We will continue on from where we left the **Boolean-cutting** shape or the **Umbilical shape**, getting to cut where you want was just part of the process, I covered it a little in the last chapter.

Now we need to alter the shape to fit to the model backdrop images better, as we have the shape split into 3 sections, we’re going to use those to make the adjustments, you will see what I mean in the following images.

While working on this and other meshes, I learned a valuable lesson and I will pass it on to you, at no extra charge, as we are going to be welding Points later on for this. So copy the Secondary Hull and place it in the next layer where we can work on it without worrying about making any mistakes.

This will make your life easier, the amount of times that I mistakenly selected a Point on a part of a model that I did not intend to touch and mistakenly welded it or worse deleted it, then only to find my error much later and its beyond fixing with the undo levels option.

You can set the undo in General Options via keyboard letter (o) check your user manual for more information.

I think it uses a lot of RAM so be-careful how much you enter.
Right now select the Aft Portion (indicated) of the Secondary Hull, the copied one not the original which should be in the previous layer (Remember I said Remember about this), Now delete these selected polygons.

I am going to take this forward some more and delete half of this object on the X-axis so we only have to work on one side.

If you don’t have the copied one, create one before going on with the Boolean-cut operation, or at the very least the forward section shown above in the image. It could save you some pain if you make a mistake.
On the forward disc select the top and bottom Points and create a line between each using the Construct Tab and the Combine Tool and Connect you can use the keyboard shortcut letter (L). You see the Connected Points in the image above. This is what we are working with now, a smaller object and smaller chance of messing it up (unless you mistakenly moved it which I have done in the past).

Now we go to the 3 Docking Umbilical’s.

One of the problems I had with this is are the Docking Umbilical straight and level or do they have a slight angle following the lateral curve of the model. After some thought I decided to go with the angle approach, covered in the last chapter.

As we spit the docking umbilical shape in to 3 sections as mentioned before this is all we need to do

Number. 1. Needs to stay in its location near the hull.
Number. 2. Will be slightly angled out so that section 4 will arrive into its proper position.
Number. 3. This line need to the vertical line you see in the backdrop image.
Number. 4. Is where the line is now according to the front backdrop image and need to be adjusted out to 5.
Number. 5. Is where we what section 4 to be.

As we are going to use the rotate tool this is what your rounder tool will look like, the Cross-Heir appear on light or white backgrounds. I created this icon as the screen-shots I take for the tutorial fail to show these things.
First we move the 3. line back slightly too where you see the backdrop image white hull line above. Its does not have to be exactly the same position, its just to move this away from the forward area of the hull. where it curves around to meet the Dish Housing.

Place your cursor over the Points indicated in the image above and use the Bottom View-port for this, press the Right Mouse Button to set the coordinates use the Cross-heirs on the Rotation Tool to help set position. Now enter 1.0º into the Angle field in the Y-axis and Apply.

All the selected Points will move out 0.7º to the right or in the bottom view to the left.

Now deselect the 2 rear Points and keep the forward ones, again click the cursor over the Points indicated, change the angle to: -1.0º,(minus to reverse the rotation direction to bring it back in line) on the Y-axis again.

You will see the during this process (if your viewing in the quad view) the remaining Segment move back into position slightly.

Now we are going to Extrude this shape out so we can use it to Boolean-cut the Secondary Hull and add the 3 box shape.
Now with the surface in **Polygon Mode** and then **Flip (f)** the surface.

You cannot really see the angle here, you might if you **Zoom** in a little.

This is how your surface should look before **Extruding**.

Now select **Extrude**, or **Shift +e** then press **(n)** for the **Numeric Tool** and enter **(6 +)** in the **X field**.
Check to see if the rear curved section fits within the shape of the Secondary Hull encase it is out of position.

We want all the Umbilical Object to cut into the hull.

Now too add the other 2 Docking Umbilical’s, use the Array Tool found under Multiply and Duplicate or the keyboard short-cut is Ctrl + Y. Select the Array Type: Radial enter 4 in to the Number field in the Z-axis and OK it.

We then have 3 new object around the Secondary Hull, select the Top and Left Side Umbilical’s and delete them.
I decided to add a new cut to the hull use the BandSaw Pro and adjust as you see above.

The reason for this is too reduce surface smoothing errors that could later cause use some difficulty’s after some additional refinements to the Boolean-cut.

But it does not end here as I will now add a few more cuts in this area using the BandSaw Pro Tool again its to reduce smoothing errors that will appear later if this large area remains as it is.

I placed the Boolean cutting shape in the background to use as a guide for placing each line cut so they rest over selected Points on the guide, the red arrows indicate where the Points are on my umbilical shape in the background, you can easily adjust the positions in each line by changing the numbers in the value area, closer to the background Points the better.

I added 5 more cuts with BandSaw Pro Tool in this area.

I am adding one more Cut intersecting the line on the background shape.
To use the Boolean-cut keep the Secondary Hull in the Main Layer and the Cutting shape (black wire-frame) in the background layer.

Go to the Construct in Top Bar Tabs, then to the Side Bar Tools under Combine, you will find Boolean. The keyboard shortcut for this is shift + B, select Subtract then press OK.

This is the result. Check to see if the Rear Curve’s are OK and Cut Properly before going on, making sure that the complete curve was cut out.
Then press keyboard (m) to **Merge the Points**. Where I eliminated **211 Points** from mine, it does not mean you’ll get the same results or then again you might.

We have some more work to do on this and to make things a little easier, we are going to remove the other side (The **non-Boolean-ed** side), there will be smaller chance of making mistakes.

Once the other side is select and use the **Set-Value (v)** with the **Axis** set to **X** and the value at **0**. you will see the selected side squash up against the Center line of the hull or the X-axis.

I always find it easier to work on one side if the model is designed that way, like the Enterprise, The Millennium Falcon from Star Wars for instance is not so easy design wise as it has a lot of random surfaces and a lot more thought is needed.
Once the surface is flattened you can delete it while it’s is still selected, we now have a smaller area to work on, check to ensure you have deleted only half the object, check around the lower cut.

Now we go to the rear curve of the cut, if you have the area in the **Red Box** is fine, but we need to move the curve further back into position.

To correct this, 1st select the Point indicated by the **Yellow Arrow** and then 2nd select the Point indicated by the **Red Arrow**, then proceed to **Weld** them, press `Ctrl + w` and OK it.

The **Weld Tool** is found under the Details in the Top Bar Tabs and in the Side Bar Tools under Points.
The **Welded Points**, now continue the process around too the narrowest area as indicated above.

Here I **Weld** the lower segment line Point to the Boolean-cut Point, to keep the curve smooth.

Then I continued to **Weld** the **Points** as before.
Continue around until the narrow area of the cut starts to widen out again.

Then repeat on the lower area of this cut before starting on the Bottom cut, Yellow Arrow is where I started and the Red Arrow is where I stopped as the cut start to widen, Green Arrows shows where I Welded the Segment Point to the Boolean-cut Point.

You can see in the cut area some smoothing problems.

Press (w) to open Statistics and select the Plus + symbol beside the 2-vertices which will be then highlighted and delete.

Like so.
Now that is done work on the lower Boolean-cut.

Carrying on with the work on the lower Boolean-cut, because we used the Set-Value Tool to flatten and delete one side (Port-Side) of the Secondary Hull, you will find a number of Points connected on the central line of the bottom cut.

Select and delete them and select the Points carefully, leave the Point indicated by the red arrow.

Start Welding the Points too bring this area of the cut into line with the main body of the hull as you did with the side cut.
This is my result, we may get some smoothing errors in these area’s but we can fix them later.

Press w to open Statistics and select the 2 Vertices which will be highlighted and delete.

Now we have to add the forward block sections of the umbilical shapes.
Select the forward inner sections, copy and then re-paste it in the next layer.

We need to select the inner sections as they are at the same lever we the Secondary Hull Boolean-cuts we made.

Like so, now Flip (f) the objects.

It is now flipped and we now Extrude it.
I entered 11 in the Extend X field.

Now repeat the Array operation to place the forward blocks around the hull.

Once done select the Top and Side block and delete them.
I copied the hull and pasted it a new layer, I also **mirrored** the hull.

Again I don’t want to ruin something I am happy with so I use the Layers to my full advantage.

Now to add these blocks to the hull, time to **Boolean** again, this time select **Union**.

Result, we have the 2 blocks attached to the main object.
Now as before after Boolean, **Merge (m)** the Points, my action removed 144 Points.

Now select the mirrored side and we will used the **Set-Value (v)** option.

We will also find some random **artifacts** we need to fix.

As before delete the flattened surface, you will/may also see one or more Points on the deleted side, delete them and note that there could be another on the remaining side.
Now select the Polygon indicated above and delete them.

Include this surface and delete.

Now select the Points moving around the block, select the Points in sequence and press (p).
A new surface will be created.

Repeat on the bottom block and again select the Points carefully.

Press (p) to create a new surface again, Flip if necessary. (this would depend on the direction you selected the Points).
You may see these Artifacts left over from when we flattened the Non Boolean surface, if so delete them.

I am going to work on this upper area of the Boolean-cut, you need to duplicate this on the lower and single bottom one.

This image should help, Photographer Unknown.
Select the large surface here and using BandSaw Pro tool add some cuts, you can see the cuts position as you work so adjust to line up with the vertical segment lines already on the hull.

1. set direction to either odd or even
2. set operation to add
   (or edit if you what to adjust individual cut lines which change color)
3. set value to adjust position to line up,
   you can type in number or scroll side to side.

Then I select these 2 Points and Connect (L) them and Connect them all on this upper area, not only there but also the lower and bottom area, also connect the others in the forward area before going on.
Connect these also, don’t use BandSaw Pro here as you will lose the setting for the large area, instead use the Cut Tool or Shift + U on the keyboard to open.

Now you will have a Cut here, you will have to Weld the Yellow indicated Points to the Red indicated Points.

Again I will say there is no one way to do this, I have choose the route I have to show what I think is an interesting way to build this model. I highly recommend that you do some experimenting again in these areas with the tools and see what happens.
like so.

I also decided to connect this segment line also, again using the **Cut tool**.

Again connect the Points.

I like to clean up as many points and segment lines as possible, but this is not always practical for those who are working to an air time, to work in this as a profession, you will need to be fast (something I am not) and learn to texture.

Find a teacher or many teachers to learn all you can about modeling and CGI in general.
I have done the same on the bottom cut, as above connect the Points.

I decided for the bottom cut just to use the Knife.
Then I selected the Points and Welded them. (if you don’t weld them Rounder may not work as expected).

1 or 2 more cuts, with the Knife Tool, select the surface block above. 5 in all.

Cut across the selected surfaces lining up with the hull segments.
Now **Weld** these Points as needed.

The **Point** indicated with the yellow arrow is really doing nothing on the curve of the Boolean-cut. Option 1: weld the yellow arrow Point to the red arrow Point, option 2: create a new line.

I am going to use option 2: select the Knife and positioned it over the two Points on the Boolean-cut, get in as close as you can.
Then I started to **weld** the old segment line to the new one, you can leave them if you wish too, but it makes for a cleaner mesh and a few less polygons.

Continue around to the bottom **Boolean-cut** and **weld**.

Now in **Polygon Mode** open **Polygon Statistics (w)** and select the 2 **Vertices**, when they are highlighted as above delete them. 2 vertices will cause **Rounder Tool** to fail.
I also want to cut a line across these Points.

Use the Knife and cut across the hull and the Points, press (m) if they are close enough any new Points will be Merged, if you went over them with the Knife tool, great, nothing will be Merged. Check them anyway.
Now select the polygons above and use the Cut Tool, I edit-adjusted it to 0.15, you can type that in the White field. How you select the polygons will decide the what value to enter into the value field, if 0.15 does not work as expected type in 0.85 or whatever number works best for you.

I am doing this so it I decide to export it to another format the chance of surface errors will be reduced, I was told that some years ago and it has become habit. None the less I find this works best for me.

Weld the Yellow Point to the Red Point at the rear, and Weld the single yellow Point forward.

like so, also weld the 2 Points indicated (lower and bottom areas too) repeat below.
On the lower area I revered the direction of the cut to 0.85.

Or if you have to enter 0.15 into the value field, now weld the Points as described above. Now go and do the same cut operation on the bottom Boolean-cut.

As we are going to use the Rounder here again we need to get the block surface prepared for it or we will have some nuances errors on the surface.

So we are now going to use the Multishift tool, select the surface area you see above and beyond those Points that are connected to the main hull, indicated by the yellow arrow.
When you have selected the Polygons press the Multishift Tool then (n) for Numeric Tool and a new window opens, 1. Inset amount: enter 5" - 2. Inset: select Contour - 3. select group polygons – 4. then make it.

You will see the selected surface bevel in slightly.

I want to add a new line across these area selected, but not over the Point that is already there but just behind it.

When we use the Rounder the we will be selecting this single Point for the operation but as I tested it, it became flatted back towards the front of the cut and ruining the look of the curve which is a big no for me.
I cut just behind the original Point across the hull.

Now before you Round the edge of the Boolean-cut and the forward block, check the Points individually to ensure you don’t have more than one on each segment, it will distort your line and at worst cause it to give you a nasty error. I got the distortion not the error, a gentle reminder something is amiss.

Now I selected all the Point moving around the edge of the Boolean-cut and the forward block.
Take care to select only the Points at the edge the exception are the Points indicated.

Select the Rounder Tool and enter the setting above, you may need to alter them to suit your own model especially the Inset distance.
This is my result, note the cluster of Points at the rear and the Forward Block, the forward block I am going to leave as it is. But the clusters at the rear need to be sorted out.

Select these Points and Weld to the Point we added earlier indicated by the Red Arrow.

The Welded Points

Now do the lower section.
Like so.

Now we will work on the bottom Boolean-cut.

Before we work on the bottom Boolean-cut you should use the Mirror X, for the forward block will need the opposite side for Rounder Tool to work as we want it.

Once done check to see if these Points are in the correct position, Set-Value (v) to the X-axis.

Now merge the Points and see what you eliminate.
Select the Points at the bottom Boolean-cut and work your way around the block and taking care to select only the edge Points, note you can clearly see the left/port side of the model in the top left view-port.

Before you use the rounder check to be sure you have only the Point you want selected. Use one of the view-ports and the magnifying glass icon to zoom out and rotate around.

With the same setting as before and apply.
Select the Disarrayed Points and Weld to the Points indicated as we did before.

The result, now do the same on the other side.

You are almost done, but there is one more task for the Boolean-cuts and that is to connect the remaining blank area, it will help reduce any smoothing errors.

Start by Merging (shift + Z) these areas together (we will restore it later).

Merge (Shift + z), do it section by section.
Finish on this one, delete the 2 remaining Points, what I want to do is to connect the Points here indicated by the green arrows.

Select the opposing Points and Connect them, you can use the letter (L) on your keyboard.

Work as far back to where the Red Arrows are, the Green Arrows indicate where you might have some difficulty in selecting only one Point on each side, don’t worry.
If you do use the Polygon Statistics Tool (w) and select the 2 Vertices and delete them. They maybe so small you barely see them even highlighted.

Select the edited surface as above and use the Cut Tool (Shift + u) it using the default setting.

Weld the Point as described above.

In the 2 large blank areas which would be under the heading of non-planer we need to fix’s so start selecting 2 Point working out from the single center on and Connect (L) with the outer ones, working your way around.
If you open your Statistics in Polygon Mode and select Non-planer you will see the surface selected the easiest way to fix is to triangulate them, as we did on the left side image, you can of course use the Triple tool (shift + T) but we get varied results and not as precise as we would like.

Now do the same on the sides, you can if you choose delete once side and just work as you did before on the single side and mirror it again.

You can see here I have already delete one side and connected the Points as I did on the bottom, now I use the Cut Tool.

As before Weld the Yellow Point to the Red Point, and forward if necessary Connect (L) these Point (green) together. Your cut could appear different from mine (above) depending on how you select the surface, and check for the 2 Vertices.
As with the bottom Boolean-cut I connect the Points.

Last few steps:

Select the surface you see above, from the area we used the Multishift tool back to the curved area, also select in the direction indicated above.

Use the Cut Tool and in the value field adjust the setting to 0.95, or 0.05 and continue.
Now this is what you should have, where you see the dark area just **Weld the Yellow Points to the Red Points.**

We cannot go further until we clean up a surface Non-planer problem, open the statistics in Polygon Mode and select non-planer, there is only one we really need to deal with for now the others can wait. Deselect all the other non-planer apart from the one (really there is 2) indicated above and delete.

I will note that lightwave did crash on me for some reason at this Point, not sure why so remember to save save and save, it can save your hard work from been lost. I have the idea that it was because I had my Polygon statistics window open for a long period before doing anything.

You can see it better here, I have already deselected the other non-planer’s, using the mouse with the right button press to deselect them, once I see that only 2 are remaining, delete them.

Then use the cut tool again to fill these 2 remaining surfaces, welding the Points as before, then mirror X the object.
Once mirrored select the surface you see above.

Using the Multishift Shift Tool enter the same setting as before: Inset amount: 5”, and Make.

Same as the upper Boolean-cut, select the polygons working around and press the Cut Tool.

You should get this result. As before Weld the Points you think you need too.
One more cut, using the **Knife Tool**, across the **Sensor Dish Housing**, its for the same reason as all the work we have been doing on the Boolean-cuts, to help prevent smoothing errors, as the smoothing angle in the surface properties only goes so far.

This is what I have now.

I hope this part was easy to follow, I mentioned it once or twice that it took me some time to work things out, most models are never really finished just abandoned.

You will find this out as you progress with this as a hobby or a business if your work in the film or TV industry.

See you in the next chapter, where we will be looking at the Hangar and Fantail, it will have its twists and turns.

**Continued in next chapter.**

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