Chapter 2: Mechanical Design

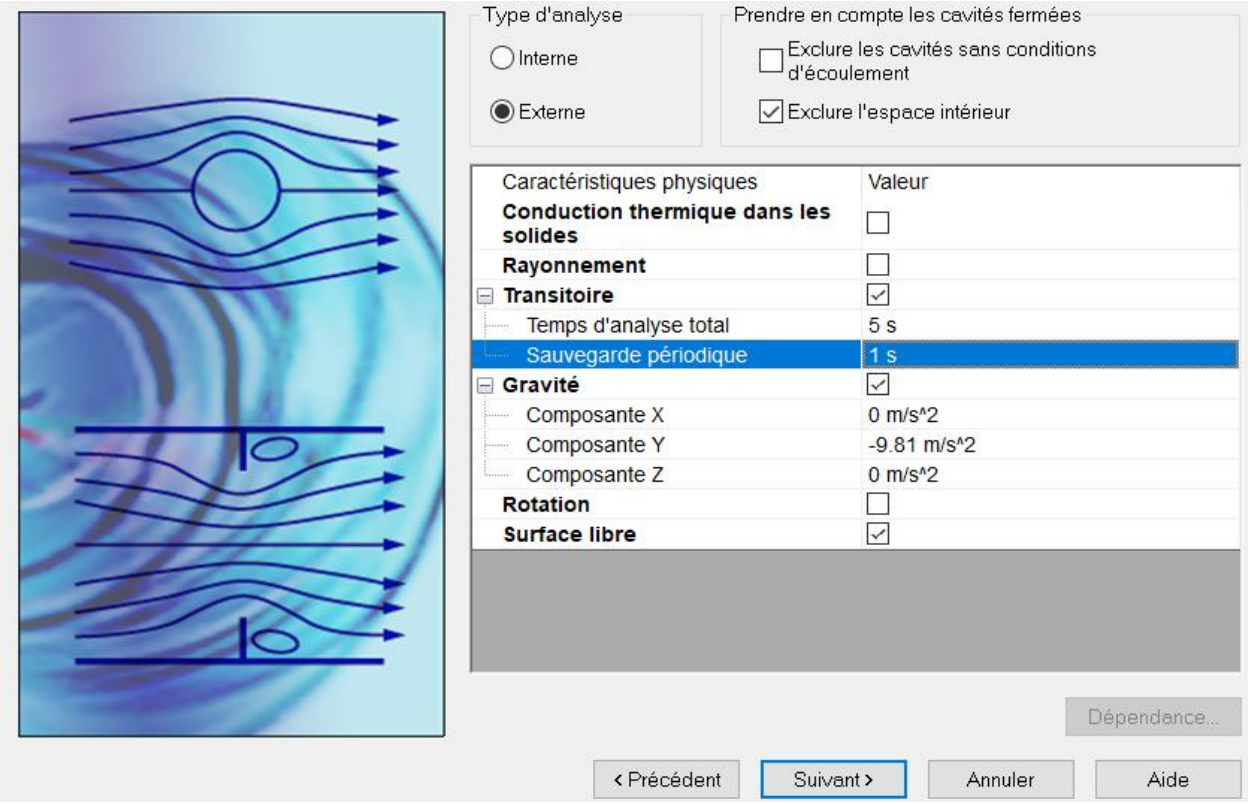
1. Introduction

The mechanical design of my cleaning boat is inspired by the streamlined and stable body of the whale shark. This marine creature, known for its wide and flat shape, served as a natural model to achieve high stability and low hydrodynamic resistance. Based on this inspiration, I designed a **catamaran-style boat** with two hulls, which provides both balance and space to mount electronic components. In this chapter, I present the mechanical choices I made regarding structure, dimensions, materials, and the use of simulation tools.

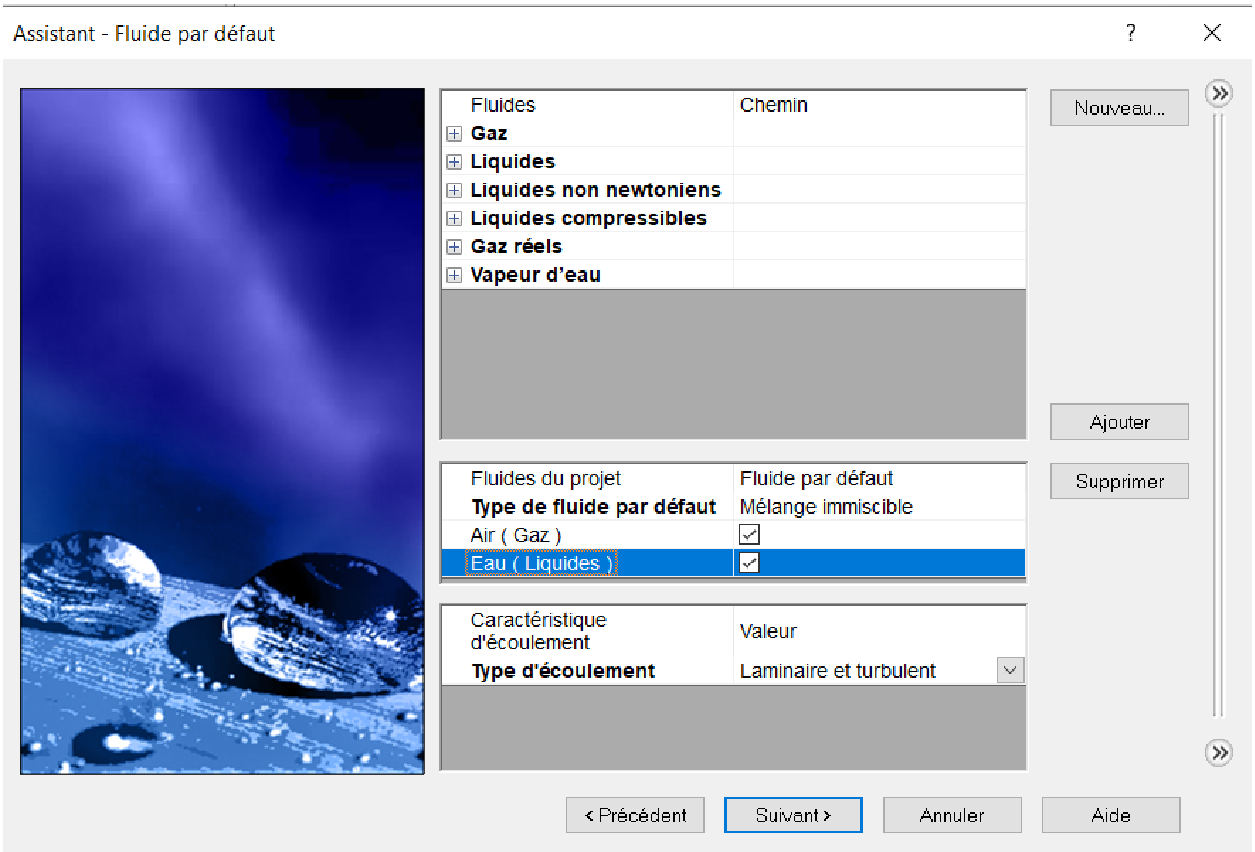
2. Structural Analysis

2.1) F low Simulation Setup

For the external flow analysis of the boat design, I selected **"External"** as the analysis type to focus on the interaction between the boat hull and the surrounding water and air. The **"Exclude internal space"** option was checked to ensure that only the outer geometry was considered in the simulation. Under **Physical Features**, I enabled the **"Transient"** simulation mode with a total analysis time of **10 seconds** and data saved every **2 seconds** to capture dynamic flow behavior over time. **Gravity** was activated with a **Y-component of -9.81 m/s²**, reflecting standard Earth gravity acting downward, which is appropriate for the boat's orientation. Additionally, I enabled the **"Free Surface"** option to simulate the interface between water and air, which is crucial for accurately representing buoyancy, drag, and wave formation around the boat hull.

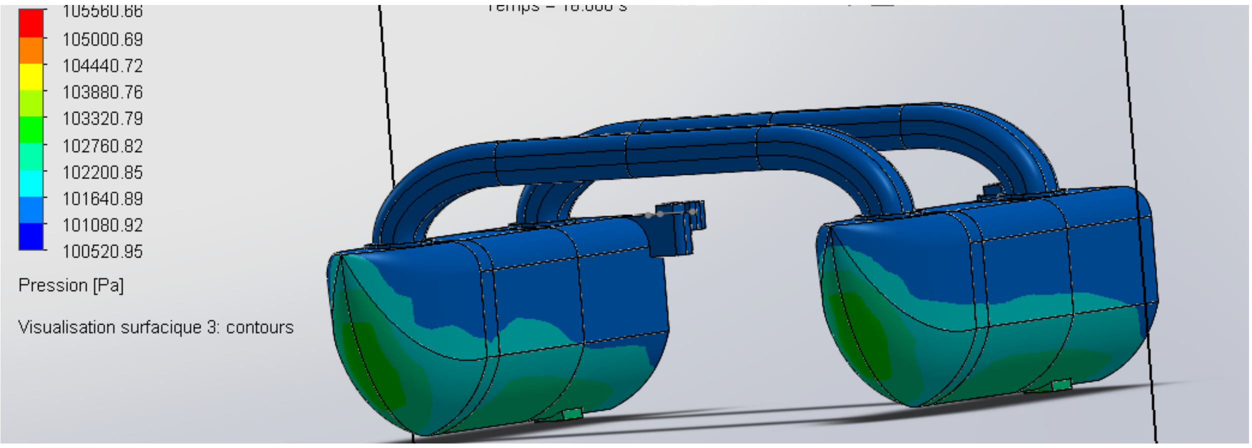


In the **fluid selection** step of the simulation setup, I included both **water and air** to accurately model the real environmental conditions experienced by the boat.



In the flow simulation, I set the **inlet velocity to 4 knots.** This value represents the boat moving at a relatively high operational speed, allowing observation of fluid behavior under dynamic conditions such as high drag and resistance forces.

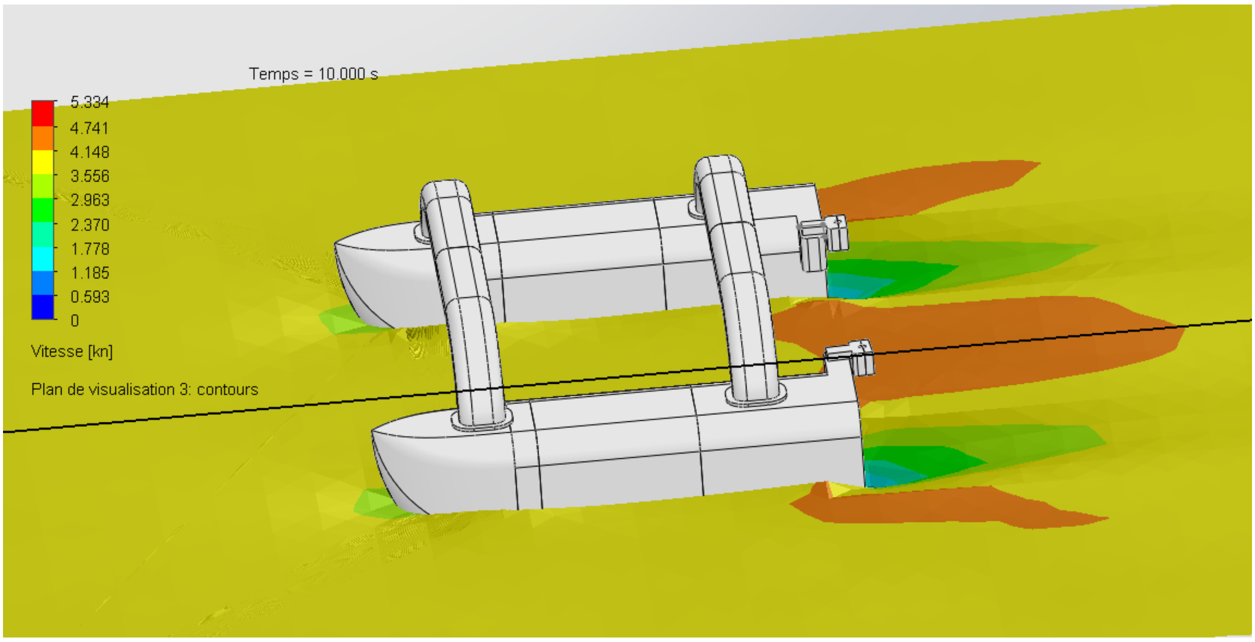
Following the velocity setup, I defined the **calculation domain**, which represents the virtual space around the boat where the simulation takes place



This image illustrates the pressure distribution on the double-hull catamaran during the external flow simulation using SolidWorks Flow Simulation. The simulation was conducted at a velocity of **4 knots (≈2.06 m/s)** with transient analysis over 10 seconds. The color gradient indicates the pressure acting on the surface of the hulls, where:

- **Red areas** represent regions of **high pressure**,
- **Blue to green areas** indicate **lower pressure** zones.

This result helps evaluate the hydrodynamic behavior of the boat design, particularly how water pressure affects the hull structure during movement. The curved, boat-optimized bow shape helps reduce drag and ensures smoother flow, as seen by the pressure gradients.



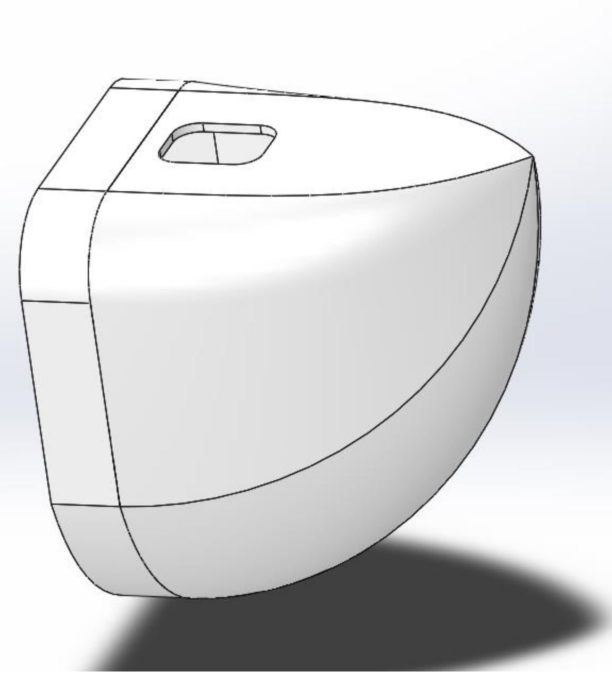
To analyze the flow behavior around the hull, a **Cut Plot – Velocity** was generated using SolidWorks Flow Simulation. This plot visualizes the velocity distribution of the fluid along a selected plane intersecting the boat model. It highlights how the water accelerates around the bow and flows along the hull, offering insight into the hydrodynamic performance of the design. Areas of higher velocity indicate regions of reduced pressure, which are critical for understanding drag forces and optimizing the shape for better flow efficiency. This visualization helps validate the effectiveness of the hull form in reducing resistance and improving movement through water.

3. Pieces Designed

In this part of the mechanical design, the focus is placed on the detailed modeling of the boat's structure, particularly the hulls and their rear sections. Special attention is given to the front of the hull (coque), as its shape plays a crucial role in hydrodynamic performance. The design aims to ensure both stability and smooth water penetration. Several studies have been conducted to refine this form before finalizing the 3D model.

3.1 Hull head

The front part of the hull is meticulously engineered to minimize water resistance and enhance the boat's efficiency in navigating through water. Drawing on principles of streamlined marine structures, the pointed and curved geometry of the bow optimizes hydrodynamic flow, reducing wave-making resistance and minimizing splash, as described by Molland (2011). This design improves stability by maintaining a balanced center of buoyancy and lowers the energy required for propulsion. The 3D model of the boat's bow, developed using SolidWorks, is presented in the figure below

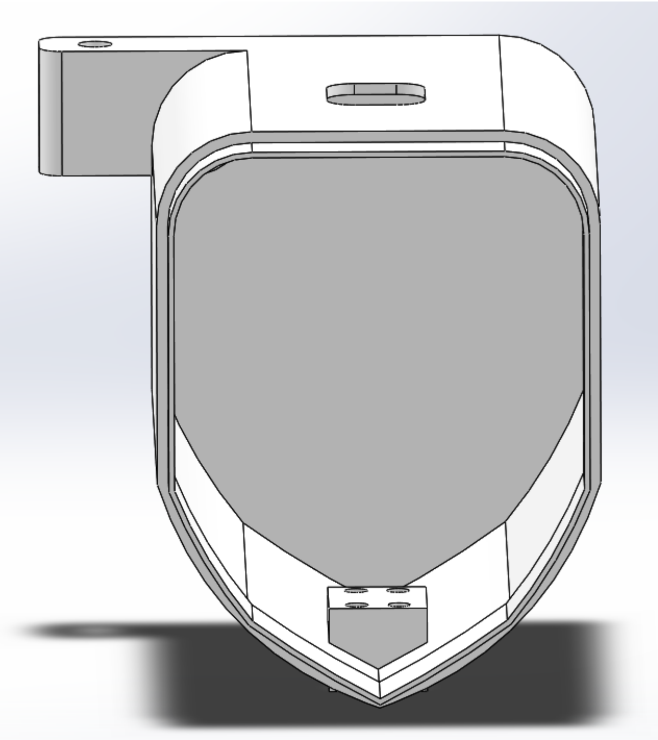


Parameter	Value
Dimension	L=20cm W=15,8cm H=20,8cm
Longitudinal Curvature Radius	18,2cm

Transverse Curvature Radius	29,4cm
Bow length	17cm

3.2 Hull Back

The rear section of the hull is a critical component, engineered to support the net and motor assembly, ensuring operational efficiency and structural integrity. The design features a reinforced platform with a curved trailing edge to optimize water flow and reduce drag during propulsion, while providing a stable mounting surface for the motor. An A-shaped transition, extending from the middle hull, enhances the liaison between the midsection and rear, improving load distribution and hydrodynamic continuity. This configuration supports dynamic loads from the net tension and motor, as validated by the integrated design.



Parameter	Value

Dimension	L=20cm W=15,8cm H=20,8cm
Support motor	4 screw holes(3*4 diameter)
Support net	1 screw hole (8mm diameter)