Numerical Study of Submarine Launched Underwater Vehicle

Muhammad Zubair Zahid and Muhammad Nadeem
Pakistan Institute of Engineering and Applied Sciences (PIEAS)
NiloreIslamabad, Pakistan
mzubair_1922@yahoo.com, mnadeem@pieas.edu.pk

Muhammad Ismail
Centers of Excellence in Science and Applied Technologies (CESAT)
Islamabad, Pakistan
ismailbaila@gmail.com

***Abstract*:Launching of underwater projectile from submarine is carried out by gas-steam ejection dynamic system. Specific velocity of projectile at water exit is essential for its successful launch. The velocity of projectile depends upon the projectile dimensions, mass and water depth which govern the pressure requirements for gas-steam ejection dynamic system. To investigate the pressure required to launch the projectile under various conditions, a two-dimensional, axisymmetric numerical model was established. The numerical simulations were performed using ANSYS Fluent software. Dynamic mesh method was applied to track the motion of projectile combined with a User Defined Function (UDF) to compute the forces exerted on the body of the projectile. To achieve specific exit velocity of projectile for number of cases by varying vehicle mass, nose shape, and water depth simulations were performed and corresponding pressure requirements for each case was determined. Results shows the nonlinearity in pressure requirements with changing different parameters to achieve same exit velocity. In addition, bubble cavity formation in water is analyzed.**

Keywords: Submarine launched Projectile; Dynamic mesh; gas cavity; Multiphase flow

I. Introduction

A forward movement of Navy in weapons is the development of such missile which can be launched from underwater via submarine. This technology has advantage of un-detectability, good tracking, spotting its target and devastation.Projectile is pushed by high pressure gas from the tube of submarine and moves through the water until it comes out of sea water. This launching technique is termed as "Cold launching of missile". Launching process consist on three stages which includes tube-exit stage, free moving water stage and water-exit stage.Tube exit stage is important one due to strong forces of pressure acting on it which effects its mechanical structure.During launching process projectile; trajectory, velocity and drag force varies and pressure fluctuation on launching pad also occurs.When projectile comes out of tube, air bubble form due to expansion of highly pressurized gas. Size of air bubble depends on the pressure inside tube which is in direct relationship with mass of vehicle and water depth. Afterwards shrinkage of air bubble happens due to hydrostatic pressure and water start moving in the tube cavity. Hence, the required pressure is essential parameter which has to be calculated in order to design a gas generator in accordance to the vehicle mass and its dimensions.

In recent years, various parameters controlling the motion of submarine launch underwater vehicle have been investigated by many researchers.Liu and Xi [1] simulated the gas-steam ejection of projectile using 3d canister, whose top surface is act as a bottom of projectile which moves upward with the gas pressure. Gas flow filled around the canister was analysed and predicted the velocity of projectile.Cao[2]et al. studied the evolution of gas cavity at the bottom of projectile when it leaves the launching tube by implementing 2D-axisymmetric model. Huang[3]et al. observed the bubble dynamic behavior of pressurized gas during tube exit stage, and found that air is sucked toward nose of projectile due to low pressure region around it. Li [4] et al. have researched the evolution of bubble cavity and calculated the variation of velocity, acceleration, and pressure on body for 1 sec of launching process.Chen [5]et al.computed bubble dynamics and thrust of underwater power-launched vehicle. Zhang and Wang [6] provides the velocity profile of underwater vehicle pushed by energy source of compressed water.

Until now no one studied the pressure profile for different shapes of nose, weights and water depth. Besides, the pressure requirements to push the projectile out of water surface in different scenarios are calculated.

II. Mathematical Model

A. Governing Equations

Hydrodynamics of cold emission launch is very complex phenomena due to multiple phase involved in it.Mixture model coupled with cavitation model is used to simulate such an intricate problem including three phases liquid, air and vapors. Continuity equation for individual phase is written as follow:

$$\frac{\partial(\rho_m \alpha_l)}{\partial t} + \nabla.(\rho_m \alpha_l V_m) = 0 \quad (1)$$

978-1-7281-4676-8/20/$31.00©2020 IEEE

$$\frac{\partial(\rho_m \alpha_v)}{\partial t} + \nabla.(\rho_m \alpha_v V_m) = m^{-}_{l\to v} - m^{+}_{v\to l} \quad (2)$$

$$\frac{\partial(\rho_m \alpha_g)}{\partial t} + \nabla.\left(\rho_m \alpha_g V_m\right) = 0 \quad (3)$$

In equation (2) right hand side indicating the net phase change rate.Equations stated above for continuity are similar to its counterpart in single phase flow expect for mass transfer and volume fraction terms. Volume of any phase is stated as:

$$V_q = \int \alpha_q dV \quad (4)$$

where

$$\alpha = \sum \alpha_q n_q \quad (5)$$

V_m is mass average velocity, α_i is phase volume fraction andρ_m is the density of mixture.

The equation of momentum is written by summing the individual equation of all three phases for air, water liquid and water vapor.

$$\frac{\partial(\rho_m V_m)}{\partial t} + \nabla.(\rho_m V_m V_m) = -\nabla P + \bar{F} + \rho g$$
$$+ \nabla.[\mu_m(\nabla V_m + \nabla V_m^T)] + \nabla.(\sum_{q=1}^{n} \alpha_q \rho_q V_{dr,q}) \quad (6)$$

where V_m^T is mixture velocity in turbulence and $\bar{F}$ is body force.Viscosity of mixture is written as:

$$\mu_m = \sum_{q=1}^{n} \alpha_q \mu_q \quad (7)$$

drift velocity of phase q is defined as:

$$V_{dr,q} = v_q - v_m \quad (8)$$

density of incompressible ideal gas changes with temperature variation. This change in density in conjunction with gravity force sets the natural convection in the system. Density of incompressible ideal gas is calculated as:

$$\rho = \frac{M}{RT} P_{op} \quad (9)$$

Energy equation is applied to calculate temperature variation of mixture during simulation. Energy equation for mixture model is written as:

$$\frac{\partial}{\partial t}\sum_{q=1}^{n}\left(\alpha_q \rho_q E_q\right) + \nabla.\sum_{q=1}^{n}\left(\alpha_q v_q\left(\rho_q E_q + p\right)\right)$$
$$= \nabla.(k_{eff}\nabla T) \quad (10)$$

where k_{eff}effective thermal conductivity calculated as:

$$k_{eff} = \sum \alpha_q (k_k + k_t) \quad (11)$$

where$k_t$is a turbulent thermal conductivity based on the type turbulent model being used.When the fluid is compressible,$E_q$is total energy calculated as

$$E_q = h_q - \frac{p}{\rho_q} + \frac{v_q^2}{2} \quad (12)$$

and for incompressible phase $E_q = h_q$ where h_q is sensible enthalpy energy of phase q.

B. Cavitation Model

Zwart-Gerber-Belmari model[7]is implemented in simulation as it converges easily and compatible with mixture model. General equation of this model is as follow:

$$\frac{\partial(\rho_v \alpha_v)}{\partial t} + \nabla.(\rho_v \alpha_v v_v) = R_e - R_c \quad (13)$$

If$P \leq P_v$

$$R_e = F_{vap}\frac{3\alpha_{nuc}(1-\alpha_v)\rho_v}{R_B}\sqrt{\frac{2}{3}\frac{P_v - P}{\rho_l}} \quad (14)$$

If $P > P_v$

$$R_c = F_{cond}\frac{3\alpha_v \rho_v}{R_B}\sqrt{\frac{2}{3}\frac{P - P_v}{\rho_l}} \quad (15)$$

where R_B is bubble radius, F_{vap} is evaporation coefficient, F_{cond} is condensation coefficient and α_v is nucleation site volume fraction.

C. Turbulance Model

In this current research, *k*-ε with Renormalization group (RNG)[8] is used. Equation for turbulent kinetic energy is as follow:

$$\frac{\partial(\rho k u_i)}{\partial x_i} + \frac{\partial(\rho k)}{\partial t} = \frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff}\frac{\partial k}{\partial x_j}\right)$$
$$+ G_b + G_k + S_k - \rho\epsilon - Y_M \quad (16)$$

Equation for turbulence dissipation rate is as follow:

$$\frac{\partial(\rho\varepsilon u_i)}{\partial x_i} + \frac{\partial(\rho\varepsilon)}{\partial t} = \frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_{eff}\frac{\partial \varepsilon}{\partial x_j}\right) +$$
$$C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \quad (17)$$

where G_k and G_b are the generation of turbulent kinetic energyand buoyancydue to mean velocity gradient respectively. Y_M represents the fluctuation dilatation in compressible turbulence to the overall dissipation rate. α_k and α_ε indicates the inverse of Prandtl number for k and ε respectively.S_k is user defined source term for k.

D. Projectile motion

In process of submarine launch forces including gas and liquid on the bottom of projectile are represented by following differential equation.

$$\frac{d(mV)}{dt} = G + F_{gas} + F_{liquid} \quad (18)$$

where *m* and *V* represents the mass and velocity of projectile. This equation is implemented in FLUENT by compiling a user defined function (UDF). Forces are computed which update the velocity of projectile at each time step.

III. NUMERICAL SIMULATION METHOD

Numerical simulation of 2D-axisymmetrical model shown in Fig. 1 is carried out according to above theoretical models. Layering technique for dynamic meshing is implemented.ICEM package in ANSYS software is used to generate the quadrilateral structured mesh. Projectile is 8.5m long, having 0.86m diameter. A number of cases is simulated by changing height, mass and nose shape of projectile.

Discretization scheme for volume fraction, pressure-velocity coupling terms is quick and SIMPLEC respectively. Second-order upwind scheme is applied for density, energy, momentum, turbulent kinetic energy and dissipation terms.

A. Boundary Conditions

Inlet of tube gives air pressure which serve as a driving force for projectile. Pressure inlet boundary condition is set as $P(t) = P$ where $P(t)$ isaccording to Fig. 2.After the completion of 0.5sec simulation, inlet pressure boundary condition is turned into wall boundary condition. All other boundaries were specified as wall boundary condition. Boundary condition for fluid domain is set as

$$P = P_o + \rho g h \tag{19}$$

Where P_o is local atmosphere pressure, h is water depth which is variant according to each case.

IV. RESULTS AND DISCUSSION

To investigate the influence of water height, mass and nose shape of vehicle on pressure requirement twenty-four different cases were simulated. Different inlet pressure profiles were used for each case to attain specific exit velocity at a position when bottom of projectile is out of water. This range of exit velocity ensures to trigger the sensor of projectile for ballistic push before it losses all of its momentum. These pressure profileswere obtained by multiplyinga factor with the given pressure profile of Li et al.[4].List of multipliers for each case is listed in Table 1. It is illustrated in the table that by increasing the weight and depth of water multiplier increases but not in specific fashion. There is a nonlinear relationship with the pressure requirements either by varying weight, depth of water or shape of nose. Its due to hydrostatic pressure keeps on changing along the distance travelled in water. Bubble dynamics dissimilaritieswith the change in inlet pressure also contribute toward its nonlinearity behavior.

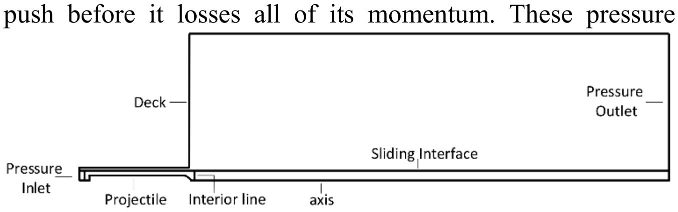

Fig. 1. Schematic diagram of geometry

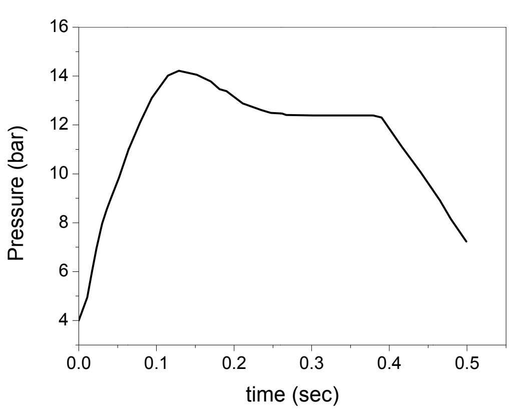

Fig.2. Pressure inlet Profile according to [4]

Figure 3 shows that the formation of Bubble cavity structure inside the water depends on the shape of nose and pressure of air inside the tube. Shape of nose does not affect as largely as pressure inside tube does. It can be seen that large bubble of air form when the pressure inside the tube is high which eventually dies up with the time. When the bubble collapse due to hydrostatic pressure of water, shock wave of water produces at that instant in two directions. Upper shock wave which moves toward the bottom of projectile and other toward the tube named as bottom shock wave. Upper shock wave may cause serious damage on the instruments placed at the bottom of projectile.Fig. 3(b and f) represents that cylindrical type air gap produce rather than larger circularone behind the projectile body due to low gas pressure requirement for 1.0-meter nose length shape projectile.It also gives an information that more the pressure required more will be the intensity of upper shock wave due to formation of larger bubble.

Table 1. Multipliers of Li et al.[4] *pressure profile, for different masses, heights and shapes of projectiles*

Depth of water (meter)	Mass of projectile (kg)	0.45-meter nose length	1.0-meter nose length
		Pressure multiplier	
10	4500	0.5	0.5
	5500	0.65	0.55
	6500	0.7	0.6
20	4500	0.67	0.65
	5500	0.85	0.75
	6500	0.9	0.8
30	4500	0.85	0.8
	5500	1	0.9
	6500	1.1	1
40	4500	1.05	0.9
	5500	1.2	1
	6500	1.3	1.08

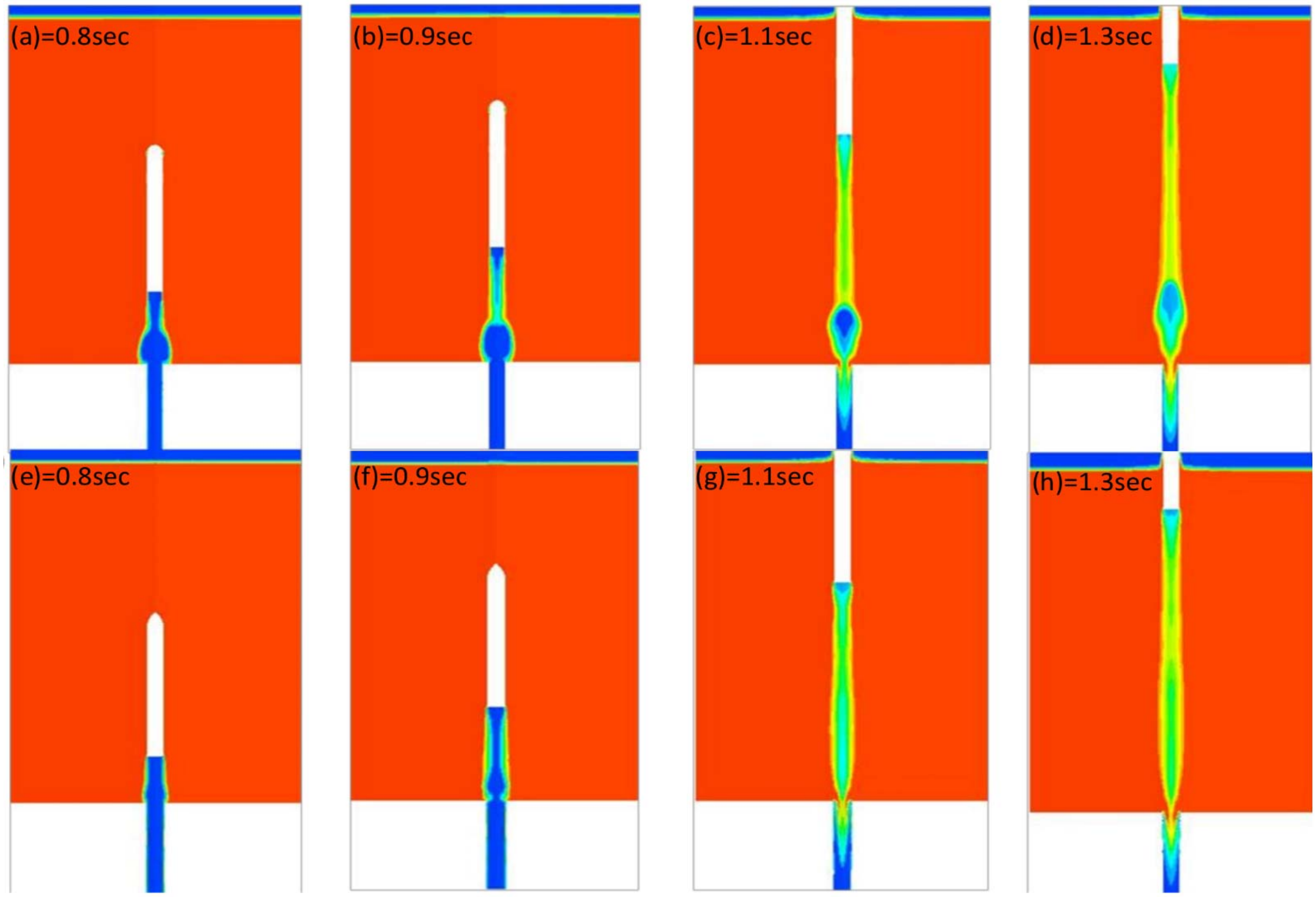

Fig. 3. Dynamics of bubble cavity projectile m=5500kg releases from h=20m (a-d) nose length 0.45m (e-h) nose length 1m

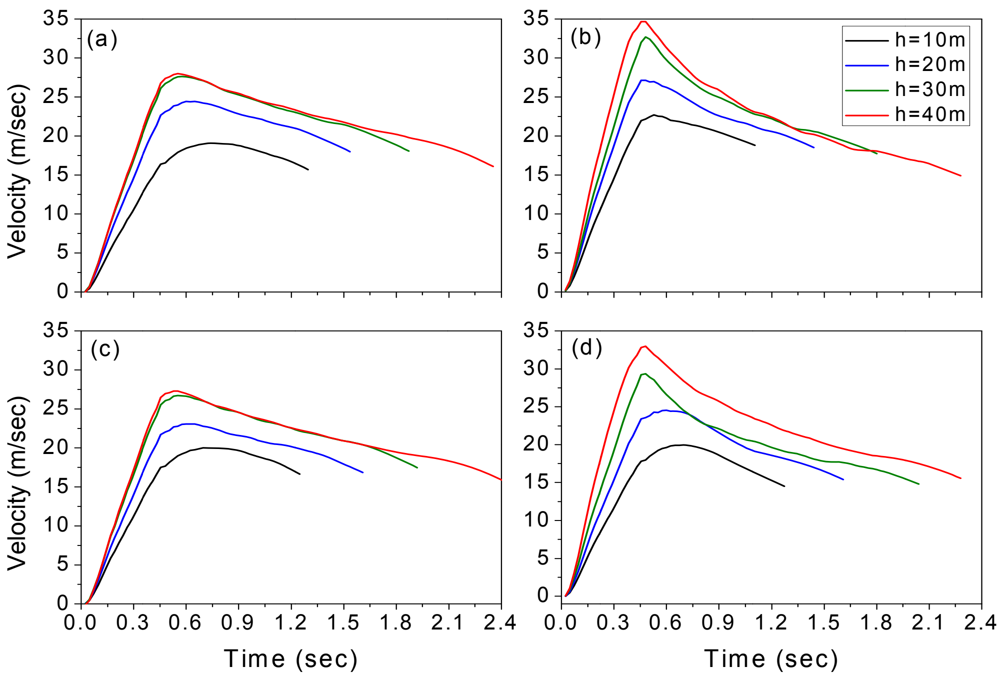

Fig. 4. Velocity profiles with same mass and different heights (a) 5500kg and 1m nose (b) 5500kg and 0.45m nose (c) 4500kg and 1m nose (d) 4500kg and 0.45m nose

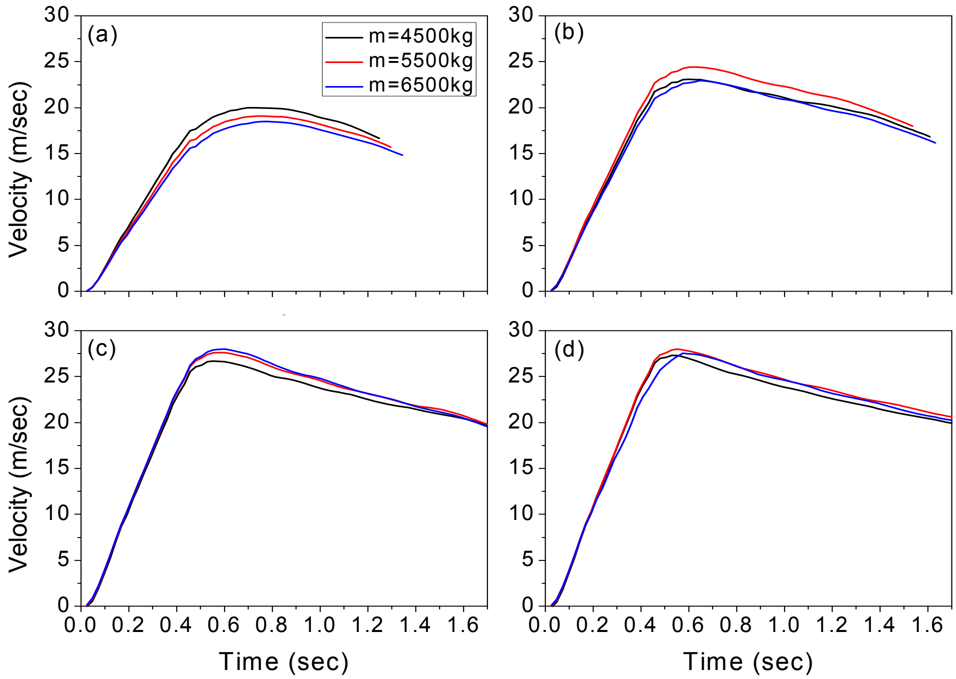

Fig. 5. Velocity profiles with same nose length 1m, same height and different masses (a) 10m depth (b) 20m depth (c) 30m depth (d) 40m depth

Figure 3(c and g) indicates that projectile having blunt type nose forces the water fields to revolve in circular path which assist the water at the bottom of projectile to collapse quickly adding the intensity of upper shock wave. On the other hand, as shown in Fig. 3(h) nose of streamline shape does not produce high curvature in streamlines of water resulting larger dispersion of vapour bubbles behind the tail of vehicle.

In the current research, one of the velocity trend is in good agreement with [4]. Initial velocity requirement increases with the increase in water depth to push the projectile out of water illustrated by Fig. 4. There is lot of jumps in velocity profiles with the projectiles having blunt type nose than streamline shape nose due to drag force variation and bubble cavity dynamics. When 0.45m nose length projectile leaves the tube at approximately 0.5sec, there is sharp decrease in velocities as drag force is large initially which decreases with the decrease in hydrostatic pressure of water.

Figure 5 illustrates that for small depth of water, uniform velocity persists for a longer time during free moving stage due to low hydrostatic pressure. Velocity profiles tends to be similar with the increase in water depth for different masses. Its due to high momentum gain at the end of tube exit stage also drag force effects on streamline shape nose is not too much significant to change velocity profiles.

V. CONCLUSION

Nonlinear relationship exists in pressure requirements for different cases to achieve same specific exit velocity of projectile. Bubble cavity formation depends on pressure and shape of nose. Upper shock wave intensity is in direct relationship with the pressure requirement.Velocity profiles for blunt type projectiles have little bit fluctuation. Velocity profiles tends to become uniform during water-moving stage for projectiles released from smaller water depth. Launching of projectiles having different weight from deeper water demands larger momentum of body so velocity profiles trends are almost similar in those cases.

REFRENCES

[1] Y. Liu and A. Xi, "An interior trajectory simulation of the gas-steam missile ejection," *J. Comput.*, vol. 8, no. 5, pp. 1321–1326, 2013.

[2] J. Y. Cao, C. J. Lu, Y. Chen, X. Chen, and J. Li, "Research on the base cavity of a sub-launched projectile," *J. Hydrodyn.*, vol. 24, no. 2, pp. 244–249, 2012.

[3] H. Hai-Long, W. Cao, and C. Wang, "Numerical Simulation of the Launching of Underwater Vehicle," *Appl. Mech. Mater.*, vol. 556–562, pp. 4404–4407, 2014.

[4] R. Li, G. Le, D. Ma, and S. Chen, "A study on the tube-exit cavity surrounding submarine-launched missile," in *3rd International Conference on Engineering Technology and Applications*, 2016, pp. 737–741.

[5] C. Chen, W. Cao, C. Wang, and Y. Wei, "Trajectory simulation for underwater vehicle with power-lunched," *J. Harbin Inst. Technol.*, vol. 23, no. 1, pp. 17–22, 2016.

[6] X. Zhang, S. Wang, and D. Yu, "2D axismmetric CFD simulation of underwater torpedo launch tube flow," *Proc. - 2009 Int. Conf. Inf. Eng. Comput. Sci. ICIECS 2009*, pp. 2–5, 2009.

[7] P. J. Zwart, A. G. Gerber, and T. Belamri, "A two-phase flow model for predicting cavitation dynamics," in *International Conference on Multiphase Flow*, 2004, no. January 2004, p. 152.

[8] V. yakhot and S. A. Orszag, "Renormalization-Group Analysis of Turbulence," *J. Sci. Comput.*, vol. 57, 1986.