A model of the water environment action on anthropomorphic robot arms

V A Kramar¹, V R Dushko¹

¹ Sevastopol State University, 33, Universitetskaya St., Sevastopol, 299053, Russia
E-mail: kramarv@mail.ru

Abstract. The article deals with the technology of constructing a 3-D velocity field mathematical model for flow around manipulators of anthropomorphic robots, such as the SAR-400 robot, when subjected to wave and constant currents. The studies show that when building control systems for manipulators operating underwater, it is necessary to take into account the arising forces acting on the manipulators as external disturbances that impede the precise fulfillment of control tasks. Methods for calculating 3-D velocity fields under the influence of wave and constant currents on the structural elements of manipulators of an underwater anthropomorphic robot as cylinders of large and small elongations are considered. A numerical simulation of the evolution of the flow velocity and pressure fields in a flow past a cylinder has been performed. The proposed approaches will allow the simulation of external disturbances acting on the manipulators of the robot working under water when constructing their control laws.

1. Introduction
Operating conditions of an underwater arm, including those of the anthropomorphic-type arm, will depend on features of the water area where the arm is to be operated. The analysis of underwater arm operating conditions, e.g. currents, is accounted for while developing its control system considering peculiarities of underwater arm operation near the seabed. So far, semi-empirical relationships have generally been used in analyzing current actions on ocean engineering facilities, which specifically in the context of this article are represented by underwater-operated anthropomorphic robot arms being modeled as cylindrical beams. Those relationships are apt to impose some considerable uncertainties on the results due to a significant scatter of drag and inertial coefficients, their strong variation with respect to flow direction and configuration of the object immersed in flow. In addition, it should be taken into account that the principal scaling law appears impractical to satisfy in laboratory studies of physical simulators. Proceeding from considerations set out in [1-4], this article discusses a 3-D velocity field analysis-based approach applied to underwater anthropomorphic robot’s structural components immersed in flow.

2. The 3-D velocity field of flow around ocean engineering facilities
Let us consider 3-D velocity field calculation methods of wave and constant currents flowing around underwater anthropomorphic robot’s structural components.
So far, there are a significant number of publications devoted to experimental, theoretical and numerous methods of investigating viscous fluid flow around various obstructions, [5] to [6]. Nevertheless, the problem cannot be regarded as “settled” because in the turbulent regime, the solution of the governing
The hydrodynamic equations becomes unstable even on finer grids and smaller time steps. Reynolds-averaging of Navier-Stokes equations is used to solve time-dependent problems associated with interaction of various currents with objects immersed therein. In the simplest case, use is made of the Boussinesq hypothesis where the tensor is expressed in terms of gradients of averaged flow velocity components and the constant eddy viscosity $v_t$. It enables us to formally replace the kinematic viscosity by some constant parameter $v_t$, which is intrinsically multiple-valued and is normally selected by imposing a requirement to obtain a stable solution for the specified boundary conditions, grid mesh and time step.

The below system of equations is used to calculate the three-dimensional velocity field $u_i, i = 1, 2, 3$ of incompressible fluid flowing around a solid obstruction:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j - \tau_{ij})}{\partial x_j} + \frac{\partial p}{\partial x_i} = S_i
\]

\[
\frac{\partial \rho E}{\partial t} + \frac{\partial}{\partial x_j} \left[ (\rho E + p)u_j + q_j - \tau_{ij}u_i \right] = S_j u_j + Q_H
\]

where $S_i$ denotes the distributed external forces per unit mass, $E$ is the total flow energy per unit mass, $Q_H$ is the heat flux per unit volume, $q_i$ is the diffusive heat flux, $\tau_{ij}$ is the viscous stress tensor defined as:

\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_l}{\partial x_l} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij},
\]

where $\mu = \mu_t + \mu_l$, $\delta_{ij}$ is the Kronecker delta function, $\mu_l$ is the dynamic viscosity, $\mu_t$ is the turbulent viscosity, $k$ is the turbulence kinetic energy. Subscripts denote the ox-, oy-, oz-axes of the three-dimensional Cartesian coordinate system.

The below $k - \varepsilon$ relationships of the turbulence model are used in calculations:

\[
\mu_l = f_\mu \frac{C_\mu \rho k^2}{\varepsilon},
\]

\[
f_\mu = \left[ 1 - \exp \left( -0.025R_y \right) \right]^2 \left( 1 + \frac{20.5}{R_T} \right),
\]

\[
R_y = \frac{\rho_y \sqrt{k}}{\mu_l}, \quad R_T = \frac{\rho k^2}{\mu_l \varepsilon},
\]

where $y$ is the distance to the solid surface; $C_\mu = 0.09$. Function $f_\mu$ allows for transition from laminar to turbulent flow. $k$ and $\varepsilon$ are obtained by solving the following system of equations:
\[ \begin{aligned}
&\frac{\partial(\rho \mathbf{u})}{\partial t} + \frac{\partial(\rho \mathbf{u} \mathbf{u})}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\nu} + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + S_i \\
&\frac{\partial(\rho \mathbf{e})}{\partial t} + \frac{\partial(\rho \mathbf{e} \mathbf{u})}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\nu} + \frac{\mu_t}{\sigma_e} \right) \frac{\partial \mathbf{e}}{\partial x_j} \right] + S_e
\end{aligned} \]

where

\[ S_i = \tau_{ij} \frac{\partial u_j}{\partial x_j} - \rho \mathbf{e} + \mu_t P_B, \]

\[ S_e = C_{el} E \left( \sum_{i,j} \tau_{ij} \frac{\partial u_i}{\partial x_j} + \mu_t C_B P_B \right) - C_{e2} f_2 \frac{\rho \mathbf{e}^2}{k}, \]

\[ P_B = -g_i \frac{1}{\sigma_B} \frac{\partial \rho}{\partial x_i}, \]

where \( g_i \) is the \( x_i \)-component of acceleration due to gravity; \( \sigma_B = 0.9 \); \( C_B = 1 \) at \( P_B > 0 \), and \( C_B = 0 \) at \( P_B \leq 0 \); \( C_{el} = 1.44 \), \( C_{e2} = 1.92 \), \( \sigma_e = 1.3 \), \( f_1 = 1 + (0.05 f_u^{-1})^3 \), \( \sigma_k = 1 \), \( f_2 = 1 - \exp(-R_T^2) \).

The diffusive heat flux is described by the following equation:

\[ q_i = \left( \frac{\mu_t}{\nu} + \frac{\mu_t}{\sigma_c} \right) C_p \frac{\partial T}{\partial x_i}, \]

where \( C_p \) is the water specific heat at constant pressure; \( \nu \) is the Prandtl number, \( T \) is the fluid temperature, \( \sigma_c = 0.9 \).

The boundary conditions for solving this time-dependent problem of the uniform flow around a cylinder are defined by steps as follows:

1. Definition of the computational domain with the size along axes \( \alpha x(L_{ox}), \alpha y(L_{oy}), \alpha z(L_{oz}) \) which is 10 to 15 times the size of the cylinder immersed in flow, and adoption of the reference Cartesian coordinate system \( (x, y, z) \). The \( x \)-direction coincides with the velocity direction, the \( y \)-direction is along the cylinder radius, the \( z \)-direction is along the cylinder axis, as shown in Figure 1.

2. Definition of the geometry of the cylinder immersed in flow, i.e. its surface coordinates in the reference coordinate system. The no-slip condition is applied on the solid surface of the object immersed in flow.

3. Specification of the constant flow velocity \( u_i = U_0 \) (along the \( \alpha x \)-axis) on the boundary faces \( AA'CC' \), \( ABCD \), \( A'B'C'D' \).

4. Specification of the Sommerfeld outflow boundary (face \( BB'D'D' \)) condition with radiation for the normal components of the velocity.

The initial conditions are also specified for the purpose of solving the problem. In the case under consideration, the initial uniform constant velocity \( (U_0) \) flow along the \( \alpha x \)-axis is specified.

The reference grid is specified as a cluster of equally spaced rectangular cells, their quantity predicated by the computational domain’s and the object’s sizes, and initially amounting to 15 000 cells. As the computation proceeds, the gradients of certain physical properties in the adjacent cells are analyzed, and where the gradient turn out greater than the respective specified values, the reference cells are broken down by use of finer meshes.
By each breaking down, a reference cell is subdivided into eight parts and provision is made for up to seven further breakdowns. Where the gradients become lower than the respective specified values, the cells are brought into coincidence.

The grid is adaptive and is varied during the computation so as to minimize cell sizes in the areas where greatest gradients of physical quantities would occur. The problem is being solved by the finite volume method wherein the discrete solutions are obtained as cell volume averages and those values are assumed to hold at the cell center. The viscous shear stress tensor, $\tau_{ij}$ is parametrized at each discrete computation step and in each mesh cell by simultaneous solving equations (1) to (4) and (9) to (11).

Let us consider the computation of velocity and pressure fields of a cylinder immersed in a steady uniform flow at the velocity $U_o$, the cylinder’s diameter $D$ and length $l_D$.

Initially analyzed is the case where $l_D/D = \lambda_D > 10$, i.e. a large aspect ratio cylinder is immersed in flow, which allows the use of the 2D approximation. Since the problem is considered for $Re = (U_oD/\nu) \geq 10^3$, the flow near the cylinder surface is turbulent, and the specification of initial conditions requires estimating turbulence parameters, i.e. the turbulence intensity $I_t$ and the mixing length $l_I$. Let us use the boundary layer equation:

$$
\begin{align*}
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= - \frac{1}{\rho} \frac{\partial p}{\partial x} + v_t \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right), \\
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0
\end{align*}
$$

(13)

where $x, y$ are the coordinates along the surface of the cylinder immersed, and normal to that, respectively; $u, v$ are the velocity components along those coordinate axes, $\rho$ is the density, $v_t$ is the turbulent viscosity.

The following dimensional scales are introduced $D, \delta, U_o, V_o, T, L, P$ and there are dimensionless variables $x^*, y^*, u^*, v^*, t^*, p^*$. The condition $x = Lx^*$ holds where $L$ is the distance at which the boundary layer of thickness $\delta$ is generated, $y = \delta y^*$, $u = U_o u^*$, $t = T t^*$, with the time scale determined based on the condition $U_o T / L = U_o T / D \approx 1$, $p = P p^*$; $v = -(U_o \delta / L) v^*$, where the normal velocity component scale is determined from the continuity equation. The estimation of the turbulent viscosity:

$$
v_t = 1.5(\delta / L)^2 DU_o.
$$

(14)
In addition, using the empirical relationship \( \delta / L = 0.344(\log \text{Re})^{-1.62} \), yields:

\[
\nu_f = 0.1775DU_o (\log \text{Re})^{-3.24}.
\] (15)

Further, application of the \( k - \epsilon \) model relationships, results in:
- the turbulence kinetic energy as

\[
k = \left( \frac{\nu_f}{l} \right) = \frac{0.266U_o^2}{\kappa^2(\log \text{Re})^{3.24}},
\] (16)

where the mixing length \( l = \kappa \delta \), \( \kappa = 0.41 \) is von Karman’s constant;
- the dissipation rate of the turbulence kinetic energy per unit volume given by:

\[
\epsilon = \frac{C_\mu k^{1.5}}{l} = \frac{0.398C_\mu U_o^3}{\kappa^4D(\log \text{Re})^{3.24}},
\] (17)

where \( C_\mu = 0.09 \);
- the minimal size of a turbulent eddy given by \( \lambda_t = (\nu^3 / \epsilon)^{0.25} \), where \( \nu = 1.4 \times 10^{-6} \text{ m}^2/\text{s} \) is the kinematic viscosity;
- the standard deviation of turbulent fluctuations of velocity \( \sigma_u \) and the turbulence intensity \( I_t \) given respectively by \( \sigma_u = \nu / \lambda_t \), \( I_t = \sigma_u / U_o \).

3. The numerical simulation of velocity and pressure fields evolution for the cylinder immersed in flow

The numerical simulation of the velocity and pressure fields evolution for the cylinder immersed in flow is reasonable to consider in terms of a dimensionless time function \( t_s = tU_o / D = t/T_w / St(\text{Re}) \), where \( St \) is the Strouhal number, \( T_w \) is the period of vortex shedding in wake formed behind the cylinder. This permits the comparison of various typical stages of the velocity field generation with experimental data obtained for similar flows.

Figure 2 and Figure 3 show the calculation results of the velocity and pressure fields for the flow past a large aspect ratio cylinder at Reynolds numbers \( \text{Re} = 10^3 \).

At \( \text{Re} = 10^3 \), \( t_s = 0.504 \), the streamlines of the flow around the cylinder are close to those of a laminar flow. There is no vortex shedding in wake behind the cylinder. The flow separation region starts to generate downstream of the cylinder (Figure 2 a).

In the pressure field, the following two regions are generated: a lower pressure region at the cylinder’s lateral sides in the increased-velocity area, and a higher-pressure region upstream and downstream of the cylinder.

At a further stage of wake generation at \( t_s = 2.5 \), the flow is separated from the cylinder surface at the top face, with two symmetrical vortices generated downstream and some little vortices generated near the separation points (Figure 2 b).
Further stages of wake generation for the flow past the cylinder are illustrated in Figure 2 for \( \text{Re} = 10^5 \). \( t_s = 3.08 \text{ s} \) (Figure 3 a) is the point of the inception of instability of the interface surface between vortices downstream of the cylinder, where a wake of vortices, whose axes are parallel to the cylinder axis, develops behind the cylinder (known as the Karman vortex street).

Typical flow velocity fields and pressure variations in a steady wake flow behind the cylinder are shown in Figure 3 b. In this example, the initial wake symmetry is broken and successive vortices are generated in the flow.

In addition to the simulation of a time-dependent flow past a large aspect ratio cylinder, a number of calculations of a 3D flow past a small aspect ratio \((\lambda_D = l_D / D = 1)\) cylinder have been performed.

For the flow normal to the small aspect ratio cylinder’s axis, the pattern is governed by flat faces which permit the flow moving in the cylinder generatrix direction. The lower the length of such a cylinder, the far more considerable is the influence of the near face flow on the general flow pattern.

Simulation results of the flow past small aspect ratio cylinders are reasonable to consider in terms of a dimensionless time function \( t_s = 0.2 t U_0 / (C_{x\lambda} D) \), determined by the period \( T_v \) of vortex shedding near the cylinder surface. This quantity, in turn, meets the condition \( C_{x\lambda} \cdot St \approx 0.2 \) at \( 0.4 \leq C_{x\lambda} \leq 1.6 \).

4. Conclusion

The article deals with a simulation approach of time-dependent 3-D velocity and pressure fields of the flow past cylinders representing anthropomorphic robot arms. The proposed approach ensures the reproduction of principal features of a behind-the-cylinder wake generation process. There are the principles of simulating 3-D flows around large- and small aspect ratio cylinders.

The proposed approach could as well be applied while developing 3-D eddy-resolving models of time-dependent velocity and pressure fields in the investigation of interaction of ocean currents with complicately configured ocean engineering systems such as anthropomorphic robots operated underwater.
5. Acknowledgments
The reported studies were supported by the Ministry of Science and Higher Education of the Russian Federation (project RFMEFI57818X0264. The agreement on grant № 075—02-2018–158).

References
[1] Kushnir V M, Dushko V R, Kramar V A 2016 Nonlinear dynamics of ocean systems Infra-M
[2] Van Oord 2002 Proceedings of ICSF-1. Intern. Conf. on Scour of Foundations 1-11
[3] Grant W D. and Madsen Ô S 1979 J. of Geophys. Res. 84(C4) 1797 – 1808
[4] Hsu T-W 1998 Flow Modeling and Turbulent Measurements VII 27-37
[5] Kushnir V M, 1999 J. Marine Systems 21 243-253
[6] Faria A F; Thornton E B; Stanton T P; Soares C V, Lippmann T C 1998 J. Geophys. Res. 103(C2) 3217-3232