Un manned surface vehicles construction and simulation

Authors': Othman Ismail. Karim said. Mahmoud Khalid

Department of Ships and Submarine, Military Technical College, Egypt othmanesmaiel208@gmail.com

Supervisor: Ibrahim elsherif, Maj.Gen.Dr.pro & Alaa Eldin elhadad, Brig. Gen. Dr

***Abstract** This paper considers the design of a low-cost and modular unmanned surface vehicle (USV) used for marine environment monitoring or survey. The hull chosen is a catamaran type since it improves stability against waves, and the shape of ship is exploited to reduce the resistance. The mechanical design is based on a two-hull construction because of its low risk of capsizing in rough water. In order to satisfy requirements of USV mission, it should have acceptable speed, endurance, maneuverability and stability under inappropriate conditions; furthermore, enough payload capacity. The computational fluid dynamics (CFD) and max surf modules are used to predict calm water resistance for the floating hull through calculations. To design the floating hull and estimating the hydrodynamics performance of it, calculations are carried out using three different mesh sizes in the range of 0.7 to 0.3. The fine grid gives most appropriate results because convergent results are obtained as the mesh size decreased. Using the designed model, a prototype of unmanned surface vehicles (USVs) was developed. The prototype will be tested to prove that.*

KEYWORDS: Autonomous surface vehicle, unmanned surface vehicle (USV), Calm Water Resistance, CFD.

I. INTRODUCTION

Unmanned Surface Vehicles (USVs): Autonomous Surface Vehicles (ASVs) that operates on refer to any water surface with no crew on board., The idea of an Autonomous Surface Vessel (ASV) started nearly half a century ago during the World War II and since then these vessels have been following the technology evolution and started to be used not only for military purposes but also for scientific research. Nowadays anyone can have an ASV and until now there are no mandatory rules regarding the usage of these drones. Scientists use them to perform a lot of different studies in the open waters., Unmanned Surface Vehicles
(USVs) are the most widely used in open water technique. It has attracted wide attention in recent decades as it tries to integrate monitoring with communication. [1]
The USVs also must able to carry various devices like battery, servo motor system for underwater propeller and the most important thing it must able to withstand with the rough sea wave condition., Survey applications have made use of a variety of hull forms like numerous catamaran type USVs have been developed to support academic interests.

II. Hull geometry

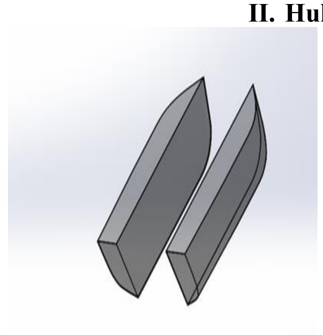


Fig 1: hull

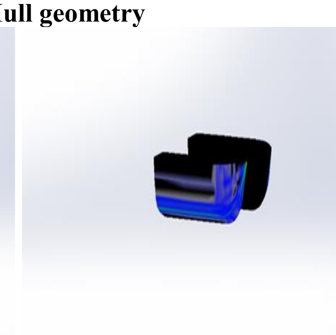


Fig 2: hull curvature

After designing the hull on Maxsurf then the hull is exported to solid work to evaluate the curvature and preparing the hull to be used in Ansys

III. Ansys fluent work flow

ANSYS Fluent solves conservation equations for mass and momentum. an additional equation for energy conservation is solved Procedure calculating the drag four hull on CFD

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t}+\nabla\cdot(\rho\vec{v})=S_m$$

Conservation of momentum in an inertial (non-accelerating) reference frame is described by

$$\frac{\partial}{\partial t}(\rho\vec{v})+\nabla\cdot(\rho\vec{v}\vec{v})=-\nabla p+\nabla\cdot(\bar{\bar{\tau}})+\rho\vec{g}+\vec{F}$$

The procedure of the Ansys is as follow

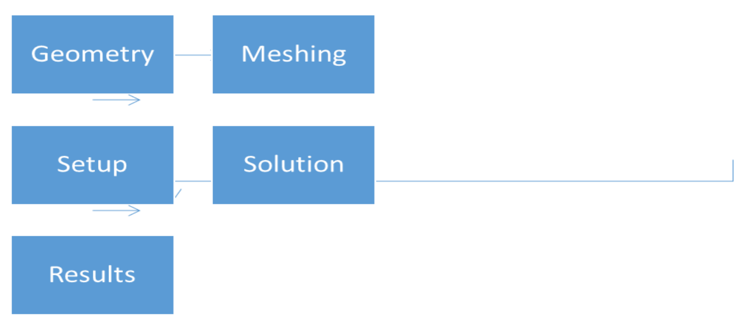


Fig 3:procedure

- Geometry and Computational Domain

In this step in all cases we defined the solid hull geometry and four domains of fluid in box shape around it -defining the type of fluids around the hull and its interface is not in this step it will be in setup Step-We subtracted the solid hull from the fluid domains maintaining the interface wall between them to define the drag force and resistance on it., The domain size had different effects in the resistance depending on the turbulence model used, showing that accurate data can be obtained even for a small length between the stern and the outlet. For the Shear Stress Transport (SST) turbulence model it was found that a domain of 1.2 L downstream, 0.5 L upstream, width of L and a depth of L/2+T gave good results

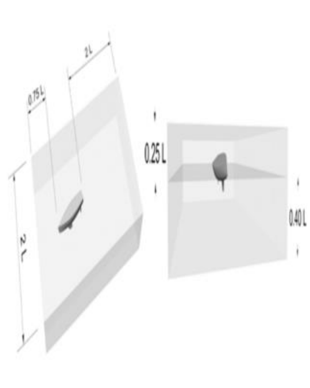


Fig 4:geometry computational domain

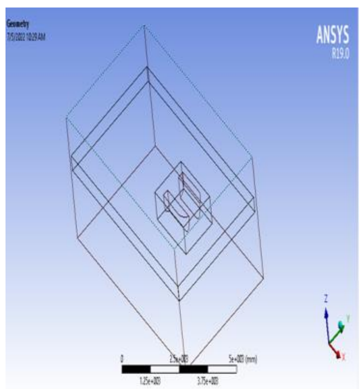


Fig 5:hull and computational domain

- Meshing

In this step we convert our problem from the physical domain to the computational domain. In our cases we generated structured hexadomain mesh which have the advantage of the high accuracy in solving step but have disadvantage with curved bodies and surfaces we could overcome this problem by using fine mesh around hull surface and the option "proximity and curvature" in mesh sizing options in Ansys meshing program. In this case we used mesh assemble with growth rate 1.2 by deferent mesh sizing in the four domain the farther the domain the coarser mesh size. we used four mesh sizes this the reason why we defined four fluid domains in the geometry step and it was done to generate less number of mesh elements and nodes with high quality concentrating on hull surface to define the resistance as accurate as possible with the available computational resources. geometry and meshing are the preprocessing step in CFD analyses

Physics Preference	Mechanical
Relevance	0
Element Order	Program Controlled
Sizing	
Quality	
Inflation	
Advanced	
Statistics	
Nodes	2522912
Elements	3542211

Fig 6:number of element

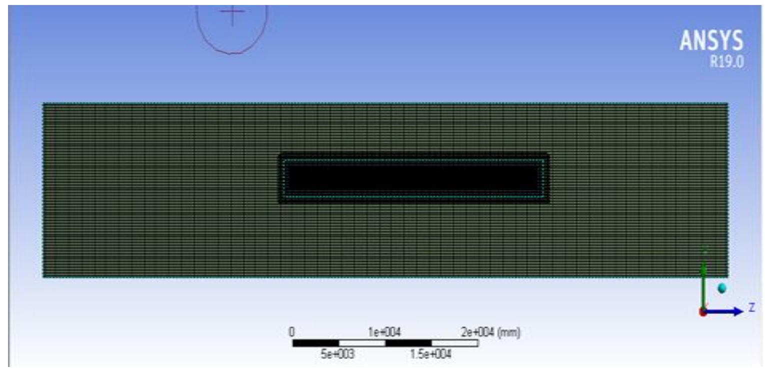


Fig 7:mesh .03

- Boundary Conditions and Solution Parameters

The fluent software is used for computations in this study. Since the motion of the free surface is governed by gravitational and inertial forces, therefore, gravity effects must be taken into account in boundary conditions. The inlet boundary condition upstream and the outlet boundary condition downstream are taken as inlet-velocity and pressure-outlet with open channel while the flow velocity is considered equal to the experimental velocity of the model. No-slip wall boundary condition is taken on the whole surfaces. Symmetry condition is invoked on the symmetric plane. Convergence of the solution is assessed by monitoring the residuals of continuity, velocity, turbulence, volume fraction and drag force. The residual convergence criterion is taken as 1e−07 [18] The model had to reach its own equilibrium, it is an inherent unsteady simulation and therefore a transient simulation was selected. Nonetheless the final equilibrium has a static behavior; once the equilibrium is reached the final position are constant.

- Drag Calculations

In this study; The calm water resistance was calculated by Versus velocity ranging from 2 m/s to 10m/s for mesh study

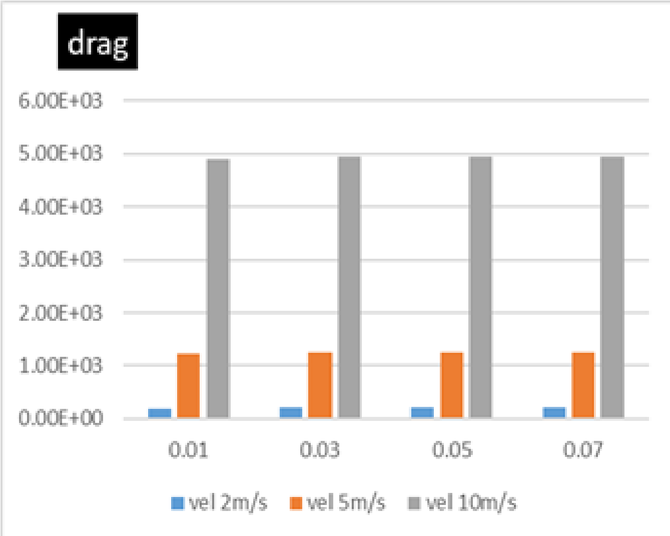


Fig 8:drag for different velocities and different mesh size

the water volume fraction of the hull is as follow

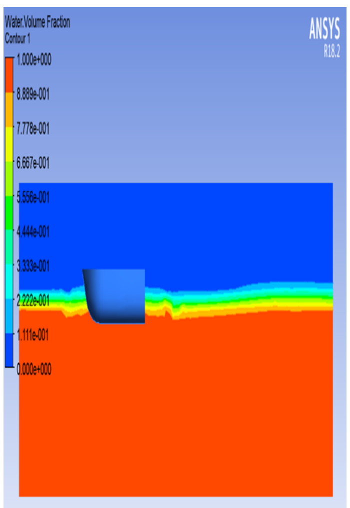


Fig 9:y-z plane water volume fraction

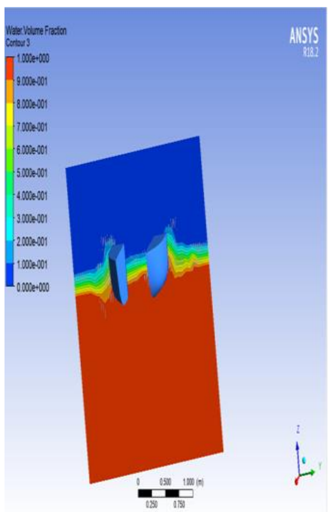


Fig 10:x-z plane water volume fraction

- For mesh size .03:
  - For velocity 2m/s

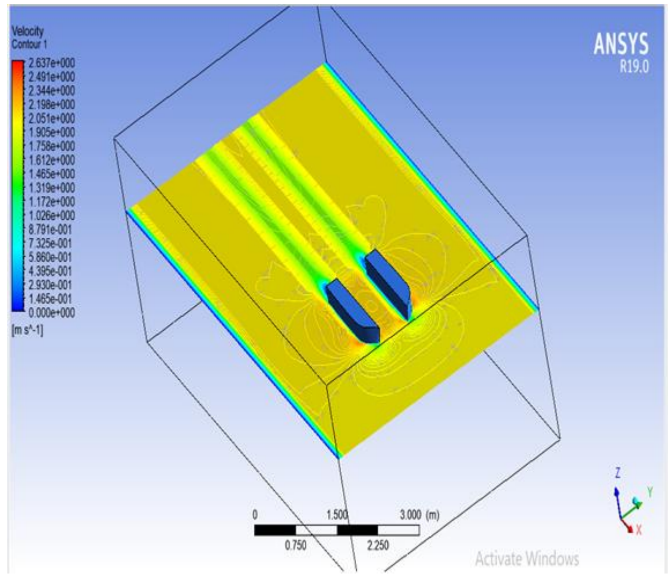


Fig 11:velocity contour for 2m/s

- For velocity 5m/s

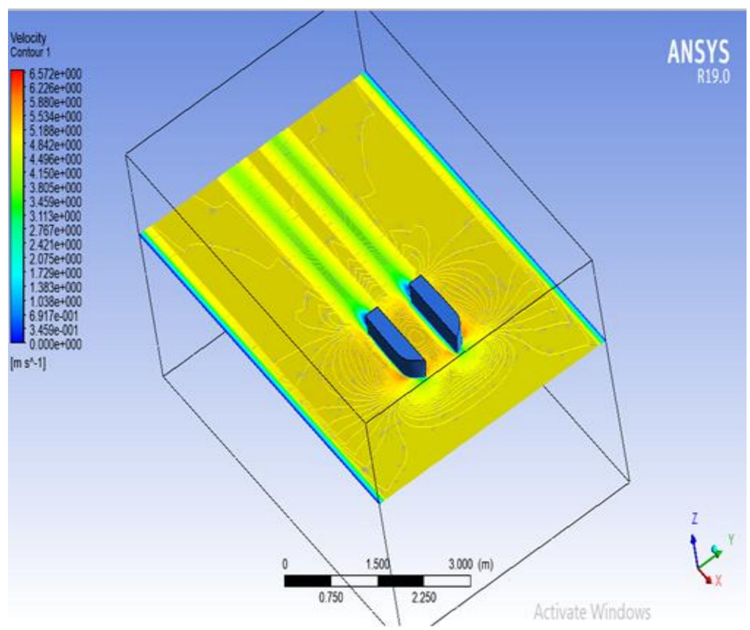


Fig 12:velocity contour for 5m/s

- For velocity 10m/s

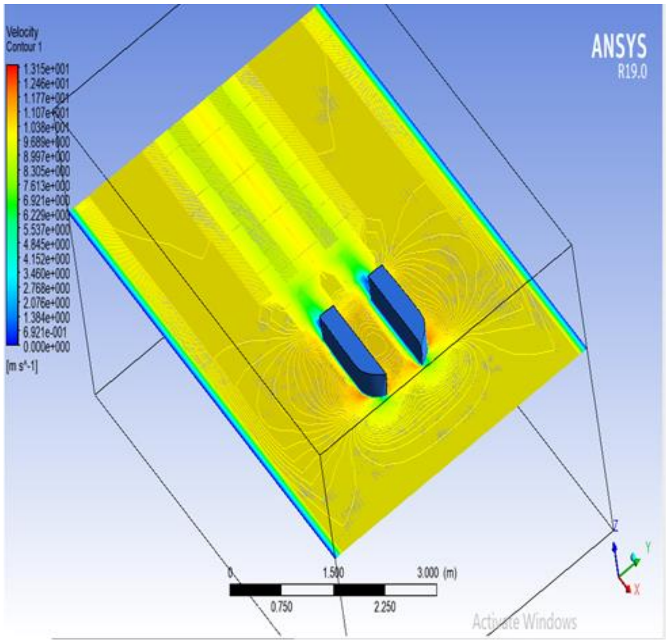
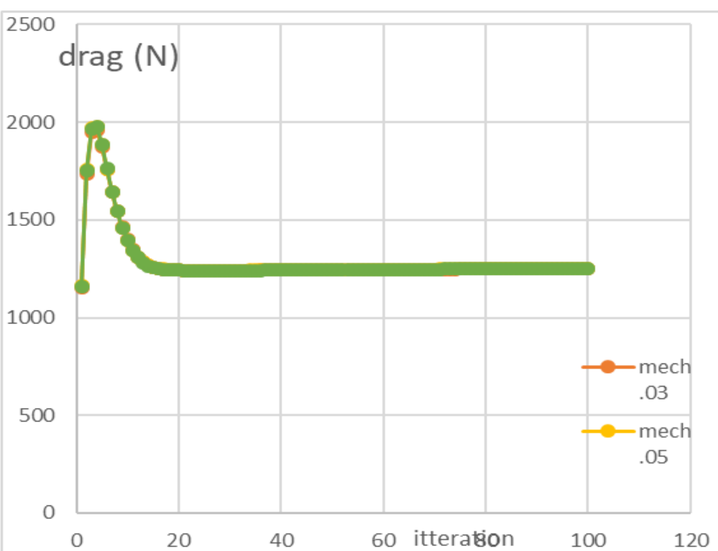


Fig 13:velocity contour for 10m/s

Fig 14:drag variation for diff. meshes

- Results of the drag

Four different mesh A comparison between three different meshes approaches has been done, showing that the data are converged for the different mesh



IV. Resistance prediction by Maxsurf Resistance

The velocity contour corresponding to the hull speed of range from 0 to 10 m/s

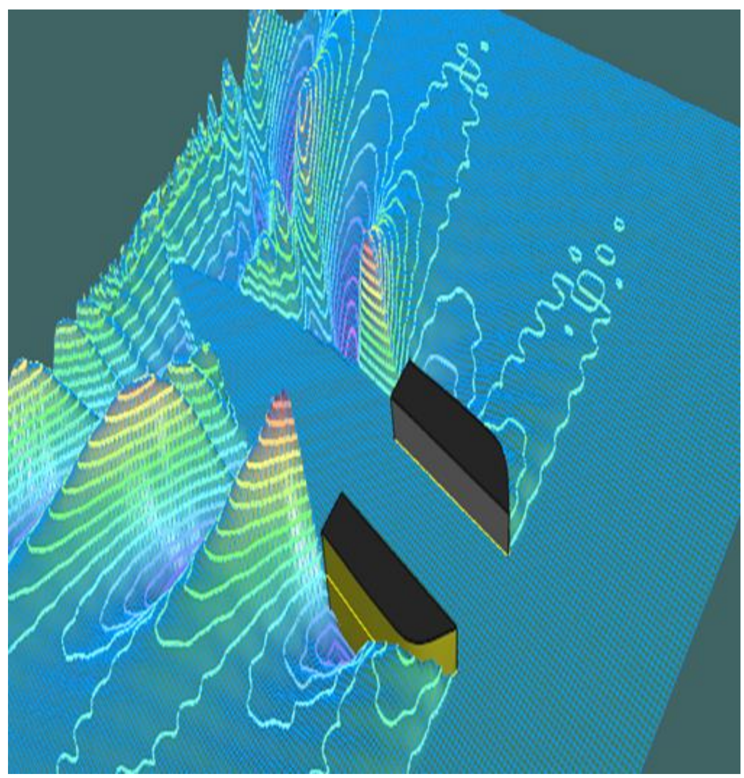


Fig 15:velocity contour Maxsurf resistance

The hull resistance of velocity ranges 0 to 10m/s using two methods slender body method and KR-barge

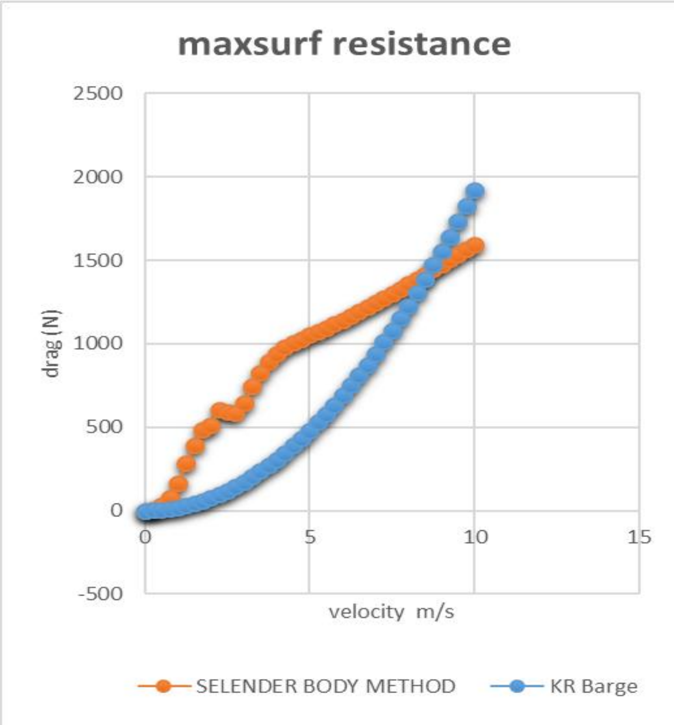


Fig 16:resistance by two method

The [Fig 17] shows the comparison between the drag results by Maxsurf (using two diff. method) and Ansys

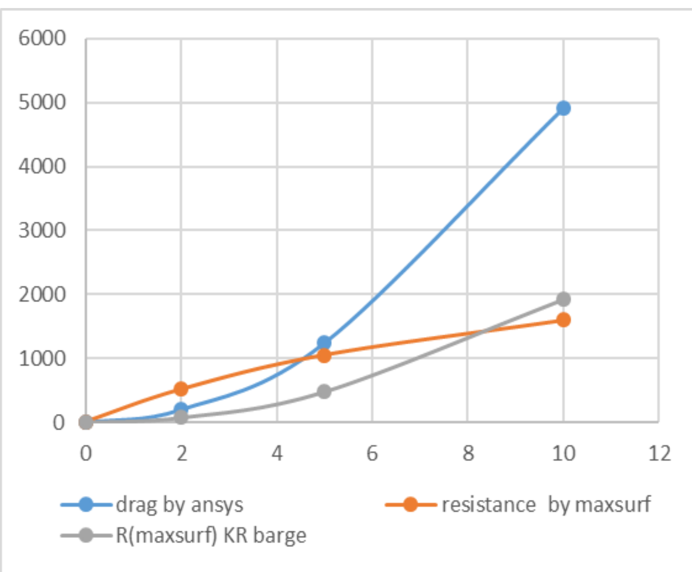


Fig 17:comparison between the drag by Maxsurf and Ansys

Conclusion

By comparison between the data given by Ansys and Maxsurf (slender body method &KR barge method) the data not converge so it's important to experimental testing of prototype to verifies the result so the future work is to production of the prototype and testing on towing tank and recorded data and making comparison with the previous data

References

[1] Resistance and Seakeeping Investigation for Optimization of the Floating Hull of Wave Glider

[2] Hydrodynamic modelling of hulls using RANSE codes

[3] Design and Implementation of Autonomous Surface Vehicle for Inland Water [Haluk BAYRAM1*]

[4] Ojeda, R.; Prusty, B. G. and Salas M.. (2004) "Finite Element Investigation on the Static Response of a Composite Catamaran under Slamming Loads". Ocean Engineering, Vol. 31. pp. 901-929

[5] Hitz G, Pomerleau F, Garneau ME, Pradalier C, Posch T, Pernthaler J, Siegwart RY, 2012. Autonomous inland water monitoring: Design and application of a surface vessel. IEEE Robotics & Automation Magazine, 19(1): 62-72.

[6] Lindemuth M, Murphy R, Steimle E, Armitage W, Dreger K, Elliot T, Hall M, Kalyadin D, Kramer J, Palankar M, Pratt K, 2011.Sea robot-assisted inspection. IEEE Robotics & Automation Magazine, 18(2): 96-107.