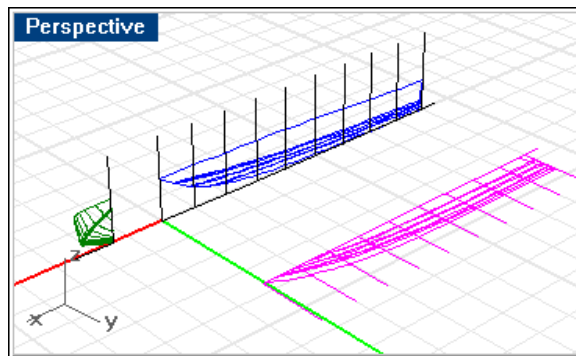
- 4 Start the **Rotate** command.
- 5 At the **Center of rotation ...** prompt, in the **Right** viewport, type **0**.
- 6 At the **Angle or first reference point ...** prompt, use **Ortho** to drag the rotation angle **90 degrees**.



Rotated profile and body plan.

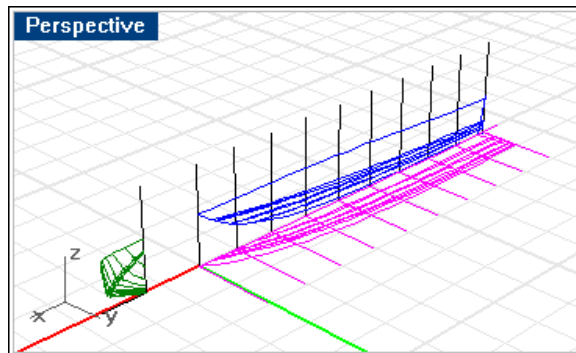
Move the curves to the center

- 1 Select all the curves in the plan view.



Select plan view curves.

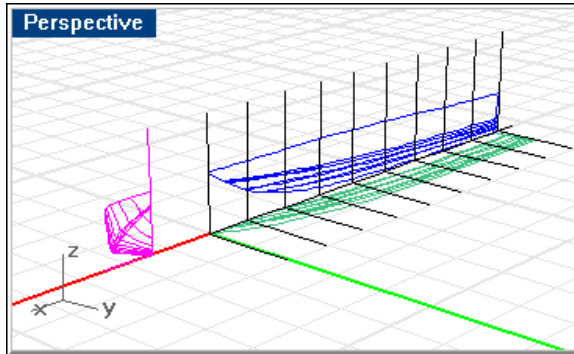
- 2 Start the **Move** command.
- 3 At the **Point to move from ...** prompt, use the **Near** object snap to select a point on the plan view centerline.
- 4 At the **Point to move to** prompt, use the **Perp** object snap to pick a point on the profile view baseline.



Move plan view curves to centerline.

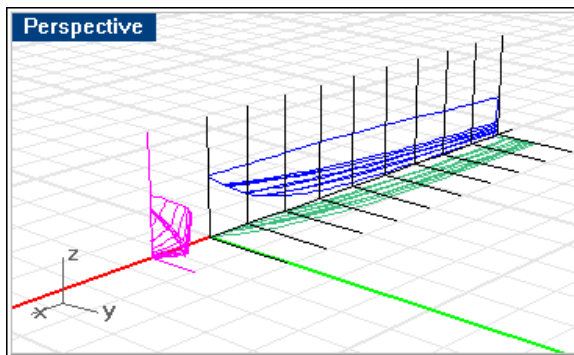
Rotate the body plan curves

- 1 Select all the curves in the body plan.



Select body plan curves.

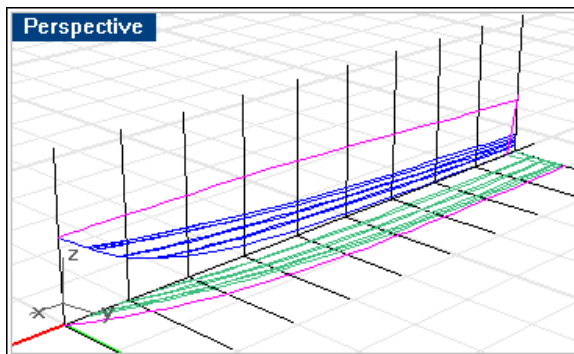
- 2 Start the **Rotate** command.
- 3 At the **Center of rotation ...** prompt, in the **Top** viewport, use the **End** object snap to pick the end of the body plan centerline.
- 4 At the **Angle or first reference point ...** prompt, use **Ortho** to rotate the curves **90 degrees**.



Rotate body plan curves.

Copy the sheer and transom profile to the new model

- 1 In the **Perspective** viewport, select the sheer in plan and profile and the transom centerline in profile.

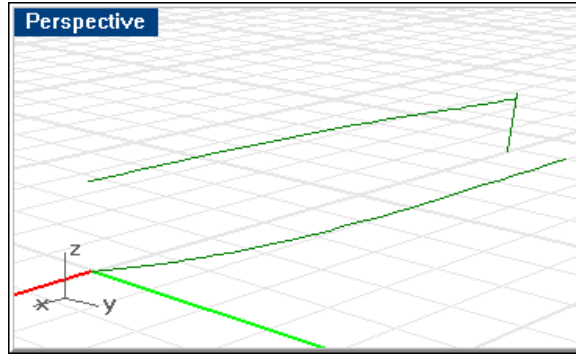


Sheer in plan and profile and transom centerline.

- 2 Press **Ctrl+C** to copy the selected curves to the Clipboard.

Paste shear and transom profile to Offshore.3dm

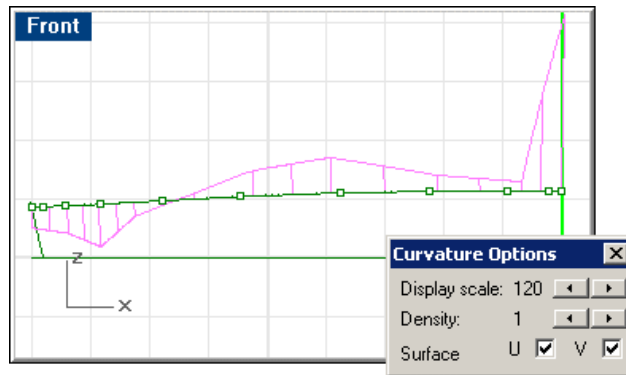
- 1 In **Offshore.3dm**, turn on layers **0**, **cXsomProf**, and **cSheer**.
- 2 Make the **cSheer** layer current and turn all other layers off.
- 3 Press **Ctrl+V** to paste the contents of the Clipboard into the current model.
- 4 Use the **ChangeLayer** command to change the shear curves to the **cSheer** layer and the transom centerline to the **cXsomProf** layer.
- 5 **Save** your work.



Pasted curves.

Fair the two-dimensional curves

Since you did not personally produce these lines, it is necessary to check the fairness of these curves before basing your modeling work on them. Use the **CurvatureGraph** and **PointsOn** commands to display the curvature graph and control points of the shear in profile. Your curve should resemble the image below.

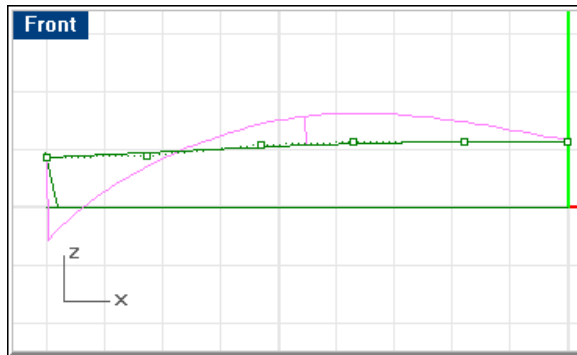


Curvature graph for shear.

The graph indicates a little work is needed on the curve, since there are two sharp changes in direction on the curvature graph. Additionally, considering how simple this curve is, the number of control points is excessive. This condition is quite common in geometry imported from CAD programs, which often display curves as polylines with no curvature. They must use many straight segments to approximate the curve.

The first step in correcting this is to **Rebuild** the curve to a higher degree with fewer points. Use the **Rebuild** command, and select the shear in profile and plan. Set the **Point count** to **6** and the **Degree** to **5**. The number of points is being reduced to the absolute minimum necessary to construct this curve without significantly changing its shape. The degree of the curve should be one less than the number of points.

The shear curve should now resemble the image below.



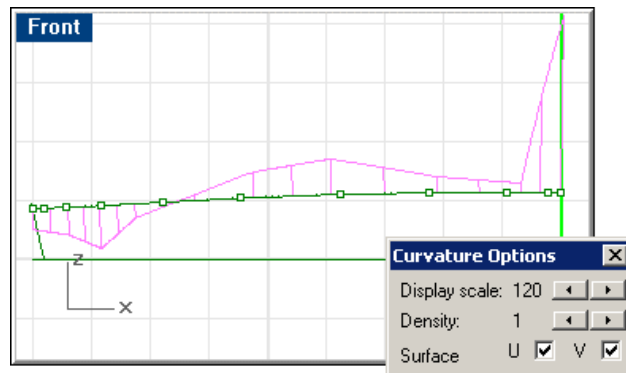
Rebuilt shear curve.

Rebuilding the curve did all the work of tweaking points to achieve a smooth curvature graph. In the original curves, the sharp changes in direction indicated there were kinks in the curve. These could be faired out by moving control points, but this would require much more time and effort and the results would be harder to use for surfacing later. You should now check the curvature graph of the plan view of the shear and modify it as required.

Step-by-Step: Fair the two-dimensional curves

Display the control points and curvature graph

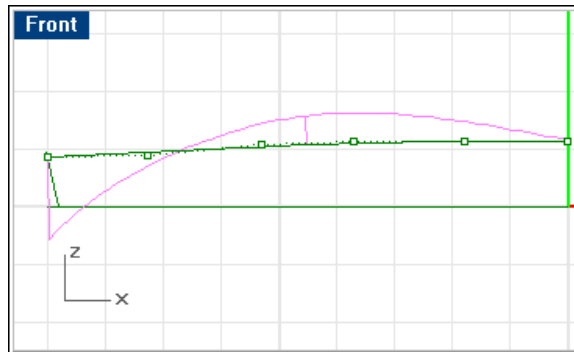
- 1 Select the shear curve in profile.
- 2 Use the **PointsOn** command to display the control points for the curve.
- 3 Use the **CurvatureGraph** command to display the curvature graph.



Curvature graph for profile shear.

Rebuild the curve

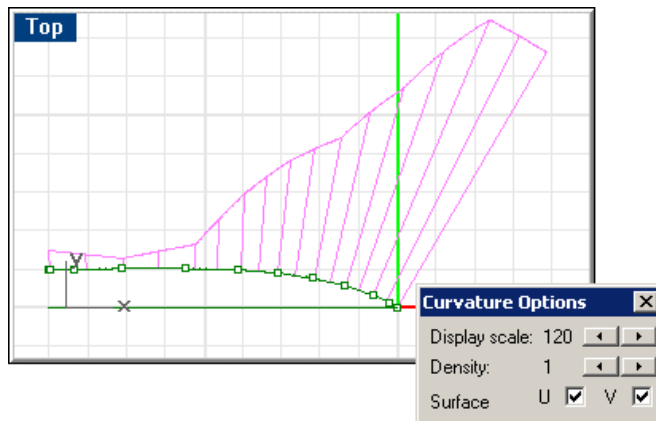
- 1 With the profile of the sheer curve still selected start the **Rebuild** command.
- 2 In the **Rebuild Curve** dialog box, set the **Point count** to **6** and the **Degree** to **5**.



Rebuilt sheer curve.

Display the control points and curvature graph

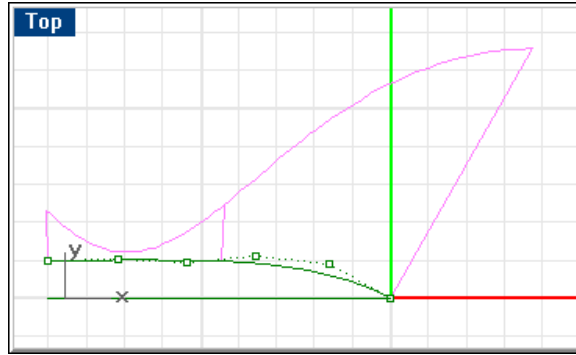
- 1 Select the sheer in plan.
- 2 Use the **PointsOn** command to display the control points for the curve.
- 3 Use the **CurvatureGraph** command to display the curvature graph.



Curvature graph for plan sheer.

Rebuild the curve

- 1 With the profile of the sheer curve still selected start the **Rebuild** command.
- 2 In the **Rebuild Curve** dialog box, set the **Point count** to **6** and the **Degree** to **5**.



Rebuilt plan shear curve.

Fair the 2-D chine curves

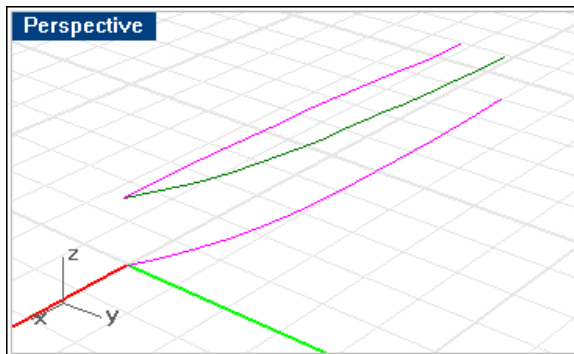
- 1 Copy the chine curves from the hull lines drawing.
- 2 Change the chine curves to the proper layers.
- 3 Use the same method to fair the upper and lower chine curves that you used for the shear.

Create the three-dimensional curves

With the **Crv2Views** command, you can create a three-dimensional curve from two planar curves. Make the **cSheer** layer current. Using the **Crv2Views** command, select the shear in plan and profile. The shear 3-D curve will be created on the layer **cSheer**.

Once you have created the 3-D shear, **delete** the 2-D curves. **Copy** and **paste** each pair of curves representing the upper and lower chines and rabbet.

We will deal with the running strakes and chine strakes after the hull has been surfaced.

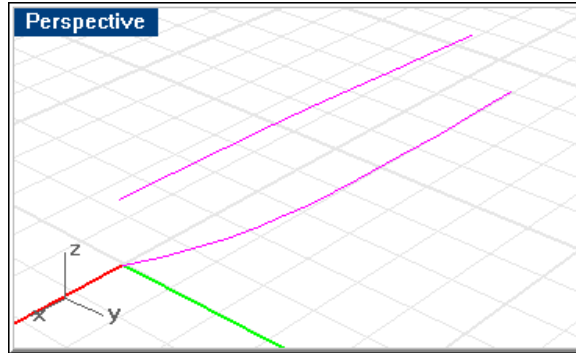


The three-dimensional shear (shown in green).

Step-by-Step: Create the three-dimensional curves

Create the 3-D shear

- 1 Turn on layer **0** and the layers **cXsomProf** and **cSheer**. All other layers should be off.
- 2 Make the **cSheer** layer current.
- 3 Select the shear plan in the **Top** viewport and the shear profile in the **Front** viewport.

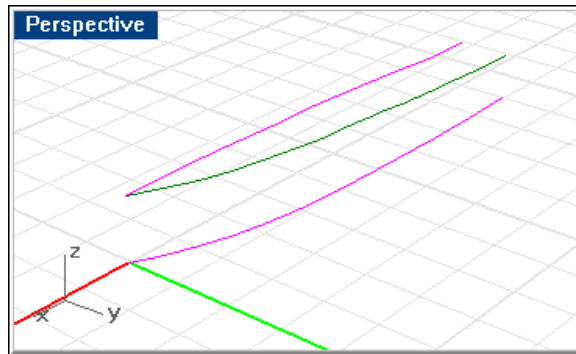


Shear in plan and profile.

- 4 Start the **Curve2View** command.

The **Crv2View** command creates the curve that describes the "intersection" of the two curves.

The original 2-D curves are highlighted to set them apart in the illustration.

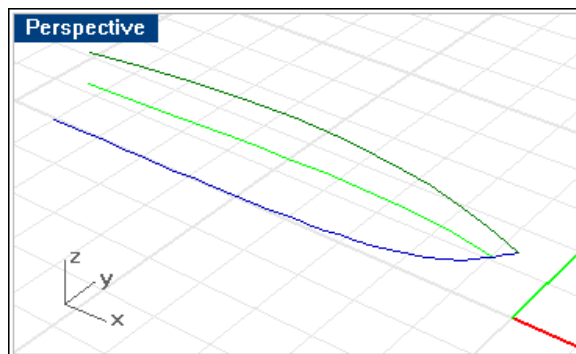


The three-dimensional shear (shown in green).

- 5 **Delete** the 2-D curves.

Create the 3-D upper chine

- 1 Make the **cUpperChine** layer current.
- 2 Select the upper chine plan curve in the **Top** viewport and the upper chine profile curve in the **Front** viewport.
- 3 Start the **Curve2View** command.

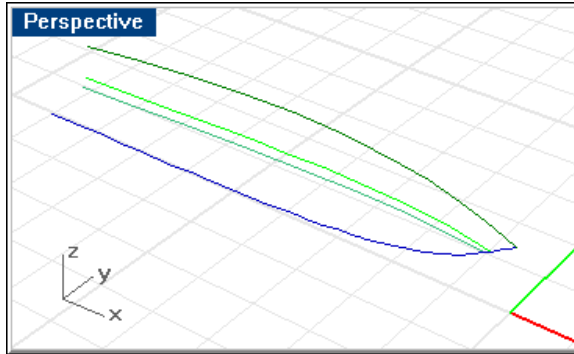


The 3-D upper chine.

- 4 **Delete** the 2-D curves.

Create the 3-D lower chine

- 1 Make the **cLowerChine** layer current so that the curve will be created on that layer.
- 2 Select the lower chine plan curve in the **Top** viewport and the lower chine profile curve in the **Front** viewport.
- 3 Start the **Curve2View** command.



The 3-D lower chine.

- 4 **Save** your work.

Import the stations

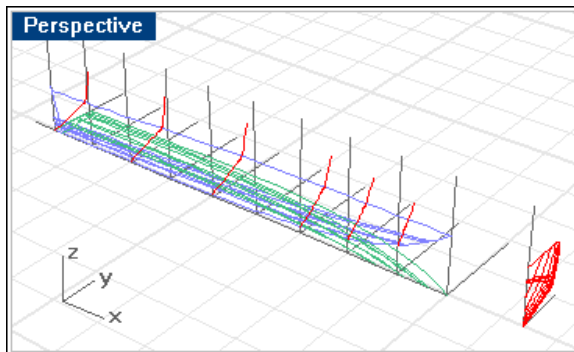
The designer provided you with stations in the lines drawing, but they are all in the same plane. You will make use of these stations in lofting the side and bottom, so the next step will be to place each station curve in its proper longitudinal location and then copy these curves from the lines drawing to your model.

The lines drawing invariably contains far more information than you need, so you will only import a few of the stations provided in the lines drawing. Beginning at the extended transom, import every other station. Additionally, import station 2 (station 0 is the tip of the bow). You will need the additional shape provided by this station for lofting the bow.

To import the stations, first place each in its proper location in the model, **Hull Lines.3dm**. To do this, in **Hull Lines.3dm**, copy each station from the body plan to its proper location. Select a station line from the body plan and **copy** it using the **End** object snap. Use the **End** object snap to the end of the profile grid line at the body plan and the end of the appropriate grid line in the profile.

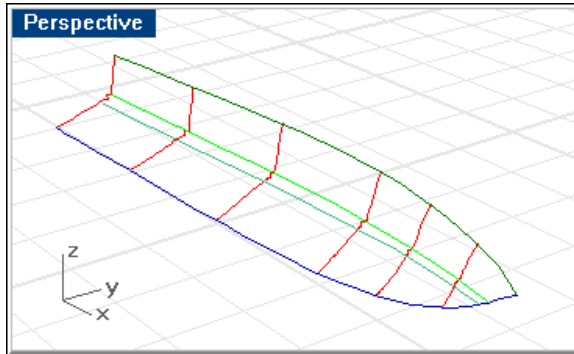
The station will be copied to its proper location along the profile view.

After you have copied each required station (1, 2, 3, 5, 7, and the transom), to its proper location, the model should resemble the image below.



Copy stations to their locations.

Copy all the located stations to the Clipboard (**Ctrl+C**). Return to **Offshore.3dm** and paste the stations (**Ctrl+V**). Change these stations to layer **cStas**. Your model should resemble the image below.

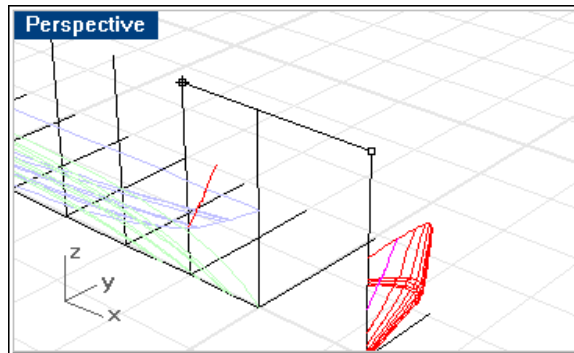


Stations pasted into new model.

Step-by-Step: Import the stations

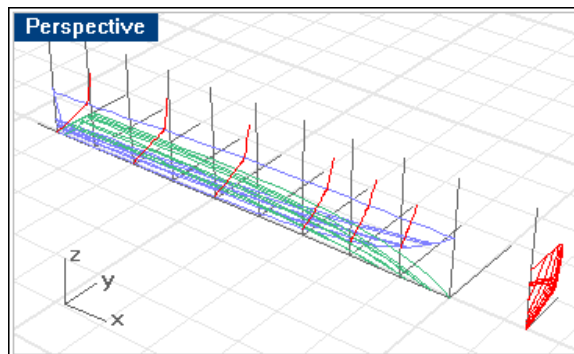
Place the stations in their proper locations in the lines drawing

- 1 In **Hull Lines.3dm**, select station 1.
- 2 Start the **Copy** command.
- 3 At the **Point to copy from ...** prompt, use the **End** object snap to pick the end of the grid line at the body plan location.



Copy station 1 into position.

- 4 At the **Point to copy to** prompt, use the **End** object snap to pick the end of the grid line at the location for station 1.
- 5 Repeat these steps for stations 2, 3, 5, 7, and the transom.



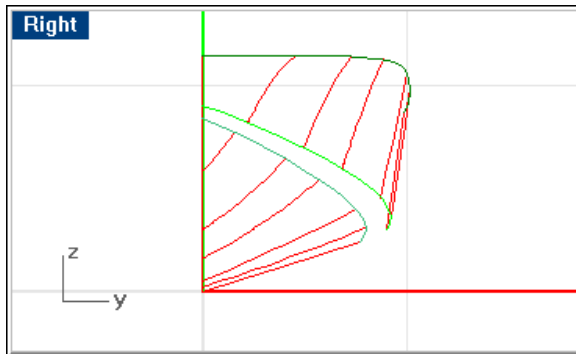
The copied stations.

Copy the curves to your working model

- 1 Select the curves you just created (transom and stations 1, 2, 3, 5, and 7).
- 2 Press **Ctrl+C** to copy these curves to the Windows Clipboard.
- 3 In **Offshore.3dm**, press **Ctrl+V** to paste the stations into the current model.
- 4 Change the stations you just pasted to layer **cStas**.
- 5 **Save** your work.

Prepare the station curves

Before lofting the side and bottom, the station curves must be broken up into bottom and side and rebuilt to ensure good geometry. To do this, **trim** all stations, removing the horizontal and vertical segments between the upper and lower chine as shown in the illustration below. After this is done, each station should consist of one continuous curve from sheer to upper chine and another from lower chine to rabbet.

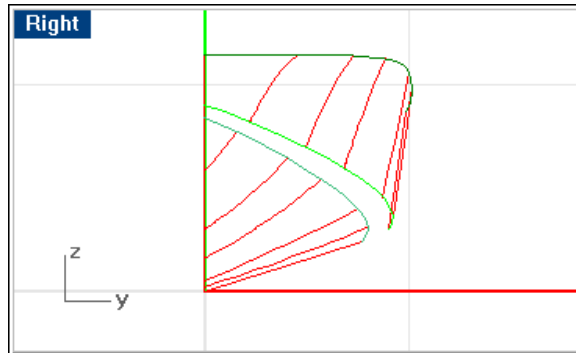


Trim the stations to upper and lower chines.

Step-by-Step: Prepare the station curves

Trim the station curves to the chines

- 1 Start the **Trim** command.
- 2 At the **Select cutting objects ...** prompt, select the upper and lower chine as shown in the image below.



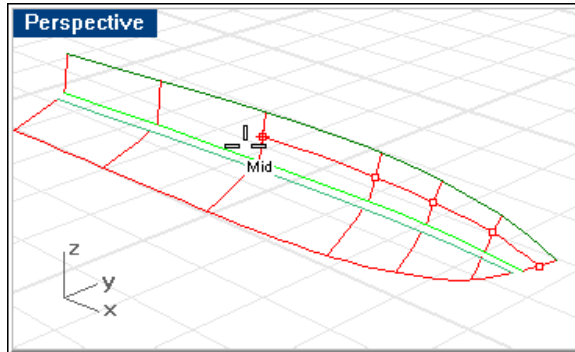
Trim the stations to upper and lower chines.

- 3 At the **Select object to trim ...** prompt, trim all stations to these two chines.

Define the side panel shape

To define the shape of the side panel, use the **InterpCrv** command to draw a curve on layer **cStas** through the midpoints of all the side stations, including the stem as shown in the image below.

Rebuild and fair this curve, keeping the shapes of the sheer and chine in mind.

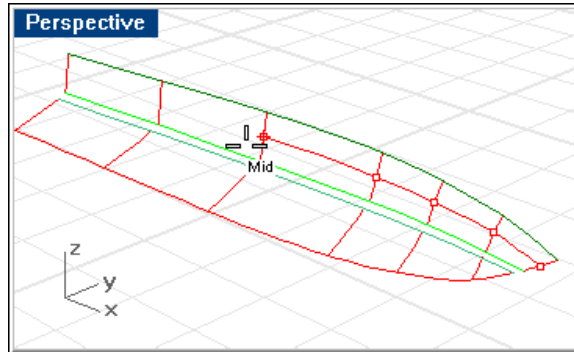


Draw the midgirth side curve to help define the side panel.

Step-by-Step: Define the side panel shape

Create the midgirth side curve

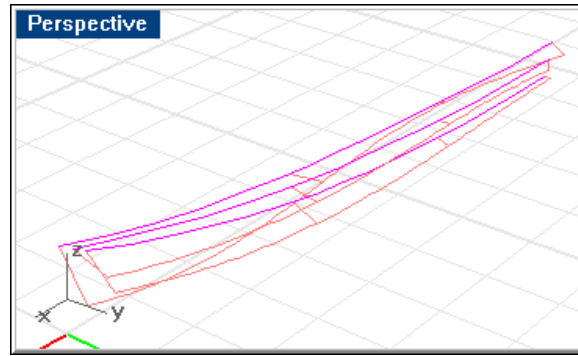
- 1 Make the layer **cStas** current.
- 2 Start the **InterpCrv** command.
- 3 At the **Start of curve ...** prompt, use the **Mid** object snap to pick the midpoint of the stem.
- 4 At the **Next point of curve ...** prompt, use the **Mid** object snap to pick the midpoint of each side station, in order, from station 1 to the stern.



Draw the midgirth side curve to help define the side panel.

Rebuild the side curves

- 1 Select the sheer, midgirth, and chine curves.
- 2 Start the **Rebuild** command.
- 3 Complete the **Rebuild Curve** dialog box specifying **6** points, degree **5**.
- 4 Use the curve fairing techniques to ensure that the longitudinal curves are fair.



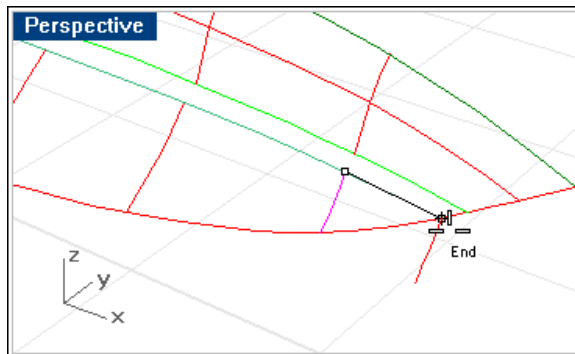
Fair the longitudinal curves.

Prepare the bottom stations

Rebuild the bottom station curves using **4** points per curve and degree **3**. Inspect the results to be sure they did not stray too far from the original geometry.

Because Rhino does not loft surfaces with pointed ends well, you need to make the bottom surface with a trapezoidal shape and trim it to a point at the end. Once the surface is lofted, it will be trimmed back to the rabbet.

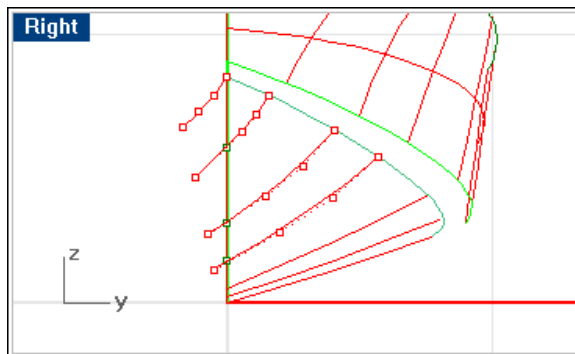
To do this, **copy** the forward most bottom station to the forward end of the chine to create a "phantom station."



Copy the station forward.

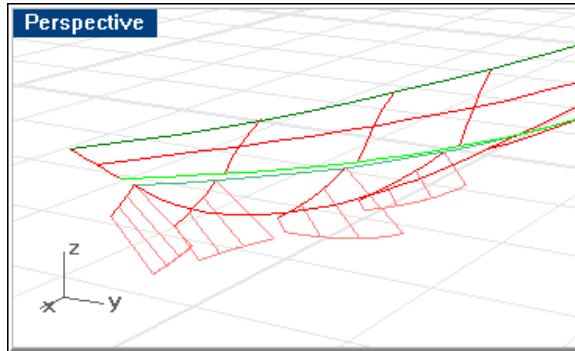
Drag the end control points of the "phantom station," the forward-most bottom station, and perhaps the next station back. See the **discussion** on hull fairing for instructions on this technique.

This extends the curves beyond the centerline and helps create a more "rectangular" hull surface.



Drag the end control points of the station curves to extend the curves beyond the centerline.

Check all of the curves for fairness.



Check the curvature graph of the curves.

Step-by-Step: Prepare the bottom stations

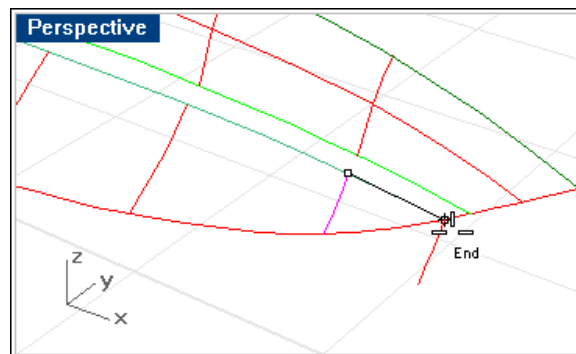
Rebuild the station curves

- 1 Select the bottom station curves.
- 2 Start the **Rebuild** command.
- 3 In the **Rebuild Curve** dialog box, set the **Point count** to **4** points and the **Degree** to **3**.

Inspect the results to be sure they did not stray too far from the original geometry.

Copy the forward station

- 1 Select the forward station.
- 2 Start the **Copy** command.
- 3 At the **Point to copy from ...** prompt, use the **End** object snap to pick the end of the curve at the bottom chine.
- 4 At the **Point to copy to** prompt, use the **End** object snap to pick the end of the bottom chine curve.



Copy the station forward.

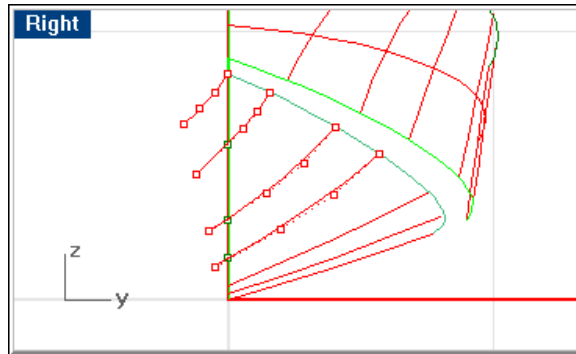
Extend the station lines

- 1 Use the **Point** command to place points at the intersections of the stations and the stem line.

This point will serve as a reference when we extend the curves.

- 2 Use the **PointsOn** command to turn on the control points of the station lines, and in the **Right** viewport, drag each station beyond centerline.

Make sure it stays intersecting the point on the stem.

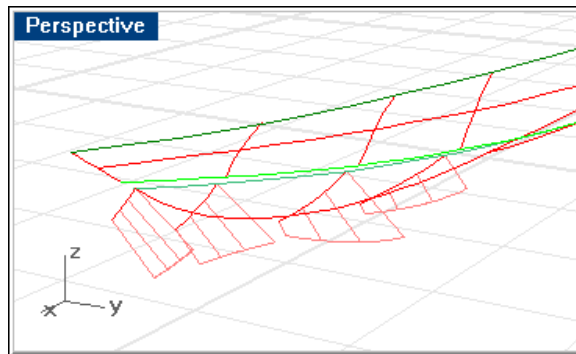


Drag the end control points of the station curves to extend the curves beyond the centerline.

See the **discussion** on hull fairing for instructions on this technique.

Check the curves for fairness

- 1 Select all the extended curves.
- 2 Use the **CurvatureGraph** command to check all of the curves for fairness.
- 3 **Rebuild** the curves and **move** or **nudge** the control points to adjust the curves as necessary.



Check the curvature graph of the curves.

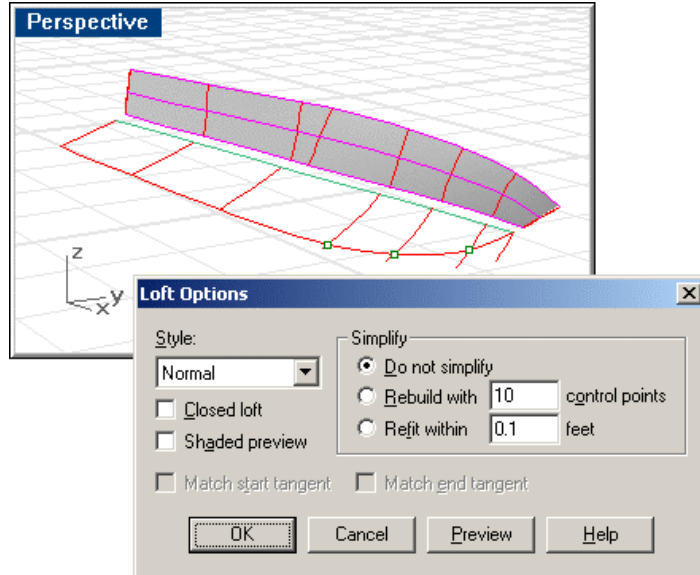
Loft the hull surfaces

Now that you have created a set of curves that define the side and bottom hull shapes, you can create the hull surfaces. Before you start, make the **sHull** layer current.

For the side, select the sheer, midgirth line, and the upper chine, and start the **Loft** command.

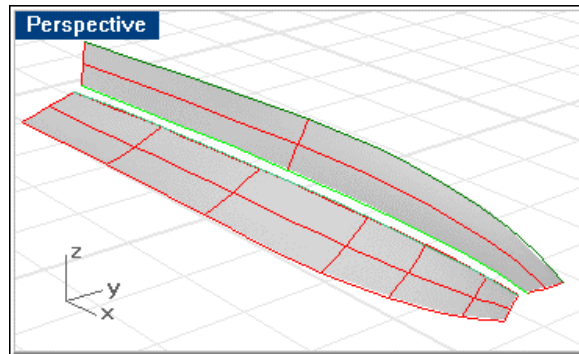
In the **Loft Options** dialog box, you have several choices to make. Select a style (the options are **Normal**, **Loose**, **Tight**, **Straight Sections**, and **Developable**) to tell Rhino the type of control you want to exert over the lofted surface. In our case, you will select **Normal**, since you have no particular constraints to maintain. This completes the side surface.

If this were a hull for which you intended to unroll the surfaces for CNC cutting, you would select **Developable**. If you have a particular set of curves that you were required to maintain on the surface (for example, if the curves were measured from an existing part), select **Tight**. If you know the curves are approximate and only intended to give the loft command guidance, select **Loose**. If this is an HVAC duct that you intend to unroll to prepare flat patterns, you select **Straight Sections**.



Loft the side surface.

The bottom is being lofted differently to accommodate the pointed end at the bow. For this surface, select all the bottom stations, including the phantom station.



Lofted surfaces.

Notice how simple the geometry is. For the bottom surface, there is one isocurve in the **U** direction one for each station in the **V** direction. The side surface is even simpler, since there were only three curves selected to perform the loft. There is only one isocurve in each direction and one at each edge.

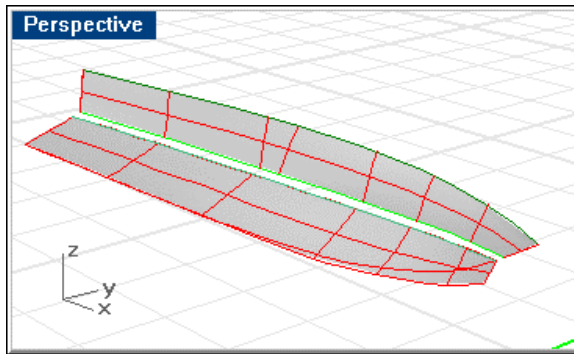
The reason this geometry is so simple is that you rebuilt your curves before you performed the loft. The simplicity of these surfaces is going to make it easy to fair this hull. Each time you move a point, the isocurve corresponding to the nearest station will change. Since, on the bottom, you have the station curves "behind" the isocurves, you can see how much you are deviating from your preliminary stations.

Note: The side could have been lofted the same way as the bottom, but was done as it was to demonstrate how easily a surface could be constructed from only three curves.

Step-by-Step: Loft the hull surface

Loft the surfaces

- 1 Make the **cHull** layer current.
- 2 select the sheer and upper chine and the midgirth curve.
- 3 Start the **Loft** command.
- 4 In the **Loft Options** dialog box under **Style**, select **Normal**.
- 5 Select all the bottom stations.
- 6 Repeat the **Loft** command.
- 7 In the **Loft Options** dialog box, click **OK**.



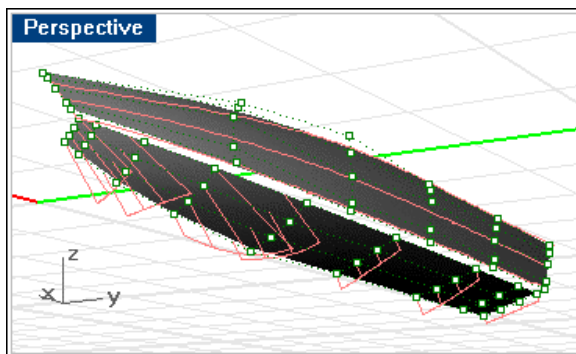
Lofted surfaces.

- 8 **Save** your work.

Adjust the surface

At this point, you should look at how fair your surfaces are. Look at a magnification of the curvature of each isocurve and analyzing how it looks relative to its neighbors. If the isocurves do what you expect and progress from one station to the next in a gradual transition, then you can assume that the surface is reasonably fair.

For now, select the surfaces and turn on **curvature graph** and **points**. In the image below, you can see the curvature graph for the surfaces on the outside of the hull.



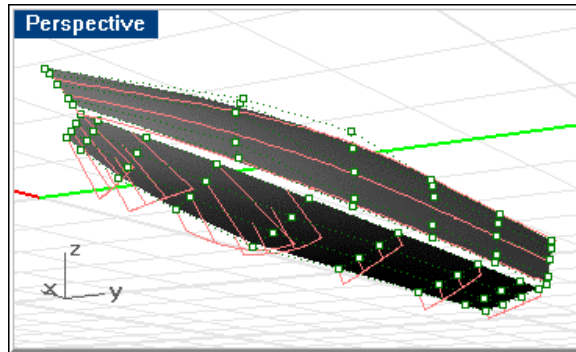
Surface control points and curvature graph.

This image shows the isocurves on the model are quite fair. You will not need to move any control points to achieve fairness in this model.

Step-by-Step: Adjust the surface control points

Turn the curvature graph on

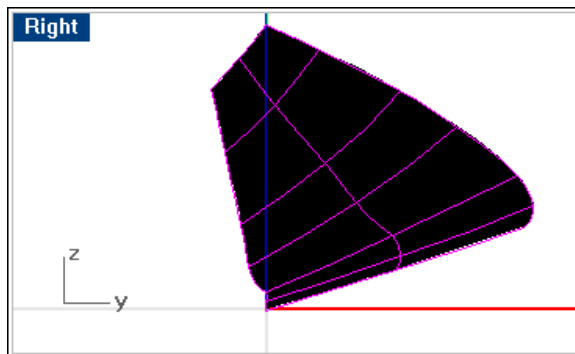
- 1 Select the lofted surfaces.
- 2 Use the **CurvatureGraph** command to display the curvature graph.
- 3 Use the **PointsOn** command to display the control points of the surfaces.



Surface curvature graph.

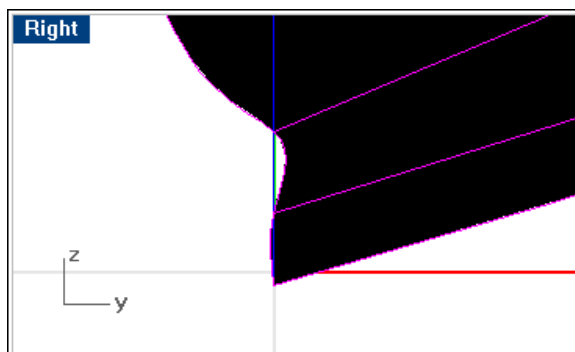
Modify the bottom surface

Since you took some liberty earlier in causing your hull to cross centerline into negative space, you should inspect the results.



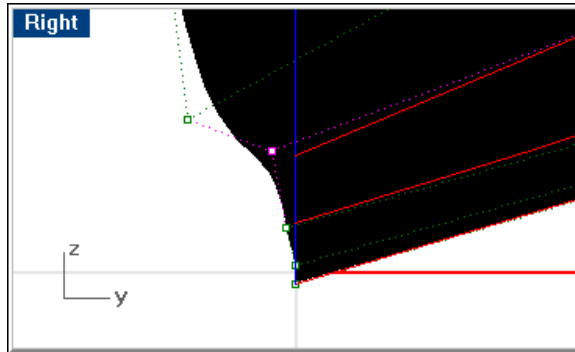
Surface in right viewport.

If you zoom in on the **Right** viewport, you may see that the inboard edge of the lofted bottom surface has a tendency to cross the centerline.



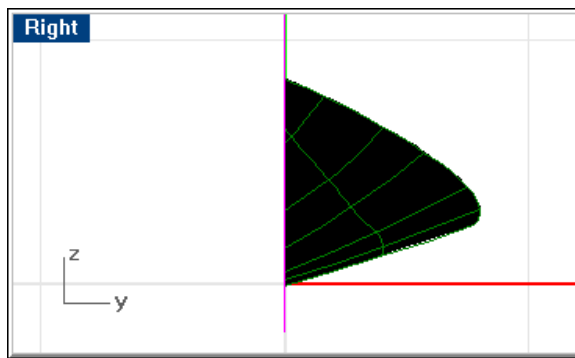
Surface crosses centerline.

To fix this, edit control points in this area to cause the entire surface to cross the centerline.



Surface problem solved.

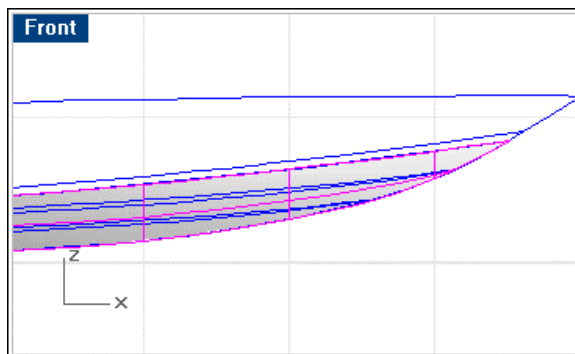
Trim the surface to a vertical centerline in the **Right** viewport.



The trimmed hull bottom surface.

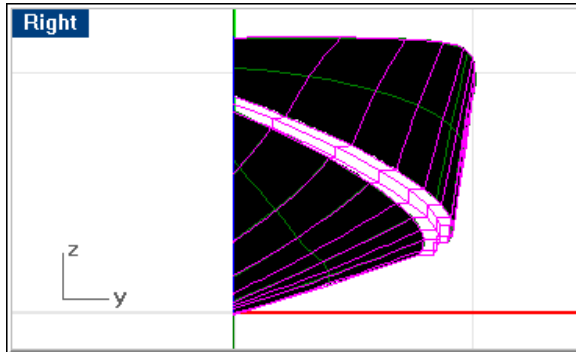
The inboard edge of the hull surface should resemble the forefoot from the designer's lines drawing. Since you did not use the stem and forefoot in your controlling geometry, how close will the resemblance be?

As you can see from the image below, the trimmed surface is almost a dead match for the forefoot that came from the designer's lines. You can judge whether the surface needs any further modification.



The trimmed hull.

However, what about the body plan? How closely do your lofted surfaces fit the stations you inherited from the lines drawing in the beginning of this exercise?



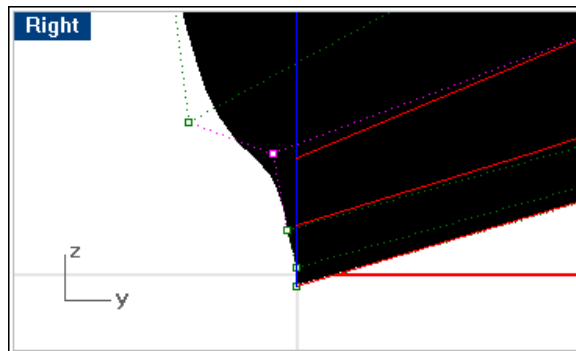
Lofted surface isocurves and body plan stations.

As you can see, we have made only minor modifications to the surface. The original intent of the stations is achieved.

Step-by Step: Fix the forefoot

Move the control points

- 1 Select the bottom surface.
- 2 Turn on the control points with the **PointsOn** command.
- 3 In the **Right** viewport, move control points until the inboard edge of the bottom surface crosses the centerline.



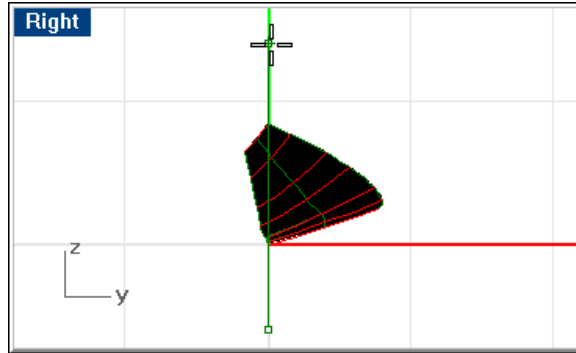
Move surface control points to make the surface cross the centerline.

Keep your eye on the stations (in other viewports) to be sure the isocurves continue to match them as you modify the surface.

- 4 Press **Esc** to turn off the control points when you have successfully nudged the inboard edge into negative space.

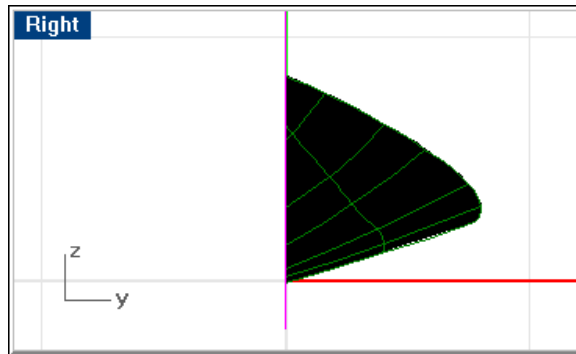
Trim to the rabbet

- 1 Start the **Line** command.
- 2 At the **Start of line ...** prompt, type **0,-3**.
- 3 At the **End of line ...** prompt, pick a point in the **Right** viewport above the tip of the sheer.



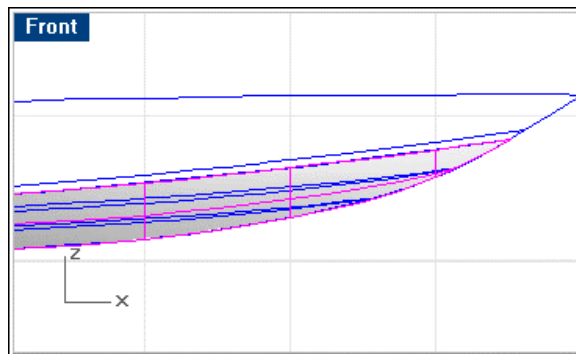
Draw a vertical trimming line.

- 4 Select the vertical line.
- 5 Start the **Trim** command.
- 6 At the **Select object to trim ...** prompt, select the bottom surface inboard of the cutting edge.

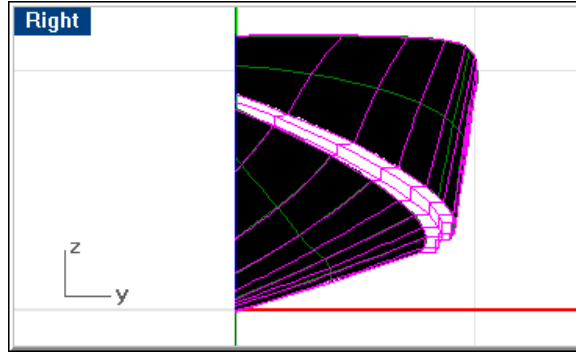


The trimmed bottom surface.

- 7 Check your work against lines in the profile and body plan.



Profile lines and hull bottom.



Lofted surface isocurves and body plan stations.

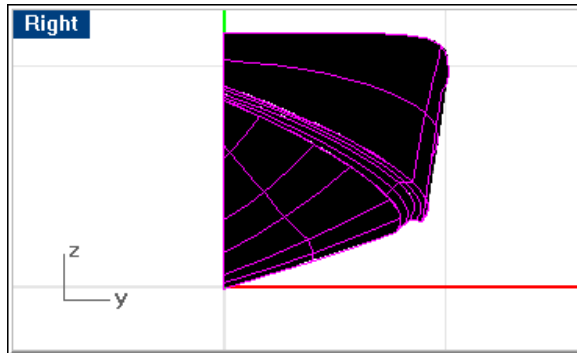
- 8 **Save** your work.

Construct the chine strakes

Now that you have built the side and bottom surfaces, adding the three surfaces that make up the chine is simple. Copy from **Hull Lines.3dm** the 2-D curves that represent the vertical surface between upper and lower chine strakes, use the **Crv2Views** command to create the 3-D curves. **Rebuild** the curves. Use the **EdgeSrf** command to create a surface between the two curves that define the vertical surface. Then use the **EdgeSrf** command to create surfaces between the edges of the vertical surface and the edges of the hull side and bottom.

Note: If you need help sorting out how the chine strakes are configured, refer to **Offshore.3dm**. The three strakes go from a right angle at the transom to what looks like a single surface at the stem.

The result should resemble the image below.



Chine strakes added.

Step-by-Step: Construct the chine strakes

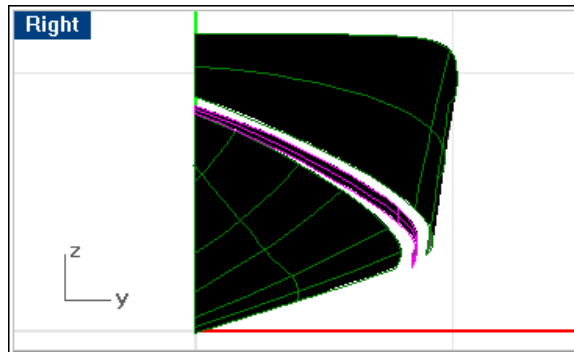
Create the 3-D strake curves

- 1 From **Hull Lines.3dm** import the 2-D curves for the vertical surface between the upper and lower chine strakes.
There should be one plan view curve for both edges of the vertical surface and a profile curve for the top and the bottom of the vertical surface.
- 2 Select the two curves that represent the bottom of the vertical surface.
- 3 Start the **Crv2View** command.
- 4 Repeat this step for the upper edge of the vertical surface.

- 5 **Delete** the 2-D curves.
- 6 Select the 3-D curves resulting from the **Crv2View** command.
- 7 Use the **Rebuild** command to change the **Degree** to **5** and the **Point count** to **6**.

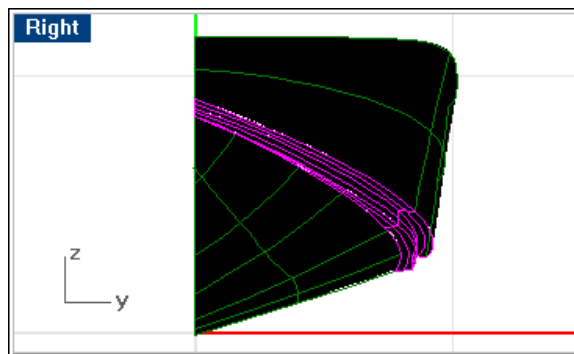
Create the strake surfaces

- 1 Select the lower and upper curves that define the edges of the vertical surface.
- 2 Use the **EdgeSrf** command to create the surface.



The vertical surface.

- 3 Select the lower edge of the surface you just created and the outboard edge of the bottom surface.
- 4 Repeat the **EdgeSrf** command.
- 5 Select the upper edge of the vertical surface and the bottom edge of the side surface.
- 6 Repeat the **EdgeSrf** command.

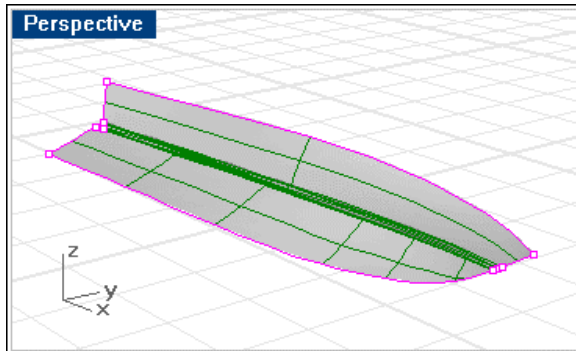


The strake surfaces.

Join the hull and check for unjoined edges

Now that you have all the surfaces needed to form one side of the boat, **join** them all together and check for unjoined edges with the **ShowEdges** command. Unjoined edges prevent analyses such as hydrostatics and Boolean operations and will make exporting to third-party software, such as solid modeling, programs impossible.

The result of the **Join** command should be a single polysurface consisting of five surfaces. The result of the **ShowEdges** command should be a continuous line around the outside edge of the model. There should be no unjoined edges in the interior of the model.



Unjoined edges on hull polysurface.

To fix any unjoined edges found, use the **ExtractSrf** command to detach two or three adjacent surfaces from the hull polysurface, use the **MatchSrf** command to match the surface edges to each other, and **join** the resulting surfaces to the hull polysurface.

Note: Because the hull bottom was trimmed, it must be **untrimmed** before fixing any edges.

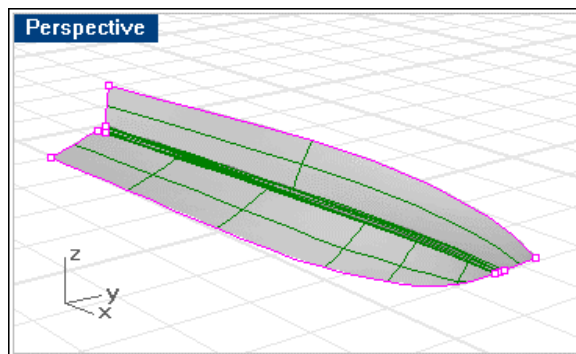
Step-by-Step: Join the hull and check for unjoined surfaces

Join the surfaces and check

- 1 Select the side, bottom, and three chine strakes.
- 2 Use the **Join** command to join the three surfaces into one polysurface.
- 3 With the resulting surface selected, use the **ShowEdges** command to check for unjoined surfaces.

Your model should resemble the image below.

The unjoined edges are highlighted. The only unjoined edge should be the one around the outside edge of the hull polysurface, not any interior surface edges.



Unjoined edges on hull polysurface.

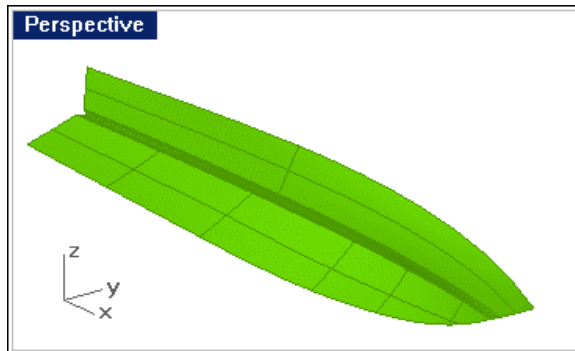
- 4 **Save** your work.

Analyze the surfaces

Now that you have faired and joined the surfaces that make up the hull, let us look into the Rhino surface curvature analysis tools.

For this process, use the **EMap** and **CurvatureAnalysis** commands. With these commands, small problems with the hull that do not ordinarily show up when shading or rendering become more pronounced. It is like turning off all the lights and standing beside the hull with a lamp on at the far end of the shop. Small ripples and flaws become much more pronounced.

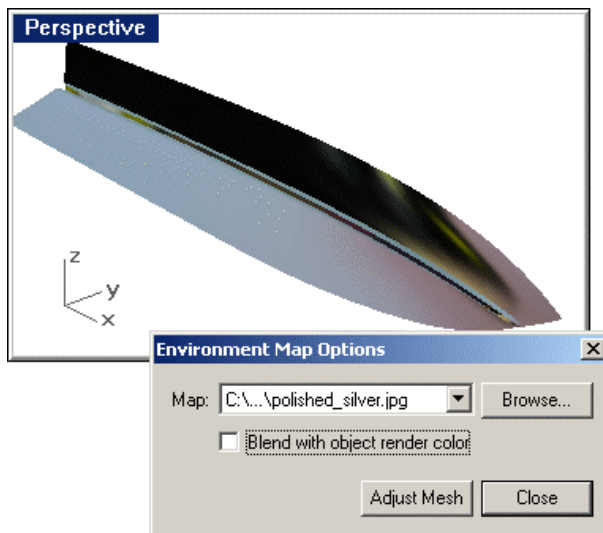
In the image below, you get a pleasant look at the hull. If you want to see just the surfaces, simply shade the view. It can be advantageous to leave the shading and wireframe displayed all the time so that you can tell where the nearest isocurve to a problem lies.



Model with shaded wireframe. No unfairness is evident.

Environment map analysis

The shaded model does not always tell you about subtle problems in the surface. The **EMap** command can be a highly effective way of finding flaws that were not apparent with the curvature graphs used earlier to analyze the surface isocurves. Flaws can be even more obvious when the model is rotated with the environment map displayed. The image below shows the hull with an environment map applied. This image used polished silver as its material. Rhino comes with various materials, which you can select in the **Environment Map Options** dialog box.

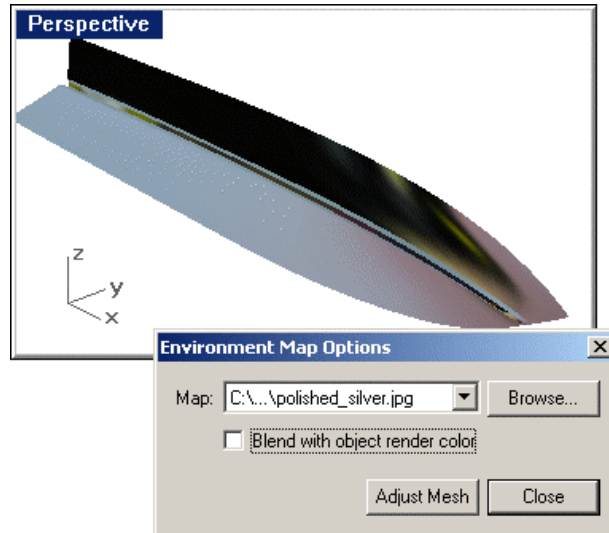


Environment map.

Step-by-Step: Environment map analysis

To display the environment map (EMap)

- 1 In the **Perspective** viewport, select the hull polysurface.
- 2 Start the **EMap** command.



Environment map.

The environment map will be calculated and then rendered in the viewport. The first time you use the **EMap** command in a model, it will take longer, since Rhino uses a specialized mesh created from the surface under consideration.

- 3 Rotate the view to examine the surface from all angles, looking for shadows that change directions in unexpected ways during the rotation.
- 4 Close the **Environment Map Options** dialog box.

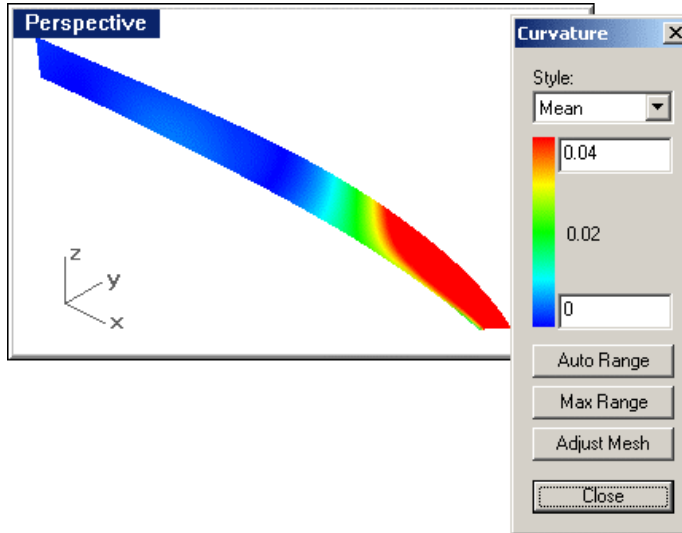
Curvature analysis

The environment map display cannot always detect flaws in the surfaces. Sometimes an even more powerful analysis tool is required. The **CurvatureAnalysis** command shows changes in curvature over the surface being analyzed. If spots of color or rapid changes from one color to another are evident, the area where these are found requires attention.

By alternately displaying the curvature analysis and editing control points, you can remove any flaws detected. As you do, it is a good idea to keep an eye on the **Right** viewport to be certain you are not introducing any unwanted characteristics to your body plan.

To use this method, you must explode the polysurface, since control points cannot be edited on polysurfaces. When you do this, there is a risk of making the surfaces so they cannot be rejoined.

The image below shows the results of the **CurvatureAnalysis** command using the **Mean** style. Here you can see there are very gradual changes from areas of high curvature (the flared bow, signified in red and green) to areas of little or no curvature (the blue areas, aft of midship). Inspect these areas on your model carefully. Any place where the curvature jumps quickly from one color to another or has spots of a particular color needs some attention.



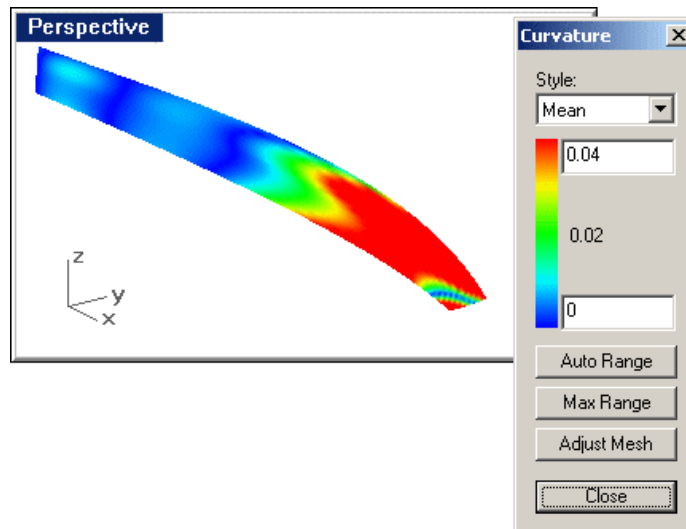
Mean surface curvature analysis display.

Step-by-Step: Curvature analysis

Curvature analysis

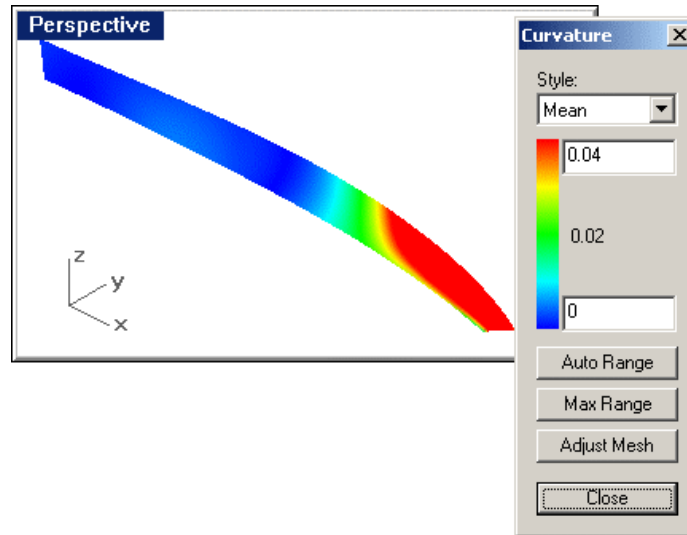
- 1 Select the hull polysurface.
- 2 **Explode** the polysurface into individual surfaces.
- 3 In the **Perspective** viewport, select a hull surface.
- 4 Start the **CurvatureAnalysis** command.

The curvature will be calculated and rendered in the viewport.



Curvature graph analysis.

- 5 Rotate the view to look at it from all angles.
Look for spots of color or rapid changes from one color to another.
- 6 Close the **Curvature** dialog box.
- 7 Edit control points as required to repair unfair areas.
- 8 Repeat the analysis until satisfactory results are obtained.



The adjusted surface.

Rejoin the hull surfaces and check

- 1 Select the four hull surfaces.
- 2 **Join** the surfaces.
- 3 With the joined hull surface selected, use the **ShowEdges** command to check for unjoined edges.

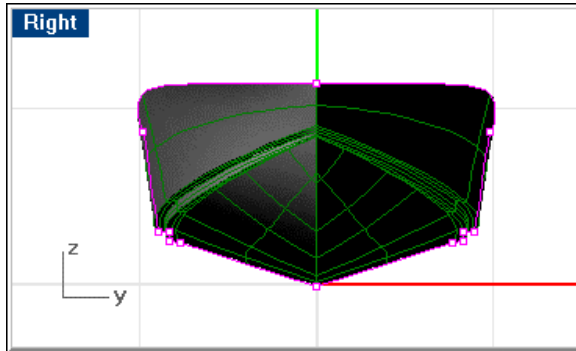
Create the transom

Now that you have seen how easy the hull surface was to build and fair, it is time to try something even easier. You have one side of the hull that extends continuously from stem to stern and from sheer to rabbet. You need to cut in a transom. Then, to calculate hydrostatics and make the model suitable for export to third-party software, you need to close the hull into a solid.

Mirror the hull around the centerline.

Join the two sides to create one continuous polysurface. This polysurface's only unjoined edges should be the sheer and transom.

Use the **ShowEdges** command. The result should resemble the image below.



Unjoined edges after joining two halves.

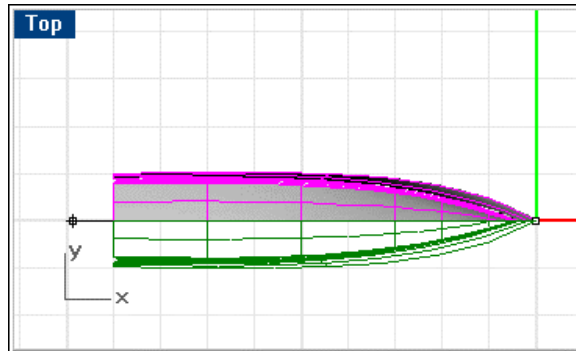
If there are unjoined edges, **explode** the model, **delete** the starboard side, **untrim** the bottom surface, and **extend** all four surfaces across centerline. **Trim** the surfaces to the centerline and **join**.

In the image, the curve highlighted shows the only naked edges in the model. If there are unjoined edges along the rabbet, work needs to be done. It is most likely because proper care was not taken to ensure that the hull surface completely crossed the rabbet before trimming. You will need to go back to that step and rework the inboard edge before proceeding. It will be imperative that we have a model with no unjoined edges when we finish.

Step-by-Step: Mirror and join the hull surface

Mirror the hull

- 1 Select the hull polysurface.
- 2 Start the **Mirror** command.
- 3 At the **Start of mirror plane** prompt, in the **Top** viewport, type **0**.
- 4 At the **End of mirror plane** prompt, use **Ortho** to drag the mirror plane line along the x-axis.

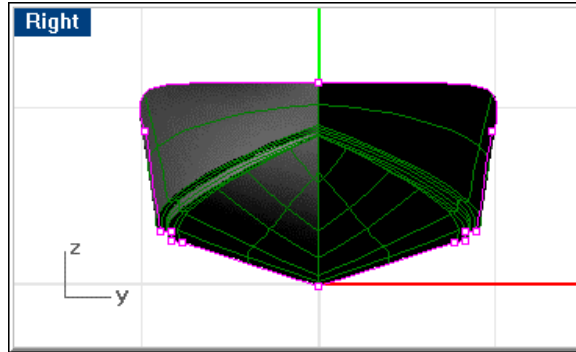


Mirror the hull halves.

Join the hull halves and check

- 1 Select the two hull halves.
- 2 Use the **Join** command to join the two sides to create one polysurface.
The only unjoined edges should be the sheer and transom area.

- Use the **ShowEdges** command.
The result should resemble the image below.



Unjoined edges after joining two halves.

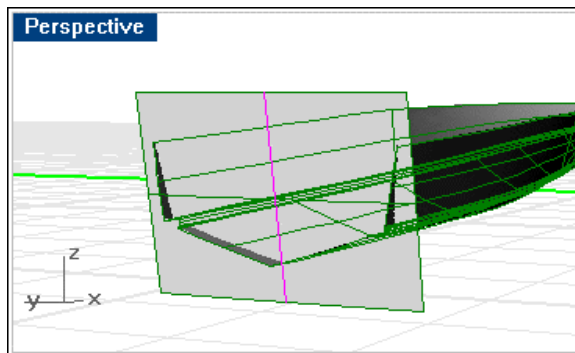
Build the transom

Construct the transom with a rectangular planar surface made from the transom profile curve imported from the lines drawing. Trim the hull to it. Construct a camber curve to define the top of the transom. Trim the transom to the hull and camber curve.

In the image below, the centerline curve was extruded to create a planar surface that will become the transom.

Extend the transom profile curve above and below the hull, to ensure success in the trimming operations that follow.

Extrude the extended centerline with the **BothSides** option to get a continuous planar surface from outboard the port side to outboard the starboard side as shown.



Extruded transom profile curve.

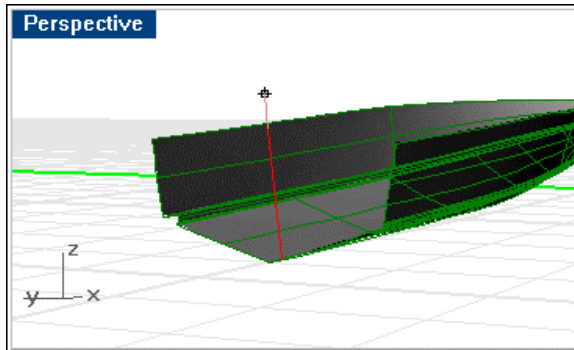
Using the resulting surface, **trim** the hull polysurface. This will give the hull its final shape.

Step-by-Step: Build the transom plane

Extend the transom profile

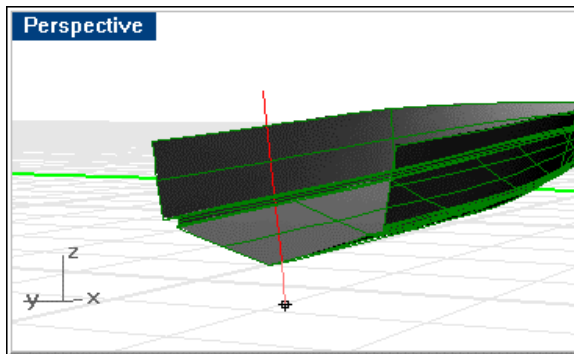
- From **Hull Lines.3dm**, copy the transom centerline profile line to your model.
- Use the **ChangeLayer** command to move this curve to the **cXsomProf** layer.
- Make layer **cHull** current.
- Start the **Extend** command.

- 5 At the **Select boundary objects or enter extension length. Press Enter for dynamic extend** prompt, press **Enter**.
- 6 At the **Select object to extend ...** prompt, pick one end of the transom profile line.



Extend the transom profile line up.

- 7 At the **End of extension** prompt, drag the line until it is long enough to clear the sheer.
- 8 At the next **Select object to extend ...** prompt, pick the other end of the transom profile line.
- 9 At the **End of extension** prompt, drag the line until it is long enough to clear the keel line.

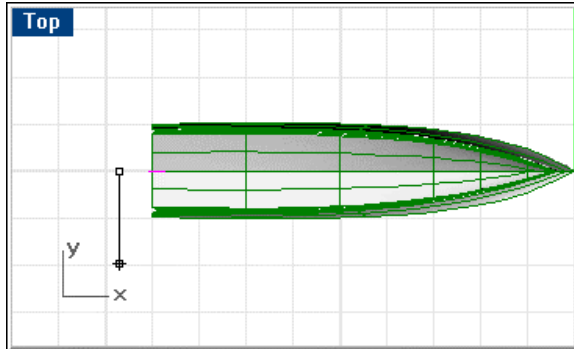


Extend the transom profile line down.

Extrude the transom profile

- 1 Select the extended transom profile curve.
- 2 Start the **Extrude** command.
- 3 At the **Extrusion distance ...** prompt, type **B** to activate the **BothSides** option.
- 4 At the **Extrusion distance ...** prompt, type **D** to show the direction of the extrusion.

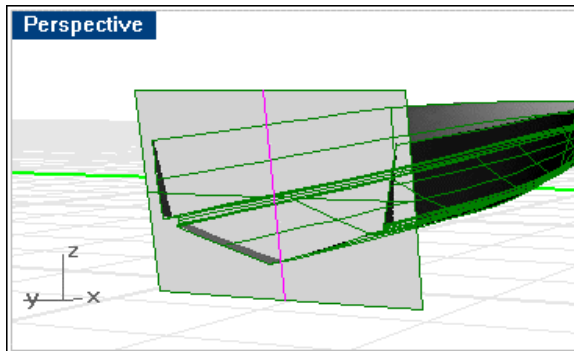
- 5 In the **Top** viewport, pick a point near centerline and use **Ortho** to pick another point directly outboard of the first.



Show the direction for the transom extrusion.

Since this is only an indication of direction, the distance between these two points is not important.

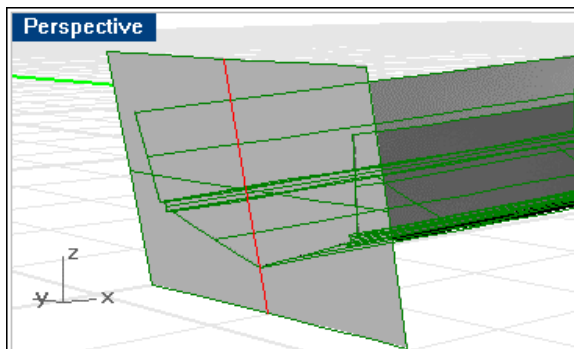
- 6 At the next **Extrusion distance** prompt, pick a point outboard of the hull so that the resulting extruded surface will be wider than the sheer.



The extruded transom surface.

Trim the hull

- 1 Select the transom surface.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, select the hull aft of the transom plane.



The hull trimmed to the transom.

- 4 **Save** your work.

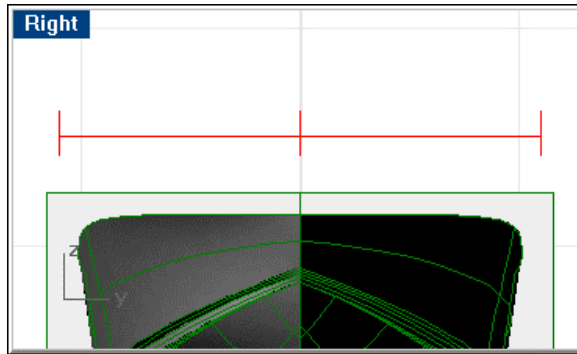
Construct the camber curve

You must create a curve on the transom from one side to the other that represents the intersection between the deck and transom. This curve will become the camber curve when you loft the deck.

Create construction lines

Draw a short vertical **line** about 12" long from the midpoint of the transom surface. In the **Right** viewport, move this line up so it is clear of the hull and **offset** it 66" to the port and starboard, similar to the illustration below. Then, draw a horizontal **line** between the lower endpoints of these two vertical lines.

In the **Right** viewport use **Ortho** to **move** the horizontal line you just created up an amount equal to the deck camber. A distance of five inches was used in the example.

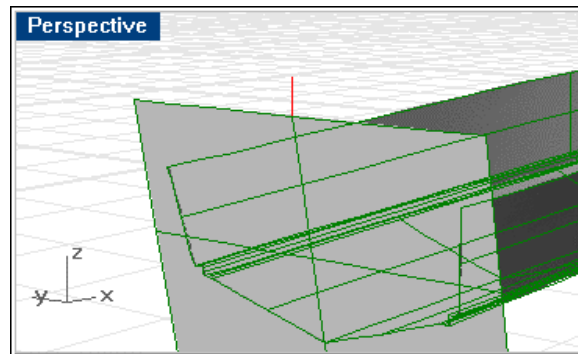


Construction lines for the camber curve.

Step-by-Step: Construct the construction lines

Draw a construction line

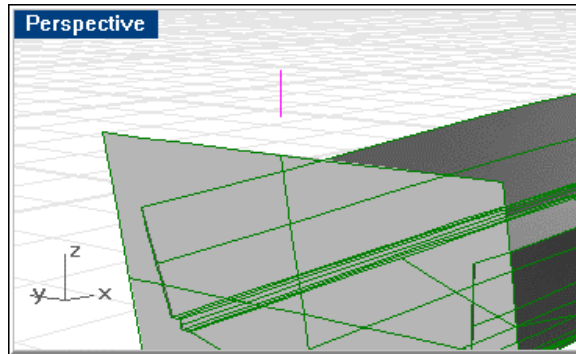
- 1 Make the **cTempDeckBms** layer current and turn on the **cCamber** layer.
- 2 Enable **Planar** and **Ortho**.
- 3 Start the **Line** command.
- 4 At the **Start of line ...** prompt, in the **Right** viewport, use the **Mid** object snap to pick the midpoint of the transom.
- 5 At the **End of line ...** prompt, pick a point approximately **12"** above the first.



Draw a vertical line from the midpoint of the transom.

Move the line

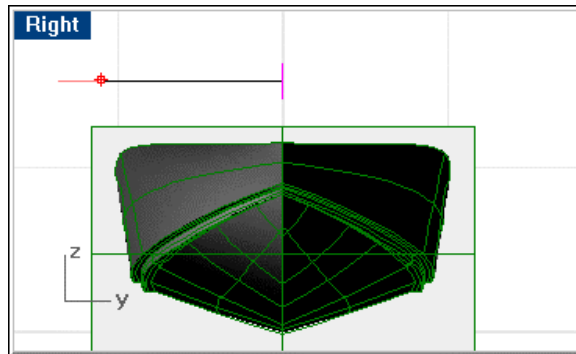
- 1 Select the vertical line you just created.
- 2 Start the **Move** command.
- 3 At the **Point to move from ...** prompt, pick in the right view.
- 4 At the **Point to move to** prompt, pick a point above the first so that the vertical line you just created is well clear of the hull as shown in the image below.



Move the vertical line up.

Offset the lines

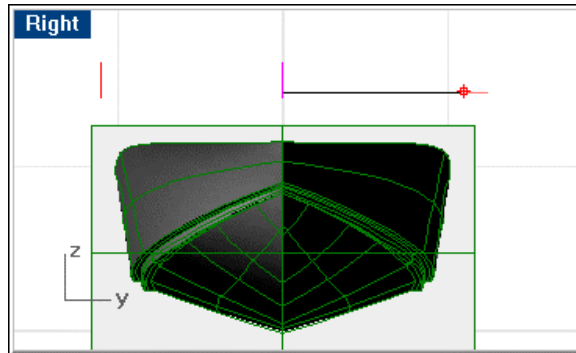
- 1 Select the vertical line you just created.
- 2 Start the **Offset** command.
- 3 At the **Select curve to offset ...** prompt, type **66"**.
- 4 At the **Side to offset ...** prompt, in the **Right** viewport, pick a point to starboard of centerline.



Offset the vertical line to starboard.

- 5 With the vertical line at the transom centerline still selected, repeat the **Offset** command.

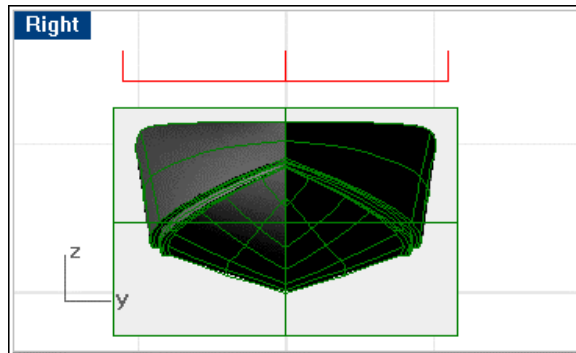
- 6 At the **Side to offset ...** prompt, in the **Right** viewport, pick a point to port of centerline.



Offset the vertical line to port.

Draw another construction line

- 1 Start the **Line** command.
- 2 At the **Start of line ...** prompt, use the **End** object snap to pick the lower end of the vertical line on the port side.
- 3 At the **End of line ...** prompt, use the **End** object snap to pick the lower end of the vertical line on the starboard side.

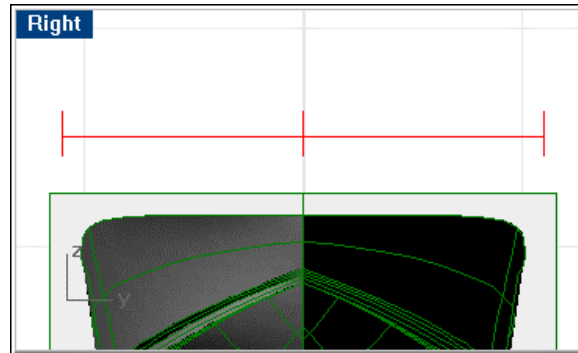


Draw a line between the bottom endpoints of the two offset lines.

Move the line

- 1 Select the line you just created.
- 2 Start the **Move** command.
- 3 At the **Point to move from ...** prompt, in the **Right** viewport, pick a point.

- At the **Point to move to** prompt, type **5"** and use **Ortho** to pick a point above the first point.

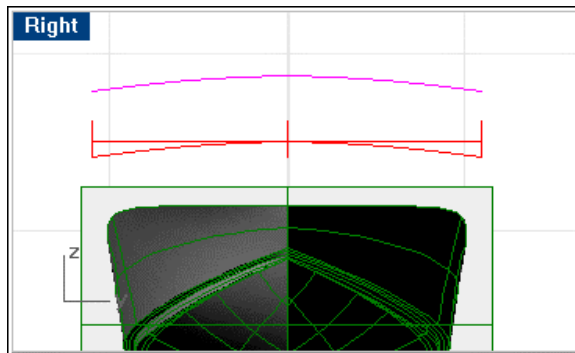


Move the horizontal line up 5 inches.

Draw the camber arc

To complete the camber curve, draw an **arc** from the lower endpoint of the starboard of the two vertical lines to the midpoint of the horizontal line and to the lower endpoint of the port vertical line. This will create a pleasing curve for our deck camber.

Make a **copy** of this arc to be used later for the deck.



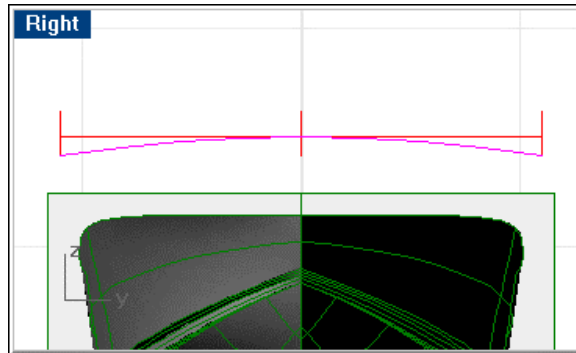
Draw the camber arc and copy up.

Step-by-Step: Draw the camber arc

Draw the arc

- Make the **cCamber** layer current.
- Start the **Arc3Pt** command.
- At the **Start of arc** prompt, use the **End** object snap to pick at the lower endpoint of the vertical line on the port side.
- At the **End of arc** prompt, use the **End** object snap to pick at the lower endpoint of the vertical line on the starboard side.

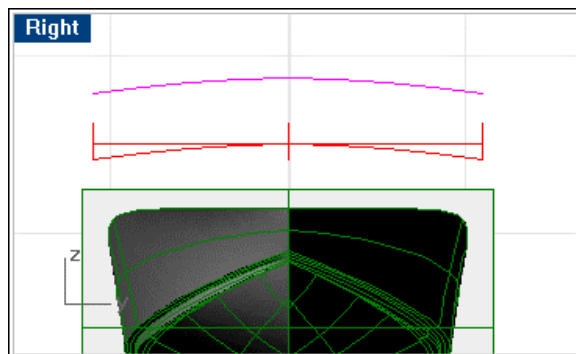
- 5 At the **Point on arc** prompt, use the **Mid** object snap to pick at the midpoint of the horizontal line.



Draw the camber arc from the vertical line lower endpoints and the moved line midpoint.

Make a copy of the camber curve

- 1 Select the arc.
- 2 Start the **Copy** command.
- 3 At the **Point to copy from ...** prompt, in the **Right** viewport, pick a point.
- 4 At the **Point to copy to** prompt, use Ortho to pick a point directly above the point selected for the origin.

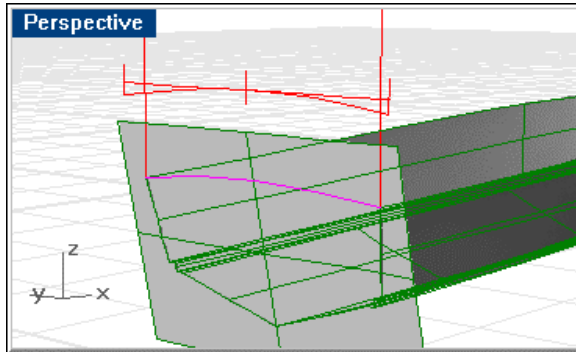


Copy the camber arc up.

Copy the camber arc into position

To place the camber curve accurately on the sheer at the transom, in **Right** viewport, draw a vertical **line** from each corner of the transom up to a point above the newly created camber curve.

Copy the camber curve. (Save one untrimmed camber curve to use at other stations later) **Trim** the copy to the two vertical lines and **move** the result to the corner of the transom.

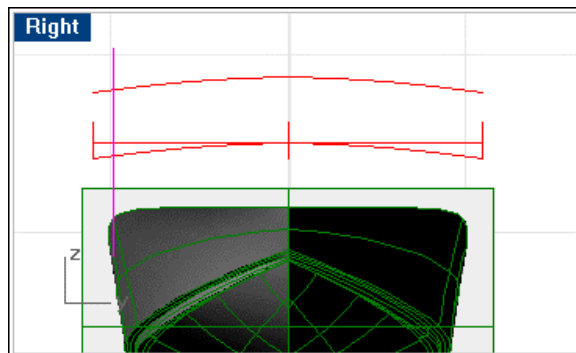


Copy the camber curve to the corner of the transom.

Step-by-Step: Copy the camber arc into position

Construct lines from the transom corners

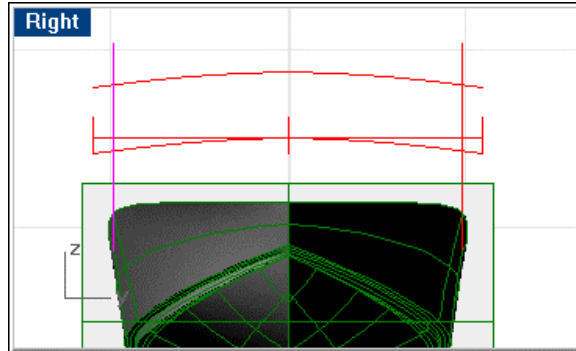
- 1 Start the **Line** command.
- 2 At the **Start of line ...** prompt, pick the starboard transom corner on the hull surface.
- 3 At the **End of line ...** prompt, in the **Right** viewport, pick a point above the copied arc.



Draw a vertical line from the starboard transom corner.

- 4 Select the vertical line.
- 5 Start the **Mirror** command.
- 6 At the **Start of mirror plane ...** prompt, type **0**.

- At the **End of mirror plane ...** prompt, in the **Right** viewport use **Ortho** to drag the mirror plane line up along the z-axis.

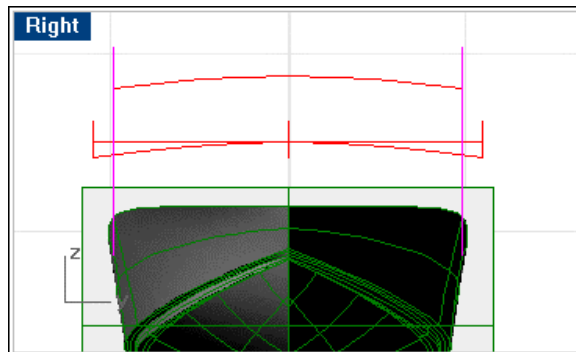


Mirror the vertical line to the port transom corner.

Trim the copy of the camber curve to the width of the transom at sheer

- Select the two vertical lines from the transom corners.
- Start the **Trim** command.
- At the **Select object to trim ...** prompt, in the **Right** viewport, pick the copy of the camber curve arc outboard of the transom corners on each side.

You should now have an arc whose chord length will fit the width of the sloped transom at the sheer.

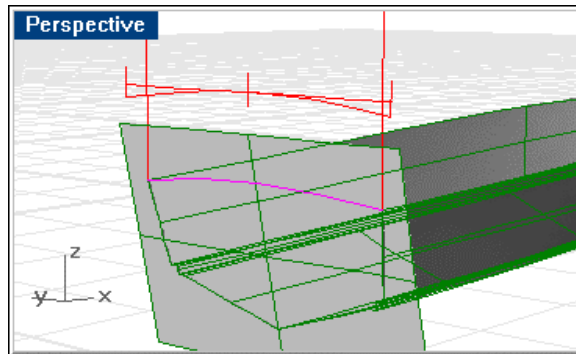


Trim the copy of the camber arc to the vertical lines.

Move the trimmed camber curve into place at the transom

- Select the copy of the trimmed camber curve.
- Start the **Move** command.
- At the **Point to move from ...** prompt, use the **End** object snap to pick the starboard endpoint of the trimmed camber curve.

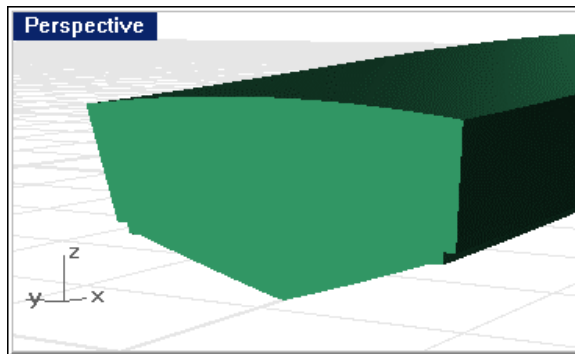
- 4 At the **Point to move to** prompt, use the **End** object snap to pick the end of the starboard transom corner.
You should now have a curve that ends at the intersection of sheer and transom on each side.



Trim the transom

Trim the transom with the hull sides and the camber curve.

You should now have a transom panel that matches the hull around its outer edges and matches the camber curve across the top.



The trimmed transom.

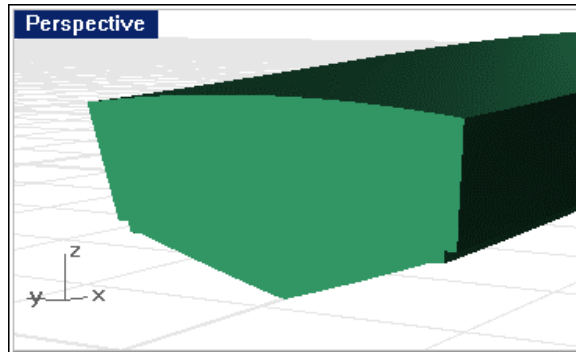
Step-by-Step: Trim the transom

Trim the transom top

- 1 Set the **sHull** layer current.
- 2 Be sure all surfaces are on layer **sHull**.
- 3 Select the transom camber curve and the hull on both sides.
- 4 Start the **Trim** command.

- At the **Select object to trim ...** prompt, select the transom outboard of the hull.

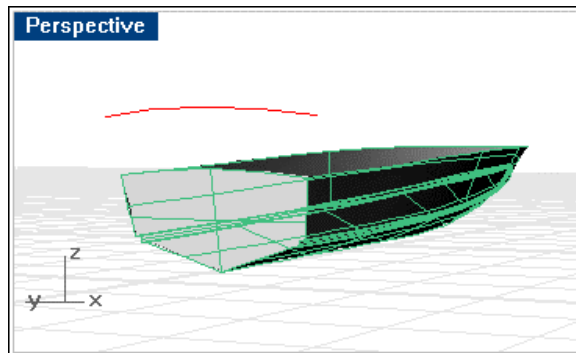
You should now have a transom panel that matches the hull around its outer edges and matches the camber curve across the top.



The trimmed transom.

Erase all the construction curves

- Delete** the construction lines created in this phase of the project except the original untrimmed camber arc.

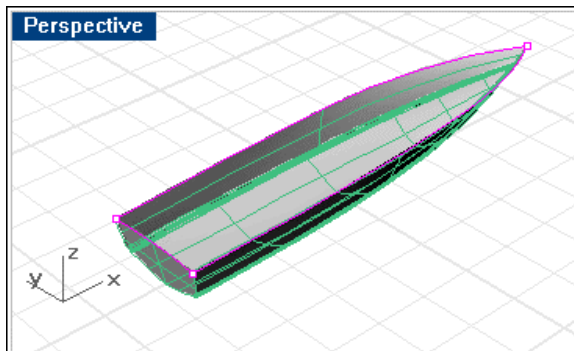


Save the untrimmed camber arc.

- Save** your work.

Join the transom to the hull

Join the transom to the hull and **check** for unjoined edges.

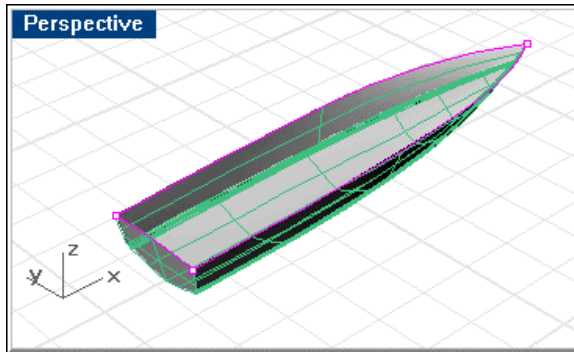


Unjoined edges after joining the transom to the hull.

Step-by-Step: To finish the hull and transom

Join the hull surfaces

- 1 Make layer **sHull** current and turn off all other layers.
- 2 Select all the surfaces.
- 3 Use the **Join** command to create a single polysurface.
- 4 Select the polysurface and use the **ShowEdges** command to check for unjoined surfaces.



Unjoined edges after joining the transom to the hull.

- 5 **Save** your work.

Create the deck

To complete the hull, you need to create a deck. This is where the spare camber curve comes in.

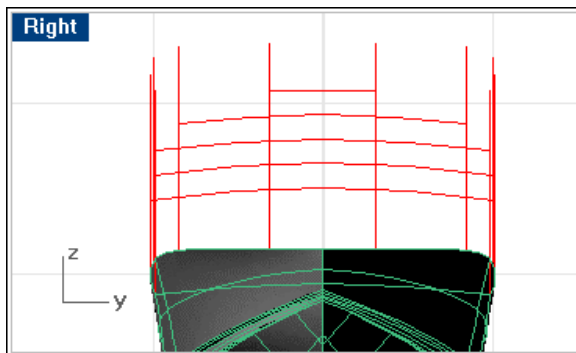
Create construction lines

At five (or more) locations along the sheer, create a vertical **line**. Draw one line and copy it however many times you need (there are four copies in the example) using **End** and **Near** object snaps.

Mirror the vertical "posts" across centerline.

Make one **Copy** of the camber curve for each pair of posts plus one (never leave yourself without an untrimmed camber curve).

Trim each camber curve to a pair of posts so that each camber curve's chord length matches the full breadth of the boat at one of the pairs of posts.

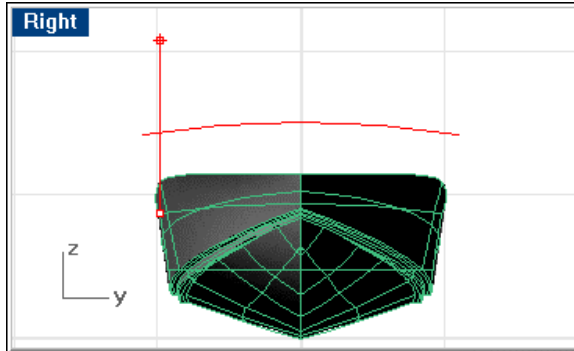


Create deck beams at various locations along the sheer.

Step-by-Step: Create construction lines

Create vertical lines to trim the camber curves to

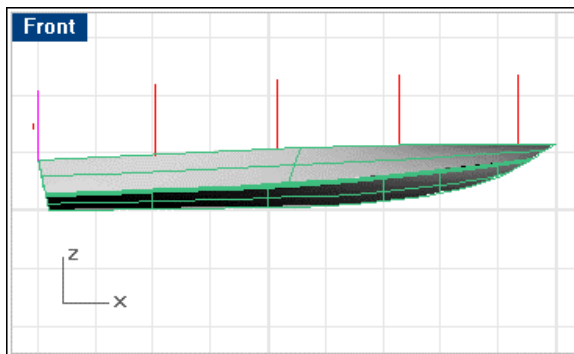
- 1 Make the **cTempDeckBms** layer current and turn on the **cCamber** layer.
- 2 Start the **Line** command.
- 3 At the **Start of line ...** prompt, use the **End** object snap to pick a point at the starboard corner of the transom at the sheer.
- 4 At the **End of line** prompt, in the **Right** viewport, use the **End** object snap to pick a point approximately 6 feet above the transom corner.



Draw a vertical line from the corner of the transom and sheer.

Copy the construction line along the sheer

- 1 Select the vertical line you just created.
- 2 Start the **Copy** command.
- 3 At the **Point to copy from ...** prompt, use the **End** object snap to pick the lower endpoint (at the sheer) of the vertical line you just created.
- 4 At the **Point to copy to** prompt, use the **Near** object snap to pick four approximately evenly spaced points along the starboard sheer.

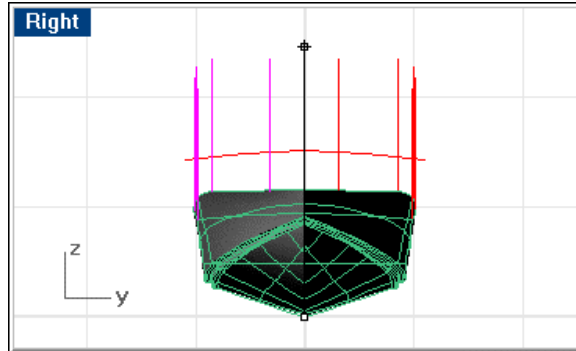


Copy the line along the sheer.

Mirror the vertical lines across centerline

- 1 Select the five vertical lines you just created.
- 2 Start the **Mirror** command.
- 3 At the **Start of mirror plane ...** prompt, type **0**.

- At the **End of mirror plane ...** prompt, in the **Right** viewport, use **Ortho** to drag the mirror plane line up along the y-axis.



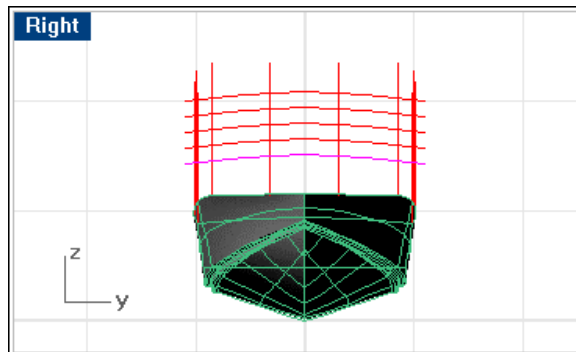
Mirror the vertical lines across the centerline.

Make copies of the camber curve

- Select the untrimmed camber curve
- Start the **Copy** command.
- At the **Point to copy from ...** prompt, in the **Right** viewport, pick a point.
- At the **Point to copy to** prompt, with **Ortho** on, pick 5 points directly above or below the origin point you just picked.

The copies of the camber curve being created should cross the vertical lines previously created.

- Select all of the copies of the camber curve you just created and change their layer to **cTempDeckBms**.



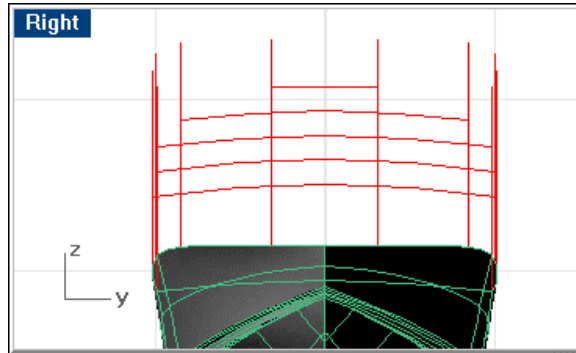
Copy the camber curve.

Trim copies of the camber curve to match the beam at each pair of vertical lines

- Select one of the vertical lines on the starboard side and the corresponding vertical line on the port side.
- Start the **Trim** command.
- At the **Select object to trim ...** prompt, select the outboard end of one of the camber curves on the starboard side and the outboard end of the same camber curve on the port side.

- 4 Repeat these steps for each of the camber curves.

Each of these curves will be a temporary deck beam to enable creation of the deck centerline.



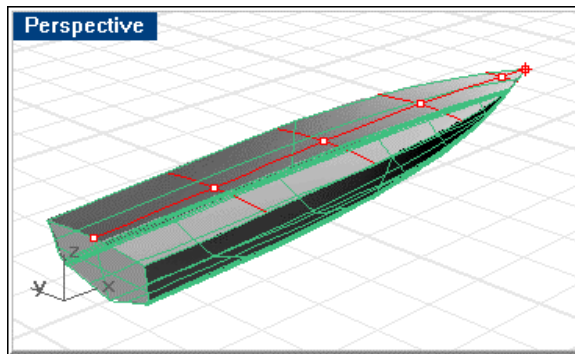
Trim the copies of the camber curves to the vertical lines.

Create deck beams

Once you have all the camber curves trimmed to these posts, **move** each camber curve from its endpoint to the endpoint of its corresponding post. In this way, you will have created a deck beam at each location on the sheer where you placed one of these pairs of posts.

These deck beams will be used to create the centerline of the deck and then will be discarded. Once the deck beams are in place, **delete** the posts.

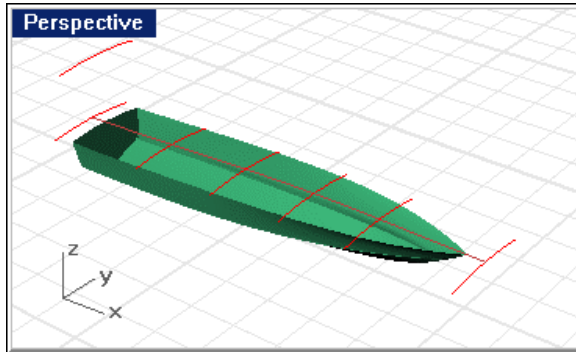
Use the **InterpCrv** command to create a deck centerline by connecting the **midpoint** of the transom to the **midpoints** of each of the deck beams and the end of the stemhead.



Temporary deck beams in place and deck centerline drawn through midpoints.

Once you have created the deck centerline, **extend** its ends beyond the ends of the hull. **Delete** the trimmed deck beams created previously.

Copy the untrimmed camber curve from its **midpoint** to various points along the deck centerline as well as one copy to each end of the deck centerline.

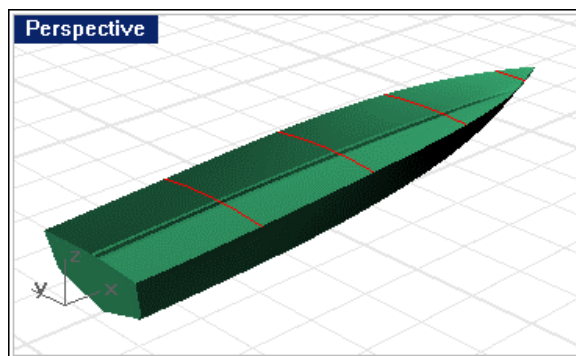


Copy the untrimmed camber curve in place of the temporary deck beams.

Step-by-Step: Create deck beams

Move the deck beams into place on the sheer

- 1 Select one of the deck beams.
- 2 Start the **Move** command.
- 3 At the **Point to move from ...** prompt, use the **End** object snap to pick a point on the end of the selected deck beam.
Be sure you do **not** select a point on the vertical line.
- 4 At the **Point to move to** prompt, use the **End** object snap to pick the endpoint of the corresponding vertical line where it meets the sheer.
Be sure you do **not** select a point somewhere else on the sheer.
- 5 **Move** each of the deck beams.
- 6 **Delete** the vertical lines associated with the deck beams you just moved.

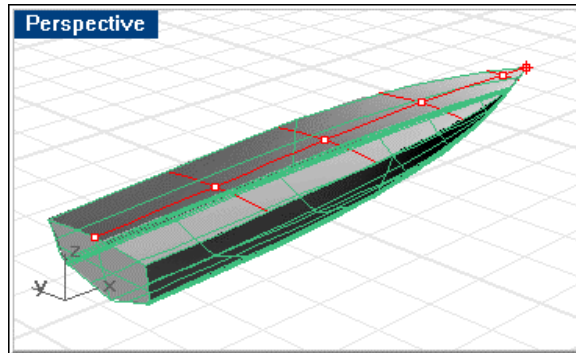


Move each trimmed curve to the lower endpoint of its corresponding vertical line.

Draw the deck centerline

- 1 Start the **InterpCrv** command.
- 2 At the **Start of curve ...** prompt, use the **Mid** object snap to pick a point at the midpoint of the transom sheer.
- 3 At the **Next point of curve ...** prompt, use the **Mid** object snap to pick a point at the midpoint of each of the deck beams.

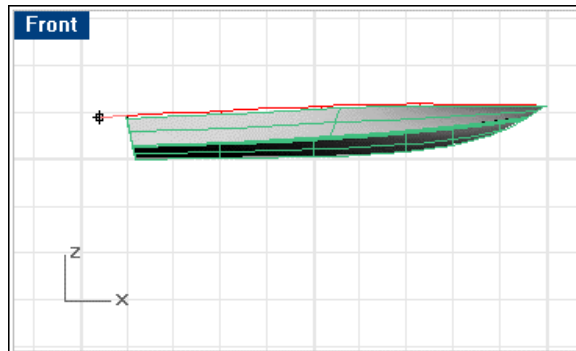
- 4 At the **Next point of curve ...** prompt, use the **End** object snap to pick the end of the stemhead.



Draw the deck centerline.

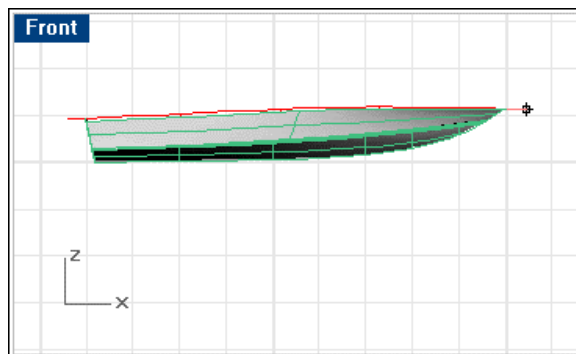
Extend the deck centerline

- 1 Start the **Extend** command.
- 2 At the **Select boundary objects or enter extension length. Press Enter for dynamic extend** prompt, press **Enter**.
- 3 At the **Select object to extend ...** prompt, in the **Front** viewport, select a point near the transom on the new deck centerline.
- 4 At the **End of extension** prompt, in the **Front** viewport, pick a point approximately one foot aft of the transom.



Extend the deck centerline aft.

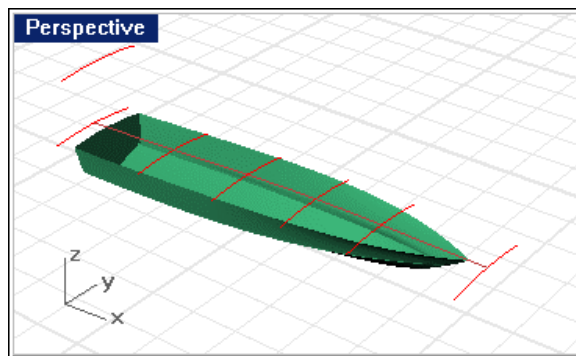
- 5 Repeat these steps for the forward end of the deck centerline curve.



Extend the deck centerline forward.

Copy full-breadth camber curves to the deck centerline

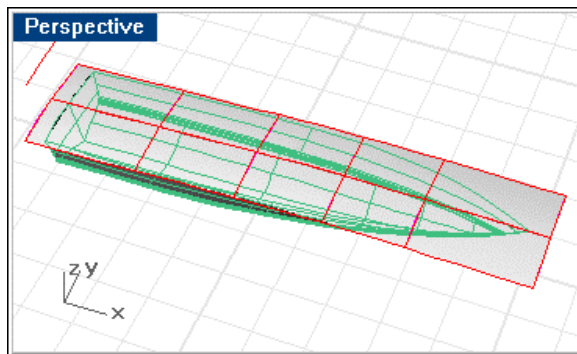
- 1 Turn off the layer **cTempDeckBms**.
 - 2 Select the camber curve that remains suspended above the hull.
 - 3 Start the **Copy** command.
 - 4 At the **Point to copy from ...** prompt, use the **Mid** object snap to pick the midpoint of the selected camber curve.
 - 5 At the **Point to copy to** prompt, use the **End** object snap to pick the aft end of the deck centerline.
 - 6 At the **Point to copy to** prompt, use the **End** object snap to pick the forward end of the deck centerline.
 - 7 At the **Point to copy to** prompt, use the **Near** object snap to pick four approximately evenly spaced points along the deck centerline.
- The **Near** object snap guarantees that the camber curve lands on the deck centerline.



Copy the untrimmed camber curve in place of the temporary deck beams.

Loft the deck beam curves

Loft these new curves.



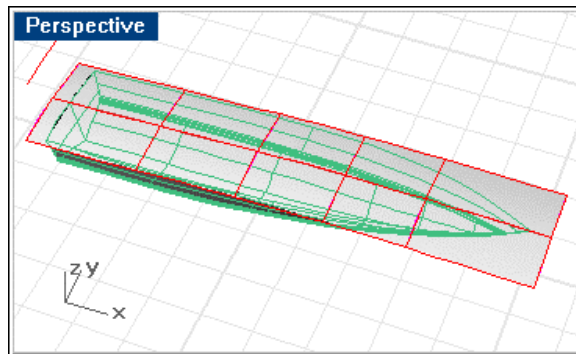
Lofted deck.

Step-by-Step: Loft the deck beam curves

Loft the full-breadth deck

- 1 Select the full-breadth camber curves.
- 2 Start the **Loft** command.

- 3 In the **Loft Options** dialog box, click **OK**.

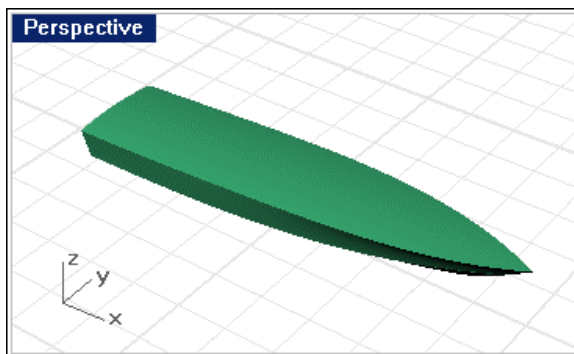


Loft the deck beams.

- 4 Select the full-breadth camber curves and change them to layer **cPermDeckBms**.

Trim the deck with the hull and join

Trim the lofted deck to the hull and **join** it to the hull. You should no longer have any unjoined edges on this model.



Deck joined to the hull. There should be no unjoined edges.

If you encounter difficulty here, **explode** the hull polysurface, **untrim** several of the surfaces, **extend** them through the deck, **re-trim** everything, and **rejoin** the surfaces.

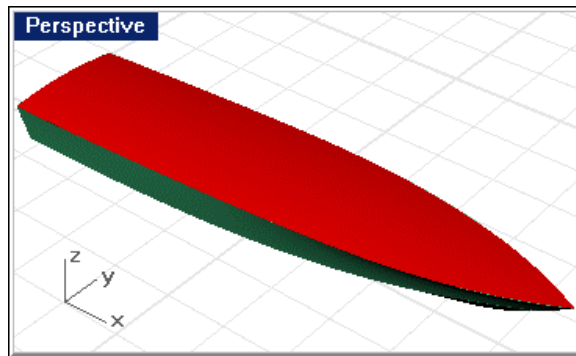
This type of “fussing” with the geometry to eliminate unjoined edges is expected. Learning to deal with these situations will enable you to move more quickly through creating models in Rhino.

Step-by-Step: Trim the deck with the hull and join

Trim the full-breadth deck to the hull

- 1 Select the hull.
- 2 Start the **Trim** command.

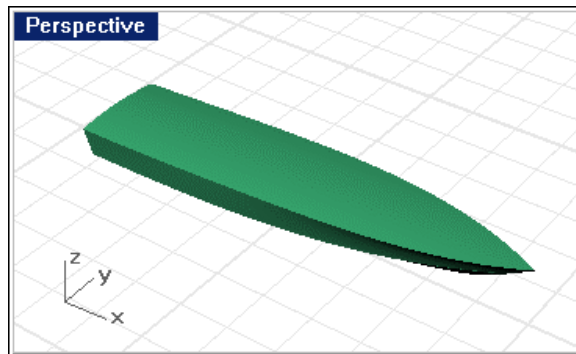
- At the **Select object to trim ...** prompt, select the deck outboard of the sheer. The deck will trim to match the sheer and transom.



Trim the deck to the hull.

Join the full-breadth deck to the hull

- Select the hull and deck.
- Use the **Join** command to join the hull and deck.
If not all surfaces are joined into a single polysurface, you must investigate which surfaces did not join and correct the problem. A single polysurface with no unjoined edges is the goal.
- Use the **ShowEdges** command to check for unjoined surfaces.
Rhino should report: *No naked edges found.*



Deck joined to the hull. There should be no unjoined edges.

- Save** your work.

Diagnostics

Before you can move on to creating the running strakes, cockpit and cabin, you must determine if you have any geometric problems. Rhino has several tools to tell you about bad geometry in the **Analyze** menu.

Use the **SelBadObjects** and **ShowEdges** commands. With these two commands, you can determine if Rhino's hydrostatics function as well as other third-party software will be able to work with your model. If you run these and get no bad objects or broken edges, you can rest. Otherwise, you must seek out the culprits (usually broken edges), fix them, and rerun the commands until the results are positive.

As with most diagnostic procedures, there is not one answer. Each situation requires its own solution.

It is important to note that programs such as SolidWorks, which will treat this hull as a solid, require that there be no unjoined edges. If there are, the model import into those programs will fail.

Step-by-Step: Perform diagnostics on the model

Analyze for bad objects and edges

- 1 Use the **SelBadObjects** command to find objects that do not pass Rhino's internal checking.
If the model contains bad objects, these will be highlighted and proper measures should be taken to correct the problems.
- 2 Select the hull/deck polysurface.
- 3 Use the **ShowEdges** command to find unjoined edges.

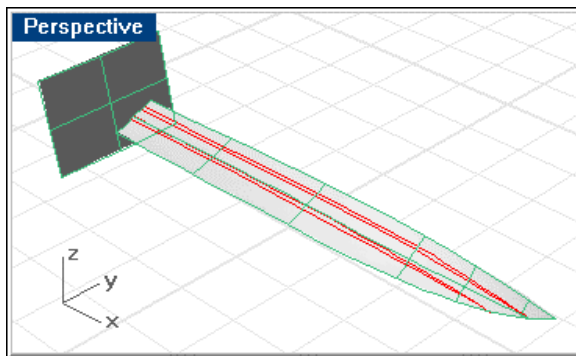
Prepare the model for the strakes

It may seem a little odd, but after the hull has been completed, you need to go back and add the running strakes. The reason for approaching it this way is that the bottom is a smooth surface onto which you will append the running strakes. It is much simpler to check the fairness of the bottom surface before dealing with the surfaces that make up each strake.

To add the strakes, it will be easier to make a copy of your model. In this way, you will have an uncluttered model in which you can do the intricate work. When you have constructed the strakes, you will join the bottom and strakes into a polysurface and copy it back to the original model using the Windows Clipboard to cut and paste the parts.

Open the strakes model, use the **ExtractSrf** command to extract one side of the bottom and the whole transom, **untrim** the transom, and **delete** the rest of the surfaces and all the curves. Next, from the lines drawing, **Hull Lines.3dm**, copy the outer and inner edges of the two running strakes in the plan view to the Clipboard and paste them into **Strakes.3dm**.

Once you have pasted the curves, **project** them to the bottom surface and **delete** the 2-D curves imported from the lines drawing.



Projected strakes lines.

Step-by-Step: Prepare the model for the strakes

Create a new model

- 1 **Save** your work.

- 2 Use the **SaveAs** command to save the model to a new file called **Strakes.3dm**.

Separate the bottom panel

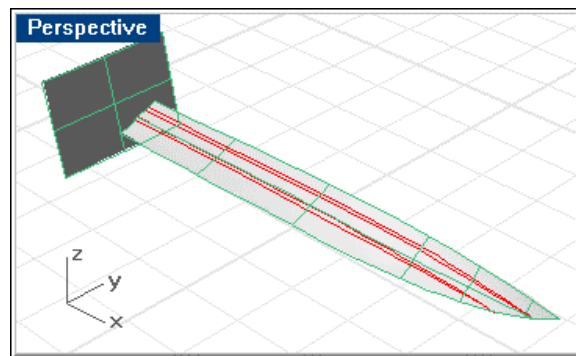
- 1 In the **Strakes.3dm** model, start the **ExtractSrf** command.
- 2 At the **Select surfaces to extract ...** prompt, select the port bottom panel and the transom.
- 3 Turn all layers on.
- 4 Use the **Zoom Extents All** command to bring all objects into view.
- 5 **Delete** everything except the port bottom and transom.

Untrim the transom

- 1 Start the **Untrim** command.
- 2 At the **Select boundary to detach** prompt, select the transom edge.

Copy the 2-D running strakes in plan view

- 1 In **Hull Lines.3dm**, in the **Top** viewport, select the inboard and outboard curves representing the running strakes.
- 2 Press **Ctrl+C** to copy these to the Clipboard.
- 3 In **Strakes.3dm**, press **Ctrl+V** to paste the curves in the plan view.
- 4 With the 2-D curves still selected, press **Shift**, and add the bottom surface to the selection.
- 5 Use the **Project** command to project the 2-D curves onto the bottom surface.
- 6 **Delete** the imported 2-D curves.



Projected strakes lines.

- 7 **Save** your work.

Model the strakes

To create the strakes, in addition to creating the surfaces for the strakes, you are going to make a gap between the hull and strake surfaces. This gap will be filled with a blend surface to create a rounded "fillet" between the strakes and the hull surface. To create the gap you are going to offset the defining curves by .25" on each side and trim the hull surface to this line. Later you will perform the same trim on the strake surfaces. Then you can create a smooth blend surface between the strake surfaces and the hull surfaces to fill the gap.

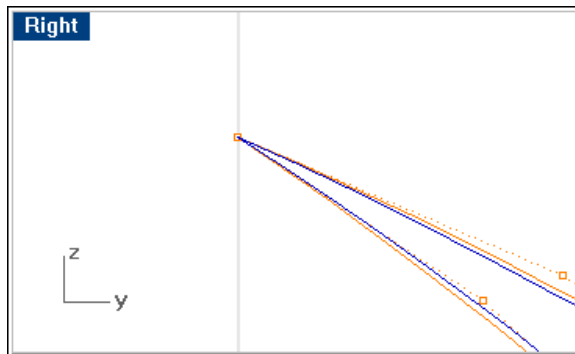
Create trimming curves for the hull

In the **Top** viewport, **offset** the outboard curve of each strake pair outboard and the inboard curve inboard. The amount you offset the curves will be the radius of the fillet. Use your own judgment (a radius of 0.25" was used in the example). Once all four curves have been offset, place the results onto layer **cHullTrimEdges**. These curves will be used to trim the bottom surface.

Rebuild all four hull-trimming curves to **Degree 3** with **10 Points**. This will allow easy manipulation of the edit points of these curves in the next step.

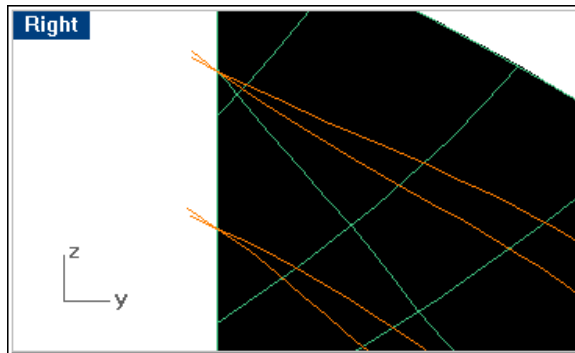
Turn on the **control points** of the four hull trimming curves. Notice that all of the strake curves come to a point right on the rabbet.

Use the **SetPt** command to change the forward ends of the hull trimming edges so they come to the same points. The ends of two hull trimming curves for the outboard strake should be coincident with the forward end of the outboard strake and likewise for the inboard strake.



Trimming curves for a running strake. All four curves come to a point at the rabbet.

Hide the strake curves and **extend** each of the four hull trimming curves so that they extend beyond the inboard and aft edges of the bottom surface, as shown below. This will facilitate trimming the bottom.



Extend trimming edges beyond the bottom surface edges, fore and aft.

Step-by-Step: Create trimming curves for the hull

Offset the strake curves

- 1 Select the outboard curve of the outboard running strake.
- 2 Start the **Offset** command.
- 3 At the **Side to offset ...** prompt, type **0.25"**

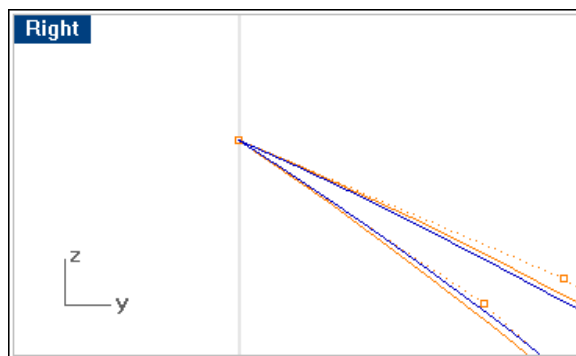
- 4 At the **Side to offset ...** prompt, in the **Top** viewport, pick a point outboard of the outboard running strake.
- 5 Select the inboard curve of the outboard running strake.
- 6 Repeat the **Offset** command.
- 7 Check that the offset distance is **0.25"**
- 8 At the **Side to offset ...** prompt, pick a point inboard of the outboard running strake.
- 9 Repeat these steps for the inboard running strake.

Rebuild the hull trimming edges

- 1 Select all four hull trimming curves you just created.
- 2 Start the **Rebuild** command.
- 3 In the **Rebuild Curve** dialog box, specify **10** points and degree **3**.

Modify the endpoints of the hull trimming curves

- 1 Select all four hull trimming curves.
- 2 Use the **PointsOn** command to turn on the curves' control points.
- 3 Select the points at the ends of the hull trimming curves for the outboard strake.
- 4 Start the **SetPt** command.
- 5 In the **Set Points** dialog box, check **Set X, Set Y, and Set Z**, click **Align to World**, and click **OK**.
- 6 At the **Location for points ...** prompt, use the **End** object snap to select the common endpoint of the outboard strake curves.
- 7 Select the points at the ends of the hull trimming curves for the inboard strake.
- 8 Start the **SetPt** command.
- 9 In the **Set Points** dialog box, check **Set X, Set Y, and Set Z**, click **Align to World**, and click **OK**.
- 10 At the **Location for points ...** prompt, use the **End** object snap to select the common endpoint of the inboard strake curves.

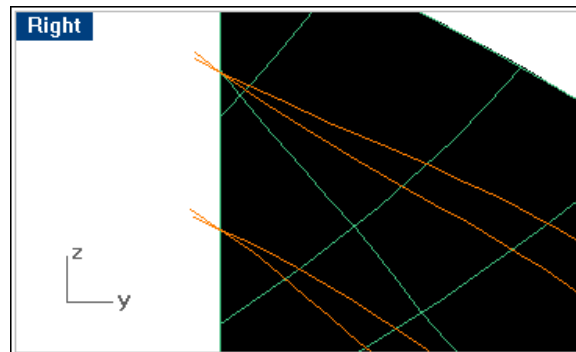


Use the SetPt command to move the endpoints of the trimming curves.

Extend the hull trimming curves

- 1 Start the **Extend** command.
- 2 At the **Select boundary objects ...** prompt, press **Enter** to start **Dynamic extend**.

- 3 At the **Select object to extend ...** prompt, select the forward end of the outboard strake's outboard trimming edge.
- 4 Pick a point forward of the endpoint of the strake.
- 5 Repeat these steps for each end of each hull trimming curves, fore and aft.

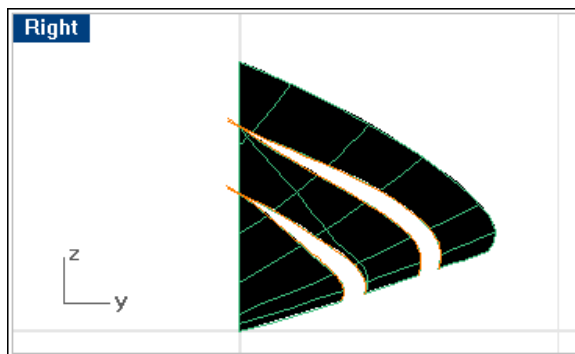


Extend the trimming curves.

- 6 **Save** your work.

Trim the hull to create a space for the strakes

Trim the bottom surface to the two pairs of hull trimming curves.



Trim the hull.

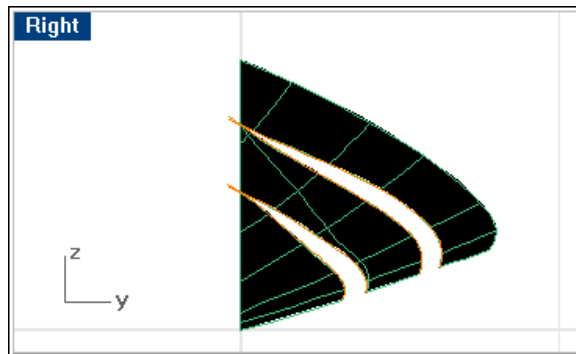
Hide the trimming curves and **show** both pairs of strake curves.

Step-by-Step: Trim the hull to create a space for the strakes

Trim the hull to the hull trimming curves

- 1 Select all four hull trimming curves.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, select the hull bottom at a point between the two outboard trimming curves and at a point between the two inboard trimming curves.

- 4 **Delete** the four hull trimming curves.

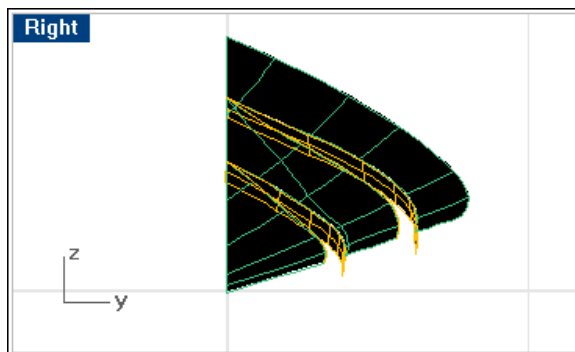


Trim the hull.

Create the strake surfaces

Extrude the outboard strake curves to make the vertical outboard surfaces of the two running strakes and **extrude** the inboard strake curves to make the horizontal surfaces.

For the vertical surfaces, give the surfaces a -5 -degree taper and a depth of 4 inches.



Extrude the strake curves.

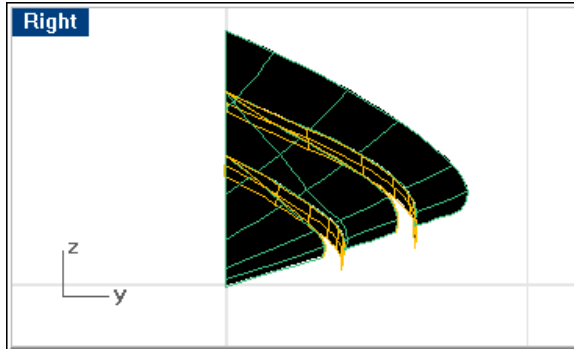
For the bottom surface, use a breadth of 4 inches.

Step-by-Step: Create the strake surfaces

Extrude the strake surfaces

- 1 Select the outboard curves of both strakes.
Be sure you do not to select the edge curves of the bottom surfaces.
- 2 In the **Top** viewport, start the **Extrude** command.
- 3 At the **Extrusion distance** *,,,* prompt, type **T** to indicate a tapered extrusion.
- 4 At the **Draft angle** $< 0 >$ prompt, type **-5** to indicate a 5-degree inboard taper.

- At the **Extrusion distance ...** prompt, type **-4"** to indicate a 4-inch extrusion down.



Extrude the stake curves.

- Select the inboard stake edges of both stakes.
- In the **Front** viewport, repeat the **Extrude** command.
- At the **Extrusion distance <-4 > ...** prompt, type **-4"** to extrude the curve away from the centerline.

Trim the stake surfaces

After the extruded surfaces have been created as described, **trim** the two surfaces to each other.

Use the **Intersect** command to create trimming curves and **extend** these curves. Creating curves that extend beyond the boundaries of the surfaces makes trimming more reliable.

Step-by-Step: Trim the stake surfaces

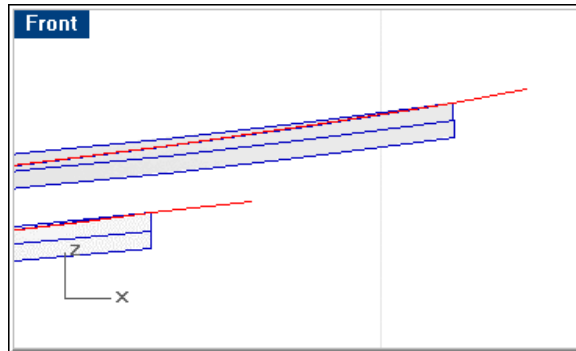
Trim the stakes

- Select the bottom and side surfaces of the inboard stake.
- Start the **Intersect** command.
This creates a curve that defines the intersection of the two surfaces.

Extend the intersection curve

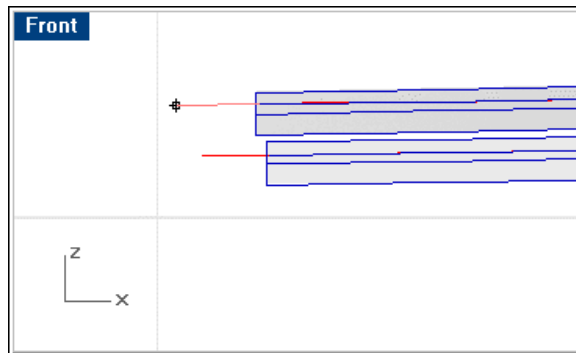
- Start the **Extend** command.
- At the **Select boundary objects ...** prompt, press **Enter** to start **Dynamic extend**.
- At the **Select object to extend ...** prompts, select the forward ends of the intersection curves you just created.

- 4 At the **End of extension** prompts, pick a point forward of the end of the strake surfaces.



Extend the trimming curves forward.

- 5 Repeat the **Extend** command.
- 6 At the **Select boundary objects ...** prompt, press **Enter** to start **Dynamic extend**.
- 7 At the **Select object to extend ...** prompts, select a point at the aft end of the intersection curves you just created.
- 8 At the **End of extension** prompts, pick a point aft of the aft end of the strake surfaces.

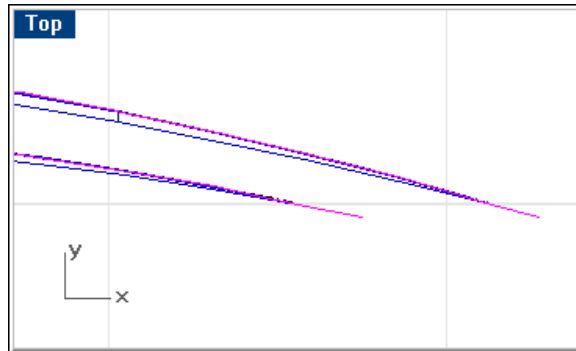


Extend the trimming curves aft.

Trim the strake surfaces with the extended intersection curve

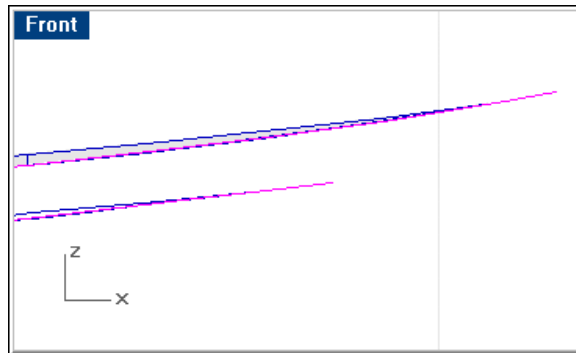
- 1 Select the intersection curves you just created.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, in the **Top** viewport, select the outboard edge of the inboard strake bottom surface.

- 4 At the next **Select object to trim ...** prompt, in the **Top** viewport, select the outboard edge of the outboard strake bottom surface.



Trim the strake surfaces in the Top viewport.

- 5 At the next **Select object to trim ...** prompt, in the **Front** viewport, select the bottom edge of the inboard strake side surface.
- 6 At the next **Select object to trim ...** prompt, in the **Front** viewport, select the bottom edge of the outboard strake side surface.



Trim the strake surfaces in the Front viewport.

- 7 **Delete** the intersection curves.
- 8 **Save** your work.

Blend the gap between the strakes and hull

The next step is to trim the strakes back to create the other half of the gap to match the gap we trimmed into the hull surfaces.

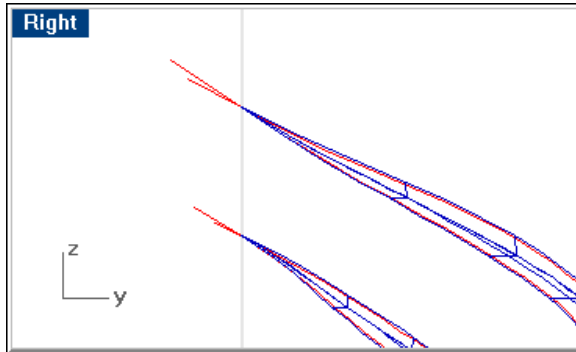
Create the trimming curves for the strakes

Now that you have the surfaces of the strakes built, **offset** the upper edges of the vertical strake surfaces vertically down the same distance as you previously offset them horizontally (.25"). Do this in the **Front** viewport. This will form the strake trimming curves for the tops of each strake. Change these curves to layer **cStrakeTrimmingEdges**.

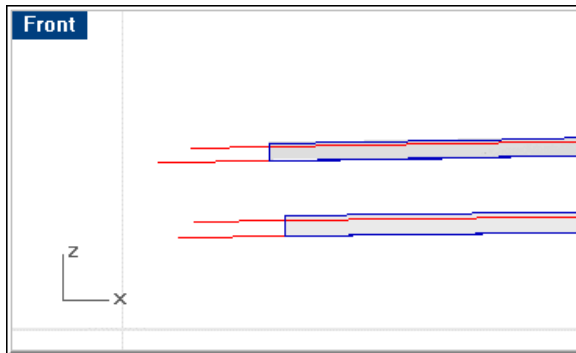
Offset the inboard edges of the horizontal strake surfaces outboard by this distance as well. This will create the trimming curves for the inboard edges of the running strakes.

Rebuild the resulting curves, and use the **SetPt** command to move their forward points to coincide the forward ends of the strakes, so that all curves and surfaces come to a point at the rabbet.

Extend the trimming edges beyond the edges of the surfaces fore and aft to ensure success in the trimming operation.



Extend the offset strake trimming edges.



Strake trimming edges constructed.

Step-by-Step: Create the trimming curves for the strakes

Create the strake trimming curves

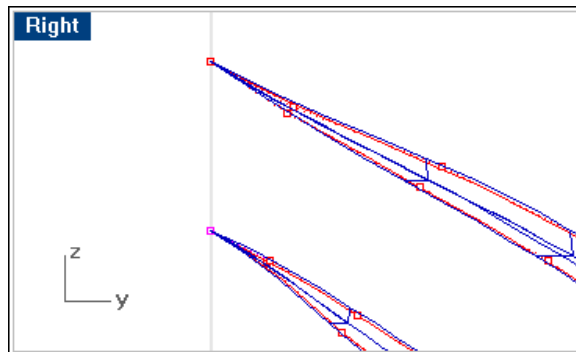
- 1 Start the **Offset** command.
- 2 Set the distance to **0.25"**.
- 3 In the **Front** viewport, select the upper edge of the vertical surface of the outboard strake.
- 4 At the **Side to offset ...** prompt, in the **Front** viewport, pick a point below the outboard running strake, so the curve is offset "down."
- 5 Repeat these steps for the inboard running strake.
- 6 Repeat the **Offset** command.
- 7 In the **Top** viewport, select the inboard edge of the bottom surface of the outboard running strake.
- 8 At the **Side to offset ...** prompt, in the **Top** viewport, pick a point outboard of the outboard running strake.
- 9 Repeat these steps for the inboard running strake.

Rebuild the strake trimming curves

- 1 Select all four strake trimming curves you just created.
- 2 Start the **Rebuild** command.
- 3 Complete the **Rebuild** dialog box specifying **10** points and degree **3**.

Modify endpoints at the forward end

- 1 Select all four strake trimming curves.
- 2 Use the **PointsOn** command to turn on the curves' control points.
- 3 Select the points at the ends of the trimming curves for the outboard strake.
- 4 Start the **SetPt** command.
- 5 In the **Set Points** dialog box, check **Set X**, **Set Y**, and **Set Z**, click **Align to World**, and click **OK**.
- 6 At the **Location for points ...** prompt, use the **End** object snap to select the endpoint of the outboard strake edges.
- 7 Select the points at the ends of the trimming curves for the inboard strake.
- 8 Start the **SetPt** command.
- 9 In the **Set Points** dialog box, check **Set X**, **Set Y**, and **Set Z**, click **Align to World**, and click **OK**.
- 10 At the **Location for points ...** prompt, use the **End** object snap to select the endpoint of the inboard strake edges.

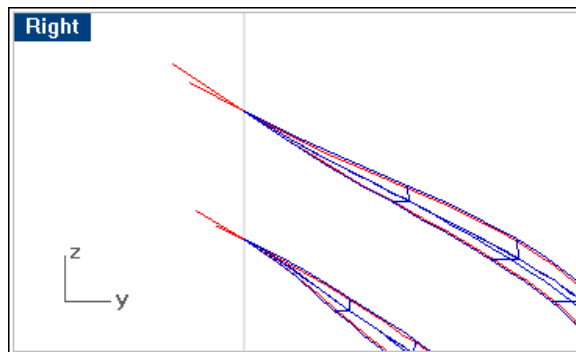


Use the SetPt command to move then end control points together.

- 11 Press **Esc** to turn off the **Control Points**.

Extend the outboard strake trimming curve

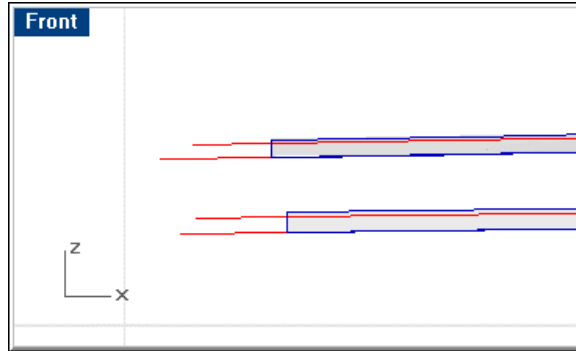
- 1 Start the **Extend** command.
- 2 At the **Select boundary objects ...** prompt, press **Enter** to start **Dynamic extend**.
- 3 At the **Select object to extend ...** prompt in the **Right** viewport, select the forward end of the outboard strake's outboard trimming curve.



Extend the offset trimming curves fore.

- 4 At the **End of extension** prompt, pick a point forward of the endpoint of the strake.

- Repeat these steps for each end of each strake trimming curve fore and aft, making the ends extend beyond the ends of the strake surfaces.



Strake trimming edges constructed.

Create a blend surface between the strakes and hull

Now that you have constructed the strake trimming edges, **trim** the strakes to these and the strake surfaces will be ready for blending to the hull.

One way to fillet two surfaces with a variable radius is to make the surfaces with edges touching, **trim** the edges back an amount equal to the radius, and create a **blend** surface between them. The actual shape of any station on the surface, while approximating a radius, is not an actual radius. Allowing Rhino the freedom to use the appropriate shape at any station means the fillet will go much more smoothly, especially in cases where the fillet requires variable radius. The surface created with the **Blend** command will be guaranteed to be curvature continuous (G2).

To finish the strakes, use the **BlendSrf** command and select two adjacent edges. Make certain the **Continuity** option is set to **Curvature**. Rhino will construct a surface between the bottom surface and strake surface that is curvature continuous (G2) from stem to stern and comes to a point at the bow.

Step-by-Step: Create a blend surface between the strakes and hull

Trim the strakes to the strake trimming edges

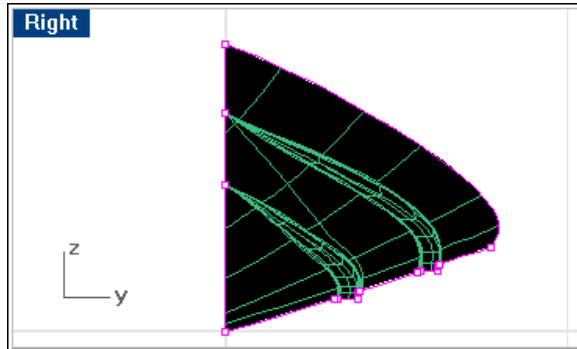
- Select all four strake-trimming curves.
- Start the **Trim** command.
- At the **Select object to trim ...** prompts, in the **Front** viewport, trim the edges of the vertical strake surfaces above of the trimming curves.
- At the **Select object to trim ...** prompts, in the **Top** viewport, trim the edges of the horizontal strake surfaces inboard of the trimming curves.

Blend the strakes to the hull

- Start the **BlendSrf** command.
- At the **Select first set of edges ...** prompt, select outboard edge of the the inboard bottom surface.
- At the **Select second set of edges ...** prompt, select the inboard edge of the inboard strake's bottom surface.
- In the **Blend Bulge** dialog box, click **OK** to accept the defaults.
- Repeat these steps for each two adjacent edges (4 blends in all).

Join and check the hull

Once you have blended all four edges (2 per strake), **join** everything and check for unjoined edges.



Finished bottom with strakes, joined. No unjoined edges.

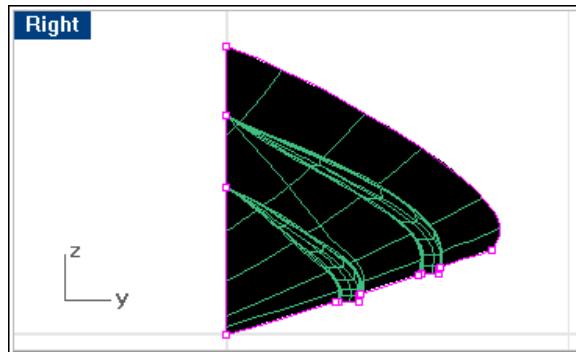
Step-by-Step: Join and check the hull

Join the surfaces

- 1 Select all the surfaces.
- 2 Use the **Join** command to join the surfaces into one polysurface.

Check for unjoined edges

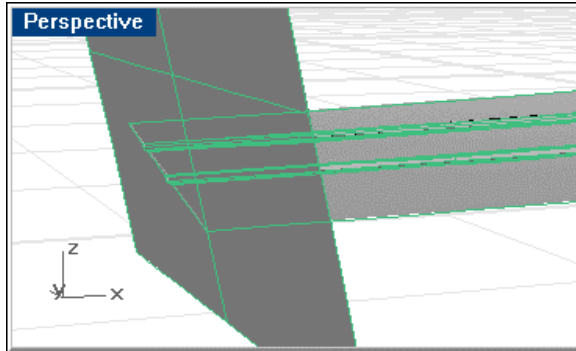
- 1 Select the bottom polysurface.
- 2 Start the **ShowEdges**.
Only the sheer, centerline and transom edges should highlight.



Show the hull naked edges.

Trim the hull to the transom surface

Trim the aft edge of the polysurface to the transom surface.



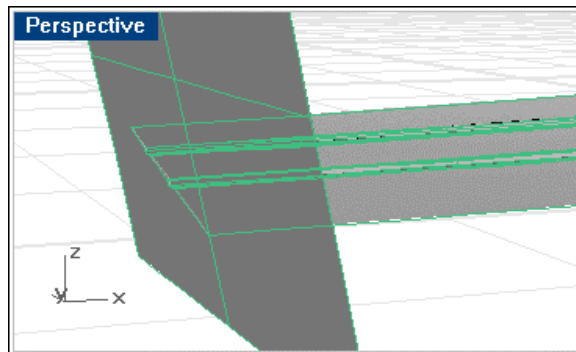
Trim the hull to the transom.

Step-by-Step: Trim the hull to the transom surface

Trim the aft edge of the bottom polysurface to the transom

- 1 Select the transom.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, select the aft edge of the bottom polysurface.

The bottom polysurface should trim to the transom.



Trim the hull to the transom.

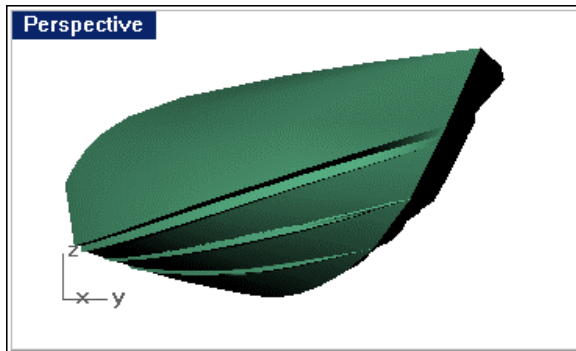
If it does not trim, **explode** the hull surface and use the **ExtendSrf** command to extend surfaces beyond the trimming surface and then **trim**.

- 4 **Save** your work.

Marry the bottom with strakes into the hull model

Copy the bottom panel to the Clipboard, open **Offshore.3dm**, **delete** the two bottom surfaces from that model, and **paste** the bottom panel from the Clipboard.

Mirror and **join** the surfaces into one closed polysurface with no unjoined edges.

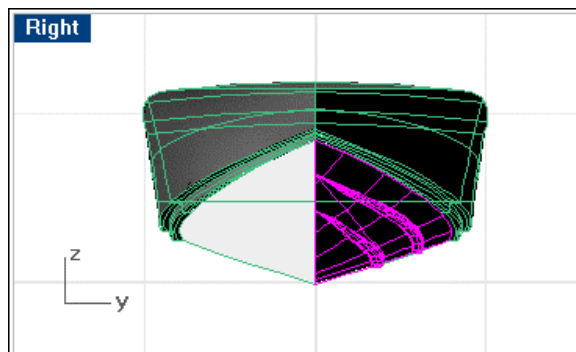


Finished hull with strakes added.

Step-by-Step: Marry the bottom with strakes into the hull model

Copy the hull bottom from the Strakes model to the Offshore model

- 1 In **Strakes.3dm**, select the bottom polysurface.
- 2 Press **Ctrl+C** to copy the selected surface to the Clipboard.
- 3 In **Offshore.3dm**, use the **ExtractSrf** command to separate the bottom surfaces on both sides of centerline from the hull.
- 4 **Delete** the two bottom surfaces.
- 5 Press **Ctrl+V** to paste the contents of the Clipboard into the current model.
- 6 Leave the bottom surface you just pasted selected for the next step.

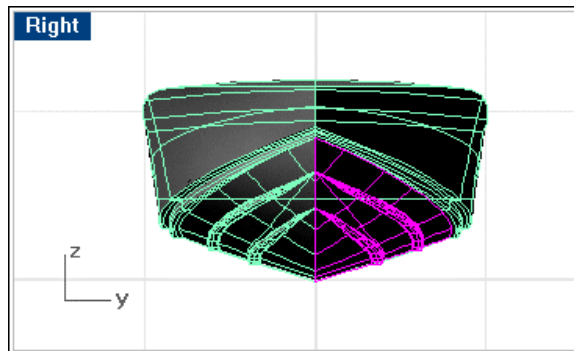


Paste the new bottom into place.

Mirror the pasted bottom

- 1 Use the **Mirror** command to mirror the bottom surface.
- 2 In the **Front** viewport, at the **Start of mirror plane ...** prompt, type **0**.

- At the **End of mirror plane ...** prompt, drag the mirror plane up along the z-axis.



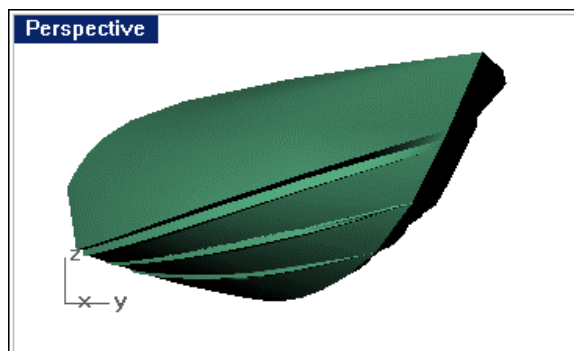
Mirror the bottom halves.

Join the hull surfaces

- Select all the hull surfaces.
- Start the **Join** command.
- Check the command prompt area to be sure all surfaces/polysurfaces were joined into a single polysurface.

Check for unjoined edges

- Use the **ShowEdges** command to check the joined hull for unjoined edges. There should be no unjoined edges.



Join all the hull parts.

- Save** your work.

Add the cockpit and cabin

To begin the cockpit, you will work from two models, similar to the way you created the running strakes.

Set up the models

Save the finished hull model **Offshore.3dm** as **Deck.3dm**.

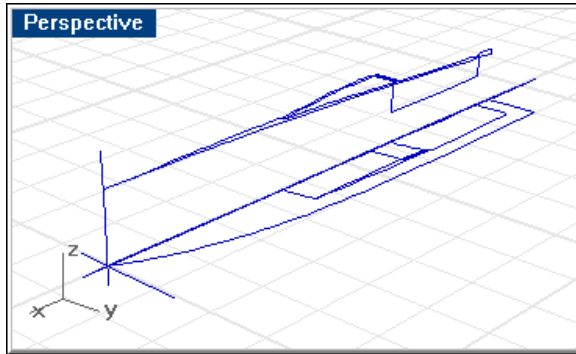
Extract the deck surface and **delete** all other geometry. The second model you will work from is called **Deck Lines.3dm**. This is a simplified lines drawing of just the deck, cockpit and cabin.

Prepare the deck lines drawing for export similar to the way you prepared the hull lines for export.

Rotate the profile so its world top y-dimensions become world top z-dimensions.

Move the plan so its centerline is at **Y=0**. Be sure not to move either view longitudinally. There is no body plan, so it is not necessary to rotate that view as you did with the hull lines.

The prepared model **Deck Lines.3dm** should resemble the illustration below.



Deck lines prepared for export.

Step-by-Step: Set the models up

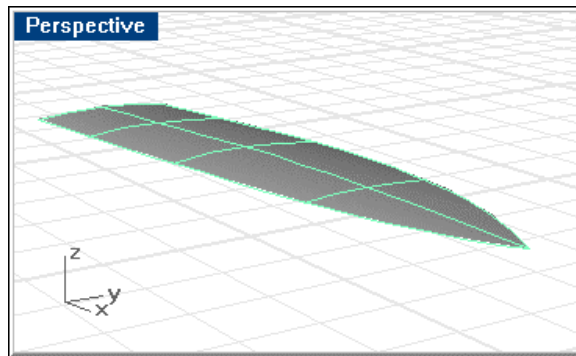
Prepare the deck model

- 1 Open **Offshore.3dm**.
- 2 Use the **SaveAs** command to create a new model called **Deck.3dm**.
- 3 Turn on all **layers**.
- 4 Use the **Show** command to show all hidden objects.
- 5 Select the joined hull polysurface.
- 6 Use the **Invert** command to deselect the deck surface and to select everything else in the model.
- 7 **Delete** the selected objects.

Extract the deck surface

- 1 Start the **ExtractSrf** command.
- 2 At the **Select surfaces to extract ...** prompt, select the deck surface.
- 3 Use the **Invert** command to deselect the deck surface and to select everything else in the model.

- 4 **Delete** the selected objects.



The deck surface.

- 5 **Save** your work.

Prepare the deck lines drawing

- 1 In **Deck Lines.3dm**, select all the profile curves.
- 2 Start the **Rotate** command.
- 3 At the **Center of rotation ...** prompt, type **0**.
- 4 At the **Angle or first reference point ...** prompt, type **0**.
- 5 At the **Second reference point ...** prompt, with the **Right** viewport active, type **90**.

Move the plan curves to the centerline

- 1 Select the plan curves.
- 2 In the **Top** viewport, start the **Move** command.
- 3 At the **Point to move from ...** prompt, use the **Near** object snap to select a point on the plan view centerline.
- 4 At the **Point to move to** prompt, use the **Perp** object snap to pick a point on the profile baseline.
- 5 **Save** your work.

Create the cockpit curves

Since it is the simplest, **copy** the plan view lines of the cockpit from **Deck Lines.3dm** and **paste** them to **Deck.3dm**.

Mirror the copied curves making up the cockpit across centerline.

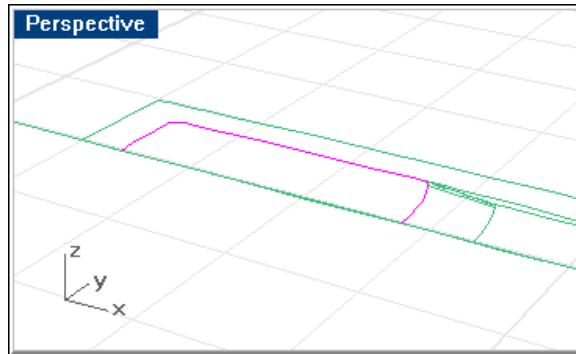
Join the curves.

Step-by-Step: Create the cockpit curves

Import the 2-D curves and prepare for extruding

- 1 In **Deck Lines.3dm**, select the curves representing the cockpit sides in plan view.
- 2 Press **Ctrl+C** to copy these to the Clipboard.

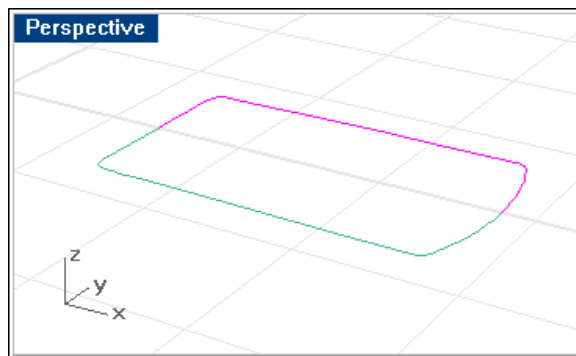
- 3 In **Deck.3dm**, press **Ctrl+V** to paste the curves from the Clipboard.



The cockpit sides in plan.

Mirror the cockpit curves

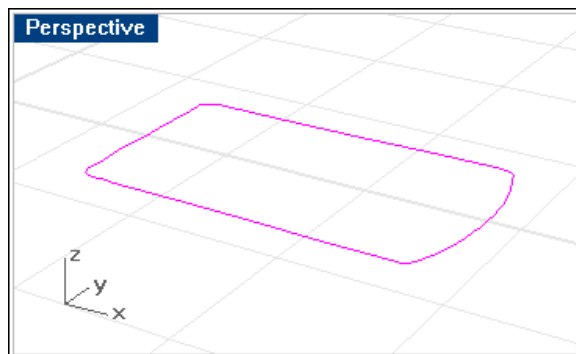
- 1 Select the curves you just pasted.
- 2 Start the **Mirror** command.
- 3 At the **Start of mirror plane ...** prompt, type **0**.
- 4 At the **End of mirror plane ...** prompt, drag the mirror line along the x-axis.



Mirror the cockpit curves. (Deck surface hidden for illustration.)

Join the mirrored curves

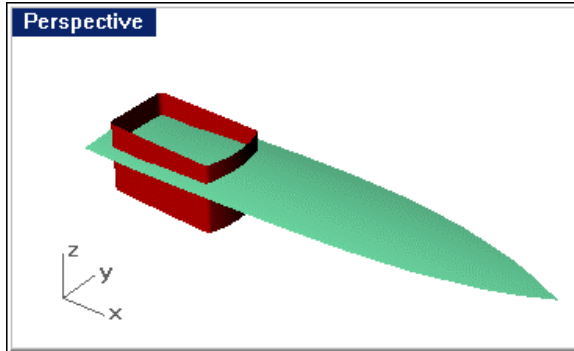
- 1 Select the mirrored curve and the result (the two sides of the cockpit plan view).
- 2 Use the **Join** command to join the two side curves into one.



Join the cockpit curves. (Deck surface hidden for illustration.)

Extrude the cockpit curve and trim

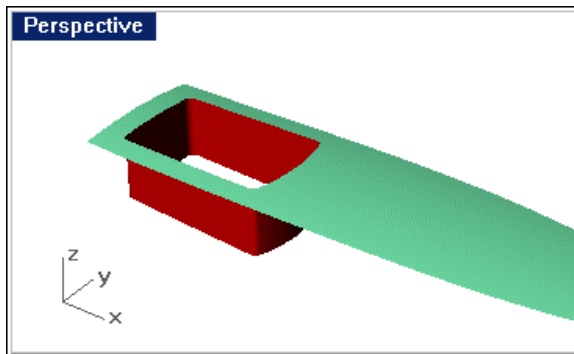
Extrude the curve to a height somewhat above the deck.



Extrude the cockpit curves.

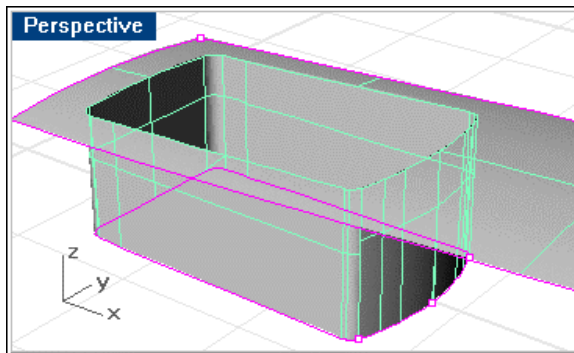
Trim the deck to the extruded cockpit sides.

Trim the cockpit sides to the deck.



Trim the cockpit sides and deck.

Join the deck and cockpit sides and check to be sure that you have no unjoined edges at the joint.



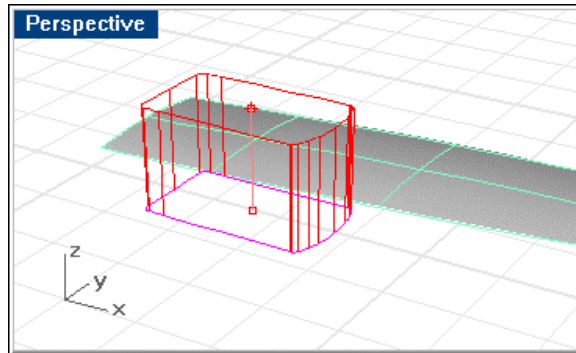
Join the deck and cockpit sides.

Step-by-Step: Extrude the cockpit curve and trim

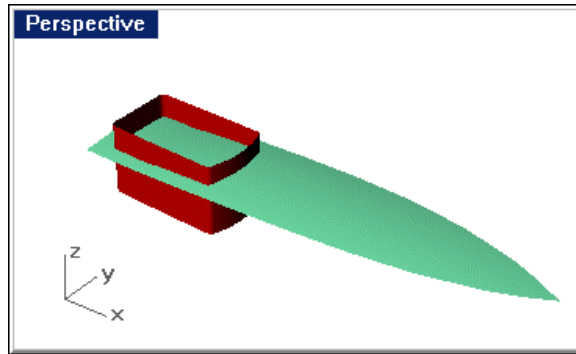
Extrude the 2-D curve and trim the cockpit sides and deck

- 1 Select the cockpit plan curve.
- 2 Start the **Extrude** command.

- 3 At the **Extrusion distance ...** prompt, in the **Front** viewport, pick a point above the deck.



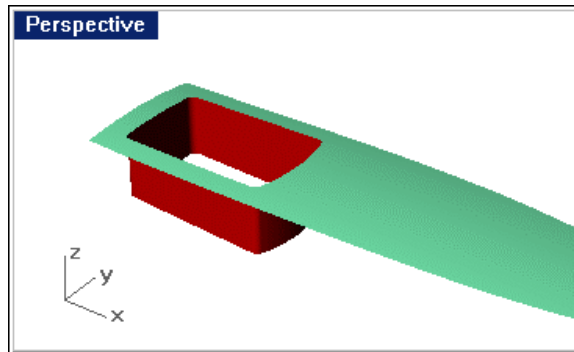
Extrude the cockpit curves.



The cockpit surface.

Trim the cockpit sides and deck

- 1 Select the deck and cockpit sides.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, select:
 - the cockpit sides at a point above the deck.
 - the deck surface inside the edges of the cockpit.



The trimmed cockpit and deck.

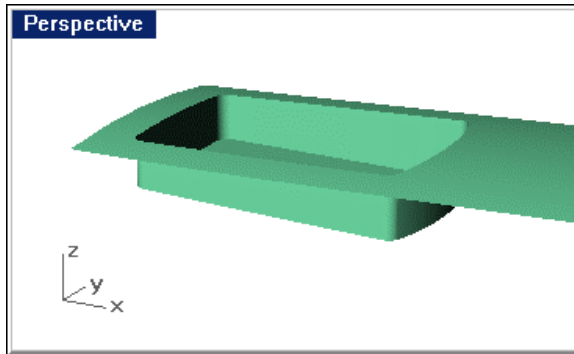
Create the cockpit sole

When you are satisfied with the deck/cockpit edge, **copy** the horizontal line that represents the cockpit sole from **Deck Lines.3dm** and **paste** it into **Deck.3dm**.

Use this line to **trim** the vertical cockpit surface. You will have to **extend** the line towards the fore and aft of the boat and then **extrude** the line using the **BothSides** option to obtain a surface to use for trimming.

Use the **Cap** command to close the bottom of the cockpit. This command closes any planar openings in a surface or polysurface.

Join all the surfaces and **check** for unjoined edges.



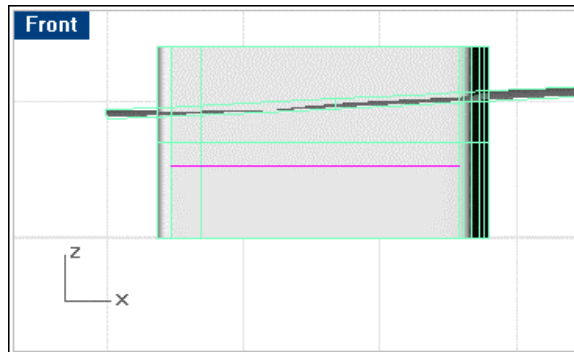
The trimmed and joined cockpit.

Delete the cockpit 2-D curves and **save** the model.

Step-by-Step: Create the cockpit sole

Copy and extend the cockpit sole line

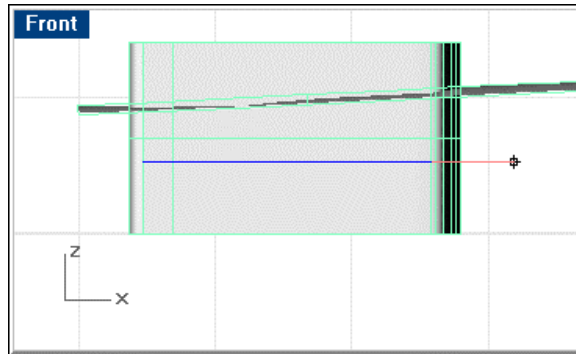
- 1 In **Deck Lines.3dm**, select the horizontal line that represents the cockpit sole.
- 2 Press **Ctrl+C** to copy these to the Clipboard.
- 3 In **Deck.3dm**, press **Ctrl+V** to paste the curves from the Clipboard.



Copy the profile curve for the cockpit sole.

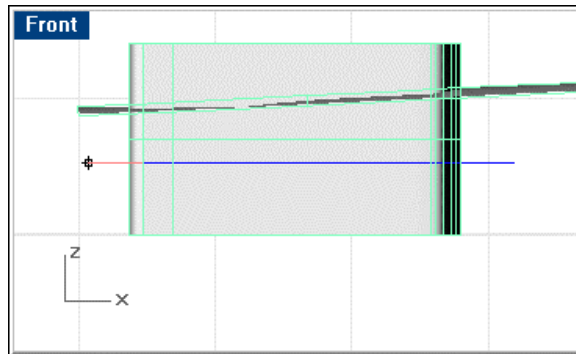
- 4 Start the **Extend** command
- 5 At the **Select boundary objects ...** prompt, press **Enter** to start **Dynamic extend**.
- 6 At the **Select object to extend ...** prompt, in the **Top** viewport, select the forward end of the imported trimming curve.

- 7 At the **End of extension** prompt, pick a point forward of the cockpit.



Extend the sole curve forward.

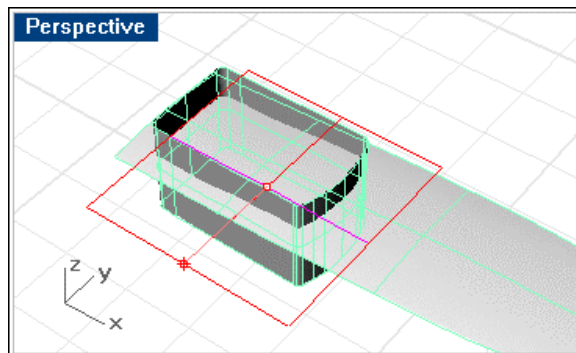
- 8 At the **Select object to extend ...** prompt, in the **Top** viewport, select the aft end of the imported trimming curve.
- 9 At the **End of extension** prompt, pick a point aft of the cockpit.



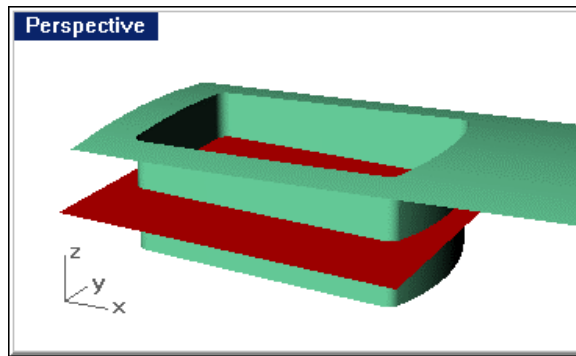
Extend the sole curve aft.

Extrude the cockpit sole line

- 1 Select the extended cockpit sole line.
- 2 In the **Front** viewport, start the **Extrude** command.
- 3 At the **Extrusion distance ... (Direction BothSides Tapered)** prompt, type **B**.
- 4 At the **Extrusion distance ...** prompt, in the **Top** viewport, pick a point outside the cockpit sides.



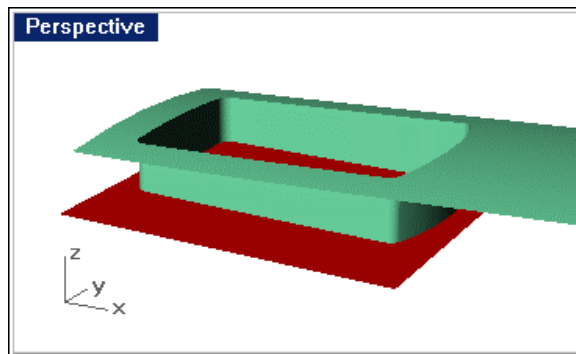
Extrude the sole curve.



The extruded cockpit sole.

Trim the cockpit to the extruded sole line

- 1 Select the extruded cockpit sole.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim** prompt, select the cockpit sides below the sole.

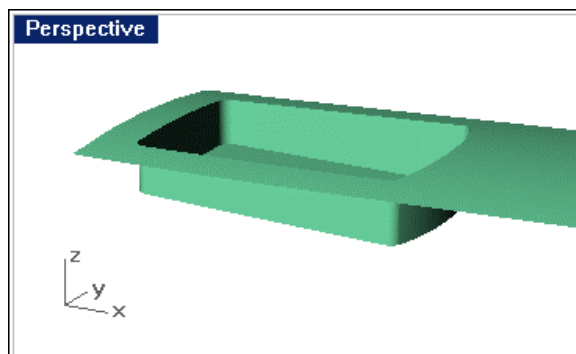


Trim the cockpit with the sole surface.

- 4 **Delete** the trimming surface.

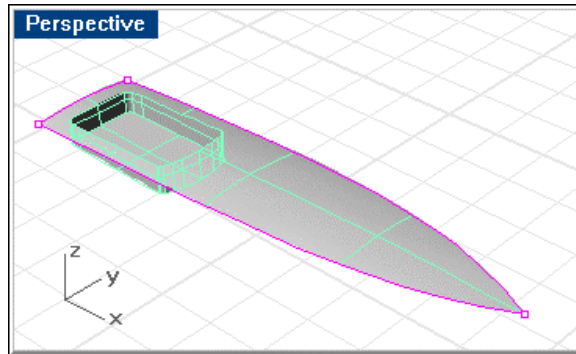
Cap the cockpit and check

- 1 Select the cockpit and deck polysurface.
- 2 Use the **Cap** command to close the cockpit sole.



Create and join the sole surface with the Cap command.

- 3 Use the **ShowEdges** command to check for unjoined edges.

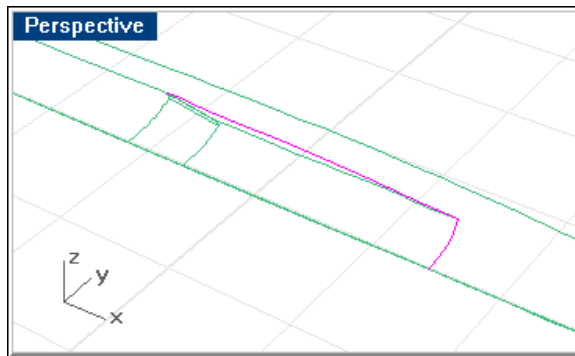


There should be no unjoined edges around the cockpit.

- 4 **Save** your work.

Create the opening in the deck in way of the cabin

Copy and paste the curves that make up the intersection between deck and cabin side/top. Since these will be projected to the deck to get 3-D curves, it is only necessary to import the plan view curves. You also do not need the curve that defines the rear edge.



The cabin side/top curves.

Mirror the curves around the centerline.

Join the curves.

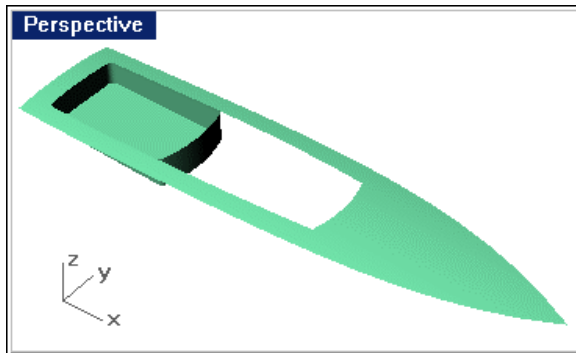
Explode the deck/cockpit polysurface. This makes the trimming process easier. **Hide** the cockpit surfaces.

Trim the deck to these curves. Use the Top view and apparent intersections.

The resulting edges will be the lower edge of the cabin side and the forward edge of the cabin top.

Show the cockpit surfaces and **Join** the deck and cockpit surfaces back together.

Delete the 2-D curves.



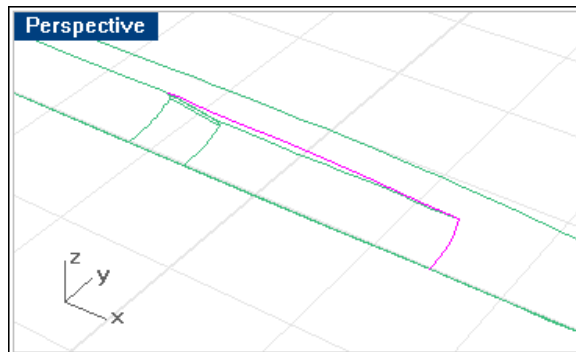
Trim deck to cabin edges.

Note: The top edge of the cockpit at the forward end will later become the bottom edge of the cabin aft bulkhead.

Step-by-Step: Create the opening in the deck for the cabin

Copy the cabin opening curves

- 1 In **Deck Lines.3dm**, select the curves in plan view representing the cabin/deck intersection.
- 2 Press **Ctrl+C** to copy the curves to the Clipboard.
- 3 In **Deck.3dm**, press **Ctrl+V** to paste the curves.

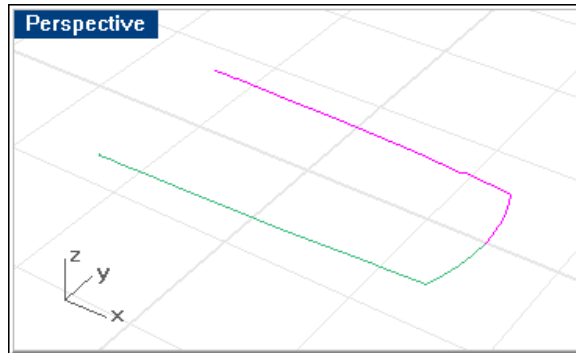


The cabin side/top curves.

Mirror the deck curves

- 1 Select the deck/cabin intersection curves you just joined.
- 2 Start the **Mirror** command.
- 3 In the **Top** viewport, at the **Start of mirror plane ...** prompt, type **0**.

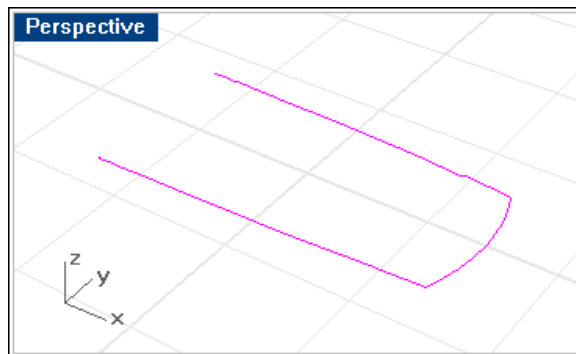
- At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line to the right along the x-axis.



Mirror the deck/cabin intersection curves.

Join the mirrored curves

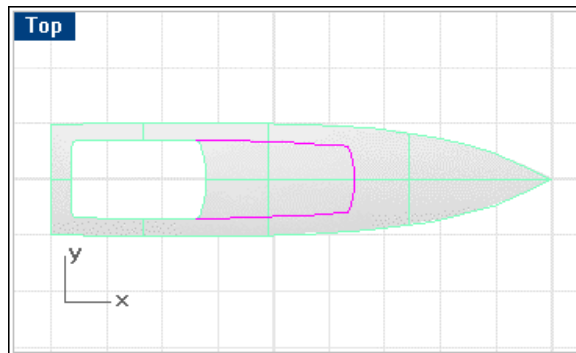
- Select the curve you just mirrored and the result of that mirror (the deck/hull intersection on both sides).
- Use the **Join** command to join the curves into one.



Join the curves.

Trim the deck surface

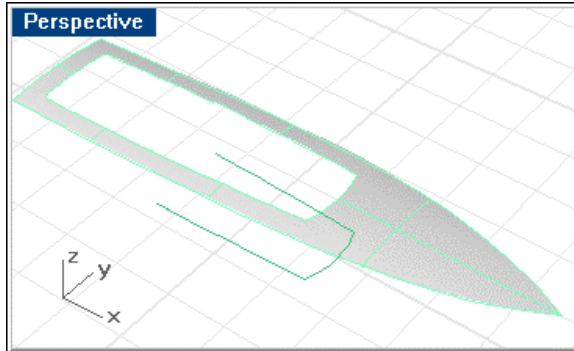
- Select the curve you just joined.



Trim the deck surface with the curves.

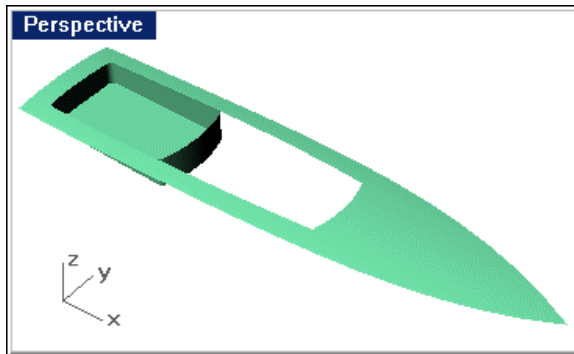
- Start the **Trim** command.

- At the **Select object to trim ...** prompt, in the **Top** viewport, select the deck inside the cutting edge boundary.



The trimmed deck.

- Show** the hidden surfaces.
- Select the deck and cockpit surfaces.
- Use the **Join** command to join the surfaces.



Join the deck and cockpit surfaces.

- Save** your work.

Create the aft cabin bulkhead

The aft bulkhead of the cabin is made up of three main surfaces. The large surface is gently arched. This surface is joined on each side by a surface of tighter radius at the outboard edge.

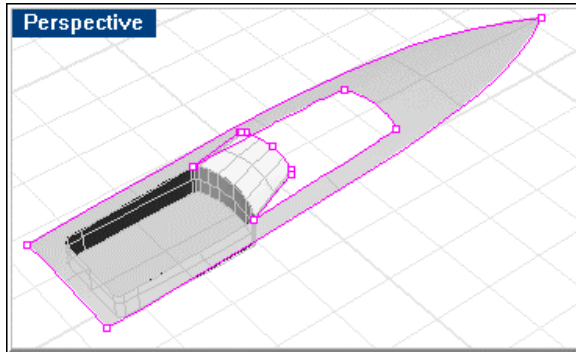
These surfaces butt into a 3/8" wide flat that extends to the cabin side.

Import the plan and profile of the top edge of the aft cabin bulkhead. You will need the top edge, the arc at the corner and the outboard edge of the flat.

Use the **Crv2View** command to create the aft edge of the cabin top (top edge of the aft bulkhead).

Make the arched surfaces using the **Loft** command, selecting curves duplicating the deck edge at the bottom and the appropriate 3-D curve at the top.

Once the first three surfaces of the aft bulkhead have been created, **join** them to the deck and **check** for unjoined edges along the intersections.



Aft cabin bulkhead joined to deck.

Step-by-Step: Create the aft cabin bulkhead

Import the 2-D curves and prepare for lofting

- 1 In **Deck Lines.3dm**, select the curves representing the top of the aft cabin bulkhead in plan and profile.
You will need the top edge, the arc at the corner and the outboard edge of the flat.
- 2 Press **Ctrl+C** to copy the curves to the Clipboard.
- 3 In **Deck.3dm**, press **Ctrl+V** to paste the curves copied to the Clipboard in the previous step.

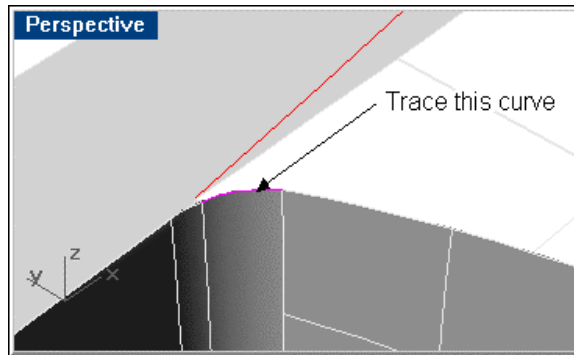
Create the 3-D curves

- 1 Select one of the curves in the plan view and the corresponding curve in the profile.
- 2 Use the **Crv2View** command to create the 3-D curve.
- 3 Repeat the **Crv2View** command for each of the curves (total of 3 curve pairs).
- 4 Use the **Rebuild** command on all the curves to set the **Point count** to **4** and the **Degree** to **3**.

Draw the cockpit front curve

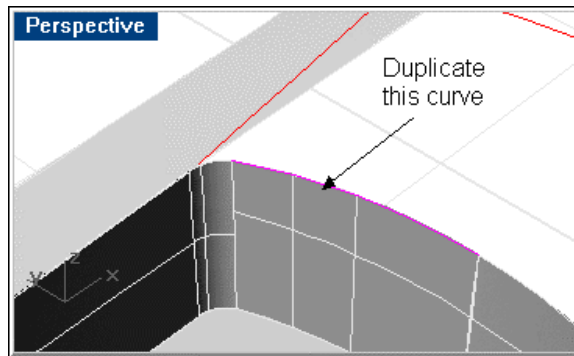
- 1 Start the **InterpCrv** command.
- 2 At the **Point of curve ...** prompts, pick points along the top edge of the cockpit at the forward end.
 - Use the **End** object snap to begin with the outboard tangent of the corner fillet.
 - Use the **Near** object snap to select four points along the top of the fillet.
 - Use the **End** object snap to finish with the inboard tangent of the corner fillet.

You will find an isocurve on the cockpit sides at each tangent to help you locate the points of this curve.



Trace the curve at the corner fillet.

- 3 Use the **DupEdge** command to duplicate the top edge of the forward end of the cockpit.



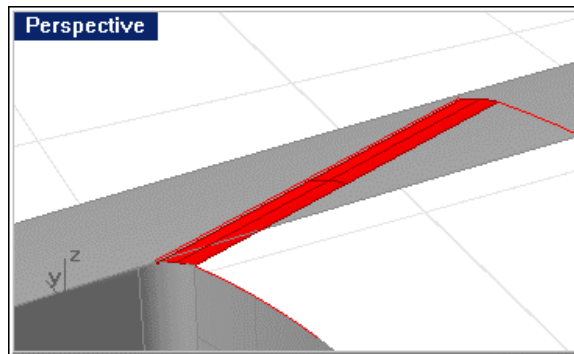
Duplicate the top edge of the cabin.

- 4 Use the **Rebuild** command on both curves to set the **Point count** to **4** and the **Degree** to **3**.

Loft the curved surfaces of the aft bulkhead

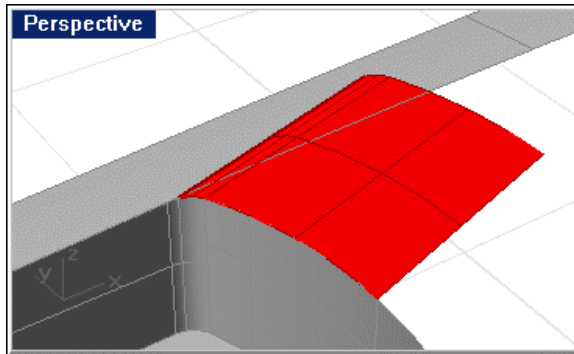
- 1 Select the two curves representing the top and bottom of the corner fillet.
- 2 Start the **Loft** command.
- 3 In the **Loft Options** dialog box, click **OK**.

You may have to start the Loft command before picking the curves and carefully pick the same end of each curve to avoid twisting the surface.



Loft the top and bottom of the corner fillet.

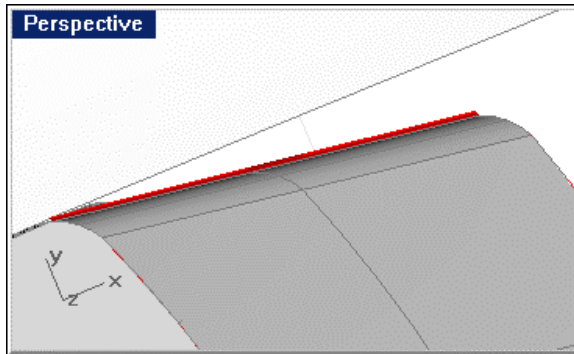
- Repeat the **Loft** for the larger surface of the aft bulkhead.



Loft the aft bulkhead surface.

Create the flat surface

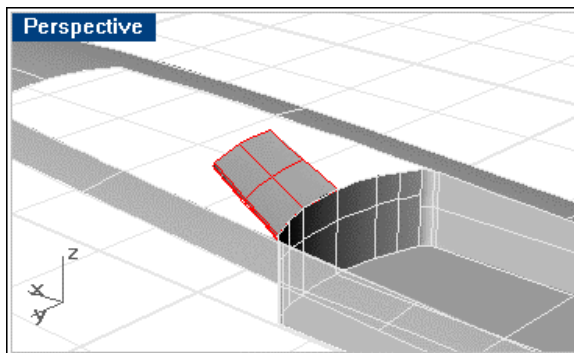
- Start the **EdgeSrf** command.
- At the **Select 2, 3, or 4 curves** prompt, select the outboard edge of the fillet and the straight curve representing the outboard edge of the flat.



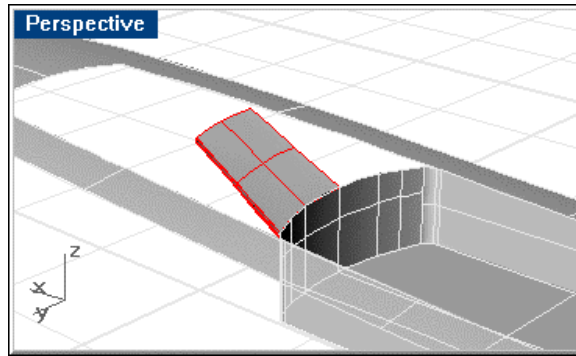
Create the small flat surface.

Extend the lofted surfaces above the cabin top

- Start the **ExtendSrf** command.
- At the **Select edge of surface to extend ...** prompt, select the top edge of the 3/8" surface you just created.
- At the **Extension factor <1>** prompt, type **1**.
- Repeat the **ExtendSrf** command to extend the corner fillet and the large curved surface of the aft bulkhead.

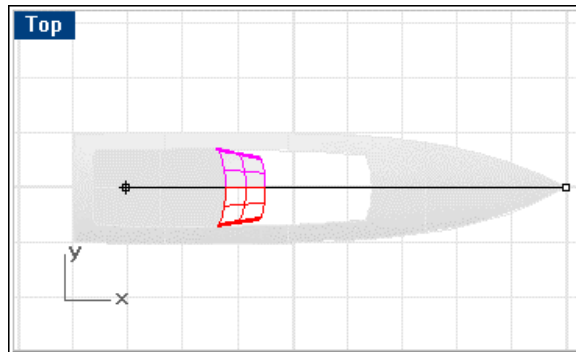


Extend the bulkhead and corner fillet.



Mirror the surfaces

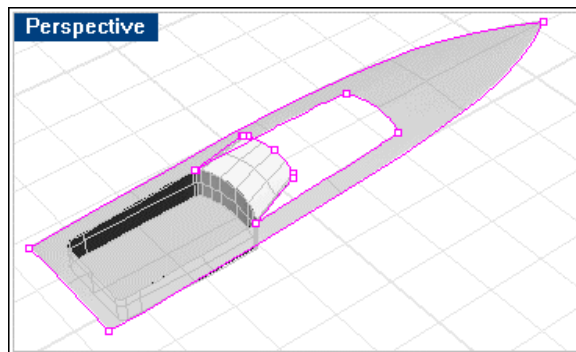
- 1 Select the three surfaces making up the port half of the aft bulkhead.
- 2 Start the **Mirror** command.
- 3 In the **Top** viewport, at the **Start of mirror plane ...** prompt, type **0**.
- 4 At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line to the right along the x-axis.



Mirror the bulkhead surfaces around the centerline.

Join the surfaces

- 1 Select the cockpit/deck polysurface and the six surfaces of the aft bulkhead.
- 2 Use the **Join** command to join all the surfaces into one.
- 3 Use the **ShowEdges** command to check for unjoined edges.



Join the surfaces.

- 4 **Save** your work.

Create the cabin top

Use the same top curve of the aft cabin bulkhead as a camber template to create the cabin top.

To do this, import the cabin top centerline from the lines drawing. **Copy** the cabin top curve. **Copy** the camber curve along the centerline. **Loft** the resulting "deck beams."

Create the cabin top camber curve

In the **Right** viewport, copy the original bulkhead top curve at a point above the model.

Project this curve to the **Right** viewport construction plane.

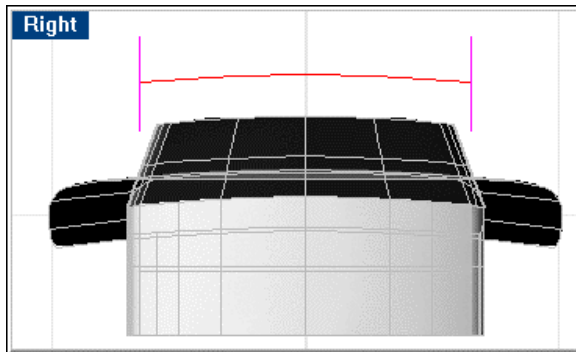
Mirror this curve across centerline.

Create a circle using the **Circle3Pt** command. Pick the outboard endpoint of the port half of the copied bulkhead top, the inboard end of the port half of the copied bulkhead top and outboard end of the starboard half.

Trim this curve to become an arc whose chord length is slightly greater than the width of the cabin side at its widest point and whose midpoint is on centerline.

Rebuild the resulting arc. This will ensure clean geometry.

Note: The locations of the vertical trimming edges shown in the image are not important as long as they are symmetrical about centerline.



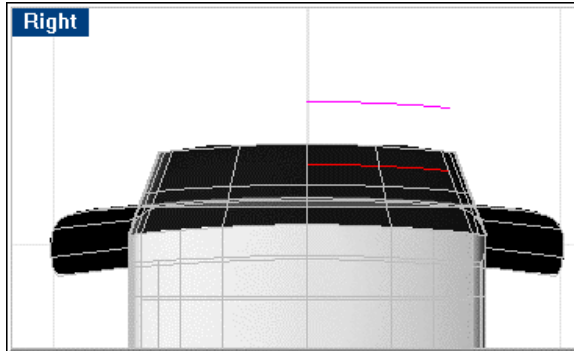
Create cabin top camber curve.

Step-by-Step: Create the cabin top camber curve

Copy the aft bulkhead upper edge and project to CPlane

- 1 Select the original curve that you used to loft the aft cabin bulkhead.
- 2 Start the **Copy** command.
- 3 At the **Point to copy from ...** prompt, in the **Right** viewport, pick a point.

- At the **Point to copy to** prompt, use **Ortho** to pick a point above the first (approx 6-12" higher).



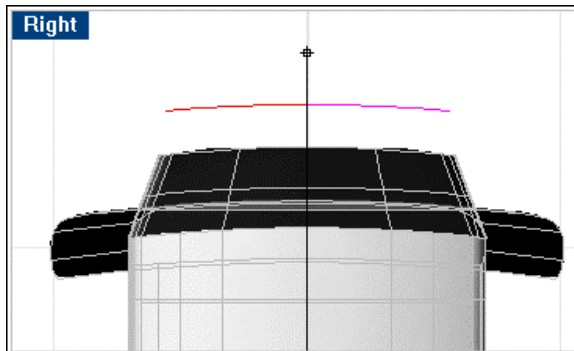
Copy the curve up.

Project the curve to the construction plane

- Select the curve you just copied.
- In the **Right** viewport start the **ProjectToPlane** command.
- In the **ProjectToPlane** dialog box, click **Yes** to delete the input objects.

Mirror the projected curve

- Select the projected curve.
- Start the **Mirror** command.
- In the **Right** viewport at the **Start of mirror plane ...** prompt, type **0**.
- At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line to the right along the x-axis.

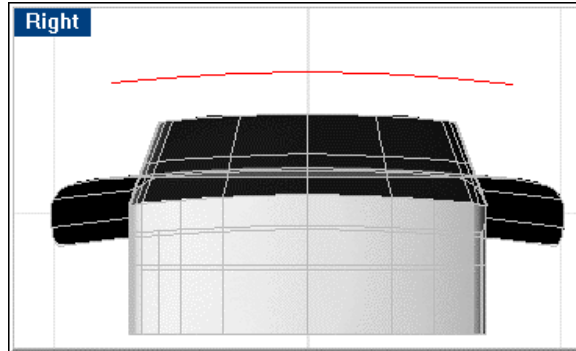


Mirror the projected curve.

Join the two curves and extend them

- Select the two curves.
- Use the **Join** command to make them into one.

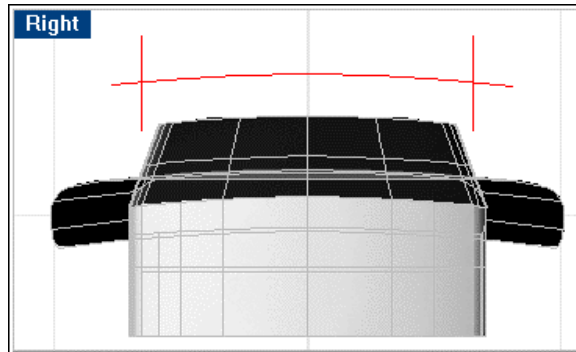
- Use the **Extend** command with the **Arc** option to extend each end of the curve beyond the cabin top sides.



Join the curve halves.

Draw and mirror a trimming line

- Start the **Line** command.
- Draw a vertical line outboard of the port upper corner of the aft cabin bulkhead.
- Select the vertical line.
- Start the **Mirror** command.
- In the **Right** viewport, at the **Start of mirror plane ...** prompt, type **0**.
- At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line up along the z-axis.

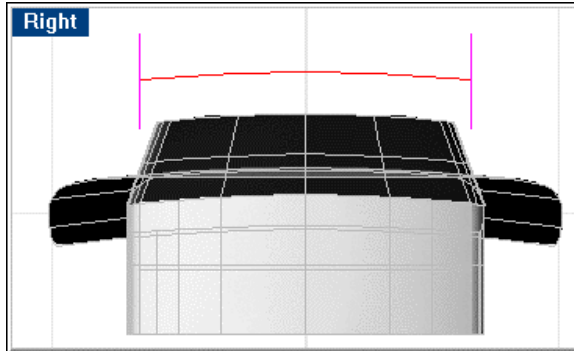


Draw a vertical trimming line and mirror it around the centerline.

Trim the camber curve

- Select the two vertical lines.
- Start the **Trim** command.

- At the **Select object to trim ...** prompt, select the camber curve outboard of the cutting edge on each side.



Trim the curve to the trimming lines.

- Select the camber curve, and use the **Rebuild** command to set the **Point count** to **4** and the **Degree** to **3**.
- Save** your work.

Create the cabin top

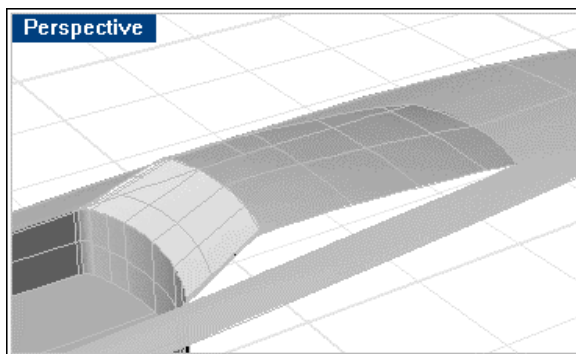
Copy and **paste** the cabin top at centerline from the lines drawing.

Copy the camber curve to three points along the length of this centerline, using a **Mid** object snap to pick the midpoint of the camber curve and a **Near** object snap along the cabin top centerline. Make one additional **copy** of the camber curve at the very aft end of the cabin-top centerline.

Copy and **paste** the cabin side top edge in plan view from the lines drawing. **Mirror** this curve about centerline and use the two resulting curves as trimming edges to **trim** the camber curves to length.

After trimming, **move** the aft camber curve aft approximately 16" using the **Near** object snap on the cabin top centerline. This will extend the cabin top beyond the aft bulkhead and allow it to be trimmed to that bulkhead.

You will recognize this methodology as similar to how the deck was constructed. **Loft** through the camber curves and the forward edge of the deck cut.

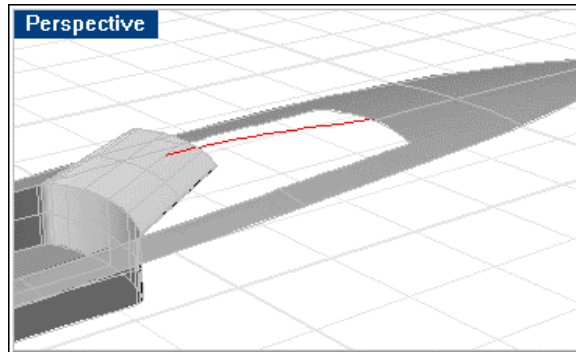


Lofted cabin top.

Step-by-Step: Create the cabin top

Import the 2-D curves

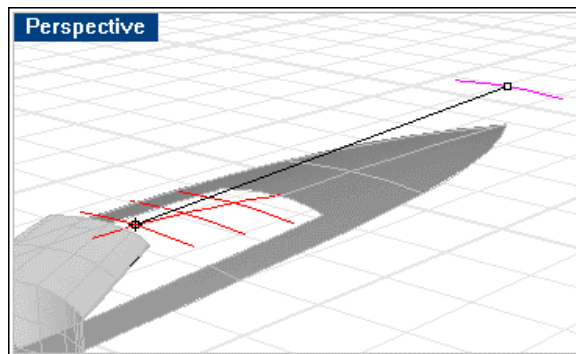
- 1 In **Deck Lines.3dm**, select the curves representing the cabin top centerline in profile.
- 2 Press **Ctrl+C** to copy the curves to the Clipboard.
- 3 In **Deck.3dm**, press **Ctrl+V** to paste the curve.



Copy the cabin top centerline.

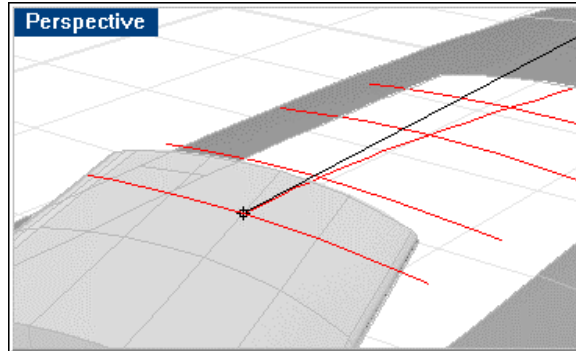
Set up the camber curves

- 1 Select the camber curve.
- 2 Start the **Copy** command.
- 3 At the **Point to copy from ...** prompt, use **Mid** object snap to pick the midpoint of the camber curve.
- 4 At the **Point to copy to** prompt, use the **Near** object snap to pick three points on the cabin top centerline.



Copy the camber curve to the centerline.

- At the **Point to copy to** prompt, use the **End** object snap to pick the aft end of the cabin top centerline.



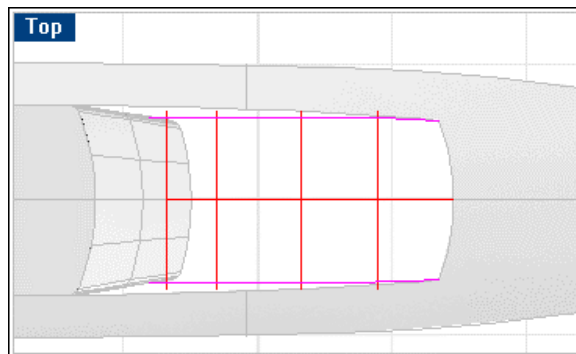
Copy a curve to the end of the centerline curve.

Copy the cabin side curve

- In **Deck Lines.3dm**, select the curve representing the cabin side top edge in plan view.
- Press **Ctrl+C** to copy the curve to the Clipboard.
- In **Deck.3dm**, press **Ctrl+V** to paste the curve.

Mirror the cabin topside curve

- Select the cabin topside curve you just pasted.
- Start the **Mirror** command.
- In the **Top** viewport, at the **Start of mirror plane ...** prompt, type **0**.
- At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane to the right along the x-axis.

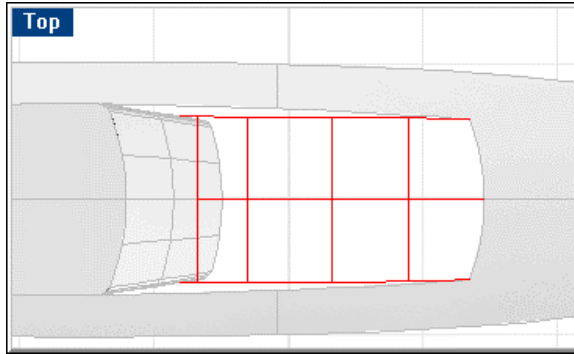


Mirror the cabin topside curve.

Trim the cabin top

- Select the two cabin topside curves.
- Start the **Trim** command.

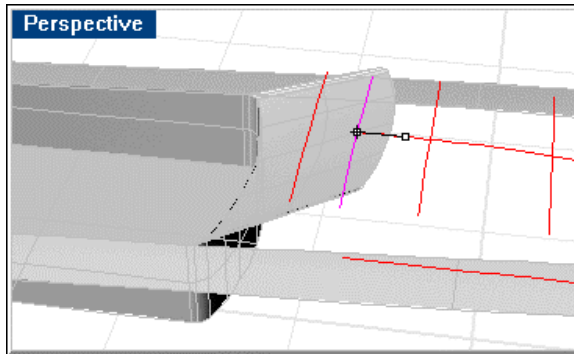
- 3 At the **Select object to trim ...** prompt, in the **Top** viewport, select the outboard ends of each of the camber curves until all curves have been trimmed on both ends.



Trim the camber curves with the cabin topside curves.

Move the aft camber curve

- 1 Select the aft camber curve.
- 2 Start the **Move** command.
- 3 At the **Point to move from ...** prompt, use the **Near** object snap to pick a point on the cabin top centerline approximately 16" forward of the aft end of the cabin top centerline.
- 4 At the **Point to move to** prompt, use the **End** object snap to pick the aft end of the cabin top centerline.

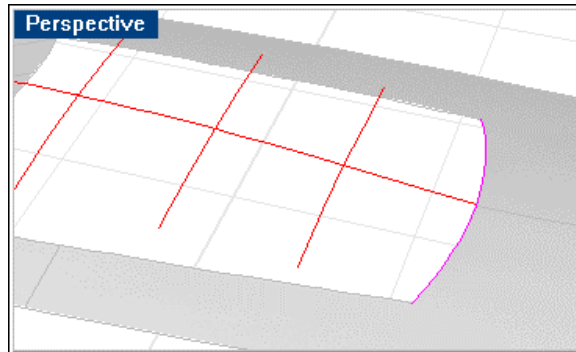


Move the aft camber curve beyond the end of the centerline.

Duplicate the forward deck edge

- 1 Start the **DupEdge** command.

- 2 At the **Select edges to duplicate** prompt, select the edge curve on the forward edge of the deck cut.
If there is more than one edge, select all the forward edges. **Join** the duplicated edges.



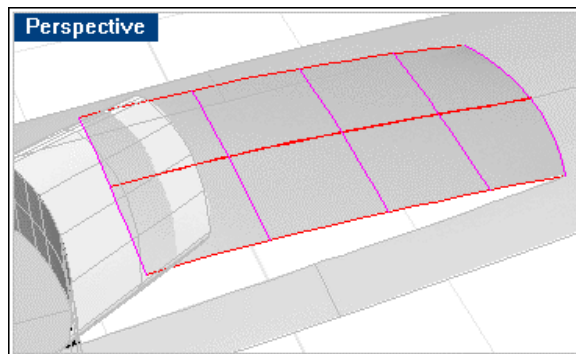
Duplicate the forward edge.

Rebuild the deck curves

- 1 Select the forward deck cut curve you just created and all the camber curves.
- 2 Start the **Rebuild** command.
- 3 In the **Rebuild Curve** dialog box, set the **Point count** to **6** and the **Degree** to **5**.

Loft the cabin top

- 1 Select the rebuilt curves.
- 2 Start the **Loft** command.
- 3 In the **Loft Options** dialog box, specify **Normal** style, and do not simplify the surface.

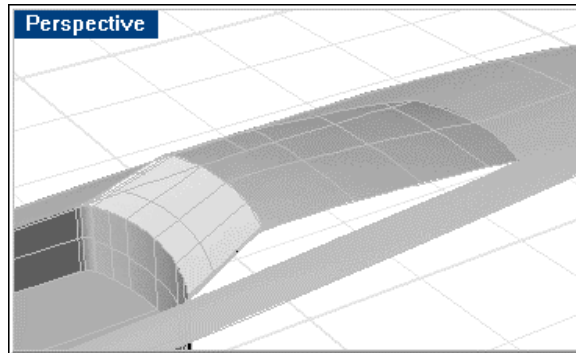


Loft the cabin top.

Trim the aft cabin top and bulkhead

- 1 Select the top of the aft cabin bulkhead and the cabin top.
- 2 Start the **Trim** command.
- 3 At the **Select object to trim ...** prompt, select the aft end of the cabin top.

- 4 At the next **Select object to trim ...** prompt, select the top of the aft cabin bulkhead.



Trim the cabin top.

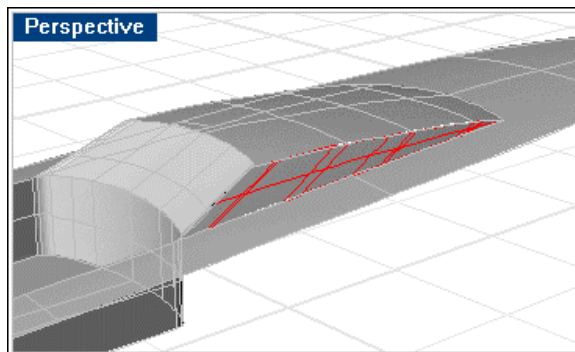
- 5 **Save** your work.

Create the cabin sides

With the upper and lower edges of the cabin sides completely defined, the sides can be constructed. To do this, use the **EdgeSrf** command, selecting the three edges of the cabin side.

The resulting surface can be checked by **joining** to its neighboring surfaces and **checking** for unjoined edges.

If none are found, **undo** the join, **mirror** the surface, and **join** both cabin sides into the deck/cabin/cockpit polysurface.



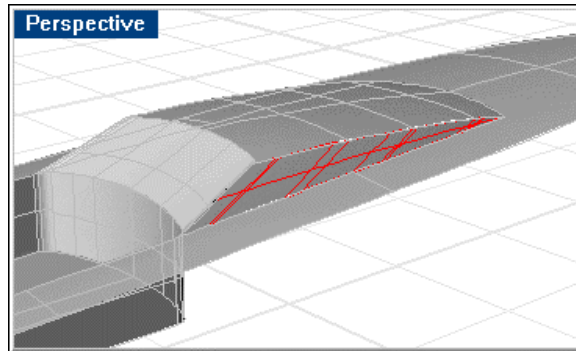
The cabin sides.

Step-by-Step: Create the cabin sides

Create the cabin side

- 1 Start the **EdgeSrf** command.

- 2 Select the edge of the cabin top, the outboard edge of the aft cabin bulkhead, and the edge of the deck cut.



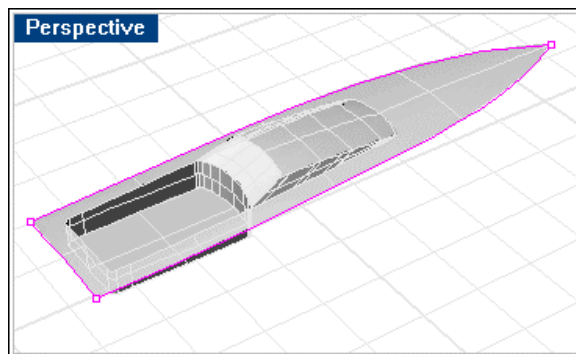
Cabin sides.

Mirror the cabin side

- 1 Select the cabin side.
- 2 Start the **Mirror** command.
- 3 At the **Start of mirror plane ...** prompt, in the **Top** viewport, type **0**.
- 4 At the **End of mirror plane ...** prompt, use **Ortho** to drag the mirror plane line along the x-axis.

Join the cabin and deck surfaces

- 1 Select the cabin and deck surfaces.
- 2 Use the **Join** command to join all the surfaces into one polysurface.
- 3 Select the cabin/deck assembly.
- 4 Use the **ShowEdges** command to check for unjoined edges.

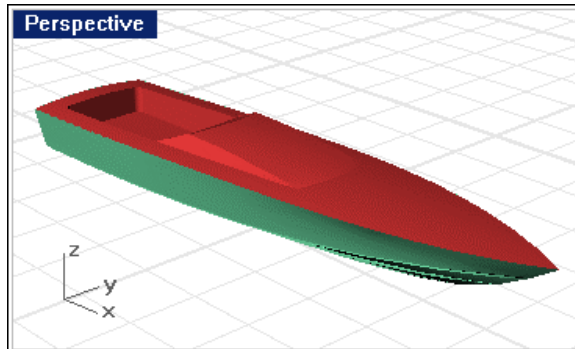


Finished deck model.

- 5 **Save** your work.

Copy the cabin/deck assembly to the original model

Now that you have finished the deck assembly, you can copy and paste it into the original model and replace the deck in that model, similar to the way you handled the running strakes.



Cabin/deck assembly exported to hull and joined.

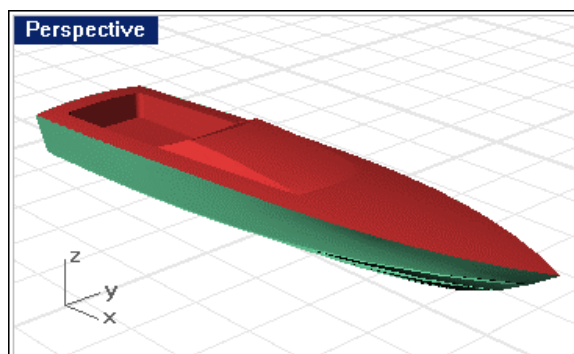
Step-by-Step: Export the cabin/deck assembly to the original model

Copy the cabin/deck assembly to the final model

- 1 In **Deck.3dm**, select the cabin/deck assembly.
- 2 Press **Ctrl+C** to copy the curves to the Clipboard.
- 3 In **Offshore.3dm**, use the **ExtractSrf** command to extract the deck surface.
- 4 At the **Select surfaces to extract ...** prompt, select the deck surface.
- 5 **Delete** the deck surface.
- 6 Press **Ctrl+V** to paste the contents of the Clipboard (the cabin/deck assembly).

Join the cabin/deck assembly to the hull

- 1 Select the hull and cabin/deck assembly.
- 2 Use the **Join** command to join the hull and deck assembly into one polysurface.



Cabin/deck assembly exported to hull and joined.

Check for unjoined edges

- 1 Select the vessel.
- 2 Use the **ShowEdges** command to check for unjoined edges.
There should be no unjoined edges.
- 3 **Save** all models.

Part V: 165-Foot Motor Yacht Hull



Introduction

This tutorial will introduce you to the methods needed to create a fair hull from a designer's lines drawing. You will import the curves needed from the lines into an empty .3dm file, fair the hull and use Rhino analysis tools and your own trained eye to be sure the hull is fair.

When the hull is finished, you will have joined both sides, added a deck and complete superstructure and checked to be sure the completed assembly is suitable for export to other programs, as well as further analysis in Rhino, should that be necessary.

The model created for this tutorial is included on the distribution CD. You can work from it or create your own while performing the tasks. To gain maximum benefit, the latter is recommended.

This tutorial assumes you have completed three of the other four tutorials on the distribution CD (Sail, Offshore, and Skiff). If you have not done so, you are urged to complete them, before starting this tutorial. Descriptions of some operations assumes you have completed these tutorials.

The vessel

The vessel to be modeled in this tutorial is similar in form to a mega-yacht. It should be noted this is not a tutorial in boat design. Most mega-yacht designers will find fault in the design of this vessel. However, the purpose of this tutorial is to instruct in the art of fairing hulls and superstructures using **Rhinoceros** modeling techniques

The files

The that make up this tutorial include:

Yacht Hull Finished.3dm – The finished hull.

Yacht Hull Lines.3dm – 2-D lines drawing from which the hull will be lofted.

Prepare the curves from the designer's lines

Open **Yacht Lines.3dm** and start a new model called **Yacht Hull.3dm**.

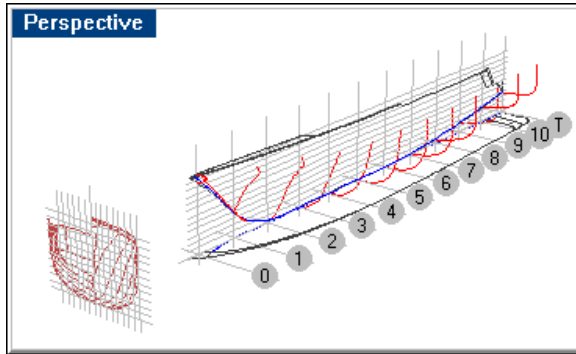
In **Yacht Lines.3dm**, orient each view as it would be as if projected from the hull to the appropriate viewing plane as follows:

Rotate the profile and body plan curves 90 degrees about the profile baseline in the **Right** viewport.

Rotate the body plan curves in the **Top** viewport 90 degrees about the body plan centerline.

Move the plan view curves in the **Top** viewport so they share their centerline with the baseline of the profile.

Copy the stations from the body plan to the appropriate station ordinate in the profile (except stations 10, 8, and 6).

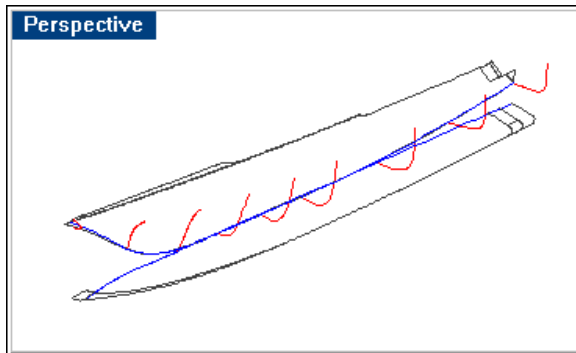


Lines drawing oriented for exporting curves.

Copy the required 2-D curves to the new model

Turn off all layers, except **cStas**, **cRabbet**, and **cEdges**.

With layers **cStas**, **cRabbet**, and **cEdges** on, in the **Perspective** viewport select the curves shown.



Curves to export.

Observe these notes when exporting these curves to your model:

- The station curves are made up of several sub-curves. You should explode before copy/pasting to the other model, since you will only need the station curves, not the whaleback, deadflat or guard strake curves.
- Rabbet and stem wrapper tangent are created on layer **cRabbet** to distinguish them from the centerline curves, which are quite close in profile.
- Because station 10 crosses bulwark corner, do not use it for lofting hull surface.

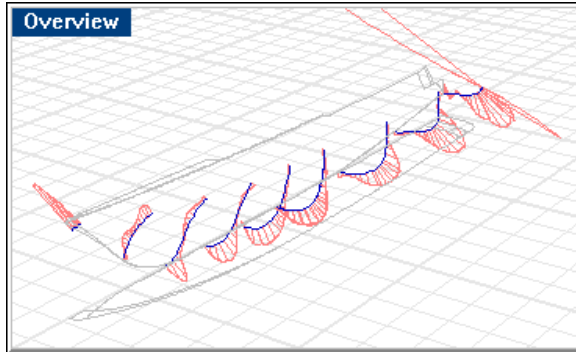
Press **Ctrl+C** to copy the highlighted curves to the Clipboard.

In **Yacht Hull.3dm** press **Ctrl+V** to paste the contents of the Clipboard into the current model.

Save your work.

Fair the two-dimensional curves

Since you did not personally produce these lines, it is necessary to check the fairness of these curves before basing your modeling work on them. Use the **CurvatureGraph** command to display the curvature graph of the stations.



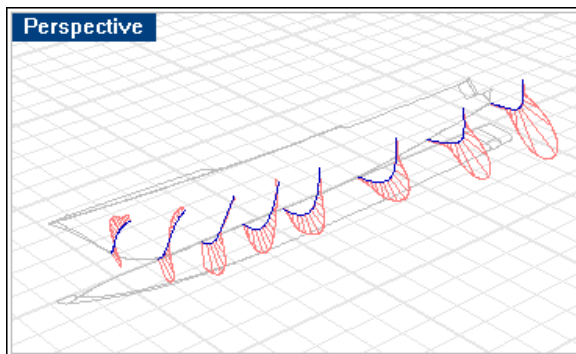
Station curves before editing control points.

The graphs indicate a little work is needed on each of the stations, near the sheer and rabbet, since there are severe changes in direction of the curvature graphs in these regions. This can be corrected by editing the control points in these regions.

Rebuild the station curves so that their control points are distributed evenly over the length of the curves. In the **Rebuild Curve** dialog box, set the **Point count** to **10** and the **Degree** to **5**.

Rebuilding the curves will cause any "corners" to become rounded. You should have already **exploded** the station curves. Since you will only be lofting the stations between the rabbet and the sheer, and since there are no knuckles in this region, **delete** the portions above the sheer and below the rabbet, since you will have no use for these curves.

Once you have completed fairing the stations, repeat the process with the 2-D curves in both the profile and plan views.



Station curves after editing control points.

Interpreting the curvature graph

If you look at the aft station, you will note that before fairing, the top and bottom of the curve have points where the **curvature graph** crosses the station several times in a short span. This tells you that the curve is "wiggling" in this area. After fairing, the **curvature graph** begins to depart from the station in a gradual manner and continues away from the station until it passes the turn of the bilge and then begins a gradual approach back to the station. It finally meets the station at the rabbet.

This indicates that the curvature of this station gradually increases from the sheer to the turn of the bilge then gradually decreases from there to the rabbet. This station is as smooth as you can get it. It may not represent the shape you had in mind, but it is fair. It is up to you to reconcile the difference between what you (or the designer) want and what is fair.

If you view the forward several stations after fairing, you note that the **curvature graph** crosses the station once. This indicates that the station is concave above this point and convex below it. As we know, this is a normal condition for a bow in this type of vessel and no particular cause for concern.

The forward station has a very wild **curvature graph**. This is because it is a very short span and built with many points (normally you would use fewer points, but in this case, you used a standard number of points for all stations when rebuilding). This being the case, the **curvature graph** will swing wildly.

To view this graph reasonably, set the **Display Scale** value in the **Curvature Graph Options** dialog box to about **112**.

Create the three-dimensional stem wrapper tangent

In some projects (the Offshore model, for example), it is necessary to create several 3-D curves from pairs of 2-D curves. In this model, it will only be necessary to create a few curves in this manner. The first example of this is the stem wrapper tangent. It will be used in creating the phantom station in the next section.

Select the plan and profile views of the wrapper tangent imported from the designer's lines.

Use the **Crv2Views** command to create the 3-D representation of this curve.

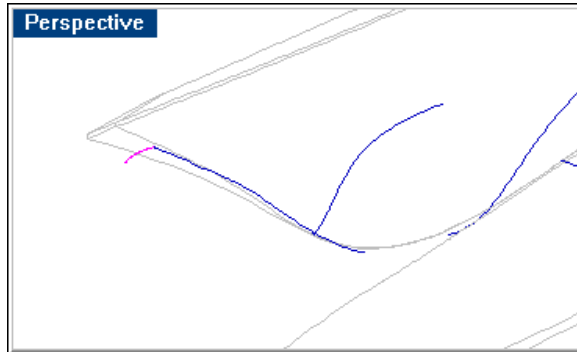


Stem wrapper tangent.

After completing this command, you should **hide** the 2-D curves to avoid inadvertently selecting them.

Construct the phantom station

Copy the forward station (station zero) to the upper end of the 3-D stem wrapper tangent you just created.



Copy the forward station to the stem wrapper tangent.

With the phantom station in place, **extend** all the other stations at the bottom so that they intersect the centerline (except station zero – extend it approximately 3" below the wrapper tangent).

The reason for extending these curves is so that the lofted surface will be entirely inboard of the stem wrapper / forefoot / rabbet over its entire length. There is no need to extend the phantom station. You will soon trim it at the bottom.

Use the **InterpCrv** command to draw the temporary inboard edge of the hull starting at the stern and moving forward. This curve connects the lower ends of all the stations. Use an **End** object snap on all the stations except the phantom station. Use a **Near** object snap when connecting this new curve to the phantom station.

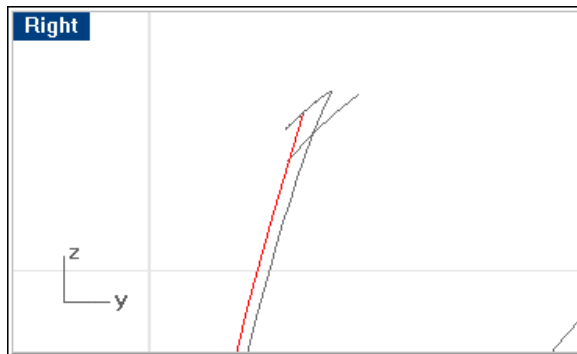
When making this last pick, "slide" the cursor up and down the phantom station in the **Right** viewport until the shape of the curve you are creating is pleasing to your eye.

To make this last pick easier, consider making another **Right** viewport. Leave this viewport zoomed out so you can see the entire body plan. Zoom in close on the actual **Right** viewport. In this way, you can get a close look at the pick while still seeing an overview of the body plan.

The exact shape of this temporary inboard edge of the hull is not important. It should make a gradual transition from centerline outboard and remains inboard of the rabbet / forefoot / stem wrapper tangent over its entire length. Once you have made the last pick on the phantom station, complete the curve.

Use the curve to **trim** the bottom of the phantom station.

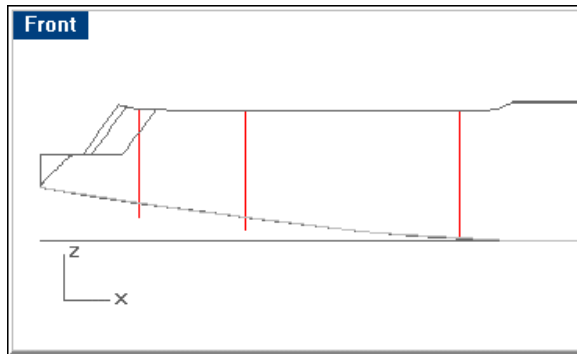
Delete the temporary inboard curve. You will have no further use for it.



Temporary inboard curve at phantom station.

Extend the station tops aft

The aft stations are drawn to the lowered shear.



Aft stations meet lowered shear.

To create the hull surface, you must extend the stations to a projected shear and use these extended stations for lofting the hull.

This will create one smooth, continuous edge for the hull surface. After the hull surface has been lofted and faired, it will be trimmed back to the lowered shear.

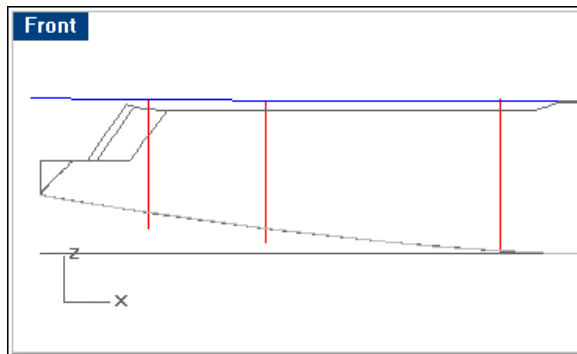
Use the **InterpCrv** command to draw a 2-D curve in the profile.

You can copy the lowered shear up one foot and use it as a template over which to draw the extended shear.

This curve can be drawn with three picks: one at the aft end of the forward shear, one above the top of the transom, and one aft of the transom.

Use the **Match** command with the **Curvature** option to force the new curve to flow smoothly from the forward shear.

Extend the stations to meet the extended shear.



Extend aft stations to projected shear.

Loft the hull surface

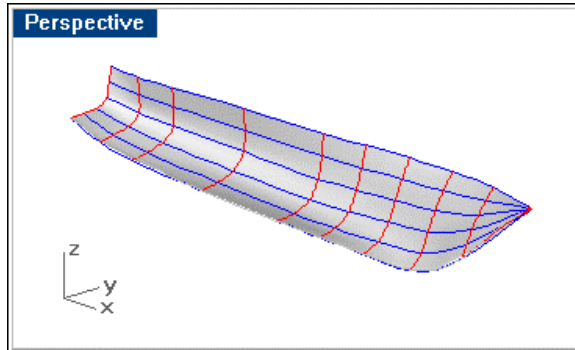
Now that you have created and faired a set of stations and extended their upper and lower ends so that they describe continuous edges, **loft** the hull surface.

Make the **sHull** layer current.

Select the faired stations including the phantom station.

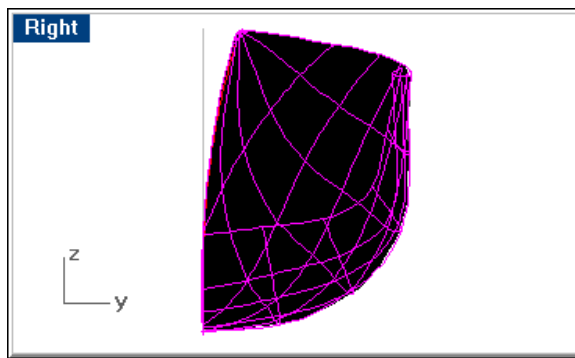
Use the **Loft** command to create the hull surface

If you get a twisted surface, undo the loft and restart the **Loft** command. Select the stations one at a time, making sure you select the same end of each curve.



Lofted hull surface.

This completes the hull surface.



The hull crosses the rabbet for its entire length.

The reason this geometry is so simple is that you rebuilt your curves before you performed the loft. The simplicity of these surfaces is going to make it extremely easy to fair this hull.

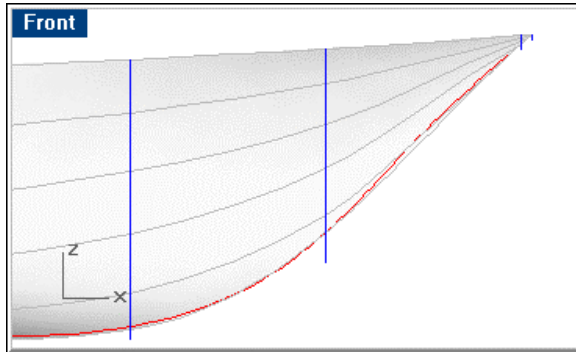
Trim to the rabbet / forefoot / stem wrapper

Trim the surface to the rabbet, forefoot, and stem wrapper tangent, causing the inboard edge of the hull surface to resemble the designer's profile.

Use the 2-D rabbet and stem wrapper tangent as the trimming curves in the **Top** viewport.

These curves need to be extended beyond their respective intersections with the hull in order to make good cutting edges.

Since you did not use the profile in your controlling geometry and since you did the trimming in the plan view, how close will the resemblance to the profile be?

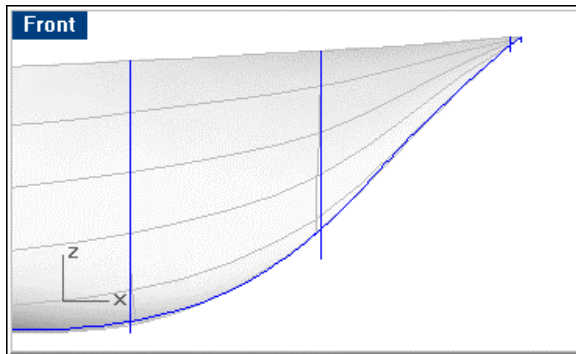


Profile after trimming.

The trimmed surface does not match the profile (shown in red). This surface will require further modification.

To fix this, **nudge** the surface control points until the edge of the surface lies on the red curve. Make these changes in the **Front** viewport to avoid inadvertently changing the y-coordinates of the surface control points.

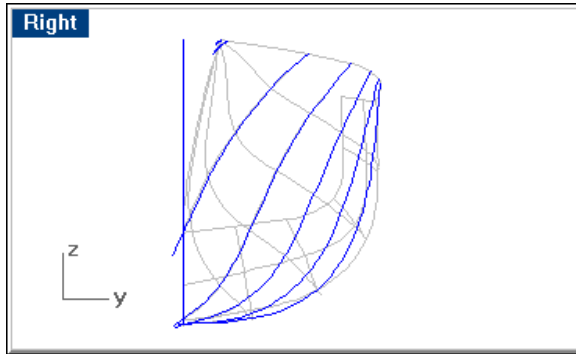
How closely you match this curve is a matter of discretion. Each time you move a point the isocurve corresponding to the nearest station will change. Since you have the stations "behind" the isocurves, you can see how much you are deviating from the designer's stations.



Forefoot condition corrected.

The control points nearest stations 1 and 2 were moved aft and forward to fill in the forefoot there. This will mean that comparisons with stations 1 and 2 are no longer valid, since the isocurves are no longer in the planes of the stations.

What about the body plan? How closely does your lofted surface fit the stations you inherited from the lines drawing?



Lofted surface isocurves and stations.

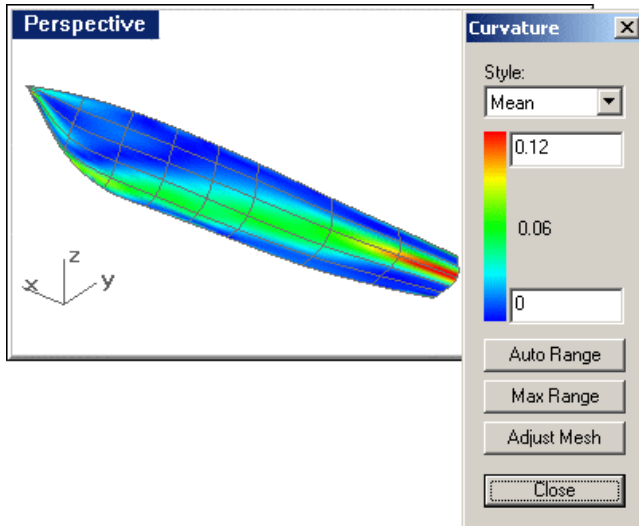
As an alternative method for shaping the forefoot, use the **InsertKnot** command to insert a knot in the area where the edge of the surface least resembles the forefoot. Editing the points on this row will keep them in a transverse plane.

The disadvantage to this method is that the isocurves will no longer coincide with the stations.

In most cases, the two station curves and the isocurves are indistinguishable. The original intent of the stations is achieved.

Analyze surface fairness

Now that the hull surface looks a little more like a hull, use the **CurvatureAnalysis** command to check the fairness.



Curvature Analysis.

Since the progressions of color are gradual, the surface exhibits an acceptable degree of fairness. The red patches are areas of high curvature, while the blue signifies very subtle curvature. Colors that blend from one to the next without sharp edges indicate fairness.

There is no such thing as absolutely fair. You could spend years getting the surface perfectly fair, without ever achieving success. The aim is to assure yourself that the finished product will be pleasing to the eye, no matter what angle the surface is viewed from.

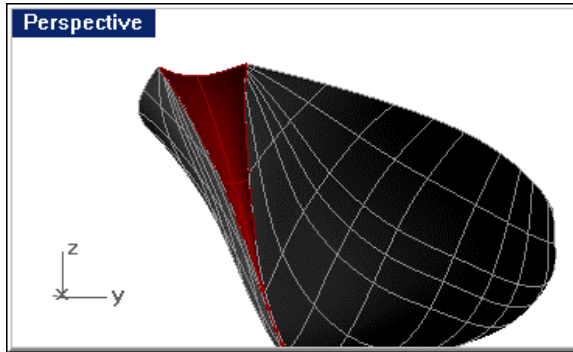
Create keel flat, forefoot, and stem wrapper

Mirror the hull surface across centerline.

Use the **BlendSrf** command between the edge curves of the two halves.

This will create a surface from stem to stern.

Trim this surface away aft of the temporary stem wrapper bottom (the top of the forefoot).

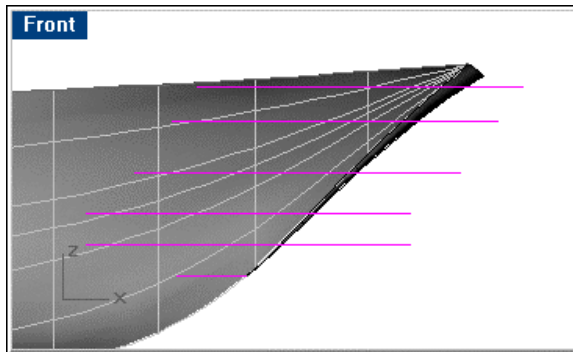


Stem wrapper.

Notice top edge of this temporary stem wrapper. It does not extend to the sheer, as you would expect.

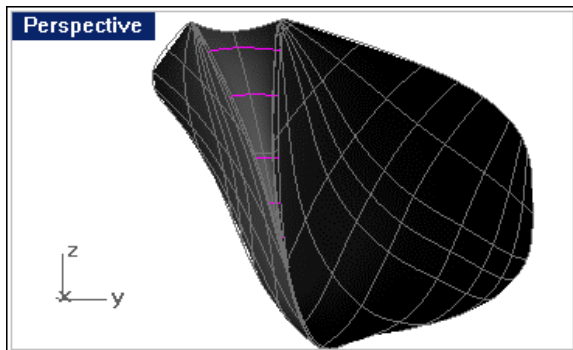
Extend the stem wrapper to the sheer

Draw a few horizontal (waterline) lines in the **Front** viewport along the length of the temporary stem wrapper.



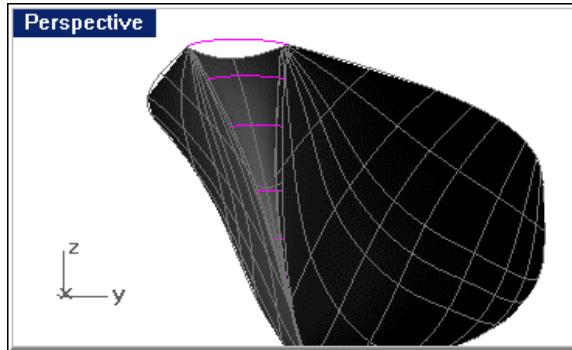
Waterlines drawn through temporary stem wrapper.

Project the lines to the temporary stem wrapper.



Waterlines projected to temporary stem wrapper.

Use the **Crv2View** command to create the 3-D stem wrapper top from the 2-D lines imported earlier.



Stem wrapper top edge.

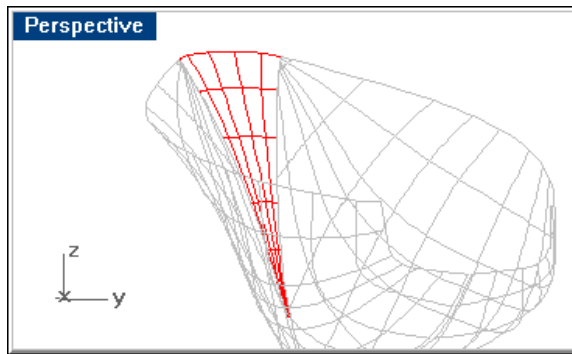
Use the **DupEdge** command on the bottom of the stem wrapper to create a very small curve.

Delete the temporary stem wrapper.

Rebuild the curves you just created.

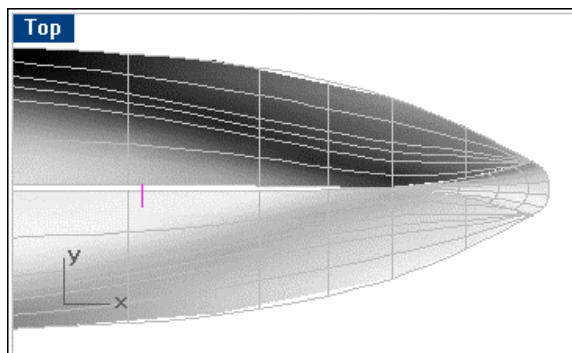
Select these curves and use the **Loft** command to create a new stem wrapper.

You can check the fairness of this surface by drawing a few horizontal lines in the profile and projecting them to the hull and stem wrapper surfaces. The joint between the resulting waterlines should be extremely smooth.



Stem wrapper lofted to upper edge.

Draw a **line** in the plan view from the port rabbet at the bottom of the forefoot to a point beyond the starboard rabbet (be sure to activate **Ortho** and **Planar** before drawing this curve, and deactivate object snaps before making the second pick).



Draw a horizontal line for the aft edge of the forefoot and trim to the rabbet.

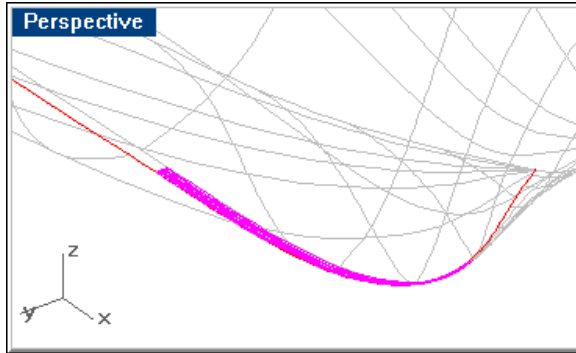
Trim this line to the starboard rabbet.

You should now have a horizontal line at the bottom of the forefoot. The exact location along the rabbet of this line is not critical.

Use the **NetworkSrf** command to create the surface between the two halves of the hull at the forefoot.

Select the port and starboard rabbet surface edges, the bottom of the stem wrapper surface and the curve you just drew.

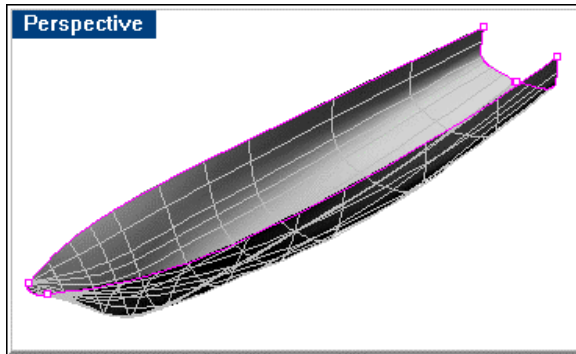
This surface will transition smoothly from the full roundness of the stem wrapper to the flat sections of the keel flat.



Use NetworkSrf to create the forefoot.

Use the **EdgeSrf** command to create a flat surface between the two halves that extends from the aft end of the forefoot all the way to the aft end of the hull.

Join all the surfaces and **check** for naked edges.



Keel flat, forefoot, and stem wrapper joined.

Once you have performed this check and corrected any naked edges, create a **plane** on centerline that extends from above and forward of the hull to a point below and aft of the hull.

Use this plane to **trim** the starboard half of the hull away.

Since you will be doing work that applies to both halves of the hull, it pays not to have to do it twice. Having only half of the hull present will simplify future modeling tasks.

Once you have trimmed away the starboard half of the hull, **explode** the remaining surface, since you will be doing some trimming to the remaining surfaces. Trimming surfaces is easier than trimming polysurfaces because surfaces can be trimmed with curves and polysurfaces cannot.

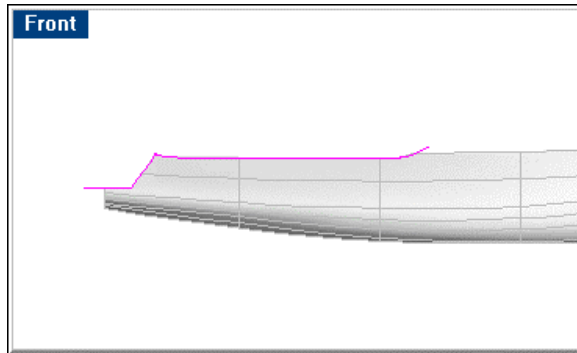
Trim the aft shear

Now that the hull surface has been faired, the shear aft needs attention. Initially, you extended the upper shear from station 5 to the transom to become one continuous edge. It is now time to trim in the transom, swim step, and shear break from station 5 aft.

Turn on layer **cEdges**.

As in most trimming operations, it will probably be necessary to extend the cutting edges beyond the edges of the surface being trimmed. This ensures the cutting edge will intersect the surface edge at both ends and result in a more reliable trimming operation.

With the swim step extended aft of the transom and aft shear transition curve extended above the shear, in the **Front** viewport, **trim** the hull surface as shown.



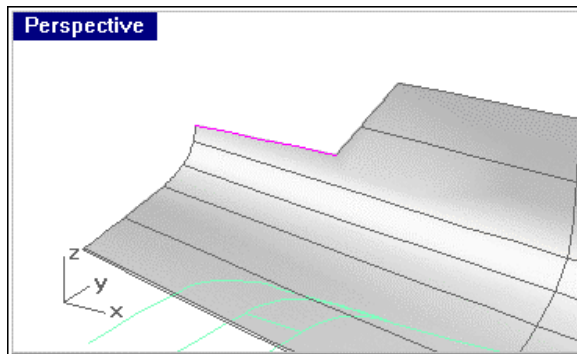
Trim the hull to the profile curves aft.

After trimming, **join** the hull surfaces back together.

Create the swim step transom chine and trim the hull

The intersection of the swim step transom with the hull will generate a chine. Since this is a visible curve, it is important that it be smooth. For this reason and because the surface it bounds must meet the hull tangentially, it will be done iteratively.

The first step is to use the **DupEdge** command to duplicate the edge of the hull surface representing the swim step side.

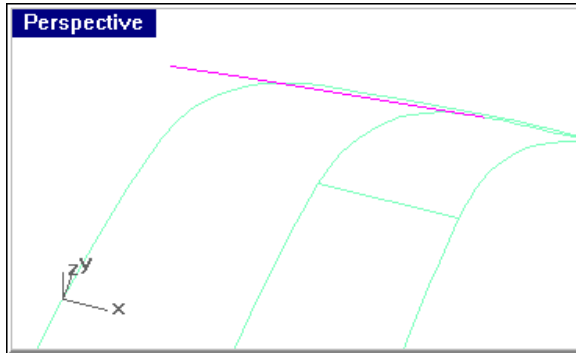


Duplicate swim step side.

In the **Top** viewport, use the **ProjectToPlane** command to project the curve to the construction plane.

The reason for not using the designer's swim step transom corner is that you may have tweaked the surface's points in this area. If so, the fillet supplied by the designer will no longer meet the surface tangentially.

Trim away the designer's transom plane curve.



Projected side curve.

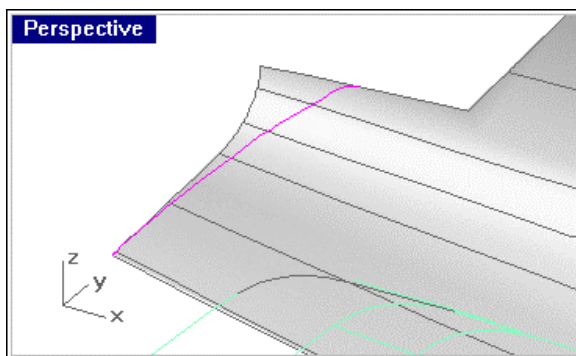
Fillet or **Blend** the remaining curves to ensure tangency.



Designer's transom blended to the projected swim step side curve.

If the flexibility of your situation allows, consider using **Blend**, instead of **Fillet**. **Blend** lets you draw a curve that is curvature continuous, instead of being just tangent. If you can prevail on the designer to allow this type of substitution, the project will benefit from the difference.

In the **Top** viewport, **project** the swim step transom corner and swim step aft edge to the hull surface.



Swim step transom corner and aft edge projected to hull.

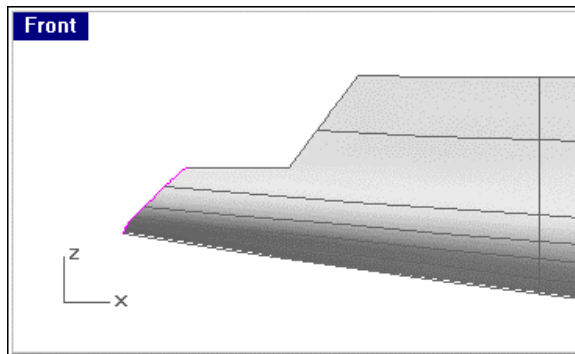
Inspect the result in profile and body plan.

If the curve is not satisfactory, tweak its control points until it is smooth. Then use the **ProjectToCPlane** command to project it to the construction plane of the view in which it is most unfair.

Tweak the points (except the end points) in that view and project it back to the surface again.

Continue this process until it is fair in all three views.

Once you have a curve that is satisfactory in all three views, **trim** the hull surface below the swim step to this curve.

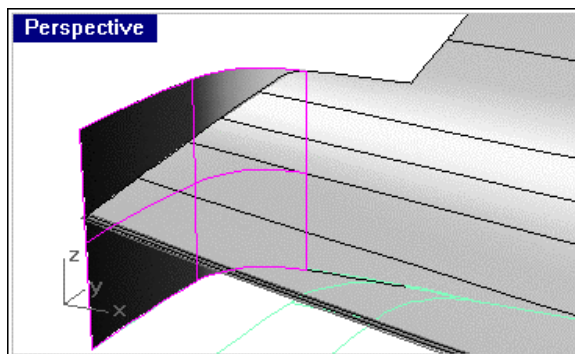


Trim the hull below the swim step.

Create the swim step transom

Select the 2-D curve representing the aft edge of the swim step/transom chine and the curve representing the swim step transom corner.

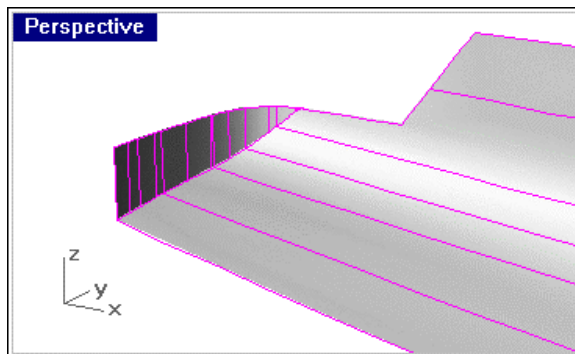
Use the **Extrude** command, specifying a vertical direction and a distance equal to the height of the swim step top as shown.



Extrude the transom curve from the base plane to the height of the swim step.

Trim the extruded swim step transom to the hull.

Join the transom to the hull assembly and **check** for unjoined edges.



Trim swim step.

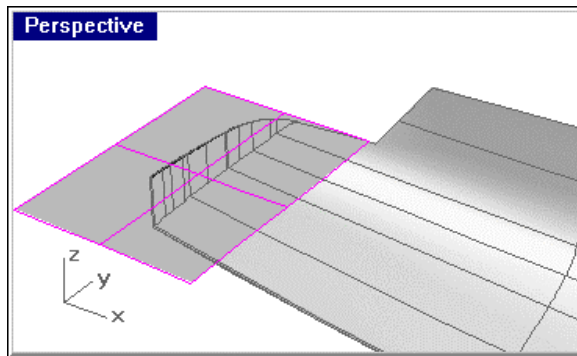
If the transom will not trim to the hull or vice versa, select both surfaces and use the **Intersect** command to create a curve representing the intersection of the two surfaces. **Extend** both ends of this curve by a small amount and use it for a trimming edge for both surfaces. Once you have done so, you should be able to **join** the transom to the hull.

Create the swim step

Turn on **Planar** and **Ortho** mode.

Use the **Plane** command in the **Top** viewport.

As the start point of the plane, select the forward end of the swim step outboard edge. For the second point, pick a point aft of the hull. Be sure this plane crosses centerline.

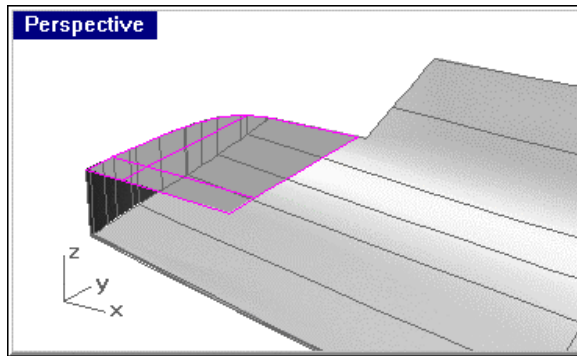


Swim step top surface.

Draw a **line** on centerline in the **Top** viewport from forward of this plane to aft of it.

Use this line to **trim** the plane to centerline and then **delete** the line.

Trim the outboard edge of the swim step top to the hull surface and the aft end of the swim step to the swim step transom.

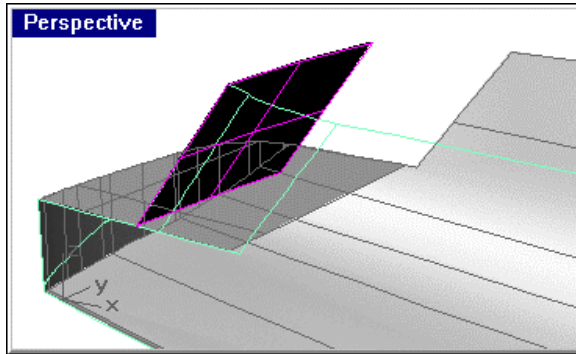


Trimmed swim step top.

Create the transom bulwark

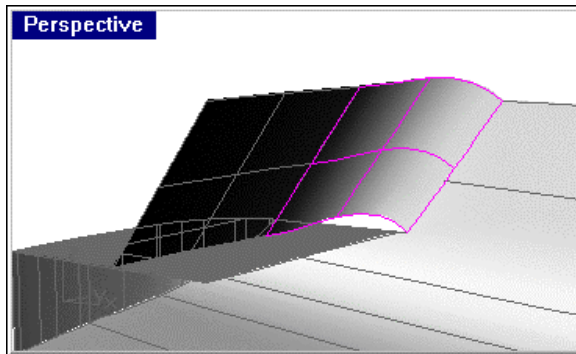
Project the transom bulwark bottom edge from the designer's lines onto the swim step.

Using the transom bulwark centerline curve as a direction vector, **extrude** the result to the top of the transom bulwark centerline as shown.



Project bottom of transom bulwark curve to swim step and extrude.

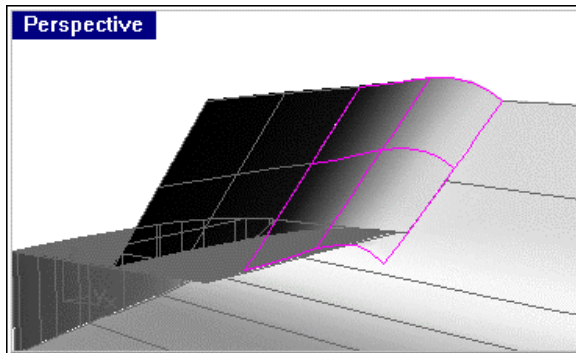
Use the **BlendSrf** command to blend the resulting surface with the transom corner forward tangent as trimmed into the hull surface and shown.



Create the bulwark corner with the BlendSrf command.

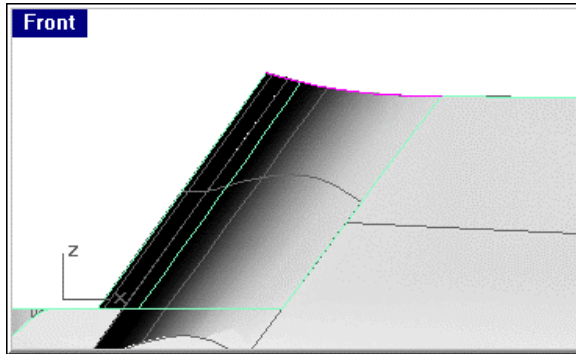
The top and bottom edges of this corner surface will not be quite correct.

To fix this, **extend** the bottom downward 2 feet to be sure the lower edge is entirely below the swim step.



Extend surface below swim step.

In the **Front** viewport, **trim** the top of this corner surface and the transom surface to the profile curve of the transom sheer as shown.

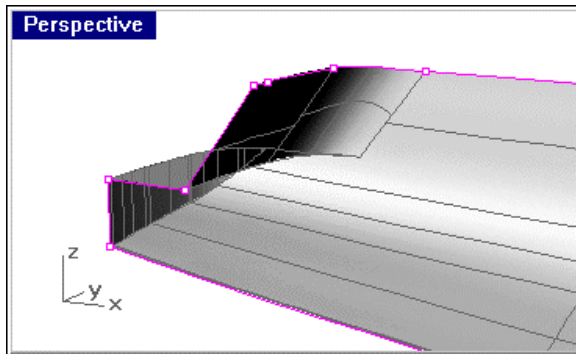


Trim the tops of the transom bulwark and corner to the designer's profile.

Trim the bottom of this surface to the swim step.

Trim the swim step forward edge to the transom bulwark and corner.

Join this bulwark and swim step to the hull and **check** for naked edges.



Naked edges after joining transom, transom bulwark and swim step.

Add the whaleback and sheer strake

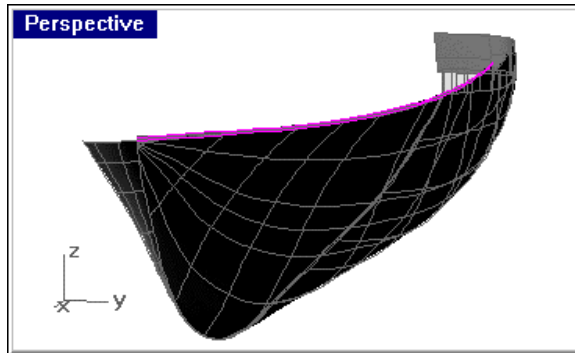
Use the **DupEdge** command to copy the edge of the hull surface at the sheer forward of station 5.

Rebuild this curve to simplify the geometry. Inspect the results to be sure they are faithful to the designer's curves.

Copy this curve vertically 3 inches.

Loft the resulting curves.

This creates the sheer strake.

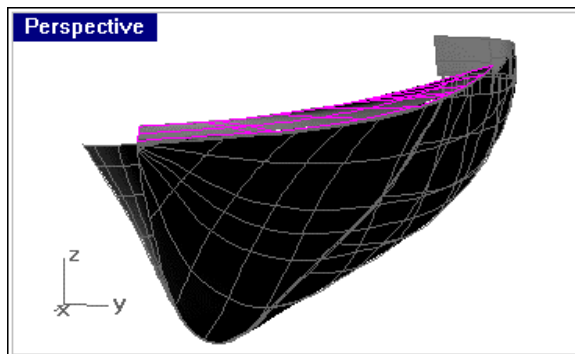


Sheer strake.

Create the whaleback shear

Use the **Crv2View** command using the curves that describe the whaleback shear.

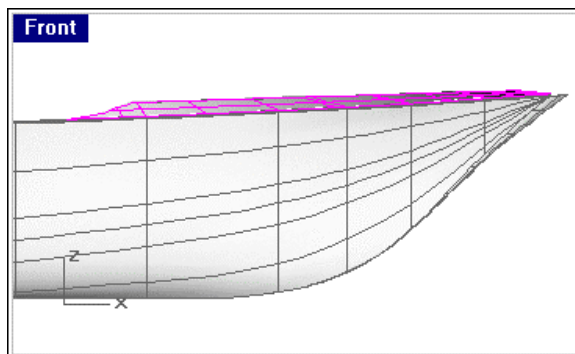
Hide the 2-D curves.



Whaleback.

Loft the whaleback using the upper edge of the sheer strake and 3-D whaleback shear. This will give you a surface whose aft edge is straight.

Trim the surface to the forward transition curve shown in the profile of the lines model.

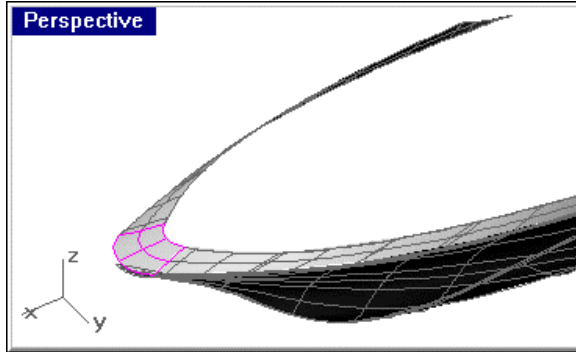


Trimmed whaleback.

Create the whaleback stem wrapper

Mirror the whaleback.

Use the **BlendSrf** command with the **Continuity** option to create a surface between the forward edges of the port and starboard whalebacks.



Whaleback stem wrapper initial surface.

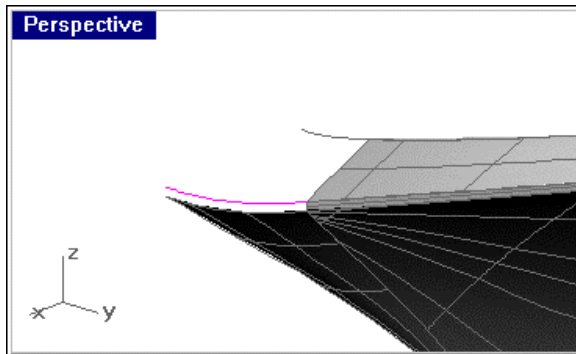
Trim the resulting surface to centerline.

Use the **DupEdge** command to copy the upper edge of this surface.

Delete the surface.

Use the **DupEdge** command to copy the upper edge of the stem wrapper.

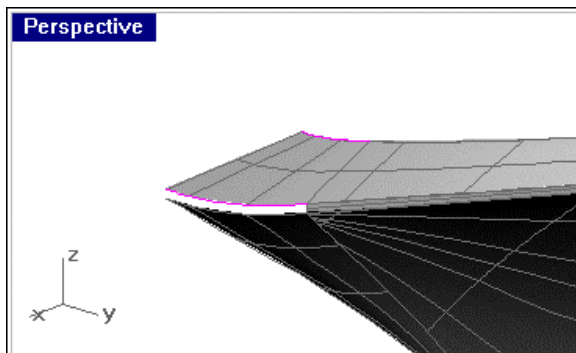
Copy the result up (vertically) 3 inches.



Whaleback stem wrapper edges.

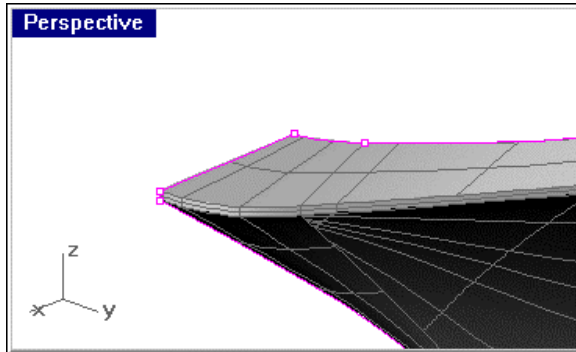
Loft these two curves to create the shear strake at the stem wrapper.

Use the edges of the whaleback stem wrapper and shear strake stem wrapper to **loft** the whaleback stem wrapper.



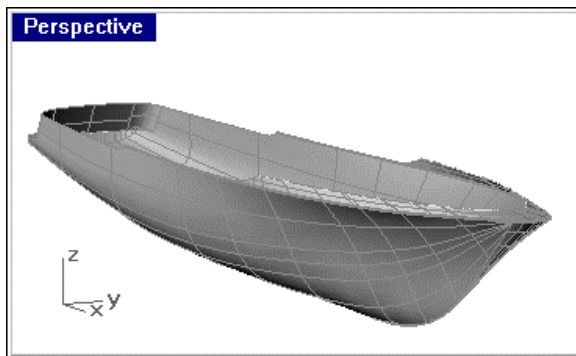
Lofted whaleback stem wrapper.

Join the whaleback and sheer strakes and their respective stem wrappers to the hull polysurface and **check** for naked edges.



Whaleback and sheer strakes.

Mirror the hull parts, **join**, and check for **unjoined** edges.



Completed hull.

Create the camber curve

Draw a vertical **line** in the **Right** viewport from 0,0 to a point above the tip of the sheer.

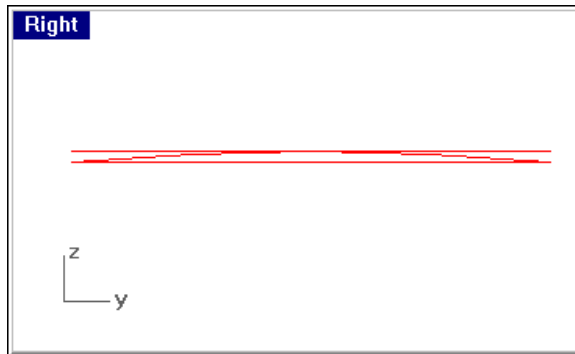
Offset this line, to starboard a distance equal to the half breadth of the vessel.

Mirror the result about centerline.

Draw a horizontal **line** from the top of the starboard vertical line to the top of the port one.

Offset this horizontal line up a distance equal to the camber of the vessel (8 inches, in this case).

Use the **Arc3Pt** command to draw an arc using the starboard endpoint of the lower horizontal line, the midpoint of the upper horizontal line, and the port endpoint of the lower horizontal line.



Camber curve.

This camber curve will be used in the rest of this tutorial to create the decks and in the superstructure tutorial, to create any other decks and housetops.

If you prefer a different camber curve, that is a matter of choice. This method of constructing the camber curve is not mandatory.

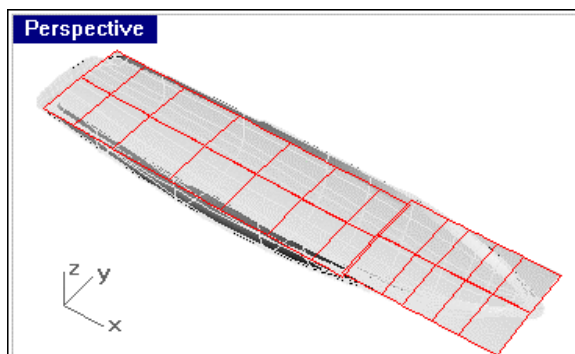
Add the decks

Since the designer gave you the decks at centerline, the construction of these decks is very simple.

Copy the camber curve to each end of each deck at centerline, using **Mid** and **End** object snaps.

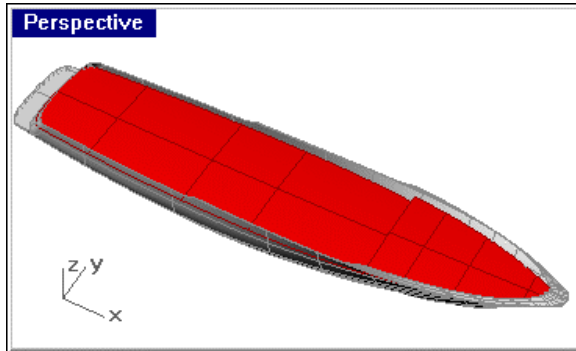
Copy the camber curve to several points along each deck at centerline (4-6 places per deck should do) using **Mid** and **Near** object snaps so that each camber curve rests its midpoint on the deck at centerline profile.

Loft the camber curves.



Deck surfaces.

Trim the resulting decks to the hull assembly.



Trimmed deck surfaces.

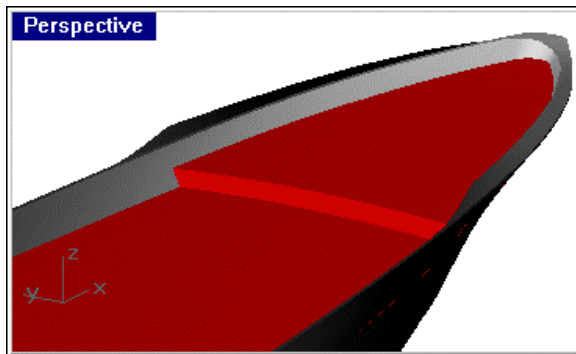
If the designer had not supplied the deck at centerline, you would have to employ a more complex method of lofting the deck. For details of this method, refer to the **Offshore** and **Sail** tutorials.

Create the step between the main deck and the raised deck

Use the **EdgeSrf** command to create the step between the edges of the deck surfaces.

Join the forward and aft decks and the break into one polysurface.

Check for unjoined edges.



Deck step created and joined.

Part VI: 165-Foot Motor Yacht Superstructure



Introduction

This tutorial will introduce you to the methods needed to create a superstructure from a designer's lines drawing and associated Rhinoceros hull model that you should already have created. You will import the curves needed from the designer's lines into an empty .3dm file, import the deck and hull from the hull model, fair the superstructure surfaces using many familiar and some new modeling techniques.

The model created for this tutorial is included on the distribution CD. You can work from it or create your own while performing the tasks. To gain maximum benefit, the latter is recommended.

Requirements

- Complete all of the other tutorials in this series. Some operations in this tutorial assume you have knowledge gained from those tutorials.
- Completed the hull model tutorial for this motor yacht.

The Vessel

This is not a tutorial in boat design. Yacht designers will find fault in the design of this vessel. However, the purpose of this tutorial is to instruct in the art of fairing hulls and superstructures using Rhino techniques.

The Files

The files make up this tutorial include:

Superstructure Lines.3dm – 2-D lines drawing from which the superstructure will be lofted.

Superstructure Final.3dm – Finished deck/superstructure model.

Fair the curves in 2-D

Before modeling the superstructure, you must be sure the designer's lines are fair.

In the designer's lines model, **Superstructure Lines.3dm** use the fairing tools you have learned about in the previous tutorials to get the curves fair to your satisfaction. This is important, since the curves in the designer's lines are disjointed and will make poor surfaces.

Prepare the 2-D lines for export

Because you have already done this a number of times already, no detailed explanation will be necessary. If you need a refresher on preparing the designer's lines for export to your 3-D model, refer to the Skiff, Offshore, and Motor yacht (hull) tutorials.

Build the main deck

Import the main deck surfaces from the hull model.

Open the model **Yacht Hull Finished.3dm**. This can be either the model supplied with the tutorial or the model you created in the previous section.

Copy the main deck polysurface to the Clipboard.

Open the superstructure model and **Paste** the contents of the Clipboard.

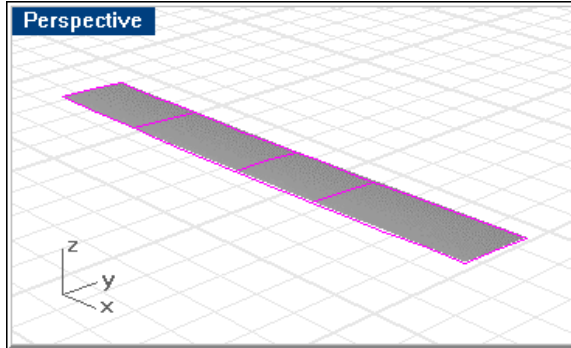
Because the two models share the same origin, the deck is guaranteed to be in the right location with respect to the remaining modeling work to be performed in this tutorial.

Explode the deck polysurface.

Delete the step and front deck.

Untrim the main deck and **extend** it approximately 2 feet forward.

Trim the starboard side of the resulting deck to centerline.



Main deck.

Construct the housetops

Construct the two decks that will form the housetops.

Copy the main deck vertically.

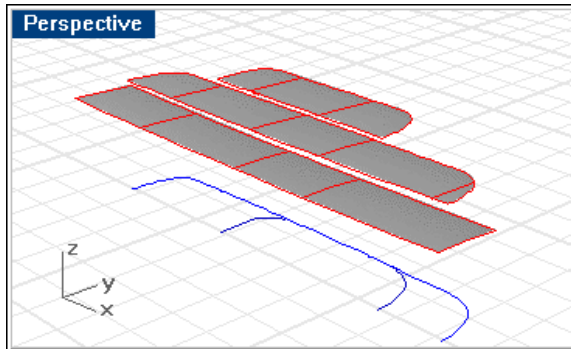
The three decks in this vessel are parallel and share a common camber curve. Get the heights of these decks from the designer's lines.

Change the new boat and bridge decks to layers **sDeckHouse1** and **sDeckHouse2** respectively.

Copy the plan view outlines of the two decks (housetops) from the designer's lines.

These will be on layers **cDeckHouse1** and **cDeckHouse2**.

Trim each deck to the corresponding plan view outline.



Copy main deck vertically and trim to plan-view outlines.

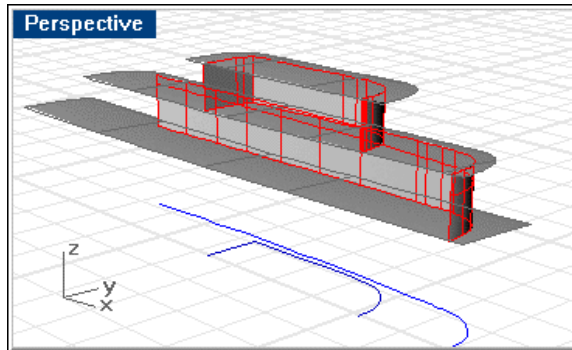
Build the deckhouse sides

From the plan view of the designer's lines, import the plan view curves that represent the intersection of the deckhouse and the main deck. These will be found on layer **cDeckHouse1**.

Extrude these lines to a point just above the boat deck to form the house sides for the first level of superstructure.

Trim the resulting surfaces to the main deck and boat deck.

Repeat this process for the second level of superstructure (the Pilothouse and Master's Stateroom).



Extrude and trim deckhouse sides.

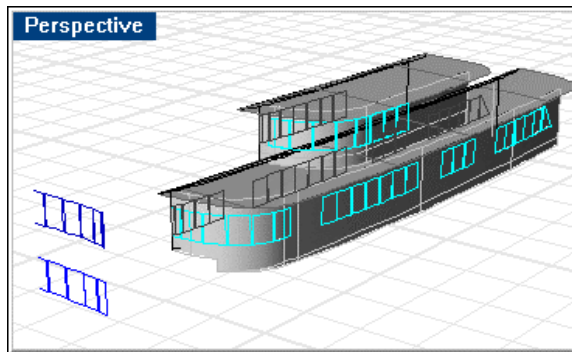
Add the windows

To add the windows, from the designer's lines, import the profile view curves that represent the window cuts. For the windows around the fronts of the two deckhouses, use the curves shown in the body plan view in the designer's lines. Once the windows have been imported, it will be necessary to rotate them in the right viewport so that they will represent the windows in the profile view.

Use the **Split** command to create the window openings.

This will leave the material for the window panes intact.

Change the windowpanes to layer **sGlass**. These can be used for rendering.



Cut in the windows on the house sides.

Build the visors

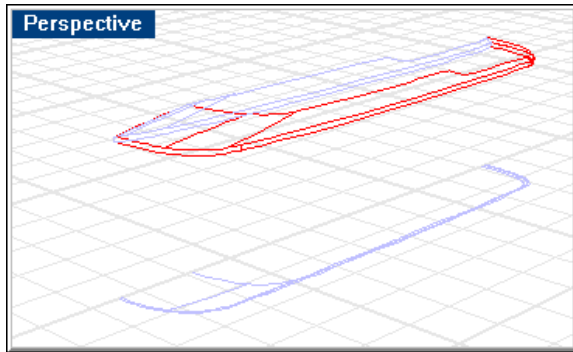
To construct the visors, import the appropriate curves from layers **cVisor1** and **cVisor2** in the designer's lines. You will need to import both the plan and profile views of the curves. Make sure you import all the curves from both views.

Once the curves are in place, use the **Crv2Views** command to construct 3-D curves from the 2-D curves. These curves will be used to **loft** portions of the visors and for constructing sections that will be used for lofting other portions of the visors.

The result of having built 3-D curves can be seen in the image below.

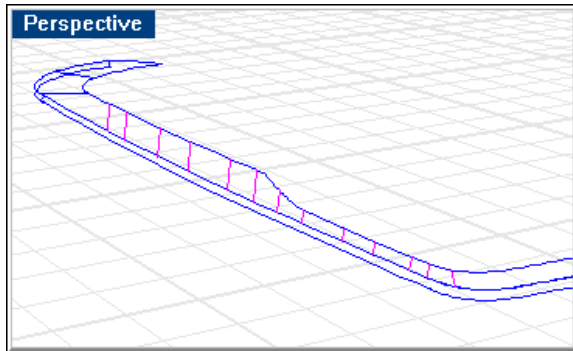
The tangents on the forward corner must be built using the **Crv2Views** command, because there is curvature to the profiles of these two curves. If there were no curvature, you could simply draw them as straight lines, after building the top and bottom of the corner piece.

Only the visor for the first level of superstructure is shown. Both visors will be built in the same manner.



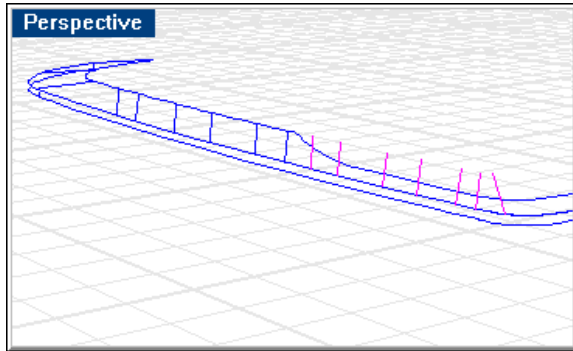
3-D curves of visor.

It is not good form to **loft** a surface from curves that have discontinuities at its edges. The curves that define the top of the visor have discontinuities; so construct sections through this portion of each visor using the **CSec** command. The illustration below shows the result of lofting without extending the curves.



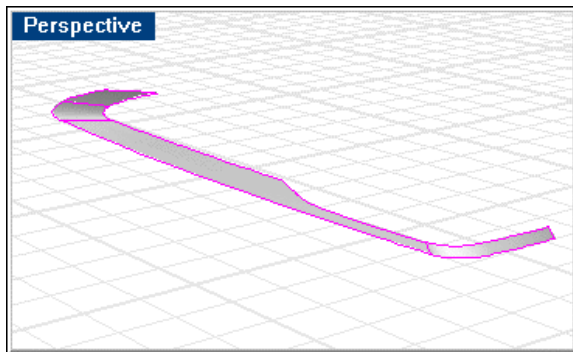
Create section curves through the visor curves.

Extend these sections so that their tops form a continuous top edge to the visor side.



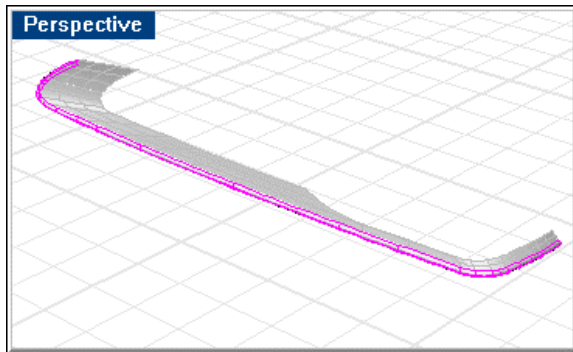
Extend sections along side to create a continuous top edge of visor side.

Use these sections to loft the side of the visor. This lofted surface will then be trimmed to the profile curves of the visor side. Once you have lofted the front, corners, side and back of the visor and trimmed to its profile curves, join everything and check for naked edges.



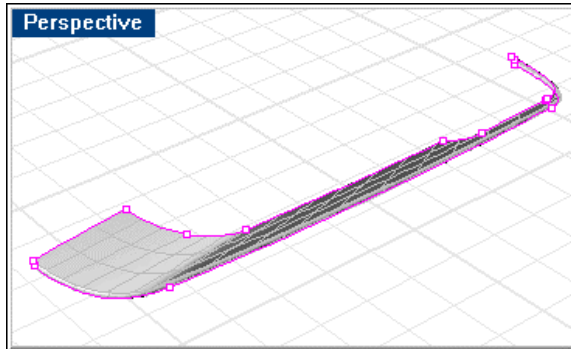
Lofted and trimmed visor.

To finish the visor, use **two-rail sweeps** around the bottom of the visor with the edges of the surfaces and the profile curve that forms the bead at the bottom taken from the designer's lines drawing. This edge is a 5" half-pipe.



Visor half-pipe edge.

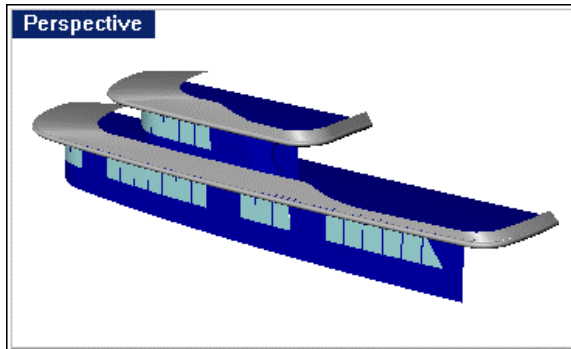
Join this half-pipe to the visor and check for unjoined edges.



Joined visor surfaces.

The visor shown is that of the first level of superstructure. The method for building the second level is identical.

Turn on all layers that begin with the letter **s**, and shade the **Perspective** viewport.

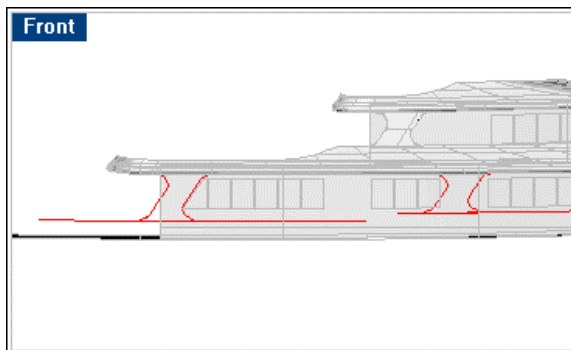


Progress.

Build the visor mullions

Import the profile curves that describe the mullion's shape from the designer's lines.

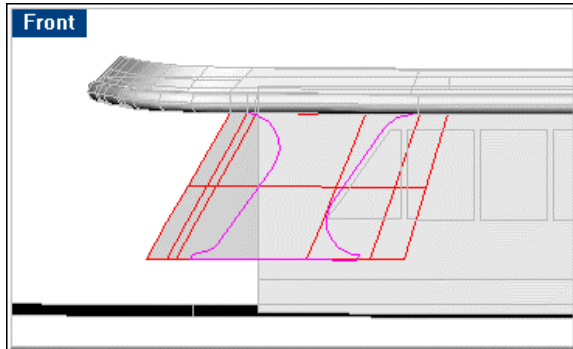
Import the 3-D shear curve from the hull model.



Mullion profile and shear curves.

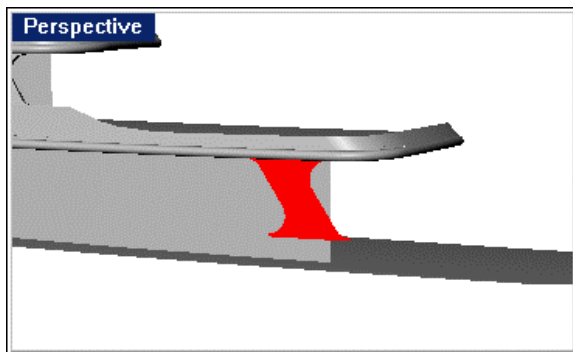
Duplicate the lower edge of the visor surface and split it so it is slightly longer than the mullion shape.

Loft a surface between the sheer and the duplicated edge of the visor.



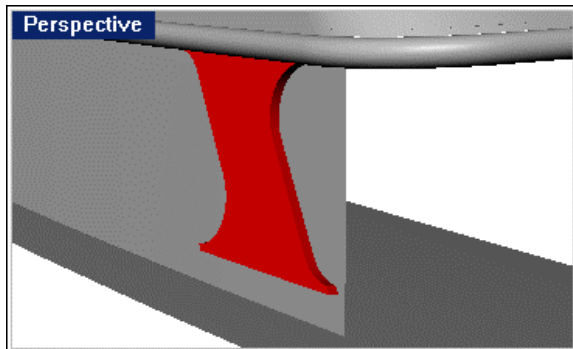
Lofted mullion surface.

Trim the surface to the shapes from the lines.



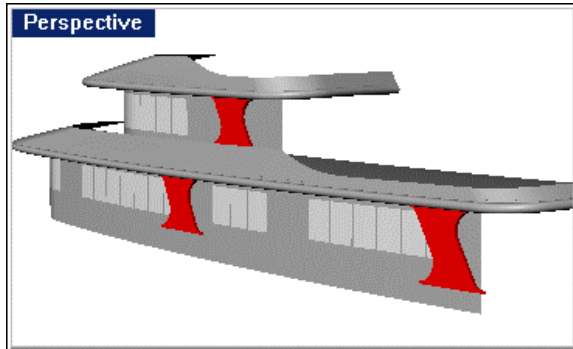
Trimmed mullion surface.

Use the **OffsetSrf** command with the **Solid** option to thicken the resulting surface a 4 inches inboard. If the surface normals point out, specify the **Flip** option when using the **OffsetSrf** command to reverse the tendency for this surface to offset outward.



Extruded mullion.

Repeat this process to create the other two mullions.

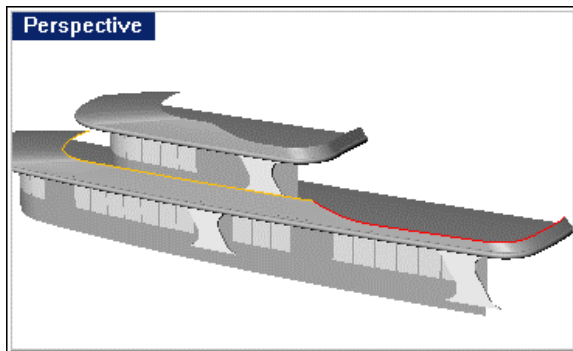


Completed mullions.

Add the caps around the visor tops

To create the lower-level visor cap, use the **DupEdge** command to duplicate the top edges of the visor.

Join the resulting edge curves into two distinct curves that have a common endpoint at the discontinuity in the sheer of the visor, approximately 2/3 of the way aft.

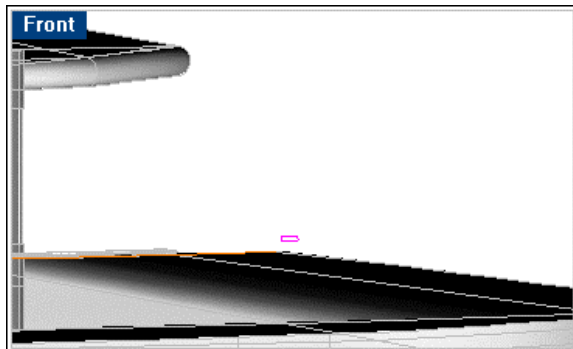


Edge curves for visor caps.

These curves will be referred to as the forward and aft visor edge curves.

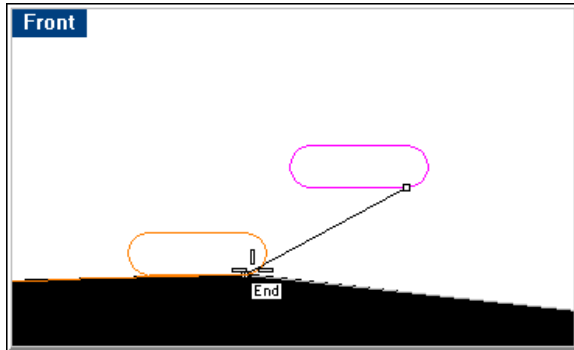
Loft the forward visor caps

In the **Front** viewport, draw a rounded **rectangle** 6" x 1.8".



Draw the visor cap profile curve.

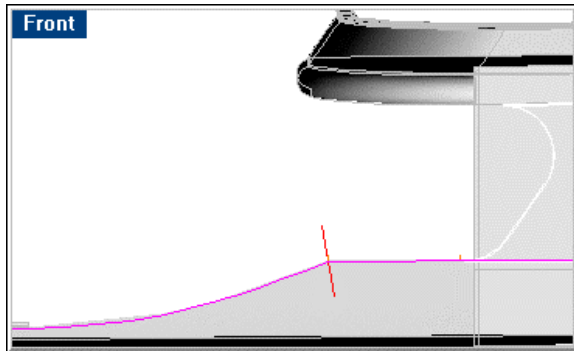
Move this rectangle so the endpoint of the horizontal lower edge matches the endpoint of the forward end of the forward visor edge curve.



Visor cap profile curve moved into position.

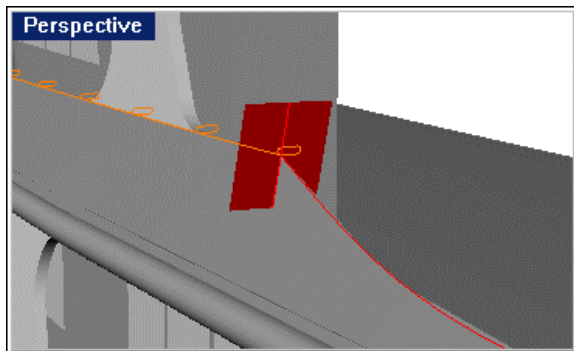
Use the **ArrayCrv** command copy this rounded rectangle along the forward visor edge curve. For the path curve, select the forward visor edge curve. In the **Array Curve** dialog box, under **Method**, for the **Number of items**, specify **16**. Under **Style**, select **Freeform twisting**.

Construct a plane that bisects the angle between the forward and aft segments of the cap. In the **Front** viewport, draw a bisecting line using the **BothSides** option. Make this line approximately 1-foot long.



Bisector line.

Extrude this line in the **Front** viewport, specifying a transverse direction (make sure Ortho is enabled).



Extruded plane.

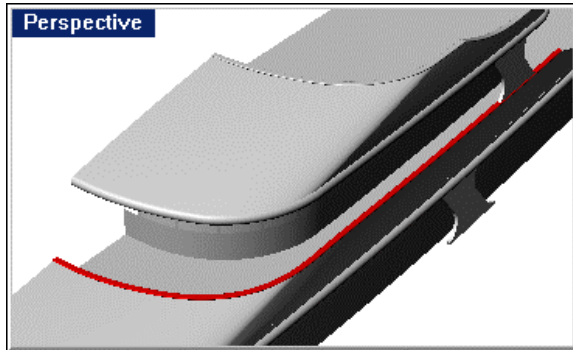
Once you have the bisecting plane, project the aft rounded rectangle that resulted from the **ArrayCrv** command onto the bisecting plane. This will be your last and common loft curve for both the forward and aft sections of the cap. In this way, the two sections will share a common end section and will join as a mitered corner.

Once you have done the projection, **delete** the aft rounded rectangle that resulted from the **ArrayCrv** command.

Delete the bisecting plane. Make sure you retain the curve that resulted from the **Project** command.

Loft the rounded rectangles, including the one that resulted from the projection.

The result should be a nice smooth cap that is 6" x 1.8" at any section normal to the forward visor edge curve.

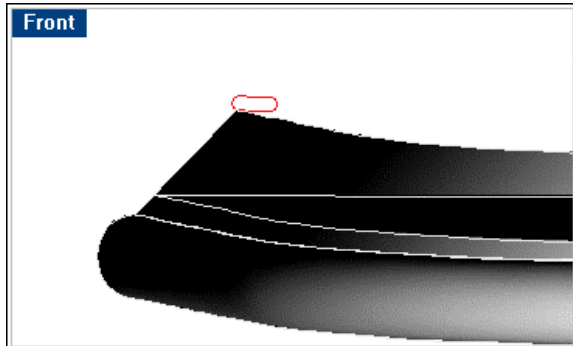


Forward visor cap.

Loft the aft visor cap

Repeat the process to create the aft visor edge cap.

Copy the rounded rectangle so the end of the lower horizontal edge matches the aft end of the duplicate edge curve.

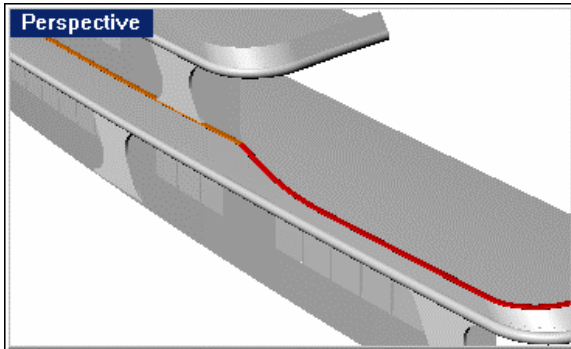


Aft visor cap profile curve.

For the forward-most loft curve, use the projected rounded rectangle that was used for the aft most loft curve in the previous loft.

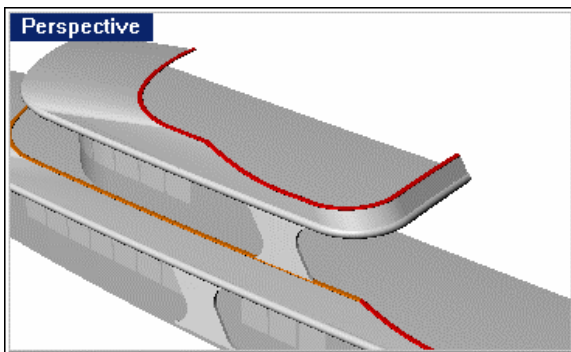
For the **ArrayCrv** command, specify **20** items.

Loft the curves to create the aft visor cap.



Aft visor cap.

Repeat the process for the upper-level visor cap.



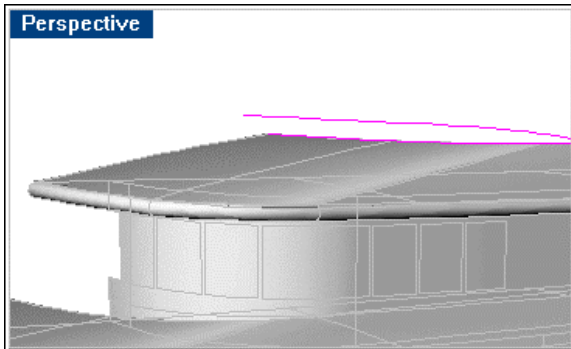
Upper-level visor cap.

Create the windshields

Lofting the windshields is quite simple. Import the 2-D curves for the tops of the windshields from the designer's lines.

Use the **Crv2Views** command to create the 3-D top edge of each windshield.

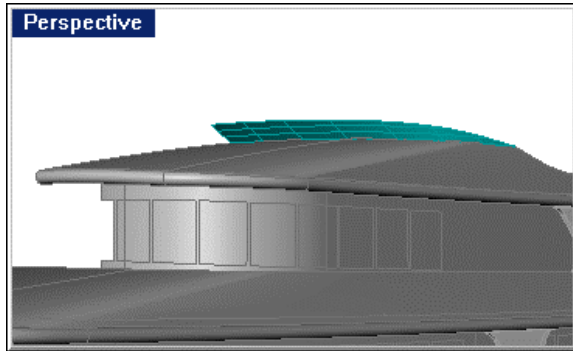
Use the **DupEdge** command to copy the top edge of the appropriate visor to create a curve for the bottom edge of each windshield.



Windshield edge curves.

Rebuild the resulting curves.

Use the **EdgeSrf** command to create a surface between these curves.



Completed windshield.

Build the railings

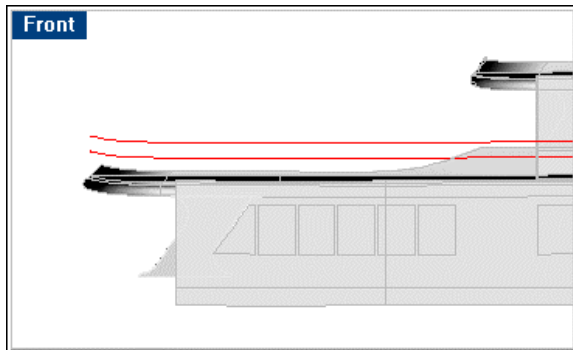
This section describes construction of the railing around the boat deck. All other railings will be constructed the same way.

To add the railings **copy** the courses in profile.

Use the **DupEdge** command to duplicate the deck edge.

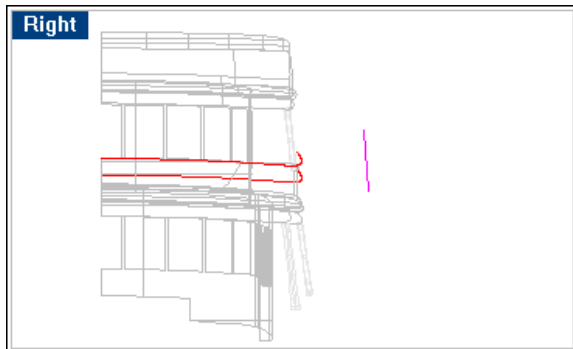
Copy the duplicated deck edge vertically 39 inches.

The intermediate courses are 15 inches and 30 inches below the upper course. Since the upper edge of the visor is above the line created by the lower course, this course need not be created.



Rail courses.

Create the stanchions by drawing a **line** in the **Right** viewport that has an inboard slope of 4 degrees. You can draw this line anywhere, since it will be copied into place along the sheer to create the stanchions.

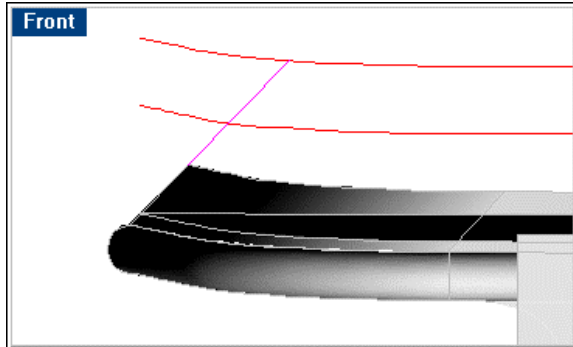


Stanchion line in right viewport.

To add the slope in profile, in the **Front** viewport, **rotate** the line to the same slope as the aft end of the visor.

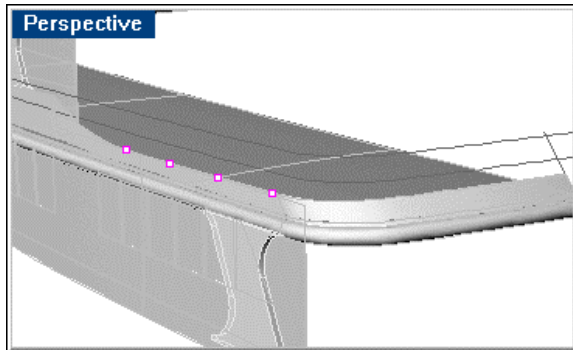
Move the curve to the upper edge of the visor.

Trim this line to match the rail height.



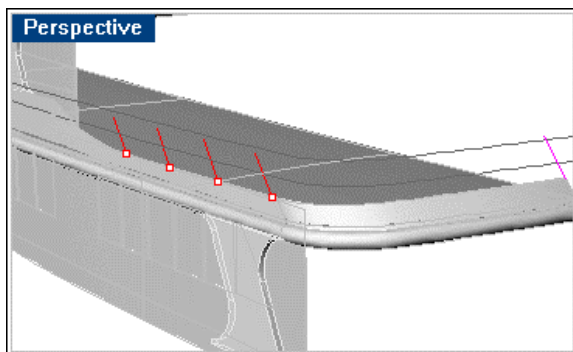
Sloped stanchion line.

Use the **Divide By Length** command to place point objects along the visor sheer. A spacing of 5 feet is suggested.



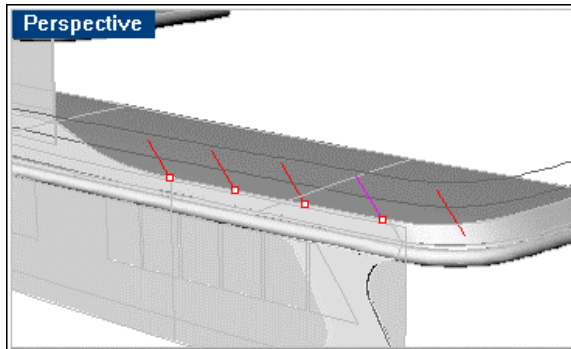
Create points on sheer.

Copy the sloped stanchion line into place at each point created at intervals along the side.



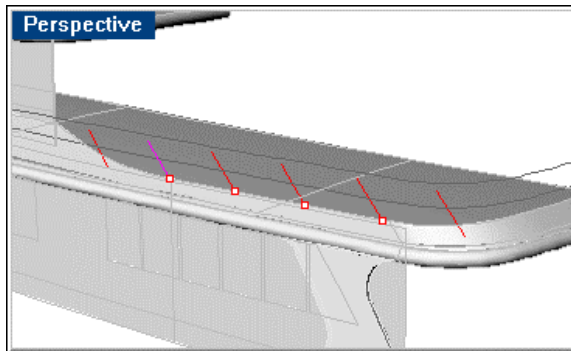
Copy stanchion lines to side.

Once all the stanchions have been placed in this manner, **copy** the last (aft) one to a location aft of the corner of the sheer from the lower end of the next-to-last stanchion (aft) to the lower end of the last stanchion. This creates a "phantom stanchion" which, when lofted, will yield a continuous surface to which you can create a tangent corner.



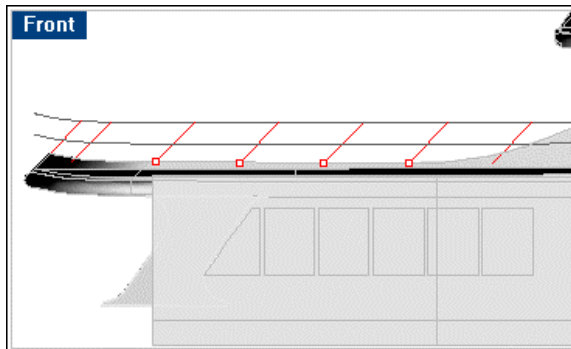
Create aft phantom stanchion.

Repeat this process at the forward end of the railing in order to create a "phantom stanchion" at that end as well.



Create forward phantom stanchion.

After you have created the stanchions, **trim** them (or **extend**, if the stanchions are too short) to the upper course previously created in profile.

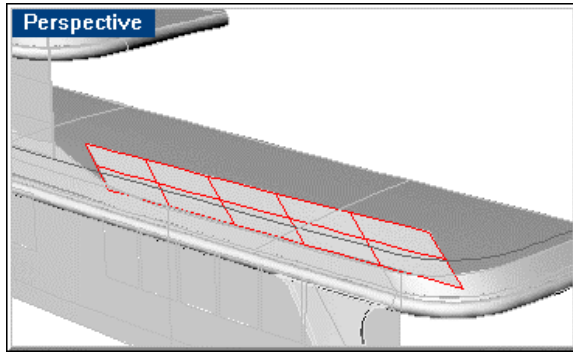


Stanchion curves.

Delete the upper course.

Move all stanchions inboard 3 inches. This movement is done so that the finished pipe stanchions will not "hang over" the edge.

Loft a surface through the stanchions. The result will be a surface onto which you will eventually project the courses.



Lofted side surface.

Create the stanchions across the aft end of the boat deck, using similar methods. The slope of the stanchions in this case will be the same as the slope of the visor surface. There will be no inboard slope to these stanchions. Make the spacing between them equal and approximately the same 5 feet as the side stanchions. Just like you did with the side stanchions, create a "phantom stanchion" outboard of the sheer.

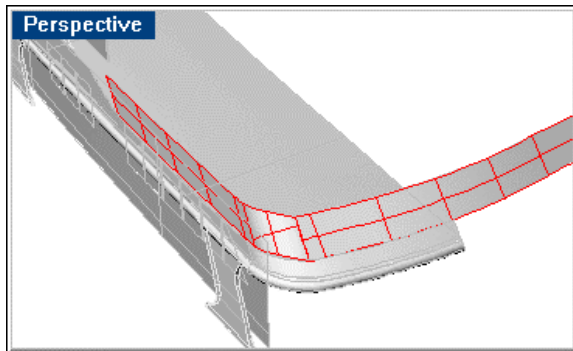
After all stanchions have been created, **move** them forward the same 3 inches.

Mirror the stanchions across centerline before lofting to ensure the lofted surface comes tangent at centerline.

Loft a surface through the aft stanchions.

With the surfaces lofted through the side and aft stanchions, **trim** each to the other and **join** them into a polysurface.

To make the corner, use the **FilletSrf** command and specify a radius of 3 feet.



Construction surface.

You now have a temporary polysurface surface onto which the courses and stanchions can be projected. Additionally, you will create a stanchion in the corner by connecting the midpoints of the upper and lower corners with a line.

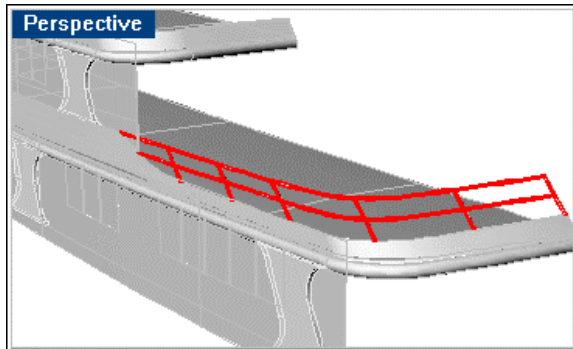
To create the upper course, **duplicate** the upper edge of the temporary polysurface.

To create the intermediate course **project** the intermediate course 2-D line onto the temporary polysurface. Project the aft courses in the **Right** viewport and the side courses in the **Front** viewport.

For the course in the corner, use the **Blend** command between the side and aft courses and **project** the resulting curve onto the corner surface.

Having completed the construction of the course centerlines of the courses and stanchions, you can **delete** the polysurface used for projection.

Use the **Pipe** command to turn all the centerlines into railings. Specify a radius of 2 inches for the upper course and 1-1/2 inches for the intermediate course and stanchions. Once the railings have all been created, trim their forward ends to the sheer. Your railings are now complete.



Completed rails.

Build the stacks

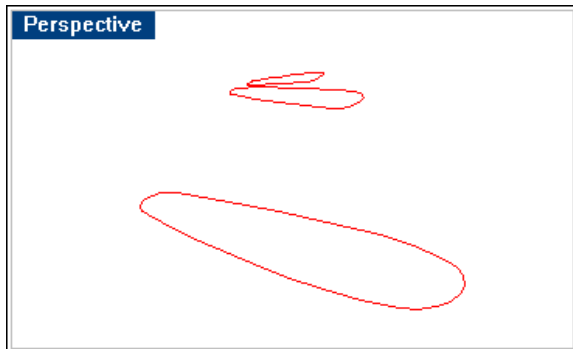
The stacks and the mast are almost a standalone project. You can choose to loft these in the superstructure model or export a few surfaces from the superstructure model and build the stacks independently.

If you choose this latter course of action, be mindful of the common origin shared by the various models. If you respect this origin, parts like the stacks can be interchanged without having to move and rotate things after insertion. This modular approach lets you model a complex project more simply.

Once the stacks are complete, import them into the superstructure model, and trim to the appropriate bulkheads and decks.

To create the stacks, import the lines from the **Superstructure Lines.3dm** model. These will be found on layer **cStack**.

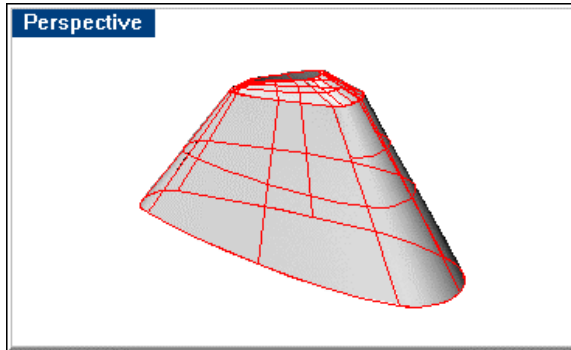
Use the **Crv2Views** command to create the edges of the stack surfaces.



Stack 3-D curves.

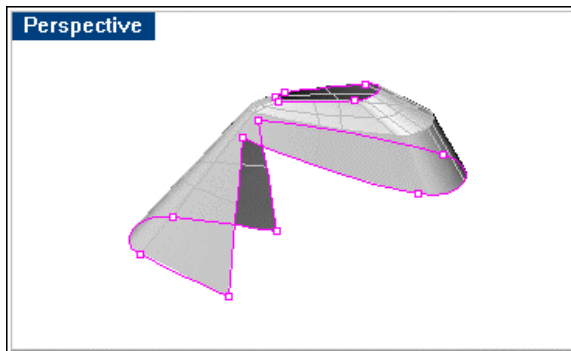
Rebuild the resulting 3-D curves, inspecting the results to be sure they do not differ appreciably from the original geometry.

Once you are satisfied with the rebuilt curves, **delete** the original geometry and use the rebuilt curves to **loft** the various surfaces of the stacks.



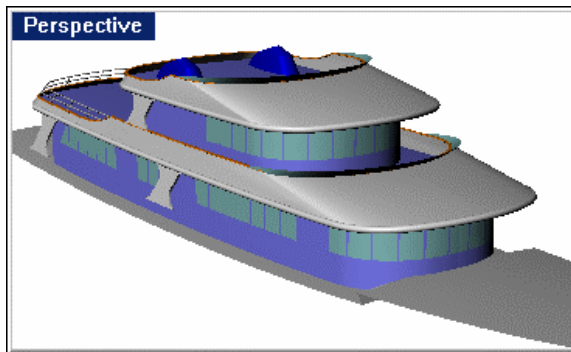
Lofted stack surfaces.

Join everything and check for **naked edges**.



Completed stack, trimmed to the decks and bulkheads.

Mirror the superstructure half.



Mirrored superstructure.

Build the mast

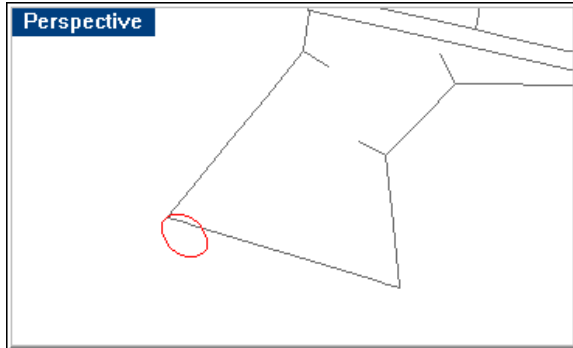
The mast lends itself to being modeled in a separate file. Other than the intersection with the housetop, there is no dependence on hull or superstructure geometry. For convenience, it is included in the superstructure model, but you could just as well create this geometry in its own file, if you prefer. If you do, remember to use the common origin, so you can easily import it into your superstructure model later.

The mast is simply a shapely box with rounded front and back edges. It sits on the housetop at centerline.

Import the profile view curves of the mast and **rotate** them so that they are on top of the house in the proper orientation in the **Front** viewport.

In the **Top** or **Perspective** viewport, draw a 1-foot radius **circle** using the **AroundCurve** option.

Using **Quad** and **End** object snaps, **move** the circle so that its aft quadrant is on the endpoint of the line.

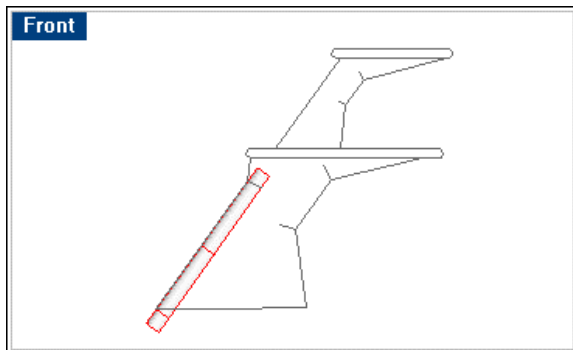


Half-round construction circle.

Draw a **line** from the port quadrant of the circle to the starboard quadrant.

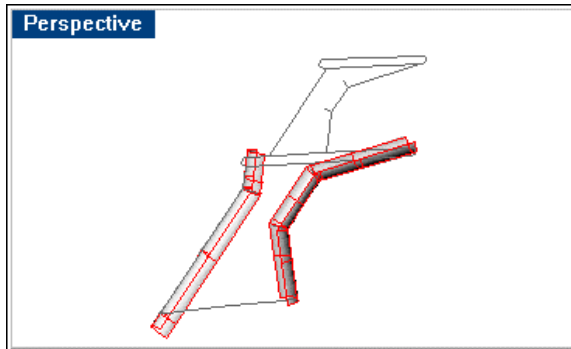
Use this line to **trim** the forward half of the circle away. This will leave a semi-circle that can be extruded along the forward profile curve to form the first rounded surface.

Extrude the circle along the mast profile curve a distance greater than the finished surface will be. You will be trimming this surface later, so it is good to have excess. You can either extend the profile curve or extend the surface after extruding it.



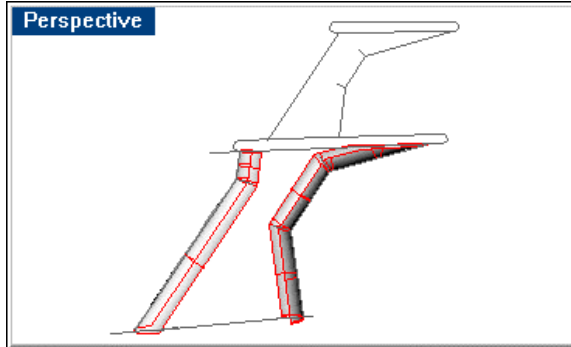
Extrude and extend the half-round.

Create the remaining aft edge rounded surfaces in the same manner as described. After they are all created, **trim** each adjacent pair of surfaces and **join** them together. Check for **naked edges**. If the joint shows no naked edges, move on to the next surface and repeat the process.



Join the surfaces.

Trim the upper edge of each leading/trailing edge of the mast to the radar platform or housetop as appropriate.



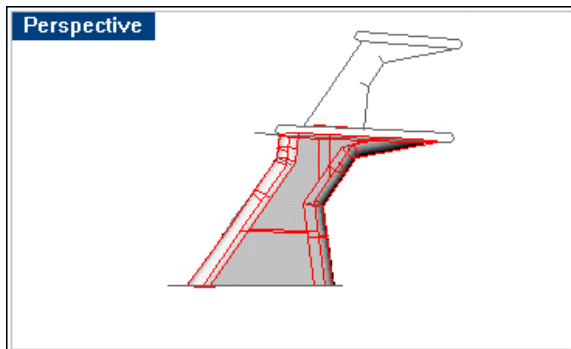
Trim the edges.

Side surfaces

Once you have created the front and back edges of the mast, in the **Front** viewport, create a plane that is large enough to be used as the side of the mast.

Move this plane into place at one side of the mast and **trim** it to the rounded edge surfaces.

Mirror the plane to the other side of the mast.

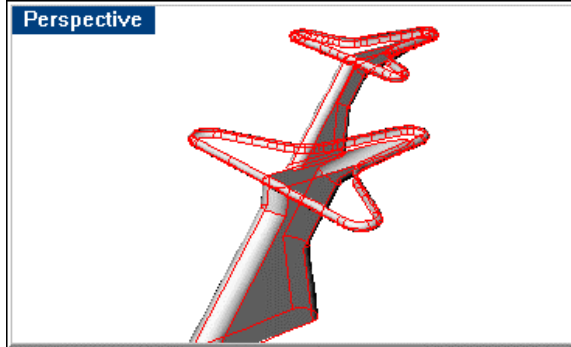


Mast side surface.

Radar flats

To create the mast platforms, **import** the edge curves from the designer's lines (plan view) and **move** them to the proper elevations. From the designer's lines (profile view), **import** the radius that is the profile curve of the platform edges.

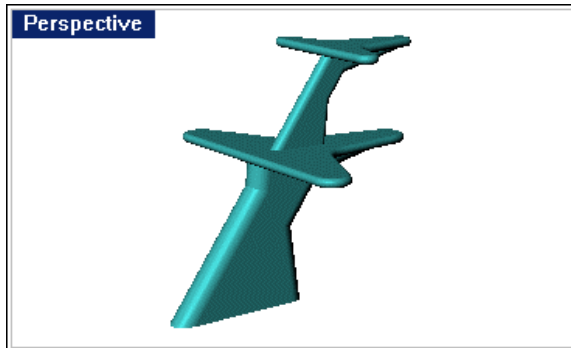
Use the **Sweep2** command to sweep the arcs around the radar flat curves to create the edges.



Sweep the edges of the radar flats.

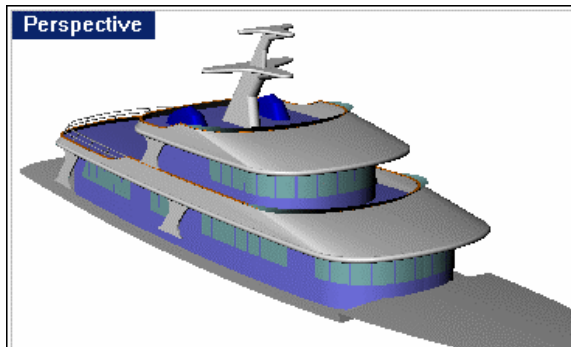
Create a horizontal **plane** that is big enough to completely cover the edges just created.

Trim this flat to the edge surfaces, **copy** it from the top of the edge surfaces to the bottom, **join** everything, and check for **naked edges**.



The completed mast.

Your superstructure is now complete.



The completed superstructure.

You can now import the superstructure into your hull model to complete your mega-yacht.



The completed yacht.

Part VII : Reference

About the author

Cliff W. Estes has been designing and lofting boats, both commercial and pleasure, for 27 years in the Pacific Northwest of the United States.

He has also developed a measurement system for taking lines from hulls and has used it to measure in excess of 50 hulls in steel, aluminum, and fiberglass.

Cliff maintains a design and lofting business and develops custom software for his clients, in addition to developing a series of Rhinoceros plug-ins for marine designers.

Cliff W. Estes
BaseLine Technology
15834 NE 67th Place
Redmond, WA 98052

cestes@baseline.com<http://www.baseline.com>

Hydrostatics from Rhino using RhinoStatics
<http://www.baseline.com/rhinoplugin>

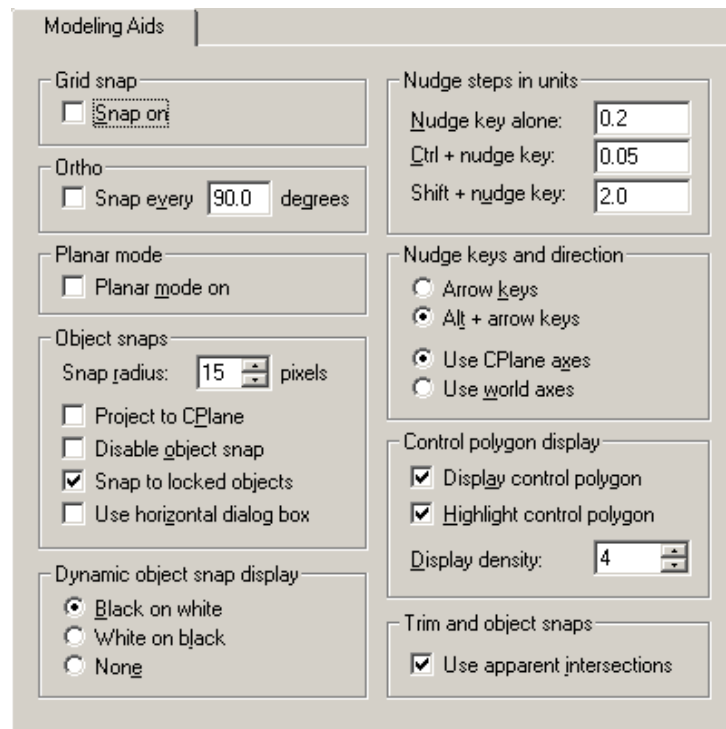
Quality Marine Fairing, Modeling and Rendering
<http://www.baseline.com/imggal.shtm>

How to nudge control points

To move objects, especially control points, small distances while fairing your curves and surfaces, use the "nudge" keys.

Press Alt+ an arrow key to move objects up, down, left, and right relative either to the construction plane in the view or the world axes.

You can change the distances and keys in the **Options** dialog box on the **Modeling Aids** tab.



Options dialog box Modeling Aids tab.

Nudge steps in units

Nudge key alone

Defines the nudge increment of plain nudge keys in Rhino units.

Ctrl + nudge key

Defines the nudge increment when **Ctrl** is pressed with the nudge keys.

Shift + nudge key

Defines the nudge increment when **Shift** is pressed with the nudge keys.

Nudge keys and direction

Arrow keys

Arrow keys alone are used for the nudge keys.

Use **PageUp** and **PageDown** to nudge in the z-axis direction.

Use **Alt+arrow keys** to rotate a view.

Alt + arrow keys

With this option **Alt+Arrow keys** are used for the nudge keys.

Use **Alt+PageUp** and **Alt+PageDown** to nudge in the z-axis direction

Use CPlane axes

The nudge keys move the selected objects in relation to the construction plane axes in the active viewport.

Use world axes

The nudge keys move the selected objects in relation to the world axes.

What to do about unjoined edges

If your model has unjoined (naked) edges where the surfaces are supposed to be joined, you will need to fix the problems.

These are some of the strategies you can use:

- Use the UnTrim command to separate the trimming curve from the surface and then trim with the curve.
- Use adjacent surfaces to trim instead of a curve.
- Extract the surfaces with the ExtractSrf command that share the naked edge, rebuild the edges (RebuildEdges command) and join.
- Use surface edges to create other surfaces wherever possible rather than duplicated edges or other curves.
- Trim with adjacent surfaces wherever possible.
- Use the MatchSrf command to match untrimmed surface edges.

Glossary

Vessels have features that may not be familiar to all readers, the following illustrations are offered to clarify the names of features used in the text.

After Perpendicular

The most common usage of the term defines the Aft end of the Design Waterline.

Body

Also known as *body plan*, this is a view in the lines drawing that depicts the front view of the boat, as if standing forward of the bow and looking aft. On the right side of the centerline are stations located forward of midship and on the left side of centerline are the stations aft of that point.

Design Waterline

The waterline or level at which the vessel is designed to float. The determination of this waterline is beyond the scope of this tutorial.

Forward Perpendicular

The most common usage of the term defines the forward end of the *design waterline*.

Lines

A drawing depicting all the edges (sheer, knuckles, chines, surface intersections, etc.) and derived curves (waterlines, buttocks, diagonals) that give the builder a sense of the shape of the boat.

Lines drawings consist of three views: plan (also known as *beam plan*), profile (also known as *sheer plan*), and *body plan*.

A lines drawing can be likened to a contour map of a piece of property.

Midship

The point, typically half-way between the forward and aft perpendiculars. Typically, though not necessarily, this point marks the largest section of the vessel.

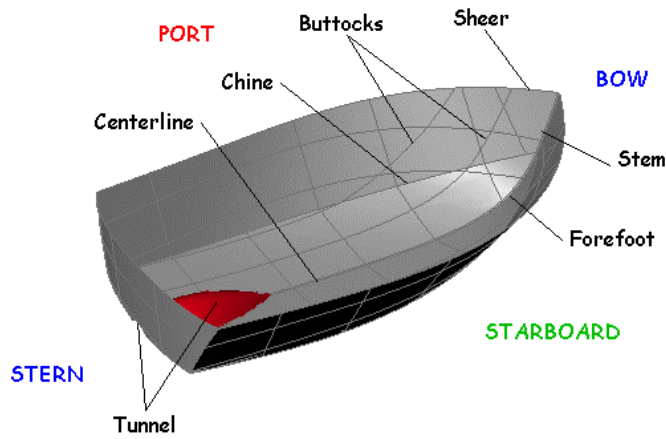
Plan

Also known as *beam plan* this is a view in the lines drawing that depicts the top view of the boat as if standing above the water and looking down. This view includes edges and waterlines. Also included are straight lines representing the buttocks and stations.

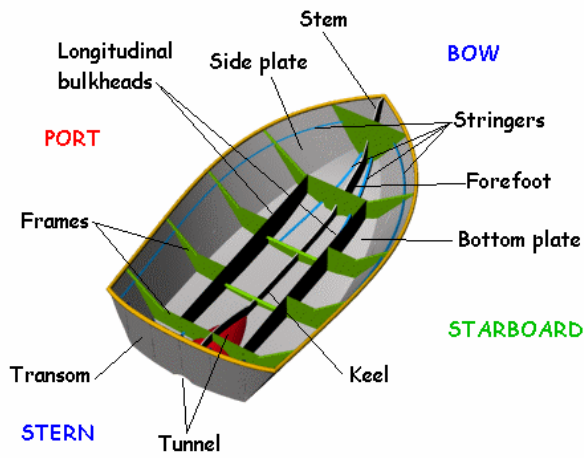
Profile

Also known as *sheer plan* this is a view in the lines drawing which depicts the elevation of the boat, as if viewed from the starboard side (typically, though not exclusively). This view includes edges and buttocks. Also included are straight lines representing the waterlines and stations.

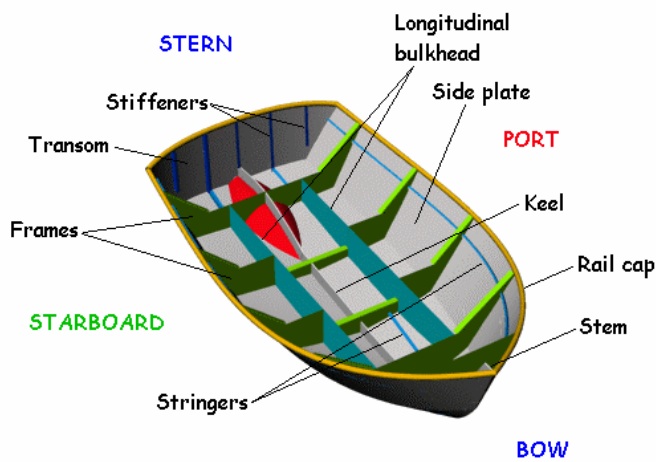
Parts of a Boat



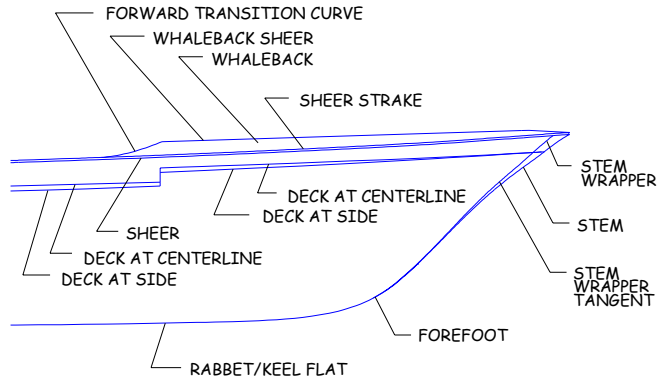
Lines.



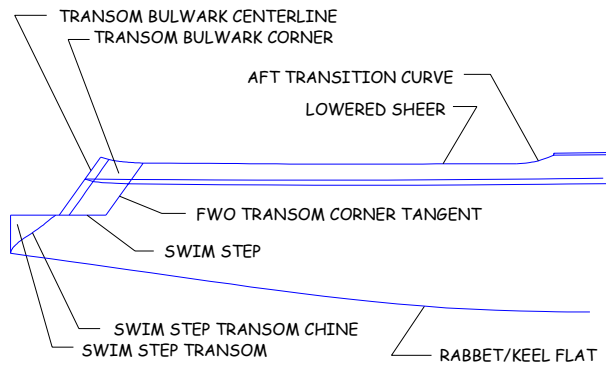
Parts from stern.



Parts from bow.



Nomenclature - bow.



Nomenclature - stern.