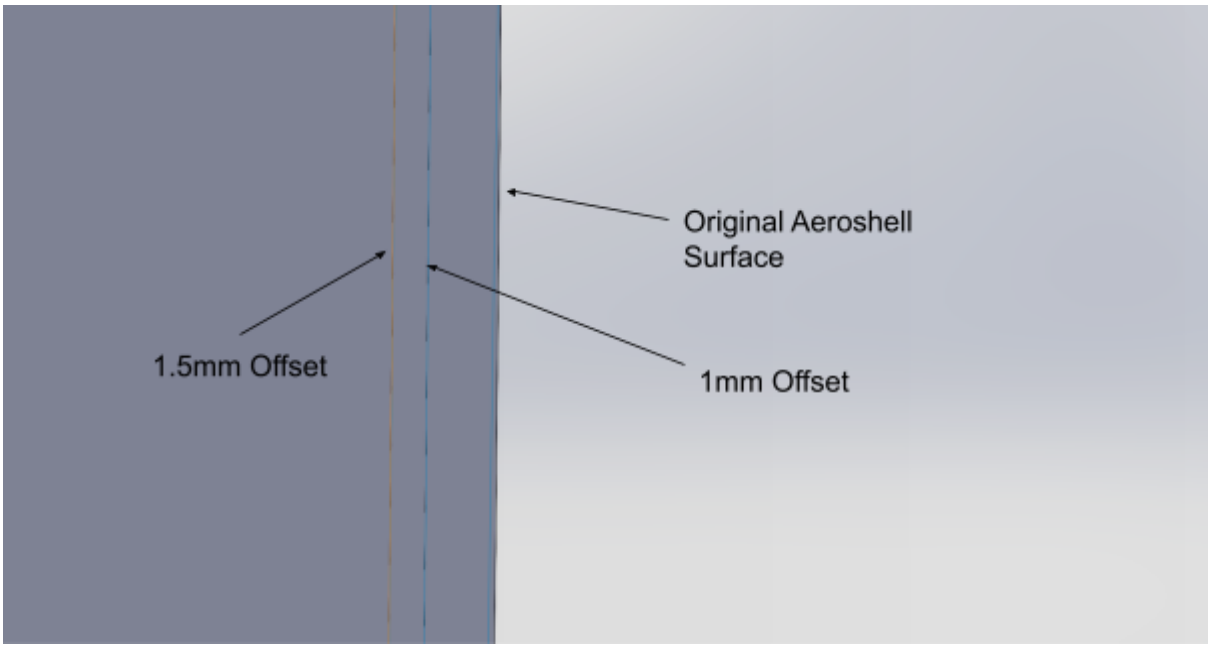
How to make the Bottomshell Plug

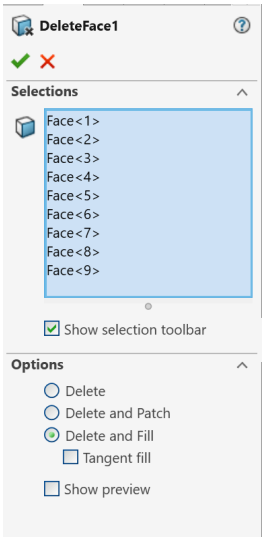
While this is a document detailing how to make a bottomshell plug, the best way to learn if you are confused is to roll-back the solidworks file and understand each step and how it was completed. Remember that Solidworks can be annoying so patience is also important.

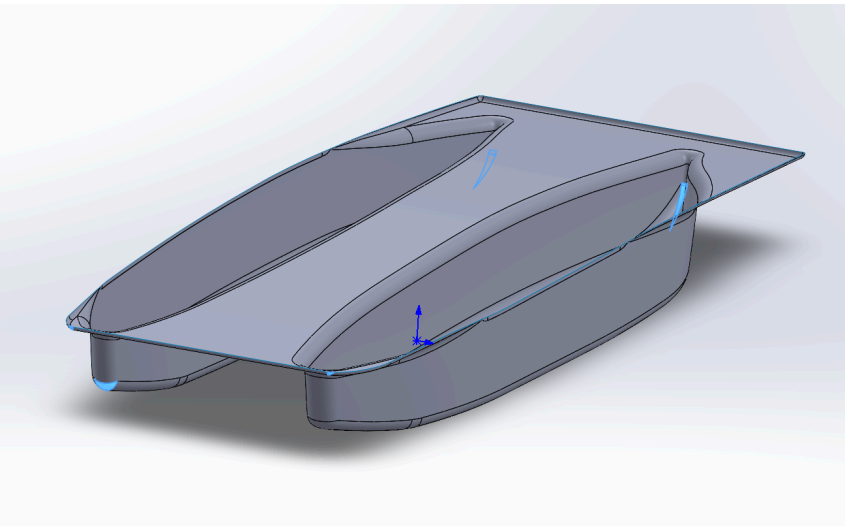
1. **Inner offset the bottomshell by 1mm and 1.5mm**
   This step is key in order to make our indents. The purpose of the indents is to outline the location of key features when making the mold. The offset gives the indent a back surface to lay on. You essentially will be cutting out the shape of a feature on the outermost surface and replacing it with an offset surface. For the bottomshell, the fairings and intake/exhaust holes use the 1.5mm offset, and all the lights (front, rear, side) use the 1mm offset.

   Using the surface offset feature, select the entire bottomshell. Offset it by 1mm first. Then hide that feature, and then repeat; selecting the surface offset feature, then selecting the entire bottomshell but offsetting by 1.5mm.



   If you are having issues with failing faces, you can use the delete face feature to help fix that. When you encounter failing faces, it is usually because there are small faces that intersect with the car when offsetting. Due to how small these faces are, we can delete these faces and patch them with minimal geometry change. Refer to the bottomshell plug issues tab in this document for further details.

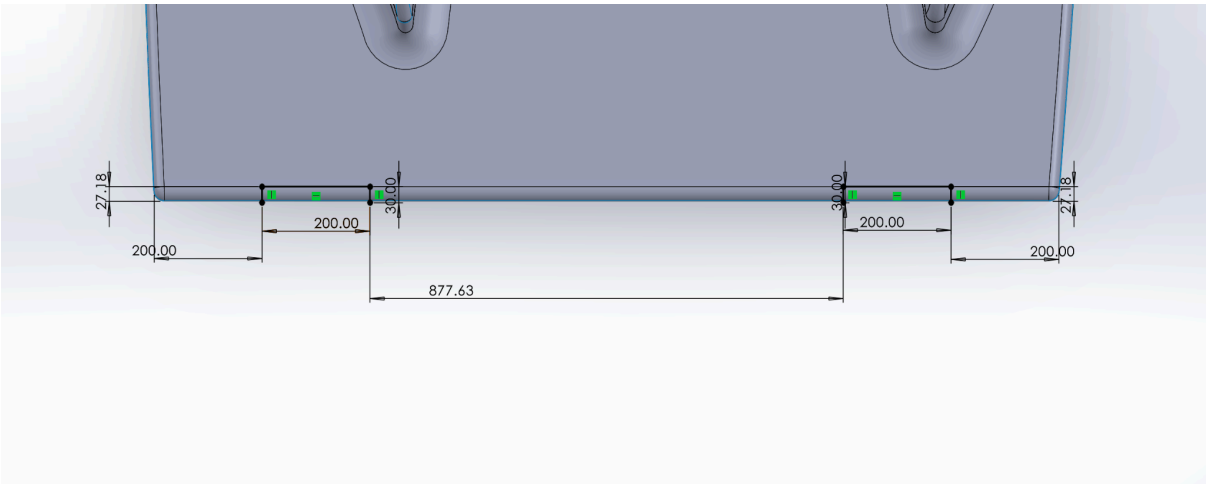




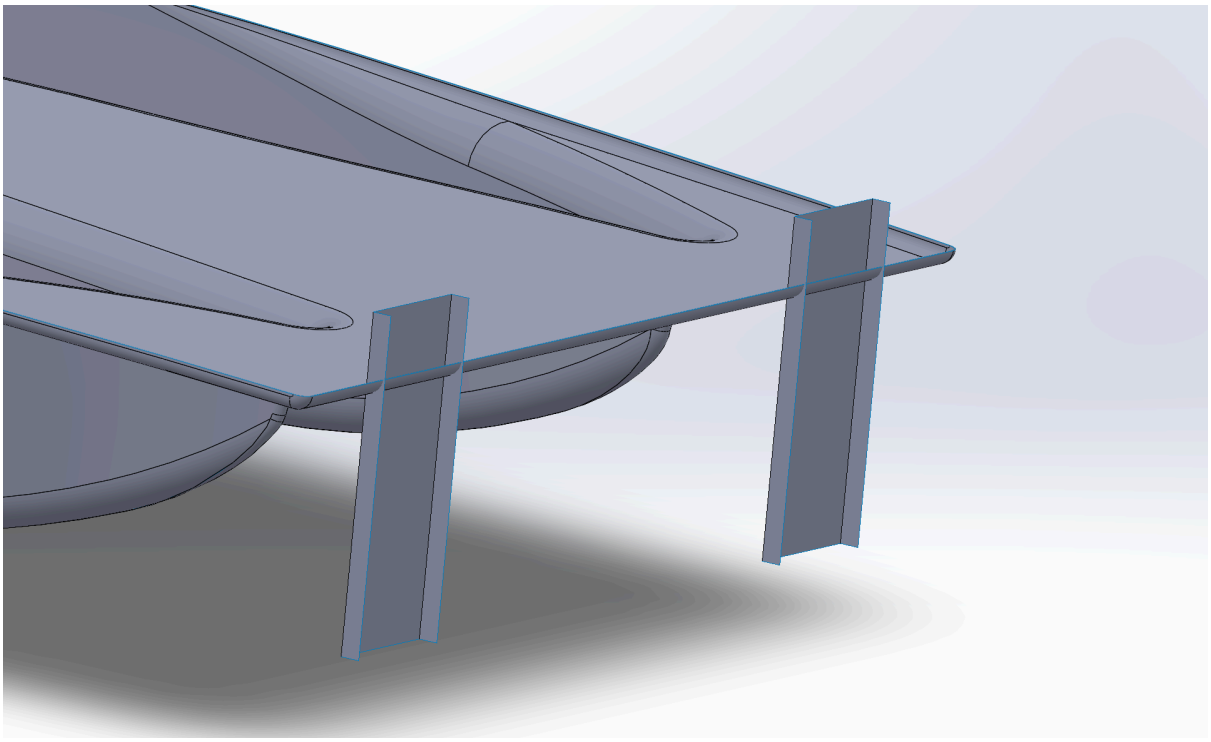
2. **Rear-Light Indents**

   Step one is to know where the lights are located. For Cascadia they are located under the bounding box configuration in the Catamaran_EP4 file.

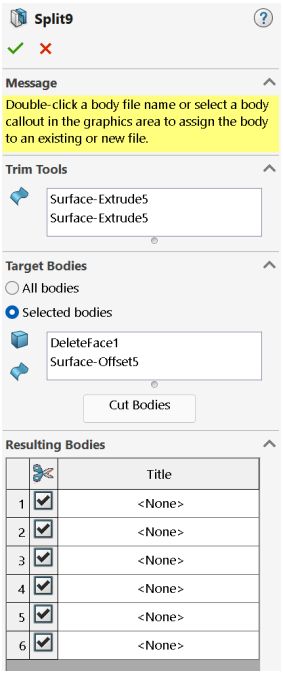
   You then want to create a sketch on the top plane. Mark out where the lights will be with a box. Make sure to make your dimensions relative to the car to ensure they are in the right location. Then, trim the outer edge of the box (away from the car) to form a u-shape.

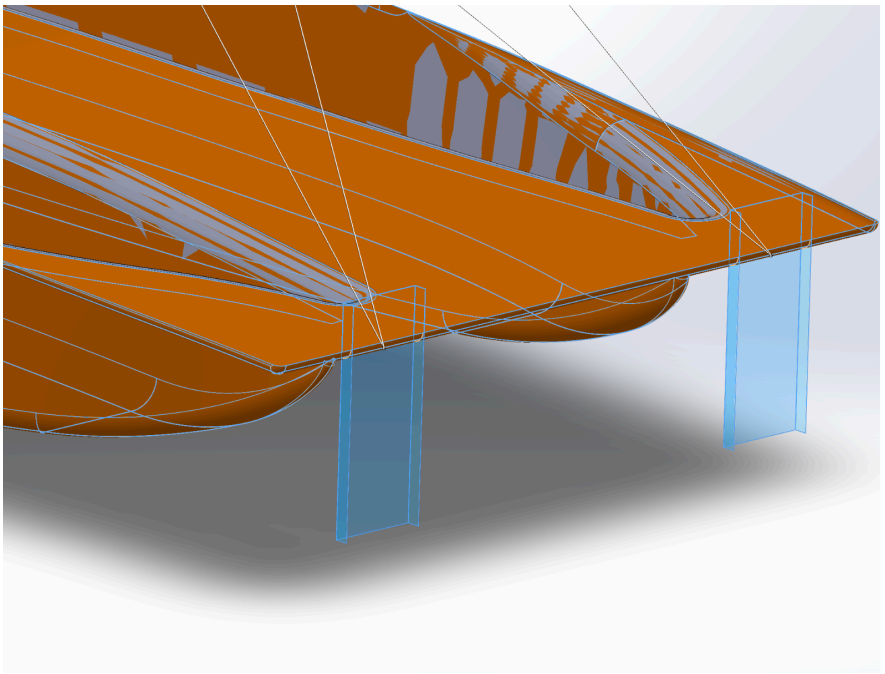


You then want to surface extrude this u-shape all the up until it cuts through the bottomshell.

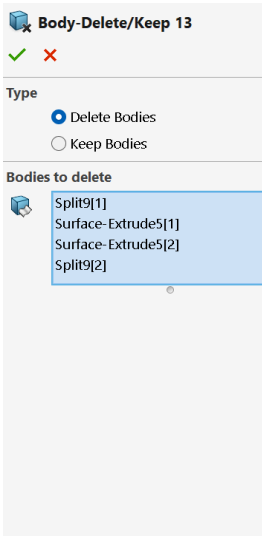


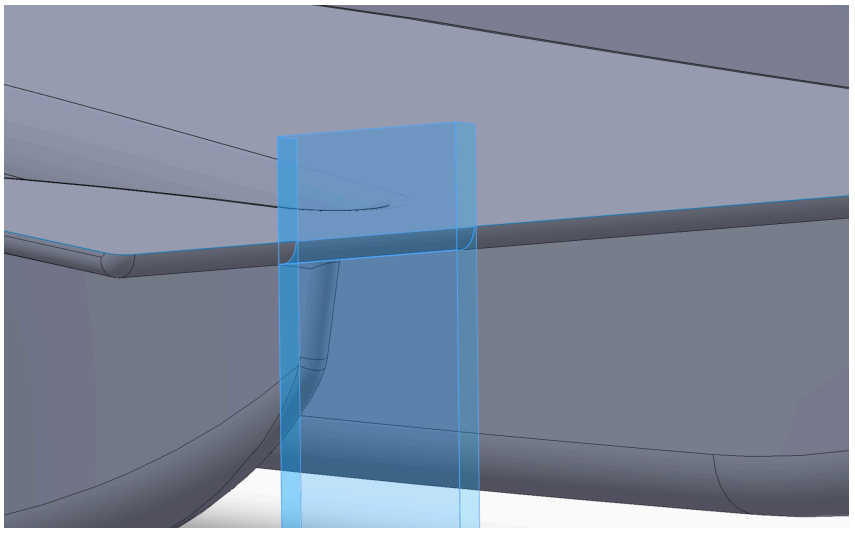
Once that is done, find the split tool feature. Use the two extruded u-shapes as the trim tool and select both the original aeroshell surface, as well as the 1mm offset as the bodies to trim. Select cut-bodies. Select all cut pieces.





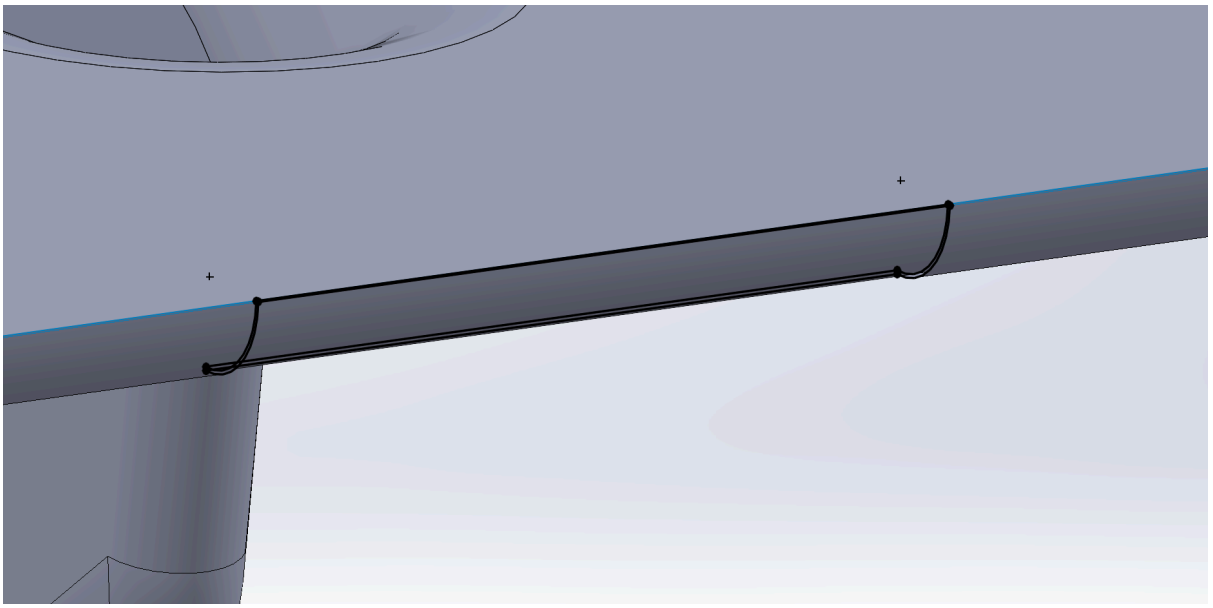
Select the delete bodies features and delete the u-shaped surface extrudes, along with the original aeroshell surface that has been split.



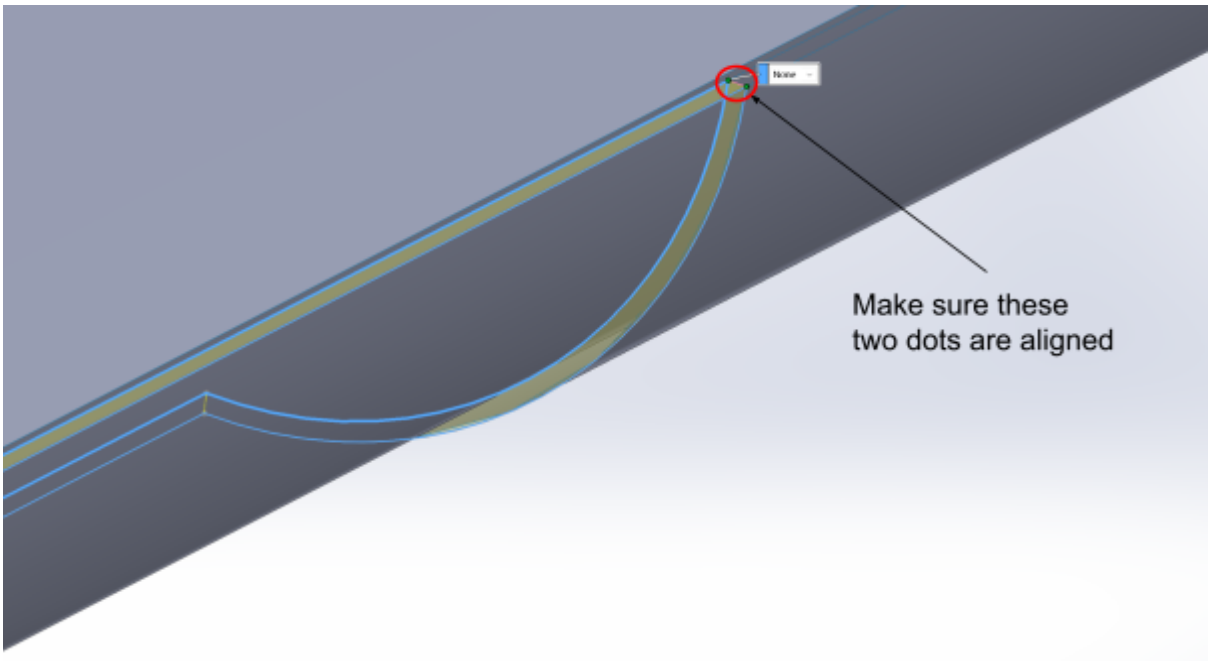


You now want to create a boundary surface between the 1mm offset face and the original aeroshell body. This can be tricky. It is best to create a 3D sketch in this scenario to draw out the lines in order for Solidworks to not get confused.

Once you've created a 3D sketch, select the convert entities feature. Once selected you can select all the edges that go around the cut surface on the original shell, and the 1mm offset surface. Complete the loop by drawing a line across the two end points.



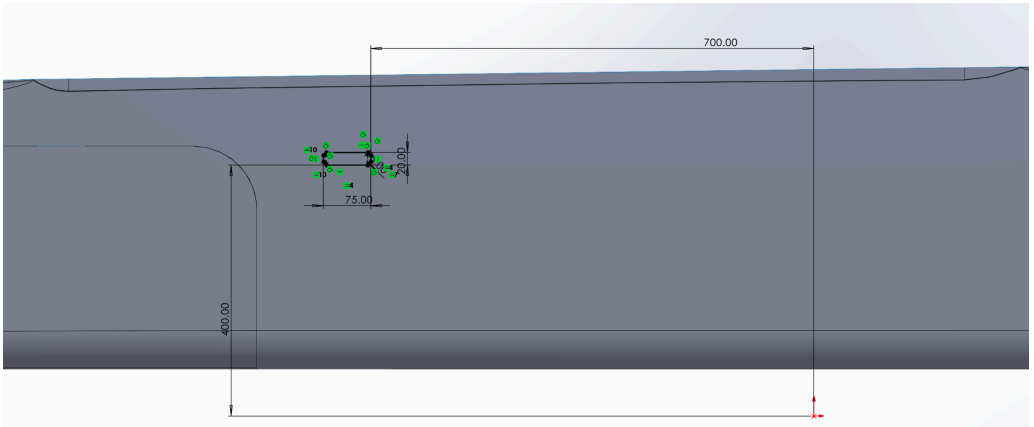
Now select the boundary surface feature. This is a very particular tool in Solidworks to the point where if you click it in the wrong order or location it will fail. For these lights, click on the line that connects the two ends together. Click the inner line and then the outer. For one of the lines you may have to right-click on the line and select the selection manager. This will allow you to select a closed loop. If you've done this correctly a yellow preview will appear. If it doesn't appear it could be for one of two reasons. First, the two green nodes are not aligned. To fix this move the nodes beside one another. Second, the loop selection has failed, in which case you have to unselect both loops and then try again.



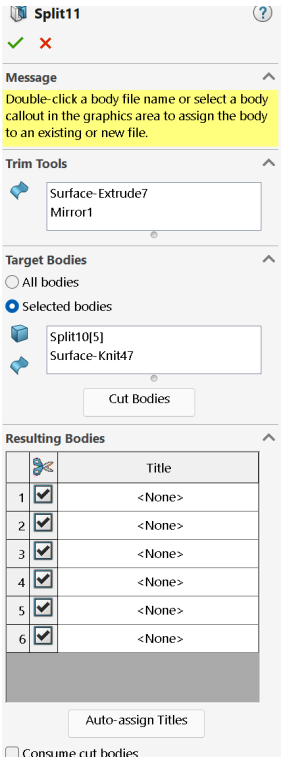
Finally knit everything together (the original aeroshell, the offset surface and the boundary surface)

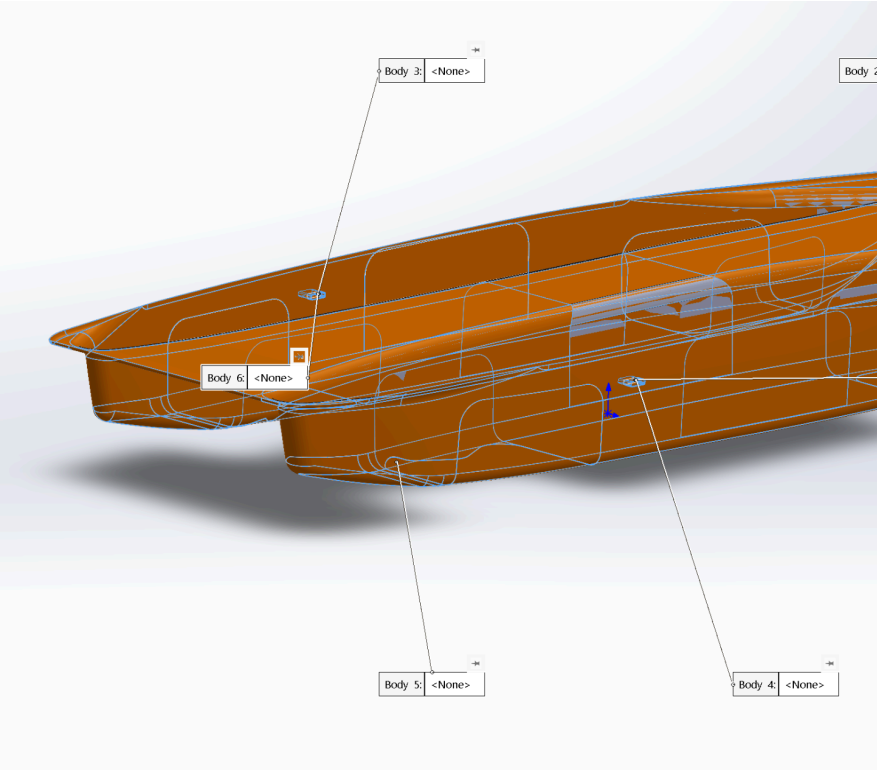
3. **Side-Light Indents**

The process for this is quite similar to the rear-light indents. The only difference is that you need to create a reference plane on the outside of the car. Then place your light box and extrude the surface. You can then mirror this surface to the other side of the car.



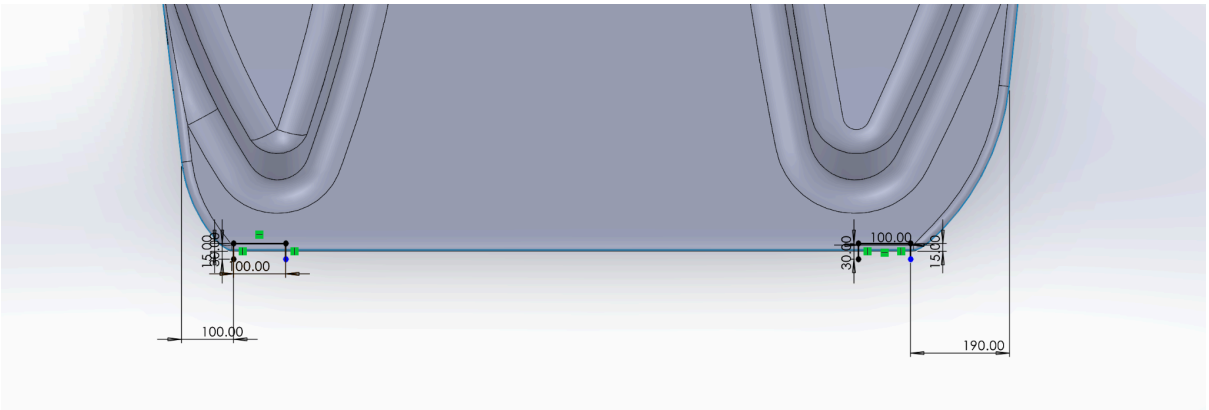
Repeat the process of splitting just the original aeroshell and the 1mm offset. When creating the boundary surface for this however, you shouldn't need to create a sketch for the loop.





Finally knit everything together (the original aeroshell, the offset surface and the boundary surface)

4. **Front-Light Indents**
   For the front lights use the exact same process as the rear-lights.You just have to change the sketch of the box where the light goes according the the bounding box.



Split10

Message

Double-click a body file name or select a body callout in the graphics area to assign the body to an existing or new file.

Trim Tools

Surface-Extrude6
Surface-Extrude6

Target Bodies

All bodies
Selected bodies

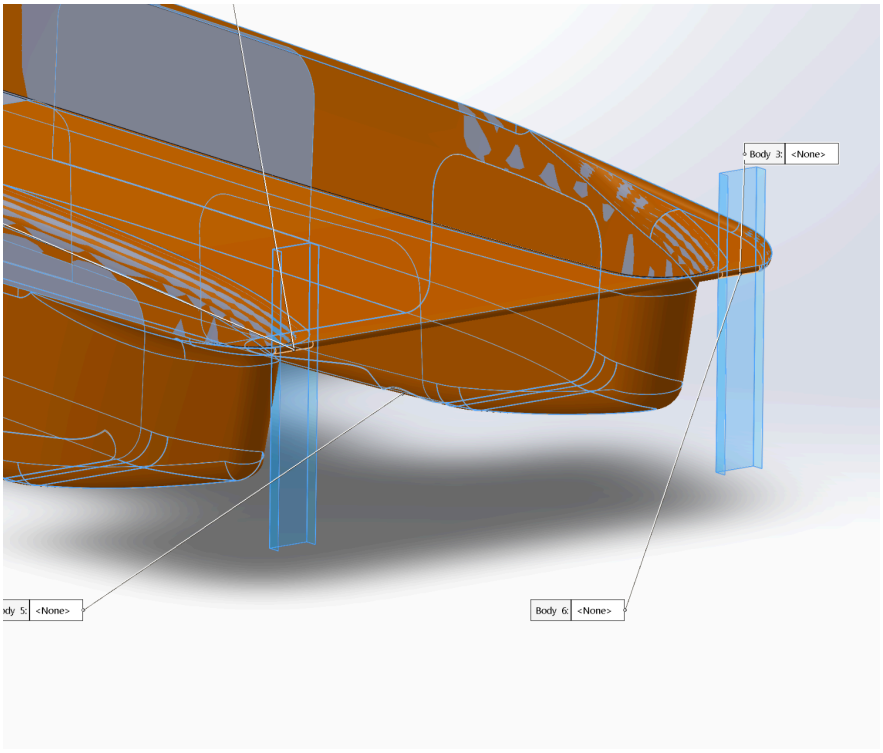
Top split[10]
Surface-Knit46

Cut Bodies

Resulting Bodies

		Title
1	☑	<None>
2	☑	<None>
3	☑	<None>
4	☑	<None>
5	☑	<None>
6	☑	<None>

Auto-assign Titles



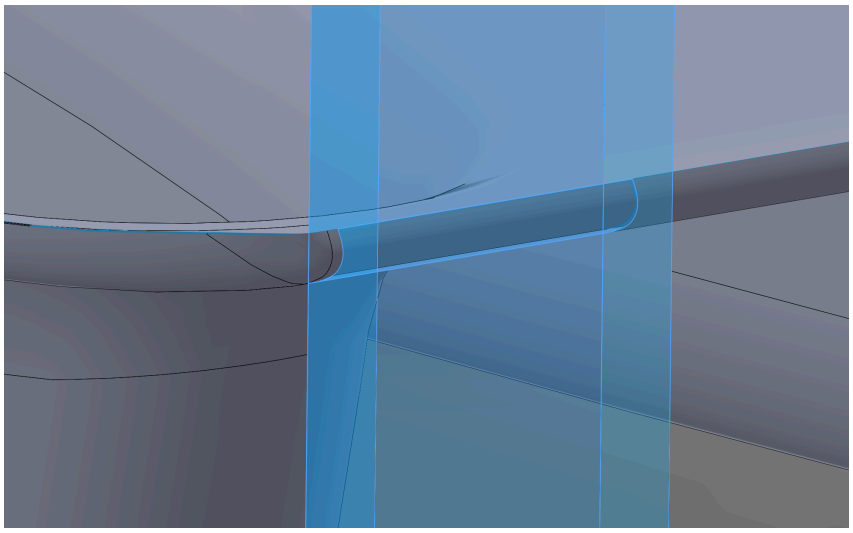
Body-Delete/Keep 16

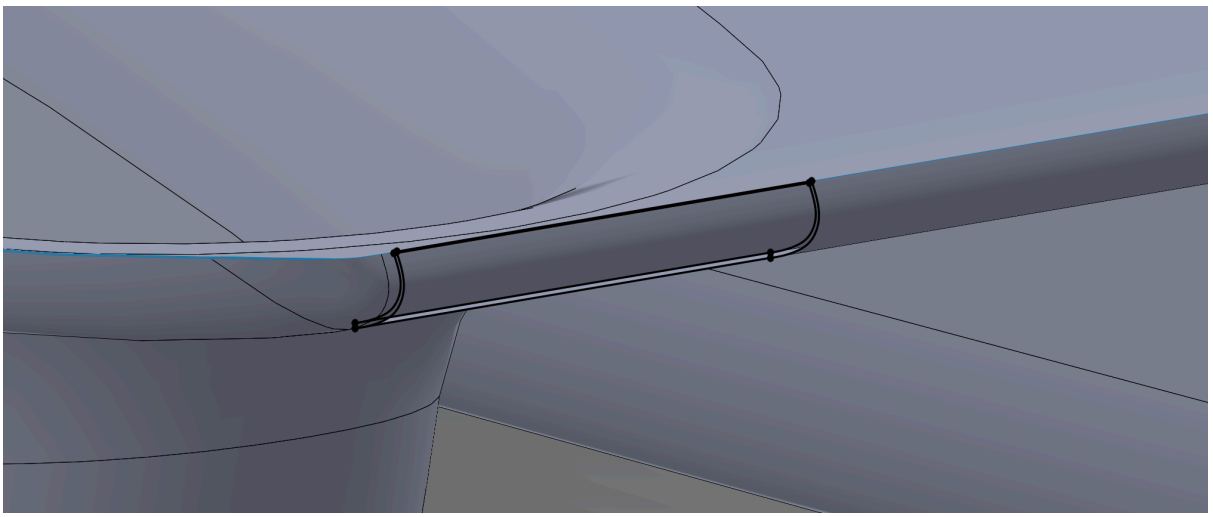
Type

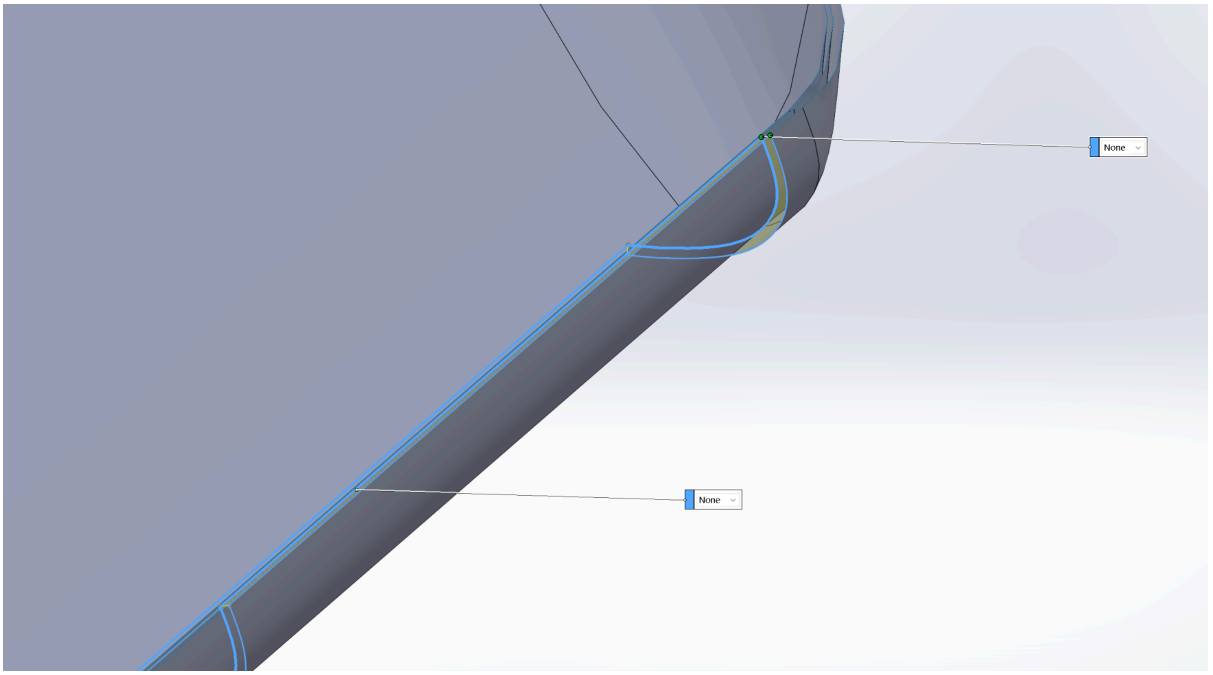
Delete Bodies
Keep Bodies

Bodies to delete

Surface-Extrude6[1]
Surface-Extrude6[2]
Split10[1]
Split10[3]

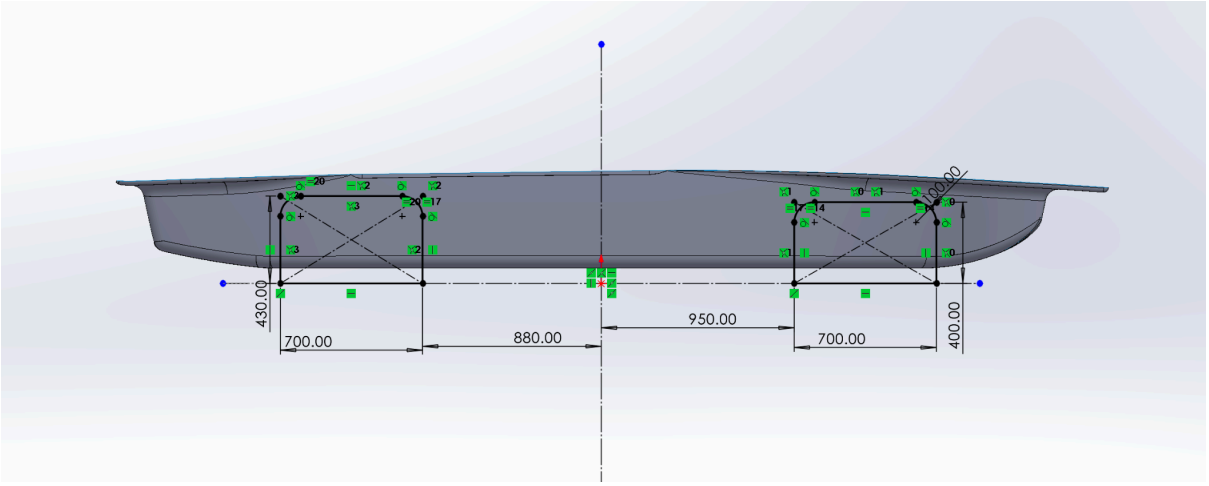






5. **Fairings + Battery Access Panel**

   The first step is creating the side and top profile of the fairings and the battery access panel. As shown in the three photos below that has been done according to the fairing dimensions for Cascadia. The side profile of the fairings and battery access panel were sketched on the right plane, and the top view was sketched on the top plane.


430.00
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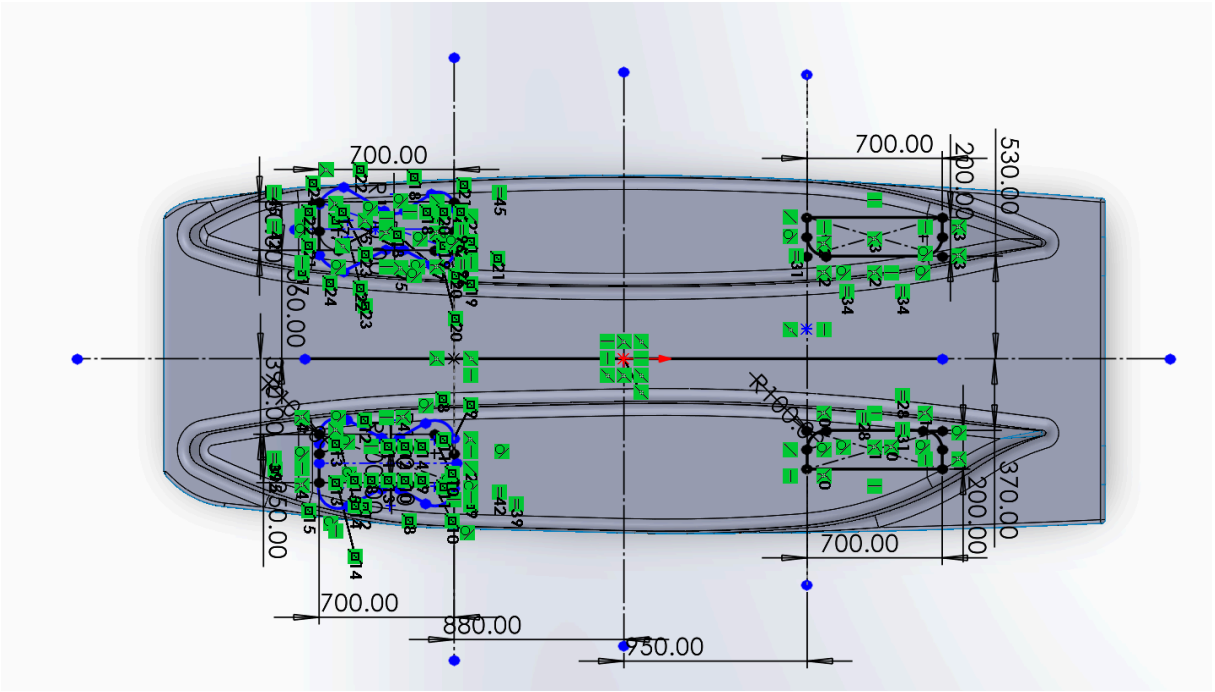
400.00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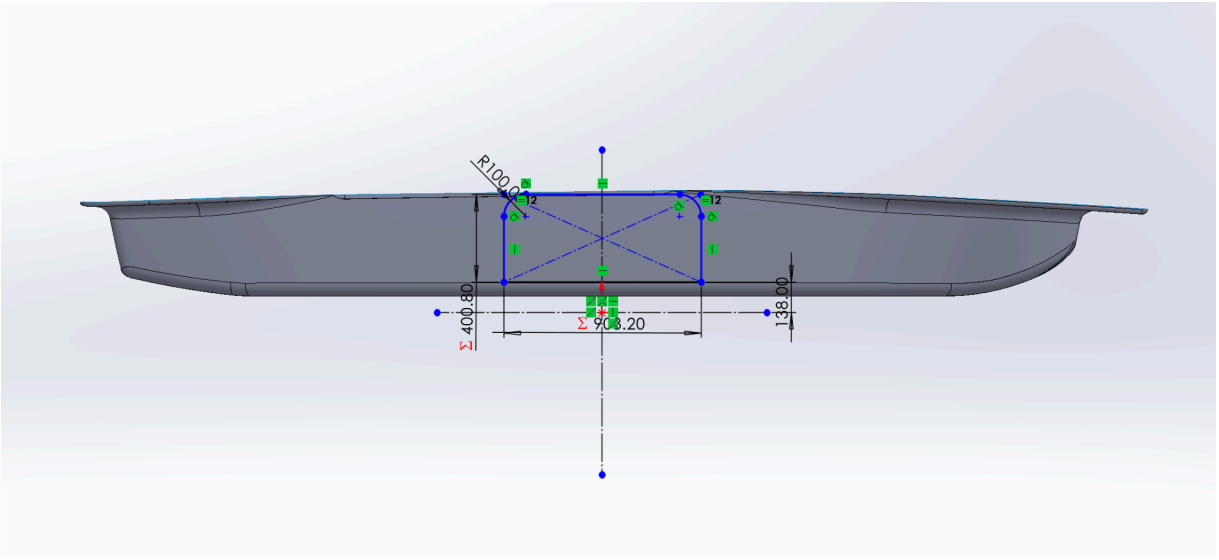
R100.00
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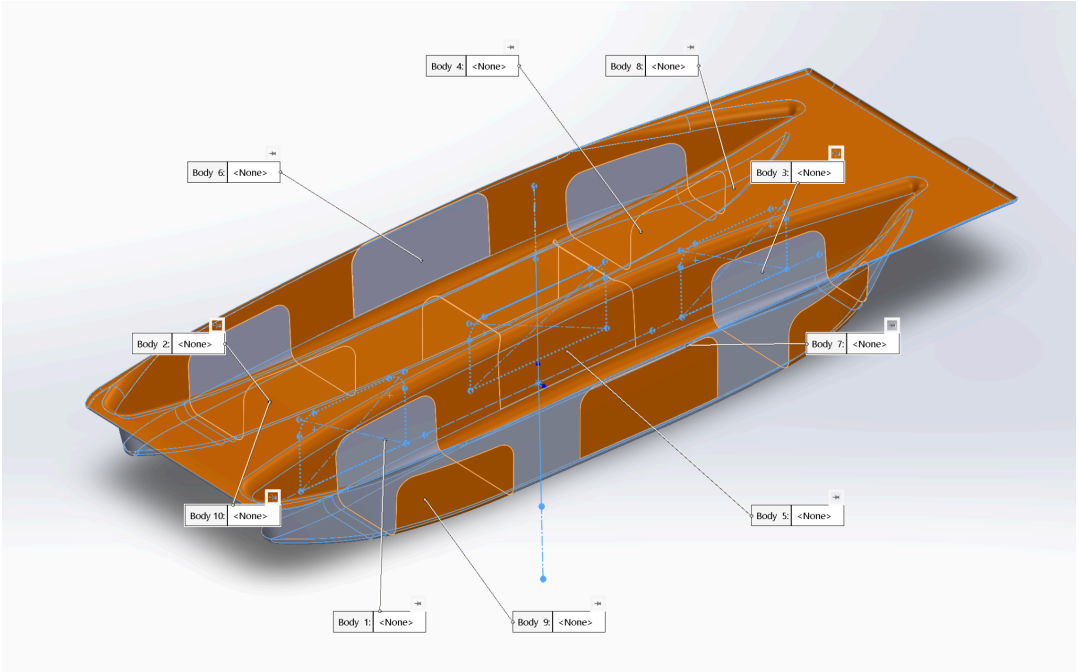
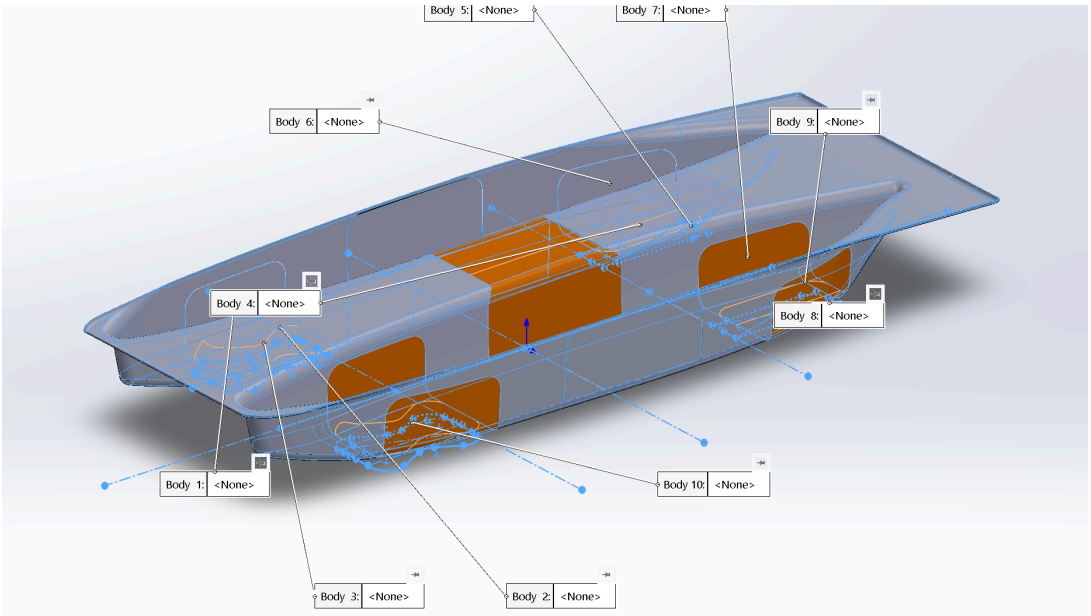
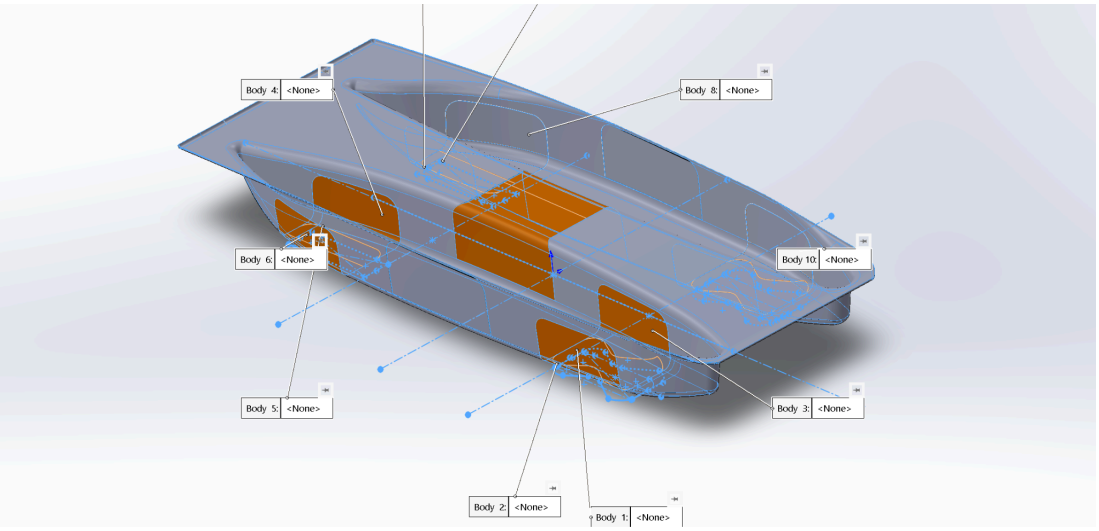
138.00

Once you do this, you want to split the aeroshell and the 1.5mm offset. You are going to split it twice. The first split, you want to select the sections referred in the image below. These would be the inside of the car except the area where the fairings split, and then the outside where the fairing is.

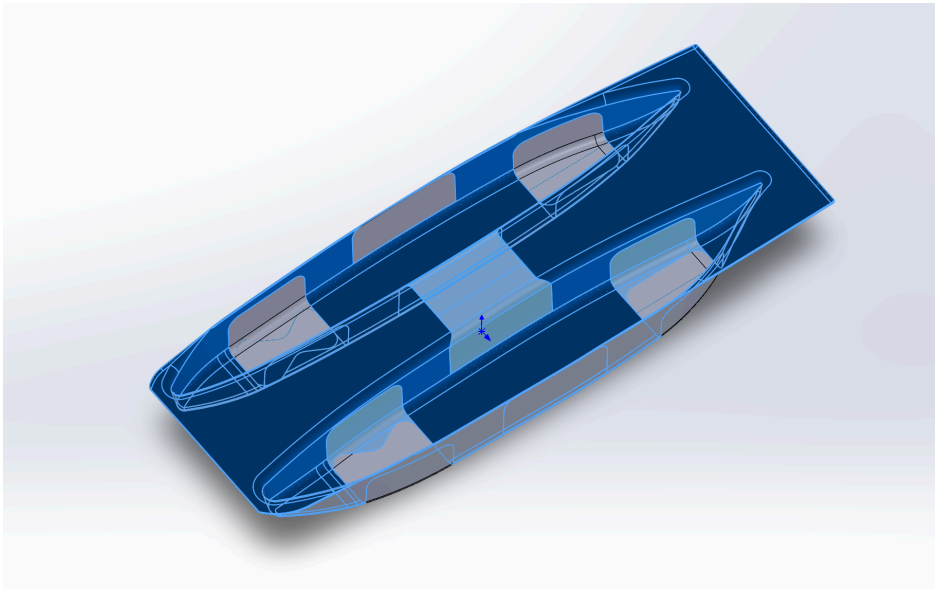


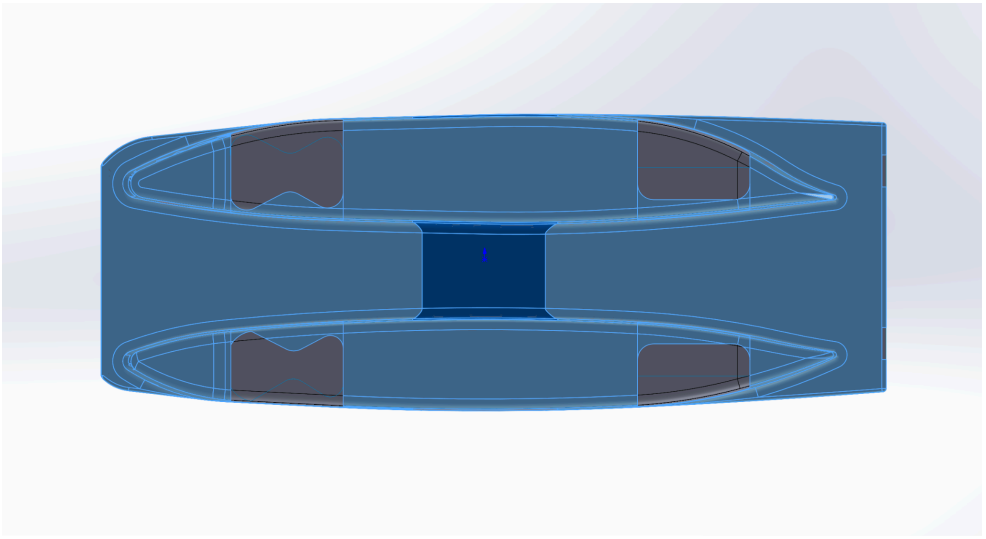
Next you want to split it again, but this time only selecting the outside of the fairing and the inside that is not going to be fairing. This is because the split will cut through the inside of the hull and the outside of the hull. In the end you only want it to split the outside. So by doing this, you can knit together the rest of the pieces after. The photo below shows this second split selection, and the photo after that is the same split but another angle.



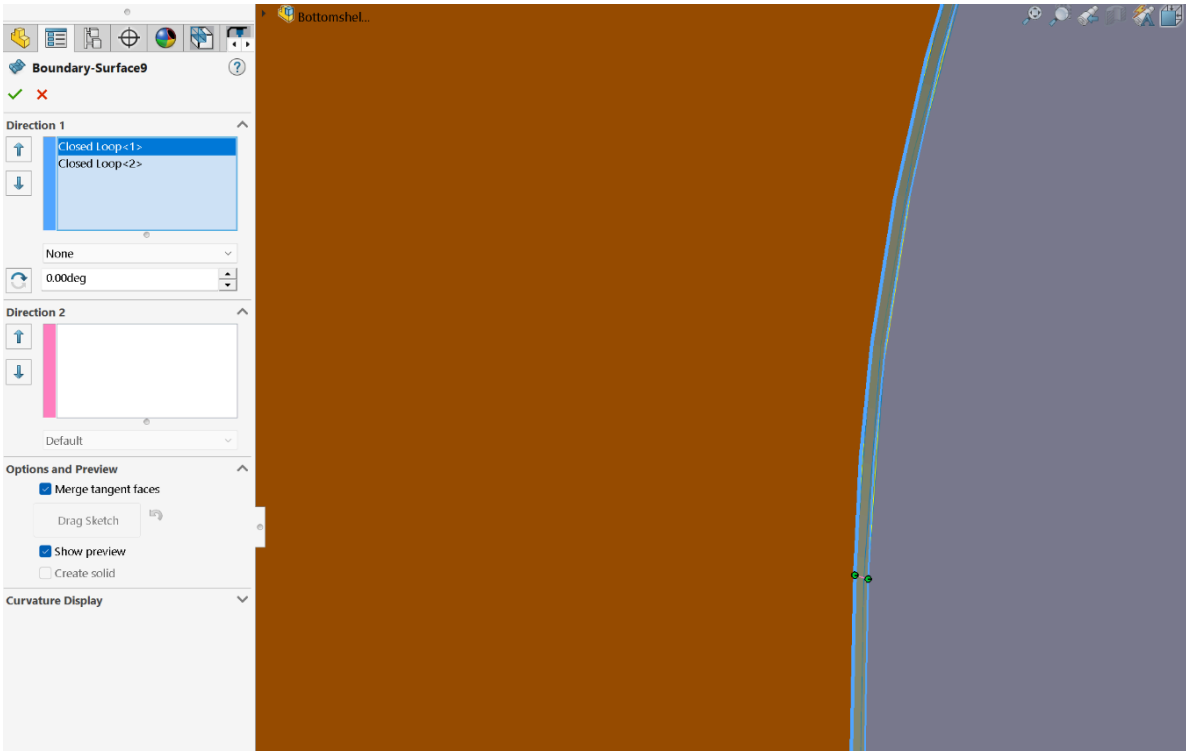


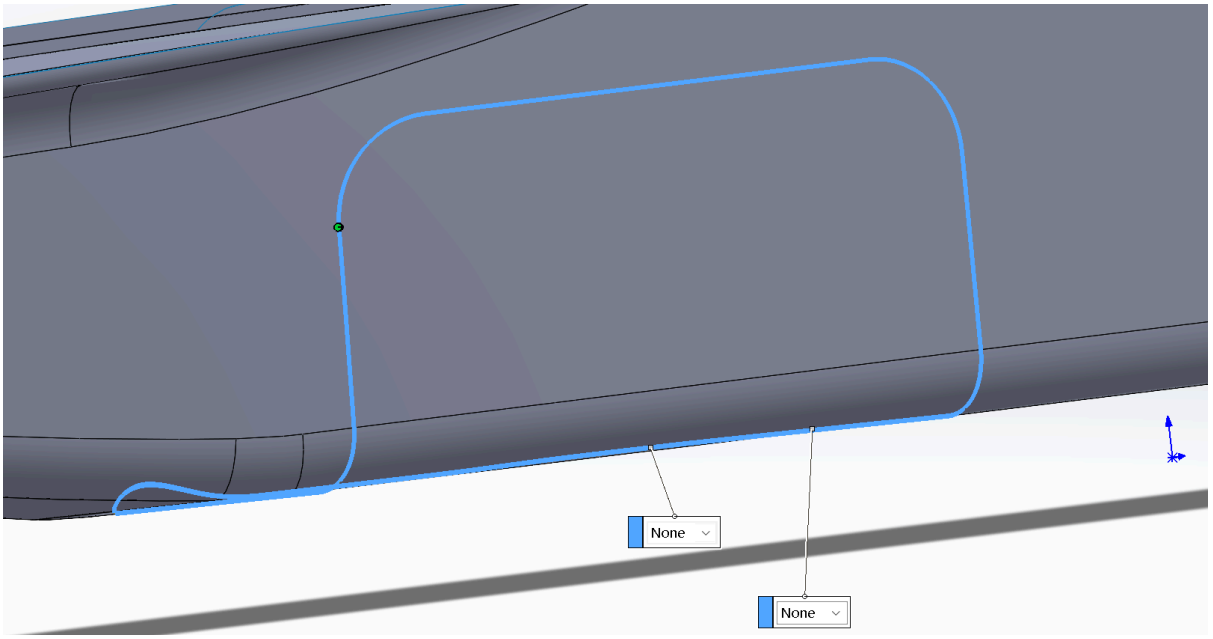
Next you want to knit **everything but the fairings and access panel** together. One knit will be the inside, and another will be the outside of the aeroshell.





Next, you wanna make all of your boundary surfaces for each fairing. It “should” be a simple process. First, try selecting each loop on its own to see if Solidworks is nice and will create a closed loop for you automatically. In my case it did not, so I created a 3D sketch that outlined the two loops that I wanted to surface between.

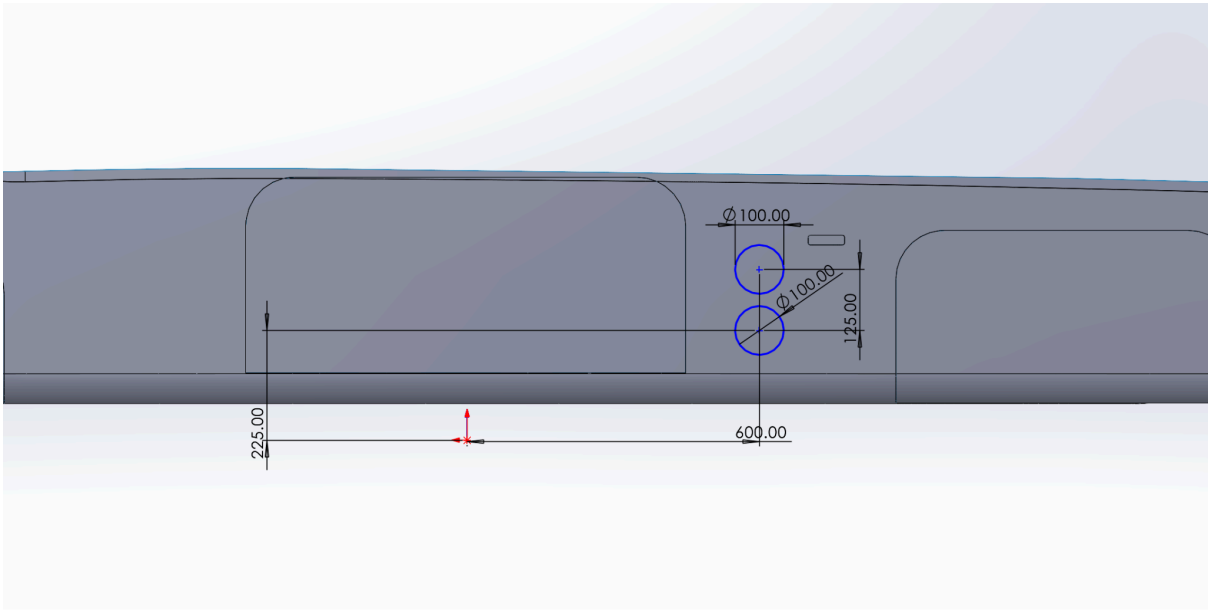


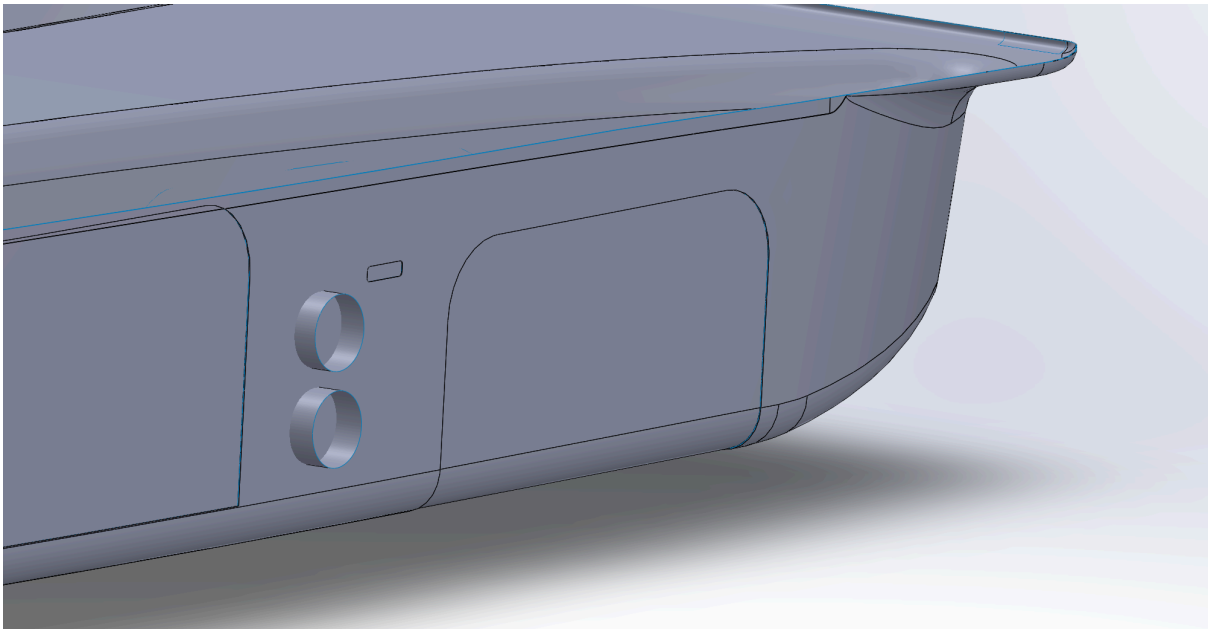


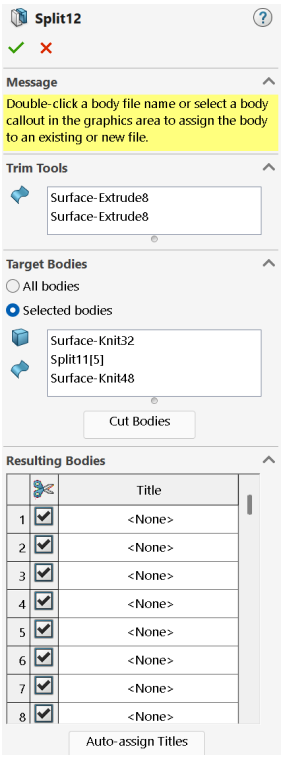
Repeat this process for all four fairings and the battery access panel. Make sure to knit each indent to the rest of the car as you progress through.

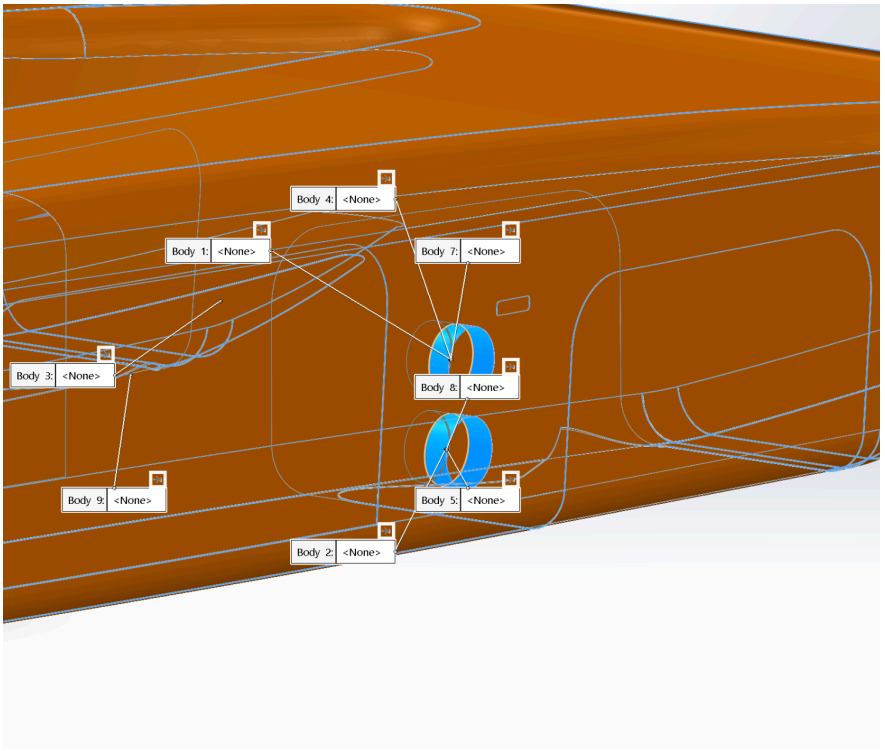
6. **Battery Intake + Exhaust**
   The process for this is very similar to all the lights except you are using the 1.5mm offset instead of the 1mm offset.

   Create a reference plane on the outside of the battery side hull. Create another reference plane on the inside of the battery hull. On the reference plane on the outside, create the sketch of the intake holes. Follow the steps for the side lights to split it. You can split the original aeroshell and both offset surfaces. You will then delete the 1mm offset surface and the original aeroshell which has now been split.

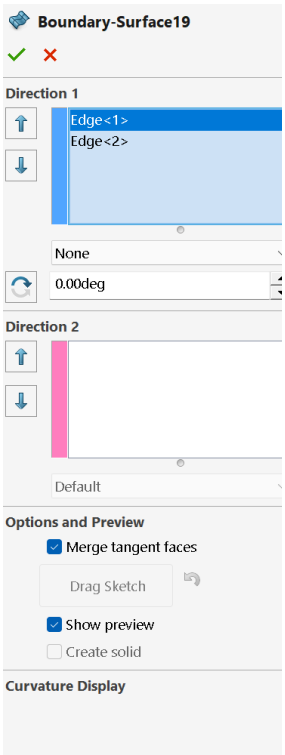


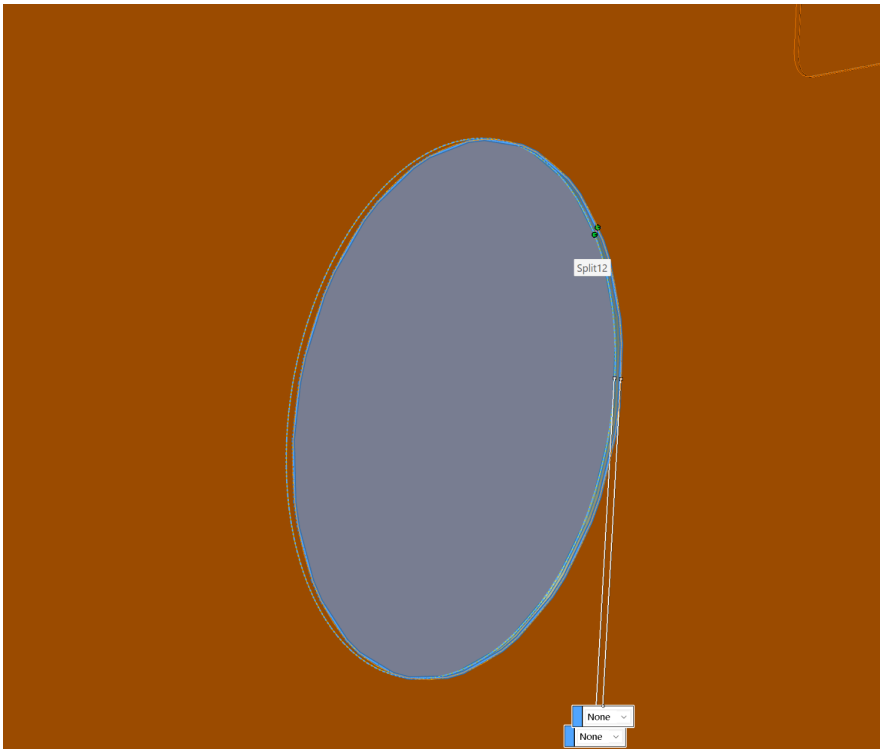






Then as per usual, create the boundary surface and knit everything together. Repeat this process for the exhaust duct on the inside of the hull.

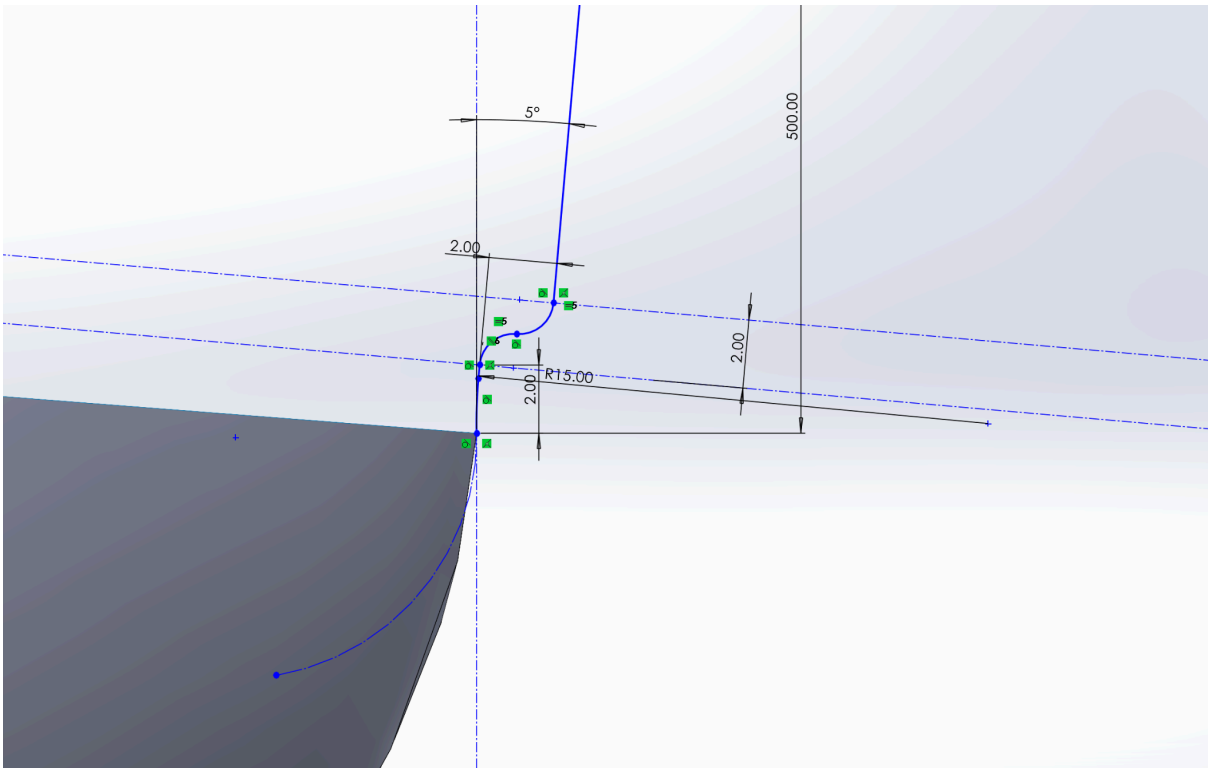




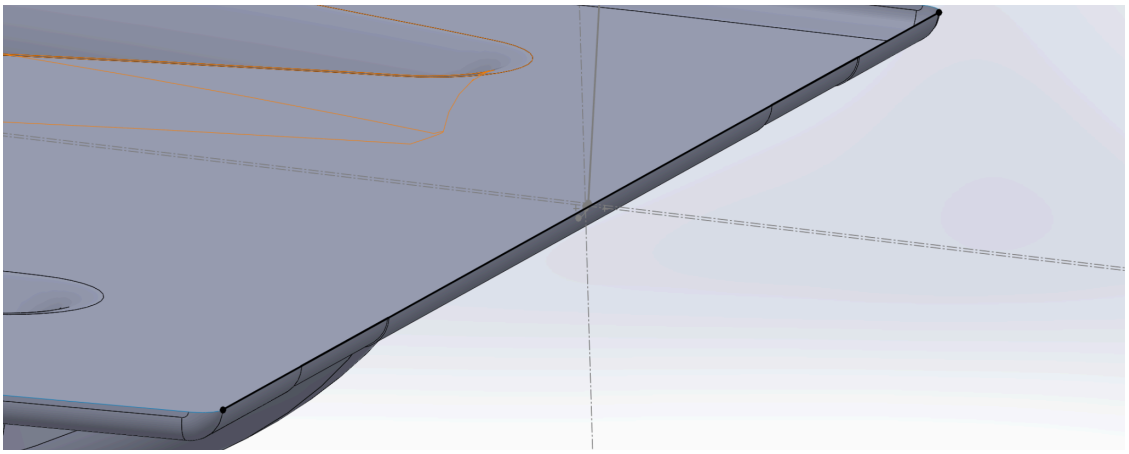
7. **Extension**

The purpose of the extension is to create a flat surface for the plug that we can epoxy on the wooden jig. Creating the extension is a very repetitive process. It is essentially two steps over and over**: 1. Surface sweep or loft. 2. Surface Knit to the rest of the car.**

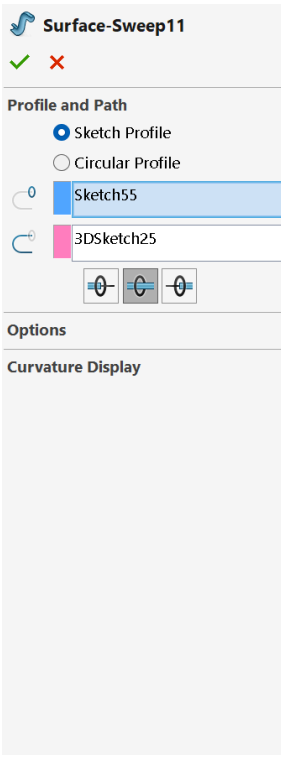
Start with the rear of the car. Create a sketch on the right plane and replicate the sketch shown below. This is the profile for the extension. Make sure to make the bottom of the profile pierce the car.

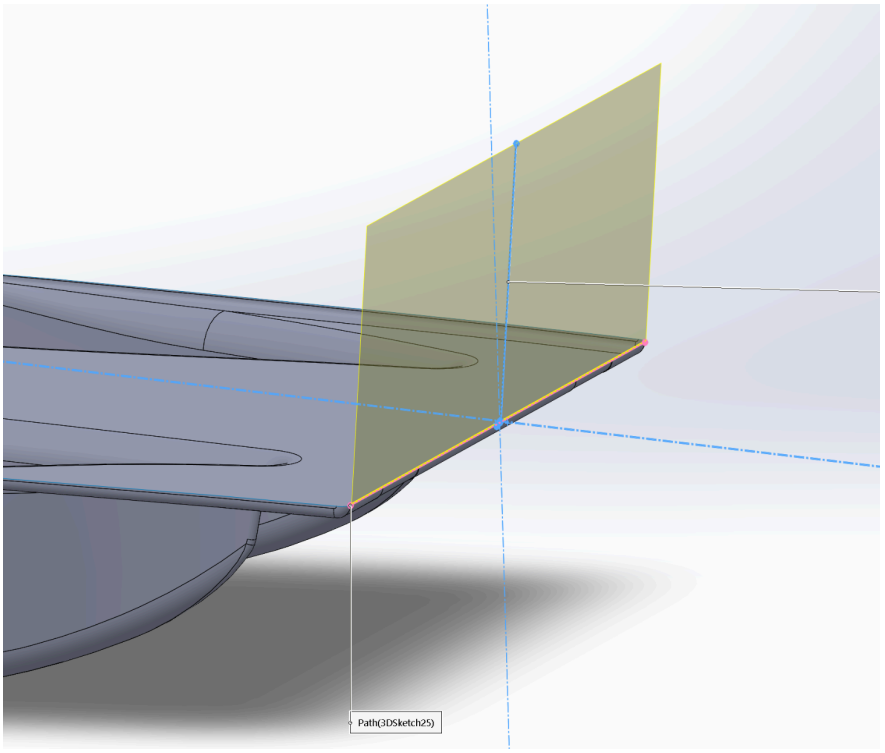


Then create a 3D sketch that runs along the path of the back of the car. You can use the convert entities feature to make this easier.



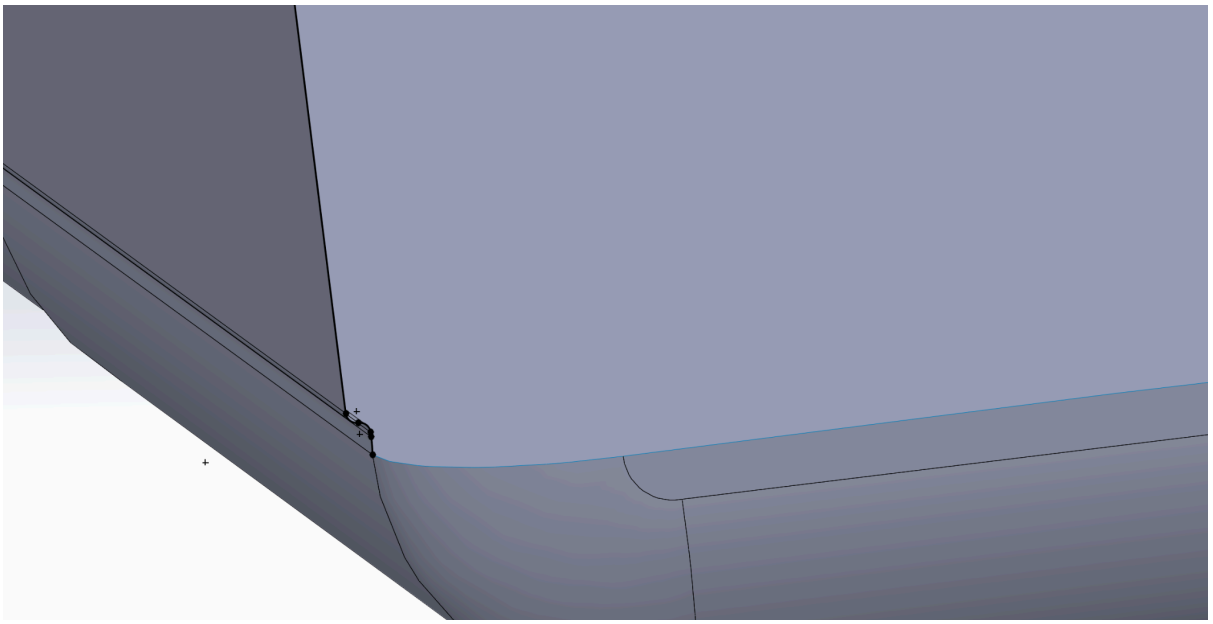
Then use the surface sweep feature. Select the profile on the right plane as the profile and then the 3D sketch as the path. Make the surface run both ways.

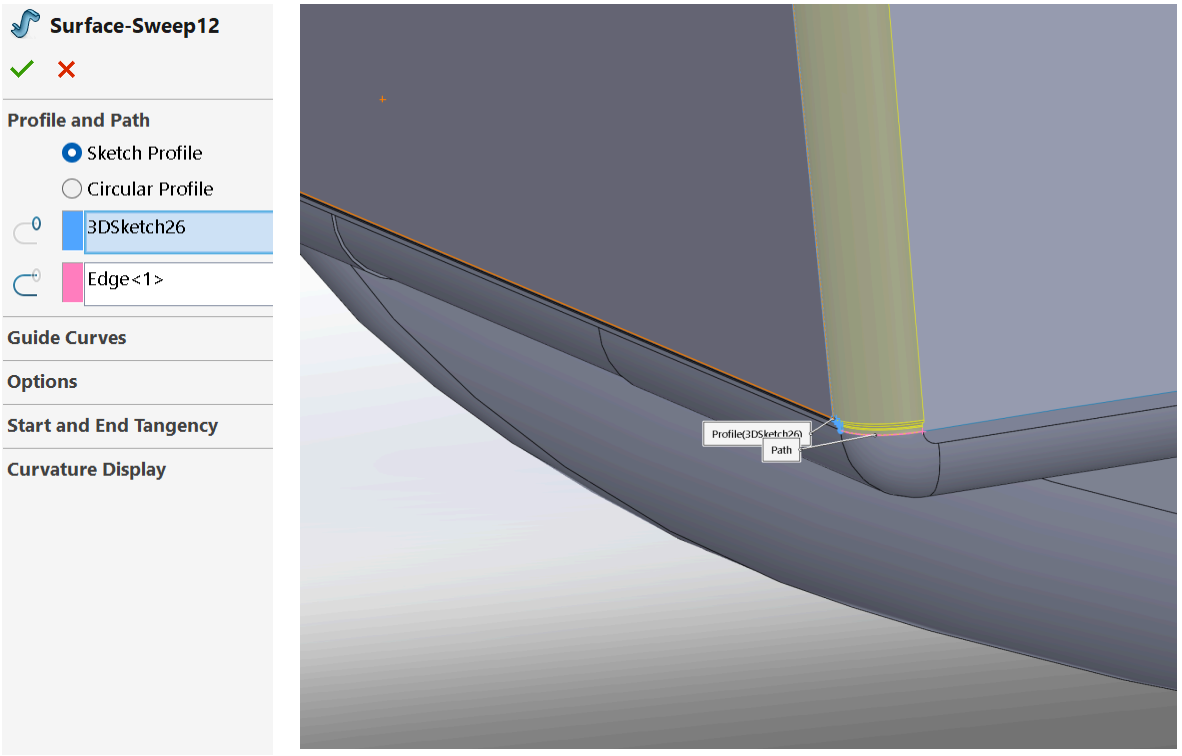




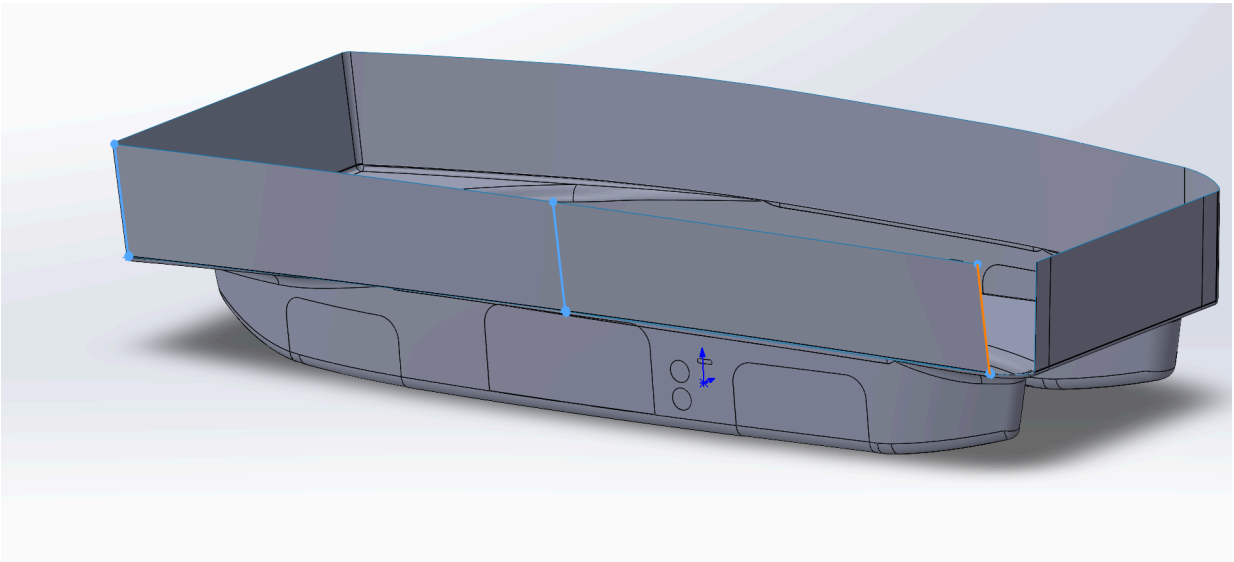
Then knit the surface to the rest of the car, and that's one side done.

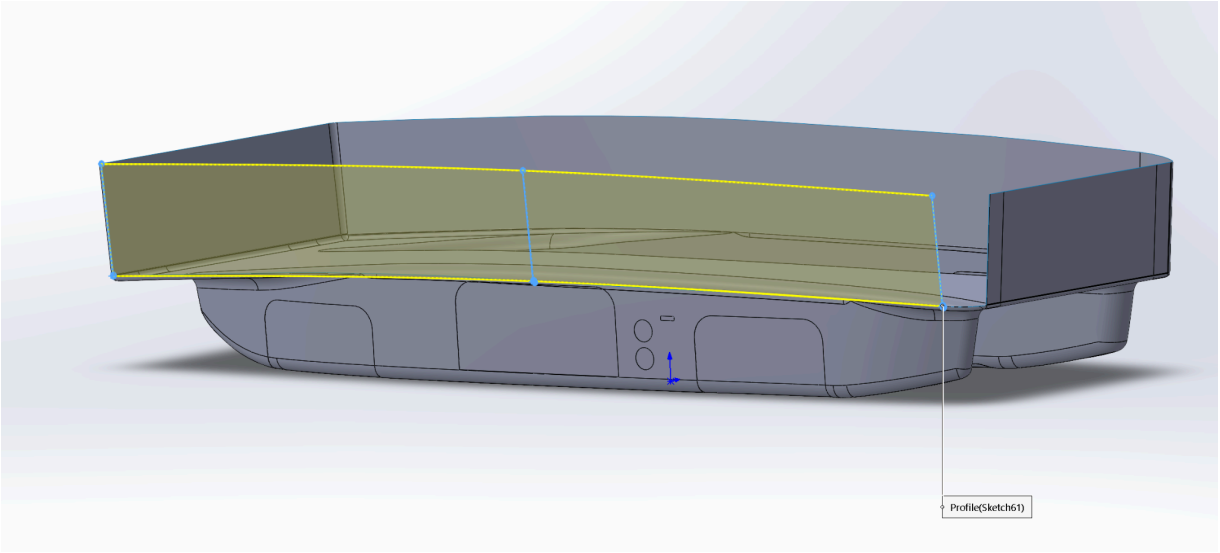
For the corners of the car, create a 3D sketch and use convert entities to create the profile of the extension again. You can then use surface sweep to create the extension: selecting the 3D sketch as your profile and then the curve as your path. Once again, knit the surface to the rest of the car when complete.





For the sides of the car, you have to add that profile at the end, middle and front of the car. Then you have a 3D sketch that is the guideline between all three of those. Then surface loft it and select the three extension profiles with the 3D sketch as your guideline. For the middle sketch I used the front plane for sketch reference. For the front I created a reference plane at that point.



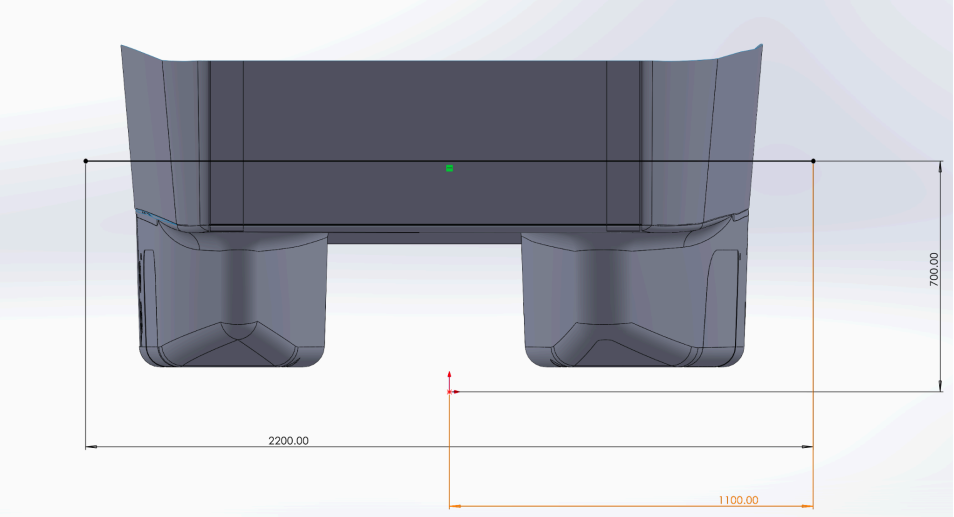


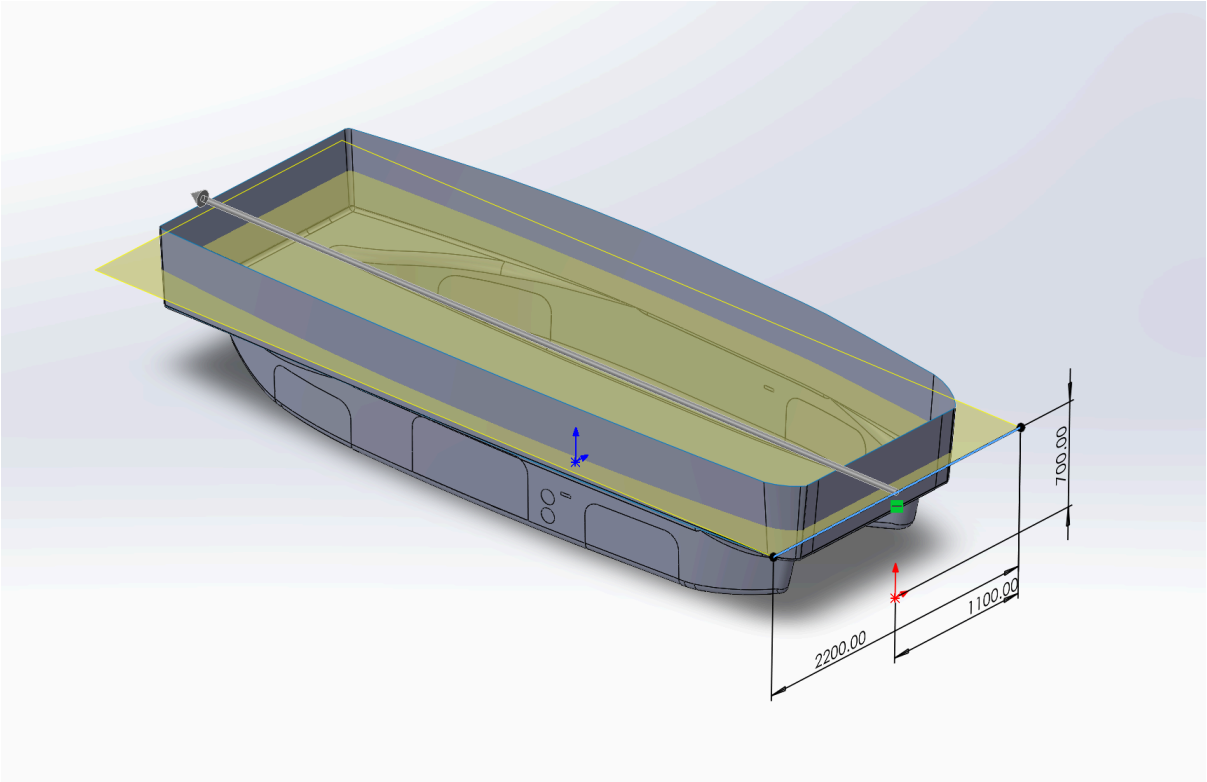
Repeat this process for the right side of the car. A trick to make your lift easier is, if you try to copy the extension profile you can just move all the entities at once(using move entities) to the new location you want. And you can also mirror entities if you switch sides.

Repeat the process for the back sweep and back corners for the front sweep, and the front corners.

The final step is to make a flat top for the extension, so that the mold can sit flush with the wooden jig that it will be epoxied onto.

The first step is to create a sketch on the front plane at the height you want to cut the extension off at. Then extrude this line all the way past the front and the back of the car. Then you want to use the surface trim tool to cut away all the excess. You want to trim the extension first to the right height before trimming the excess surface extrude.





Surface-Trim7

Trim Type

- (•) Standard
- ( ) Mutual

Selections

Trim tool:

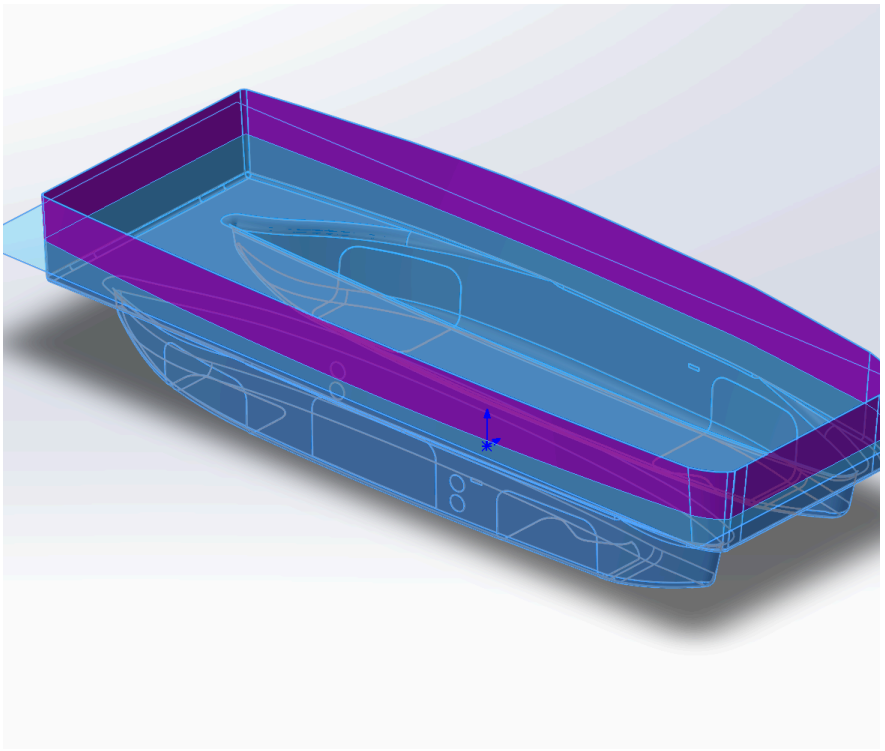
Surface-Extrude10

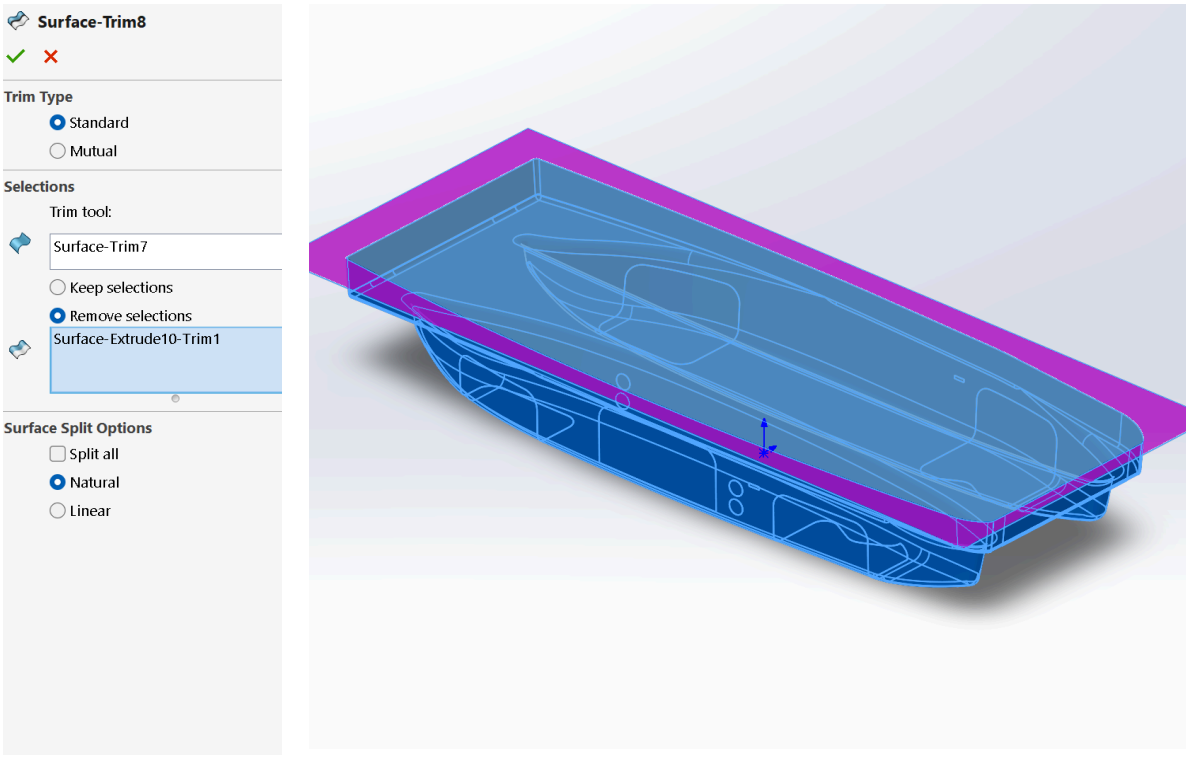
- ( ) Keep selections
- (•) Remove selections

Surface-Knit59-Trim0

Surface Split Options

- [ ] Split all
- (•) Natural
- ( ) Linear





Finally knit everything together and you have a completed bottomshell plug!

Final Product: