Surface Modeling - II

The goal of this exercise is not to teach you how to build a boat, but to learn how to fit curves and surfaces. To do this, you need to understand how surfaces are described within a CAD system. In this exercise we'll focus on NURBS (Non-Uniform Rational B-Spline) curves & surfaces.

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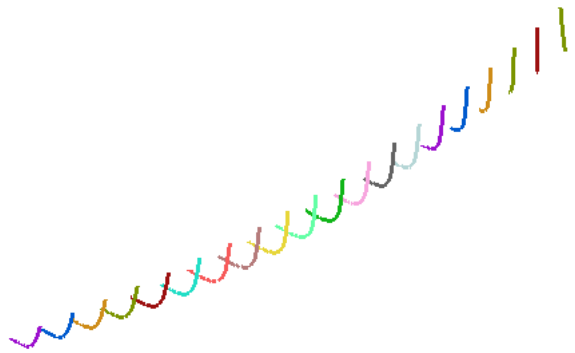
1. Step 1: Full Speed Ahead

NOTE:

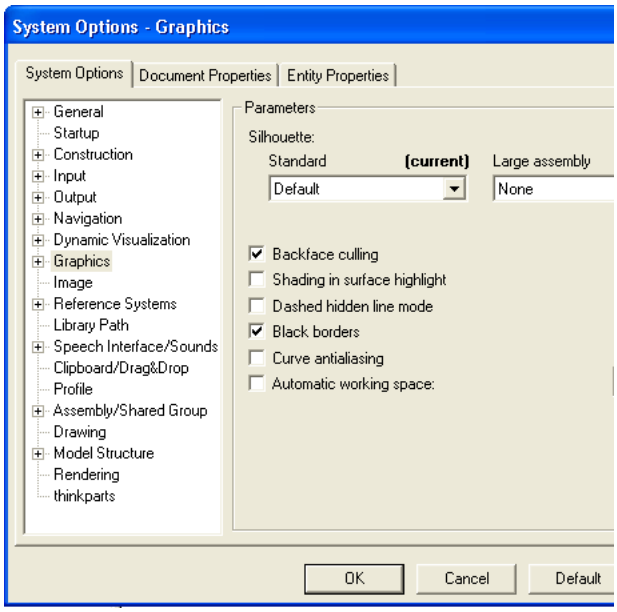
With a double click on the exe file you can run the webtraining session. ThinkDesign will be open with the right model to start.

If request to open a file, you can find it in the C:\MyTraining path.

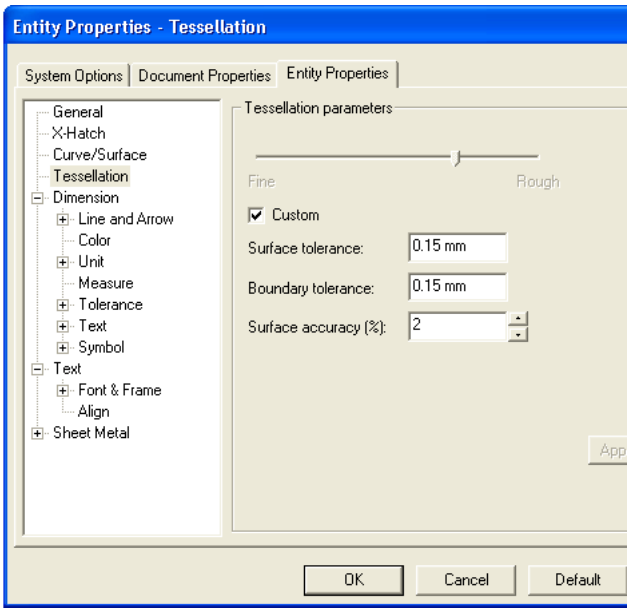


Just a quick set up needed. Since the boat we are creating is so large (about 20 Meters), we need to do an adjustment to the Tessellation settings in the Graphics tab of Options to view the shaded model properly.

- Increasing the Bounding box size as shown so that the model can be seen in full when shaded or active Automatic working space.



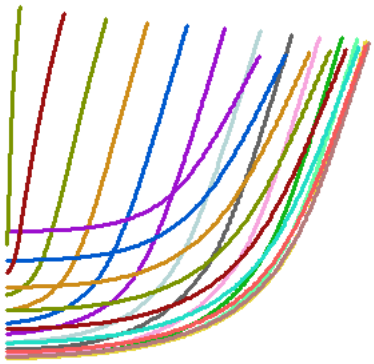
- Increasing the Boundary tolerance and Surface tolerance for a large model will improve performance.
- Hit OK.



First, we'll start with the hull sections provided by the engineer and create a U Curve surface.

The hull section curves are in halves, so let's start by hiding the port half.

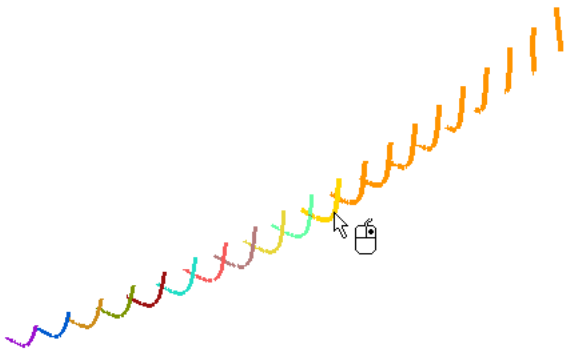
- Change to **View on Work Plane**.
- Use the keyboard step actions to rotate the view by holding the Alt key and hitting the → right arrow key three times.
- **Hide Entities** the left half of the sections.



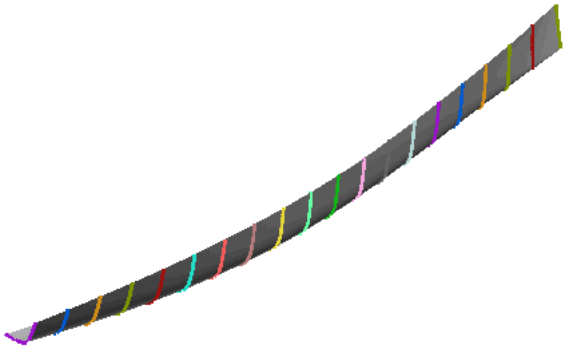
Now let's create a **Lofted Surface** surface from these curves. New surface will interpolate the curves exactly.

- Start **Lofted Surface**.
- Pick Select As Grid and Boundary Set A. Select by window all curves.

> **Note: Order of selection.**
>
> If it is important a properly order to select the curves especially with complex model, select the curves, one at a time, from front to back.
>
> If the preview will be curled, pick More Options and Surface Type Options. Checking Manual Ordering/ Orientation there will be the possibility to correct wrong orientation of sections.



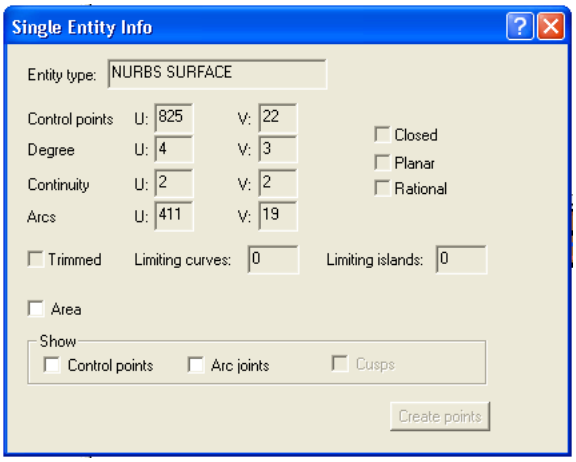
- Pick More Options. In the 'Type' option, you can find the method that Thinkdesign will use to create the surface; Grid in this case.
- Still under More Options but in Surface Type Options set Parameterization as Intrinsic mode.
- Hit OK.



The surface looks pretty good, but let's look a little deeper with **Single Entity Info**.

- Right click on the surface and start the **Single Entity Info** command

The Single Entity Info dialog box displays important information about the surface. First of all, we can see that it's a NURBS surface. We can also see that there are many control points and arcs, especially in the U direction.



Why are there so many control points?

Since we used interpolation, the number of control points is based on the curves used to create the surface. If the curves are not relatives (i.e. all having the different number of arcs and control points), the number of control points is calculated using a sort of sum. We will go into a little more detail on this in the next step.

With that many control points, we would call this an heavy surface. Let's see how we can clean up this surface with **Convert into NURBS**.

Typically, NURBS conversion is done automatically to specialized surfaces when required for editing purposes (based on the Convert to NURBS). However, you can manually convert surfaces to NURBS, and you can even use Convert to NURBS on an existing NURBS surface to redefine and refit.

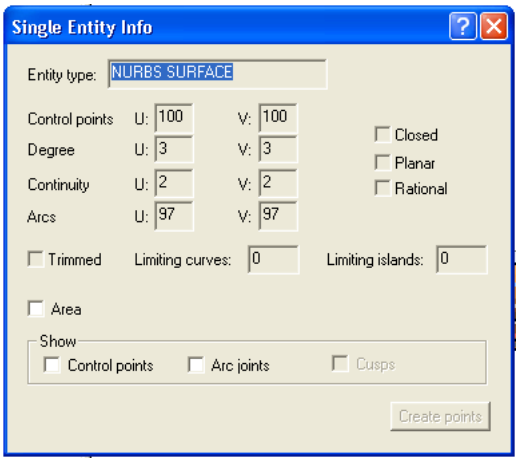
- Start the **Convert into NURBS** command.

There are three Fix modes to fit surfaces: tolerance, parameters or advanced. Let's start with Fix tolerance.

- Set Fix tolerance.

- Select the hull surface as the Surfaces.
- Let's start with the default values of 0.001 for the Tolerance and Max. U and Max. V to 100 for the maximum number of control points in both U and V directions.
- Press Preview.
- Select Warning to verify the Achieved accuracy in the approximation (0.01 mm).
- Hit OK.

Check the surface with **Single Entity Info** and you can see that the surface now has 100 control points in both U and V directions.



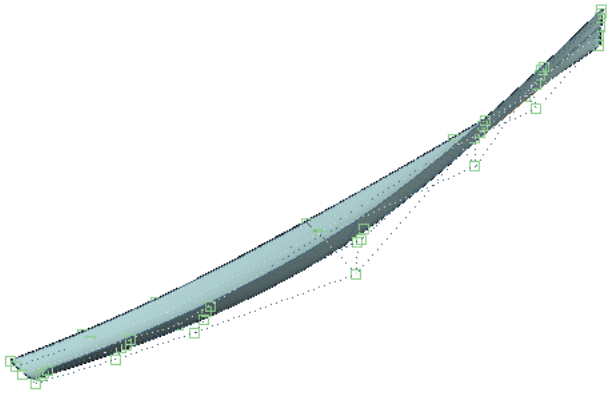
But 100 control points and 97 arcs in each direction is still pretty heavy. Let's see if we can do better.

- **Undo** the first conversion, then start **Convert into NURBS** again and select the hull surface.
- We're working in millimeters on a 20 Meter surface, so increase the Tolerance to 0.2. Keep Max. U and Max. V to 100.
- Press Preview.
- Warning will not present now. It means that the approximation has been done with 0.2 mm tolerance; so the original and computed surfaces, at most, have a gap of about 0.2 mm.
- Hit OK.

Now let's look at the 'Parameters' mode. With this option, you can control the number of arcs, the degree of each arc and the continuity for both U & V directions. You also get immediate feedback on the achieved precision for the settings you choose.

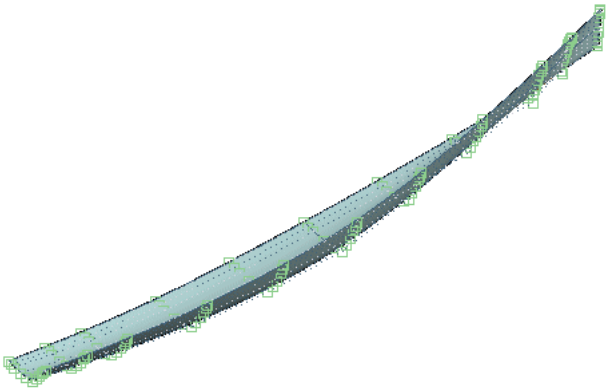
- **Undo** the last conversion.
- Start **Convert into NURBS** again.
- Set the Fix to 'Parameters'.
- Select the hull surface.
- Press Preview.

- Check the Show Control Points.



Look at the Achieved tolerance, now it is at 26,33 mm. Our goal is to reduce this error..

- Set Arcs U and Arcs V to 4.
- Press Preview..
- Check the Control Points.



That's a thin net of control points. Let's try increasing the control points in the U direction first.

The precision isn't much better, but let's take a look at the control points.

That's better, but still not good enough. We'll increase the V parameters this time.

- Set Deg. U and Deg. V to 4.
- Set Arcs U and Arcs V to 8.
- Press Preview..

That looks lot better. Look at the Achieved tolerance, now it is at 1,30 mm. Let's take another look at the control points.

That's a nice, regular net of control points and precision that we can live with. Remember that the changes are not applied until you hit Apply or OK.

Now you know the basic mechanics of surface fitting.

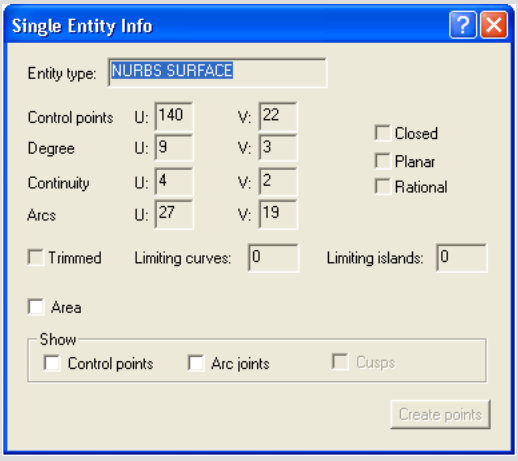
- We won't use this surface, so you can **Delete** it before moving on to the next step.

Note: Intrisic or ..

In the Bi-rail surface that you generated before, you found lot control points, close to 800.Let's try to generate the surface again.

- Start **Lofted Surface**.
- Select the curves as in the last to make Grid type.
- Under More Options, Surface Type Options set Parameterization as Curvilinear mode.
- Hit OK.

Now, using **Single Entity Info**, you'll find different data with less control points.

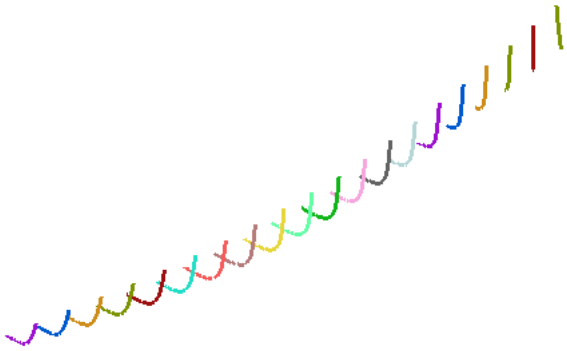


Using this method, thinkdesign tries to approximate automatically the selected curves second to their length.

2. Step 2: Another Approach

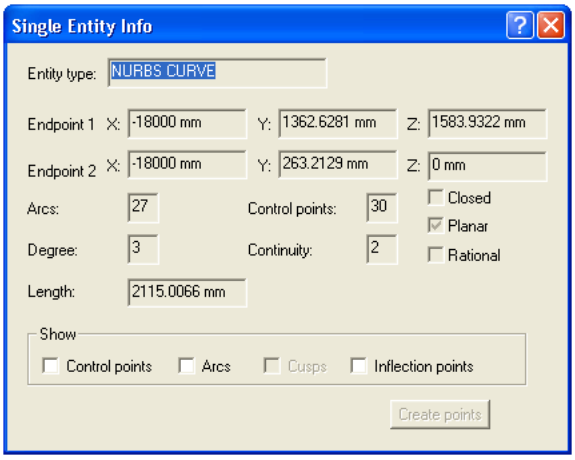
Now that we know we can fit a heavy surface to reduce the number of control points, let's look at a different approach. In this step, we'll refit the original curves before we make the surface. This approach also gives us a lighter surface.

To understand why the surface was so heavy, we need to look at the original curves. We'll use the **Single Entity Info** command to examine a few of the curves.



- Right click on the first curve and start the **Single Entity Info** command.

We can see that this curve has 27 control points with 30 arcs at a degree of 3 and a continuity of 2



Now let's look at the last curve.

- With the command still active, select the last curve. Make sure you select the whole curve, not a snap point.
- This curve has different control points and arcs but keep same degree and the continuity. This strange behaviour is for all curves.

- Select the green curve (color 10), the tenth curve from the bow (front).

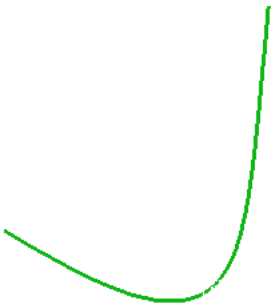
We see that this curve has 34 control points with 31 arcs at a degree of 3 and a continuity of 2.

When we made the **Lofted Surface** surface Grid mode, the system had to calculate the correct number of control points based on the differing number of control points in each curve. Whenever possible, it's better to create surfaces from curves that are relatives, meaning they have the same number of control points and arcs.

To achieve this, let's try fitting the curves prior to creating the surface. We'll start by fitting a single curve so we can look at each option in detail.

- **Hide Entities** all the curves except the green curve (color 10), the tenth curve from the bow (front).
- Change the current Color and the line Width to 3.

- **Fit View** the remaining curve.



Now we'll use the Curve fitting option of the **Fit Curve** command to refit this curve.

- Start the **Fit Curve** command.
- At the prompt, select the green curve.

Once the curve is selected, you are prompted to enter the number of points to fit. This number determines the number of points thinkdesign will use when evaluating the original curve. These are not actual control points, but intermediate points along the curve that thinkdesign uses to fit the new curve.

Note: Number of points to fit.

Remember this rule of thumb: if you have a smooth, heavy curve to fit, use a higher number of fitting points to make the resulting curve close to the original. If you have a curve with fewer control points and sharp edges and/or inflection points, use a lower number to try to smooth the curve. The length of this curve is over 3 meters (3000 mm), so we'll use the maximum number of points, which is 300..

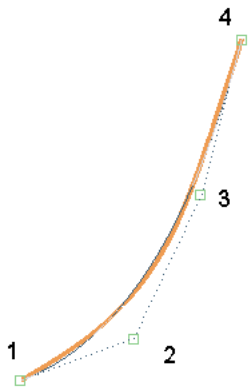
- For the No. of points to fit, enter the maximum value of 300.

The next prompt asks for how many arcs you want for the curve. The higher the number of arcs, the closer we get to the original, with little approximation. The default value is the number of arcs in the original curve. This is an iterative process, so we'll start low.

- For the No. of arcs, enter 1.

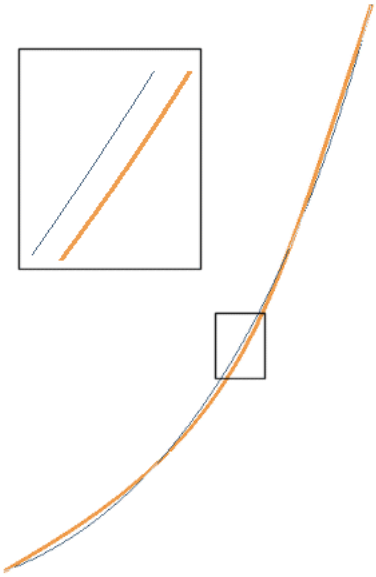
Now we're prompted for the degree and continuity for the new curve. The degree is one more than the number of control points, so a Degree 3 curve has four control points. We'll start low with this one, too.

Pick ⊞ Quality Checks and check ☑ control points.



We need a very smooth, continuous curve, so we'll use a continuity value of 2.

- Set the Degree to 3.
- Set the Continuity to 2.



- Expand Quality Checks and check for the values of Maximum distance, about 44 mm, and Mean square error

With these settings, we don't get very close to the original curve. You can see the new curve.

- Increase the No. of arcs number to 4.
- Set the value of Degree and Continuity to 4 and 2 respectively.

- Expand Quality Checks and check for the values of Maximum distance, about 0,5 mm, and Mean square error.
- Expand More Options and check Hide Source Curve so don't overlap original and new curve.
- Hit OK.

This looks good, very close to the original curve. Do not Undo now.

Now let's take a look at the NURBS formula. This is a simplified version you can use to calculate the number of control points of a curve.

NP=(deg-con)*na+con+1

Where:

NP = Number of control points

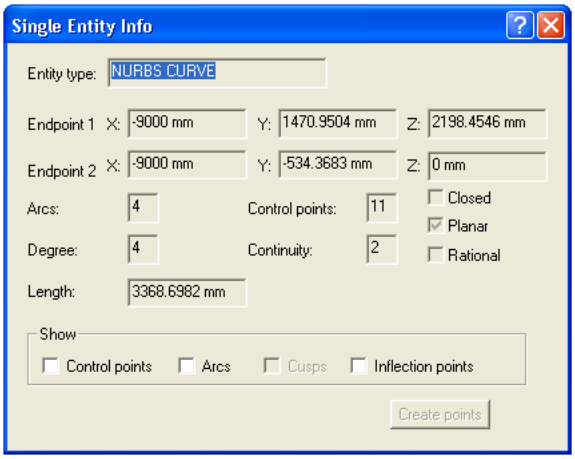
deg = Degree

con = Continuity

NA = Number of arcs

Let's use **Single Entity Info** to check our work.

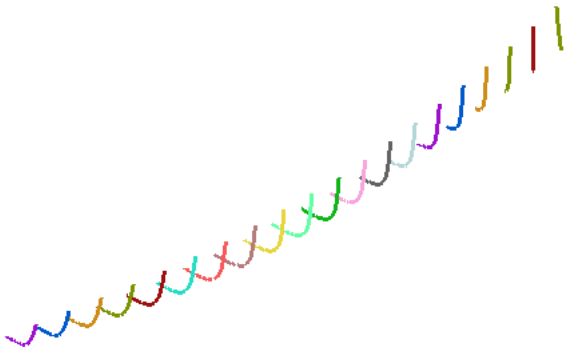
- Right click on the new curve(red) and start the **Single Entity Info** command.



So, you can see from the formula that increasing the degree or the number of arcs increases the number of control points. This can be helpful when you are trying to decide what values to use when fitting curves or surfaces.

We can also use the **Fit Curve** command to refit multiple curves, making the curves relatives. Before we start, let's get rid of the curve we just created and make the rest visible.

- Use **Undo** to remove the curve we just created.
- **Unhide Entities** the rest of the right-half curves



Now we'll use the **Fit Curve** command again to fit all the curves at once. We'll use the same settings we used for the single curve.

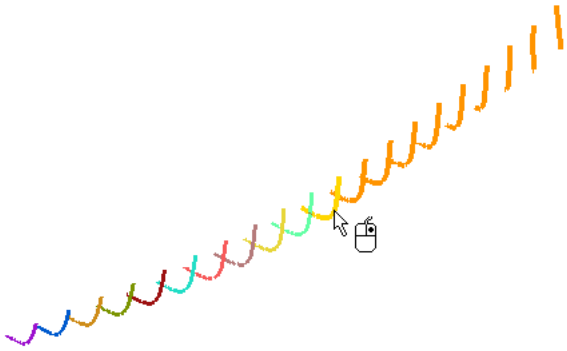
- Start the **Fit Curve** command, hit the Curve button and select all of the curves.
- Change No. of points to fit 300, No. of arcs 4, Degree 4 and Continuity 2.
- With these settings, we get about a Maximum distance of 2.35 mm and Mean square error of 0.04. Not bad for a 20 Meter boat.
- Expand More Options and check Hide Source Curve so don't overlap new curves.
- Hit OK to end the command.

With our approximated curves, let's make another surface.

- Turn off the hull sections (Layer 100) by **Output Layers**.

- Change the Color to 1 and the Width to 1.

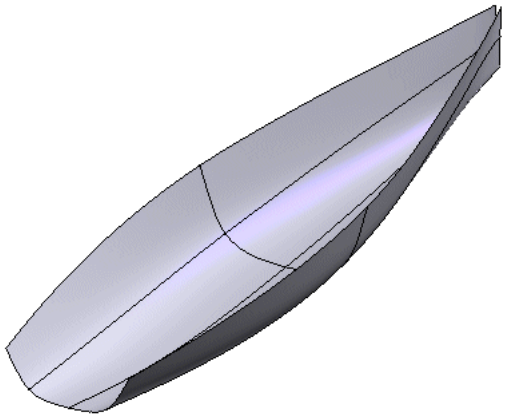
- Start the **Grid Lofted Surface** command.
- Still under More Options but in Surface Type Options set Parameterization as Intrinsic mode.



- Select all new curves.
- Hit OK.

Finally, let's mirror the surface to get the other half of the hull.

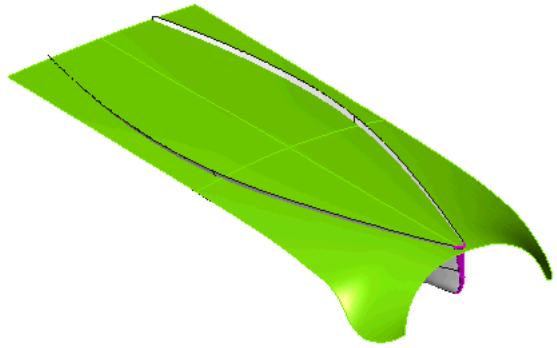
- Select the surface and start **Mirror Entities**.
- Set the Symmetry plane as Perpendicular to Z Axis in the workplane 0,0,0 origin.
- Set the Point to **Work Plane Origin**.
- Check Copy.
- Hit OK.



Before we move on hide all the curves using the **Hide Entities** command.

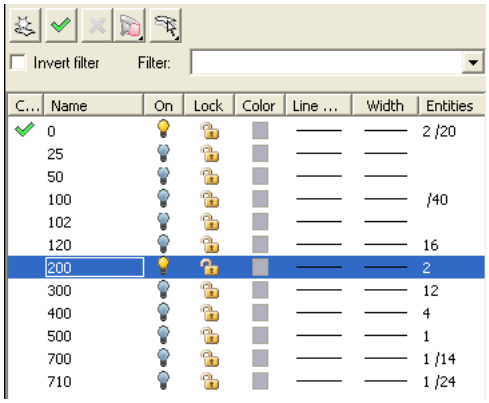
3. Step 3: Time to Have Some Fun

In this step, we'll get a little creative with the bow and add the deck.



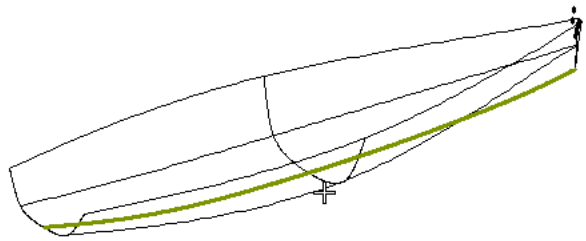
We have some starter curves for the bow on a Layer, so let's turn it on now.

- Turn on the ideal bow profile (Layer 200).



As you can see, we don't have the complete bow profile, just some starter curves. We'll use **Connect Curve** to finish the curves, but first we need a **Boundary Curve** curve to connect.

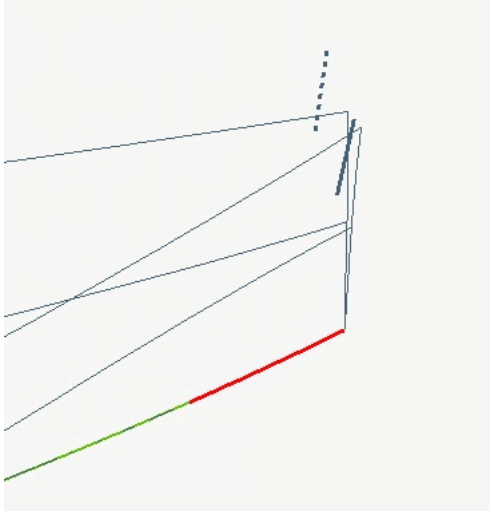
- Set the Color to 3.
- Start the **Boundary Curve** command.
- Pick Select option under Boundaries.
- Select one in the bottom of the hull surfaces.
- Pick near the bottom edge to create the boundary.
- OK.



Let's trim the curve back to the **Work Plane Origin** with **Split Curve** and **Delete**.

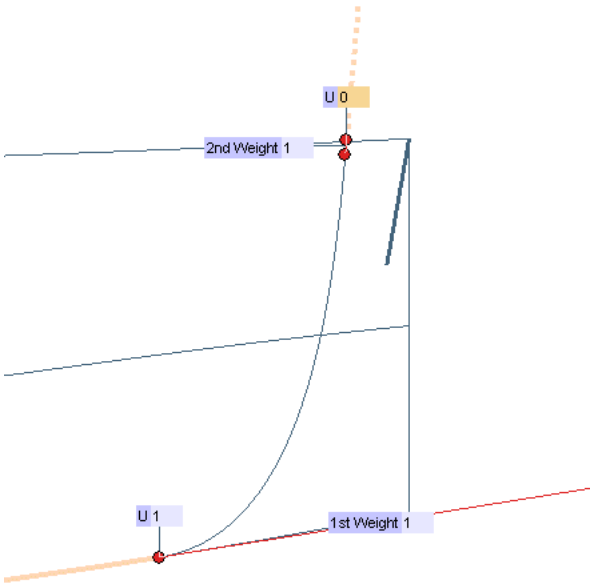
- Start the **Split Curve** command.
- Select the curve and say continue.
- For the point at which split the curve prompt, select the **Work Plane Origin**.
- OK.

Delete the right side curve as shown.



Now we'll create the first **Connect Curve** curve between the dashed line and the boundary curve. We'll use this curve to trim the hull surfaces.

- Start **Connect Curve**.
- Select the lower portion of the dashed line.
- Select the green boundary curve where splitted.
- Leave the Weights of both curves at 1.



- Hit OK.

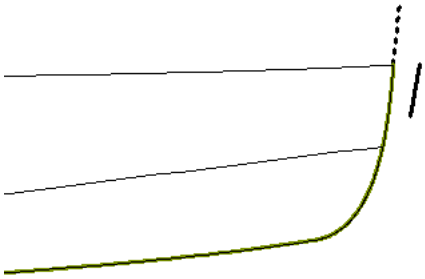
Note: Connect Curve.

Snapping the end points, if you repeat the last step, the command capture automatically the parametric position (0 or 1) and all possible degree of continuity with the selected curve.

It is possible to select the boundary of the surface directly without having to create them before.

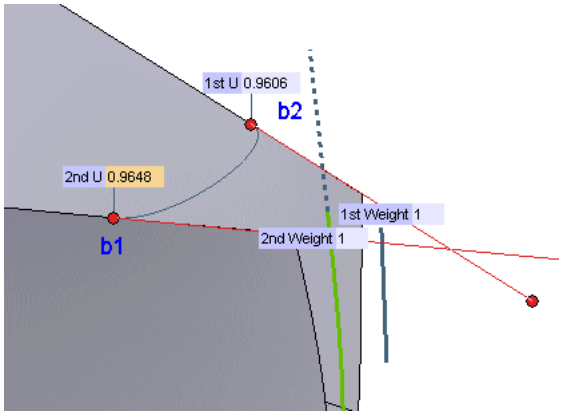
Now let's trim the hull surfaces back to this new curve with Trim Surface with limits. Note that the view is important when trimming surfaces with curves that do not lie on the surface.

- Change to a Work Plane View (hit F8) and **Zoom Window** on the bow.
- Start **Modify**›**Surface**›**Trim with Limits**.
- Select the connect curve as Limits.
- Select the hull surfaces as Surfaces with a windows selection from right to left.
- Select the region to keep, using again the windows but picking the left portions of the hull surfaces.
- Hit OK



That flat bow won't be very fast since it's flat in the front, so we'll need to create a rounded surface there. Let's make another **Connect Curve** curve with the other line to get an idea of the type of rounded surface we are looking to create for our bow. .

- Start again **Connect Curve**.
- Select both top hull boundaries.
- Change the Weights of both curves at 1.
- Hit OK



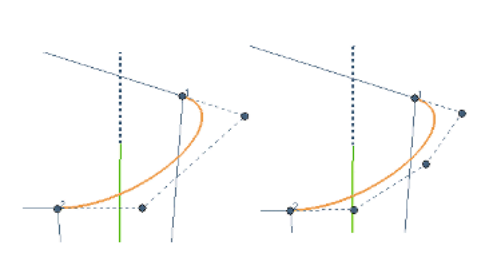
Note: Control points and flexibility.

That's a start, but we need to add a little more flexibility to the connect curve by adding one more control point to it using the **Curve Flexibility** command.

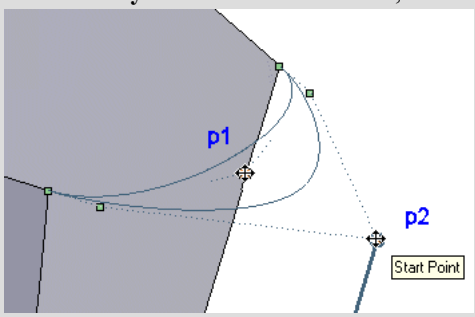
- Start the **Curve Flexibility** command.
- Select the top hull connect curve we just created.
- Change the Degree to 4.
- hit OK.

A rapid mode, using **Curve Flexibility**, to increase the number of control points on curves.

Other mode with **Modify Curve Control Points** picking one time on Increase degree.



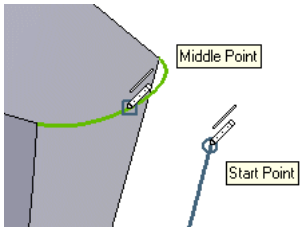
Start **Modify Curve Control Points**, select middle control point and move from p1 to p2.



This isn't the better mode so that the curve come to p2.

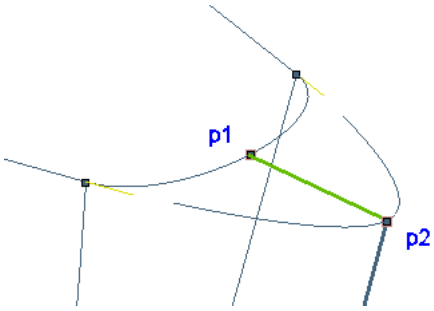
Delete the curve and create it again with **Connect Curve** as in the previous step. Let's use other mode.

- Go to **Two-point Line** command.
- Select middle and end points as in below image.



Our goal is to change the shape of the connect curve but without to alter the tangent direction.

- Start the **Modify Curve Interpolation Points** command.
- Select Add Interpolation Points as mode
- Select the end point of the new line (same to the middle point of the last connect curve).
- A new point will be shown with same graphic ot the end points. Select it again and move from p1 to p2.
- Hit OK.



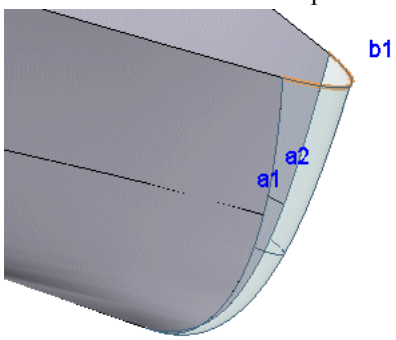
Note: Alternative - Global Shape Modeling.

Without using the last two commands (**Curve Flexibility** and **Through Point Curve**), you could use the **Advanced Modeling** with Point on Curves and achieve the same result, especially if we use an associative modeling.

With the curve modified, we need to create the rounded bow surface. we'll create one using the **Lofted Surface** in the Proportional mode.

- Change the Width to 1 and the Color to 6
- Start the **Lofted Surface** Surface command.
- Select the front edges of the two hull surfaces (a1 and a2) as the Boundary Set A in the Selection list.
- Select the top hull connect curve (b1)we just edited as the Boundary Set B.
- Pick More Options and as Type choose Proportional.
- Hit OK.

If the preview will be curled, pick More Options and Surface Type Options. Checking Manual Ordering/ Orientation there will be the possibility to correct wrong orientation of sections.

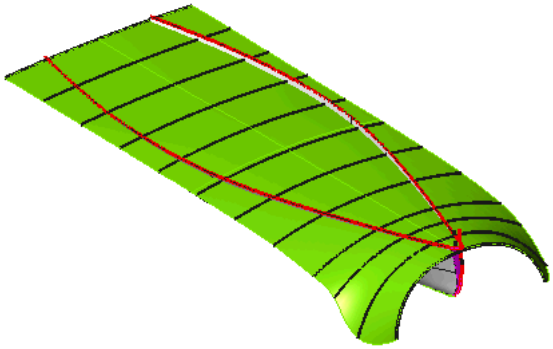


Now let's create the deck. We've got some curves for the deck on **Output Layers**, so let's turn them on and create an Grid surface.

- Turn on the deck curves on **Output Layers** (Layer 300).
- Set the Color to 3.
- Start **Lofted Surface**.

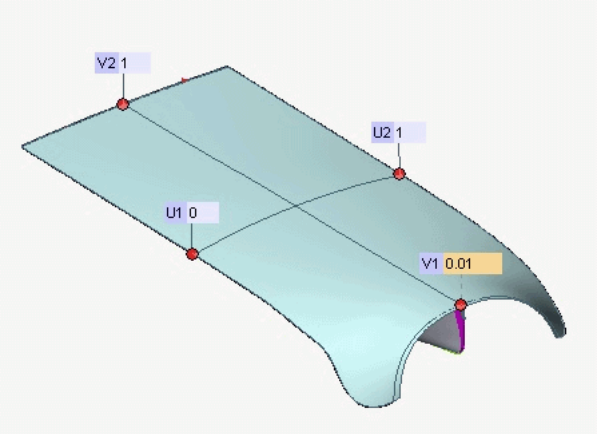
- Select the curves of the deck.

- Hit OK.



It looks like we need to extend the surface a little at the bow. We'll use the Parameter mode of **Trim/Extend Surface** for that.

- Start **Trim/Extend Surface** and Set the Mode to Parameter .
- Select the bow boundary on the deck surface.
- Set the v1 parameter to -0.0.1.
- Hit OK.

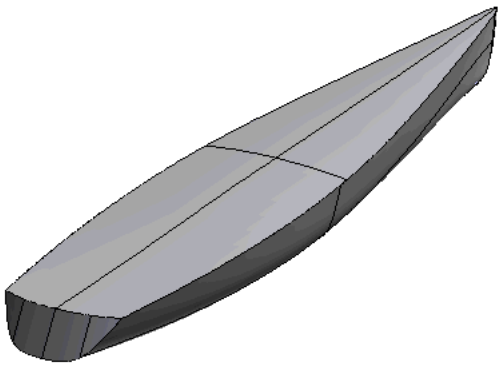


We're really getting somewhere now. In the next step, we'll make it solid and do some work on the keel. Before we move on, let's clean up.

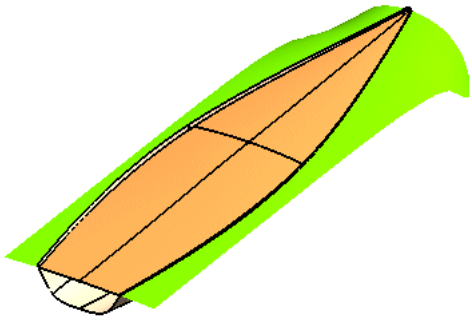
- Turn off the deck curves (Layer 300).
- Turn off the ideal bow profile (Layer 200).
- **Hide Entities** the rest of the curves, leaving just the surfaces.

4. Step 4: Make it Float

Our racing boat is taking shape, but it still needs some work before it will float. In this step, we'll make the surfaces solid and use Booleans to create a solid closed hull.



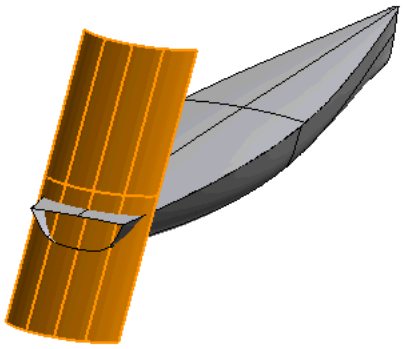
- Start **Modify**➪**Surface**➪**Trim with Limits**.
- Select the three hull surfaces as Limits.
- Select the deck surface as Surfaces.
- Select the internal deck area to keep.
- Hit OK



- Start Trim Surface with Limits.
- Select the deck trimmed surfaces as Limits.
- Select the three hull surfaces as Surfaces.
- Select the internal area for each surfaces to keep.
- Hit OK

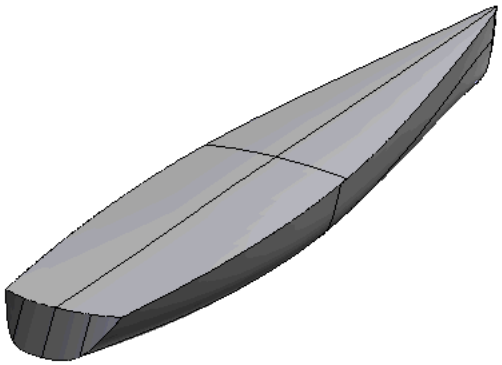
The boat won't float with that big hole in the stern, so we better close it up. There's a surface for the stern on a **Output Layers**; let's turn it on and make it solid.

- Turn on the stern surface **Output Layers** (500).
- Start again **Modify**➪**Surface**➪**Trim with Limits**and repeat the double trim.



We'll start by using the **Make Solid** command to obtain a single solid hull.

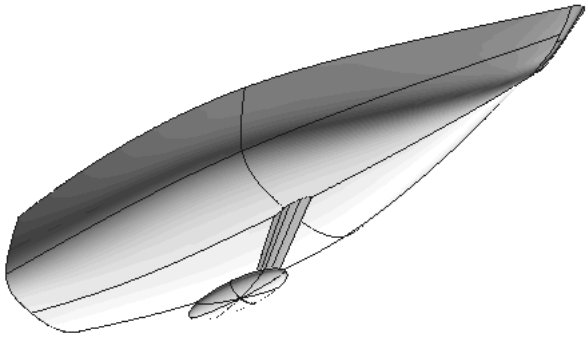
- Change the Color to 1.
- Start **Make Solid** and select all four boat surfaces.
- Hit OK.



Looking good! In the next step, we'll add the keel.

5. Step 5: Add the Keel

In this step, we'll use some existing curves and profiles to create the keel. We'll use a **Generic Protrusion**, then we'll create surfaces using **Rotational Surface**, deform them by stretching and **Solid Union** them all together.



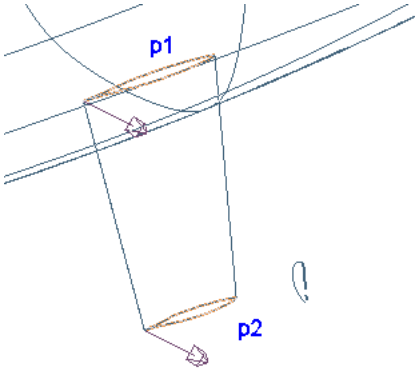
The keel curves and profiles are already on a **Output Layers**, so let's take a look.

- Turn on the keel curves (Layer 400).

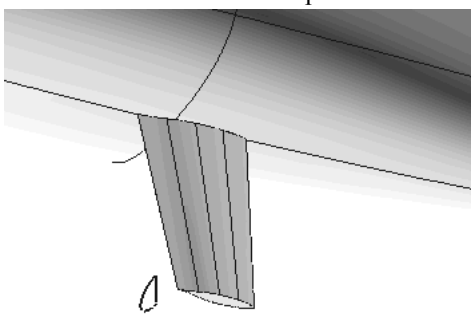
We'll use the two green, tear-shaped profiles to create a **Generic Protrusion**

- Start the **Generic Protrusion** command.
- Select the two tear-shaped profiles as the Boundaries in the Selection list, making sure to select each at a similar point.
- Make sure the Blending is set to Ruled in the Selection area.

If the spine preview does not connect on ../Common points on the profiles, as shown by the arrows, use Start points in the selection list and pick the points to align it and to correct it.

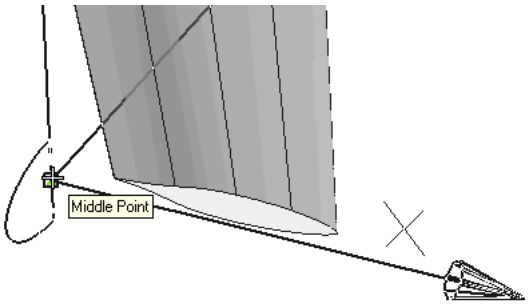


- Select the hull as the Solid in the Selection list
- Hit OK to create the protrusion.



Now we'll work on the base. For starters, we need to move the Work Plane to the curves.

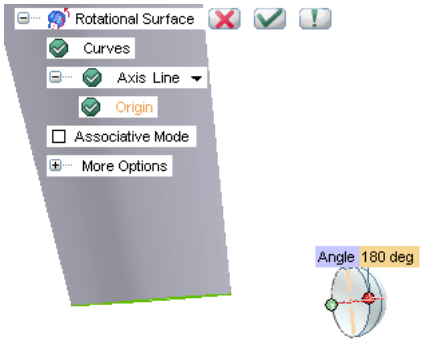
- Right click on the Work Plane and select Edit from the context menu.
- Pick Transformation in the Selection List and select the **Mid Point Snap** of the vertical reference line as Origin.
- Apply the Work Plane command.



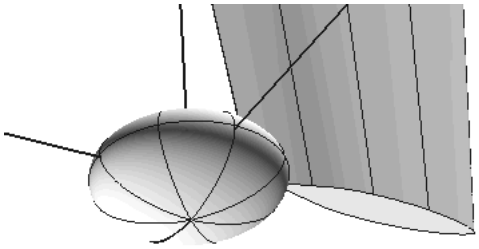
- By By Quick Edit Work Plane, rotate the Work Plane of the -90 degree around the Y axis.
- Press ESC to confirm.

To finish up the keel, let's create two 180° **Rotational Surface** surfaces from the same curve.

- Start **Rotational Surface** command and select the elliptical curve as the Curves in the Selection list.
- Set the Axis to Line and select the vertical reference line as the Line in the Selection list.



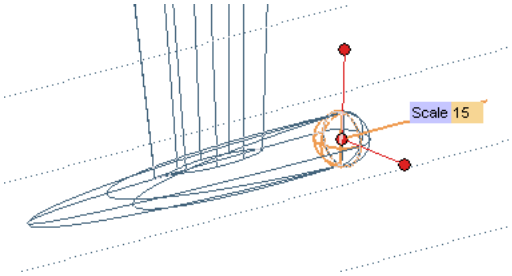
- Set the Angle180 and Apply.
- Select the same Curves and Axis again, change the Angle-180 and hit OK to create the second mirrored surface.



Now we'll Stretch them with **Scale Entities**.

- Select the rotational surface, the one towards the stern, and start **Scale Entities**.

- Right click on Handle Origin say Reset.
- Select the **Work Plane Origin** for the handle Origin point.
- Set the Scale: to 15 along Z direction and say OK.



Now let's make a solid from these two surfaces and **Solid Union** the two keel solids to the hull.

- Start **Make Solid** and select the two surfaces.
- Start **Solid Union** and select all the solids to make a single solid.

Nice work. In the last step, we'll create the cabin.

6. Step 6: Create the Cabin

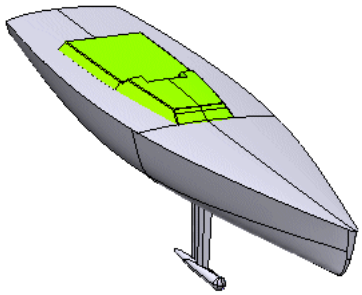
This exercise is running a little long. So, we've created a bit on the last step and supplied two solids to complete the cabin.

Start by turning on the two **Output Layers** with the cabin solids.

- Turn on Layer 700 and Layer 710.

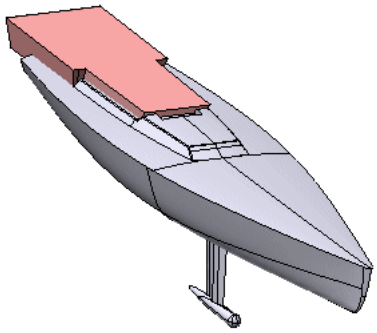
Let's use Boolean Operation to create the complete sailboat.

- Start the **Solid Union** command.
- Select the main boat solid and the green cabin solid.



Continuing with boolean operation.

- Start the **Solid Difference** command.
- Select the main boat solid and the red cabin solid.



And there you have it. Great job!