NUMERICAL STUDY OF FLUID FLOW AROUND A DIVER HELPER

Hesamedin habibi\textsuperscript{1,} Ali Yari\textsuperscript{2,} S. Hosseinzadeh\textsuperscript{3}* Saeed Darzi BurKhani\textsuperscript{4}

\textsuperscript{1,4}Department of Mechanical Engineering, University College of Rouzbahan, Sari, Iran
\textsuperscript{2}Department of Mechanical Engineering, University of Shiraz, Shiraz, Iran
\textsuperscript{3}* Corresponding author, Department of Mechanical Engineering, Islamic Azad University, West Tehran Branch, Tehran, Iran

ABSTRACT

Having access to high speed diving without the use of mechanical science and discovery centers have been considered. Production of simple, yet effective tool to reduce energy consumption and associated diver is very valuable. Assistant diver device that works with human muscle power, includes a pair of ballets. This system reduces the energy required to dive to less than half as the speed increases to 2 to 5 knot. Using numerical methods can answer a lot of questions and a simulation of the dynamic behavior of the device. In this article, modeling of fluid flow around the Diver helper of FLUENT software and using Dynamic Mesh have been done.

Flow lines show an increase in the angle of the fins and causes development of vortices behind them. Pressure Cantor can also be used in the analysis of the fins. The drag coefficient ballet based on the device at various angles in a period is reported in charts.

KEYWORDS

Hydro-foil, drag coefficient, lift coefficient, fin swimming, Computational Fluid Dynamics

1. INTRODUCTION

Diver helper is a portable device that can be used in reconnaissance operations, rescue and military substantially to work. Using a companion diver diving speed can be increased to twice the maximum speed as be stable. Any other equivalent device with these characteristics is unique. Much of this research has been on the swim fins. Ballet swimming fins like that help moving the water in underwater activities such as swimming, diving and water sports covered on foot.

Pendergast et al. investigated effects of geometry, mass, density and roughness factor on the speed and efficiency of economic analysis examined as the use of swim fins \cite{4, 5}.

Toshiyant et al. and Zamparoet al. performed two separate studies comparing the use of the fins with their bare feet \cite{1, 2, and 3}. However, the exact impact on the performance characteristics of swimming fins quantifies the important issue. To date, the most common way to assess swim fins driving force is based on studies of "swimmer fins" as a global system which includes biomechanical and physiological responses and physiological point of view, oxygen consumption to assess the amount of energy used in swim fins \cite{6, 7, 8, and 9}.

But only few studies on the relationship between the physiological and mechanical characteristics of ballet or swimming techniques have been done on the subject. The last parameter directly

DOI : 10.14810/ijmech.2016.5105
related to the performance of the ballet requires a biomechanical study. Kinematic studies, analysis of swimming techniques are proved with this model [1, 2].

However, this type of analysis investigated the relationship between cause and effect parameters and did not clear properly and can only be assessed through a variety of dynamic levels. In early studies, the maximum forces in swimming with fins different place are compared [10, 11, 12, and 13].

But the real situation is very different condition of water flow around a swimmer and reported values may be overestimated [14]. In another article, the measured values have been reported in swimming motions [15].

Using a methodology that is used in another article is considered for analysis [2, 3] can be used for economic analysis and calculation of the efficiency of swimming with fins at low speeds to be used. However, the assessment of the effects on the overall performance of the ballet alone will be more difficult. The alternative method is considered only in the fin system.

Given the interaction of the fluid and to address this deficiency, swimming fins as different authors are under the terms of the quasi-static mode (wind tunnel waterway) to evaluate the drag coefficient and lift [16, 17]. In spite of such a manner, the impact of transitory is taken into consideration [18].

It is noteworthy that the former swimmers care swimming techniques that can affect the test and this may lead to differences in the results. The various techniques used by the swimmer make it difficult to differentiate the effects of fin effects.

In this paper designed a new device called a Diver helper, it can be replaced as the original design of the device with fluid flow modelling FLUENT software according to using Dynamic Mesh around.

2. PRODUCTION OF GEOMETRY

One of the most important phases in numerical analysis is modeling object which is analyzed. By way of analysis, modeling is more sensitive. Model must have certain characteristics. There are many applications for modeling and production volume.

The project is due to the need for high accuracy and volume of the body of Catia software that is used to generate geometry. The advantages of this software can include high precision surface grinding, ability to work with the original parts, CNC machine connections and generating output files that have the capability to mesh software that can be mentioned.

Turning the volume level and the output file was sent to the software Gambit. Gambit is one of powerful software applications in the field of Mesh. The software output as input software Ansys-Fluent is considered. How the system designs the first leg of the diver companion embedded in a rail mounted on the diver's legs, is due to bending the knee to move in the water there. This type of diving water is drawn from the tail of a dolphin. The main objective for the installation of various components is stabilizing at the end of the leg introduced as a companion diver machine in Figure 1.

1. The core of the device is that the various components are mounted on it.
2. The device includes a pair of fins, and is the main cause for the device.
3. Trans-ferrous Bush attached to the front of the device that is placed in it.
4. Trans-iron is the main interface and ballet.
5. Copper wire that connects the spring ballet interface
6. Shock device that can determine the flow of the ballet
7. Leg of that diver to use it to your device closes
8. The page that leg through the bolts attached to the main axis
9. Bolts M6 and M5 that connects the main axis parts.
10. Stabilizing device which prevents distortion when scuba diver is on track and to be desired.
11. Piece L-shaped stabilizer is attached by bolts to the core.

Figure 1. a) Three-dimensional design of software Catia

b) Introduction of system components

3. DOMINATED EQUATIONS

Assuming incompressible flow, mass conservation equations (continuity) and conservation of momentum (Navier-Stokes) are as follows:

\[ \text{div } U = 0 \]  

\[ \frac{\partial (\rho u)}{\partial t} + \text{div} (\rho u U) = -\frac{\partial P}{\partial x} + \text{div} (\mu \text{grad}(u)) \]  

\[ \frac{\partial (\rho v)}{\partial t} + \text{div} (\rho v U) = -\frac{\partial P}{\partial y} + \text{div} (\mu \text{grad}(v)) \]  

\[ \frac{\partial (\rho w)}{\partial t} + \text{div} (\rho w U) = -\frac{\partial P}{\partial z} + \text{div} (\mu \text{grad}(w)) - \rho g \]
In this equation, \( \mathbf{U} \) is velocity vector, \( P \) is static pressure on the fluid, \( \mu \) is viscosity and \( w \) and \( v \), \( u \) are the velocity vector components. To model the movement of float, over time is used to solve the equations of motion of rigid body. Rigid body motion of the coordinate system is used to check system. The inertial coordinate system is connected to the ground and XYZ coordinate system is a free system that has accelerated the pace.

Equation of motion of the machine axis as a function of time is:

\[
y = 4\sin(2\pi t)
\]

Disruption to the existing two-equation \((k - \varepsilon)\) model has been modelled. In this model, the turbulent times turbulent flow energy is according variables \(K\) and dissolution rate of viscous turbulent energy \((\varepsilon)\) States:

\[
\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho \mathbf{U} k) = \nabla \cdot \left( \mu \frac{\partial k}{\partial s} \right) + P_k - \rho \varepsilon
\]
\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \mathbf{U} \varepsilon) = \nabla \cdot \left( \mu \frac{\partial \varepsilon}{\partial s} \right) + \frac{\varepsilon}{k} \left( C_{\varepsilon 1} (P_k) - C_{\varepsilon 2} \rho \varepsilon \right)
\]

4. **AREA MESHING SOLUTION**

One thing that is very important is to select the dynamic analysis of floating range and Mesh solutions.

For example: if the border around the object was small, the results are affected, causing large errors in results, and if the line is selected, the calculation time and therefore costs will increase. Therefore, we must seek an optimal size to be floating. By examining the documents (of ponds stretch) to the front, the total length to the back of the float, is four times the overall length of the vessel to the top and the bottom, the size of the vessel and to the side, twice the length of the float which is selected.

In Figures 2 and 3, we show the grid around the body of the device that is displayed. Due to the complex geometry of Meshing with tetragonal mode, tetrahedral elements have been used around the body.

In addition to reducing the number of cells in remote areas of the fuselage, an organized Mesh with cubic elements is used.
Figure 2. Production Meshes around the machine

Figure 3. Production of fins on the Mesh

5. INDEPENDENCE OF MESH

The total number of elements used in this case is 1532541. Meshes have the structure and size of the cells around the body of the device which is smaller. To evaluate the effect of improving the Mesh solution, the number of nodes is determined by Mesh, as suitable respectively is increased. To examine the independence of the Mesh, it has static mode and has been investigated in four different Meshing. By comparing the drag acting on the float, about 3/06 percent error between the solutions in the Mesh shows a large seal. But the important thing here is time. High computational mesh number was low due to the error between the Meshes. As seen in diagram 1, the error between stages 3 and 4 is very low. Taking into account that, the time as a factor for computing Meshing can be used to stage 3.

Diagram 1. Drag exerted on the device in four different Meshing
6. RESULTS AND DISCUSSION

To solve this problem in the dynamic of temporal resolution, fluent software is used to step the right time. If the resolution is according to the first time a step taken in the last step, we should reduce the amount of acceptable error system.

Figure 4. The pattern of flow lines around the device in landscape mode

Figure 4 and 5 lines flow around the device in landscape mode also shows a 30 ° angle. Horizontal mode, parallel lines and the flow to the body stick is seen at an angle of 30 degrees centered and phenomena such as separation from the end of the ballet shows the formation of vortex and Finally loss occurs downstream of the vortex. Physical instability causes this flow field and thus forces the surface of the device which also will be unstable.

Figure 5. The pattern of flow lines around the machine at an angle of 30 degrees
Figures 6 and 7, respectively, show pressure and velocity contours as three-dimensional image as the shell of the device. Darker color represents areas with high pressure and low speed more and more bright colors for areas with speed and pressure. As it is clear is that the central axis swing upward pressure on the upper area of the fins around axis is created. It is noteworthy that back ballet created by the shock process that creates thrust.

![Figure 6. Cantor pressure on the theme of the 0520/1 second time in the 4-degree angle Ballet](image)

Diagram 2 shows curve drag coefficient of resonant mode during a period of ballet as full swing. As can be seen, the drag coefficient is a non-periodic fluctuates. Physical instability in mentioned fluctuations is due to the flow field and thus forces the surface of the device that also will be unstable.

![Figure 7. Contour speed on the theme of the 0520/1 second time at 45-degree angle at the ballet](image)
7. CONCLUSION

In this article, flows of new devices that can replace the companion diver fins swimming in a swinging motion is studied. Lines vortex fluid flows around the fin shows that is possible by increasing the angle of the flaps created and expanded. The pressure field and the drag coefficient of the reciprocating motion bullet as contour and curve has been reported here. This can empirically is tested on the device and is checked as the actual flow around it and compared with modelling software.

REFERENCES


