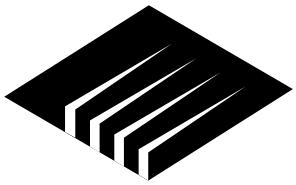
User's Guide

Autoship

FOR WINDOWS™

Release 8.0



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Introduction

Introduction to Autoship/Autoyacht 8.0

This chapter contains the following topics:

What's in This Guide?

The purpose of this guide is to assist users in installing, starting, running, and using Autoship/Autoyacht release 8.0 (for convenience, only the term "Autoship" will be used).

Once the program is running, the fully online and context sensitive help system is the primary source of assistance and reference.

This printed guide specifies the system requirements needed to run Autoship, gives instructions for installing and starting the program, and outlines the range of available online and printed help resources. The section entitled Tutorials contains six step-by-step exercises that introduce the basics of designing and modelling with Autoship.

Users upgrading from previous versions of Autoship will want to review the "What's New" section of this user guide; for more details

on program changes and bug fixes, see the History.doc and Readme.doc files.

Both this guide and the online help system (Autoship Help) cover all features within Autoship 8.0 Pro; for a list of differences between Autoship Pro, Autoship Standard, and Autoyacht versions, see the Introduction section of Autoship Help.

Welcome to Autoship Systems Corporation

Since its founding in 1980, Autoship Systems Corporation ("ASC") has become the world's leading PC software developer for the naval architecture and marine shipping industries.

Our mission is to provide quality software solutions for the marine industry. Autoship Systems Corporation is the foremost supplier of marine design and construction programs worldwide, with over 1,700 installations in more than 50 countries and an international network of Autoship dealers. Our roots are in providing CAD/CAM products for the ship/boat design market, where ASC was the first to deliver a ship design tool on a PC platform, first to develop a Windows-based ship design tool, and was a leader in using the industry-standard Nurbs 3-D hull modeling tool. Recently ASC broadened its product lines and services, which now include on-board stability and stowage planning software for ship owners and marine cargo handlers.

Autoship Systems' success is derived from building upon continuous leading-edge innovation, unsurpassed quality, and a strong tradition of responding to our customer's needs. This tradition of success is continued with the new functionality found in this latest release of the Autoship program.

What's New in Version 8.0

Introduction

This chapter provides an outline of the functions, features, and tools that are new or modified in Autoship release 8.0. For detailed information on these topics, refer to the individual subjects in the online help system (see below). See also the History.doc and Readme.doc files (found in \Program Files\ASC\Autoship8 \Documents) for a listing of fixed bugs.

Autoship Help - The Online Help System

Autoship Help, provided with this release, is a fully online context sensitive help system integrated into the Autoship program. There are two main ways to access Autoship Help:

1. Run Autoship Help

Click on the Help menu in Autoship, and then select Autoship Help.

2. Context Sensitive Help

Point at the part of the screen that help is required for and press F1; or use the help buttons in the dialogue boxes: both will activate the help system, which will display the highest level of help contained for the desired topic. Navigate from there to more specific information using any of these methods:

- the system of links to related topics
- the Table of Contents, provided on the left of the help screen
- the Index tab at the top of help screen
- the Search tool tab at the top of the help screen

Changes in Autoship 8.0:

New Offset Report Format

The format in which offset tables are presented in reports has been changed to be closer to naval architecture practice.

ORC IMS Offsets Export

The Offshore Racing Council International Measurement System offset file provides data on hull, keel and rudder for use in their Velocity Prediction Program (VPP). This file has the extension ".off", and is meant to be partnered with a ".dat" file generated by ORC's ORCEDIT.EXE program.

Changes to Geometry File Export

Important Note: These changes significantly alter the way the Autoship program behaves.

Previously, Autoship made one Part (name "HULL", unless changed by user at a prompt) containing one Component (name C1). Now a separate component is created for each object in the selected group. Thus each surface, polyobject or group contained in the export group becomes a different component.

Instant Hydrostatics

Autoship 8.0 provides instant hydrostatics information through the small "H" button located in the bottom row of controls.

Surface Edge Match to Curve

The Attach Surface dialogue has been changed to support a new feature of the program. In addition to the old choices of "Attach to Surface" and "Attach to Curve", there is now a third option, "Match to Curve". Choosing this option, selecting a curve to match, entering a tolerance causes the program to match the selected surface edge to all or part of the curve. This feature has been added to compensate for the loss of ability to trim attached surfaces.

Surface Blends

There is a new tab in the Create Surface dialogue, labeled "Blend", providing the ability to produce fillets and chamfers.

Grouping of Control/Edit Points

It is now possible to select and move multiple edit and control points.

Trimmed NURBS Surfaces

Previous to this release, trimming a surface produced a polyobject, which could be regenerated but not edited. Now the trimming operation is performed directly on the NURBS surface. Further editing of the surface or trim curve(s) results in an immediate update of the trim.

Surface Mesh Layout

In previous releases surface mesh points were spaced equally in parameter space, with extra points at chines. This has been replaced with a system in which there is a mesh node at each knot, including chines, and the number of mesh points between knots depends on the degree of the section.

DXF Output of Flattened Developable Surfaces

This now brings up the DXF output dialogue which allows the user to select many of the same output options as for other objects.

Cutting Surface-embedded Curves at Crossings

In select mode, the bottom right tool icon, which was previously used only for trimming surfaces, now has two functions.

Measurement Tool

Next to the information button on the bottom row of controls is a small button labeled "M". Clicking on this control brings up a frame containing measurement information. Clicking on the "M" button again removes the measurement frame.

Contact Information

There are two main ways to contact Autoship Systems Corp.:

- Contact our technical support department for technical assistance.
- For other inquiries, contact our head office.

Technical Support:

Our technical support department is available for direct telephone inquiries on weekdays from 08:00 to 17:00 Pacific Standard Time. You can also fax or email questions at any time:

Tel:	(+1) 604 254 4171 (Extension 104)
Fax:	(+1) 604 254 5171
Email:	techsupport@autoship.com

Contacting Autoship's Head Office:

If you would like more information about Autoship Systems Corporation and our products, please do not hesitate to contact us at:

Telephone:	(+1) 604 254 4171 (Extension 102)
Fax:	(+1) 604 254 5171
Surface Mail:	Autoship Systems Corporation Suite 312, 611 Alexander Street Vancouver, BC, Canada V6A 1E1
Email:	
Sales Dept:	sales@autoship.com
Our web site:	www.autoship.com

Installation

This chapter contains sections that detail:

- system requirements necessary for running Autoship
- installation instructions
- hardware lock installation

System Requirements

The following table shows the minimum and recommended system requirements for running Autoship:

	Minimum	Recommended
CPU	• Pentium 90 – recommended minimum	Pentium 400 or higher (for best results)
Memory	32 Mb (Windows 95) 64 Mb (Windows NT)	64 Mb (Windows 95) 128 Mb (Windows NT)
Free Hard Disk Space	10 Mb	
Graphics	1024x768 resolution, 256 colors *	1024x768 resolution, 32k colors
Mouse	Any Windows pointing device	

*****Operation is possible at 256 colors, but for rendering to function correctly 32k colors or more are needed.

Program Installation

1. First, you need to install the hardware lock onto your system.

   Note: If you do not install the hardware lock, Autoship will run in test drive (or "demo") mode and will not be able to save any files or print any reports. (See "Installing the Hardware Lock" below for hardware lock installation details.)

2. Then, with Windows running:

   - If you are installing from a CD, run **Setup.exe** found on the CD. If you are installing from floppy disks, run **Setup.exe** found on the Authorization disk.
   - If you are installing from files on your hard drive, run **Setup.exe** on your hard drive.

3. Follow the instructions given by the install program.
4. When you get the message that installation is complete, respond by clicking **OK**.
5. After the installation is complete, you will see the Autoship icon on the screen.
6. To start Autoship, click on the appropriate icon or option from Start – Programs – Autoship.

Hardware Lock Installation

Description

The hardware lock is a small electronic device that is connected to the parallel port of your computer. (Note: If you have a network hardware lock, only one lock is used for all computers on the network. See below for details.) You need the hardware lock in order for Autoship to run in normal operating mode. Otherwise, the program will only run in demo mode. Keep the lock in a safe place when not in use. As outlined in the Licensing Agreement, you are exclusively responsible for the hardware lock. You may be required to purchase an additional software license if the hardware lock is lost or stolen.

Precautions

- Before beginning the installation, close all running programs.
- Only network locks may be daisy-chained. For a local station installation, a single lock may be configured for all the programs the user is authorized to use – consult your dealer.
- Certain local printers may interfere with the operation of the lock. If this is the case, try a different printer driver, or install another parallel port.

Windows 95, Local Station

1. Connect the hardware lock to the parallel port.
2. If you had a printer connected to the same parallel port, connect the printer to the free end of the lock.
3. Install Autoship as discussed above.

Windows NT, Local Station

1. If the Autoship installation program detects a Windows NT operating system, you will be prompted: "Would you like to copy the NT lock driver files to your computer?" You must respond "Yes", unless you have previously installed the NT lock driver. (The NT lock driver is required so your NT operating system can communicate with the parallel port.) You will then be asked where you want to copy the NT lock driver files to.
2. Follow the instructions to complete the installation of Autoship onto your computer.
3. If the program runs only in Demo mode, see the Readme.doc file located in the same directory you copied the NT Lock Driver files to.

Hardware Lock Installation, Network System

If you have purchased a network lock, you will have received a separate floppy named "Network Lock Driver".

1. Connect the network lock onto the parallel port on any one of the computers in your network.
2. The Network Lock Driver floppy contains 3 directories:

   - \DOS (for Windows 3.11)
   - \NW (for NetWare)
   - \Win32 (for Windows 95 and Windows NT)

   Each directory contains a readme.txt file. Follow the directions appropriate for your operating system.
3. Install Autoship onto each station on your network from which it will be run, following the installation instructions above.

Getting Started

Introduction

This chapter contains sections that detail:

- Starting Autoship
- Using Autoship and accessing the context-sensitive Help system
- Exiting Autoship
- Upgrading Autoship

Starting and Exiting Autoship

To start Autoship, go to Start – Programs – Autoship and click on the Autoship 8.0 icon.

To become familiar with the main screens, features, and organization of Autoship, point at any active window, dialogue, or feature on the screen and press F1. The highest-level of available Help information will appear for the feature you selected, and will contain links to more detailed information.

To exit Autoship, go to File – Exit. Be sure to save any changes made to your project files before closing down the program.

References and Help Resources

This section describes the help resources available to Autoship users. Autoship Help (the online help system) is the primary resource, and it can be accessed at any time.

How to Access Online Help:

Run Autoship Help

Click on the Help menu in Autoship, and then select Autoship Help.

Context Sensitive Help

Point at the part of the screen that help is required for and press F1; or use the help buttons in the dialogue boxes: both will activate the help system, which will display the highest level of help contained for the desired topic. Navigate from there to more specific information using any of these methods:

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Types of Help Available:

Online Familiarization and Overview

- The onlne help section entitled Introduction to Autoship gives an overview of the software. Reviewing the table of contents will provide a good introduction to the functional organization of the Autoship program.

- Index and search utilities are also available within the Help system.

Program Reference: Context-Sensitive Help

- Autoship includes context sensitivity in the main screen (see "How to Access Oneline Help" above).

- Detailed information on the software's features, and how to use them, is available in the "Usert Interface" section of the help system. The help system can be accessed by using the Help Menu in the main screen menu bar, or by using context sensitive help (see "How to Access Online Help" above).

- The Basic Functions section in the Help system explains how to perform many of the basic operations involved in using Autoship.

Online Technical References

The following technical and theoretical references are provided in the Help system:

- Basic Concepts – Autoship as a surface modeler, Nurbs .vs. polygonal models.

- Technical Reference – theoretical basis and detailed explanations of the functions performed on surface models by Autoship code.

Tutorials

The tutorials in this guide are a series of increasingly challenging step-by-step projects that help develop the skills needed to use Autoship as a design tool. It is assumed that users are already familiar with the main screens and tools of the system, and know how to use the Help system to answer basic questions. Example tutorial files are found in \Program Files\ASC\Autoship8\Tutorials.

Frequently Asked Questions

Within the online help system is a list of the most frequently asked questions regarding the use of Autoship, with the answers provided by Autoship Systems Corporation's technical support department.

Contact Information – Technical Assistance

Technical assistance is available as outlined in “Contact Information” on page 7.

Further Training

For advanced and complex projects, you should take an Autoship training course. Contact Autoship Systems, or your Autoship dealer for schedules and locations of upcoming courses, or for customised training.

Where to Find Help Resources

There is a complete range of Help resources available to assist users at all levels:

Help Found Online – The Autoship/AutoYacht Help Sytem

- General Familiarization and Overview
- Program Reference - context-sensitive online Help containing specific information on how to use Autoship tools, interface, features, and basic functions.
- Technical references – help with the purpose, details, and theory behind key concepts and features in Autoship.

Help Found in Print – This User Guide

- Getting Started – Installation and Familiarization information.
- Detailed Tutorials (in this User Guide).
- Note that this guide is also available in .pdf format on your Autoship/Autoyacht program CD.

Autoship Systems Corporation
Contacts and Technical Assistance

- Autoship System Corporation's technical assistance via telephone, email, or fax.
- Autoship System Corporation's instructional courses on the Autoship program.

Updating Autoship

When you update Autoship, the installation procedure is the same as for a new installation. All necessary program files are updated. No data you have saved will be adversely affected by updating the installation.

Tutorials

Introduction

This chapter takes you through six tutorials: Chined Hull Planing Boat; Round Bilge Hull; Asymmetric Hull Catamaran; Pocket Cruiser; Ro-Ro Carrier; and Oil Rig. Completed examples of these tutorial projects are found in \Program Files\ASC\Autoship\Tutorials. It is suggested that you first create a new folder to contain the tutorial projects you will construct (we suggest the filename MYTUTS).

We recommend **that Autoship Pro and Standard users** do these tutorials in the order that they are presented, as the sophistication of the design methods increases accordingly.**Autoyacht users** should begin with tutorial one; however, because Autoyacht does not support developable surfaces, take care to create the bottom using a ruled surface. Tutorials 2,5, and 6 cannot be constructed in Autoyacht due to the size of the vessels.

With Autoship there are always, generally, several ways to accomplish a given task. Often the best approach will be determined by the requirements of subsequent steps. In these tutorials, we have not attempted to show the only, or the best way to deal with a certain aspect of the model, but instead have undertaken to provide a broad base of approaches from which the knowledgable user can choose the one best suited for their particular circumstances.

(See note on next page)

Note: When more information is required than is available in these tutorials, access Autoship Help through one of these three methods:

- press F1 and help will appear for the window, dialogue, or feature currently active on the screen
- use the Help menu in the Menu Bar
- use the Help buttons provided in the dialogues

Tutorial 1 - Chine Hull Planing Boat

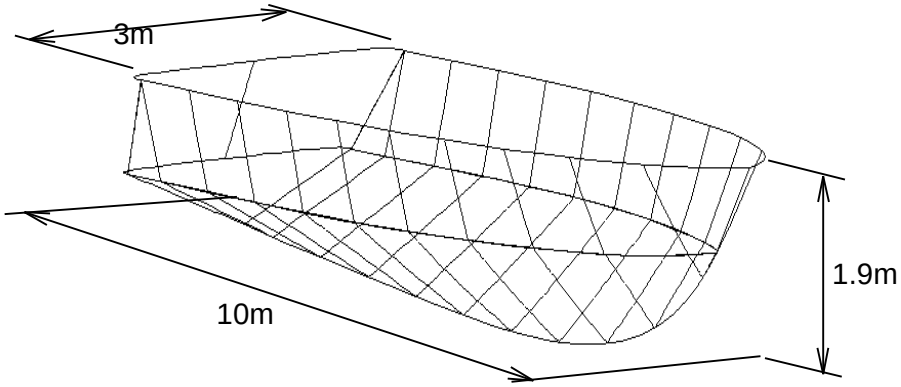

Figure 1-1 Chine hull planing boat

1.1 Introduction – Chine Hull Planing Boat

In Tutorial 1, you will:

- Design a deck edge
- Design a chine
- Design a centreline
- Create the bottom panel as a developable surface
- Create the side panel as a ruled surface
- Add a cylindrical transom
- Obtain offsets
- Obtain hydrostatics

Tutorial 1 shows how to create a hull using developable and ruled surfaces and demonstrates how to create and fit a cylindrical transom. It also introduces the idea of attaching the end of a curve to a point and discusses the topic of tangency at the end of a curve. In addition, it shows how to obtain offsets and hydrostatics.1.2 Starting Out

The model we've constructed, **AS7TUT1.PR3**, is a very simple, little planing hull. You may want to review this model before continuing.

➲ To begin:

1. Select ***File-New***.
2. Select ***Settings-Units*****,** choose ***Meters*** and ***Tonnes***, and click ***OK***.
3. Select ***Settings-Preferences***.
4. Click the ***Coordinate System*** button.
5. Click the ***Naval /Aircraft (US)*** radio button. This orients the coordinate system so that aft, starboard, and up are positive directions; while forward, port, and down are negative directions.
6. Click ***OK*** in the *Coordinate System* dialogue and click ***OK*** again in the *Preferences* dialogue.
7. Select ***File-Save***. In the *Save Project* dialogue box that appears, type **TUT1.PR3** for the File Name. Save the project in your new directory called **MYTUTS**.
8. In the *Project Info* dialogue , under *Name*, type a descriptive message for the project (for example, **10m Planing Hull**). Notice that under *Details*, 0 objects have been defined.
9. Click ***OK***.
10. In the *Save Project* dialogue that appears, type **TUT1.PR3** and specify the **MYTUTS** directory to store this project in.

1.3 Designing a Deck Edge

This exercise introduces the concept of working to reference points. In some instances, you may want to have several curves end at the same location. Attaching the ends of the curves to a common point accomplishes this purpose.

➲ To create a Reference Point at the Forward End of the Deck:

1. Click on the ***Create Mode*** button, which places Autoship in Create mode.
2. Click the ***Create Point*** button.
3. In the *Create Point* dialogue, select the ***Free*** tab, type the following information, and click ***OK***.

Field	Entry
Point Name	Deckfor
Location	L = A0.3; T = 0; V = 0.9

4. Hold down the ***Shift*** key and click on the ***Zoom Extents*** button to zoom out in all views. (See Figure 1-2.)

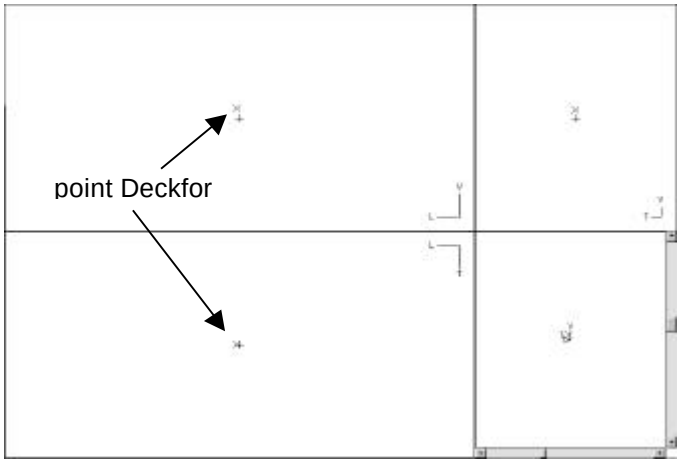

Figure 1-2: The point Deckfor. Note that the "x" is the point Deckfor, while the "+" is it's "base point".

➲ To create a curve to represent the deck edge from bow to stern:

1. Click the ***Create Mode*** button.
2. Click the ***Create Curve*** button.
3. In the *Create Curve* dialogue, select the ***Free*** tab, type the following information and click **OK.**

Field	Entry
Curve Name	Deck Edge
Parameters	Control Points = 5; Max Degree = 3
End 1	L = 1; T = 1.5; V = 0.8
End 2	L = 9; T = 1.5; V = 0.8

4. Hold down the ***Shift*** key and click on the ***Zoom Extents*** button. (See Figure 1-3.)

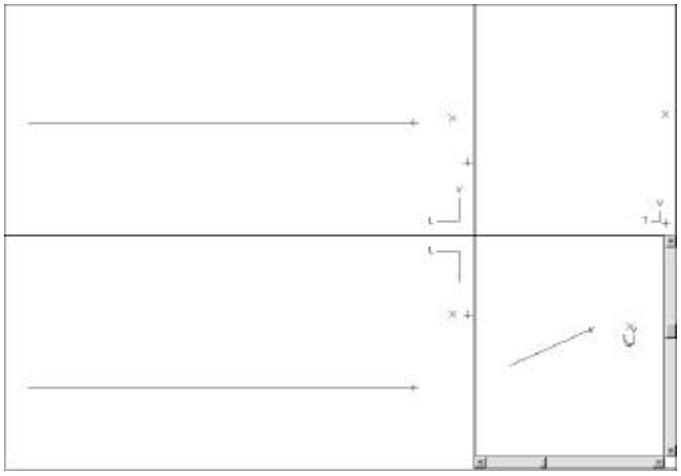

Figure 1-3: The curve Deck Edge

➲ To attach the forward end of curve Deck Edge to reference point Deckfor Curve:

1. With the curve *Deck Edge* selected, click the ***Edit Mode*** button. Ensure that you are working with control points rather than edit points: click the ***Control/Edit Points*** button and see the button change; for edit points, the points show on the curve in the button; for control points, they do not. On the main screen, control points are indicated by a circle at each vertex, edit points are indicated by a square.
2. Select the first vertex (vertex 0).
3. Click the ***Attach*** button.
4. In the *Attach Curve* dialogue, Under *Action*, pick ***Attach to Point***, under *Attach to point*, select ***Deckfor*** and click ***OK***. (See Figure 1-4.)

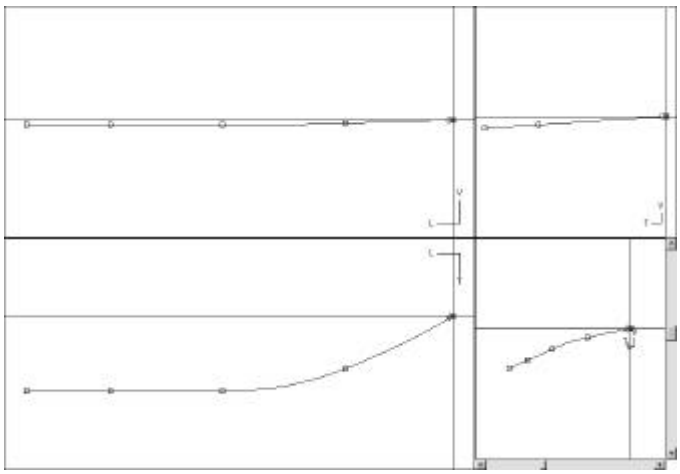

Figure 1-4: Attaching curve Deck Edge to point Deckfor

Note: we could have attached End 1 of the curve Deck Edge to the point Deckfor in the Create Curve dialogue. However, we wanted this curve to have shape and if we had made the attachment in the dialogue then the curve would have been a straight line between Deckfor and the coordinates 9a, 1.5, 0.8.

1.4 Creating the Chine Curve

We will create the chine in a multi-step process: first, we will create the point Chinefor to represent the forward-most point of the chine; then we will copy the deck edge; next we will the copy to the correct position; and then we will attach the forward end of the chine curve to the point Chinefor; and finally, we will re-shape the Deck Edge and Chine curves.

➲ To create a Reference Point for the forward end of the chine:

1. Click the ***Create Mode*** button.
2. Click the ***Create Point*** button.
3. In the *Create Point* dialogue, select the ***Free*** tab, type the following information, and click ***OK***. (See Figure 1-5.)

Field	Entry
Point Name	Chinefor
Location	L = 1; T =0; V = 0

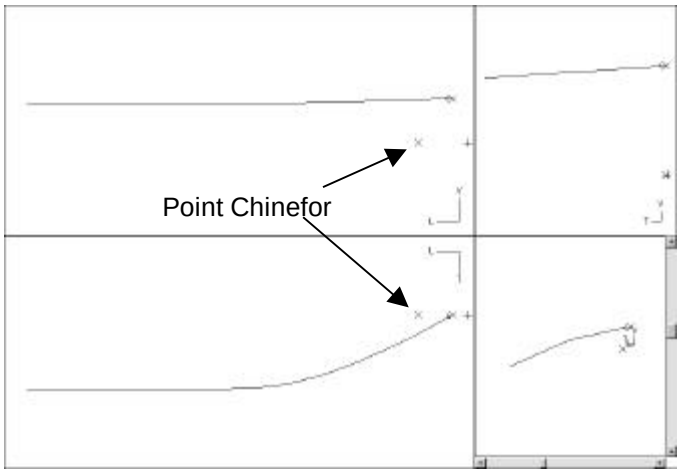

Figure 1-5: Reference point at the forward end of the chine

➲ To copy the Deck Edge curve to use as the Chine:

1. Click the ***S Side*** button to better see the objects of interest.
2. Select the curve ***Deck Edge***.
3. Hold down the **Shift** key and click on the ***Clone*** button to copy the *Deck Edge* curve.
4. In the ***Copy*** dialogue , under *Name for Copy* change the name to **Chine** and click ***OK***.

➲ To move the curve Chine:

1. Make sure that the curve *Chine* is selected. (If it isn't, press the **Spacebar** repeatedly until the *Object display box* at the top-right of the screen shows *Chine*.)
2. **Right-click** on the ***Move*** button.
3. In the ***Move Selection*** dialogue, for ***V*** enter ***–1.3*** and click on ***OK***. (See figure 1-6.)

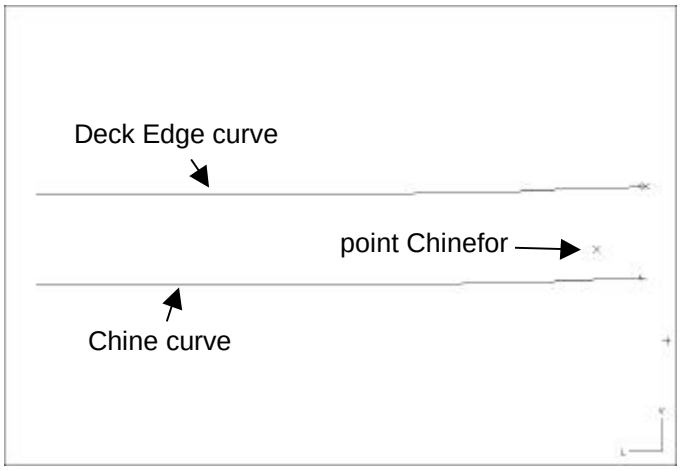

Figure 1-6: Chine curve moved down

➲ To attach the curve Chine to the point Chinefor:

1. Switch to ***Edit*** mode.
4. Select vertex 0 of the curve *Chine*, by clicking on that vertex. Watch the readout at the bottom of the screen to ensure vertex 0 is selected.
5. Click the ***Attach*** button.
6. In the *Attach Curve* dialogue , Under *Action*, pick ***Attach to Point***, under *Attach to point*, select ***Chinefor*** and click **OK**.

➲ To move the curve Chine inboard 0.3m:

1. Click the ***Top*** view button.
2. Switch to ***Select*** mode.
3. With the *Chine* curve selected, right-click on the ***Move*** button. In the ***Move Selection*** dialogue , for ***T*** enter **P0.3** and click ***OK***. (See Figure 1-7.)

Note: since the forward end of the Chine is attached to Chinefor, the curve will change shape when it is moved.

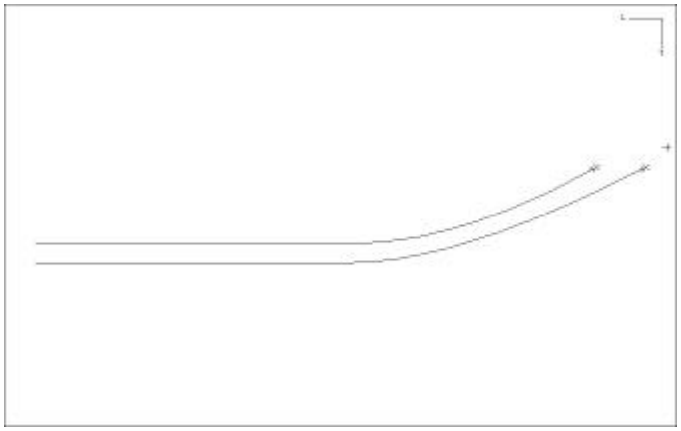

Figure 1-7: Chine and Deck Edge curves

➲ To edit the Deck Edge Curve:

1. Select the curve *Deck Edge*.
2. Switch to ***Edit*** mode.
3. Ensure ***Control Point*** mode is selected by clicking the ***Control/Edit Points*** button, if necessary. (For Edit Points mode, the button shows the vertices lying on the curve. For Control Points mode, the vertices do not lie on the curve.)
4. Move vertex 1 to L = 1.5A and vertex 2 to L = 4A by selecting each point and using the arrow keys on the keyboard. Watch the readout at the bottom of the screen to ensure the correct coordinates are obtained.
5. Fill out the *Chine* curve in the same manner. (See Figure 1-8.)

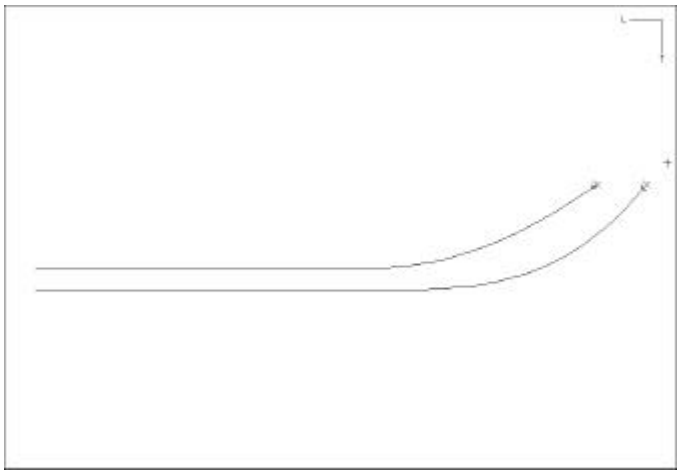

Figure 1-8: Deck Edge and Chine filled out

6. You may wish to switch to ***S Side*** view and edit the shape of the *Chine* curve by moving the vertices up or down.

1.5 Creating a Curve to Represent the Centreline

➲ To create a curve:

1. Click the ***S Side*** view button.
2. Click the ***Create Mode*** button.
3. Click the ***Create Curve*** button.
4. In the *Create Curve* dialogue , select the ***Free*** tab, type the following information, and click ***OK***. (See Figure 1-9.)

Field	Entry
Curve Name	Centreline
End 1	L = 1; T = 0; V = -1
End 2	L = 9; T = 0; V = -1

5. Following the method described previously, attach Vertex 0 of curve *Centreline* to the point *Chinefor*.

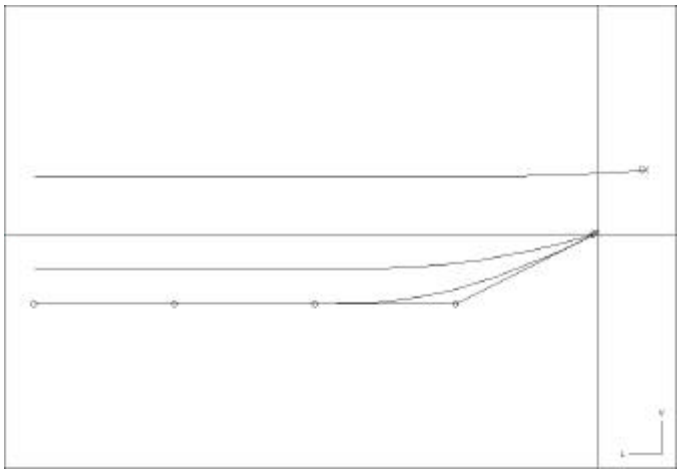

Figure 1-9: Centreline curve

1.6 Ensuring Fairness Along the Stem

This design calls for the stem section below the chine to fair smoothly into the straight stem section above the chine. Currently, the Centreline curve defines the stem below the chine and the straight stem is a line drawn from the forward end of the Deck Edge curve to the forward end of the Chine curve. In Autoship, the end of a curve lies at the same position as the end control point, and is tangent to a straight line drawn from the end control point to the next control point. Therefore, by re-positioning the second control point of the Centreline curve, we can ensure the stem section below the chine is tangent to the straight stem above the chine. To achieve this, the second control point could be located anywhere below the chine on the extension of the straight stem line.

➲ To position the vertex:

1. With the curve *Centreline* selected, and in ***S Side*** view, make sure you are in *Control Point* mode.
2. Select ***vertex 1***.

   Note: We have calculated the intersection of the straight stem line with V = -1 (the current height of vertex 1) to be at L = 1.83A

3. Move the control point forward to L = 1.83A by clicking in the L coordinate box near the upper left of the screen, typing ***1.83A*** and ***Enter***.
4. Adjust the curvature of the fore foot by moving *vertex 2* of the *Centreline* directly forward. (See Figure 1-10.)

   Tip: use the left-right arrow keys on the keyboard to avoid moving the vertex up or down.

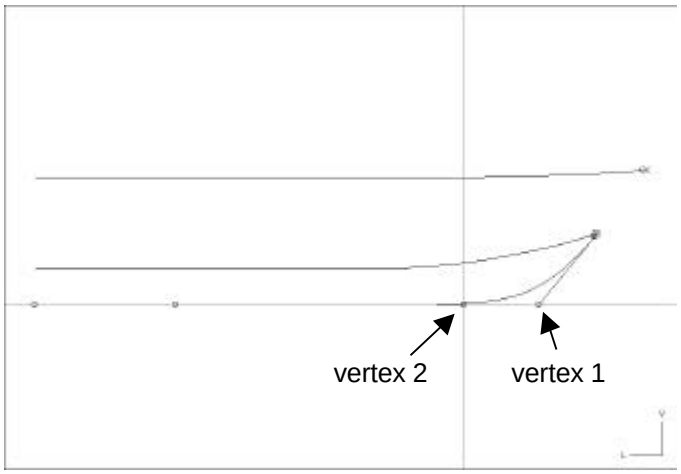

Figure 1-10: Fairing the stem

1.7 Creating the Bottom as a Developable Surface

In this exercise, we will create the bottom panel as a developable surface. Developable surfaces are favoured when working with sheet material, such as aluminum or plywood as the surface may be easily formed. In Autoship, developable surfaces are formed between two curves. If you change the shape of either of the curves after creating the surface, you can update the surface by selecting the surface and clicking on the Edit – Regenerate menu option.

➲ To create a developable surface:

1. Click the ***Create*** mode button.
2. Click the ***Create Surface*** button.
3. In the *Create Surface* dialogue, select the ***Develop*** tab, type the following information, and click ***OK***:

Note for Autoyacht Users: if you are an Autoyacht user, you will not be able to create a developable surface; instead, click on the Ruled tab and select the curves Chine and Centreline.

Field	Entry
Surface Name	Bottom
Curves	Chine Centreline
Tolerance	0.0005
Start Options	Straight
End Options	Extrapolate Curvature

Note: Start Options governs the beginning edge of the surface (r = 0), End Options governs the ending edge (r = 1). “Straight” causes the edge of the surface to be a straight line between the ends of the two curves. “Extrapolate Curvature” creates the surface as if it extended beyond the end of the curves but truncates it at the ends of the curves, thus creating a curved edge.

4. Click the ***Mesh*** button to see the ruling lines. (See Figure 1-11.)

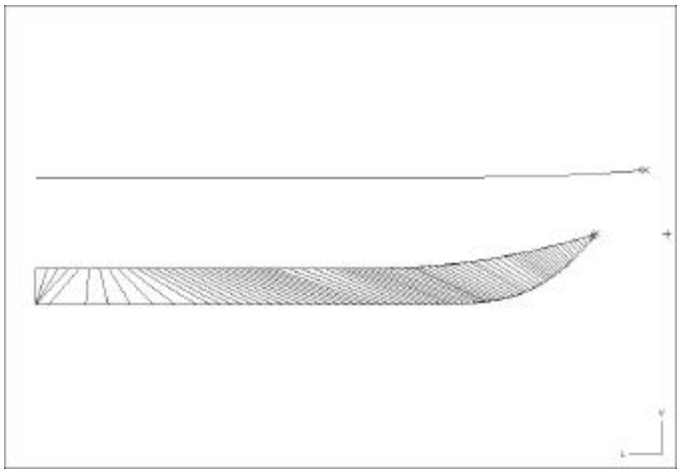

Figure 1-11: Bottom surface, profile view

1.8 Creating the Topsides as a Ruled Surface

Ruled and developable surfaces are similar, but not identical. Both surfaces are created by constructing a series of ruling lines between two curves. The rows of the surface are derived from the ruling lines. For a ruled surface, the ruling lines connect a given parameter value on one curve to the same parameter value on the other curve. The arrangement of the ruling lines on a developable surface depend on the relative shape of the two curves and are determined by extensive calculation. Since developable surfaces are merely sections of cones and cylinders linked together, the shape can be manufactured from sheet material with just simple straight-line bending. A ruled surface usually has compound curvature, which means the material must be plastically deformed to manufacture the shape.

➲ To create a ruled surface:

1. In ***Create*** mode, click the ***Create Surface*** button.
2. In the *Create Surface* dialogue, select the ***Rule*** tab, type the following information, and click ***OK***.

Field	**Entry**
Surface Name	Topsides
Curves	Deck Edge Chine

3. Click the ***Mesh*** button to see the ruling lines. (See Figure 1-12.)

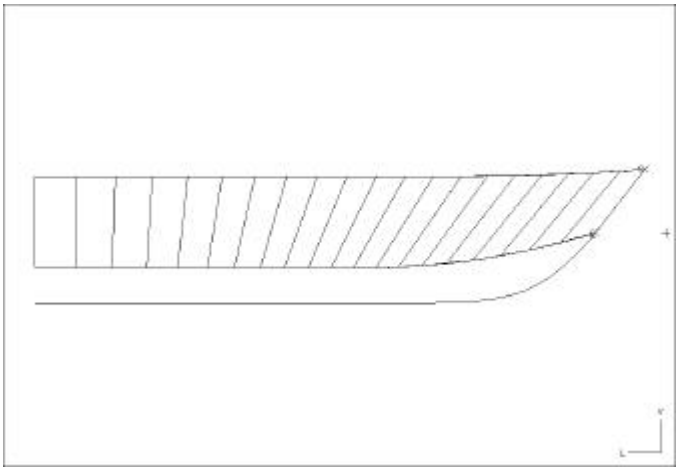

Figure 1-12: Topside surface, profile view (note the visibility of the Bottom surface has been turned off)

1.9 Seeing the Results

1. Click the ***Mesh*** button to turn off the surface mesh.
2. Right-click on the ***Contours*** button to open the Contours dialogue .
3. Set stations, waterlines, and buttocks appropriate to your model and click ***OK***.

   Tip: Click on the *Stations* radio button, set *Station Spacing* to ***1***, click on the *Group* button, set *Last* to ***10*** and *Step* to ***1*** and click on ***OK***. Click on the *Buttocks* radio button, click on the *Group* button, set *Last* to ***2*** and *Step* to ***.5*** and click on ***OK***. Click on the *Waterlines* radio button, click on the *Group* button and click on ***OK***.
4. Click the ***4 View*** button.
5. With both surfaces selected, click the ***Contours*** button.

6. Experiment with modifying the lines by moving the two reference points ***Deckfor*** and ***Chinefor*** and editing the shape of the curve ***Chine*** to see how they affect the two surface panels. Remember to update each surface by selecting it and then clicking on ***Edit - Regenerate***.

1.10 Adding Flair to the Topsides

In this exercise we will add another column of vertices to the Topside surface so that we can introduce some flair at the forward end.

➲ To add another column of control points and fair the shape:

1. Turn off the contours by clicking on the ***Contours*** button.

2. Select the surface ***Topsides***, and in ***Edit*** mode, pick the upper column of control points.

Note: The rows and columns of a surface may not be oriented the way you expect. For ruled and developed surfaces, the rows always match the curves used, thus the columns are oriented the opposite way. For columns, the *Rows / Columns* button should show vertical lines.Click on the ***Add Row or Column*** button (make sure you are working with columns, not rows).

---

3. Insert a column at a proportion of .35. (This percentage works out well for the flair shape.)

4. With *Topsides* selected, switch to ***Edit*** mode.

5. In the top view, move the *second vertex* (Row 1, Col 1), and perhaps the third also, aft and inboard slightly. (See figure 1-13.)

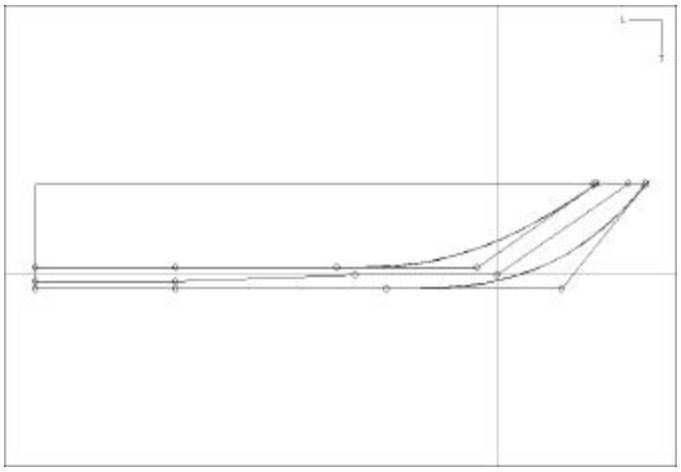

Figure 1-13: Editing column 1 of Topside surface

6. View the results by turning on the *contours*.

You will notice that the stations display as straight lines rather than curves. This is because the curvature and resolution of the surface Topsides are still at the low values that a ruled surface is created with.

7. Go to the ***Attributes*** dialogue (***Edit – Attributes*** menu) and, for the surface *Topsides*, set the *CDeg* to ***2*** and *CRes* to ***20***.

Note: if you move the Col 1 vertex at the stem to develop curvature in the stem, you will destroy the smooth transition of the stem to the fairbody. If a curved stem is desired, you will have to calculate a new position for vertex 1 of the curve Centreline based on the relative positions of surface Topsides Col 1 and Col 2 vertices at the stem, move the vertex and then regenerate the Bottom surface.

---

8. Turn off contours and switch to ***Select*** mode to begin the next exercise.

1.11 Adding a Transom

This exercise introduces the Arc curve type. The arc will be extruded to form a cylindrical transom which will then moved into position and trimmed off at the hull. This transom will have a 4m radius and be raked aft 9 degrees.

➲ To create the 4m radius curve:

1. In ***Create Mode***, click the ***Create Curve*** button.
2. In the *Create Curve* dialogue, select the ***Arc*** tab, type the following information, and click ***OK***: (see figure 1-14.)

Field	Entry
Name	Transom Shape
Plane	*Top*
Centre	*L* = 0, *T* = 0, *V* = 0
Angles	*Start* = 0 *End* = 25
Semi Axes	*Circular* checked *Radius* = 4

Note: we estimated that an angle of 25 degrees would produce a curve slightly wider than the half-breadth of the hull.

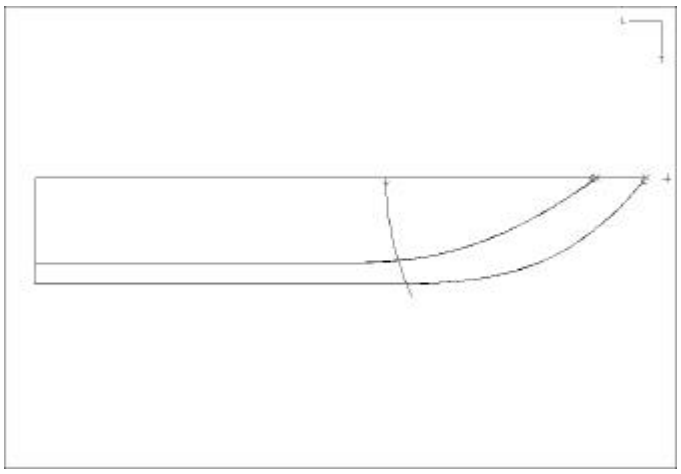

Figure 1-14: Transom Shape curve

➲ To create the transom surface:

1. In ***Create Mode***, click the ***Create Surface*** button.
2. In the *Create Surface* dialogue, select the ***Extrude*** tab, type the following information, and click ***OK***: (see figure 1-15.)

Field	Entry
Surface Name	Transom
Curve	*Transom Shape*
Vector	L = 0, T = 0, V = 2
Column Mesh	*Number* = 2 *Max Degree* = 1

Note: we could have created the transom at the intended rake angle by specifying an "L" component for Vector. However, this would not have resulted in a true cylindrical surface.

3. Click the ***S Side*** view button.

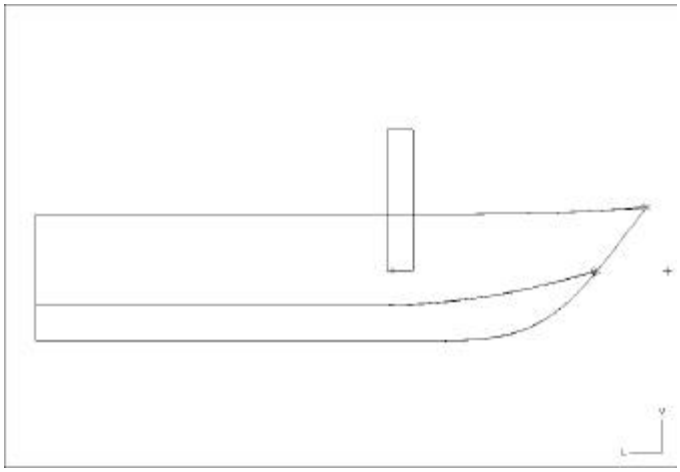

Figure 1-15: The Transom surface

➲ To rotate the Transom surface:

1. Select the ***Transom*** surface.
2. Right-click on the ***Rotate*** button.
3. In the ***Rotate*** dialogue, type the following information and click **OK**: (See Figure 1-16.)

Field	Entry
Axis	*Transverse*
Angle	9
Centre	*L* = 0, *T* = 0, *V* = 0

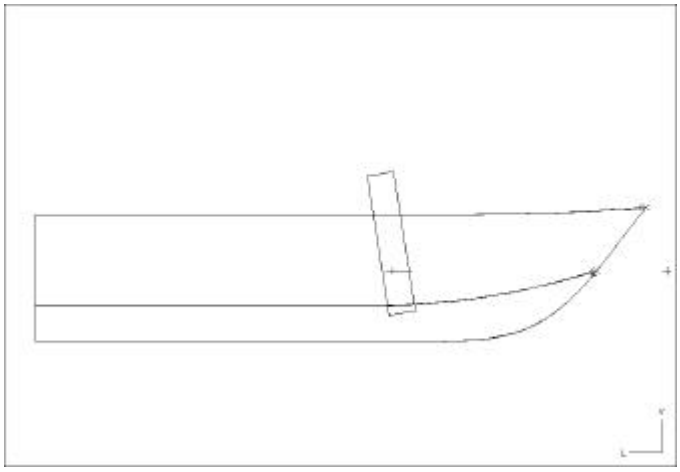

Figure 1-16: The Transom surface rotated

➲ To reposition the Transom surface:

1. With the *Transom* surface selected, click the ***Move*** button.
2. Move the *Transom* surface to a position at the aft of the vessel where it extends above the Topside surface and below the Bottom surface, and then click the mouse button.

Tip: Remember that the displacement of the object originates at it's base point – which in this case is at the origin. (See figures 1-17 and 1-18.)

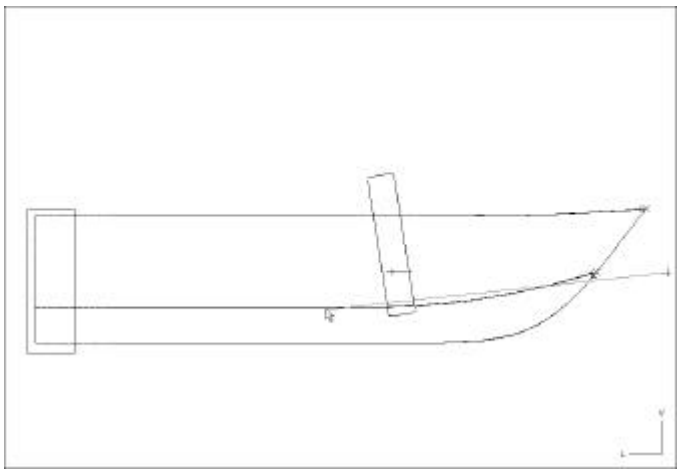

Figure 1-17: Moving the Transom surface

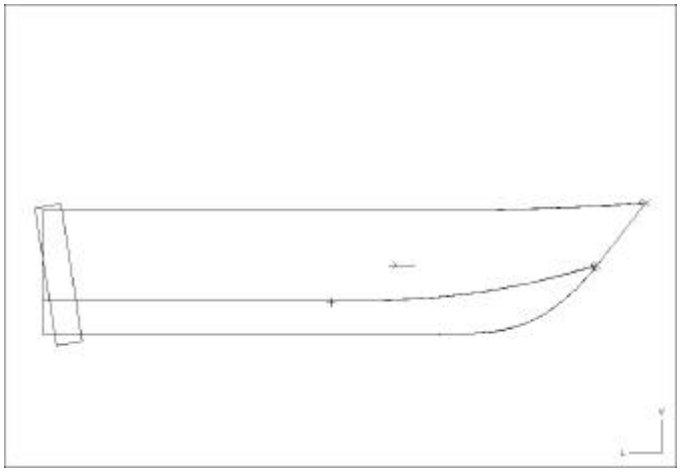

Figure 1-18: The Transom surface repositioned

In order to trim the transom to the hull, we must obtain the curves describing the intersection of the Transom with both the Topside and Bottom, and then revise these curves to produce a single curve with which to trim the transom.

➲ To generate the intersection curves:

1. Select both the *Transom* surface and the *Topsides* surface.
2. Click the ***Surf-Surf Intersect*** button.
3. In the ***Surf-Surf Intersect*** dialogue, ensure that the correct surfaces - *Topsides* and *Transom* - are displayed, enter a meaningful tolerance, i.e***. 0.0005***m and click on ***OK***.

This process will generate two intersection curves – one embedded on the Transom surface, the other embedded on the Topsides surface. We should rename them using more meaningful names.

We will use a naming convention that will appear in subsequent tutorials: the curve Topsides Int1 should be renamed to Transom-Topsides and the curve Transom Int1 should become Topsides-Transom.

4. Select ***Edit – Attributes,*** to call up the Attributes dialogue. Click on the ***curves*** button within that dialogue.
5. Click on the name *Topsides Int1* and then click on the ***Properties*** button.
6. Rename *Topsides Int1* to ***Transom-Topsides*** and click ***OK***.
7. Similarly, rename *Transom Int1* to ***Topsides-Transom***.
8. Select both the Transom surface and the Bottom surface.
9. Click the ***Surf-Surf Intersect*** button.
10. In the ***Surf-Surf Intersect*** dialogue, ensure the surfaces Bottom and Transom are selected, enter a meaningful tolerance, i.e. ***0.0005***m and click on ***OK***.
11. Rename the curve ***Transom Int1*** to **Bottom-Transom**.
12. Rename the curve ***Bottom Int1*** to **Transom-Bottom**.

➲ To revise the intersection curves:

1. In the *Attributes* dialogue, with *Curves* selected, click on the ***None Visible*** button, and then click in the ***Vis.*** column for *Transom-Bottom* and *Transom-Topsides* to turn on their visibility. Click on ***OK***.
2. At the main screen, click on the ***Show Surfaces*** button to turn off the visibility of surfaces.
3. Click on the ***Show Points*** button to turn off the visibility of points.
4. Switch to ***Front*** view.

5. Select the ***Transom-Bottom*** curve and switch to ***Edit*** mode.
6. Click on the ***Join Curves*** button.
7. In the ***Join Curves*** dialogue, under *New Name* rename the curve to ***Transom Trim***, under *Join Transom-Bottom* + select the curve ***Transom-Topsides***, un-check ***Keep Originals*** and click on ***OK***.

In order to use the new curve Transom Trim to trim the Transom surface, it must be a “closed” curve. This means both ends of the curve must either contact a surface edge or be positioned at the same location. In the absence of a deck surface with which to obtain an intersection curve to trim off the top of the Transom surface, we will extend the top of the curve Transom Trim to the centreline edge of the surface.

8. Switch to ***Edit*** mode.
9. Click on the ***upper-most vertex***. Ensure you have selected the end vertex by pressing the ***F4*** key a few times and watching the *Vert* display at the bottom of the screen – when the number does not change, the end vertex is selected.
10. Insert another vertex at the same location as the end vertex: press the ***F3*** key once and then click on the ***Add Control Pt*** button. In the *Insert* dialogue, set *Proportion* to ***1*** and click on ***OK***.
11. Press the ***F4*** key to move to the new end vertex.
12. Move the new end vertex to the centreline: click in the *T* coordinate box at the upper left of the screen, type ***0*** and ***Enter***.

13. If necessary, use the arrow keys on the keyboard to move the vertex up or down to cause the upper portion of the curve to be horizontal.

14. Switch to ***Select*** mode. (see figure 1-19)

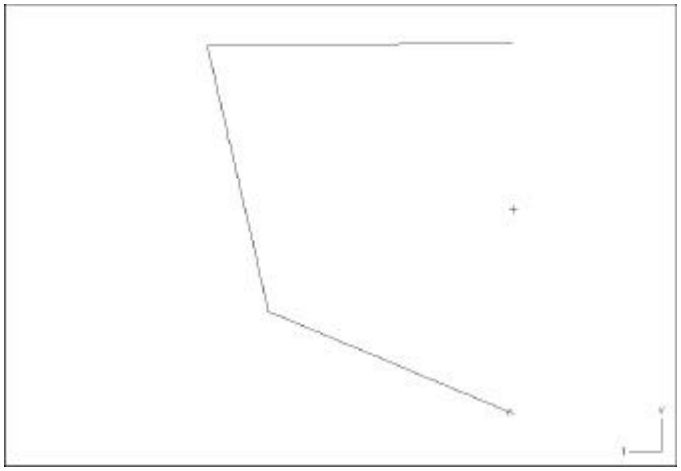

Figure 1-19: Revised Transom Trim curve

➲ To trim the Transom surface:

1. Turn on the visibility of surfaces by clicking on the ***Show Surfaces*** button.
2. Select the surface ***Transom***.
3. Click on the ***Trim*** button.
4. In the *Trim Surface Transom* dialogue, under *Available Curves* select ***Transom Trim*** and click on ***OK***. Answer ***Yes*** to the appropriate *Keep this Domain ?* prompt and ***No*** to the other.

➲ To trim the Topsides surface:

1. Click on the ***4 View*** button.
2. Select the surface ***Topsides***.
3. Click on the ***Trim Surface*** button.
4. In the *Trim Surface Topsides* dialogue, under *Available Curves* select ***Topsides-Transom*** and click on ***OK***. Answer ***Yes*** to the appropriate *Keep this Domain ?* prompt and ***No*** to the other.

➲ To trim the Bottom surface:

5. Select the surface ***Bottom***.
6. Click on the ***Trim Surface*** button.
7. In the *Trim Surface Bottom* dialogue, under *Available Curves* select ***Bottom-Transom*** and click on ***OK***. Answer ***Yes*** to the appropriate *Keep this Domain ?* prompt and ***No*** to the other.
8. Create a group of the three surfaces:
   a. Select the three surfaces – select one, and then while holding down the shift key, select he other – you must click on the outline of the surface. Note that the object box at the top of the screen displays “Multiple Select”.
   b. Select ***Arrange – Group***.
   c. In the New Group dialogue, enter a name for this group. For now, ignore the other settings in the dialogue.
9. View the contours.
10. View the rendering.

1.12 Generating Offsets

In Autoship, you can obtain offsets for both surfaces and curves. In this exercise, we will add the centreline, chine and deck edge curves to the group just created and then obtain offsets for the entire group.

➲ To add curves to the group:

1. With the group just created selected, click on the button showing ***Group*** in the top right corner of the screen to go the the ***Group Editor***.
2. Add the ***Centreline*** curve: under ***Objects***, click on the ***Curves*** radio button; from the pick list below that, select ***Centreline***; click on the ***Add*** button.
3. Add the ***Chine*** and ***Deck Edge*** curves the same way and click on ***OK***.

➲ To obtain an Offsets report:

1. Click on the ***Report - Create – Offsets*** menu option. At the ***Offset Table*** dialogue, set a meaningful tolerance and click on ***OK***.
2. The Report Editor will appear and show the offsets table. Note that the first three tables indicate station, buttock and waterline locations. The fourth table shows heights and the fifth table shows half-breadths at the currently defined stations, buttocks and waterlines.

1.13 Obtaining Hydrostatics

Autoship offers two type of hydrostatics. The Instant Hydrostatics option displays hydrostatic values on a “floatable” screen and is for use while fairing surfaces. The Report method produces a full hydrostatics report in the report editor.

➲ To obtain Instant Hydrostatics:

1. Click anywhere on the main screen, or click on the "**X**" button at the top right of the report editor to return to the main screen.
2. Click on the small ***H*** button near the lower left of the screen to open the Instant Hydrostatics window.
3. In the *Instant Hydrostatic* window that appears, click on the ***Group*** button.
4. In the *Choose Hydrostatics Group* dialogue that appears, select the group you last created and click on***OK***. (Note that you must actually click on the name.)
5. In the *Instant Hydrostatic* window, click on the ***Update*** button.

   Note: the *Depth* box shows the "default" depth value originally entered with the Project Info when setting up the project. This is the distance from the origin to the waterline. You can type a different value in the Depth box, or restore the default by clicking on the R button to the right of Depth. The Update button calculates the hydrostatics for this depth. Alternately, you can type in values for Mass and LCG and click on the S button to the right of Mass to have Autoship find the depth and trim corresponding to the given mass and LCG and then show the hydrostatics for that waterline.
6. Click on the small ***H*** button near the lower left of the screen to close the Instant Hydrostatics window.

➲ To obtain a Hydrostatics Report

1. Select the group you last created.
2. Click on the ***Reports - Create - Hydrostatics*** menu option.

3. In the *Hydro Parameters* dialogue that appears, set the depth to ***-.4*** and click on ***OK.***

4. The Report Editor will open up and display the hydrostatics just calculated. Re-size the report window to full size by clicking in the middle box in the upper right corner. The scroll bar at the right can be used to shift the report up or down.

5. When finished, click on the ***Clear Report*** button (the "x" button) near the upper left corner and answer***No*** at the *…save changes* prompt. This will erase this report.

   Note: a new report is appeneded to the bottom of the existing report. Clearing the report ensures you will see the next report immediately upon opening the report editor.

6. Click on either the "-" button or the "X" button at the upper right to return to the Autoship main screen.

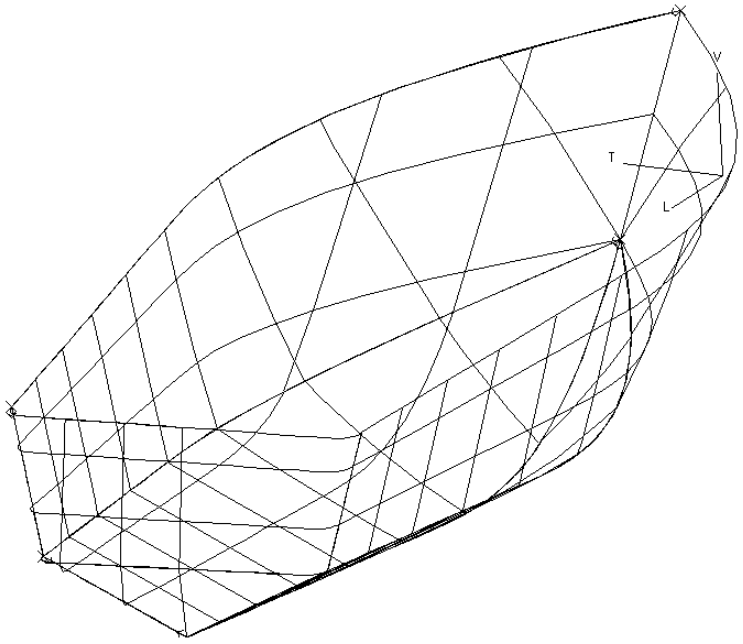

Figure 1-20: Chine hull planing boat

1.14 Summary

You have been shown how to create two more types of surfaces—the developable and ruled surfaces—and you have seen how these surfaces depend upon curves. You have also worked with embedded curves and have explored the uses of reference points for attachments. And you have also seen how to obtain offsets and hydrostatics.

Tutorial 2 introduces additional methods of creating and refining curves and surfaces: it demonstrates two methods for forming a hull shape (by extruding a surface from a midship section, and by lofting a surface through given sections); and one method to create and fit a deck.

Tutorial 2 – Round Bilge Hull

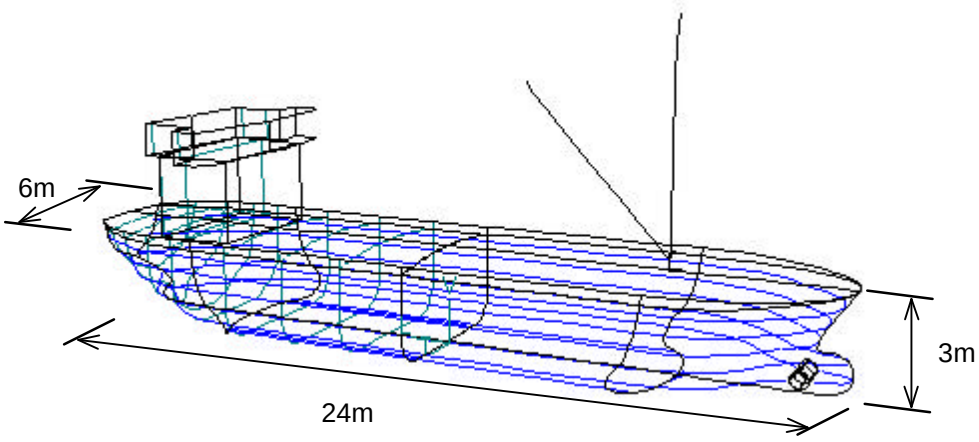

Figure 2-1: Round Bilge Hull

2.1 Introduction - Round Bilge Hull

The project in Tutorial 2 is a general cargo vessel of radius-bilge design. It also has a bulbous bow with thruster and a faired-in stern bulb.

Tutorial 1 showed how to create a hull using developable and ruled surfaces, and demonstrated how to create and fit a cylindrical transom. It also introduced the idea of attaching the end of a curve to a point and discussed the topic of tangency at the end of a curve. In addition, it showed how to obtain offsets and hydrostatics.

Tutorial 2 introduces additional methods of creating and refining curves and surfaces: it demonstrates two methods for forming a hull shape (by extruding a surface from a midship section, and by lofting a surface through given sections); and one method to create and fit a deck.

The learning objectives in Tutorial 2 are to:

- Create a midship section with a radius bilge
- Generate a forebody shape as an surface extruded from the midship section
- Develop the bow rounding
- Work with shape and fairness
- Control flat of side
- Generate the aftbody shape as a surface lofted from curves
- Add a bow thruster by projecting a curve
- Add a bow thruster by surf-surf intersection
- Add a deck with camber and shear

The model we have constructed, **AS8TUT2.PR3**, has the following particulars:

overall length:	25m
breadth:	6m
depth:	3m
bilge radius:	1m
origin:	under keel amidships

You may want to review this model before continuing.

2.2 Starting Out

➲ To create a new project:

1. Select ***File - New*** to prepare Autoship for a new project.
2. Select ***Settings - Units***. Pick Meters and Tonnes, and click ***OK***.
3. Select ***Settings - Preferences***.
4. Click the ***Coordinate System*** button.
5. Click the ***Naval/Aircraft (US)*** radio button. This orients the coordinate system so that aft, starboard, and up are positive directions while forward, port, and down are negative directions.
6. Click ***OK*** in the Coordinate System dialogue box and click ***OK*** again in the Preferences dialogue box.
7. Select ***File - Save As***. In the *Save Project* dialogue box that appears, type **TUT2.PR3** for the File Name. Save the project in your new directory called **MYTUTS**.
8. In the *Project Info* dialogue box, under *Name*, type a descriptive message for the project, such as “Tutorial 2”. Note the *Design Depth* box at the bottom of the dialogue – this is to provide a starting point for the Instant Hydrostatics calculations. Click ***OK***.

2.3 Creating the Midship Section

In this exercise you will learn how to create and form curves by creating a midbody curve and matching it to a circle. You will also learn how to create a surface by extruding the midbody curve forward to form the forward half of the hull.

➲ To form the Midbody Curve:

1. Click the ***Create Mode*** button.
2. Click the ***Create Curve*** button.
3. In the *Create Curve* dialogue box that appears, select the ***Free*** tab to create a free curve.
4. Type the following information and click ***OK*** when you have finished.

Field	Entry
Curve Name	Midship Curve
Parameters	Control Pts = 3 MaxDegree = 2
End 1	L = 0, T = 0, V = 0
End 2	L = 0, T = 3, V = 3 (the values of max beam and depth)

Note: For this model, the longitudinal origin is at midships. We will create the midship curve at L = 0, thus positioning it at midships. This first step will also set up the basis for the model's coordinate system.

Your screen should look like Figure 2-2.

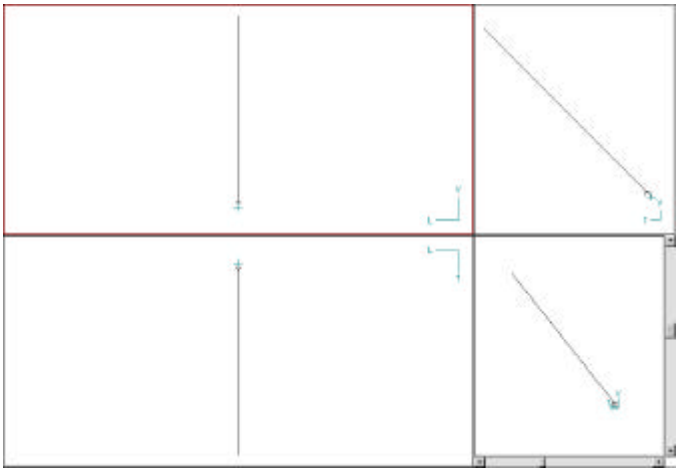

Figure 2-2: Midship Curve

To form the basic hullshape at midships:

1. Click the ***Front*** view button.
2. If the Midship Curve is not selected (when selected, the object's name appears in the Object box at the top of the screen, and it's coloured red on screen) select it by pointing at it with the mouse and clicking the left mouse button.
3. Click the ***Edit Mode*** button. This places the program in Edit mode.

   Ensure that you are working with control points rather than edit points. Click the ***Control/Edit Points*** button and see the button change. For edit points, the points show on the curve in the button; for control points, they do not. On the main screen, control points are indicated by a circle at each vertex, edit points are indicated by a square.
4. Select the middle vertex and drag it to max beam and to the baseline. Watch the readout at the bottom of the screen. Release the mouse button when the is at vertex at *T = S3* and *V = 0*. (See Figure 2-3.)

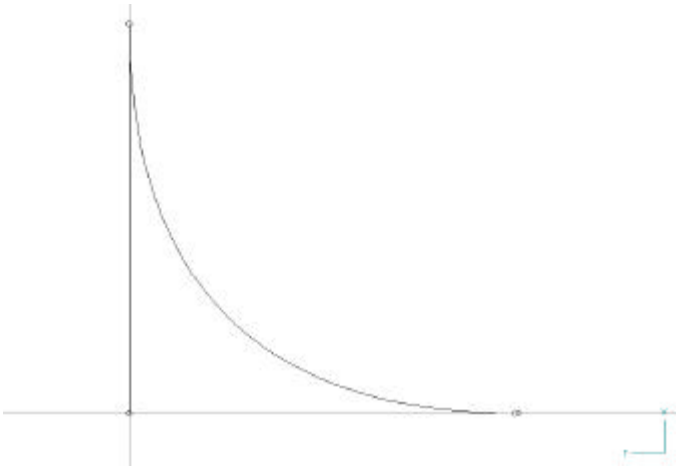

Figure 2-3: Front view of Midship Curve

It may help if you are working in Grid Mode. Click the ***Snap/Grid*** button to toggle between Snap and Grid so that the button displays Grid.

> **Note:** Select a point by clicking on it. Drag a vertex by pointing at the selected vertex, holding the left mouse button down, pulling the vertex to the intended position, and then releasing the mouse button. You can also move a vertex by selecting it, re-positioning the cursor with the arrow keys on the keyboard and then pressing ***Enter***.

4. Make the middle vertex into a knuckle point by clicking the ***Toggle Corner*** button. Note that the vertex changes to green in color.

➲ To form a radius corner:

1. While still in *Edit Mode*, with the middle vertex selected, click the ***Radius Corner*** button.
2. In the *Fillet* dialogue box, specify a radius of ***1.0***, and click **OK**.

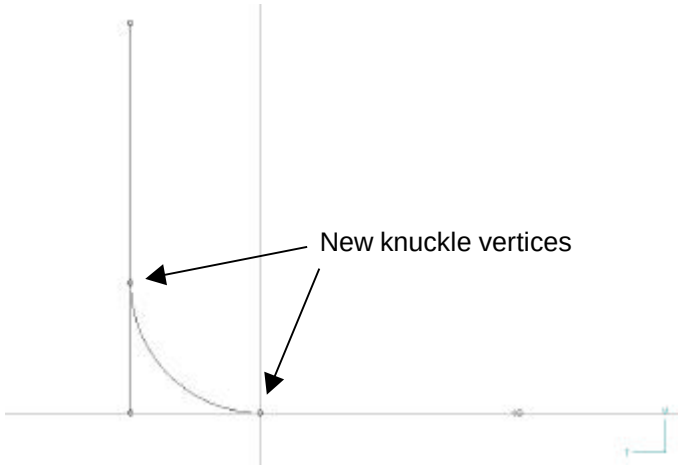

Figure 2-4: Midship Curve with two new vertices as knuckles

Note: This adds two new vertices as knuckle points and turns the selected corner vertex back into a regular vertex. (See Figure 2-4.) A surface extruded from this curve will have two knuckle lines (rows) along the entire length of the extrusion. With a little work, we will now make their knuckle points disappear.

➲ To make a Bilge Circle template:

1. Click the **Create Mode** button.
2. Click the **Create Curve** button.
3. In the Create Curve dialogue box that appears, select the **Arc** tab, type the following information and click **OK** when you are finished. (See Figure 2-5.)

Field	Entry
Curve Name	Bilge Circle
Plane	Front
Centre	T = S2 V = 1
Semi-Axes	Circular = checked Radius = 1.0
Angles	Start = 0 End = 360

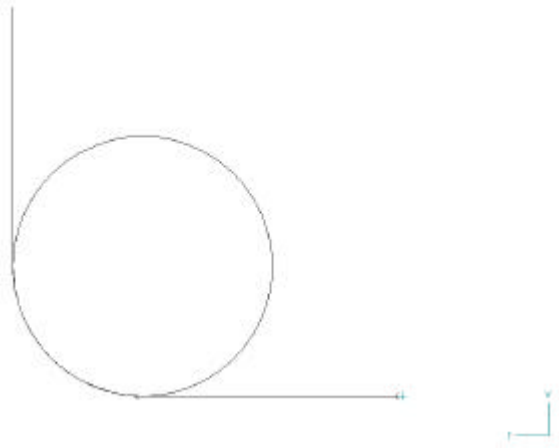

Figure 2-5: Bilge Circle Template

➲ To match the Midship Curve to the Bilge Circle Template:

1. Select ***Midship Curve***. (Press the space-bar to scroll through the objects)
2. Go to ***Edit Mode***.
3. To insert a new vertex after vertex 1, click on vertex 1. (Make sure you are still in Control Point mode.)

Observe that the vertices are numbered starting at 0, not 1. Look at the *Vertex Selection* display at the bottom of the screen to make sure you have the correct vertex.

4. Click the ***Add Control Pt*** button.
5. In the *Insert* dialogue box, select ***Refine*** and click ***OK***.

➲ To form the Midship Shape: In order to avoid having longitudinal knuckles in the hull shape, we must remove the knuckle points in the Midship Curve.

1. Select **vertex 1** and click the **Corner** button. Then select **vertex 4** and click the **Corner** button. This turns the two knuckle points into regular points.
2. Use the arrow keys on the keyboard to move the lower points (vertices 1 and 2) transversely and the upper points (vertices 3 and 4) vertically until the bilge radius portion of the midship curve is the same shape as the template curve. You may have to adjust the snap value in order to move the vertex by small distances. (See Figure 2-6.)

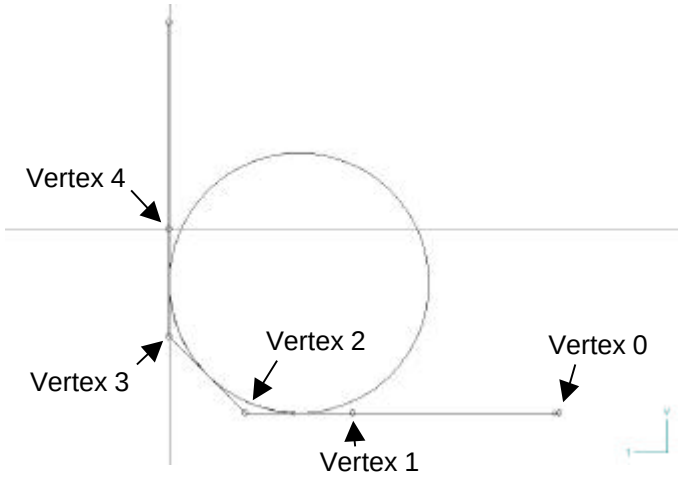

Figure 2-6: Matching the Midship Curve to the template

Note: The curve generated in Figure 2-6 is tangent to the straight line segment between adjacent, normal vertices, at the halfway point of the straight line segment. The fillet operation creates an exact circular arc while the matching operation creates a parabolic approximation.

2.4 Generating the Forebody Surface

This exercise introduces surfaces by using the extrusion surface type. The extrusion method creates a surface by advancing a given curve along a straight path, thus sweeping out the surface. We will extrude the midship curve to create a surface that will become the forward part of the hull.

➲ To create the surface:

1. Click the ***Create Mode*** button.
2. Click the ***Create Surface*** button.
3. In the *Create Surface* dialogue box, select the ***Extrude*** tab, type the following information, and click ***OK***:

Field	Entry
Surface Name	Foreship
Curve	Midship Curve
Vector	L=F12 (the entire forebody length from midship)
Col Mesh	Number = 3, Max Degree = 2

Note: This creates a surface with three columns, as specified, and six rows copied from the six vertices on Midship Curve. (See Figure 2-7.)

4. Click the ***Para*** view button. Rotate the view with the scroll bars at the right and bottom to get a good look at the surface.

5. Click the ***Edit Mode*** button.

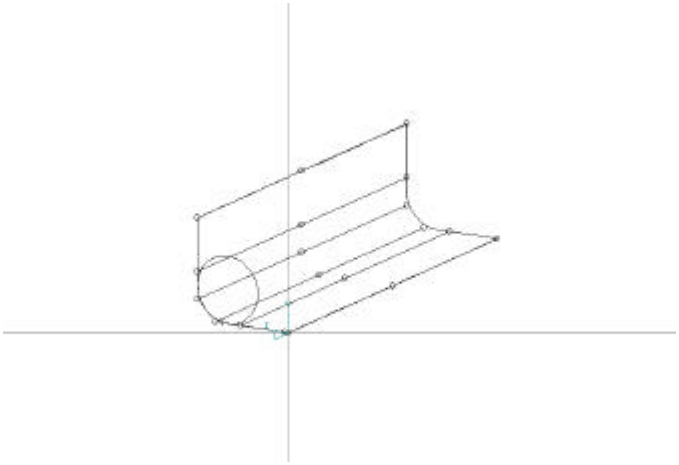

Figure 2-7: The Foreship Surface

➲ To close the Forward End (i.e. start to form the bow):

1. Click the ***S Side*** view button.
2. Ensure you are working with columns by clicking the ***Rows/Columns*** button, if necessary.

   Note: In the Rows/Columns button, rows appear as horizontal lines and columns as vertical lines.

3. To select the column at the forward end (right side), click its top control point. The readout at the bottom of the screen should display *Row 5*, *Col 2*.
4. Click the ***Isolate*** button to isolate only the forward-most column.

   Note: The Isolate button now shows only one point, in the middle of a cross.

5. Click the ***Front*** view button.
6. Reposition the top control point (verterx 5) of this column at the centreline by typing 0 in the T coordinate box in the

coordinate boxes at the top left of the screen, and then hitting Enter. (See Figure 2-8.)

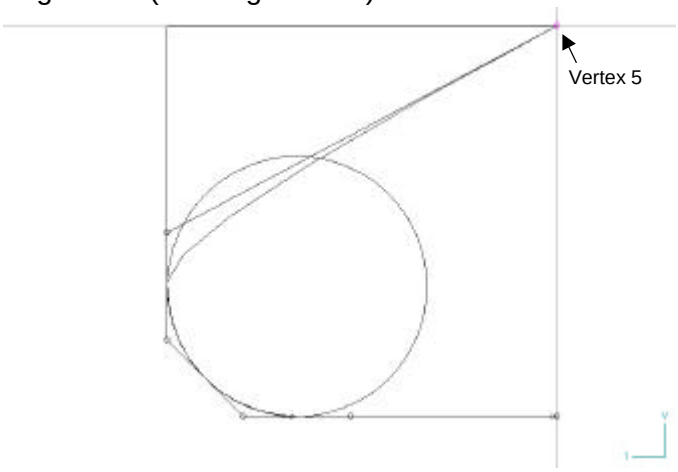

Figure 2-8: Top vertex moved to centreline

8. With the topmost vertex selected, click the ***Straighten*** button and then click on the vertex at the baseline/centreline. This will move all the vertices in between the two vertices you selected to the centreline. (See Figure 2-9.)

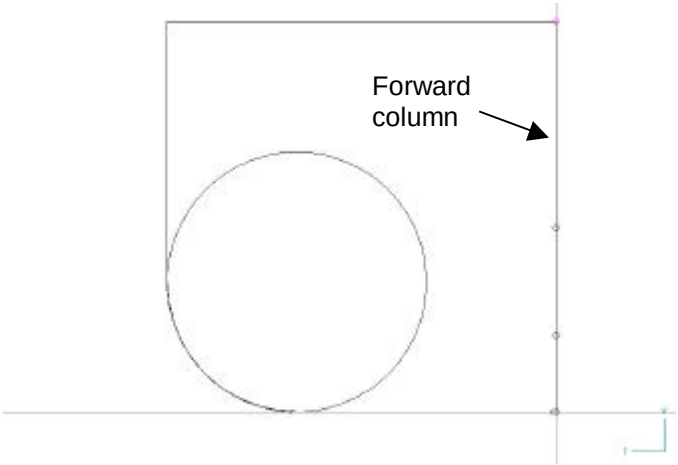

Figure 2-9: Forward column straightened

9. Click the ***Top*** view button to see the effect. (See Figure 2-10.)

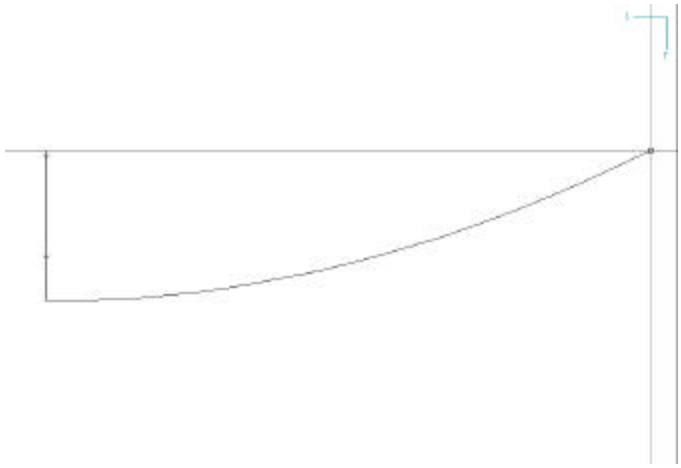

Figure 2-10: Top view

➲ To form the Stem:

1. Click the ***S Side*** view button. (Make sure *Foreship* is still selected and you are in *Edit Mode*.)
2. Click the ***Control/Edit Points*** button to switch to Edit Point mode.
3. Select the bottom vertex. Then, watch the Row display at the bottom of the screen and press the F4 key repeatedly to switch to the next vertex up the column. Note the position of each vertex, as some are obsured by others. Press the F3 key to switch to the next vertex down the column.
4. Move the edit points longitudinally and vertically until you get the desired bow profile. You can drag the vertices with the mouse, or move them with the arrow keys on the keyboard. You may have to reduce the Snap value to have the vertex move by a small enough distance. Adjust the Snap value by clicking on the up-down arrows just to the right of the Snap window, or by pressing the “+” or “-“ keys

on the keyboard, or by typing a new value into the Snap value box. (See Figure 2-11.)

Figure 2-11: Side view

5. Ensure that the bottom of the bulb fairs smoothly into the flat bottom by switching to ***Control Point Mode*** and positioning the Row 0 and Row 1 vertices at V = 0.
6. Change back to displaying all columns by clicking the ***Isolate*** button.
7. Select ***File - Save*** and save your work so far.

2.5 Controlling Bow Rounding

In this exercise, we will round out the bow (in plan view) by adding another column of vertices and then use this column in conjunction with the column at the centreline to form the bow radius and also to ensure tangency across the centreline.

➲ To add a new column to the Stem:

1. Select the middle column (Col 1) by clicking on one of it's vertices. (To do this, Foreship must be selected and you must be in Edit Mode.)

2. Click the ***Add Row or Column*** button. (Make sure you are working with columns, not rows.)

Note: The Add Row/Column button is not activated when a vertex at the 'ending' edge of a surface is selected (i.e. row 5.) You must be on any other vertex in the column to have the Add Row/Column button activated.

3. In the *Insert* dialogue box, select ***Proportional Interpolation*** and specify a proportion of ***1*** and click ***OK***. (This places the control points of the new column at the same location as those of the "next" column - the column at centreline. You can see this by toggling between Control Point Mode and Edit Point Mode.) (See Figure 2-12.)

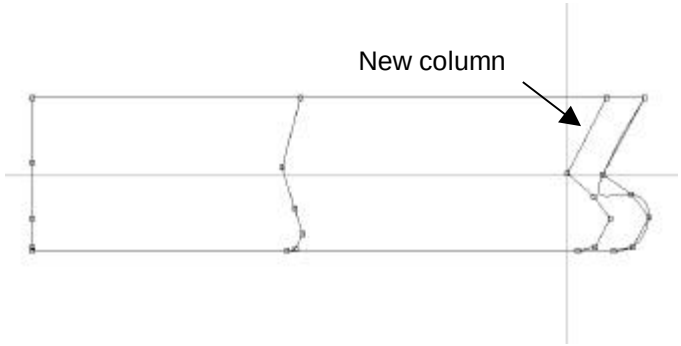

Figure 2-12: New column added

4. Click the ***Lock*** button.

In the *Surface CP Locking* dialogue box, select ***Relative to Next***, check the ***Longitudinal*** and ***Vertical*** check boxes and click ***OK***. Note that the vertices change to purple to indicate that they are locked vertices - now none of the points on this column can be moved longitudinally or vertically - only transversely. (At it's edges, a surface is always tangent to the straight line drawn between the edge control point and the next point inside the surface. Thus, this arrangment with the vertices on the new column exactly outboard of the vertices on the centreline

column forces the forward edge of the surface to meet the centreline at 90 degrees, as seen in plan view.)

➲ To form the shape:

1. Click the ***4 View*** button and then in the *Front* view window.
2. Switch to ***Edit Points Mode***.
3. Working only on the new column in the Front view, move the vertices away from the centreline to create the necessary rounding. Remember that these points are on the surface. Tangency is ensured across the bow because these points can only be moved transversely.
4. Switch to ***Select*** mode.
5. Click the ***Surface Mesh*** button to see the result. (See Figure 2-13.)

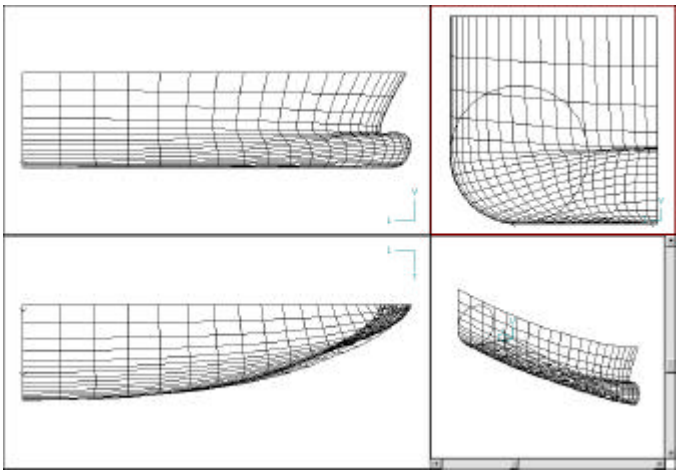

Figure 2-13: Creating bow rounding

2.6 Controlling Shape and Fairness

This exercise examines some of the functions and tools used for fairing.

➲ To fair the shape of Foreship:

1. Turn off ***Surface Mesh***.
2. Switch to ***Edit*** mode.
3. Select the column ahead of the midship column (Col 1).
4. Click the ***Add Row or Column*** button. (Remember, you can't add another column when the 'ending' edge of the surface is selected.)
5. In the Insert dialogue box that appears, select ***Proportional Interpolation*** and specify a proportion of ***0.75***. Click ***OK***.
6. Select ***Settings - Contours***. To help visualize the surface, we will now set some stations, waterlines and buttocks.
7. In the *Contours* dialogue box:

   Set stations as follows: note that *Location of Sta.0* is set to 0 (midship) and *Station Spacing* is set at 1m. Click on the ***Stations*** radio button, click ***Group***, set *Last* to ***12F***, and click ***OK***. This will create a group of stations from 12F to 0 at every 1m.

   Set buttocks: Click the ***Buttocks*** radio button, click the ***Group*** button, and set *First* to ***0.0***, *Last* to ***2.5***, and *Step* to ***0.5***, and then click ***OK***. This creates buttocks from 0 to 2.5s at every .5m.

   Set *Waterlines* at ***0*** to ***2.5*** at a *Step* of ***0.5***. When you are done, click ***OK***.
8. Click the ***Contours*** button to make all contours visible.
9. In *Front*, *S Side* and *Top* views, fair the shape by modifying the control (or edit) points on the column you just inserted until you achieve the forward shape that you want.

Tip: To maintain what will be the flat of side, do not move the vertices of columns 0 or 1 transversely or vertically.

Auto*ship*'s fairing tools consist of the *Porcupine Plot*, the *Curvature Window*, *Gaussian Curvature* and *Mean Curvature*.

- To get a porcupine plot, when viewing *Edit* points, click the ***Curvature/Normals*** button on the Display toolbar. It is best to use this feature with the *Isolate* button on. The porcupine plot displays the curvature of the row or column as a series of spines radiating from the row or column. The length of the spine indicates the relative curvature along the row or column. Note the setting ***Settings – Invert Curvature Display*** which plots the spines as 1/curvature, thus showing the relative 'straightness' along the row or column.
- The Curvature window is turned on and off from the ***View – Curvature Window*** menu option. It plots the relative curvature along the row or column as a function of row/column length. Vertex positions are shown as vertical tick marks. This window can be re-sized and moved.
- ***Gaussian Curvature*** or ***Mean Curvature*** are found under the ***View*** menu. A rendered image will appear in the *Para* view window. Both options display the curvature as a color plot. Mean curvature shows the average of row curvature and column curvature and is best used for seeing the relative curvature across the surface. Gaussian Curvature is the product of row curvature and column curvature and is good for an indication of "buildability" – the more white (flat or single curvature) appearing, the easier the surface is to construct. Note the setting ***Settings – Preferences – Curvature Display – Color Compression Index*** which sets the scaling factor for curvature to visible spectrum for the color plot.
- Refer to the on-line help for details about using these fairing tools.

9. To see the results of your work, click the ***Para*** view button and rotate to the desired view. Then ***Select View - Render*** to render the surface. (See Figure 2-14.)

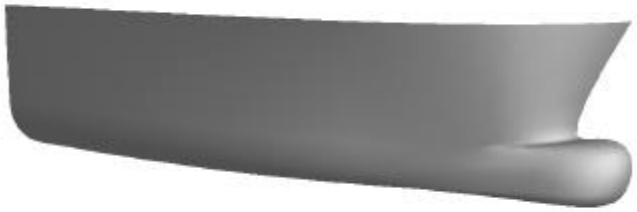

Figure 2-14: Rendered view of the hull

2.7 Controlling Flat-of-Side

A flat-of-side for the foreship section is most easily accomplished with the aftmost three columns. In our model, Column 0 is located at midship, column 1 will become a knuckle line and so define the forward end of the flat-of-side, and column 2, further forward, will be used to ensure the curved side blends smoothly into the flat side. If the rows of control points passing through these three columns are always in a straight line then the surface will be tangent and fair at the knuckle column.

➲ To control Flat-of-Side:

1. Click ***View - Render*** to return to normal view, if necessary.
2. Click the ***S Side*** button.
3. Select the *Foreship* surface and switch to Edit Mode by double-clicking on the *Foreship* surface.
4. Ensure that you are in *Column* mode by clicking the ***Rows/Columns*** button, if necessary.
5. Select *Column 1* and click the ***Toggle Corner*** button.

6. In *Top* or *Side* views, move the vertices longitudinally (but not transversely or vertically) until they are at the correct location to define the forward end, or tangency, of the flat-of-side. Do not worry about getting the locations exact – you can move them again later. (See figure 2-15).

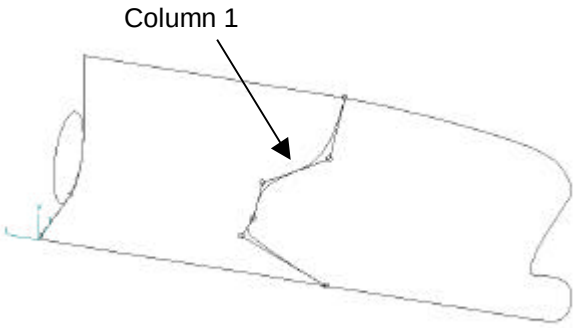

Figure 2-15: Editing Column 1 to define the Flat –of-Side.

Note: You can ensure that the vertices only move longitudinally by locking the points transversely and vertically or by using the arrow keys rather than the mouse.

Now we will add another column ahead of the knuckle column to ensure tangency across the knuckle column:

1. With *Column 1* selected, click the ***Add*** button.
2. In the *Insert* dialogue, select ***Proportional Interpolation***, specify a proportion of ***0*** and click ***OK***.
3. Select the top vertex in column 2 and click the ***Isolate*** button. (You may have to use the F3, F4 and F5, F6 keys to select this vertex as it resides as the same position as the top vertex in column 1 and clicking on the vertex always picks the one with the lower number.)

4. Drag the entire column forward by pointing at the top vertex in column 2 and, while holding the left mouse key down, move the cursor a little way directly forward (be sure not to move up or down, keep it at V = 3). Then hold down the ***Shift key*** and release the mouse button.

Note: To ensure flatness and tangency, you must keep column 2 control points at the same half-breadths and heights as like-numbered column 0 and column 1 control points.

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5. In *Side*, *Top* and *Front* view, arrange the points of column 2 and 3 to fair in the forward sections. Remember not to move column 2 vertices transversely or vertically. (Figure 2-16.)

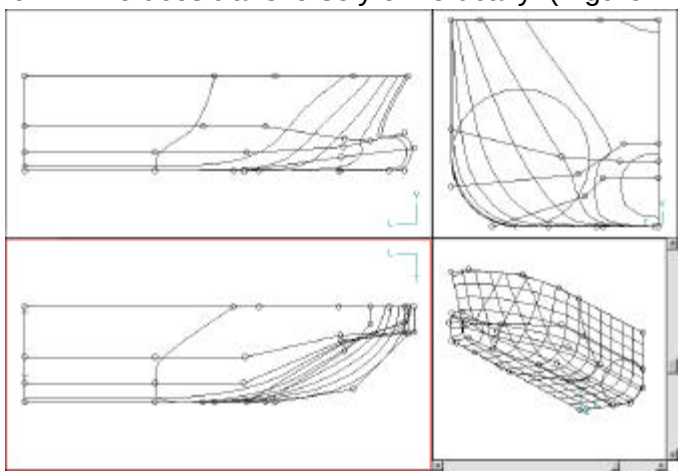


Figure 2-16: Fairing the forward sections

2.8 Generating the Aftbody Shape

You could generate the aft body as you did the foreship. However, we suggest another method: a lofted surface. A lofted surface is generated from a set of curves by linking like-numbered parameter values on the curves.

We will create a hull shape aft by lofting a surface between two curves: one at midship, the other at the aft profile. We will then add additional columns to fill and form the shape.

➲ Creating a Stern Curve:

1. Click the **Create Mode** button.
2. Click the **Create Curve** button.
3. In the Create Curve dialogue box, select the **Free** tab.
4. Type the following information in the dialogue box and click **OK**:

Field	Entry
Curve Name	Stern Curve
Parameters	Control Points = 8; Max Degree = 2
End 1	the centreline-baseline at the aft end of the vessel (L=10A, T=0, V=0)
End 2	the top of the transom at centreline (L=12A, T=0, V=U3)

5. Click the **S Side** button and go to **Edit mode**.
6. Zoom in on the curve and drag the control points on the curve to form the profile shape of the aft end of the vessel. Don't move the topmost or bottom-most points. (Zoom by clicking on the zoom button and then dragging a zoom box diagonally in the appropriate view.)
7. In Front view (or Back view) move vertices six and seven outboard to form the transom. (See Figure 2-17.)

Note: You may want to make the vertex at the bottom of the transom into a knuckle.

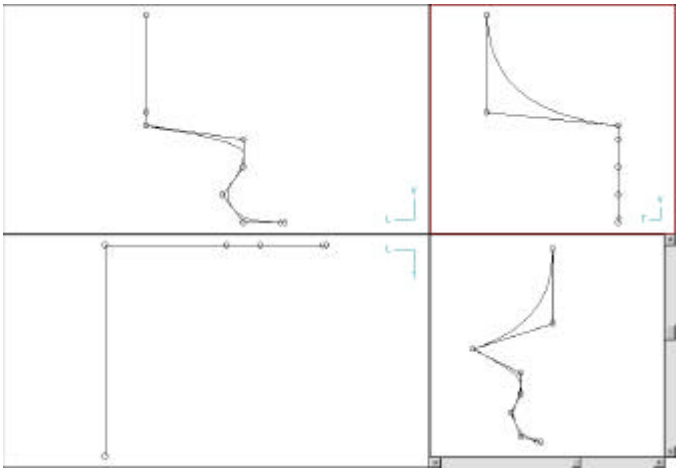


Figure 2-17: Creating the Stern Curve

➲ To create the Aft Hull Surface:

1. Click the ***Create Mode*** button.
2. Click the ***Create Surface*** button.
3. In the *Create Surface* dialogue, name the surface ***Aftship*** and select the ***Loft*** tab.
4. Under *Method*, select the **True Loft** radial button.
5. In the *Available Curves* list box, click ***Midship Curve*** and then the ***Add*** button, and then ***Stern Curve*** and ***Add*** again. Notice that *Row Max Degree* has changed from *0* to *1*.

   Note: This creates a surface starting at Midship Curve and ending at Stern Curve. As this surface has only two columns— one at the midship and one at the stern—the highest that Row Max Degree can be is 1 (one less than the number of columns).
6. Click ***OK***.

➲ To add additional columns to the Aft Hull Surface:

1. Click the ***Edit Mode*** button.
2. In the *S Side* view, select the column at the forward end of the *Aftship* surface and click the ***Add*** button. (Make sure you are in *Column* mode, not *Row* mode and do not have the upper edge vertex selected.)
3. In the *Insert* dialogue box, select ***Proportional Interpolation***, and specify a proportion of ***0***. Click ***OK***. This creates another column at the same position as the forward column.
4. Click the ***Isolate*** button to work on the new column (Col 1) without interference from the column that was copied. (Make sure Column 1 is selected.)
5. Select the *top vertex* and pull it straight aft to roughly where you want the flat-of-side to end and then hold down the ***Shift*** key and release the mouse button. (Be careful not to move the vertex vertically: watch the cursor coordinates and compare the cursor's vertical value to the vertex's vertical coordinate.)
6. Click the ***Corner*** button to turn column 1 into a knuckle. This creates the flat of side and tangency.
7. In both *S Side* view and *Top* view, modify the points of the column to form the aft end of the flat-of-side. Remember to move the vertices only longitudinally (not vertically or transversely).
8. With *Column 1* selected in *Edit* mode, click the ***Add*** button.
9. In the Insert dialogue, select ***Proportional Interpolation***, specify a proportion of ***0*** and click ***OK***.

Note: If necessary, add another column ahead of the aftmost column to complete fairing. Also, you can add more rows; however, be aware that they may influence the midship area.

10. Move the *Column 2* vertices directly aft to enable fairing the aft sections. This is easiest done using only the right-left arrow keys on the keyboard.

In Top view the surface will appear as straight lines (not curved). This is because the Row Max Degree is only 1. We will attend to this now.

11. Select ***Edit - Attributes***.
12. In the *Attributes* dialogue box, set *CDeg* for Aftship to ***2***. This causes the surface to be curved in top view, rather than straight lines between columns. Also, set *RRes* to ***60*** to allow better display of the surface shape. Click on ***OK***.
13. To help in fairing, go to ***Settings - Contours*** and add another group of stations from ***0*** to ***12a***.
14. Turn on the contours by clicking on the ***Contours*** button. (See Figure 2-18.)

The object in fairing this surface is to get the rows and columns to flow as smooth lines. Start with the Control Points to be sure the basic mesh is smooth, then progress to smoothing out the Edit Points. Fine tune the vertex positions by looking at the Mean Curvature.

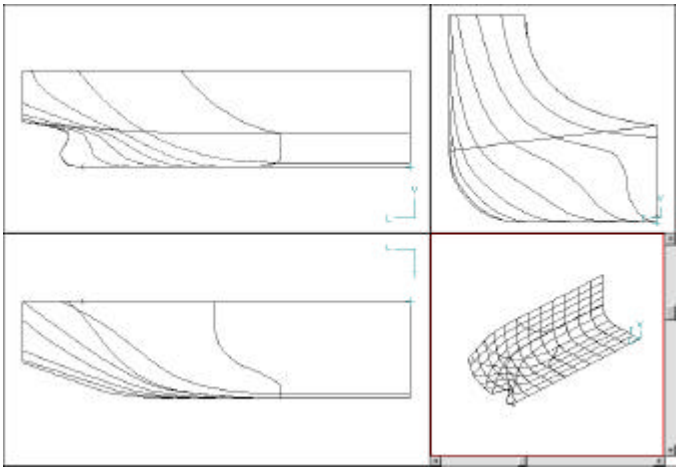


Figure 2-18: Aftship Surface

Because of the knuckle in the Stern Curve, the surface Aftship was created with a knuckle row. You may find it easier to fair the Aftship surface if you turn the knuckle row (Row 7) into a non-knuckle: select the row and click on the Toggle Corner button. However, if you turn the knuckle off, you will notice the aft edge of the surface no longer follows the Stern Curve. The easy solution is to fair the surface and then turn the row into a knucle line again. You should then make sure that the knuckle line is not causing unwanted chine by positioning the knuckle vertex on a straight line between it's neighbours. To do this, straighten the portion of the column which includes the two vertices on either side of the knuckle vertex: in ***column*** mode, click on the vertex below the knuckle vertex, then click on the ***Straighten*** button and then on the vertex above the knuckle vertex. You should repeat the straightening operation in one other view to make sure the knuckle vertex lies on a straight line between it's neighbours. (See figure 2-19.)

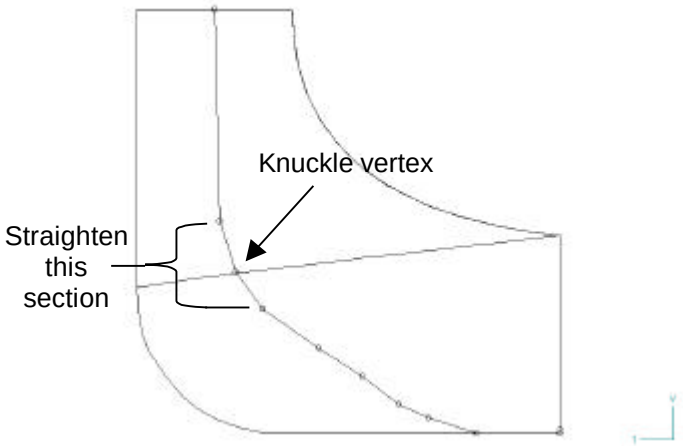


Figure 2-19: Straightening across the knuckle line

2.9 Adding a Bow Thruster

This exercise demonstrates how to project a curve onto a surface and shows one possible use for a projected curve. You will create two curves – a template curve and its projection onto the hull – and then rule a surface between them to create the thruster.

➲ To create the template curve:

1. Click the **Create Mode** button.
2. Click the **Create Curve** button.
3. Select the **Arc** tab.
4. In the *Create Curve* dialogue, type the following and click ***OK***.

Field	Entry
Curve Name	Circle
Plane	Side
Centre	On the centreline, at an appropriate position for the bowthruster (we use: 10.5F, 0T and 0.6U)
Angles	Start: 0°; End: 360°
Semi Axes	Circular: Checked Radius: An appropriate radius, we used: 0.25 m

5. Change to ***S Side*** view and zoom in on the area around the newly created curve. (See Figure 2-20.)

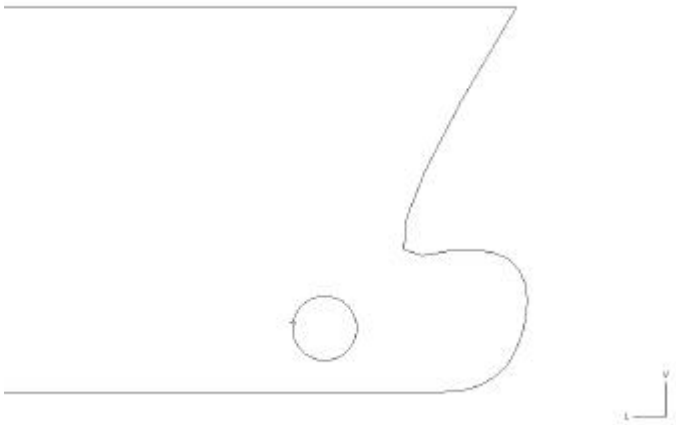


Figure 2-20: Circle curve

If you would like to re-position the curve, do the following

1. ensure that the *Circle* curve is selected and click the ***Set Base Point*** button.
2. Click just aft of the *Circle* curve. This will move the base point from 0,0,0 to the location just clicked on.
3. Click the ***Move*** button.

4. When you move the cursor into the view, you will see a "rubber band" emanating from the curve's "base point". Position the bilge circle at the new location and click the left mouse button. You will note that the base point has moved also.

Note: We moved the base point first because the rubber band originates from the base point, and as the base point did not appear because of the current zoom, then the rubber band would have started from off the screen. Note that the rubber band indicates the relative displacement of the curve – the curve will move **by** this amount from it's current location, not **to** the position where the cursor is.

➲ To project the Circle curve onto the hull surface:

Projecting the Circle curve onto the hull surface produces a new curve that is embedded on the hull. An embedded curve will adapt itself to any changes in the host surface.

1. In ***Create Mode***, click the ***Create Curve*** button.
2. Select the ***Projected*** tab.
3. In the *Create Curve* dialogue, type the following and click ***OK***:

Field	**Entry**
Curve Name	Thruster Curve
Projection	Side
Type	2
Source Curve	Circle
Host	Foreship
Acceptable Tolerance	0.001

Note: A type 1 projected curve, can be altered only by editing the source curve. If you edit the source curve, you may have to click the Update button to update the projected curve. A type 2 projected curve is a separate, editable curve that is embedded on the surface. A type 2 projection can be updated (to adapt to changes of the source curve or host surface) by selecting the curve and clicking on Edit – Regenerate Object.

4. To see the result, click the ***Para*** button and zoom in on the forward end of the hull. (See Figure 2-21.)

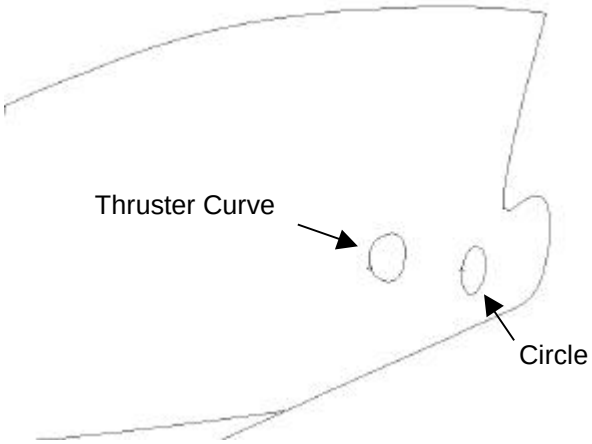


Figure 2-21: Circle projected onto Foreship Surface

➲ Using the Thruster Curve to cut the hole in the hull surface:

1. Select the ***Foreship*** surface.
2. Click the **Trim** button. In the *Trim Surface Foreship* dialogue box, pick ***Thruster Curve*** and click ***OK***. The program will show various trim options and prompt you with “Keep this domain”. Answer No to the option not wanted, and Yes to the option wanted.
4. Select ***View - Render*** to see the trim. (See Figure 2-22.)

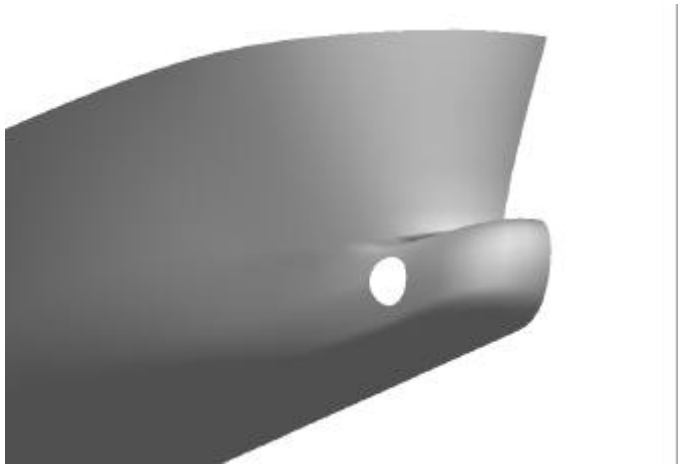


Figure 2-22: Rendered trimmed Foreship surface

➲ To create the Tunnel as a Ruled Surface:

1. Turn off Render by selecting ***View - Render*** again.
2. In ***Create*** mode button, click the ***Create Surface*** button.
3. Select the ***Rule*** tab.
4. In the *Create Surface* dialogue, type the following and click **OK**:

Field	Entry
Surface Name	Thruster
Curves	Circle; Thruster Curve

5. Select ***View - Render*** to see the result.

Note: If the surface Thruster appears as a tie, or hourglass, then you must change the direction of one of the curves and regenerate the surface. The direction of a curve is shown by a small arrow near the beginning of the curve when in Select mode. The arrows on both curves should point the

same direction. To change the direction of a curve, select the curve, then in ***Edit*** mode click the ***Reverse Ends*** button. Then update the Thruster surface by selecting it and clicking on ***Edit – Regenerate Object***. Note that if you reverse the embedded curve used to trim the surface, you will have to re-do the surface trim.

6. In Select mode, select both the new Thruster surface and the Foreship surface. (Select one, and then while holding down the Shift key, select the other – you must click on the outline of the surface.) Note that the object box at the top of the screen displays “Multiple Select”.
7. Select ***Arrange - Group***.
8. In the *New Group* dialogue, change the name to ***Hull Fwd*** and click ***OK***. Note the *Make Comps Invisible* checkbox. If checked, the visibility of the two surfaces will be turned off in the *Attributes* dialogue.
9. If it is not selected, select the new group **Hull Fwd** and select **View-Render** to see the result. (See Figure 2-23.)

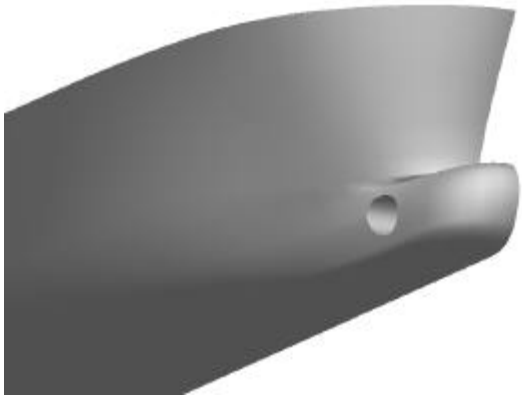


Figure 2-23: Rendered Hull Fwd group

2.10 Adding a forward deck with Camber and Sheer

This exercise introduces the camber curve type, the sweep surface type and the method of trimming one surface to another. Auto*ship* can create parabolic camber curves, or you can design your own camber curve. In this exercise, we will use a parabolic curve. In it's simplest form, a sweep surface advances one curve (the pattern curve) along another curve (the guide curve), thus "sweeping out" a surface. We will sweep a camber curve along a sheer at centreline curve to create the forward deck, obtain the intersection with the hull and then trim the deck at that intersection.

1. Turn off the render by clicking on ***View – Render***, switch to ***S Side*** view and click the ***Zoom Extents*** button.

➲ To create the parabolic Camber Curve (the pattern curve):

1. Click the ***Create Mode*** button.
2. Click the ***Create Curve*** button.
3. Click the ***Camber*** tab.
4. In the *Create Curve* dialogue, type the following and click **OK**:

Field	**Entry**
Curve Name	Deck Camber
Projection	Front
Dimensions	Half : radio button selected Breadth A value slightly larger than the vessel's half-breadth, we used 3.1m Height: For a 1:50 camber, use .06

➲ To create the sheer Curve (the guide curve):

1. Click the ***Create Mode*** button.

2. Click the ***Create Curve*** button.
3. Click the ***Free*** tab.
4. In the *Create Curve* dialogue, type the following and click **OK**:

Field	Entry
Curve Name	Fwd Sheer at Centreline
Parameters	Control Points = 4; Max Degree = 2
End 1	L = 0, T = 0, V = U2.9
End 2	L = F12, T = 0, V =U2.9

5. Create some sheer in the curve by repositioning the forwardmost vertex to V = U3.0: In ***S Side*** view and ***Edit*** mode, select vertex 3 and use the up arrow key on the keyboard, hit ***Enter*** when done.

➲ To create the deck surface:

1. Click the ***Create Mode*** button.
2. Click the ***Create Surface*** button.
3. Click the ***Sweep*** button.
4. In the *Create Surface* dialogue, type the following and click ***OK***:

Field	Entry
Surface Name	Foredeck
Pattern Curves	In the upper box, pick Deck Camber
Guide Curves	In the upper box, pick Fwd Sheer at Centreline
Options	Parallel: Checked

Note: Parallel will maintain the original upright orientation of the camber curve as it sweeps out the surface, just as deck beams would be placed. If Parallel was not checked, the relationship between the two curves would be maintained, thus, in order to keep perpendicular to the guide curve as the guide curve bends, the pattern curve would rotate (in side view) as it was swept along the guide curve.

5. View the surface in ***4-view*** to see the camber and sheer. It may help to turn on the ***Mesh***. (See figure 2-24.)

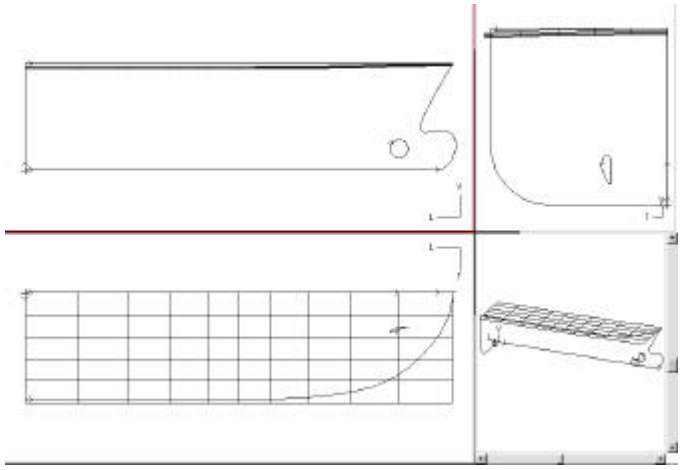


Figure 2-24: The Foredeck surface

➲ To trim the deck surface:

1. Select ***Foredeck*** and ***Foreship***.

Note: If, when you created the group *HullFwd*, the *Make Comps Invisible* checkbox was checked, the visibility of the *Foreship* surface will be turned off so you will not be able to select *Foreship* from the main screen. To turn on *Foreship*'s visibility, go to the *Attributes* dialogue and check the *Vis*. column for the surface *Foreship*.

2. Click the ***Surf-Surf Intersect*** button. In the *Surf-Surf Intersect* dialogue box, make sure the two surfaces shown are Fored*eck* and *Foreship*, set the tolerance to ***0.005m*** and click ***OK***.

This will create two curves which describe the intersection of the Foreship and Foredeck; one curve, Foreship Int1, is embedded on the Foreship surface, the other curve, Foredeck Int1, is embedded on the Foredeck surface. To avoid future confusion, we suggest you rename the curves using the following convention:
SurfA Int1 → SurfB-SurfA, and SurfB Int1 → SurfA-SurfB, where then first name indicates the surface the curve is embedded, the "-" is reserved to indicate an intersection curve, and the second-named surface is the surface intersected.

3. Rename curve Foreship Int1 to Foredeck-Foreship: click on ***Edit – Attributes***, click on the name *Foreship Int1* and then click on the ***Properties*** button. In the Curve Properties dialogue, under Name, type ***Foredeck-Foreship*** and click on ***OK***.
4. Similarly, rename *Foredeck Int1* to *Foreship-Foredeck*.
5. Select the surface *Foredeck*.
6. Click the ***Trim Surface*** button.
7. In the *Trim Surface Foredeck* dialogue, select the curve ***Foredeck-Foreship*** and click on ***OK***. Again, you will be shown various trim options, answer ***Yes*** at the appropriate one and ***No*** to the others.
8. Select the surface *Foreship*.
9. Click the ***Trim Surface*** button.
10. In the *Trim Foreship Surface* dialogue, select both of the curves ***Foreship-Foredeck*** and ***Foreship-Thruster Curve*** and click ***OK***. Answer ***Yes*** at the appropriate trim option shown and ***No*** to the others. (See figure 2-25.)

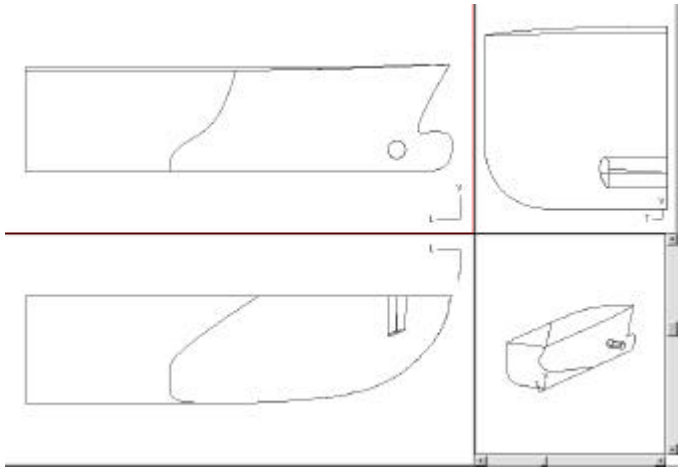


Figure 2-25: Trimmed Foreship and Foredeck

2.11 Adding an Aft Deck with Camber and Sheer

Use the same procedure to create the surface Aftdeck. In our example, we kept the Aft Sheer at Centreline curve horizontal.

2.12 Creating a group of the entire vessel

➲ To create the group:

1. Select one of the surfaces you have created.
2. Click on the ***Arrange – Group*** menu option.
3. In the *New Group* dialogue, type the name ***Main*** and click on ***OK***.
4. With the group *Main* selected, click on the button now marked ***Group*** in the upper right of the screen to open the **Group Editor** dialogue.
5. In the *Group Editor* dialogue, under *Objects - Select*, click on the ***Surface*** radio button.

6. In the pick box directly below that, pick one of the surfaces (other than the one already indicated in the object list below that).
7. Click on the ***Add*** button. The selected surface will be added to the object list.
8. Add the other surfaces the same way.
9. Click on the ***Stbd + Port*** check box to have all the selected surfaces mirrored to the other side of the vessel.
10. Exit the *Group Editor* dialogue.
11. Switch to ***Para*** view and render the group. (See figure 2-26.)

Figure 2-26: Rendered group Main

2.13 Summary

This completes the second tutorial. The first and second tutorials introduced the program's tools for constructing a model, and described how to use the building blocks of the system– objects – by creating and using simple curves and surfaces. Tutorial three completes this introduction to hull design with Autoship. In tutorial three, hull design is accomplished using extrusion, mirroring surfaces, integrating panels, and by using ruled and swept surfaces.

Tutorial 3 - Asymmetric Hull Catamaran

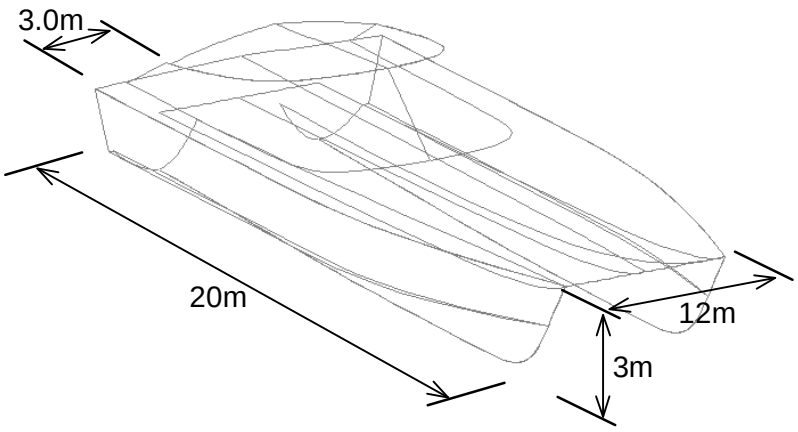


Figure 3-1: Asymmetric Catamaran

3.1 Introduction – Asymmetric Hull Catamaran

In Tutorial 3 we show how to:

- Design the outboard half of the hull by extrusion
- Generate the inboard half of the hull by modifying a mirrored copy of the outboard half
- Design the tunnel as an integration of several panels
- Fit a deck between a centerline and a deck edge curve
- Construct a pilot house using a ruled and a swept surface

We have constructed this model as **AS7TUT3.PR3**. You may want to review it before continuing.

3.2 Starting Out

➲ To begin:

1. Select ***File - New***.
2. Select ***Settings - Units***.
3. In the *Units* dialogue, select ***Meters*** and ***Tonnes*** and click ***OK***.
4. Select ***Settings - Preferences***.
5. In the *Preferences* dialogue, click the ***Coordinate System*** button.
6. In the *Coordinate System* dialogue, click the ***Naval / Aircraft (US)*** radio button and click ***OK***.
7. Click ***OK*** in the *Preferences* dialogue.
8. Click ***File – Save As***.
9. In the *Save Project* dialogue, save the project as ***TUT3*** in the ***MYTUTS*** directory.
10. In the *Project Info* dialogue, type a description such as ***20m Catamaran***, and click ***OK***.

3.3 Designing the Outboard Hull

The outboard half of the hull will be constructed by extruding a simple cross-section curve.

➲ To create the cross-section curve:

1. In *Create Mode*, click the **Create Curve** button.
2. In the *Create Curve* dialogue, type the following information, and click **OK**:

Field	Entry
Curve Name	Section
Tab	Free
Parameters	Number = 4, Max. Degree = 2
End 1	L = 0; T = 1.6; V = 3
End 2	L = 0; T = 0; V = 0

3. Switch from *Snap* to *Grid* mode (value of 0.1) by clicking on the ***Snap*** button at the top left of the screen.

Note: By using Grid instead of Snap, you can easily re-position vertices to rounded, or even value, co-ordinates. Snap causes the vertex to jump relative to its initial position. Grid jumps to an absolute position as defined by a grid with an interval equal to the Snap setting.

4. In *Front* view, add another vertex and use the ***Toggle Corner*** button to form a chine flat and then reposition the vertices to form the hull section at midship, as depicted in Figure 3-2.

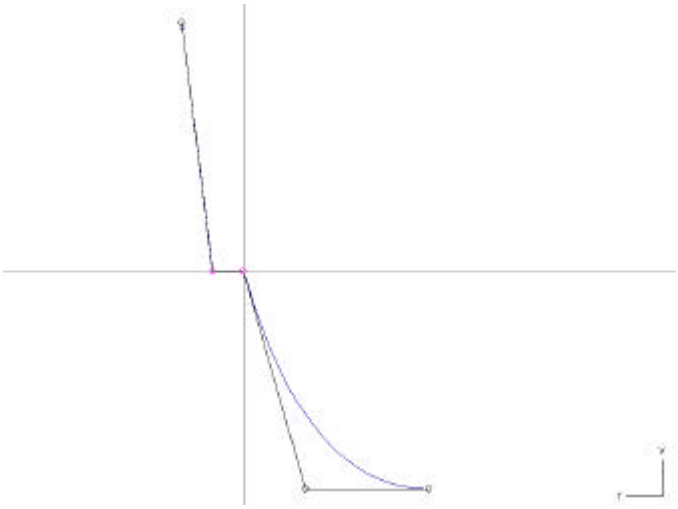


Figure 3-2: Section Curve

3.4 (a) - Creating the Outboard Hull Surface

➲ To create the outboard hull surface:

1. In *Create Mode*, select ***Create Surface***.
2. In the *Create Surface* dialogue, type the following information, and click ***OK***:

Field	Entry
Name	Hullout
Tab	Extrude
Curve	Section
Vector	L: 20F
Column Mesh	Number: 4; MaxDegree: 3

3. Switch to ***Edit*** mode and ***Column*** mode, select the forward column, and then click the ***Isolate*** button.
4. In *Front* view, move the topmost vertex to the centreline: select vertex 0 click in the ***T*** cooridinate box at the top left of the screen, type ***0*** and hit ***Enter***.
5. With the topmost vertex selected, click the ***Straighten*** button and then click on the lowest vertex to straighten all the vertices to the centerline.
6. In *Side* view, move the points longitudinally to give the desired bow profile.
7. In *Side*, *Top* and *Front* view, form the hull to achieve the shape you want for the outboard half of the hull. Try to set the top edge of the hull around midship at T = 1.6. We will be trimming off the top edge of the hull, so do not bother to shape it in profile. You should also position the forward-most vertices of both chines at the same location. (See Figure 3-3.)

Note: We added another column forward to give us better control over the bow shape.

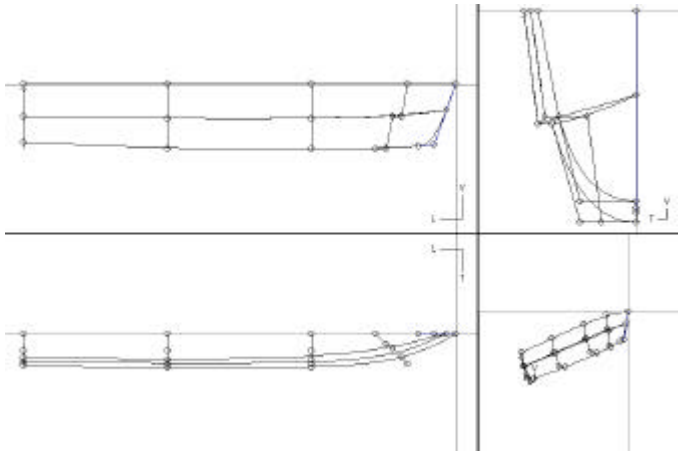


Figure 3-3: Outboard hull surface

3.4.(b) - Ensuring fairness at the forward-lower corner of the surface.

In Tutorial 2, special emphasis was placed on positioning the vertices at the forward end of the centreline curve to ensure that the stem below the chine blended smoothly into the stem above the chine. A similar situation exists on the surface Hullout at the corner where the forward edge meets the lower edge. Currently, this corner is at the aft end of the curved-forefoot. In order to make the forward edge transition smoothly into the bottom edge, we must position the corner vertex and it's closest neighbouring vertices on the forward edge and bottom edge on a straight line. An easy method would be to position all three vertices at V = 0. However, we will outline a more general method.

1. With ***Hullout*** selected, in ***Edit*** mode, and ***Column*** mode, with ***Control Points*** selected, click on the vertex at the bottom column immediately aft of the stem column (Row 4,

Col 3 in our model) and write down the L and V coordinates. (See figure 3-4.)

2. Click on the vertex at the bottom of the stem column (Row 4, Col 4 in our model) and write down the L coordinate. (See figure 3-4.)

3. Click on the next vertex above the bottom of the stem (Row 3, Col 4 in our model) and write down the L and V coordinates. (See figure 3-4.)

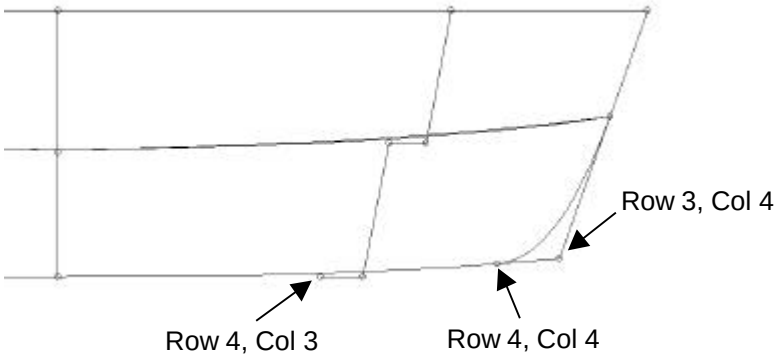


Figure 3-4: Control points at stem

4. Create a new curve using the ***Free*** tab. Set *Control Pts* to ***3***. For *End 1*, type in the first set of coordinates you wrote down. For *End 2*, type in the last set of coordinate you wrote down.

5. With the new curve selected, switch to ***Edit*** mode, and ***Edit Points***, if necessary, and select the middle vertex.

6. Set the middle vertex to the L coordinate you wrote down for the corner of Hullout: click in the *L* coordinate box at the upper left of the screen to highlight the L value, type the new coordinate and hit ***Enter***.

7. Straighten the entire curve: select one of the end vertices, click on the ***Straighten*** button and click on the vertex at the other end of the curve.

8. Copy the *V* coordinate of the middle vertex of the curve to the corner vertex of the Hullout surface: select the middle vertex, click in the *V* coordinate box, hold down the *Ctrl* key and press the *C* key (this copies the coordinate to the "clipboard). Switch to ***Select*** mode, select the ***Hullout*** surface, switch to ***Edit*** mode and select the vertex at the corner where the forward edge meets the bottom edge (Row 4, Col 4). Click in the ***V*** coordinate box, hold down the ***Ctrl*** key and press the ***V*** key (this pastes the value from the "clipboard) and hit ***Enter***.

9. Go to the ***Attributes*** dialogue and turn off the visibility of the curve used for this straighten process.

This effectively straightens the forward-lower corner of the surface Hullout. Note that if you move either of the three vertices used for the straightening operation, then you will have to re-do the straightening operation.

3.5 Generating the Inboard Hull Half

To have both sides of the hull match up properly at the centerline, we will copy the outboard half of the hull and then mirror it to create the inboard half, and then rework the inboard shape.

➲ To copy and mirror the outboard half of the hull:

1. In ***Select Mode***, select ***Hullout***.

2. Copy the outboard hull: hold down the ***Shift*** key and click the ***Clone (Shift – Copy)*** button.

3. In the *Copy* dialogue, rename *Copy of Hullout* to ***Hullin*** and, click ***OK***.

4. With *Hullin* selected, in *Front* view, click the ***Flip Horizontal*** button.

5. If both chines end at the same location at the stem, then in ***Edit*** mode and ***Row*** mode, select one of the knuckle lines (rows) and click the ***Delete Row or Column*** button to remove it. If you did not position the forward-most vertices of both chines at the same location, then you cannot delete either chine as it will change the profile of the stem.

6. To decrease the breadth of Hullin, use the scale function: In *Select Mode*, right-click the ***Scale*** button. Set the transverse scale to *0.875* and click ***OK***.

   Tip: Before scaling *Hullin*, make sure that it's Base Point is located at T = 0: with *Hullin* selected, in *Select* mode, right-click on the *Set Base Point* button. Set the *T* value to *0*, if it is not already.

7. Edit the shape as you see fit. Take care to not move any of the vertices at the centreline, otherwise the two halves of the hull will not match. (See Figure 3-5.)

In order to have the same shape for the bow profile on Hullin as on Hullout, the distribution of vertices must be the same. This means the second vertex from the top must be a corner vertex and must not move from it's original position. And since this vertex must be a corner vertex, then the entire row must be a knuckle row. If you do not want to have a chine at that row, then the columns must be straigtened across the corner vertices, as follows:

8. In ***Edit*** mode, switch to ***Column*** mode, click on the ***Isolate*** button and switch to ***Edit Points***, if necessary.

9. Using the aftmost column as an example, in ***Front*** view, click on the vertex immediately above the chine vertex (the chine vertex is the green one), then click on the ***Straighten*** button, then click on the vertex immediately below the chine vertex. Repeat this straightening procedure in ***S Side*** view.

10. Repeat this procedure for the other columns.

Straighening across the chine will effectively cause the chine to disappear. Note that if you edit the shape of the surface after straightening, you will have to re-do the straighten operation.

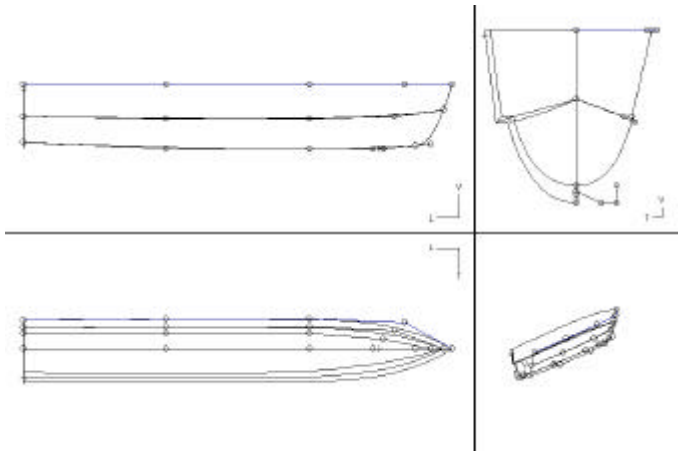


Figure 3-5: Mirrored inboard half of the hull

3.6 Designing a Tunnel and Fillet Panel

First, we will move the hull to its correct position. Next, we will create a tunnel by extruding a surface from a centerline curve. Then, we will construct a fillet panel between the tunnel and the inboard hull side using the Blend surface type.

3.6.(a) - Moving the Hull to its Correct Position

At this point, the hull centerline is at T = 0. It is to be moved to 4.0 starboard.

1. In ***Select*** mode, select both the surfaces *Hullin* and *Hullout*.
2. Right-click on the ***Move*** button.

3. In the *Move* dialogue, set the *T* value to ***4*** and click on ***OK***.

Now Hullin and Hullout are at centred at 4m to starboard while the boat's centerline is at T=0.

3.6.(b) - Creating the Tunnel Surface

1. Create a new curve:

Field	Entry
Curve Name	Tunnel Shape
Tab	Free
End 1	L=F20; T=0; V=U3.0
End 2	L=0; T=0; V=U1.9

2. Switch to ***S Side*** view and reshape the curve as indicated in figure 3-6.

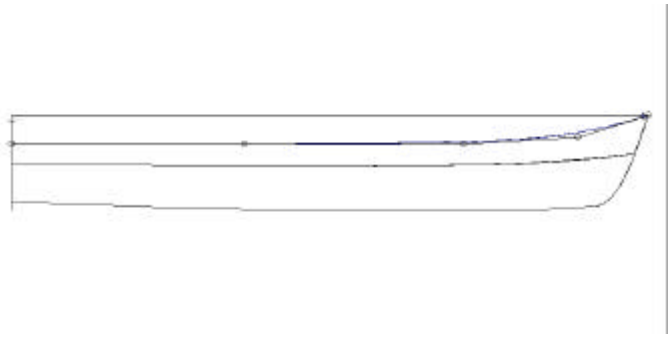


Figure 3-6: Tunnel Shape curve

➲ To create the surface Tunnel:

1. Create a new surface: (See Figure 3-7.)

Field	Entry
Surface Name	Tunnel
Tab	Extrude
Curve	Tunnel Shape
Vector	L = 0; T = S4.0; V = 0

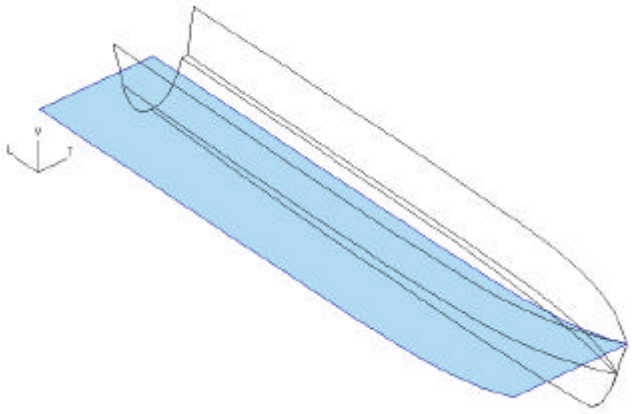


Figure 3-7:The Tunnel Surface,

To create a smooth corner between the Tunnel and Hullin, we will create a fillet panel using the Blend surface type. For this design, this fillet panel will have a 0.25m radius over most of it's length but will taper to 0 at the forward end. Two surfaces to be "blended" must have congruent edges, such as at a chine or at a trimmed intersection. Therefore, to blend the Hullin and the Tunnel, we must first obtain the intersection of the Tunnel and Hullin surface and then trim the surfaces to the intersection.

➲ To obtain the intersection:

1. Switch to ***4 View***.
2. Select both the ***Tunnel*** and ***Hullin*** surfaces.
3. Click on the ***Surf-Surf Intersect*** button.
4. In the *Surf-Surf* dialogue, verify the surface names and click on ***OK***.
5. In the *Attributes* dialogue, rename the curve *Hullin Int1* to ***Tunnel-Hullin*** and *Tunnel Int1* to ***Hullin-Tunnel***.

➲ To trim the surfaces:

1. Select the ***Hullin*** surface
2. Click on the ***Trim Surface*** button.
3. In the *Trim Surface Hullin* dialogue, pick the curve ***Hullin-Tunnel*** and click on ***OK***.
4. Answer ***Yes*** to the appropriate *Keep this Domain* prompt and ***No*** to the rest.
5. Repeat steps 1 to 4 for the surface *Tunnel*.

➲ To create the fillet panel

1. Create a surface:

Field	Entry
Surface Name	Fillet
Tab	Blend
Surfaces	Tunnel Hullin
Radii Variation	Cubic

Radii	1 = 0.25 2 = 0.25 3 = 0.25 4 = 0
Type	Fillet
Tolerance	0.001

2. The intersection of the two surfaces will be highlighted and you will be promted with *Blend here?* Answer ***Yes*** if the indicated path is correct. (See Figure 3-8.)

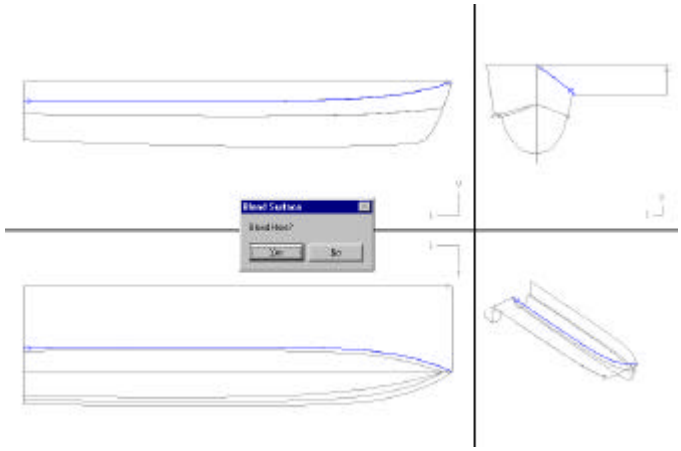


Figure 3-8: Fillet Panel

A Blend surface operation creates one surface and two curves. The surface fits between the two curves, but is not joined to either of them. One of the curves is embedded on one surface and the other curve is embedded on the other surface. These curves can be used to trim the surfaces being filleted to the junction with the fillet panel. We will now re-trim the Tunnel surface at it's junction with the Fillet and then re-trim the Hullin surface at it's junction with the Fillet.

➲ To re-trim the surfaces:

1. Select the ***Hullin*** surface.
2. Click on the ***Trim*** button.
3. In the *Trim Surface Hullin* dialogue, de-select the curve ***Hullin-Tunnel***, select the curve ***Fillet 0*** and click ***OK***. (If you had selected the two surfaces in the reverse order when generating the Blend surface, then the curve embedded on Hullin would be named Fillet 1.)
4. Answer ***Yes*** to the appropriate *Keep this Domain* prompt and ***No*** to the rest.
5. Repeat this operation for the Tunnel surface.

   Note: If the surface trim does not seem to take effect, the likely cause is that the trimming curve is not “closed”. A closed curve means that either both ends of the curve contact an edge of the surface or the curve begins and ends the same location. If you encounter a case where the surface trim does not work, first check the location of each end of the trimming curve and, if necessary make adjustments to close the curve.

6. Save the project: click on ***File*** – ***Save***.

3.7 Adding an Upper Deck

The upper deck appears to be a relatively simple shape, however it has some complex restrictions. This surface will have three columns: an inboard one along the centreline, an outboard one along the surface Hullout, and a middle one exactly half way in between. To ensure tangency across the centreline, the vertices on the middle column must be positioned at the same height as the centreline column. To ensure proper control of the shape, the rows must be perfectly transverse. To ensure easy control of the shape,

we must limit the number of rows to a manageable number, say five or six. We will employ a few tricks to accomplish this.

➲ To create the Deck Edge curve:

1. Create a new curve:

Field	Entry
Curve Name	Deck edge
Tab	Embedded
Host	Surface radio button selected Hullout
End 1 End 2	R = 0; c = 0 R = 0; c = 1

This will create a curve which runs diagonally across the surface.

2. In *Edit* mode, re-shape the curve to describe the deck edge. Make sure both ends of the curve contact an edge of the Hullout surface. (See Figure 3-9.)

Figure 3-9: Embedded curve on Hullout

➲ To create a temporary curve:

1. Create a new curve by matching the Deck Edge:

Field	Entry
Curve Name	Temp
Tab	Match
Type	Free
Source Curve	Deck Edge

Note: Copying an embedded curve produces another embedded curve. A Match curve does not have the restriction of being only embedded.

2. In *S Side* view, shape this curve to represent the profile of the deck centreline. Do not move the vertices longitudinally or transversely. You might find it advantageous to use the *Porcupine* plot to see the curvature: click on the ***Curvature*** button. Since this curve is so flat, try inverting the curvature by clicking on ***Setings – Invert Curvature Display***. This displays the relative straightness, rather than the relative curvature. (See Figure 3-10.)

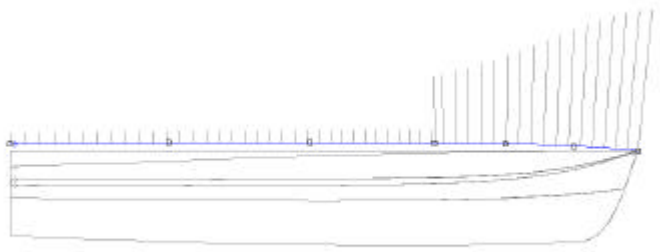


Figure 3-10: The Temp curve

➲ To create a Centreline curve:

1. Create a new curve by copying the *Temp* curve: with *Temp* selected, hold down the ***Shift*** key and click on the ***Clone (Shift – Copy)*** button. In the *Copy* dialogue, name the new curve ***Centreline***.

2. Move the entire curve to the centreline: with *Centreline* selected, click on the ***Scale*** button and for the *T* value type ***0*** and then hit ***Enter***. You may wish to view the result by switching to ***Top*** view or ***4 View***.

➲ To create the Deck:

1. Create a new surface:

Field	Entry
Name	Deck
Tab	Rule
Curves	Centerline Temp
Column Mesh	Number: 3 Max Degree: 2

2. Attach the outboard edge to the curve *Deck Edge*: switch to ***Edit*** mode, select the outboard column (Col 2), click on the ***Attach*** button and in the *Attach Surface* dialogue, under *Action* click on ***Attach to Curve*** and under *Attach To* select **Deck Edge** and then click on OK. (See figure 3-11.)

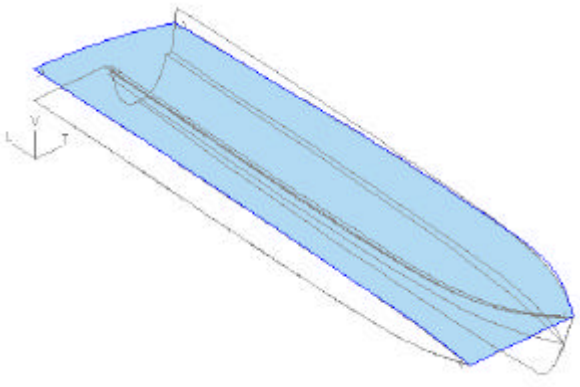


Figure 3-11: Deck Surface

➲ To clean up:

1. Detach the outboard edge of the deck: with Col 2 selected, click on the ***Detach*** button.

2. Delete the curve Temp: select the curve ***Temp***, and click on the ***Delete*** button.

3. Trim the Hullout surface off at the Deck Edge curve: select the surface ***Hullout***, click on the ***Trim Surface*** button, in the *Trim Suface Hullout* dialogue select the curve ***Deck Edge*** and click on ***OK***. Answer ***Yes*** to the appropriate *Keep This Domain* prompt and ***No*** to the others.

Notes:

The shortcut to positioning the middle column exactly half-way between the edges is to hve the program insert it. The trick to setting the vertices on the middle column to the same height as the centreline column is to start with a surface in which the rows are transverse and horizontal. The best way to set up a surface in which the rows are transverse is to create a ruled surface between two curves that have the same number of vertices placed in identical

longitudinal positions. The best way to obtain two curves with the same number of vertices placed in identical longitudinal positions is to copy one to create the other.

Because the Deck Edge curve is attached, a ruled surface generated from it would have hundreds of rows. A ruled surface generated from two free curves usually has only five or six rows.

Often, attaching the edge of a surface to a curve disrupts the arrangement of the vertices on the edge being attached. Because the Deck surface was generated from a copy of the Deck Edge curve, attach the edge of Deck to Deck Edge did not distrupt the arrangement of the edge vertices.

3.8 Adding a Pilot House

We will create the house front and sides by ruling a surface between a curve embedded on the deck and a free curve some distance above the deck. The top will be created as a swept surface.

➲ To create the Housefront surface:

1. In *Top* view, create a new curve

Field	Entry
Curve Name	Housebot
Tab	Embedded
Host	Surface radio button selected Deck
Parameters	Number: 4; Max. Degree: 2
End 1	r = 0; c = 0
End 2	r = 1; c = 1

2. Modify this curve to represent the lower edge of your pilot house. (See figure 3-12.) To ensure tangency across the centreline, keep vertex 2 at the same longitudinal position as vertex 3 (the centreline vertex.)
3. Create another curve, *Housetop Brow*, as a free curve with 4 *Control Pts* and *MaxDegree* 2.

Tip: for easy editing, set End 2 to a different L value than End 1.

4. In *Top* and *S Side* view, form *Housetop Brow* as the upper edge of the house side. (See Figure 3-12.)

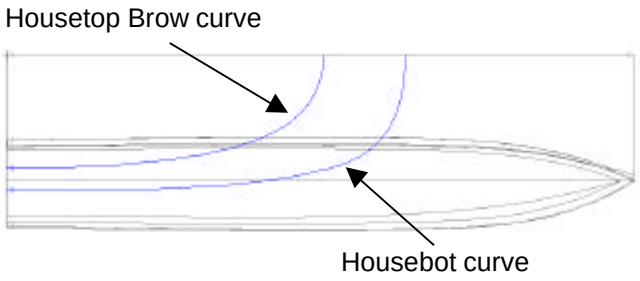


Figure 3-12: Housebot and Housetop Brow curves

5. Create the surface Housefront: (See Figure 3-16.)

Field	Entry
Surface Name	Housefront
Tab	Rule
Curves	Housetop Brow Housebot

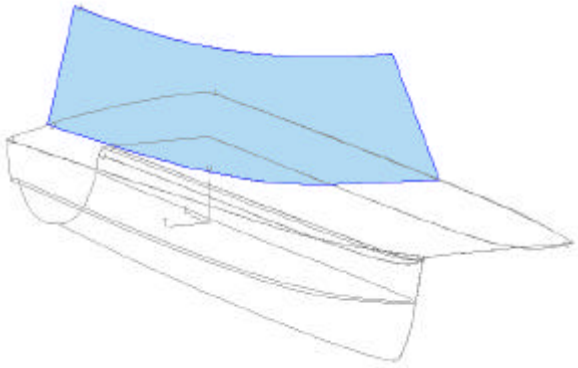


Figure 3-16: Housefront surface

➲ To create the Housetop surface:

First, we will copy the Housetop Brow curve and reshape it to form the profile of the house top at the centreline. Next, we will create a new curve which describes the shape of the house top at the aft end. Finally, we will sweep this second curve between the Housebrow curve and the curve at centreline to produce the house top surface.

1. Copy the Housetop Brow curve: select ***Housetop Brow***, hold down the ***Shift*** key and click on the ***Clone (Shift – Copy)*** button. In the *Copy* dialogue, name the new curve ***Housetop Centre***.
2. Set *Housetop Centre* to the centreline: right-click on the ***Scale*** button. In the *Scale* dialogue, set ***T*** to ***0*** and click on ***OK***.
3. In ***S Side*** view, edit the heights of the vertices to form the profile at centreline. Do not move vertex 3, and try not to move the vertices longitudinally.
4. Switch to ***Front*** view.
5. Click on the ***Create*** mode button and then the ***Draw Curve*** button.
6. Click once at the aft end of the curve *Housetop Centre*.
7. Click at a position just above and inboard of the aft end of the curve *Housetop Brow*. Try to click at the same height as the where you clicked on *Housetop Centre*.
8. Click on the aft end of the curve *Housetop Brow*.
9. Right-click anywhere in the view window.
10. In the *Curve Parameters* dialogue, name the curve ***Housetop Shape***, set *Maximum Degree* to ***2***, click on the ***Control Points*** radio button and click on ***OK***.

11. If necessary, switch to ***Edit*** mode and modify the shape. (See figure 3-14.)

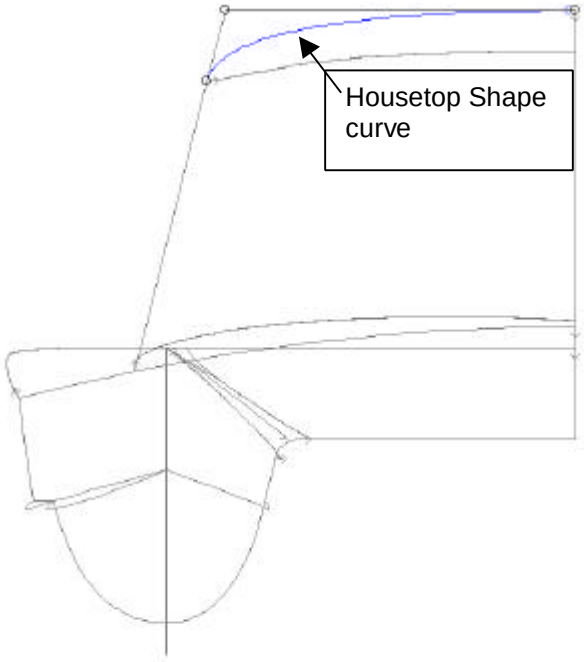


Figure 3-14: Housetop Shape Curve

12. Create a new surface: (See Figure 3-15.)

Field	Entry
Surface Name	Housetop
Tab	Sweep
Pattern Curves	Housetop Shape
Guide Curves	Housetop Centre Housetop Brow
Options	Scaled: checked

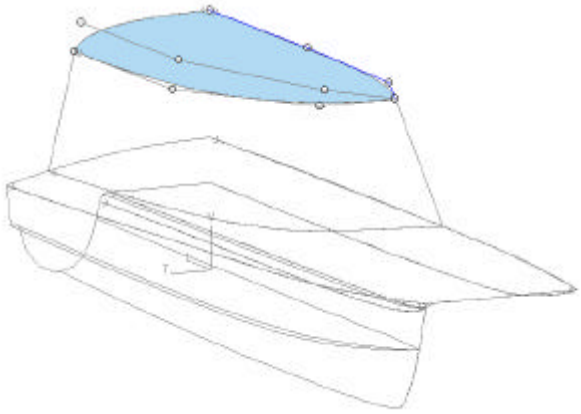


Figure 3-15: Housetop surface

3.9 Starboard and Port Side

We will complete the catamaran by creating a group, Main, which contains all the surfaces we have created, and then set the group to mirror the assembly.

1. Create a group, ***Main***, of the seven surfaces *Hullout*, *Hullin*, *Tunnel*, *Fillet*, *Deck*, *Housefront*, and *Housetop*: in ***Select*** mode, select as many of the surfaces as you can, then click on ***Arrange – Group***, in the *New Group* dialogue, set the name to ***Main*** and click ***OK***.
2. With **Main** selected, click **Edit-Attributes**.
3. In the *Attributes* dialogue box, click the ***Group Editor*** button.
4. In the *Group Editor* dialogue box, include the remaining surfaces in the group: click on the Surface radio button, then select one of the remaining surfaces from the pick box, then click on the Add button. Repeat for the remaining surfaces.

5. Click on the ***Stbd + Port*** check box and click on ***OK***.
6. Click on ***OK*** at the *Attributes* dialogue.
7. Save the project: click on ***File*** – ***Save***.
8. Render the group. (See Figure 3-16.)

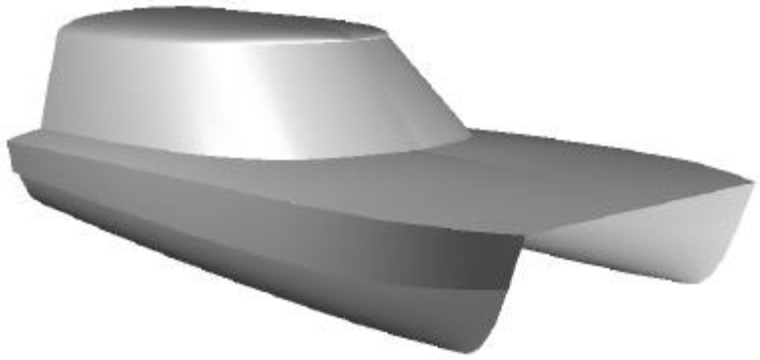


Figure 3-16: Rendered catamaran

3.10 Summary

In Tutorials 1 to 3, you have been shown a number of ways to create and edit ships and boats. These tutorials are intended to be an introduction to Auto*ship*. They do not describe the definitive methods for producing hulls. Using what you have learned in these tutorials, you should experiment with the program to develop the best methods for working with your particular type of vessel.

Tutorial 4 - Pocket Cruiser

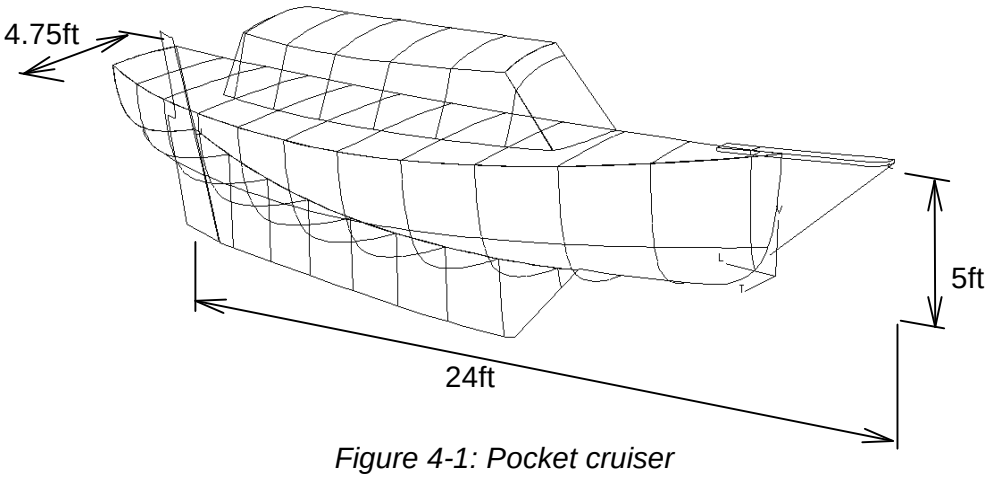


Figure 4-1: Pocket cruiser

4.1 Introduction – Pocket Cruiser

In Tutorial 4, what we will design is a small, heavy displacement cruising yacht. This tutorial goes beyond the basic hull lines and completes the superstructure as well. The shell is built up much as a real boat would be constructed. First the hull is created, and then the transom, keel, deck, and cabin are added. You can add the mast, rigging, and sails later, but they are not included as part of this tutorial.

The model we have constructed, **AS7TUT4.PR3**, has these dimensions:

- Displacement: 4.0LT
- LOA: 24ft
- Beam: 9.5ft

We suggest that you examine **AS7TUT4.PR3** before reading further.

4.2 Starting Out

➲ To begin:

1. Select ***File - New***.
2. Select ***Settings - Units*** and set the units to ***Feet*** and ***Pounds***.
3. Select ***Settings - Preferences***.
4. Click the ***Coordinate System*** button.
5. Click the ***Naval /Aircraft (US)*** radio button.
6. Click ***OK*** in the *Coordinate System* dialogue and click ***OK*** again in the *Preferences* dialogue.
7. Click on ***File*** – ***Save As***. Click ***OK***. In the *Save Project* dialogue, type ***TUT4.PR3***, and specify the ***MYTUTS*** directory to store this project in. In the *Project Info* dialogue field, type a descriptive message for the project and click on ***OK***.

4.3 Creating the Hull

1. Create a new surface:

Field	Entry
Name	Hull
Tab	Dimensions
Size	L = 24, T = 4.25, V = 5
Column Mesh	Number: 4; MaxDegree: 3
Row Mesh	Number: 4; MaxDegree: 2

Note: The number of rows and columns is based on previous experience. However, since you can easily add or delete rows or columns, you can start with fewer if you wish.

4.4 Shaping the Hull

➲ To shape the bow:

1. Hold down the ***Shift*** key and click on the ***Zoom Extents*** button to zoom out in all four views.
2. With *Hull* selected, switch to ***Edit Mode*** and change to ***Column*** mode if necessary.
3. Select the top vertex on the bow column (Col 0), and click the ***Isolate*** button.
4. Set this vertex to the centreline: type ***0*** in the *T* coordinate box, and click ***Enter***.
5. In *Front* view, straighten the whole column to the centreline: with the top vertex selected, click on the ***Straighten*** button and then click on the lowest vertex.
6. In ***S Side*** view, shape the bow profile by editing column 0 by eye. It's best to work with the control points. (See figure 4-2.)

Tip: It is easiest to edit the shape of a row or column with Isolate on, but you may find it beneficial to toggle between Isolate and non-Isolate during the next few steps.

Figure 4-2: Editing the Bow Profile

Note: in order to avoid a knuckle at the bottom of the stem in profile view, the vertices of Col 0/Row 1, Col 0/Row 0 and Col 1/Row 0 should all lie on a straight line. You can position them fairly close by eye, or draw a free, straight curve to use as a template, or use a hand calculator to find the exact position of one vertex based upon the postions of the other two. Refer to Tutorials 2 nd 3 for a further explanation.

7. Change to ***Rows***, select *Row 0* and form the fairbody shape.
8. Switch to ***Top*** view and shape the sheer line (Row 3, in our model.)
9. Switch to ***S Side*** view and shape Row 3 to get a pleasing sheer line. Experiment with using the **Planar** button to force the sheer into a plane: select the vertex on either Column 1 or 2 and click on the ***Planar*** button.

   The Planar routine calculates a plane based upon the positions of the two end vertices and the currently selected vertex and then adjusts the heights of the remaining surface-edge vertices to lie on that plane.
10. Switch to ***Front*** view and re-shape column 3 to form the aft end of the hull. In steps 7, 8 and 9, you positioned the bottom and top vertices in profile, so you should not move them now.

    Note: we will be adding a transom as a separate surface, so the hull must extend aft of where the transom corner will be. Do not worry about the profile at the aft end of the hull.
11. Also in ***Front*** view, re-shape columns 1 and 2 to form the shape in the mid-body. (See figure 4-3.)

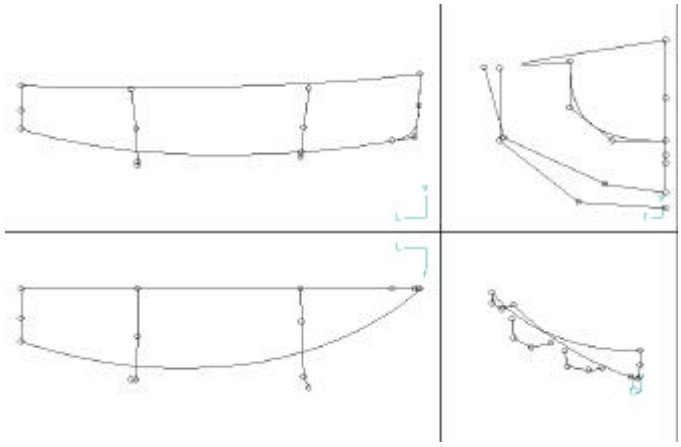


Figure 4-3: The Hull Surface

4.5 Instant Hydrostatics

4.5(a) – Group

At this point, it is time to think about displacement. We will use the instant hydrostatics feature. This routine works only on a group, so we must create a group.

➲ To create a group:

1. With the *Hull* surface selected, in *Select* mode, click on the ***Arrange – Group*** menu item. Rename the group to ***Main***, un-check the *Make Comps Invisible* check box and click on ***OK.***

4.5.(b) - Running Instant Hydrostatics

1. Click on the small ***H*** button near the lower left of the view window to open the *Instant Hydrostatics* window.
2. In the *Instant Hydrostatics* window, click on the ***Group*** button.

3. In the *Choose Hydrostatics Group* dialogue, select ***Main*** from the pick list and click on ***OK***.
4. In the *Instant Hydrostatics* window, note the *Depth* display in the lower left corner.
5. Click on the ***Update*** button.

   Note: If the *Invalid Parameters* message appears, the likely cause is that the hull is not actually sitting in the water at the specified Depth. Set the Depth to a higher value and click on Update again.

   We are aiming for 4.0 LT displacement. It is not very likely that the displacement at the draft you have set is exactly 4.0 LT, however we can get Autoship to find the draft corresponding to 4.0 LT.

6. Near the bottom centre of the *Instant Hydrostatics* window, set the *Mass* to ***4.0*** and click on the ***S*** button. After a brief calculation, new results will appear. The Depth value now displayed is the amount you need to shift the hull vertically to hit the target displacement at the design waterline.
7. On the main screen, select the Hull surface.
8. Right-click on the ***Move*** button. In the *Move Selection* dialogue, set the *V* value to the opposite of that shown for *Depth* in the *Instant Hydrostatics* window, i.e. if Depth is U.041, set the V to D.041. Click on ***OK***.
9. Click on the small H button to close the Instant Hydrostatics window.

   Note: If the value shown for *Trim* in the *Instant Hydrostatics* window is very large (> 1 degree) then it may take a few iterations of step 8 to hit the target draft.

4.6 Adding the Transom

We will create a cylindrical surface, Transom, by extruding a curve, Transom Shape. Then we will move the Transom to its correct position and intersect it with the Hull. The top of the transom will be defined by an embedded curve, Transom Top. The surface will them be trimmed off at a combination of the curves Transom Top and Transom-Hull intersection.

➲ To create the cylinderical transom:

1. Create a new curve:

Field	Entry
Name	Transom Shape
Tab	Arc
Plane	Top
Semi-Axes	Circular check box checked Radius = 30
Center	Somewhere 30 ft ahead of the transom
Angles	0 and 10°

2. Create a new surface named ***Transom*** by extruding the *Transom Shape* curve upward by 5 feet.
3. Right-click the ***Rotate*** button.
4. In the dialogue box, specify a ***Transverse*** axis and an angle of ***15***°, and then click **OK**.
5. In *S Side* view, move the transom so that it intersects the hull. You may find the transom easier to move if you re-position the base-point closer to the transom.
6. Select the ***Hull*** and ***Transom*** surfaces.
7. Click the ***Surf-Surf Intersect*** button, set the tolerance to 0.1 in. and click ***OK***.

8. This generates two intersection curves: *Transom Int1* (embedded on Hull) and *Hull Int1* (embedded on Transom). Rename *Transom Int1* to **Hull-Transom**, and *Hull Int1* to **Transom-Hull**.
9. Trim *Hull* off at the intersection with the transom: select the ***Hull*** surface, click on the ***Trim Surface*** button, in the *Trim Surface Hull* dialogue, select the curve ***Hull-Transom*** and click on ***OK***. Answer ***Yes*** at the appropriates *Keep This Domain* prompt and ***No*** to the others.

We cannot yet trim the transom surface as we do not have a curve which completely describes where to trim the surface.

➲ To establish the top of the transom:

1. In *Create* mode, click the **Create Point** button.
2. In the *Create Point* dialogue, type the following information, and click **OK**:

Field	Entry
Name	Transom Corner
Tab	Embedded
Host	Curve radio button selected Transom-Hull

3. Working in *Front* view, move the point *Transom Corner* to the top end of the curve *Transom-Hull*.

Tip: pull the point up above the end of the curve and it will jump back to the end of the curve.

4. Create an embedded curve, ***Transom Top***, embedded on the surface ***Transom*** and having ***3*** vertices.
5. Attach the outboard end of *Transom Top* to the point *Transom Corner*.

6. In *Front* view, edit *Transom Top* to the desired shape of the transom at deck level. You should position the middle control point at the same height as the vertex at centreline in order to maintain tangency across the centreline. (See Figure 4-4.)

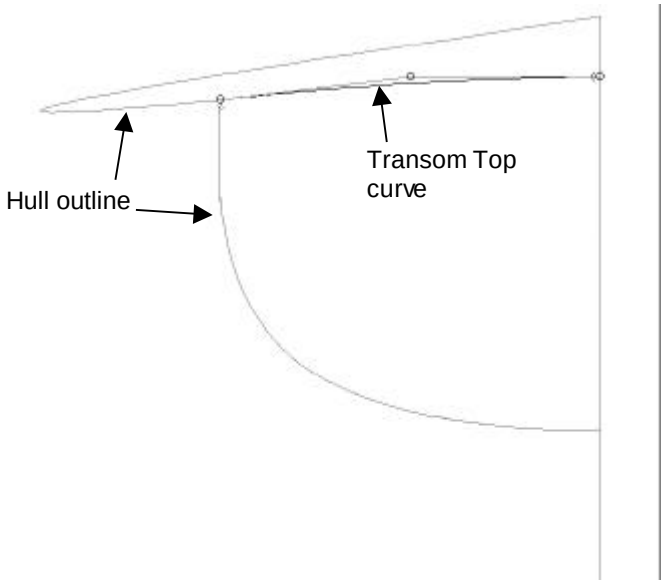


Figure 4-4: Transom Top curve

➲ To trim off the excess transom:

1. In ***Edit*** mode, with the outboard vertex of *Transom Top* selected, click the ***Join Curves*** button. In the *Join Curves* dialogue, name the new curve ***Transom Trim***, pick ***Transom-Hull*** and click on ***OK***.
2. Select the surface *Transom* and click the ***Trim Surface*** button, in the *Trim Surface Transom* dialogue, select the curve ***Transom-Hull*** and click on ***OK***. Answer ***Yes*** at the appropriates *Keep This Domain* prompt and ***No*** to the others.

4.7 The Keel

To create the keel, we will first layout the bottom of the keel with a free curve, then the top with a curve embedded in the hull, and finally create the keel as a ruled surface between these two curves.

1. Create a free curve, ***Keel Bottom***, at *V* = ***-4***, to establish a 4 ft draft. We specified 4 vertices and degree 2 and put the two ends at L = 22 and L = 9.

2. In ***Top*** view, edit the curve to the desired shape.

   You could have used a foil curve, but a true foil shape is not desirable for such a long keel.

3. Create an embedded curve, ***Keel Top***, with 4 vertices and degree 2, embedded on the surface *Hull*, and edit it to the desired shape.

4. Create a ruled surface, ***Keel***, between the curves *Keel Top* and *Keel Bottom*.

   If the surface is "twisted" (it's outline looks like a bow tie), it is because the two curves run in opposite directions. To fix this, select one of the curves, go to edit mode and click on the ***Reverse Ends*** button. Then select the Keel surface and click on ***Edit*** – ***Regenerate Object***.

5. Add another column at the same location as the bottom column.

6. Select the bottom column, switch to ***Isolate*** and straighten the bottom column between the forward-most and aft-most vertices. Pick a vertex near the middle of the column and switch to ***Rows*** to check your work.

4.8 The Deck

We will construct the deck from edge curves. We already have Transom Top. We need to create Deck CL, Deck Fwd and Deck Edge.

➲ To create Deck CL:

1. Create a point, ***Deck at Bow***, embedded on the surface *Hull*, and move it to the top of the stem.
2. Create a point, ***Transom Top at Center***, embedded on the curve *Transom Trim*, and move it to the top of the transom at the centerline.
3. Create a new curve:

Field	Entry
Name	Deck CL
Tab	Free
End 1	select: Deck at Bow check: Attach to point
End 2	select: Transom Top at Center check: Attach to point

➲ To create the curve Deck Fwd:

1. Create a free curve, ***Deck Fwd***, as a degenerate curve: check the *Degen. (Point)* check box in the *Create Curve* dialogue.

   Note: A degenerate curve is a curve that is condensed to a single point - all of its vertices are forced to lie at a given point in space.

2. Attach ***Deck Fwd*** to the point *Deck at Bow*.

➲ To create the curve Deck Edge:

- Create a curve:

Field	Entry
Name	Deck Edge
Tab	Surf Row/Col
Type	Embedded
Axis	Row
Parameter	0
Host Surface	Hull

➲ To create the Deck surface:

1. Create a new surface:

Field	Entry
Surface Name	Deck
Tab	Dimensions
Size	L = 24; T = 5; V = 0
Column Mesh	Number: 3; MaxDegree: 2
Row Mesh	Number: 3; MaxDegree: 2

2. Move it roughly into place.
3. Attach the four edges of the surface *Deck* to the following curves:
   - Col 0 → Deck Fwd
   - Col 2 → Transom Top
   - Row 0 → Deck CL
   - Row 2 → Deck Edge

Note that you may need to drag some of the vertices past the appropriate corners in order to pull the surface into the correct shape.

This exercise has established a rather complex chain of dependencies. The hull depends on the transom and two curves embedded on it. The keel and the deck both depend on the hull. The keel also depends on the curve Bottom of Keel. To check what an object's dependents are, select the object and click on the **Information (I)** button on the lower left of the screen.

4.9 The Cabin

By now you should have a good feel for some of the things you can do with Auto*ship*. By using the **Information** button and Attributes, you should be able to figure out how the cabin was created in **AS7TUT4.PR3**.

4.10 Groups

Groups provide a way of putting together the parts for viewing, calculating weight and volume, and exporting to Auto*hydro* and Auto*build*.

1. Create a group, **Main**, which includes all the surfaces made in this tutorial. Render the *Main* group to see your vessel.
2. Mirror the *Main* Group by selecting ***Edit – Attributes*** and in the Group Editor, click on the name ***Main***, click ***Stbd + Port*** and then click ***OK***. The result is a complete boat.
3. After creating the *Main* group, make a copy of this Group and call it ***Heeled 15***. Then, with the group ***Heeled 15*** selected, right-click the ***Rotate*** button. Type ***15 degrees*** and click ***OK***. The result is the boat heeled over 15 degrees. This lets you look at heeled waterlines and even see Heeled Buttocks! Try Quick Hydrostatics on this group.

Remember to save your work.

4.11 Summary

This concludes Tutorial 4. If you have done all four tutorials, you have now seen many different ways to design a hull in Autoship. With a little practice, you should be able to design your own hull quickly and easily.

Tutorial 5 - Ro-Ro Carrier

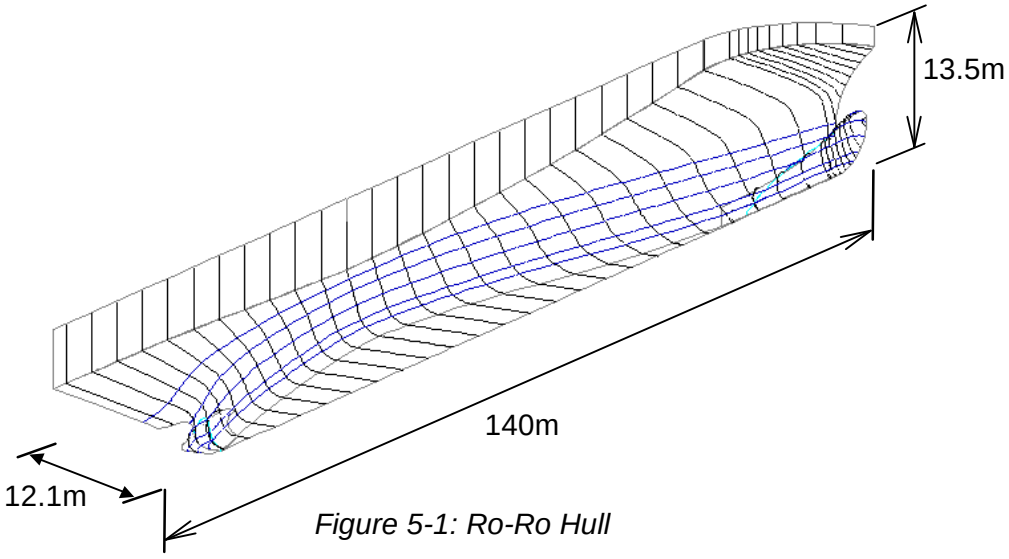


Figure 5-1: Ro-Ro Hull

5.1 Introduction – Ro-Ro Carrier

AS7TUT5.PR3 is intended as a ro-ro vessel. To create a voluminous hull, this design features a vertical panel running from bow to stern which joins the formed lower portion of the hull along a knuckle line. The hull also has a bulbous bow and a stern bulb.

Tutorial 5 shows one way to generate such a hull form. Take a look at the **AS7TUT5.PR3** file before continuing. The hull is composed of four surfaces and the bow and stern bulbs are added as separate surfaces.

Remember that this tutorial is for guidance only, and that with experience and skill you will be able to develop more suitable design techniques for your own work.

5.2 Starting Out

➲ To begin:

1. Create a ***New*** project.
2. Set units to ***Meters*** and ***Tonnes***.
3. Select ***Settings - Preferences***.
4. Click the ***Coordinate System*** button.
5. Click the ***Naval /Aircraft (US)*** radio button.
6. Save the file.

5.3 Midship Section

As in Tutorial 1, we will extrude a midship curve to generate the hull:

1. Create a ***Free*** curve, ***Midship Section***, with ***3*** control points and degree ***2***, having *End 1* at *L* = ***55F***, *T* = ***12.1S***, *V* = ***6.3U*** and *End 2* at L = ***55F***, *T* = ***0***, *V* = ***0***.
2. Select the middle control point and position it at *V* = ***0*** and about *T* = ***10***. Change it to a knuckle (***Toggle Corner*** button), and then apply a fillet (***Radius Corner*** button). We used a 3.0m fillet. (See figure 5-2.)

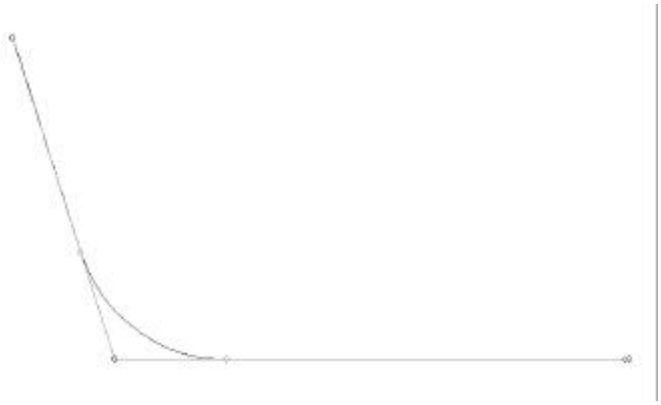


Figure 5-2: Midship Section Curve, Front View

You could generate a hull surface by simply extruding this curve forward or aft from amidships. However, the resulting surface would have two knuckles or chines, one just above and one inboard of the vertex you used to generate the radius. To avoid this problem, we will copy the curve Midship Section and edit out the knuckles while using Midship Section as a template:

3. Copy the curve *Midship Section* to ***Midship***: with the curve *Midship Section* selected, click on the ***Clone (Shift –Copy)*** button, name the curve Midship.
4. Switch to ***Edit*** mode and in ***Front*** view zoom in on the bilge radius area.
5. Select vertex 1 and click on the ***Toggle Corner*** button.
6. Select vertex 3 and click on the ***Toggle Corner*** button.
7. Using the arrow keys on the keyboard, move vertex 3 inboard until the shape of the lower portion of the curve *Midship* roughly matches *Midship Section*. Try to keep track of how far the vertex is moved.
8. Using the arrow keys on the keyboard, move vertex 1 up by the same amount vertex 3 was moved inboard. Straighten the curve between vertex 0 and vertex 2.

9. Repeat steps 7 and 8 until the shape of the curve*Midship* matches the curve *Midship Section*. (See figure 5-3.)

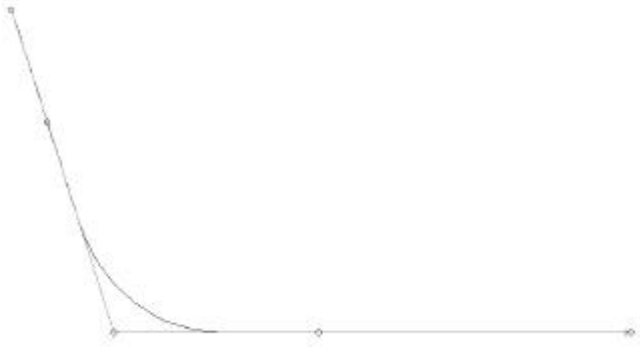


Figure 5-3: Midship Curve, Front View

5.4 The Hull Forward

It is fairly common to design a shape using surfaces that fit to each other along their original, untrimmed edges. In order to ensure that the two edges match without gaps, there are special considerations that must be taken into account. One method of fitting the edges together is to use the Attach routine. However, this has two shortcomings: the two surface edges may “wobble” and leave minute gaps; and the attached edge definition is not supported in the IGES file type – thus the fitted edges cannot be exported as an IGES file. Another method is to use the Match routine. This will match the surface edge to the object indicated within a set tolerance, but may also add extra vertices, thus making the surface more difficult to fair. The only way to absolutely guarantee that the two surface edges are identical is for the edges to be identical in degree of curvature, number of vertices and position of vertices.

There are tricks to use to construct a shape from multiple surfaces that ensure that the edges match properly. We will experiment with them in the forward portion of the hull.

First, we will create the surface Hull Forward Upper as a ruled surface between two curves: Deck Edge Fwd and Knuckle Fwd.

➲ To create the curve Deck Edge Fwd:

1. Create a ***Free*** curve, ***Deck Edge Fwd***, with ***3*** control points and degree ***2***, having *End 1* at *L* = ***55 F***, *T* = ***12.1S*** and *V* = ***13.5*** and *End 2* at *L* = ***137.5 F***, *T* = ***0*** and *V* = ***13.5***.
2. In ***Top*** view, move vertex 1 to *L* = ***137.5F*** and *T* = ***12.1S***.
3. Insert another vertex between vertex 1 and vertex 2 and move this vertex longitudinally to form the bow rounding. We suggest you keep the shape of this curve simple so that this edge-matching method will be easier to understand. (See figure 5-4.)

Figure 5-4: Deck Edge Fwd Curve, Top View

➲ To create the curve Knuckle Fwd:

1. Copy the curve *Deck Edge Fwd*, name the new curve ***Knuckle Fwd***.
2. Move *Knuckle Fwd* downward 2m.
3. In *S Side* view, edit *Knuckle Fwd* to the desired shape. Adjust the vertices in the vertical direction only. You may find it advantageous to add a control point or two. Be sure that the vertex 0 is at L= F55, T = 12.1S, V = U6.3 and that

vertex 1 is at T = 12.1S, V = U6.3 so that *Knuckle Fwd* connects with *Midship*. (See figure 5-5).

Figure 5-5: Knuckle Fwd Curve, S Side View

➲ To create the surface Hull Forward Upper:

1. Create a new surface, ***Hull Fwd Upper***, by extruding the curve ***Knuckle Fwd*** upwards by 2m.
2. ***Attach*** the upper column of *Hull Fwd Upper* to the curve ***Deck Edge Fwd*** to position the top edge of the surface, and then undo the attachment.

Note: Extruding the surface from the curve Knuckle Fwd ensures the bottom edge of the surface matches the curve Knuckle Fwd.

We will now create the surface Hull Fwd Lower by sweeping the curve Midship forwards along the curve Knuckle Fwd. This will create a surface that is identical at the aft end to the Midship Curve and identical at the top edge to both the curve Knuckle Fwd and bottom edge of the surface Hull Fwd Upper. The forward edge will be matched to a template curve, Stem.

➲ To create the curve Stem:

1. Create a *Free* curve, ***Stem***, with ***5*** control points and degree ***2*** having *End 1* at *L* = ***137.5 F***, *T* = ***0*** and *V* = ***11.5*** and *End 2* at *L* = ***130F***, *T* = ***0*** and *V* = ***0***.
2. Edit the shape as shown in figure 5-6.

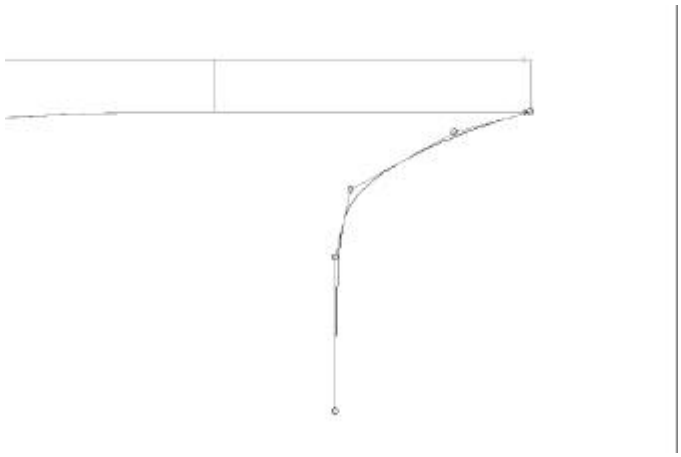


Figure 5-6: Stem Curve, S Side View

➲ To create Hull Fwd Lower:

1. Create a *Sweep* surface, ***Hull Fwd Lower***, for the *Pattern Curve* use *Midship*, for the *Guide Curve* use *Knuckle Fwd* and check the *Parallel* check box.
2. Zoom in on the forward end and drag the vertices of the forward-most column (col 4) into a close match to the curve *Stem*. You may find it easier with ***Isolate*** on. Be sure to position all the vertices at the centreline.
3. Use the Match feature to match the shape of column 4 to the curve *Stem*: click on the ***Attach*** button, in the *Attach Surface* dialogue, pick ***Match***, specify the curve ***Midship*** and click on ***OK***.

Note: The Match operation works well in this instance because the columns of surface Hull Fwd Lower have five vertices and the curve Stem also has five vertices.

3. Use ***Straighten*** to move all the vertices of the bottom row to the centreline. Note that you will have to move the vertex on column 3 aft.
4. Edit the shape of column 2 and 3 to achieve the shape desired. Do not move the vertices in the top row or else the top edge will no longer match the curve Knuckle Fwd.

Note: In Body view, the control points on columns 0 and 1 must be kept identical to ensure the forward portion of the hull will match up with the aft portion of the hull at midships.

5. If desired, add another row inboard of the row at centreline and convert it to a corner or knuckle row to define the flat bottom.

5.5 The Bow Bulb

We will create the bulbous bow as a separate surface and then trim it to the Hull Forward Lower surface.

➲ To create the Bulb at the bow:

1. In ***Front*** view, ***Draw*** a curve, ***Bulb Section***, to represent the sectional shape of the bulb. Use 3 vertices, degree 2 and make it approximately 4m high by 2m wide. Make sure the two end vertices are at T = 0 (Refer to Tutorial 3, Creating the Housetop Surface for instruction on how to Draw a curve) (See figure 5-7).

Figure 5-7: The Curve Bulb Section

2. Create a new surface, ***Bulb Fwd***, by extruding the curve *Bulb Shape* longitudinally. We specified a length of 20m, 3 columns and degree 2.
3. Move the surface into position.
4. Close the forward end of the shape: add another column at the same location as the forward-most column and set all of it's vertices to the same vertical coordinate. We uses 2.5. Make sure they are all at T = 0.
5. In ***S Side*** view, shape the top, bottom and forward edges by eye. Insert other columns as needed. The forward-most two columns define a smoothly closed shape. If you move any of the vertices on either of these column longitudinally you will distort the front end of the bulb.
6. In ***Top*** view, shape the middle row.
7. Ensure tangency across the top: add another row at the same position as the top-most row and move it's vertices outboard a little. Use these vertices to alter the sectional shape.

8. Ensure tangency across the bottom: add another row at the same position as the bottom-most row and move it's vertices outboard a little. Use these vertices to alter the sectional shape.

   Tip: Keep the aft-most vertex of this row at T = 0 to obtain a more usable intersection with the hull.

➲ To fit the Bulb:

1. Select both ***Hull Fwd Lower*** and ***Bulb Fwd***.
2. Click the ***Surf-Surf Intersect*** button to generate the intersection curves.

   Note: You may find an additional set of intersection curves positioned along the centreline, or the intersection curves may extend along the centreline at the bottom. These portions are not needed, so they can be deleted.

3. Rename *Bulb Int1* to ***Hull Fwd Lower-Bulb Fwd*** and *Hull Forward Lower Int1* to ***Bulb Fwd-Hull Fwd Lower***.
4. Select ***Bulb Fwd*** and click on the ***Trim Surface*** button.
5. In the *Trim Surface Bulb Fwd* dialogue, select the curve ***Bulb Fwd-Hull Forward Lower*** and click on ***OK***. Answer ***Yes*** to the appropriate *Keep this Domain* prompt and ***No*** to the others.

➲ To create the trimmed Hull Forward Lower:

1. Using the same method as 3 and 4 above, trim *Hull Forward Lower* at the intersection with the Bulb. You may have to re-work the aft end of the intersection curve to obtain a closed curve.
2. Select **View-Render** to see your work in perspective.

5.6 The Hull Aft

A similar procedure could be followed to design the stern section. However, we will introduce another method which can be used to build up the hull shape from curves while ensuring the different parts match where they join. In this method, the hull will be defined with a single surface, yet constructed from more than two curves.

➲ To create the curve Deck Edge Aft:

1. Create a ***Free*** curve, ***Deck Edge Aft***, with ***3*** control points and degree ***2***, having *End 1* at *L* = ***55 F***, *T* = ***12.1S*** and *V* = ***13.5*** and *End 2* at *L* = ***2.5 A***, *T* = ***12.1*** and *V* = ***13.5***.

➲ To create the curve Knuckle Aft:

1. Copy the curve *Deck Edge Aft*, name the new curve ***Knuckle Aft***.
2. Move *Knuckle Fwd* downward 6m.
3. In *S Side* view, in Edit mode, click on the middle control point and add another control point using the Refine option. Edit *Knuckle Aft* to the desired shape. Adjust the vertices in the vertical direction only. Be sure that both the forward-most vertex and the next one aft is at V = U6.3 so that *Knuckle Aft* connects smoothly with *Knuckle Fwd*. (See figure 5-8).

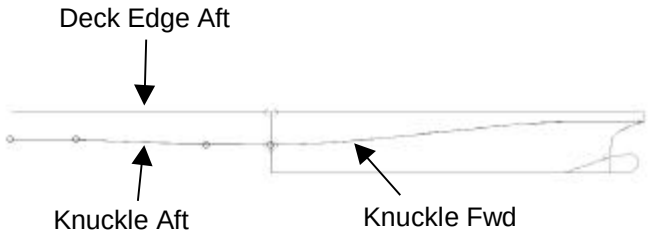


Figure 5-8: Deck Edge Aft and Knuckle Aft Curves

➲ To create the curve Knuckle Aft:

1. Create a ***Free*** curve, ***Centreline Aft***, with ***6*** control points and degree ***2***, having *End 1* at *L* = ***55 F***, *T* = ***0*** and *V* = ***0*** and *End 2* at *L* = ***2.5 A***, *T* = ***0*** and *V* = ***5***.
2. Arrange the vertices as shown in figure 5-9.

Figure 5-9: Centreline Aft Curve

➲ To create the surface Hull Aft:

1. Create a ***Sweep*** surface, ***Hull Aft***, using *Midship* as the *Pattern Curve* and *Centreline Aft* as the *Guide Curve*, check the Parallel checkbox.

You will notice that the surface is created below and to the port of the curve Centreline Aft. Remember that a Sweep surface sweeps the first end of the Pattern curve along the first Guide curve. In this case, the first end of the curve Midship is up at the knuckle line – not at the baseline. In this case, there is an easy fix:

2. Select the curve ***Midship*** and go to ***Edit*** mode.
3. Click on the ***Reverse Ends*** button.
4. Select the surface ***Hull Aft***.
5. Select the ***Edit*** – ***Regenerate*** Object menu option.

This will re-construct the surface in the originally intended manner. Now, we must form the surface.

6. Attach the upper edge of the surface to the curve *Knuckle Aft*: switch to ***Edit*** mode, select the upper-most row (Row 4), click on the ***Attach*** button, pick ***Attach to Curve***, and for *Attach to* pick ***Knuckle Aft***.
7. Move the aft-most two contol points of Row 3 downwards so that they lie below the curve *Knuckle Aft*. Try to position them so Row 3 is smooth.
8. In *Front* view, check each column of control points to make sure there are no bumps or hollows. Turn on the surface mesh and inspect it in each view to make sure it has a "smooth flow".

Now we continue to build the Hull Aft surface higher.

9. Add another row at the same position as the top-most row (Row 4): select Row 3, click the ***Add Row or Column*** button and for *Proportion*, input ***1***.
10. Select the new top-most row, (Row 5). You may find it easier to locate Row 5 by toggling to *Edit* points.
11. ***Detach*** Row 5 and then ***Attach*** it to the curve *Deck Edge Aft*.

12. Turn Row 4 into a corner: select Row 4 and click on the ***Toggle Corner*** button.

This completes the Aft Hull surface. You have now seen two ways to construct a large surface from several curves, and assure that the overall shape is contiguous and smooth.

5.7 The Stern Bulb

We will create the stern bulb in three steps:

1. Define the lower profile of the stern bulb by creating a curve called ***Stern Bulb Profile*** in side view. See Figure 5-10.

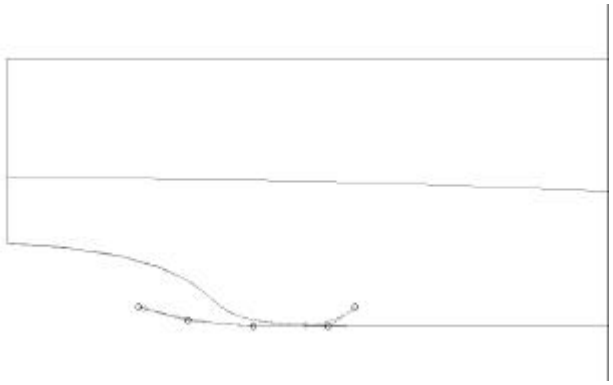


Figure 5-10: Curve Stern Bulb Profile

2. Create a surface, ***Bulb Aft***, as a type Rotate surface; pick Longitudinal for Axis and specify an angle of 180 for Rotation.

Tip: move Stern Bulb Profile's base point to the height of the centreline of the bulb – by default, the surface of rotation is be created by revolving the curve about it's base point.

3. If the *Bulb Aft* is not already in the correct position, re-position it and then intersect it with *Hull Aft*. Remember to rename the intersection curves.
4. Trim both the *Bulb Aft* and *Hull Aft*.

5.7 Summary

This concludes Tutorial 5. Have a good look at the project to get an idea of how the groups are constructed. If you want to test the hydrostatics, the design draft is 5.4 m.

Tutorial 6 - Oil Rig

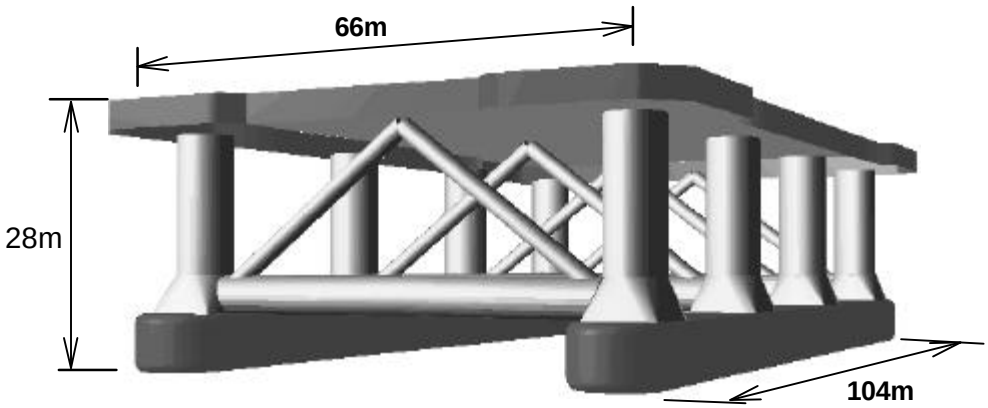


Figure 6-1: Oil Rig

6.1 Introduction – Oil Rig

Autoship can be used to create accurate models of large, complex structures which are not boats or ships. This oil rig is one such example. This model was created by constructing one quarter of the structure and then copying, mirroring, and grouping it to form the rest of the structure. In this way, you can create a model from a minimum number of individual components.

This tutorial also shows how to export a complex structure to Autohydro.

AS7TUT6.PR3 and this tutorial show one way to construct a model for such a structure.

6.2 Starting Out

➲ To begin:

1. Click on ***File*** – ***New*** to initialize Autoship.
2. Set units to ***Meters*** and ***Tonnes***.
3. Set the coordinate system to **Naval /Aircraft (US)**.

Our approach for the construction of this rig will be to create the elements needed for one quarter of the structure and then assemble them into a group. Then, by using copy and mirror operations upon the group, we will produce the entire model that will be used for visual purposes. Finally, we will re-assemble the basic elements to produce a different model of the same structure which can be exported to Autohydro.

6.3 Pontoon

First, we will create two curves, Midship and Waterline and then sweep Midship along Waterline to create a surface that represents the aft, forward one-eighth of one pontoon. Then, through a series of clone and mirror operations, we will create the aft half of one pontoon.

➲ To create the curve Midship:

1. Create a new curve, ***Midship***, with ***3*** *Control Pts* and *Max Degree* ***2*** having *End 1* at ***0***, ***5***, ***0*** and *End 2* at ***0***, ***0***, ***5***.
2. Move the middle *Control Point* (not *Edit Point*) to 0, 5, 5, toggle it to a corner and then use the***Radius Corner*** button to form a *Radius* of ***1.0***.

➲ To create the curve Waterline:

1. Create a new curve, ***Waterline***, with ***3*** *Control Pts* and *Max Degree* ***2*** having *End 1* at ***0***, ***5***, ***0*** and *End 2* at ***50***, ***0***, ***0***.

2. Move the middle *Control Point* to ***50***, ***5***, ***0*** and then toggle it to a corner and then use the ***Radius Corner*** button to form a *Radius* of ***5.0***.

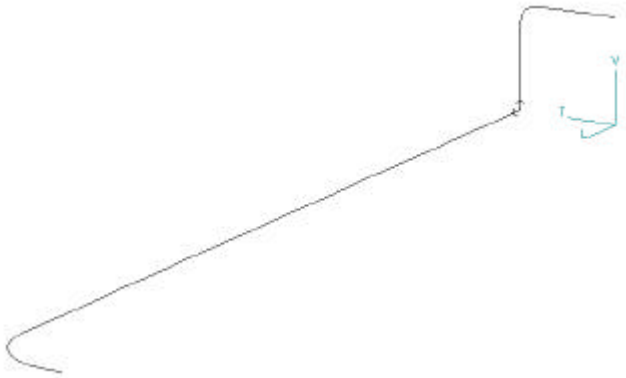


Figure 6-2: The Curves Midship and Waterline in Para View

➲ To create half of the upper aft portion of the pontoon:

1. Create a ***Sweep*** surface, ***Upper Surf***, by sweeping the curve *Midship* along the curve *Waterline*.

➲ To create half of the lower aft portion of the pontoon:

1. With the surface *Upper Surf* selected, click on the ***Clone*** button. In the New Group dialogue, give the name ***Lower Clone***.
2. Click in the *S Side* view window, if it is not the currently selected view.
3. With *Lower Clone* selected, click on the ***Flip Vertical*** button.

➲ To create the other half of the aft portion of the pontoon:

1. Select both *Upper Surf* and *Lower Clone* and click on the ***Clone*** button. In the New Group dialogue, give the name ***Port Clone***.
2. Click on the *Front* view window.
3. With *Port Clone* selected, click on the ***Flip Horizontal*** button.

➲ To create the half of the the pontoon:

1. Create a group, ***Pontoon Half***, consisting of *Upper Surf*, *Lower Clone* and *Port Clone*.
2. Move *Pontoon Half* 33.0m to starboard. (Tip: right-click on the ***Move*** button.)

6.4 The Column

We will create one column and copy it to produce the others. The column will be constructed in two parts: the lower and the upper part.

➲ To create the cone (lower part):

1. Create a ***Rectangle*** type curve, ***Cone Bottom*** to represent the bottom of the cone. We set *Plane* to ***Top***; *Centre* at **42.0**, **0.0**, **5.0**, specified ***Square*** and a *Height* of ***7.5***. You can round the corners of the bottom curve if you like.
2. Apply a 0.5m radius at each corner.
3. Create an ***Arc*** type curve, ***Cone Top***, to represent the top of the cone and the bottom of the cylinder. We set *Plane* to ***Top***; *Centre* at ***42.0***, ***0.0***, ***10.0***, *specified* ***Circular*** and a *Radius* of ***3.0***.
4. In *Top* view, compare the arrows at the beginning of *Cone Bottom* and *Cone Top* to ensure that both curves start at the

same relative location - aft or forward, and that both curves point the same way.

5. Create a ***Develop*** surface, ***Cone*** between *Cone Bottom* and *Cone Top*.

➲ To create the cylinder (upper part):

1. Create an ***Extrude*** surface, ***Cylinder***, by extruding the curve *Cone Top* ***18m*** vertically.

6.5 The Cross Beams and Diagonals

We will create one cross beam and one diagonal and copy them to produce the others.

➲ To create a cross beam:

1. Create an Arc curve, ***Cross Beam Template***. We set *Plane* to ***Side***; *Centre* at ***42.0***, ***0.0***, ***7.5****, specified* ***Circular*** and a *Radius* of ***2.0***.

2. Create an ***Extrude*** surface, ***Cross Beam***, by extruding the curve *Cross Beam Template* ***33m*** to port.

3. Obtain the ***Surf-Surf*** intersection curves of *Cross Beam* and *Cone* and trim off *Cross Beam* at the intersection.

➲ To create a diagonal:

1. Create a curve, ***Diagonal Axis*** to represent the axis, or centreline of the diagonal tube. We set *Control Pts* to ***2***, *Max Degree* to ***1***, *End 1* to ***42.0***, ***0.0***, ***5.0*** and *End 2* to ***42.0***, ***P33.33***, ***28.23***.

Note: we will need to trim off the top of the Diagonal and so need to extend it past where it will be trimmed at L = 42.0, T = 33.0P, V = 28.0. In this case, we chose to extend the curve 10%, or T = 0 + (1.1 x 33.0P) = 33.33P and V = 5U + (1.1 x (28.0U – 5.0U)) = 28.23U. (The 5U is the V at End 1.)

---

2. Create a ***Tube Sweep*** surface, ***Diagonal*** using the curve *Diagonal Axis*. We set *Radius* to ***1.0***.

3. Obtain the ***Surf-Surf*** intersection curves of *Diagonal* and *Cross Beam*. Note that Diagonal penetrates Cross Beam twice, so there will be more than one intersection curve generated. To avoid later confusion, delete the unnecessary curve(s).

4. Create a ***Dimensions*** type surface, ***Temp***, with *L* = ***6***, *T* = ***-4*** and *V* = ***-4***, *Column Mesh*: *Number* ***2***, *Max Degree* ***1***; *Row Mesh*: *Number* ***3***, *Max Degree* ***1***.

5. Move *Temp* by ***39.0A***, ***29.0P***, ***28.0U***.

   Note: We want to move the centre of the surface (L = 3.0, T = P4.0, V = 0) to L = 42.0, T = P33.0, V = 28.0. Hence the displacement is L = (42.0 – 3.0) = 39.0, T = (P33.0 – P4.0) = P29.0, V = (28.0 – 0) = 28.0.

6. Obtain the intersection of *Temp* and *Diagonal* and rename the curves.

7. Trim *Diagonal* off at the intersection with *Cross Beam* and the intersection with *Temp*. You may wish to delete the curve embedded on *Temp* and the surface *Temp*.

➲ To create multiples of the columns, cross beams and diagonals:

1. Create a group, ***Column & Bracing***, which contains the surfaces *Cone*, *Cylinder*, *Cross Beam* and *Diagonal*.

2. Clone the group *Column & Bracing*, name it ***Column & Bracing 2*** and move it 28.0m forward.

3. Move *Column & Bracing* 33.0m to starboard. Note that *Column and Bracing 2* also moves, since it is a clone. (Refer to figure 6-3.)

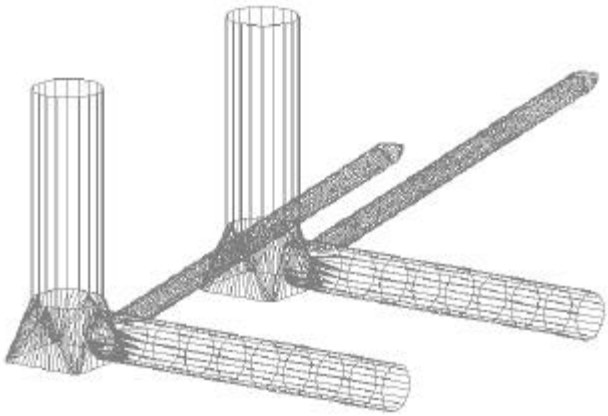


Figure 6-3: Groups Column & Bracing and Column & Bracing 2 in Para View.

6.6 The Deck Assembly

The deck assembly includes two parts: the horizontal deck and the vertical surface, Deck Skirt. As with the pontoons, columns and braces, we will create one quarter of each structure and use that to produce to entire structure.

➲ To create the vertical surface Deck Skirt*:*

1. Create a curve, ***Deck Edge***, using the ***Draw Curve*** option, that represents the outer edge of one quarter of the deck. (Once you have drawn the curve, click the right mouse button, name the curve Deck Edge, and set Maximum Degree to 1, and Method to Control Points. See the Autoship Reference Manual for more details.) Refer to figure 6-4.

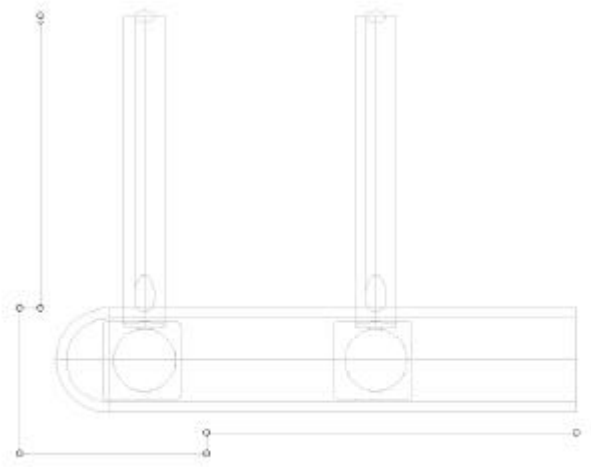


Figure 6-4: The Curve Deck Edge in Top View

2. Toggle vertices 1,2, 3,4 and 5 into corners.
3. In the Attributes dialogue, set the degree to 2 and resolution to 40.
4. Apply a radius of 1.5m at vertices 2, 3 and 4.
5. Move the curve Deck Edge upwards 28.25m.
6. Create an ***Extrude*** surface, ***Deck Skirt***, by extruding the curve Deck Edge downward by 3.0m.

➲ To create the horizontal surface deck:

1. Create a curve, ***Deck Template*****,** having *End 1* at ***0, 0, 28*** and *End 2* at ***60, 0, 28***.
2. Create and ***Extrude*** type surface, ***Deck***, by extruding *Deck Template* 50.0m to starboard.
3. Obtain the intersection of *Deck* with *Deck Skirt* and trim off *Deck* to that intersection.

4. Create a group, ***Deck Quarter***, containing the trimmed surface Deck and the surface Deck Skirt.

6.7 Assembling One Quarter of the Rig

All the components needed to make up one-quarter of the rig have been created. We will now assemble the components into one quarter of the rig and copy and mirror the quarter to produce the entire rig.

1. Create a group, ***Quarter***, containing the group *Pontoon Half*, *Column & Bracing*, *Column & Bracing 2* and *Deck Quarter*.
2. Clone the group *Quarter*, and name the new group ***Quarter 2***.
3. In the *S Side* view, with *Quarter 2* selected, click on the ***Flip Horizontal*** button.
4. Make another group, ***Main***, which contains *Quarter* and *Quarter 2*.
5. With *Main* selected, in the *Group Editor* dialogue, click on the ***Stbd + Port*** button.

Your oil rig is complete!

6.8 Assembling an Autohydro model

Autohydro models are comprised of parts and components. A component simply describes an enclosed volume in space. A Part is composed of a component, or a set of components, and is assigned specific attributes, thereby creating a complex volume with specific behaviour. An important attribute is the side designation – whether the part should exist only on the port side or the starboard side, or be mirrored across the centreline.

Autoship exports a selected group to Autohydro as a single-part model, and each object contained in the group is exported as a component of the single part. Therefore, it is important that each object contained in the group defines a single, enclosed volume. Further, the “side” of the part will be determined by the group: If the Stbd + Port check box for the group is turned on, then the part will be a centreline part, otherwise it will be a starboard part.

Consequently, we need to construct a different model in which each object contained in the export group describes a single volume. We could construct half of the model and have it mirrored across the centreline, but then we would not have access to the individual pontoons or columns in case we wished to fit tanks inside them. Instead, we will model each volume separately: our model will consist of separate components that describe:

- Each entire pontoon (2 components);
- Each vertical column (8 components);
- Each cross beam (4 components);
- Each diagonal (8 components);

We will not bother with the deck structure.

Tip: To make it easier to visualize the following operations, we suggest you go to the Attributes dialogue and turn off the visibility of all groups except for Pontoon Half.

➲ To create the pontoon components:

1. Create a clone, ***Pontoon Half 2***, of the group *Pontoon Half* and flip it to the other side of L = 0.
2. Create a group, ***Pontoon.S***, which contains the group *Pontoon Half* and *Pontoon Half 2*.
3. Create a clone, ***Pontoon.P***, of the group *Pontoon.S,* and flip it to the other side of T = 0.

➲ To create the column components:

1. Create a group, ***Column 1.S***, containing the surface *Cone* and the surface *Cylinder* and move 33.0m to starboard. Move it's base point to T = 0.
2. Create a clone, ***Column 2.S***, of the group *Column 1.S* and move it 28.0m forward.
3. Create a clone, ***Column 3.S***, of the group *Column 2.S* and move it 28.0m forward.
4. Create a clone, ***Column 4.S***, of the group *Column 3.S* and move it 28.0m forward.
5. Create a clone, ***Column 1.P***, of *Column 1.S* and flip it to the other side of the centerline.
6. Create a clone, ***Column 2.P***, of *Column 2.S* and flip it to the other side of the centerline.
7. Create a clone, ***Column 3.P***, of *Column 3.S* and flip it to the other side of the centerline.
8. Create a clone, ***Column 4.P***, of *Column 4.S* and flip it to the other side of the centerline.

➲ To create the cross beam components:

1. Create a clone, ***Cross Beam Clone***, of the surface Cross Beam and move it 33.0m to starboard.
2. Create a group, ***Cross Beam 1***, of *Cross Beam Clone* and in the Group Editor, check the *Stbd + Port* checkbox.
3. Create a clone, ***Cross Beam 2***, of *Cross Beam 1* and move it 28.0m forward.
4. Create a clone, ***Cross Beam 3***, of *Cross Beam 2* and move it 28.0m forward.

5. Create a clone, ***Cross Beam 4***, of *Cross Beam 3* and move it 28.0m forward.

➲ To create the diagonal components:

1. Create a clone, ***Diagonal 1.S***, of the surface *Diagonal* and move it 33.0m to starboard. Move it's base point to T = 0.
2. Create a clone, ***Diagonal 2.S***, of the group *Diagonal 1.S* and move it 28.0m forward.
3. Create a clone, ***Diagonal 3.S***, of the group *Diagonal 2.S* and move it 28.0m forward.
4. Create a clone, ***Diagonal 4.S***, of the group *Diagonal 3.S* and move it 28.0m forward.
5. Create a clone, ***Diagonal 1.P***, of *Diagonal 1.S* and flip it to the other side of the centerline.
6. Create a clone, ***Diagonal 2.P***, of *Diagonal* 2.*S* and flip it to the other side of the centerline.
7. Create a clone, ***Diagonal 3.P***, of *Diagonal 3.S* and flip it to the other side of the centerline.
8. Create a clone, ***Diagonal 4.P***, of *Diagonal* 4.*S* and flip it to the other side of the centerline.

➲ To create the export group:

1. Create a group, ***Autohydro***, which contains all the groups/clones you have formed in this section except for Pontoon Half 2. Do not check the *Stbd + Port* checkbox.
2. With the group *Autohydro* selected, click on the ***File*** – ***Export*** – ***Autohydro*** menu option.
3. In the Export Autohydro file dialogue, give a name and indicate a directory to store the file in and click on OK.
4. At the prompt "User Defined Sections?", click on No.

5. At the prompt “Name for GF1 part”, click on OK.
6. Start Modelmaker, load the file just written and experiment with the various views.

Note: You may notice that some of the components are not defined with many sections. This is because you answered No to the prompt “User Defined Sections”. To improve the definition, in Autoship open the Contours dialogue and set up a series of stations that provides coverage over the entire length of the pontoons (~3m interval) while being more dense around the columns, cross beams and diagonals and at the ends of the pontoons (~.3m interval).

6.9 Summary

This finishes tutorial 6 – the last of the tutorials. You have now explored many of the important features of Autoship and have experimented with various different methods of constructing a model. You have probably realized by now that Autoship is a very powerful program and that there is a lot to learn. As with many things, the only way to learn Autoship is to work with it, and that will take time. One option to shorten the time is to take a training course. The best option, however, is to experiment and seek help when things go wrong. Good luck and have fun!

Appendix:
Auto*3D* Module

Auto3D is a specialized module for working with Autoship DRA files created in the Autoship DOS version, or Autoyacht (DOS), as well as Autoship for Windows. You can perform the following functions:

Rendering Auto3D shows solid (or shaded) views of models. You can also use it to produce a rotating slide show of your Autoship model.

Plotter Output Auto3D generates output for plotters, either directly or to a file.

CAD Output Auto3D generates output to various CAD programs.

Running Auto3D

To run Auto3D, double-click on the Auto3**D** icon in the Autoship program group. The main display appears:

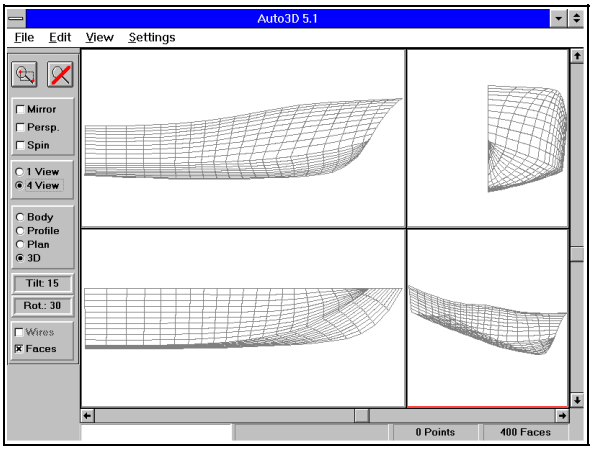
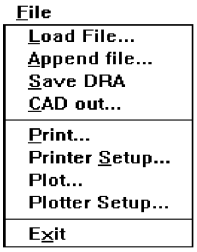


Figure 1: The Auto3D main display

The File Menu

The file menu is used for importing and exporting files, and to print files.



The File menu contains the following items:

Load File	Load an initial DRA file.
Append File	Add another DRA file to the initial file.
Save DRA	Save DRA file.
CAD Out	Export the model to a variety of CAD formats.
Print	Print the model on the screen.
Printer Setup	Bring up the Windows Printer Setup dialog box.
Plot	Bring up the Plotter Preview dialog box to set up for a plot.
Plotter Setup	Bring up the Plotter Setup dialog box for configuring Auto3D for your plotter.
Exit	Exit Auto3D.

➲ *To plot a view:*

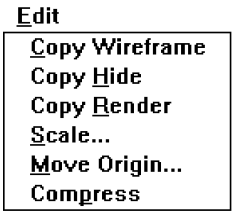
- Before plotting a view, select **File-Plotter Setup** and ensure the configuration settings are correct for your plotter.
- Then select **File-Plot**.
- In the Plotter Preview dialog box that appears, specify how the view is drawn.
- When all the settings are correct, click **OK**.

➲ *To export to a CAD program:*

- Select **File-CAD Out**.
- In the CAD Exchange dialog box that appears, select the type of exchange and the destination to send the file to. If you just want a typical lines plan of Body, Plan and Profile, use the 2D exchange, with a file for each view, as it is the fastest and easiest method.

The Edit Menu

The Edit menu is used to copy views of the model or its parts to the clipboard for use in graphics programs. It is also used to scale the model or move the origin.



Copy Wireframe Copy the wire mesh view of the model into the clipboard, for use with graphics programs.

Copy Hide Copy the wire mesh view into the clipboard, with the hidden lines removed, for use with graphics programs.

Copy Render Copy the rendered view of the model into the clipboard, for use with graphics programs.

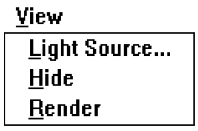
Scale Scale the model longitudinally, transversely and/or vertically.

Move Origin Move the origin by a specified amount. (The numbers you specify are always added to the origin, so if you want to move the model forward, you must type in negative numbers.)

Compress Compress DRA files that become unmanageably large.

The View Menu

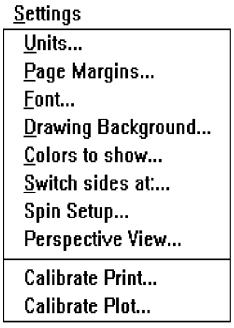
The View menu is used to control the way the model is viewed, or rendered. Rendering produces a shaded image of your vessel. If the DRA file was created with Auto*ship* for Windows, and if the surface mesh was turned on, then after you load it into Auto3D you can render it by selecting **View-Render**. The shading depends on the light source you selected with **View-Light Source**.



Light Source	Set the direction of the light source for rendering.
Hide	Toggle between showing and removing hidden lines.
Render	Generate a solid representation of the model.

The Settings Menu

The Settings menu sets user preferences.



Units	Set the units being used for the model.
Page Margins	Set up the page margins for the printed page and select the specific printer to use.
Font	Set the font style and point size
Background	Set the background colour for the screen.
Colours to Show	Select which colours will show in the drawing. Note that each line type is associated with one colour.
Switch Sides At	For Body view only, specify where to switch from showing contours on one side of the centreline, to showing contours on the other side. Typically, the mid-point of the model.

Spin Setup	Create a slide show of 13 rotational views of your model. Specify the rotation angle and the length of time each view remains on the screen.
Perspective View	Vary the distance from which you see the 3D view of the model.
Callibrate Print/Plot	Scale the model to fit the plotter or printer.

The Side Icon Box

The Side Icon Box contains tools for controlling the way your model appears and for manipulating your model.

Zoom-In Zoom in on areas within current view

No Zoom Return the view to its full extent.

Mirror When selected, shows both sides of the model.

Perspective Gives a perspective view of the model.

Spin If Spin Setup is defined, starts the slide show of your model.

1View Show single view of model.

4Views Show four views of the model.

Note: Double-clicking in 1 View changes to 4 Views. Double-clicking in 4 Views changes to 1 View.

Single View Select a view to show for single view.

Tilt Shows the amount of tilt in the current view. (Use the bottom bar controls to tilt your model.)

Rotate Shows the amount of rotation in the current view. (Use the side bar controls to rotate.)

Wires/Faces Select whether to show wires or surface faces for the model.

Note: Only models exported from Auto*ship* for Windows with mesh selected will have faces enabled.