Journal of Naval Science and Engineering
2016, Vol. 12, No.2, pp.1-26
Gemi İnşa Mühendisliği/Marine Engineering

INVESTIGATION OF FLOW DISTRIBUTION AROUND A SUBMARINE

Abdi Kukner*, Adem DURAN, Tarkan ÇINAR*****

* kukner@itu.edu.tr, Faculty of Naval Architecture and Ocean Engineering, Istanbul Technical University
** admdrn@gmail.com, Faculty of Naval Architecture and Marine Engineering, Yildiz Technical University
*** tarkancinar66@gmail.com, Faculty of Naval Architecture and Marine Engineering, Yildiz Technical University

Date of Receive: 19.09.2016 **Date of Acceptance: 01.11.2016**

ABSTRACT

In this paper the flow field around a submarine has been investigated. Pressure distribution and its impact on submarine hull form have been studied. The accurate and efficient prediction of hydrodynamic pressure and forces on a submarine has been achieved by investigating the flow related to the interaction of the vertical flow shed from the sail and the cross-flow boundary layer of the hull. Therefore this study aims to simulate the flow field of a submarine by using finite volume method. Finite Volume Stress Analysis Method and k-ω turbulence model have been used to simulate turbulent flow past the submarine hull surface. A submarine hull with overall length of 80 meters and diameter of 10 meters was chosen. It has aft body length 11m and sail length 7m. The speed range of the submarine is 0 to 30 knots with 5 knots increments. Calculated pressure coefficients along the submarine hull are discussed to show the effect of the sail lateral position and the stern appendages. It is also discussed a Reynolds Averaged Navier-Stokes (RANS) code application in the design of an "Advanced Sail" for a submarine.

Keywords: Flow distribituon, submarine, pressure distribution, RANS.

ÖZ

Bu makalede, bir denizaltının gövdesi üzerine etkiyen akışkan basıncı ve kuvvetlerinin daha iyi anlaşılması için denizaltının etrafındaki akış dağılımı detaylı bir şekilde incelenmiştir. Denizaltıya etkiyen hidrodinamik basınç ve kuvvetlerin doğru ve etkili bir şekilde tahmin edilebilmesi için denizaltının yelkeni üzerindeki düşey akış ve teknenin sınır tabakasındaki enine akışının etkileşimi ile ilgili olan akış incelenmiştir. Bu çalışmada Sonlu Hacim Yöntemi (FVM) ve Hesaplamalı Akışkanlar Dinamiği (CFD) kullanılarak denizaltı etrafındaki akış alanı simüle edilmiştir. Sonlu Hacim Stres Analiz Yöntemi ve k-ω türbülans modeli kullanılarak denizaltının gövdesinin yüzeyini takip eden türbülanslı akışın simülasyonu yapılmıştır. Boyu 80 m, eni 10 m, kıç kuyruk uzunluğu 11 m, yelken uzunluğu 7 m olan bir denizaltı modeli seçilmiş ve RhinoCeros^{TM} programı kullanılarak çizimi yapılmıştır. Denizaltının hızı 0'dan başlayarak 30 knota kadar 5 knot arttırılmak suretiyle hesaplamalar yapılmıştır. Günümüzdeki güçlü bilgisayarların getirdiği kolaylıklardan dolayı akışkan probleminin tam olarak Navier-Stokes denklemiyle sayısal hassas çözümü geniş bir yelpaze alanı içerisinde yapılabilmektedir. Denizaltı gövdesi boyunca hesaplanan basınç katsayıları yelken ve kıç takıntılarının etkisini göstermek için ele alınmıştır. Ayrıca, bir denizaltı için "Gelişmiş Yelken" tasarımında RANS (Reynolds Averaged Navier-Stokes) kodunun uygulaması ele alınmış ve irdelenmiştir.

Introduction

A submarine is a vessel capable of independent operation underwater. It is used as a surface naval weapons platform or as a tool of exploration and recreation. Their stealth plays an important role in a modern naval force. Therefore submarine is a warship with a streamlined hull design to operate completely submerged in the sea for long periods, equiped with a periscope and typically armed with torpedoes or missiles. Most large submarines have a cylindrical body with hemispherical (and/or conical) ends and a vertical structure, usually located amidships having navigation and other equipment devices as well as periscopes. Sometimes known as the conning tower. This vertical structure is called "sail" in U.S. Navy, "fin" in European Navies.

The propeller of submarine, vertical and horizontal control panels are located at stern. As the thrust is generated, water pushes over the planes, creating an upward or downward force that helps the sub gradually surface or dive. The fins can be tilted to change the angle of attack at which it climbs or dives.

The starting point of all scifientic studies is a literature survey to understand the status quo of the investigated topic. It is important to understand the reasons for the shape of submarines at different stages of their development and why changes were made. To neglect full scientific studies would be a serious mistake in the design of any future replacement submarine.

In submarine hydrodynamics, turbulence and vortex dynamics play an important role. The classic picture of turbulence starts from a sequence of bifurcations in a “smooth” flow, each of which introduces flow structures of smaller and smaller scales. Designers had begun to change nose and tail cone shape to improve the performance of submarine at operational speeds. Other major sources of resistance may be improved. The establishment of the detail performance of a submarine can be started by using computational fluid dynamics to obtain pressure distribution and to calculate the drag characteristics which will serve as the comparative foundation for any new design. All features affecting the shape of submarine are discussed including the boundary layer, laminar flow, transition, turbulence and separation and how the flow should be as quiet and smooth as possible. At the beginning the pressure distribution around submarine body without sail, and appendages were investigated. The next step was; the sail, tailplanes and foreplanes were added to obtain pressure distribution around the submarine and to observe how effects and changes in flow distributions. Design looks like a jigsaw puzzle where altering one piece requires alterations in all surrounding features to make a workable complete design. It is clear that scientific studies has to be a starting point for any future submarine design. A review of relevant literature has been completed which covered priorities in design and showed how enhancement of one feature interacts with other features and may even result in an overall loss of performance despite the perceived advantage of the enhanced feature. Hydrodynamic aspects were then discussed starting with the shape and reasons what should be the beam-to-depth ratio (B/D) to give minimum resistance as possible.

As it is well known, flow around submarines is exceedingly complicated, even at simple flow conditions, and the need to reduce submarine signatures from flow-induced noise put high demands on the computational model. Most of the boundary layer on a submarine is predominantly turbulent because of the high Reynolds (Re) number, which typically is encountered in ship hydrodynamics. At the bow, the flow is usually laminar, but rapidly undergoes transitions into a fully turbulent boundary layer, which often makes it reasonable to assume a fully turbulent boundary layer along the entire hull. The boundary layer is further affected by pressure gradients (mainly around the bow and the stern) and the hull curvature, potentially causing a vortex separation usually resulting in distortion of the propeller inflow.

Prediction of Submarine Resistance

Whenever a body is placed in a flow, the body is subject to a force from the surrounding fluid. In general, the force acting on a body is resolved into a component D in the flow direction U and the component L in a direction normal to U. The component D is called drag and L is called lift. The most important difference between the resistance of a surface ship (or submarine on the water surface) is that for a deeply submerged submarine will not have wave resistance. Therefore the submerged submarine resistance will sum up total skin friction and total submerged pressure. Skin friction drag acts tangentially at the surface and is proportional to the wetted surface.

The total pressure has form resistance or form drag and induced resistance or induced drag. The form drag is the viscous pressure resistance due to the shape of the submarine. The induced drag is the resistance caused by lift. This could be on appendages that are generating lift due to misalignment with the flow, or to the hull, that may be generating lift due to symmetry.

The resistance of a submarine can be determined either by model testing, or by Computational Fluid Dynamics (CFD). In this paper, CFD techniques have been used to estimate the resistance of the deeply submerged submarine. As the resistance of a deeply submerged submarine is dominated by the frictional component, there are a number of difficulties with this, in particular the choice of empirically based turbulence model. However, in principle it is possible to use CFD to obtain results at full scale Reynolds numbers, something which is not possible using model experiments [1]. There is one of the current complications with CFD is that there is no standard method for predicting submarine resistance. This is largely because both computing power, and CFD techniques, are developing rapidly. Thus, great care needs to be taken when investigating the effect of the change in resistance due to a change in hull shape.

Numerical Model

The use of computational tools to evaluate submarine flows have been tremendously increased over the last decade since the capacity and speed of computers were raised. Thus the applications of Computational Fluid Dynamics (CFD) to the naval industry was guiding the design of submarine. In view of these developments, CFD can offer a cost-effective solution to many problems in underwater vehicle hull forms. However, effective utilization of CFD for naval hydrodynamics depends on proper selection of turbulence model, grid generation and boundary resolution. The most common turbulence modeling approach of today is RANS, which is based on Reynolds Averaged Navier-Stokes equations and is often adopted in traditional Computational Fluid Dynamics (CFD). The first fully appended submarine RANS calculation was done by Gorski et al [3] for the submarine configuration, which was extensively measured to provide a data-base to test CFD methods. The dependent variables are divided into a mean part and a fluctuating component representing deviations from this mean. The advantage of RANS is however that the approach is fast, and it is available in most CFD codes. In particular, with the advent of parallel computational capabilities, viscous RANS simulations have seen a larger role in predicting these flow fields.
In this study, Reynolds Averaged Navier Stokes (RANS) equations and continuity equation for mean velocity of the unsteady, incompressible fluid have been used as governing equations in order to determine the mean cartesian flow field, U_i, and the mean pressure (P) of the water around the hull. The well known SST (Shear Stress Transport) k-ω model have been considered to simulate the turbulance flows.

$$\frac{\partial U_i}{\partial x_i} 0 \quad (1)$$

$$\rho\frac{\partial U_i}{\partial t}+\rho\frac{U_i U_j}{\partial x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left\{\mu\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\right\}-\rho\frac{\overline{\partial u_i' u_j'}}{\partial x_j}+f_i \quad (2)$$

where f_i represents external forces. The influence on turbulence on the mean flow is given in equation represents external forces. The influence on turbulence on the mean flow is given in equation (2) by the Reynolds stress tensor $\overline{\rho\left(u_i' u_j'\right)}$. There are many turbulence models to provide solutions to the Reynolds stresses.

The k - ω model is well-suited for prediction in the vicinity of the wall, while the k - ε model is for the remaining area near the boundary region. The k- SST-model is using blending functions to be able to use the k-ω model near the wall and the k-ε in the free stream and to get a smooth transition between them. Therefore it is a hybrid between the k- ε and the k-ω model. The SST k - ω model is known to be fairly effective for better prediction of adverse pressure gradient and flow separation. This model has been designed to promote turbulence in the congestion zone of fluid flow.

The SST k-ω turbulence model is a two-equation eddy-viscosity model developed by Menter [4] to effectively blend the robust and accurate formulation of the k-ω model in the near-wall region with the free-stream independence of the k-ε model in the far field. To achieve this, the k- ε model is converted into a k-ω formulation. Transport equations for the SST k-ω model are given by:

$$\partial/\partial t\,(\rho k)+\partial/(\partial x_{\downarrow}i)\,(\rho k u_{\downarrow}i)=\partial/(\partial x_{\downarrow}j)\,((_{\downarrow}k\ \partial k/(\partial x_{\downarrow}j))+G_{\downarrow}k-Y_{\downarrow}k+S_{\downarrow}k$$

$$\partial/\partial t\,(\rho\omega)+\partial/(\partial x_{\downarrow}i)\,(\rho\omega u_{\downarrow}i)=\partial/(\partial x_{\downarrow}j)\,((_{\downarrow}k\ \partial\omega/(\partial x_{\downarrow}j))+G_{\downarrow}\omega-Y_{\downarrow}\omega+D_{\downarrow}\omega+S_{\downarrow}\omega \quad ($$

In these equations, $\tilde{G}_k$ represents the generation of turbulence kinetic energy due to mean velocity gradients, G_ω represents the generation of ω ,

Γ_k *and* Γ_ω represent the effective diffusivity of k and ω, respectively, Y_k and Y_ω represent the dissipation of k and ω due to turbulence, D_ω represents the cross-diffusion term, S_k and S_ω are user-defined source terms.

The Model of Submarine Hull

A standard submarine hull model was used as a prototype for computations. The bow of our submarine model has been chosen as ellipsoidal and the stern has been chosen paraboloidal in shape with a portion of parallel mid-body. Since CFD method was used for the computations, This method is a very grid dependent technique. Therefore CFD method needs to be meshed in proper ways to get reliable and converged results. The largest errors occur where the largest gradients are. For this reason, the resolution should be increased in such regions. Only a restricted amount of cells can be used due to restrictions in computational power. Therefore it is beneficial to have a denser grid where e.g. the curvature of the surface is high and having larger cells closer to the middle of the surface. The discretization of the geometric domain of the submarine has been divided into 950.000 hexahedral meshes and every simulation has been iterated three hundured times. Since the CFD calculations on the computer takes a lot of time and needs more memory. It is 1/50 scale model rather than the actual size of the submarine has been used for the computations. The hull model has an overall length L of 1.6 m and maximum diameter D of 0.20 m. The sail is located in front of the hull with a length of 0.24 m..

The profile of the submarine model hull is shown in Figure 1. Also shown is the profile of nose cone, tail cone and sail shape.

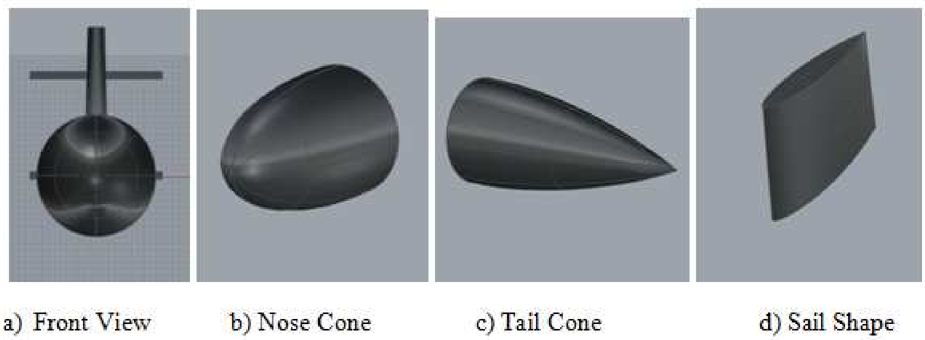


Figure 1. CAD Model of the Submarine Hull

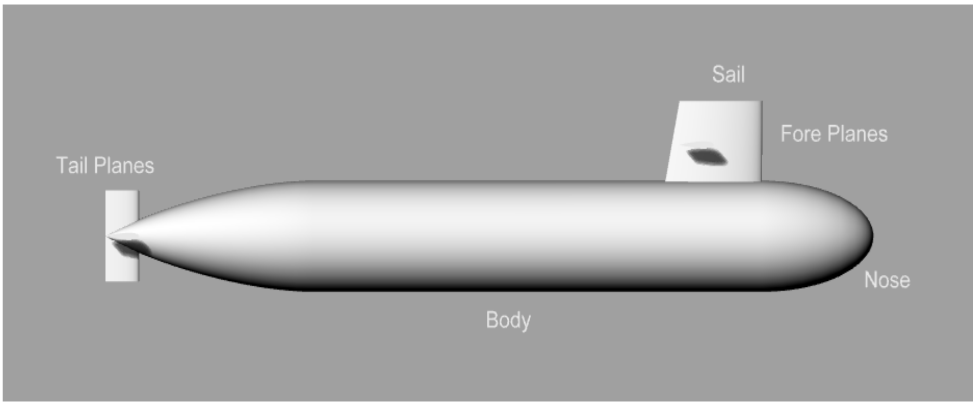
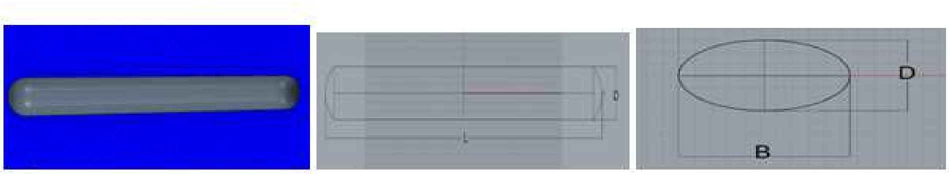


Figure 2. Submarine Control Surfaces

Numerical Computations

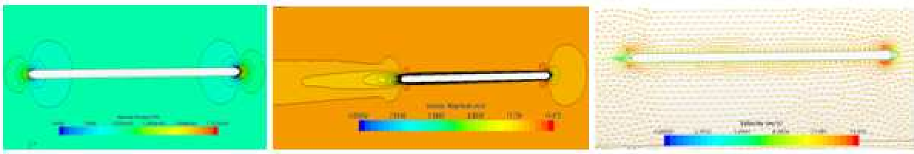
The numerical calculations were attempted by the following certain steps. The first step was the bare submarine body which was taken as a cylinderical shape for the flow calculations to observe how to change flow distribution according to different B/D ratios. It was analysed according to 1, 2, 4 and 8 ratio values. The subsequent steps, the numerical computations for hull with sail, with hull-sail and aft planes, and finally having all necessary control surfaces components of submarine form have been carried out seperately. All numerical computations were performed, in the following figures, on the actual size of bare submarine body. For each case, the results are shown in Figures 3 to 7 respectively.



Mesh configutaion L/D B/D

a) Pressure distribution b) Velocity distribution c) Velocity vector distribution (B/D=1)

a) Pressure distribution b) Velocity distribution c) Velocity vector distribution (B/D=2)



a) Pressure distribution b) Velocity distribution c) Velocity vector distribution (B/D=4)

a) Pressure distribution b) Velocity distribution c) Velocity vector distribution (B/D=8)

Figure 3. The pressure, velocity and velocity vector distributions around a cylindericsl submarine body (without sail, tails and other appendages) for different B/D ratios.

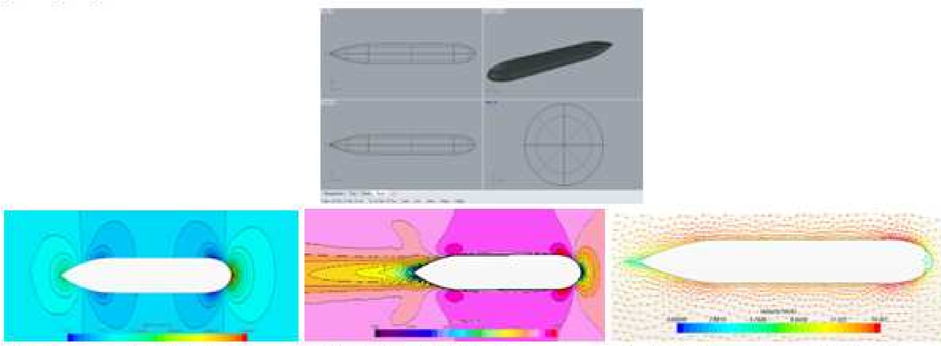
In these figures, it can be seen that how the absolute pressure and velocity distribution change around the submarine when the submarine body is assumed to be fixed and flow is coming from front of it. The computed values of absolute pressures and drag forces (resistance) for different B/D ratios of bare submarine body is given in table 1. The values given in this table are calculated at 25 knots of submarine speed.

Table 1. The absolute pressures and forces (resistances) values for different B/D ratios of bare submarine body

B/D	$V_{min}(m/s)$	$V_{max}(m/s)$	$P_{min}(Pa)$	$P_{max}(mPa)$	*Drag force (N)*	*Percent of Changing in force*
1	0	15.698	39795	1838	235.39	-
2	0	15.358	43192	1752	168.59	%29
4	0	14.673	54406	1747	129.43	%24
8	0	13.957	59520	1589	126.38	%3

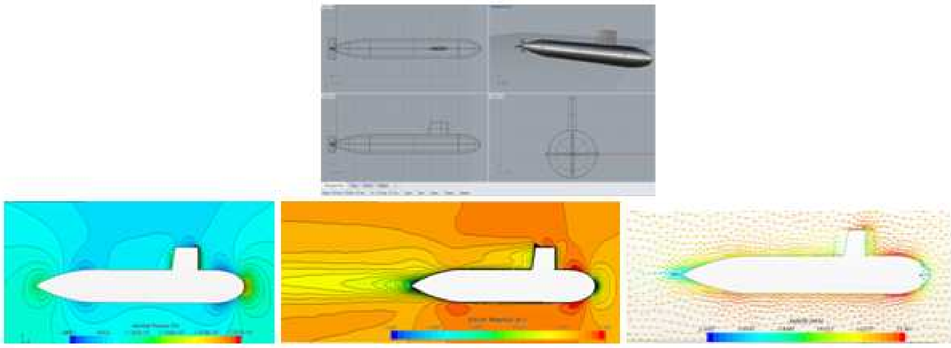
The values given in table 1 states that, consequently increasing B/D ratio of the bare submarine body will lead to reduced the drag force (resistance). This shows that the resistance depends on the pressure distribution around the body eventhough the minimum pressure values are increasing and the maximum pressure values are decreasing acording to B/D ratios geting higher.

- - -



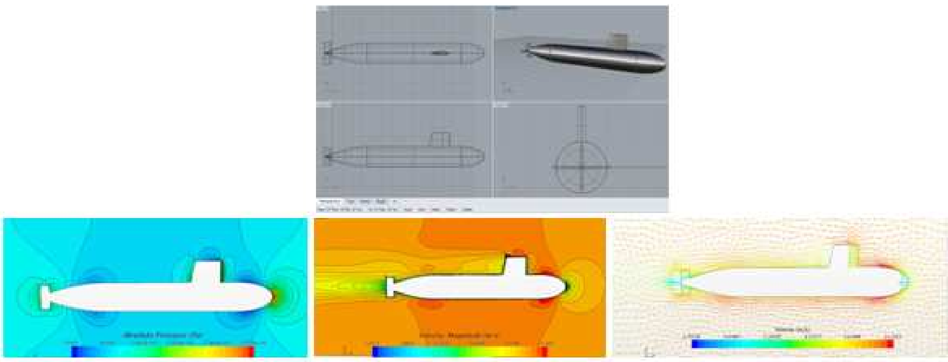
a) Pressure distribution b) Velocity distribution c) Velocity vector distribution

Figure 4. The pressure, velocity and velocity vector distributions around the bare submarine body (without sail, tails and other appendages).



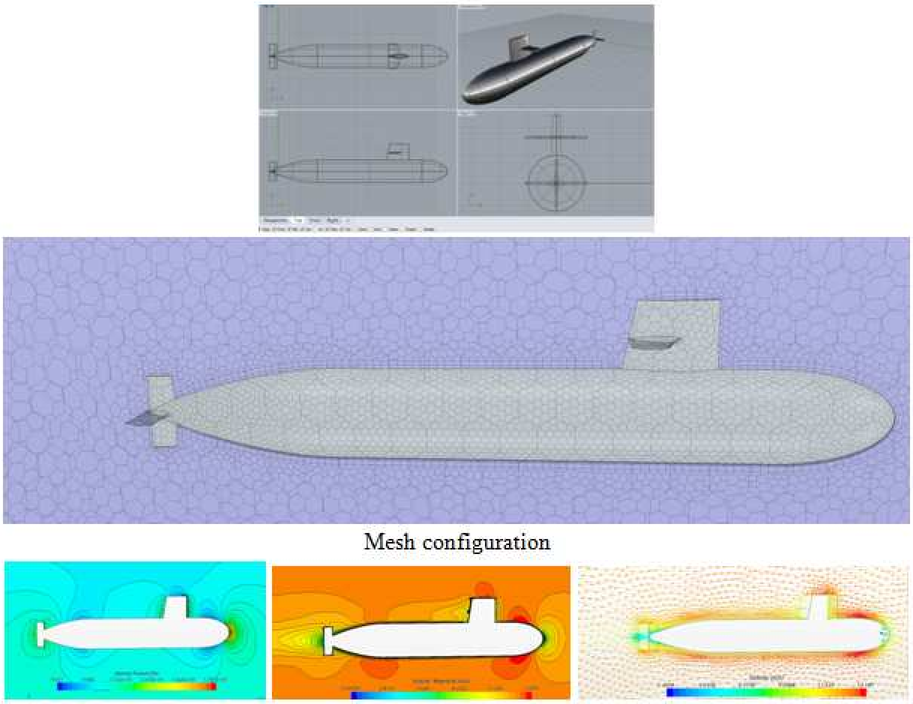
a) Pressure distribution b) Velocity distribution c) Velocity vector distribution

Figure 5. The pressure, velocity and velocity vector distributions around the submarine body with sail.



a) Pressure distribution b) Velocity distribution c) Velocity vector distribution

Figure 6. The pressure, velocity and velocity vector distributions around the submarine body with sail and tails (aft planes).



Mesh configuration

a) Pressure distribution b) Velocity distribution c) Velocity vector distribution

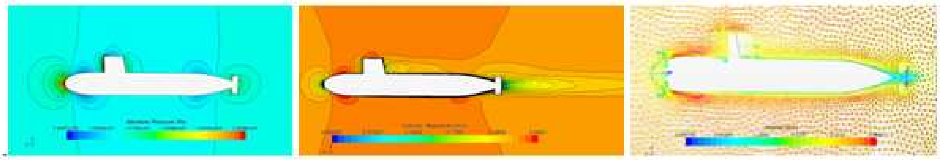
Figure 7. The pressure, velocity and velocity vector distributions around the submarine body with sail and tails (Hull+Sail+Sail Planes+Aft Planes).

Table-2 A comparison of all results according to submarine's components

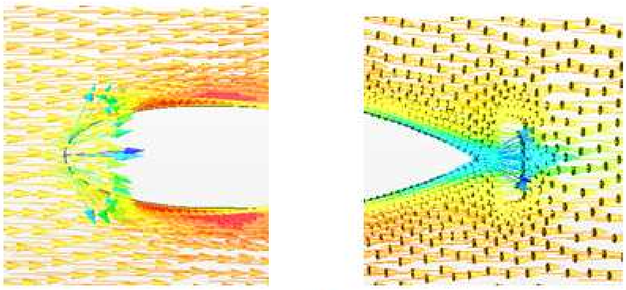
	$V_{min}(m/s)$	$V_{max}(m/s)$	$P_{min}(Pa)$	$P_{max}(mPa)$	*Force(N)*	*Percentage difference in force*
Hull	1.8506	14.386	77120	179.08	126.64	-
Hull+Sail	1.9269	14.434	68863	176.76	152.5	%20
Hull+Sail+Aft Planes	1.2536	14.409	74505	179.36	179.9	%17
Full	1.4668	14.187	70443	177.64	204.72	%14

As can be seen from Table 2 that the resistance increases because the sail area creates additional surface to create more resistance. On the other hand, the pressure values do not change dramatically as it was obtained without having sail. The full body of the submarine gives less pressure value than bare hull and Hull + Sail + Aft Planes form.

Flow Distribution Around Submarine According to Its Speed Variation
The absolute pressure and velocity distributions around the model submarine have been computed for the different submarine velocities from 0 to 2.18 m/s with increments of 0.364 m/s (corresponds to 5 knots of submarine speed). The results for each case are shown in figures 8 to 12.

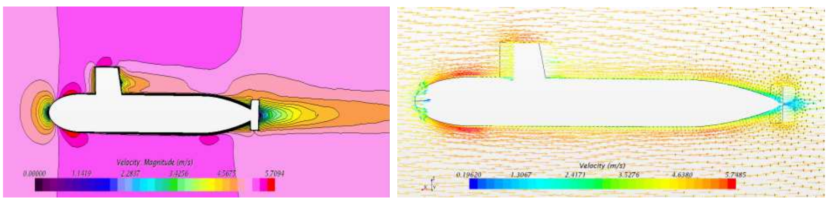


a) Pressure distribution b) Velocity distribution c) Velocity vector distribution



d) Fore and Aft view of the Vectorel Velocity

Figure 8. The pressure, velocity and velocity vector distributions around the submarine model at 0.364 m/s speed.



a) Pressure distribution b) Velocity distribution c) Velocity vector distribution

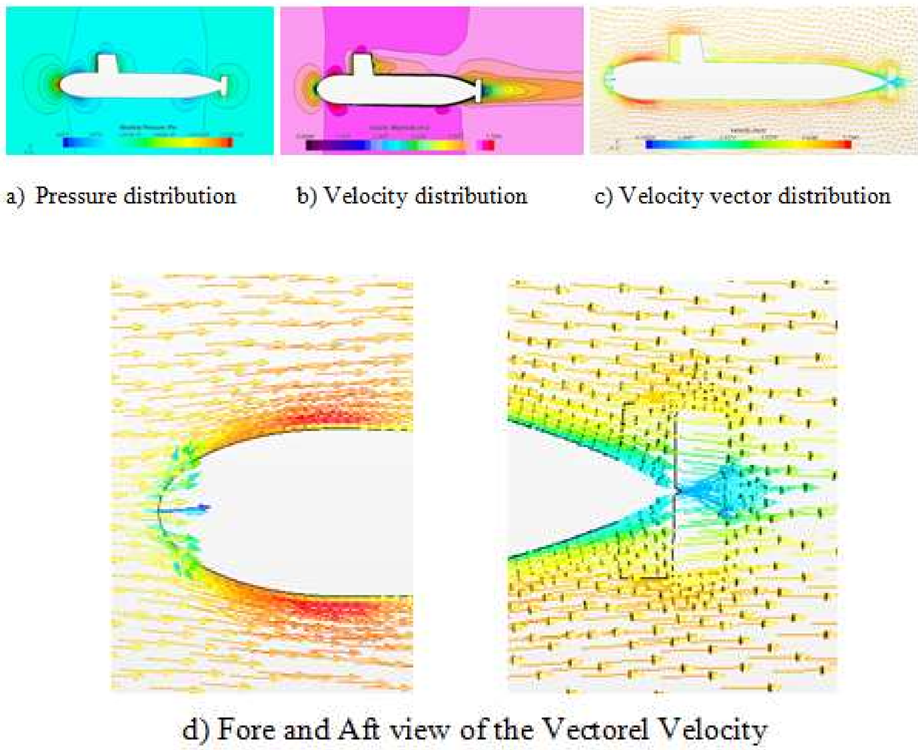


Figure 9. The pressure, velocity and velocity vector distributions around the submarine model at 0.728 m/s speed.

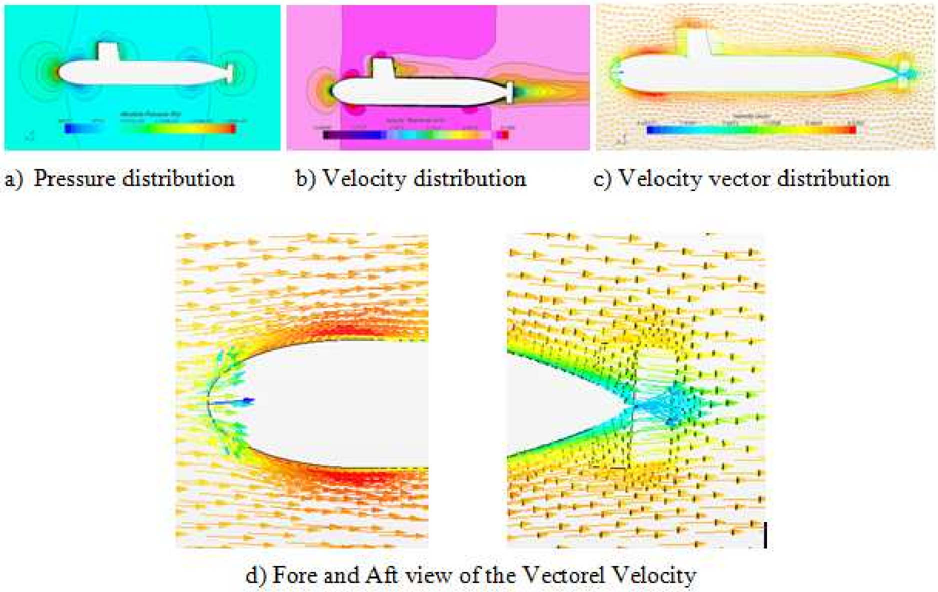


Figure 10. The pressure, velocity and velocity vector distributions around the submarine model at 1.0912 m/s speed.

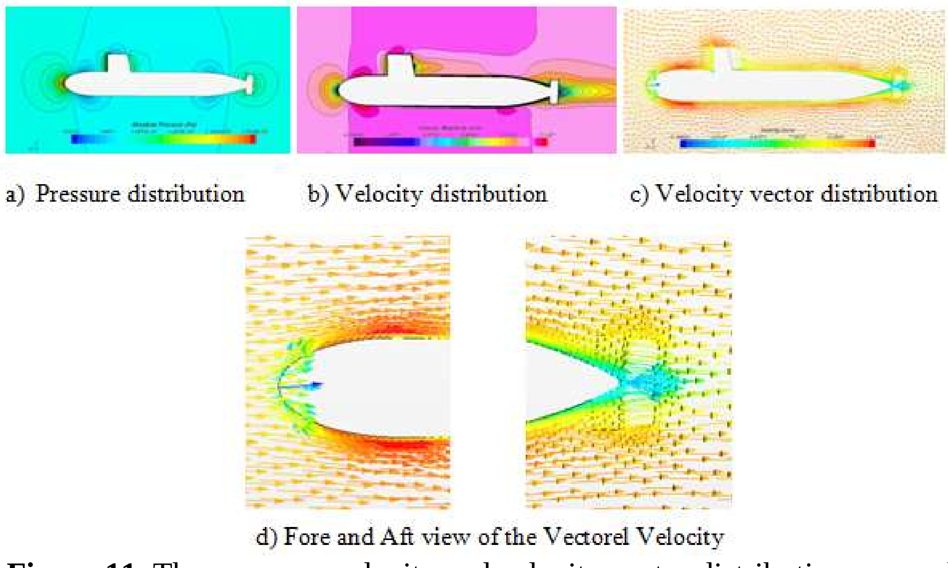


Figure 11. The pressure, velocity and velocity vector distributions around the submarine model at 1.455 m/s speed.

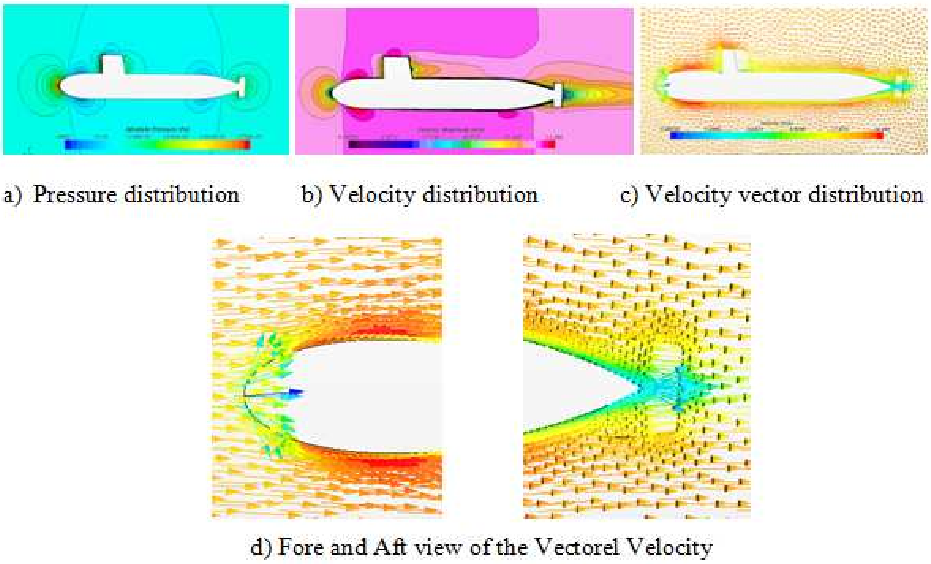


Figure 12. The pressure, velocity and velocity vector distributions around the submarine model at 1.818 m/s speed.

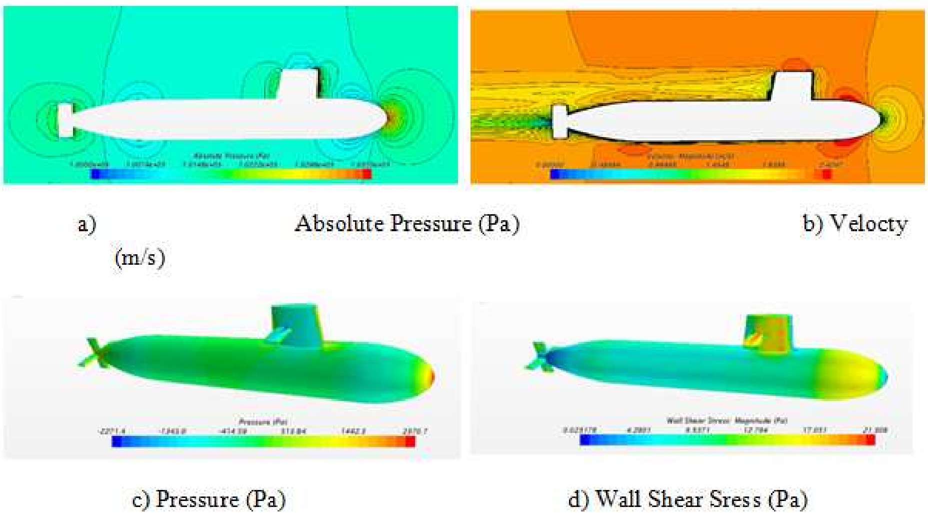


Figure 13. The pressure, velocity and velocity vector distributions and Wall Shear Stress around the submarine model at 2.1824 m/s speed.

The computed values of drag and lift forces (resistance) for different speed of submarine model is given in table 3.

Speed (m/s)*	*Drag Force (N)*	*Lift Force (N)*	*Absolute Pressure Pa (min)*	*Absolute Pressure Pa (max)*	*V m/s (max)*	*Pressure (Pa) (min)*	*Pressure (Pa) (max)*	*Thrust (N)*
0,7274	0,9934	7,528	1,0118e+05	1,0159e+05	0,80699	-248,97	263,67	0,4070
1,0912	2,1005	16,9896	1,0100e+05	1,0192e+05	1,2117	-564,10	593,15	0,8994
1,4549	3,5955	30,2581	1,0074e+05	1,0238e+05	1,6161	-1006,5	1054,2	1,5826
1,8186	5,4765	47,3348	1,0041e+05	1,0297e+05	2,0207	-1575,8	1647,0	2,4572
2,1824	7,7347	68,2035	1,0000e+05	1,0370e+05	2,4247	-2271,4	2370,7	3,5217

Table 3. The absolute pressures and drag and lift forces (resistances) values for different speed of model submarine.

* The speed values are given in the table for the model submarine. They correspond to 5 to 30 knots of actual submarine speeds with 5 knots increments.

The values of maximum pressure, drag force and lift force acting on the submarine model are given interms of Reynolds number in Figure 14-16, respectively.

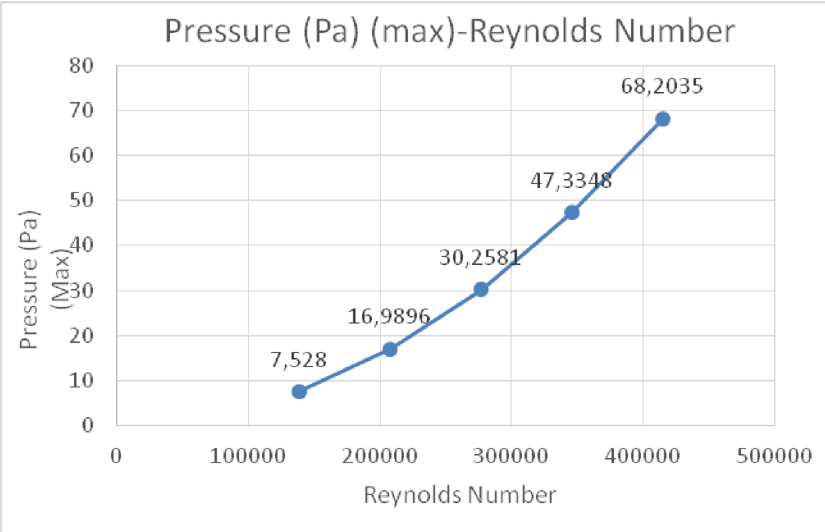


Figure 14. Reynolds Number vs Pressure (Max) (Pa)

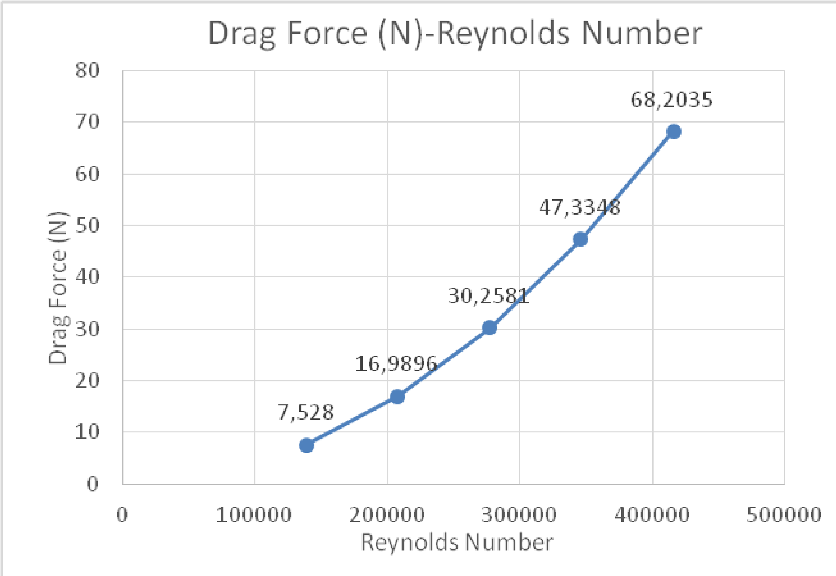


Figure 15. Reynolds Number vs Drag Force (N)

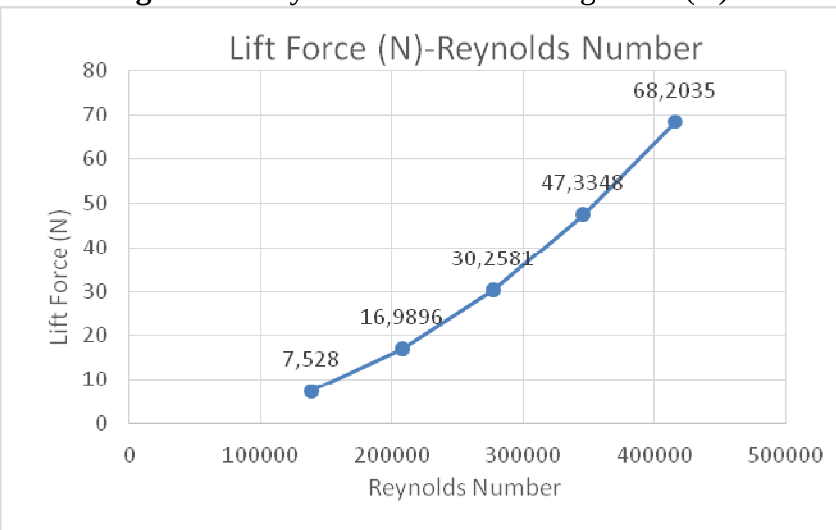


Figure 16. Reynolds Number vs Lift Force (N)

Investigation of the Sail Position According to Flow Distribution

This study was subsequently expanded to investigate the effect of sail position on design of submarine hydrodynamic. It is well known that the details of the position and shape of the submarine sail will depend on the number of masts, type of power source, type of periscope as well as effects on steering and dynamic stability. Indeed it provides a bridge platform for conning the submarine on the surface and a supporting structure for about number of masts. It may also support the forward control fins. In the past, the location of the sail has been dictated by through-hull penetration masts like periscopes which could only be located in certain positions. This should not apply in the future because of improved designs of such systems to provide non hull penetrating masts. Choosing the correct position and height is important. If too tall it affects the centre of mass and may cause a greater snap roll [8]. Any non penetrating mast needs to be properly supported Arentzen and Mandel [6] report that the drag of these large appendages may be between 15-30 % of the bare hull drag.

In this study, six sail positions were examined, the first position has been taken from the nose point by L/(6.9) m for determination of the flow and absulate pressure distributions around it (see table 4). Then the sail position has been changed to backward by taking equal increment from its position at each step for the computation of the flow and absulate pressures distributions. On the other hand, the computations were carried out for three different sail cross sections such as NACA0012, NACA0018 and NACA0024. The velocity and absolute pressure distribution values obtained from CFD computations depending on changing the position of the submarine sailing are shown in figure 19 and 20, respectively where the sailing cross section has been taken as NACA0018. Before it can be considered the design of the submarine sail, it is important to review the basic phyics of the flow around foil sections. For example it is assumed that the foil has constant section, and is long enough; in this case, the flow around all sections of the sail foil is the same, and this is describe as 2D flow. Studying 2D flow can give many insights about the effect of the section shape on the performance.

Table 4. Sail locations

Position	Sail location distance from fore point to the back of submarine model	The distance between successive positions
Position 1	232,2 mm	113,5 mm
Position 2	345,7 mm	113,5 mm
Position 3	459,3 mm	113,5 mm
Position 4	572,8 mm	113,5 mm
Position 5	686,4 mm	113,5 mm
Position 6	800 mm	113,5 mm

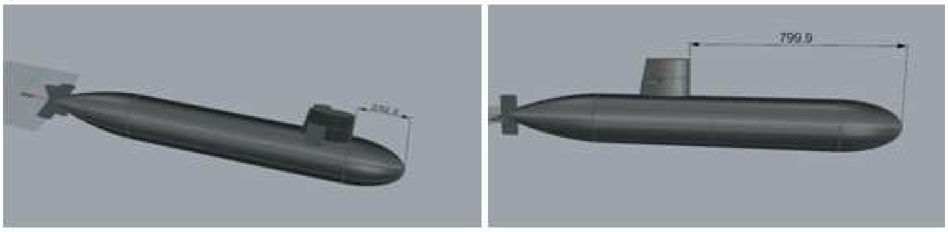


Figure 17. The first and last sail positions distance from the front of the submarine

Table 4. NACA Profiles to be used in model sail

	Thickness	Airfoil Lenght	Thickness / Airfoil Lenght
NACA0012	8,4 mm	70,0 mm	0,12
NACA0018	12,6 mm	70,0 mm	0,18
NACA0024	16,8 mm	70,0 mm	0,24

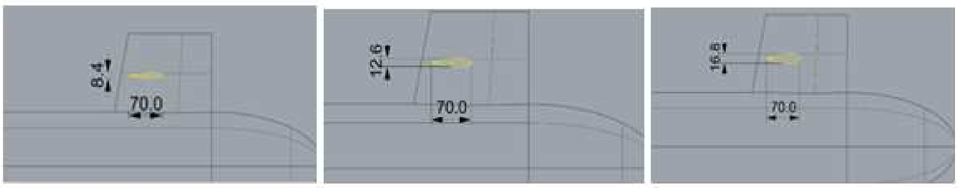


Figure 18. Sail dimension for three NACA Profiles

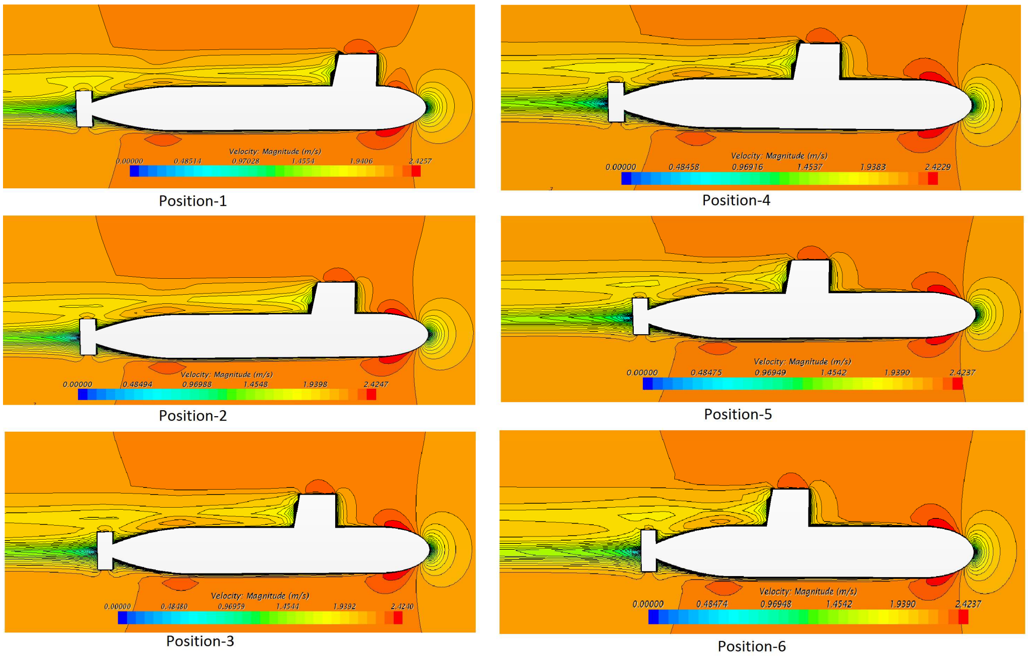


Figure 19. Velocity distribution with respect to the sail positions.

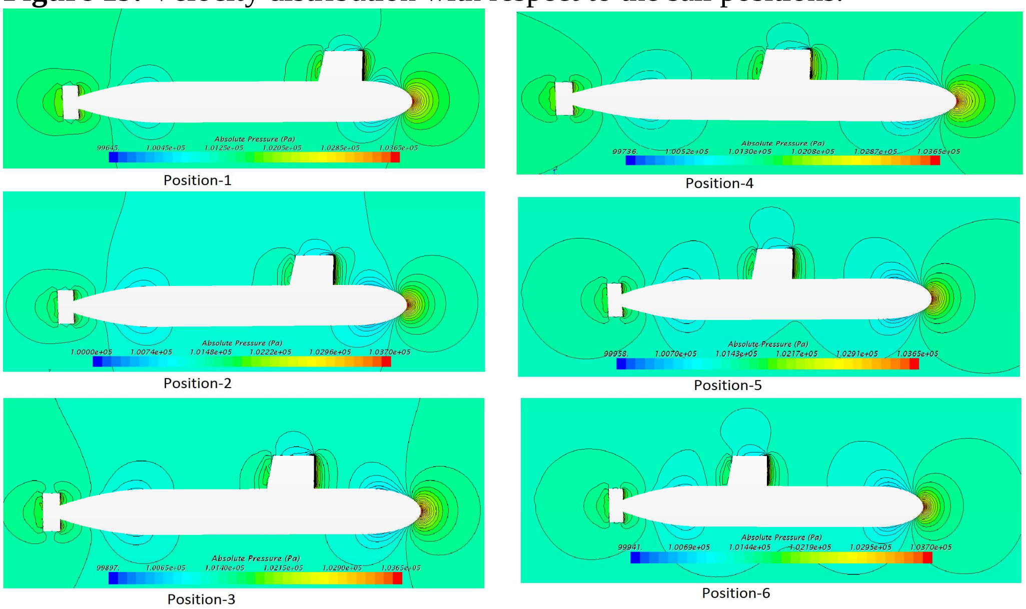


Figure 20. Absolute pressure distribution with respect to changing of sail position

From the position of the sail of the submarine with the flow lines were calculated by taking the values of absolute pressure in certain places.

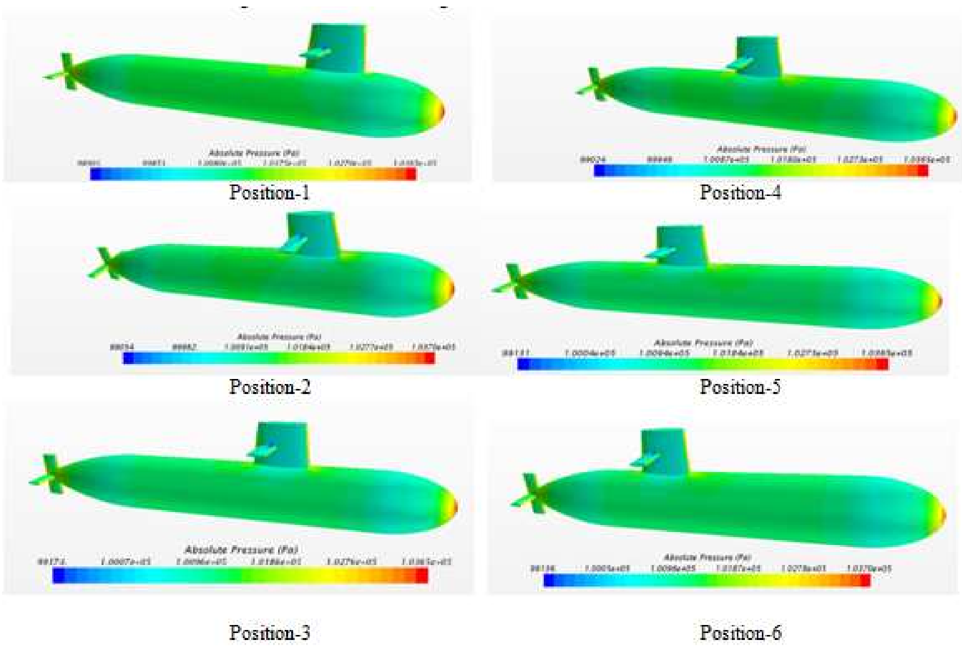


Figure 21. Absulate pressure distribution around the submarine in 3-D for different position of the sail having NACA0018 cross section

Conculusion and Recommendations

The increasing capacity and speed of computers raised the use of computational fluid dynamics (CFD) to the maritime industry. In the last decades, many developments have been observed in different areas of incompressible flow modeling including grid generation techniques, solution algorithms and turbulence modeling, and computer hardware

capabilities. One important conclusion is that CFD gives the quite accurate predictions, but requires many CPU-times. It can offer a cost-effective solution to many problems in underwater vehicle hull forms. However, effective utilization of CFD for naval hydrodynamics depends on proper selection of turbulence model, grid generation and boundary resolution. The most common turbulence modeling approach of today is RANS (with the SST k-ω turbulence model), which is based on a statistical treatment of the fluctuations about an average flow; it is expected that RANS will be the preferred, and fully sufficient, engineering tool for most design aspects. The advantage of RANS is however that the approach is fast (since only the mean flow is sought), and it is available in most CFD codes. This method can accurately predict the velocity field and absolute pressure distribution around a submarine and its resistance components. It also gives the possibility to visualize problem areas, such as separation zones. All CFD calculations were performed at model-scale Reynolds numbers of $\sim 10^7$. The study easily can be extended to full-scale Reynolds numbers.
The ratio of beam to diameter (same as length to diameter) bears a strong effect on the total resistance. The more wetted surface the greater the skin friction. This can be seen from the computational results of the submarine model used in this study (see table 1), the resistance of the bare submarine body decreases with increasing B/D (and L/D) ratio. This states that bare submarine's body resistace depens on the pressure distribution around the body eventhough the minumum pressure is increasing and the maximum pressure is decreasing. Therefore if the displaced volume of the submarine is contained in a long thin shape, then the skin friction is greater than for a shorter, beamier shape of the same volume which has less wetted surface. It is proposed that a new shape be considered of beamer shape or shorter length and greater diameter which will reduce the total drag force closer to the ideal.

In case of full submarine body including the sail and appendages, when the speed increases the resistance of the submarine increases as expected but the minimum pressure falling and maximum pressure increases opposite to the bare body case. This states that sail and appendages play an important role in submarine design. Besides, the mesh blocks in the vicinity of the sail affected by geometry and mesh topology changes. The mesh away from the sail remained unchanged, leading to more consistent CFD results.

Apart from the hull shape, important items like the sail and control surfaces need to be optimised for position, size and shape to maximise operational effectiveness and minimise resistance. The details of the position and shape of the sail will depend on the number of items beeing built inside of sail. These details should be considered after the testing of the model of the bare hull. As a tentative first move the sail is drawn moved forward by approximately L/7 m from the front of the submarine in order to maintain the lateral stability and counter-balance the loss in lateral area aft. Indeed, the sail position, shape and size might be well provided according to the required volume for advanced future payloads. The sail of the submarine can now be discussed as it plays a major role in producing drag and hence its design is critical. Research has proven that a sail may contribute up to 30% of total submarine resistance.

As is known, it is one of the major problems in submarine noises. Flow noise is primarily caused by turbulence, and the general shape of the hull is less of a cause of turbulence than poor detailing. Many class of submarines have had the "old style" sail for many years, with only comparatively minor attempts at streamlining. Their sails have sharp corners to produce noise. It would be obvious that the "rounded" "streamlined" sail would produce less noise. Flow separation is the big cause of unavoidable turbulence, and unfortunately there isn't much you can do about that beyond a certain point. One question will rise in our mind from a hydrodynamic point of view, which is better? The hull shape leads to flow noise which is caused by flow disturbance thus the hull shape effects the submarine speed.

To validate the CFD code on similar sail shapes and positions calculations will be compared with experimentally obtained data at the same from in a wind tunnel or in a water channel. This data comparison includes flow visualization, axial velocity and surface pressures. The agreement will demonstrate that RANS codes can be used to provide the significant hydrodynamics associated with these sail shapes and positions. To improve the design several modifications can be done on sail position are evaluated using the RANS code. Based on the predicted secondary flow downstream of the sail as well as the drag a new design is chosen, without having to build and test the inferior shapes, reducing time and cost for the program.

References

1) "Submarine Hydrodynamics" by Martin Renilson, SpringerBriefs in Applied Sciences and Technology, New York Dordrecht London, 2015.
2) P.N.Joubert, "Some Aspects of Submarine Design Part 1. Hydrodynamics" by DSTO Platforms Sciences Laboratory, 506 Lorimer St, Fishermans Bend, Victoria 3207 Australia. 2004.
3) Gorski, J. J., Coleman, R. M., and Haussling, H. J., "Computation of Incompressible Flow Around the DARPA SUBOFF Bodies," David Taylor Research Center Report, DTRC - 90/016, 1990.
4) Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", AIAA Journal, Vol. 32, No. 8, pp.1598-1605, August, 1994.
5) N. Vlahopoulos and C. G. Hart., "A Multidisciplinary Design Optimization Approach to Relating, Affordability and Performance in a Conceptual Submarine Design", Journal of Ship Production and Design, Vol. 26, No. 4, November 2010, pp. 273–289.
6) Arentzen, E. S. and Mandel, P. E., "Naval Architectural Aspects of Submarine Design", Trans. SNAME, pp. 622-692, 1960.
7) D. Ang, L. Chen and J. Tu., "Unsteady RANS Simulation of High Reynolds Number Trailing Edge Flow", 15th Australasian Fluid Mechanics Conference, The University of Sydney, Sydney, Australia, 13-17 December 2004.
8) P. N. Joubert, "Some Aspects of Submarine Design Part 2. Shape of a Submarine 2026", Defence Science and Technology Organisation DSTO-TR-1920, Australia, 2006,
9) Hervey, J. B., "Submarines", Brassey's, London, 1994.