STAR-CCM+ TRAINING MARINE SIMULATION CASES

Enrico di Capua

Marine online trial
Simcenter STAR-CCM+

OVERVIEW

Objective: CFD training using Siemens STAR-CCM+

AGENDA

- STAR-CCM+
- Case 1 – Hydrodynamic Performance
- Case 2 – External Aerodynamics
- Case 3 – Hydrofoil Design Exploration
- Case 4 – Self-Propulsion
- Conclusions

WHAT IS STAR-CCM+ ?

- A multiphysics Computational Fluid Dynamics (CFD) software by Siemens
- Provides simulation of fluid flow, heat transfer, and solid mechanics
- Handles complex geometries and realistic operating conditions
- Used by leading companies to accelerate innovation

CASE 1 HYDRODYNAMIC PERFORMANCE

PURPOSE

The simulation was carried out to assess hull safety and comfort by verifying that wave slamming does not generate vertical accelerations above 1 g.

KEY MODELS

- Implicit Unsteady
- RANS K-Epsilon
- Volume of Fluid (VOF)
- Overset Meshing
- Segregated Flow Solver
- VOF Waves – Fifth Order

WAVE PROPERTIES

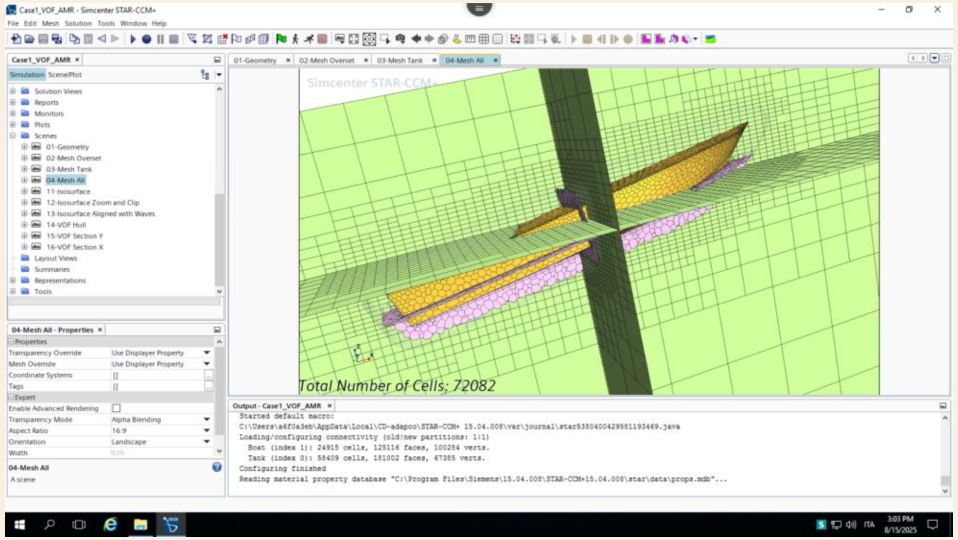
- Oblique waves at 45 degrees onset to the boat
- Wave height: 0.5m
- Water depth: 100m
- Wave period: 4s

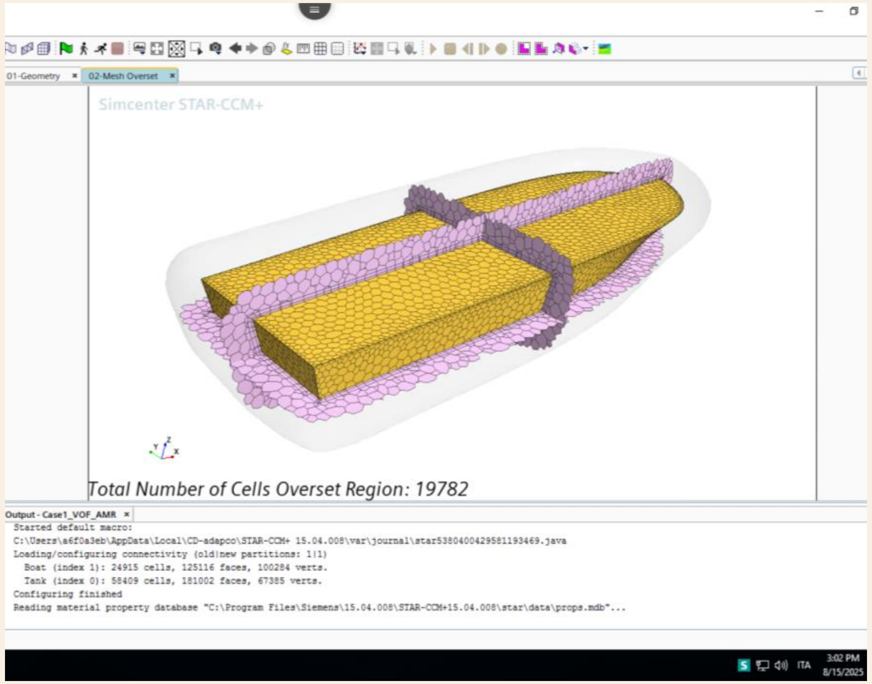
BOAT

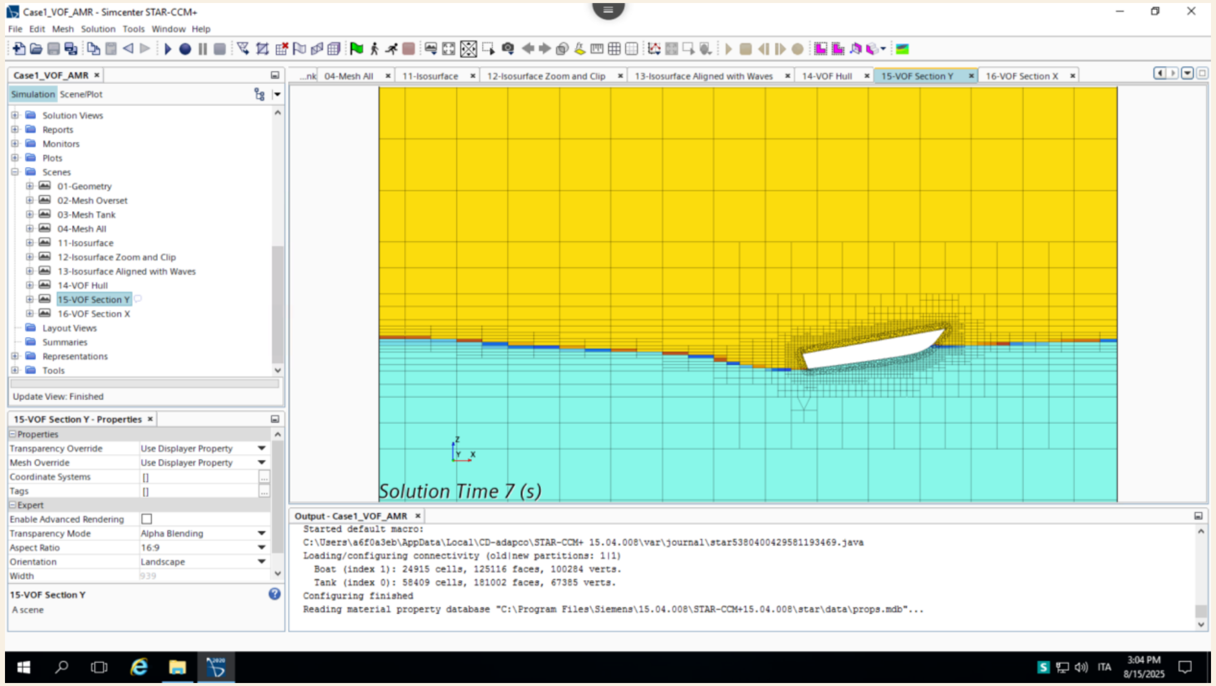
- Velocity: 20 knots
- Mass: 2500 Kg
- Dimensions: 5.6m x 1.8m x 0.6m

PROCEDURES

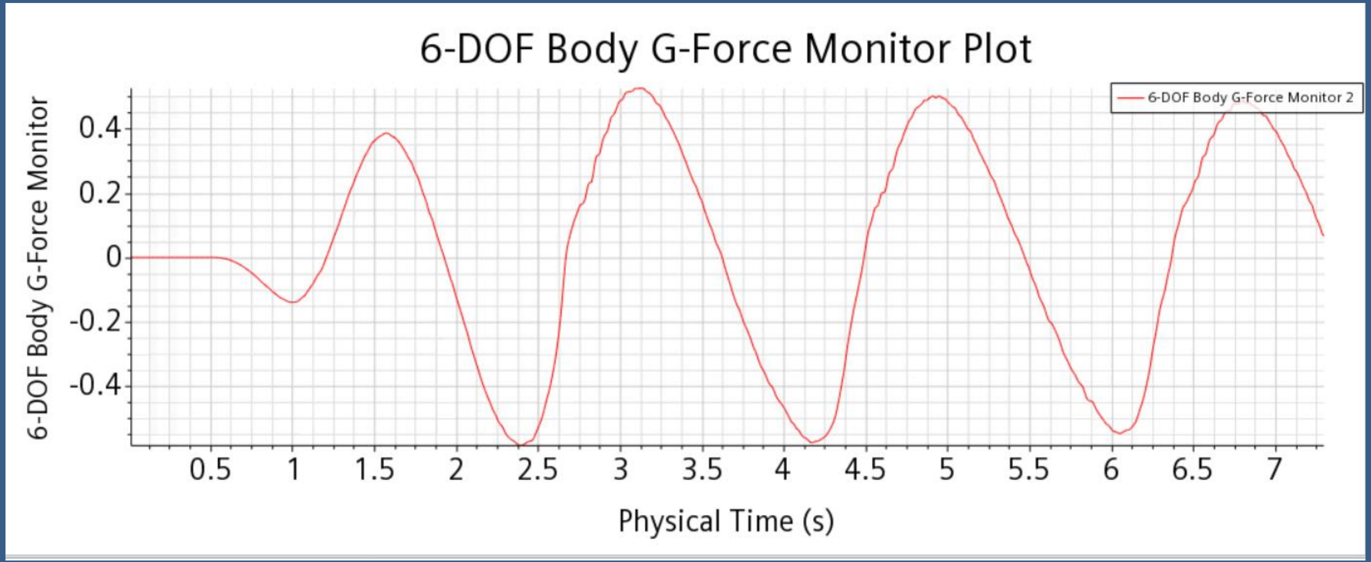
- The hull geometry was imported and positioned in oblique waves at 45°.
- An overset mesh with Adaptive Mesh Refinement (AMR) was applied to capture the free surface motion
- 6DOF model enabled realistic hull motion (roll, pitch, heave).







RESULTS



1. The boat experienced moderate roll (±3.8°), pitch (–12.5° to +0.2°), and a maximum heave of 0.5 m

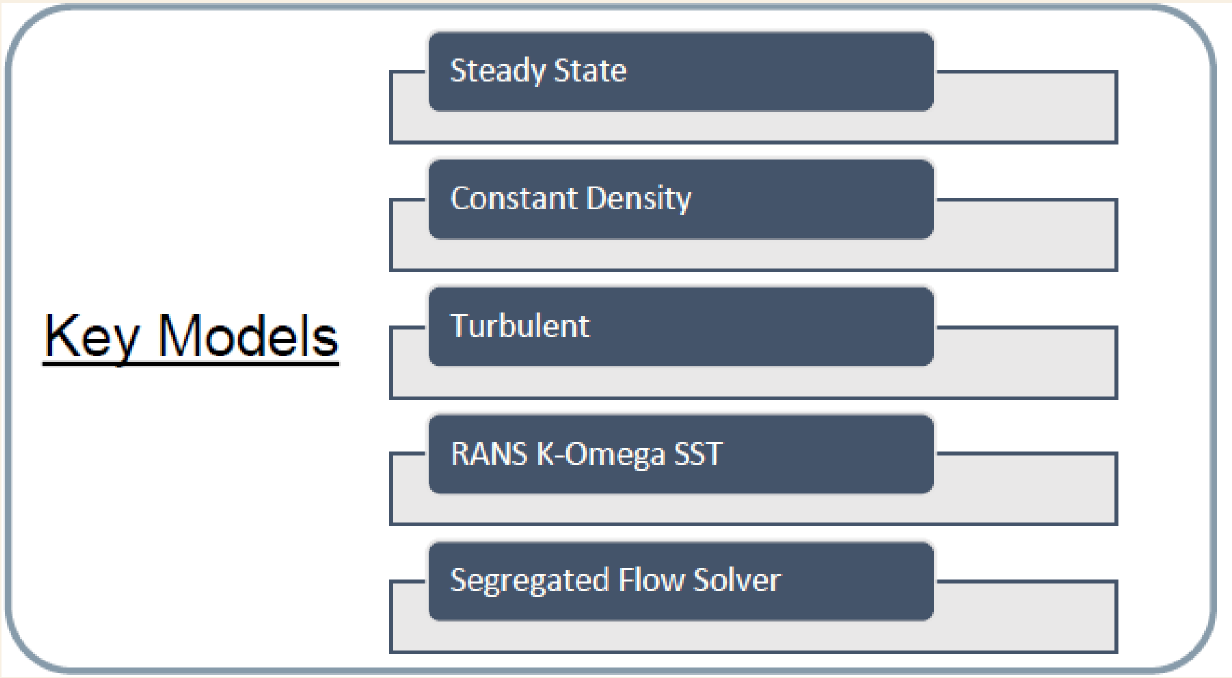
2. The peak vertical acceleration was 0.58 g, remaining safely below the 1 g limit

CASE 2
EXTERNAL AERODYNAMICS

PURPOSE

The simulation aimed to ensure that the velocity in the vertical direction on the rear deck of the vessel did not exceed +/- 1.80 m/s

KEY MODELS



WIND CONDITIONS

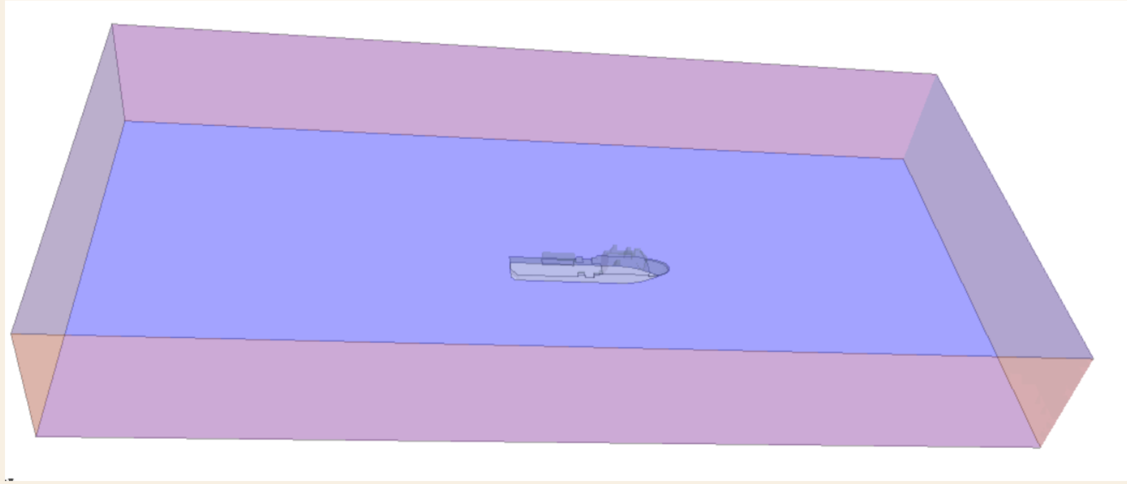
- Wind speed: 10mph
- Wind angle: 15 deg

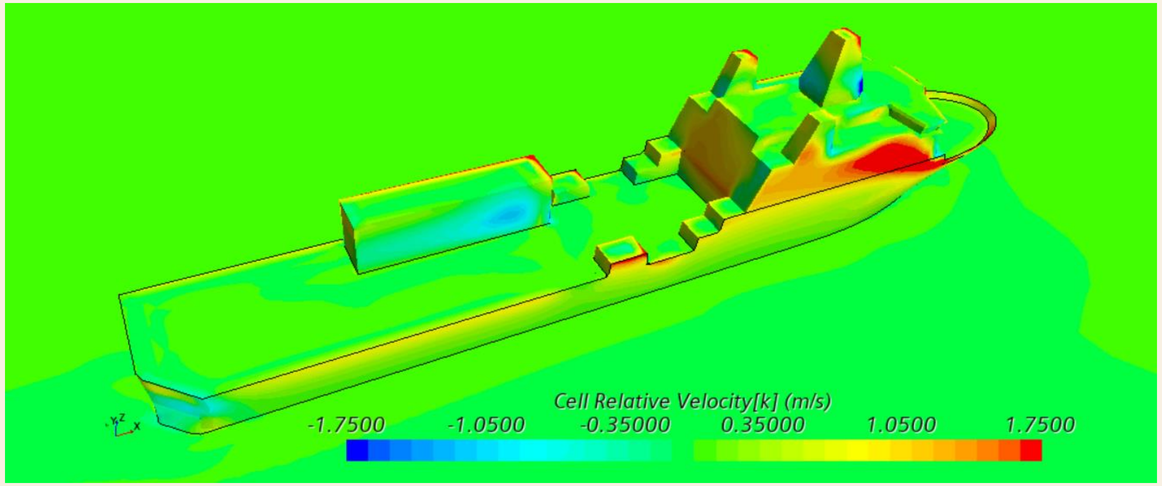
BOAT

- Dimensions: 200m x 35m x 55m

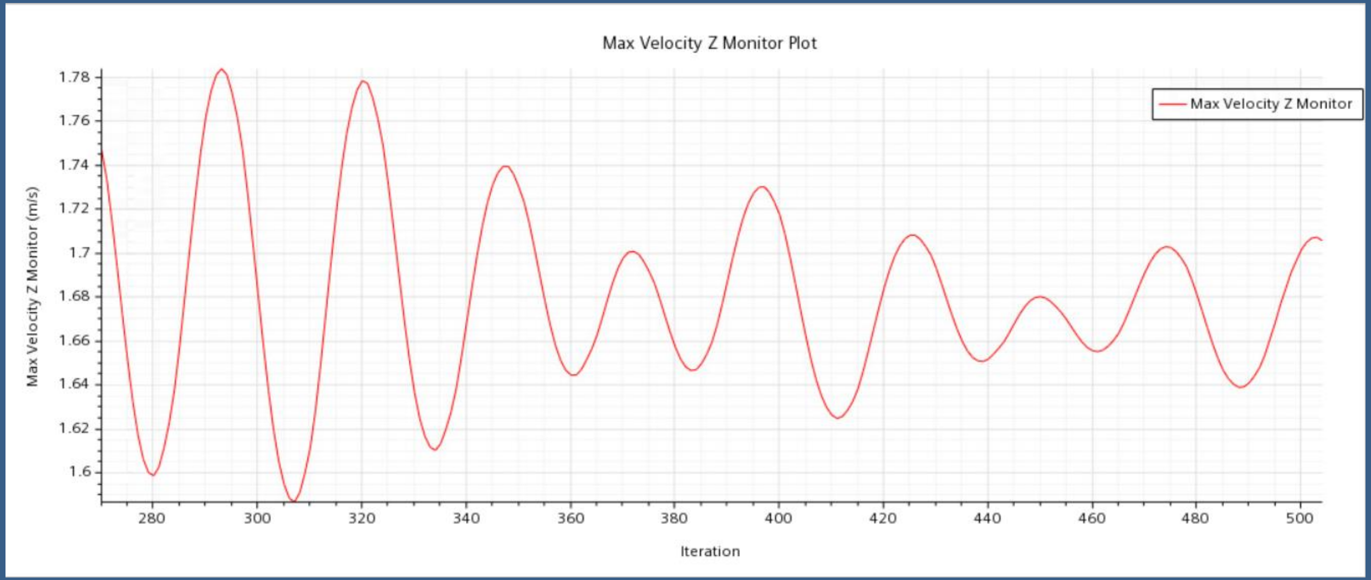
PROCEDURES

- External flow domain created with refined boundary layer mesh
- Steady-state CFD model applied to capture pressure and velocity distribution
- Post-processing focused on drag coefficient and flow visualization





RESULTS



1. Minimal fluctuation therefore the simulation is stable and converged
2. The maximum vertical velocity has converged to 1.7m/s
3. it is clear that the entire rear deck of the MRV is within the safety margin of 1.80m/s in the vertical

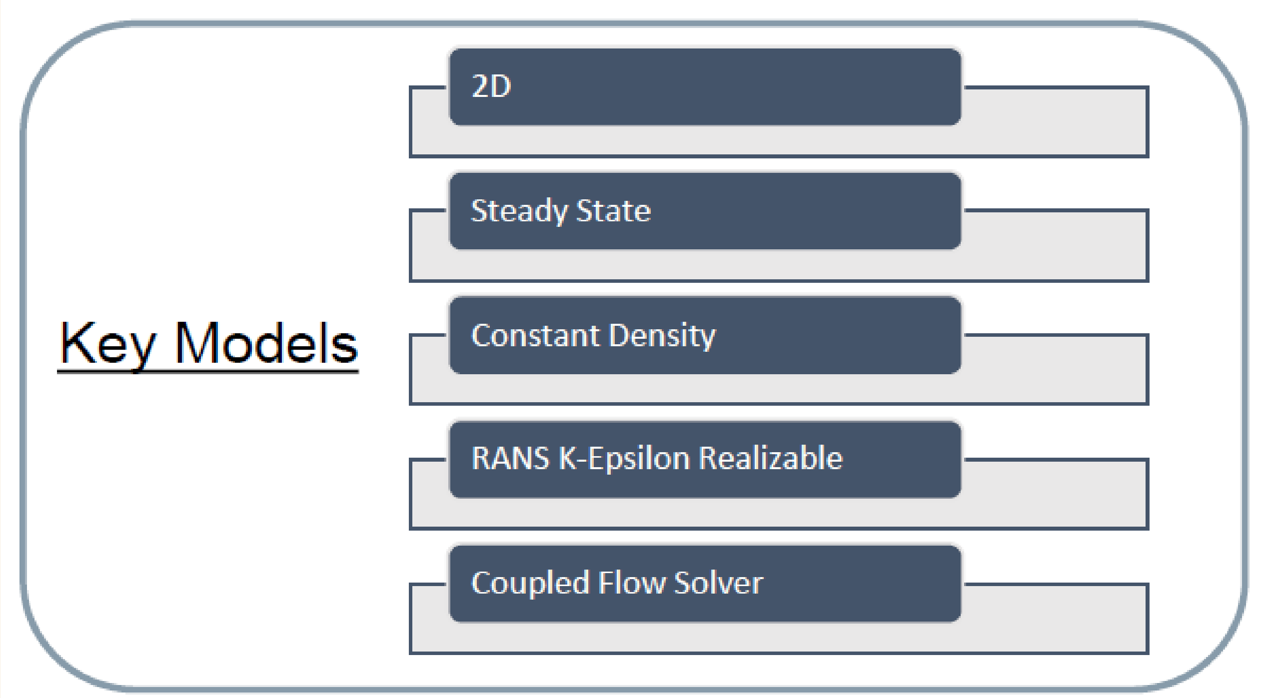
CASE 3 HYDROFOIL DESIGN EXPLORATION

PURPOSE

The simulation assessed hydrofoil performance to balance lift and drag for efficient hydrodynamic design. Objective:

- Lift Coefficient >= 0.3
- Minimize Drag Coefficient

KEY MODELS



DESIGN AND BOUNDARY

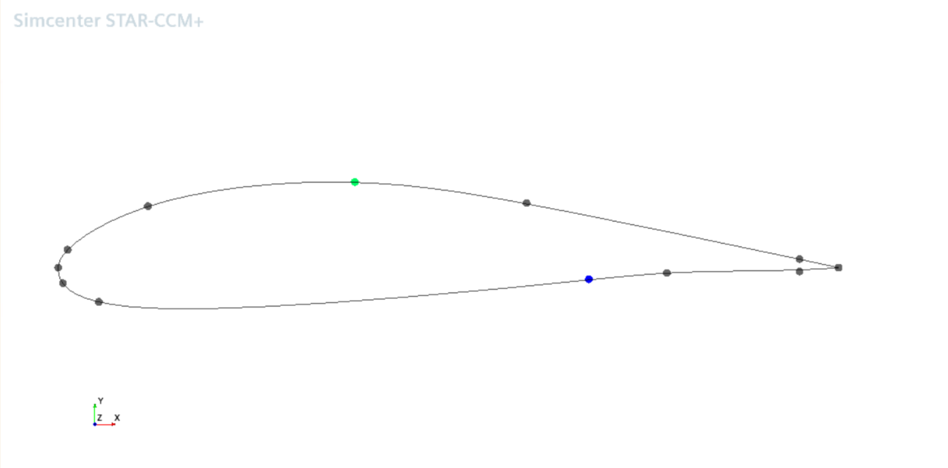
- Constant angle of attack: 0 deg
- Modifying 2 points on the hydrofoil profile
- Water velocity: 10 m/s
- Angle of attack: 0 deg

HYDROFOIL SHAPE

- Thickness: 0.19 m
- Chord length: 1 m

PROCEDURES

- Hydrofoil geometry modeled under realistic flow conditions
- Section planes and pressure contour visualizations created
- Parametric study performed to explore performance variations



Output Table - All

Design#	Name	State	Cdx100	Cl	Drag	Lift	NIters	Signed Distance Constraint-Down	Signed Distance Constraint-Up	Total Solver Elapsed Time	Performance
17	Design 17	✓	0.828951206...	0.310637538...	424.8374931...	15920.17387...	522.0	-0.009162713289316919 m	-0.013222494850840807 m	43.339999999999998 s	-0.96447461...
10	Design 10	✓	0.830276654...	0.326116648...	425.5167853...	16713.47822...	523.0	-0.01535123842184394 m	-0.01324382671358385 m	42.590000000000003 s	-0.96601675...
19	Design 19	✓	0.834354765...	0.272835388...	427.6068174...	13982.81364...	547.0	0.003505519939932866 m	-0.01309785297825097 m	44.073999999999984 s	-82.9614435...

RESULTS

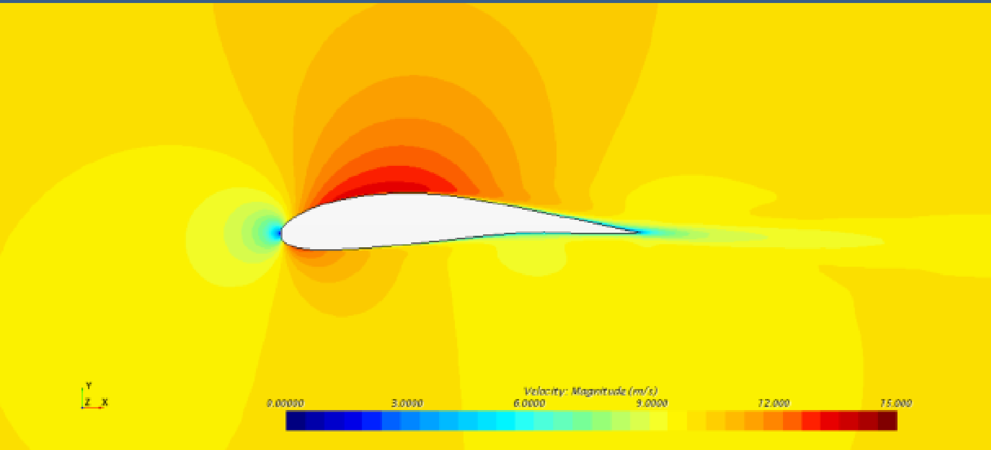
If we take a look at our starting hydrofoil, Design 1:

- Cdx100 = 0.8595
- Cl = 0.3869

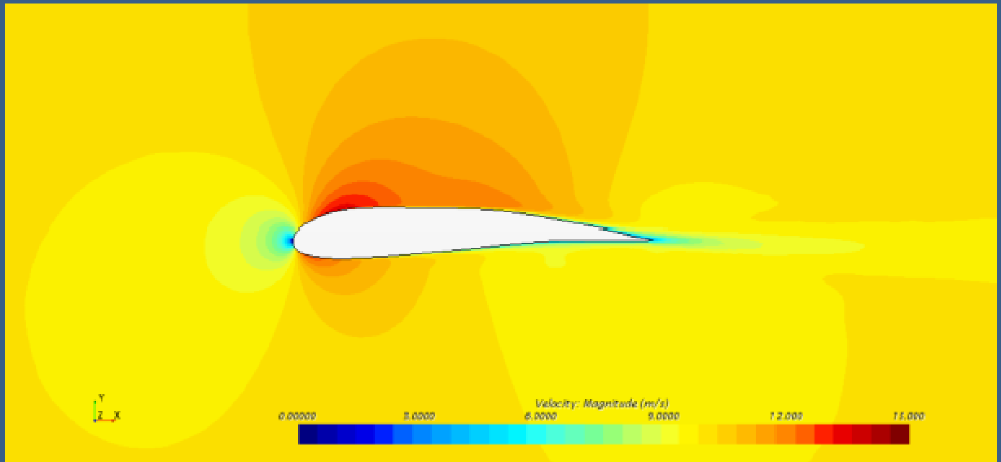
In comparison our best hydrofoil design, Design 17:

- Cdx100 = 0.8290
- Cl = 0.3111

This optimization has reduced drag 3.6% while keeping a minimum Cl of 0.3 in just 25 iterations.



Design 1: Velocity magnitude scene shows a generous amount of lift generated due to an increase in local velocity



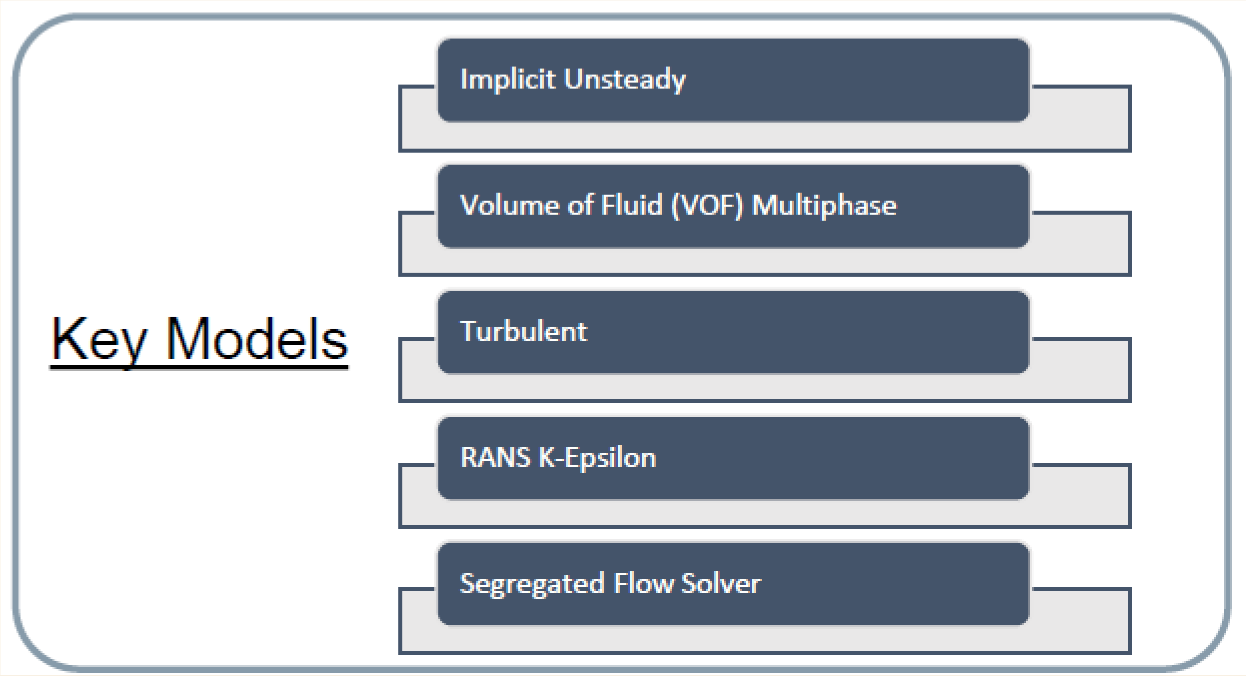
Design 17: Velocity magnitude scene shows lower amount of drag generated due to a thinner hydrofoil profile section

CASE 4
SELF PROPULSION

PURPOSE

The simulation investigated coupled hull-propeller interaction to analyze propulsion efficiency

KEY MODELS



CONDITIONS

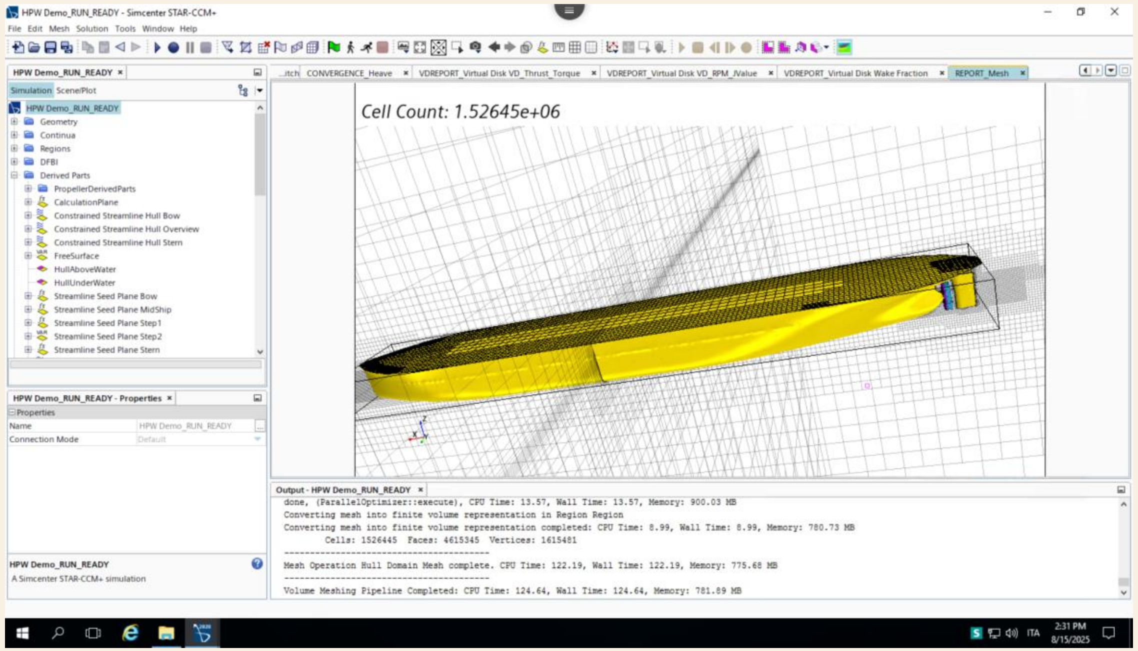
- Air Temperature: 26.5C
- Type of water: Sea
- Sea density: 1023 kg/m 3
- Ship speeds: 9.3, 11.6, 13 knots

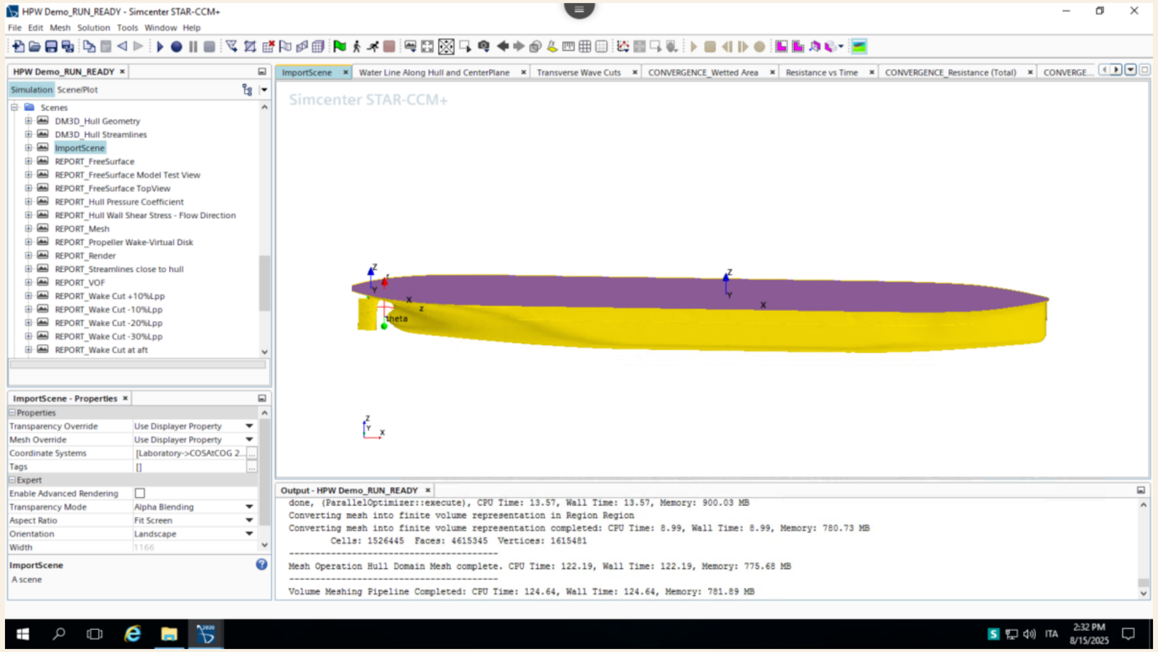
SHIP

- Dimensions: 150m x 31m x 23m

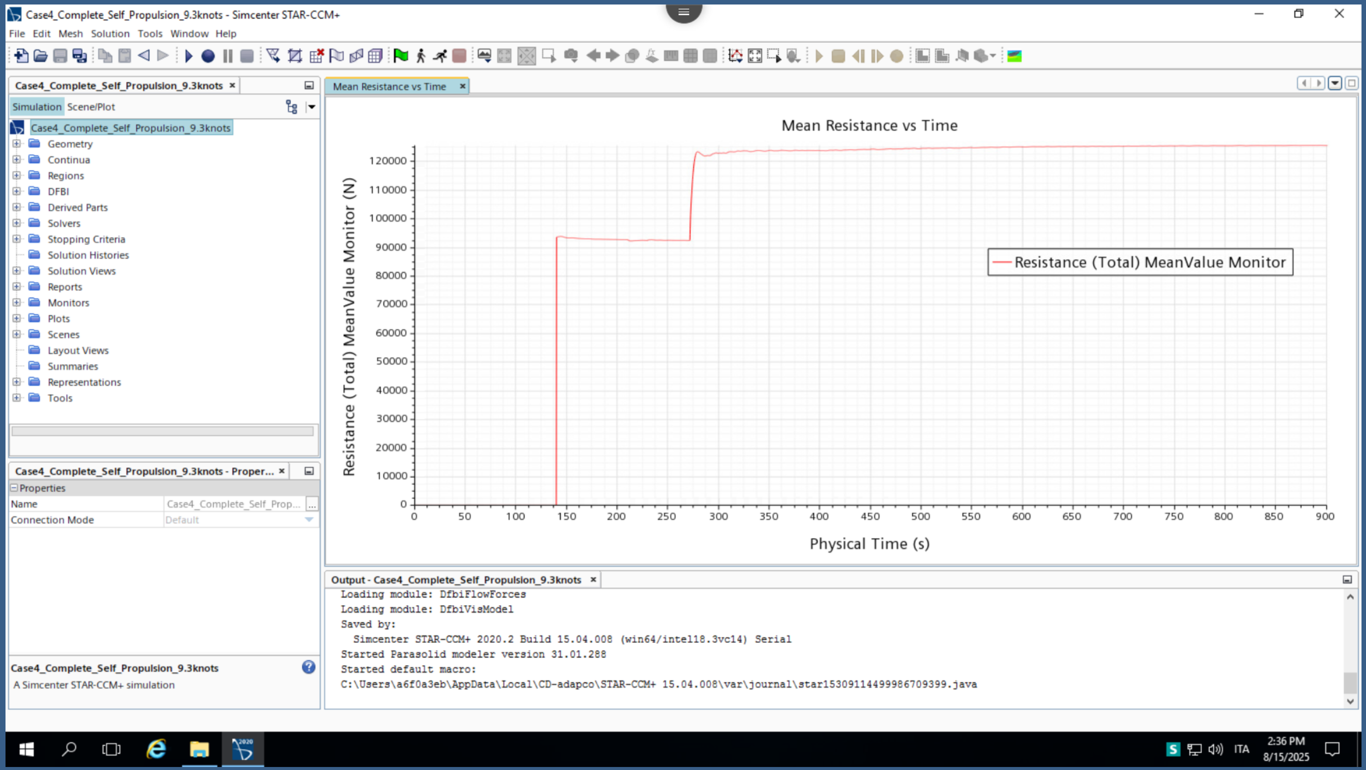
PROCEDURES

- Propeller modeled using an overset mesh to capture rotating motion
- Coupled CFD simulation performed with hull and propulsion system





RESULTS



1. Thrust curves demonstrated propeller effectiveness across operating points
2. CFD analysis confirmed reliable performance for integrated hull–propeller system.

CONCLUSIONS

- Completed four marine CFD simulations using Siemens STAR-CCM+.
- Each case explored a different aspect of marine design: hydrodynamics, aerodynamics, hydrofoil performance, and self-propulsion
- Gained hands-on experience in:
  1. Generating overset and adaptive meshes
  2. Applying 6DOF models and monitoring hull motion
  3. Post-processing results and extracting performance insights
- Developed a solid understanding of CFD workflows and marine engineering analysis, enhancing technical and visualization skills for future projects.

THANK YOU

Enrico di Capua - enricodicapua6@gmail.com

www.linkedin.com/in/enrico-di-capua

https://bit.ly/portfolio-enrico-di-capua