Flow analysis of new type propulsion system for UV’s

M Eimanis\textsuperscript{1} and J Auzins\textsuperscript{2}

\textsuperscript{1} PhD Student, Institute of Mechanics, Riga Technical University, Riga, Latvia
\textsuperscript{2} Dr.sc.ing., Professor, Institute of Mechanics, Riga Technical University, Riga, Latvia

E-mail: marcis.eimanis@gmail.com

Abstract. This paper presents an original design of an autonomous underwater vehicle where thrust force is created by the helicoidal shape of the hull rather than screw propellers. Propulsion force is created by counter-rotating bow and stern parts. The middle part of the vehicle has the function of a cargo compartment containing all control mechanisms and communications. It’s made of elastic material, containing a Cardan-joint mechanism, which allows changing the direction of vehicle, actuated by bending drives. A bending drive velocity control algorithm for the automatic control of vehicle movement direction is proposed. The dynamics of AUV are simulated using multibody simulation software MSC Adams. For the simulation of water resistance forces and torques the surrogate polynomial metamodels are created on the basis of computer experiments with CFD software. For flow interaction with model geometry the simplified vehicle model is submerged in fluid medium using special CFD software, with the same idea used in wind tunnel experiments. The simulation results are compared with measurements of the AUV prototype, created at Institute of Mechanics of Riga Technical University. Experiments with the prototype showed good agreement with simulation results and confirmed the effectiveness and the future potential of the proposed principle.

1. Introduction
Innovations and the development of new products usually require concepts that are not yet implemented at the time. Of course, prototyping is quite expensive and time-consuming, therefore it is no longer possible to test new products in the field without previous verification of their behavior in computer simulation. This paper presents a new type of drive for underwater vehicles that borrows from the Escherichia coli and Spirochaetes bacteria. Currently the most widely used drive for water vehicles is the propeller drive which is also the closest prototype for the drive presented in this paper. This propulsive force generator is also used for automated underwater vehicles, such as ships, submarines and other water vessels. However, propellers are efficient with relatively low speeds. As the speed of movement increases, cavitation bubbles are created on the surface of the propeller, affecting the propeller and causing material erosion. This significantly reduces the service life and safety of the propeller. At Riga Technical University the design of AUV was developed using two reflection-symmetric parts with a helical shape [1]. The parts are connected with motorized rotational joints. The vehicle contains a central part (see Fig.1) with the hull built from elastic material. The central part contains a Cardan (universal) joint with two servomotors.
Figure 1. The structure of Durbis-2. 1 – rear and front screw, 3 – mid-body front part, 4 – mid-body rear part, 5 – cross/ vector kite gimbal, 6 – servo, 7 – servo connector, 8 – shaft connecting connector and gimbal ring, 9 – analog signal receiver, 10 – battery, 11 – PDM type speed controller for DC motors, 12 – hull with DC motor and reducer, 13 – drive shaft, 14 – screw axis.

The motorized Cardan joint implements two-directional bending of the hull and allows maneuvering the vehicle. The direction control algorithm uses angular orientation sensor information. There are several scientific papers studying the performance of propellers in various media and circumstances, for example, [2, 3, 4], but the drive proposed in this paper is completely new, not implemented at this point in time in practice, therefore it is necessary to apply new computer modelling, fluid-structure interaction and flow analysis methods. The main fundamental difference from existing propulsion systems is that the hull body itself is the propulsive force generator.

The simulation of the underwater dynamics of a rapidly rotating and bending vehicle is a hard problem. Rotation causes turbulence and vortex motion of the fluid and a very complex interaction between the fluid and the vehicle body and between parts of the vehicle. Currently it is possible to simulate only a straight motion of the vehicle with commercial CFD software and the simulation is very time-consuming. The ICP/TP connection between multibody software and CFD software [5] is possible, but the stability and accuracy of such a calculation approach is highly dubitable. Therefore the dynamics of the AUV with 10 degrees of freedom was simulated using the multibody system simulation software MSC Adams. The fluid-mechanism interaction was simulated using metamodels of water resistance forces and moments, obtained by approximation of results of numerical experiments with CFD software Flow-3D. Although currently there is no research and analysis of the turbulence and cavitation effects available, the system shows very good maneuverability and efficiency with good energy consumption parameters. The idea of this system is described in [6] and [7], while fluid-structure interaction and drive control simulation are described in more detail in this work using the metamodeling approach.

2. Control principle for helicoidal AUV

The control principle is based on bending the Cardan joint, where it is necessary to orientate the bow (front) rotational axis in the target direction. The rotation axis, which is responsible for tilting the bow, is perpendicular to screw rotation axis unit vector $b$ and to vector $a$ from the center of vehicle to the target. The unit vector $T$ of this finite rotation is equal to the cross product of both vectors, divided by the norm of vector $a$, Figure 2.

$$ T = \frac{1}{\|a\|} b \times a $$

(1)
Figure 2. Kinematic diagram of Durbis-2. 1, 5 – bow and stern screws, 2 and 4 front and rear drive sections, 3 – Cardan journal cross (spider), \( \varphi_y \), \( \varphi_z \) – tilt angles around Cardan joint axes, \( a \) – vector from the middle point of the Cardan joint to the target point, \( b \) – unit vector of bow screw rotation axis.

The control drive torques significant and the stern part will bend in the opposite direction. Assuming that the control system is equipped with sensors giving both current angles \( \phi_y \) and \( \phi_z \) of the servo drives, as well as the projections of direction vector \( a \) on the axis of the central coordinate system (fixed on the Cardan journal cross), the control servo drives can implement the required angle or rotation speed. In the case of speed control, the relatively simple control law can be implemented:

\[
\begin{align*}
\dot{\phi}_y &= -k_c \arcsin(T_y) \\
\dot{\phi}_z &= k_c \arcsin(T_z)
\end{align*}
\]  

(2)

Here \( k_c \) is a constant coefficient. This algorithm, instead of axis vector \( b \), can use the vehicle linear velocity vector. A very important problem is the sensor system. It should include three-axis orientation sensors, because the vertical \( z \)-axis of Cardan joint cannot always be oriented in vertical direction. This is due to the capability of the middle parts to rotate around the longitudinal axis of the vehicle. This is the reason why simple manual control of the vehicle is practically impossible, because the turn \( \phi_z \) for the middle body is not always equal to yaw and \( \phi_y \) is not always equal to pitch rotation.

3. Surrogate models of water resistance

Since it is practically impossible to simulate a moving mechanism with 10 degrees of freedom in CFD programs taking into account full interaction between fluid and mechanism links, in practice the metamodeling (surrogate modeling) technology is broadly applied [8]. This implies the creation of simplified models of water resistance, obtained by planned experiments with CFD software. Several papers explain advantages of surrogate modeling over other methods and the different aspects of surrogate modeling [9, 10]. The authors of [8] used the classic Response surface method based on second order polynomial approximations of computer experiments with CFD software ANSYS FLUENT.

The created polynomial surrogate models for water resistance forces \( F_i \) and torques \( T_i \) are as follows

\[
\begin{align*}
F_i &= -A_i^T V_i - \|V_i\| B_i^T V_i \\
T_i &= -C_i^T \omega_i - \|\omega_i\| D_i^T \omega_i
\end{align*}
\]  

(3a)

where \( F_i \) – column-vector of total water resistance force on the \( i \)-th body, \( T_i \) – column-vector of total water resistance torque acting around the center of mass of the \( i \)-th body, \( V_i \) - column-vector of translation velocity of center of mass of \( i \)-th body, \( \omega_i \) - column-vector of translation velocity of center
of mass of $i$-th body, $A_i$, $B_i$, $C_i$, $D_i$ – columns of coefficients, obtained by least-square approximation. All vector projections are calculated in body (moving) coordinate systems. Creating such surrogate models, the flow change from vehicle bending in the Cardan joint is not taken into account.

4. Simulation of rectilinear motion

Commercial CFD software ANSYS FLUENT, COMSOL Multiphysics, STAR-CD, Flow-3D and others provide limited capacity for modelling fluid-mechanism interaction dynamics. In these programs it is possible to insert into a fluid flow rigid bodies with all 6 degrees of freedom or constrain the motion excluding some translation or rotation degrees. The so-called General Moving Object (GMO) components can be of a mixed motion type, namely, have translational and/or rotational velocities that are coupled in some coordinate directions and locked in the other directions. A body-fixed reference system (“body system”), defined for each moving object, and a space reference system (“space system”) are employed. Therefore, these rigid bodies can be coupled with the ground using rotational or translational joints. Unfortunately, it is not possible to connect two moving bodies among themselves using rotational joints.

To model rectilinear motion using software Flow-3D we used the following approach. Three main parts: bow (front) screw, middle body and stern (rear) screw are placed in alignment, allowing only rotation around longitudinal axis $x$. The middle body is standing still. In this model none of the three parts have any contact with each other and there are no contact forces or friction forces between them. External and opposite torques $Q$ and $-Q$ around $x$-axis act on, respectively, the bow (left) screw and the stern (right) screw. The mesh domain size in $y$ and $z$ directions was built in accordance with the recommendations given in [11] - approximately 1.4 times larger than the diameter of vehicle. The specified pressure conditions instead of wall-type boundary conditions were used. Also the idea that is applied in all wind tunnel experiments is used, i.e. the object may be moving through a stationary fluid, or the fluid may be flowing past a stationary object. In our simulations the vehicle is placed so that its center of mass is fixed in the inertial coordinate system and the water flows past a stationary vehicle. CFD software calculates the fluid pressure and velocity in all mesh points as well as the total resistance forces and torques acting on all GMOs. Also the reaction forces and torques created by motion constraints are calculated. And in our case there are three components of support reaction force and two components of reaction torque for each part of vehicle. Here the most important are the reaction force components in longitudinal direction, the other forces and torques are relatively negligible. We need to assume that the modelled movement is equivalent to vehicle traveling with constant drive torques $q_1 = q_2 = Q$, so it is necessary that the sum of reaction forces (called residual control $x$-force in the program Flow-3D [12, 13]) in longitudinal direction would be equal to zero (see Figure 3):

$$R_{x1} + R_{x2} + R_{x3} = 0$$  \hspace{1cm} (4)

We managed to satisfy this condition by experimentally determining the (external) drive torque value $Q$ for each given fluid velocity value $V_x$, trying out different software input settings.

![Figure 3. Top view of Durbis-2 in the finite volume mesh of the software Flow-3D.](image-url)
For turbulence modeling the Chen-Kim modification of Renormalized group (RNG) $k$-$\varepsilon$ model [12, 14] has been used, with following coefficient values:

$$\left(\sigma_k, \sigma_\varepsilon, C_{e1}, C_{e2}, C_{\mu}, \beta\right) = (0.7194, 0.7194, 1.42, 1.68, 0.0845, 4.38, 0.012)$$

Figures 4, 5 show the water streamlines and flow horizontal velocity contours for the fluid common velocity 0.6 m/s. Images show that the rear screw works inefficiently and pulls along the water behind it.

Figure 4. Water streamlines around Durbis-2. Velocity 0.6 m/s.

Figure 5. Fluid $x$-velocity contours in the middle section.

Figure 6 shows the stabilization of rotational velocity of bow and stern screws and the oscillation of summary longitudinal reaction force for the given flow velocity 0.6 m/s and drive torque 0.146 Nm.

Figure 6. Stabilization of the screw rotation speed (left) and summary support reaction (right).
Figure 7 shows the approximated dependence of the drive torque on the velocity of rectilinear motion. Only four numerical experiments were used for obtaining this second order polynomial approximation. 0.6 m/s flow speed was assumed, taking into account the movement of the prototype in real environment conditions, but the other two flow speeds – 0.3 m/s and 0.8 m/s - were freely chosen as points from the possible movement speed range of the prototype.

![Figure 7](image_url)

**Figure 7.** Approximate dependence of drive torque (left) and screw angular velocity (right) on the linear velocity of AUV.

The right graph of Fig. 7 shows the approximated linear dependence between the velocity of rectilinear motion and angular velocities of bow and stern screws. As can be seen, at equal (opposite) drive torques, the rear screw rotates significantly slower than the bow (front) screw. The fluid pressure and shear force analysis showed that the contribution of the rear screw in the creation of the thrust force is also significantly smaller.

5. **Dynamic simulation with MSC Adams**

The world’s most famous and widely used 3D Multibody Dynamics (MBD) software tool, MSC Adams, is used for dynamical analysis of mechanisms and machines, including mechanisms with rigid and flexible links, drive and control systems. Here the mathematical model in the form of internal systems of differential-algebraic equations is built automatically according to the kinematic diagram which can be imported from any CAD software.

The simulated mechanism contains 5 rigid-body links, connected with 4 rotational joints, see Fig. 8, 9.

![Figure 8](image_url)

**Figure 8.** Durbis-2 in MSC Adams. Outside view. Cardan journal cross not shown.
For bow and stern parts, there is one rotational joint for each which plays the role of engine, and in the middle section there are two rotational joints which implement the function of the cardan journal cross for changing the movement direction. The same set of forces as for bow screw is also applied to the stern screw. Therefore, the model has 10 degrees of freedom. Using the created surrogate models and the possibilities of the function builder in MSC Adams, external forces describe the water resistance forces and torques. Fig. 10 shows the “General force” property for the bow screw, which is in fact the implementation of quadratic approximation of flow simulation results (3a, 3b).

The thrust and bending electro-drive dynamics were not simulated in detail. It was assumed that the thrust drives generate the necessary torque and orientation drives generate the necessary rotation speed. The angular velocity of bow and rear screws was varied in the range from zero to 20 RPM. When modelling rectilinear motion, at ±10 RPM the linear velocity of vehicle was obtained as about 0.6 m/s. Figure 11 shows the vehicle following a target, using an automatic control law (2). The spherical target moves in three dimensions according to sinusoidal law \( x(t) = 0.7 \sin(0.5t) \), \( y(t) = 0.7 \cos(0.5t) \), \( z(t) = 0.46 \sin(1.5t) \). The optimal value of the coefficient of control law (2) \( k_c =110 \) was
found using computer experiments. As can be seen in the aforementioned figure, the automatic control law (2) makes the model maintain a stable path while following the moving target.

Figure 11. Following a moving target.

6. Conclusions

- Dynamics computer simulations provided sufficiently good results to predict the behavior of the model in a real medium. These results were also confirmed by the physical prototype.
- The full dynamics simulation leads to conclusions about the high performance of the model that includes functionality, maneuverability and energy efficiency, as also proven by the model prototype in real open water environment.
- Even though only 2 degrees of freedom from 10 were retained in the fluid flow impact simulation, it was nevertheless very time-consuming and computationally intensive.
- The proposed drive principle may be employed not only in a water medium, but also for “crawling” on land, through pipes or to dig through a loose medium, for example, grain, carrying out manipulations required in industry and science.
- Shape optimization of the hull and both screws would also be necessary. With the current design where the vehicle has almost completely symmetrical bow and stern parts, the stern screw has a mostly supportive function and the main thrust force is created by the bow screw. If the symmetry principle is abandoned, it would be possible to create a more energy-efficient shape for the double-helical vehicle.

References

[1] Eimanis M 2014 Development of new type propulsion device for underwater vehicles Master thesis Riga Technical University https://ndr.rtu.lv/lv/view/11681/
[2] Subhas S, Saji V F, Ramakrishna S and Das H N 2012 CFD Analysis of a Propeller Flow and Cavitation International J. of Computer Appl. 55 pp 26-33
[3] Chao X, Cees B and Sherman CP C 2014 Fluid dynamics analysis of a counter rotating ducted propeller 29th Congr. of the Int. Council of the Aeronautical Sc. (ICAS2014) Russia pp 1-8
[4] Sulaiman O O, Kader A S A, Wan Nick W B and Saharuddin A H 2012 CFD Simulation for Cavitation of Propeller Blade Global J. of researches in Engineering (Automotive Engineering)12 pp 1-10
[5] Elliott A, Slattengren J and Buijk A 2006 Fully Coupled Fluid/Mechanical Response Prediction for Truck-Mounted Tank Sloshing Using Cosimulation of MSC.ADAMS® and MSC.Dytran® SAE Technical Paper 2006-01-0932, doi:10.4271/2006-01-0932.
[6] Auzins J, Eimanis M 2015 Dynamical Simulation and Optimization of Double-Helical AUV Computational Methods in Marine Engineering VI (MARINE 2015) Italy pp 1128-1139
[7] Eimanis M, Auzins J 2016 Flagella Inspired Propulsion for AUVs 10th Symposium on High-Performance Marine Vehicles (HIPER’16) Italy pp 342-354
[8] Carroll J and Marcum D 2013 Developing a Surrogate-Based Time-Averaged Momentum Source Model from 3D CFD Simulations of Small Scale Propellers Lecture Notes in Engineering and Computer Science: Proceedings of The World Congress on Engineering London pp. 1622–27
[9] Forrester A and Keane A 2009 Recent advances in surrogate-based optimization Progress in Aerospace Sciences vol 45 pp 50–79
[10] Queipo N V, Haftka R T, Shyy W, Goel T, Vaidyanathan R and Tucker P K 2005 Surrogate-based analysis and optimization Progress in Aerospace Sciences vol 41 pp 1–28
[11] Watanabe T, Kawamura T, Takekoshi Y, Maeda M and Rhee S H 2003 Simulation of steady and unsteady cavitation on a marine propeller using a RANS CFD code 5th Int. Symp. on Cavitation (CAV2003) Japan pp 1-4
[12] Flow Science Inc. 2011 FLOW-3D user’s manual 10.1 edition (Santa Fe, N.M.: Flow Science, Inc.)
[13] Wei G 2005 A fixed-mesh method for general moving objects in fluid flow. Modern Physics Letters B, Vol. 19, Issue 28-29 pp. 1719-1722
[14] Chen Y S, and Kim S W 1987 Computation of Turbulent Flow Using an Extended K-E Turbulence Closure Model NASA Report CR 179204