I J E I T
The International Journal of Engineering and Information Technology
journal homepage:www.ijeit.misuratau.edu.ly
MISURATA UNIVERSITY

Motion Analysis of a Single Capsule in a Horizontal Hydraulic Pipeline

Sufyan A. Abushaala
Department of Mechanical Engineering,
University of Misurata، Misurata, Libya,
s.abushaala@eng.misuratau.edu.ly

Abstract— **A Computational Fluid Dynamics (CFD) based solver is used to numerically analyze the motion of a single spherical capsule in a horizontal hydraulic pipeline. A novel numerical model is employed with the aid of the dynamic mesh technique for calculating the pressure and the velocity of the moving capsule. The numerical model comprises a 2m long straight test section of 50mm diameter. The results from the numerical model were close to those from the experimental data from the test rig developed in the study. The capsule is assumed to be a point mass rigid body and the coaxial-annular-flow model around the capsule is used to derive the boundary conditions for water flow at the capsule discontinuity region. The results are compared with experimental work done.**

Index Terms—**CFD, capsule motion, 6DOF, HCP.**

I. INTRODUCTION

The capsule pipeline is a system which transports solids in capsules carried through a pipeline by a flowing fluid (usually water). The hydraulic capsule pipeline usually uses water as the carrier fluid and can easily carry heavy materials like ore. The capsule is suspended by water flow without contact with the wall in a horizontal pipeline even if the capsule is heavier than water. Thus, a hydraulic capsule pipeline does not always require wheeled capsules. On the other hand, a pneumatic capsule pipeline requires wheeled capsules because air is lighter than water. Liu et al [1] gives a detailed review of hydraulic capsule transport in general, and there are now many references available. Most of them, however, concern steady flow and very little is known about the unsteady motion of individual capsules in a pipeline or about transient water flow caused by the capsule motion. In this paper the author pursues the motion of a single spherical capsule in a horizontal straight pipeline from the feeding point to the pipe end. It was assumed that the capsules are spherical and that their posture in the pipe is always constant, and hence the 'lift off' and then the steady-flying. The analysis is now fully established and is applicable to this work.

Received 07 Feb, 2024 ; Revised 31 Mar, 2024 ; Accepted 14 Apr , 2024.
Available online 23 Jun , 2024

Lur'e et al [2] proposed a numerical analysis of pneumatic capsule transport which is based on the method of characteristics, and here the author applies their method with a little modification. A capsule is moved by fluid forces (pressure driven), consisting of the pressure and shear forces, acting on its surface. However, if the drag coefficient was used to calculate the forces in the unsteady motion of the capsule, a complicated problem concerning unsteady forces arises. This is due to the added mass which is important in this research because the density differences between the fluid and the capsule is relatively small. This problem is avoided if the forces are directly calculated by integrating the pressure and shear stresses acting on the capsule surface. The forces are related to water flow around the capsule, which is determined by the capsule geometry.

A flow model that does not pass water through the clearance between the capsule and the pipe wall is one of the simplest. In this research, the model for the fluid forces turns out to be pressure force alone and the capsule velocity is not necessarily equal to the water velocity. A model that calculates the clearance flow using the equation of orifice discharge is called an orifice flow model and is often used for the pneumatic capsule. In this model the capsule velocity is predicted and it is always in the direction of the water flow through the pipe. In this model, the capsule velocity may exceed the water velocity and, as is well known, this is not always the case for the hydraulic capsule. Therefore, it is necessary to devise a more elaborate model. In this paper, a pressure-based solver has been selected for solving the capsule flow. There are many models available in the commercial CFD packages to model the turbulence, each one of those models has its own advantages and disadvantages.

Due to the formation of a wake region downstream of the capsule, and the flow separation, k-ε model has been selected for simulating the turbulence for the current problem [3]. The reason for choosing k-ε model is for its power for simulating the wake regions accurately. In addition, results are given from experiments on single-capsule motion in a horizontal pipeline with a constant head. These results are compared with the numerical analysis. Transient modelling enables development of prediction models that incorporate transient effects such as start-up requirements.

II. THEORY

1) *Basic equations*

The governing equation for the translational motion of the center of gravity is solved for in the inertial coordinate system:[4]

$$\dot{\vec{v}}_G = \frac{1}{m}\Sigma \vec{f}_G \qquad (1)$$

Where $\dot{\vec{v}}_G$, is the translational motion of the center of gravity, m is the mass, and $\vec{f}_G$ is the force vector due to gravity. The angular motion of the object, $\dot{\vec{\omega}}_B$, is more easily computed using body coordinates:[5]

$$\dot{\vec{\omega}}_B = L^{-1}\left(\Sigma \vec{M}_B - \vec{f}_G \times L\vec{\omega}_B\right) \qquad (2)$$

Where L is the inertia tensor, $\vec{M}_B$ is the moment vector of the body, and $\vec{\omega}_B$ is the rigid body angular velocity vector. The moments are transformed from inertial to body coordinates using the following equation.

$$\vec{M}_B = R\vec{M}_G \qquad (3)$$

$$\begin{bmatrix} C_\theta C_\psi & C_\theta S_\psi & -S_\theta \\ S_\varphi S_\theta C_\psi - C_\varphi S_\psi & S_\varphi S_\theta S_\psi + C_\varphi C_\psi & S_\phi C_\theta \\ C_\varphi S_\theta C_\psi - S_\varphi S_\psi & C_\varphi S_\theta S_\psi + S_\varphi C_\psi & C_\phi C_\theta \end{bmatrix} \qquad (4)$$

In the previous equation, R represents the following transformation matrix [4] :

Where, $C_x=\cos(x)$ in generic terms, and $S_x=\sin(x)$. The angles ϕ, θ and ψ are Euler angles that represent the following sequence of rotations:

- rotation about the x-axis (rolling)
- rotation about the y-axis (pitching)
- rotation about the z-axis (yawing)

After the angular and the translational accelerations are computed from Equations (1) and (2), the rates are derived by numerical integration.

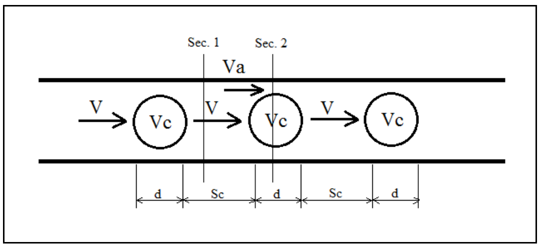

Figure 1. The capsule flow analysis

2) *The flow modeling*

In this research, ANSYS FLUENT® has been used to model the flow of a sphere in a horizontal hydraulic pipe, when the flow is turbulent. The pre-processing in CFD is broken into two main steps, i.e., creation of the geometry and the meshing of the flow domain.

A pipe of 0.1m diameter fully filled with water of a length of 2m is considered. The first half of the pipe is considered as the area of interest (considered as a test section), and the second half is representing the outlet pipe. The test section been used for the flow diagnostics of capsule transport is 1m long and it has the same properties as that studied by T. Asim [3].

The pipe in Figure. 1 is considered to be hydro-dynamically smooth. The capsule has been presented as a sphere of a diameter of half of the pipe diameter positioned near the pipe entrance. The geometries of both pipe and capsule have been created using the Design Modeler facility available in ANSYS 17.1. The area of interest (the test section) of the pipe has been meshed with tetrahedral elements. As it can be seen from the same figure, inflation has been adopted for the geometry mesh with a first layer thickness of 0.003938m of three layers around the capsule for better flow analysis. FLUENT® has been used for the grid generation with face sizing of 1mm around the capsule, and the resulted grid consists of around two million elements.

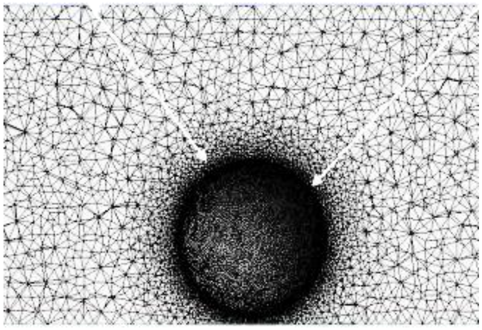

Figure 2. Grid system

For unstructured grids, the computation time associated with re-meshing may therefore become excessive. In addition, the re-meshing suffers from an accuracy loss in the physical conservation laws, where the local computational accuracy will possibly be reduced due to drastic grid size variation. However, for relatively high Reynolds numbers as in this study, the computation will become expensive because of the fineness of the grid that is required to resolve the flow behavior in the boundary layer region. This has been considered in the simulation by separating the layers around the capsule. These layers will not be deformed or re-meshed.

The motion of the boundaries can be rigid (i.e., the capsule moving in a pipeline filled with water), or deforming (i.e., the interior around the capsule). In either case, the nodes that define the cells in the domain must be updated as a function of time, and hence the dynamic mesh technique solutions are inherently unsteady. Whenever the capsule and its surrounding layer mesh are displaced, the mesh outside the boundary layer is smoothed and re-meshed.

The dynamic mesh is an embedding technology that has been used for simulations that require simplifying the grid generation for complex geometries [6]. The mesh deforms at each time step with respect to the specified motion rather than that the whole mesh system is regenerated. This approach allows preserving the grid topology for the capsule. The dynamic mesh technique, particularly for capsule flow simulation, has been proposed in the present study using an unstructured mesh that locally coupled with six-degrees-of-freedom (6DOF) solver. The built-in (6DOF) solver is available for applications with unconstrained motion, such as the capsule motion in this study.

The capsule and the pipe wall have been presented as walls with no slip conditions. The capsule interacts with the carrier fluid when the flow takes place. The wall of the capsule undergoes a rigid body motion and displaces according to the calculations performed by 6DOF solver [10]. The pipeline inlet has been treated as a velocity inlet. The pipe outlet has been treated as a pressure outlet and it has been kept at atmospheric pressure. The pipe has been considered as a hydro-dynamically smooth pipe having a wall roughness constant of zero. A zero-gauge pressure has been set for the pressure outlet boundary condition, and the intensity of turbulence is 5% at inflow boundaries for 0.1m hydraulic diameter of the pipe [11].

III. METHOD OF COMPUTATION

1) Basic strategy

The velocity of the flow within hydraulic pipelines is such that the compressibility effects can be neglected in such pipelines. Therefore, a pressure-based solver has been selected for solving the capsule flow [12]. There are many models available in the commercial CFD packages to model the turbulence, each one of these models has its own advantages and disadvantages.

The k-ε model been chosen belongs to the class of two equation models and it solves one transport equation for turbulent kinetic energy k and another for dissipation rate ε [7]. k-ε model is relatively simple to implement and leads to stable calculations that converge easily in many applications; however, it is valid only for fully turbulent flows. Realizable k-ε model shares the same turbulent kinetic energy equation as the normal k-ε model with an improved equation for ε. It has an improved performance for flows involving boundary layers under strong adverse pressure drops or separation, rotation, recirculation and strong streamline curvature [4].

The k–ε family is not valid in the near-wall region, whereas Spalart-Allmaras, for example, and k–ω models are valid all the way to the wall providing a sufficiently fine mesh. In this study, the Enhanced Wall Treatment option has been used to overcome this issue. Enhanced Wall Treatment options is suitable for low Reynolds flows with complex near-wall phenomena where turbulent models are modified for the inner layer (both buffer layer and viscous sublayer) [8]. However, it generally requires a fine near-wall mesh capable of resolving the viscous sublayer, which has been handled by adding inflation layers, and face sizing around the capsule.

2) Boundary conditions

The boundary types that have been specified in the present investigation are illustrated in Table 1. The capsule and the pipe wall have been modelled as walls with no slip conditions. The capsule interacts with the carrier fluid when the flow takes place. The wall of the capsule undergoes a rigid body motion and displaces according to the calculations performed by 6DOF solver.

TABLE 1. Named selections for the boundary conditions

Boundary Type	Boundary Name
Velocity Inlet	Inlet to the Pipe
Pressure Outlet	Outlet of the Pipe
Stationary Wall	Wall of the Pipe
Translating/rotating walls in the direction of the flow	Capsules

The pipeline inlet has been treated as a velocity inlet with a fully developed parabolic velocity profile. The pipe outlet has been treated as a pressure outlet and it has been kept at atmospheric pressure. The pipe and the capsule walls have been considered to be hydro-dynamically smooth, having a wall roughness constant of zero. A zero-gauge pressure has been set for the pressure outlet boundary condition, and the intensity of turbulence is 5 % at inflow boundaries for 0.1 m diameter of the pipe.

IV. EXPERIMENTAL SETUP AND MODEL VALIDATION

A flow loop has been designed and developed with the means of a 3 m long transparent test section of 2 inches inner diameter. The experiment setup has been developed to acquire data for validating the numerical model developed in this study. The pipes and the fittings used in building the flow loop are made of clear impact resistant polyvinyl chloride (PVC) with maximum operating pressure of 16 bar. The centrifugal pump has a maximum operating pressure of 16 bar at a maximum pumping fluid temperature of 120°C and at a maximum ambient temperature of 40 °C. The rated power of pump's motor is 37 kW at 2900 rpm, where the efficiency of the motor ranges from 92 % to 93.7 %, whereas the minimum efficiency index (MEI) of the pump is ≥ 0.4. The pump flow rate has been varied to get appropriate velocity inlet (V_{av}). This was achieved through the use of an 11 kW Siemens Optima Pump Test Rig which has a flexible and sophisticated control system, integrating a programmable logic controller and human interface to enable specific timing and control setting functions. The centrifugal pump is connected to the capsule injection system via a digital turbine flow meter.

Two long-range optical (OPB720A-06Z) sensors have been installed on the test section in the measuring section with a spacing of 1 m between them. According to T. Asim [9], the flow takes fifty times the pipe diameter of the pipe length to become fully developed. As the pipe diameter is 0.05 m, therefore, the flow is expected to be fully developed about 2.5 m away from the injection point. The signals from the sensors are transmitted to the computer through a printed board via an externally powered USB data acquisition system (NI-USB-6353, X Series DAQ). The data are evaluated with the aid of a program developed for this purpose. The output data have been then used to compute the capsule velocity along with the pressure drop. Signals come from both the pressure transducer and the optical sensors of the measuring section transmitted through an industrial automated printed circuit board created for this purpose. This board is interfaced to the computer using a system-design platform supported by LabVIEW visual programming language.

Once the capsule is fed into the capsule injection system, while the secondary loop is shut, the pneumatic knife gate valve is opened leading the capsule into the primary loop. The capsule is then collected on the top of the water tank from the capsule ejection mechanism, while the water is drained into the tank.

Both pressure drop and capsule velocity measurements are carried out on the 1m measuring section shown in Figure 3. Two pressure taps are created with 1 m spacing between them; these taps are then connected to a differential pressure transducer through piezometric hoses. The pressure transducer is able to measure pressure changes at a range of 0 to 6 bar with an accuracy of 0.5 %. Two fiber optic sensors are installed on the measuring section with a distance of 1 m between them. Signals are then transmitted from a printed circuit board to the computer in order to calculate the capsule velocity and the pressure drop.

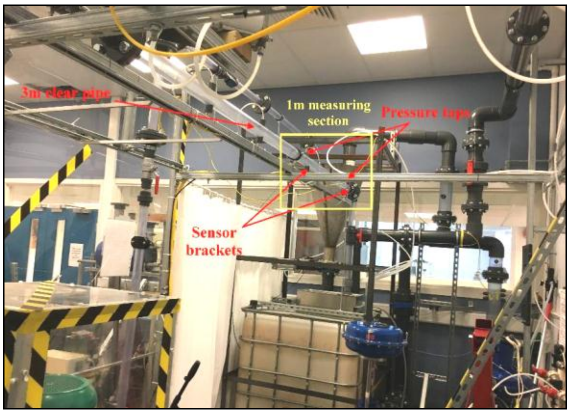

Figure 3. The test section with the measuring section

One of the most important steps while conducting numerical studies is the benchmarking of the results. This means that some of the results obtained from the numerical simulations are compared against experimental findings to confidently authorize that the numerical model represents the physical model of the real world. Hence, all the geometric, flow and solver-related parameters/variables become important in benchmarking studies. For the current study, the numerical model has been validated against the experimental findings for both the pressure drop and the capsule velocity in a pipeline carried out by the author. The numerical model has been set for the conditions listed in Table 2.

TABLE 2. Validation tests

Name/property	Value/Range/Comment	Units
Specific gravity	1	N/A
Capsule-to-pipe diameter ratio	0.5	N/A
Average flow velocity	2, 3 and 4	m/sec
N (Number of capsules)	1	N/A

The cases in Table 2 have been numerically solved. Figure 4 depicts the variations in the pressure drop within the pipeline, from both CFD and experiments, at various flow velocities. It can be seen that the CFD results are in close agreement with the experimental results, with an average variation of less than 10 %.

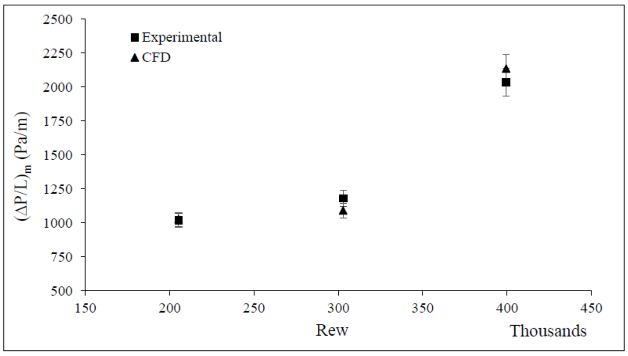

Figure 4. Validation of the CFD results against the experimental results for pressure drop

Figure 5 shows the variations in the capsule velocity within the pipeline, from both CFD and experiments, at various flow velocities. It can be seen that the CFD results are in close agreement with the experimental results, with an average variation of less than 5 %.

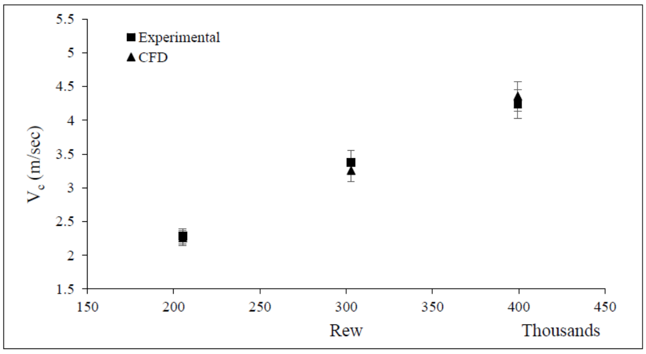

Figure 5. Validation of the CFD results against the experimental results for the capsule velocity

V. RESULTS AND DISCUSSION

Figure 6 shows time history of the named capsule translational velocity at inlet water velocity of 1 m/sec. The capsule attains a constant velocity in a relatively short time. The water velocity is higher than the capsule velocity just after the opening of the gate valve, but is soon outstripped by the capsule until it reaches the steady flying phase (after 2.5 sec). The capsule velocity then attains constant amount slightly greater than the operating velocity (5% to 10% of the operating velocity).

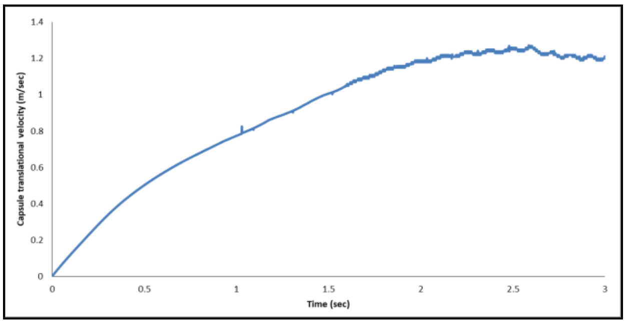

Figure 6. Time history of capsule at water inlet velocity of 1 m/sec for computation

Figure 7 shows trajectories of an equi-density capsule of k 0.5 at velocity inlet of 1 m/sec. It is found that for a given inlet velocity, the capsule trajectory is dependent on time regardless the shape or operating velocity.

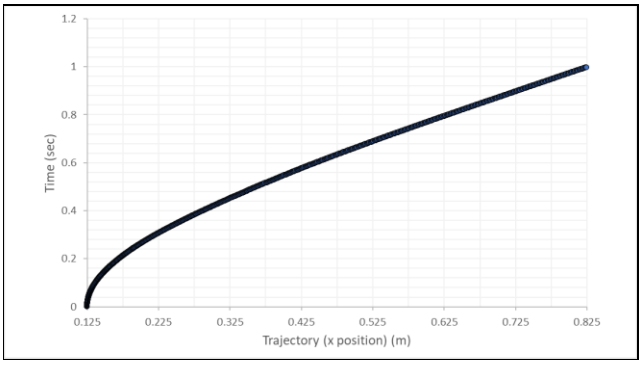

Figure 7. Trajectory of an equi-density capsule of k of 0.5 at 1m/sec velocity inlet

Figure 8 depicts V_c/V_w, against V_w where V_c and V_w are the capsule velocity and the inlet velocity respectively. The final computed capsule velocity is found to be proportional to the water inlet velocity at different operating conditions. The computation was done for three inlet velocities of 1, 2 and 3 m/sec. Similar results were obtained by Ellis et al [13] for capsules with density equal to that of the water.

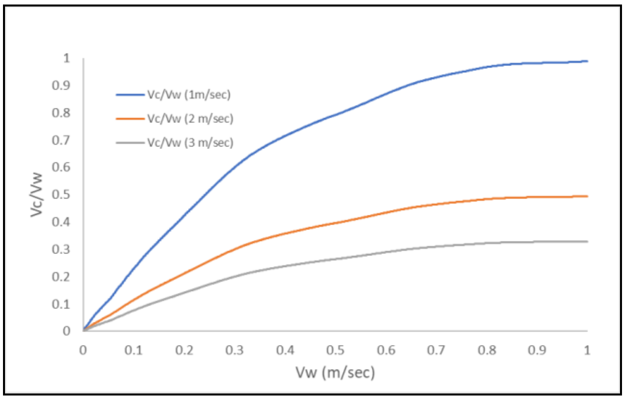

Figure 8. Velocity ratio against the inlet water velocity

Figure 9 shows the time history of named capsule acceleration at different inlet flow velocities. The total acceleration is mostly in x-direction only as the most capsule motion is considered in x-direction. The capsule tends to have a considerably high acceleration regardless the inlet velocity used in the beginning of the capsule motion. The capsule then attains a constant velocity in a short time (steady flying phase).

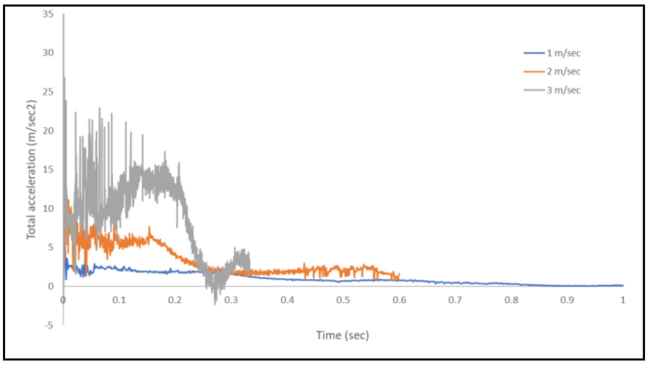

Figure 9. Time histories of acceleration due to pressure and shear forces

VI. CONCLUSION

This study provides a numerical model for the motion analysis of a single spherical capsule in a horizontal hydraulic pipeline. The analysis has been carried out with the aid of a well-validated computational fluid dynamics model for better understanding of the origins of the forces governing the entire unsteady flow. The model has been validated against the experimental data obtained from the test rig been developed for this study. The numerical study has been supplemented by expressions for the capsule to water velocity ratio. This study has employed a standard k-ε model in FLUENT to predict the flow characteristics when the flow is turbulent. The study has dealt with the effect of capsule trajectory on the flow velocity through the test section.

The capsule is found to attain higher velocity in shorter time at any flow inlet velocity in horizontal pipes. This indicates that the change in the capsule velocity is proportional to the average flow velocity of the carrier fluid (water).

REFERENCES

[1] pipeline to transport coal." J. Pipelines;(Netherlands) 1.4 (1981).
[2] 2. LUR'E, M. V. & GOL'DZBERG, V. L. 1971 Izv. Akad. Nauk SSSR, Energetika i Transport 4, 99 (in Russian).
[3] 3. Khalil, M.F., et al., Prediction of Lift and Drag Coefficients on Stationary Capsule in Pipeline. CFD Letters, 2009. 1(1): p. 15-28.
[4] 4. FLUENT, A., ANSYS FLUENT Theory Guide, Release 14.0, in ANSYS Inc., USA. 2011.
[5] 5. Guide,A.U.;Availablefrom:http://www1.ansys.com/customer/content/documentation/130/wb2_help.pdf.
[6] 6. Miedema, S.A., Slurry transport. 2016.
[7] 7. Grinis, L. and U. Tzadka. Vortex Flow Past a Sphere in a Constant-Diameter Pipe. in The Twenty-first International Offshore and Polar Engineering Conference. 2011. International Society of Offshore and Polar Engineers.
[8] 8. ANSYS, I., ANSYS Fluent Tutorial Guide. 2015.
[9] 9. Asim, T., Computational Fluid Dynamics Based Diagnostics and Optimal Design of Hydraulic Capsule Pipelines. 2013, The University of Huddersfield.
[10] 10. Taimoor Asim, Abdualmagid Algadi, Rakesh Mishra, Effect of capsule shape on hydrodynamic characteristics and optimal design of hydraulic capsule pipelines, Journal of Petroleum Science and Engineering, Volume 161, 2018, Pages 390-408,

[11] 11. Ahmed, N., Asim, T., Mishra, R., & Nsom, B. (2019). Flow visualization within a ventricular inhaler device using open source CFD for performance enhancement. International Journal of Condition Monitoring and Diagnostic Engineering Management (COMADEM), 22(2).

[12] 12. Yan, Y., Liu, Y., Jiang, H., Peng, Z., Crawford, A., Williamson, J., ... & Islam, S. (2019). Optimization and experimental verification of the vibro-impact capsule system in fluid pipeline. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 233(3), 880-894.

[13] 13. Journal of Fluid Mechanics , Volume 30 , Issue 3 , 29 November 1967 , pp. 513 - 531

BIOGRAPHIES

Sufyan Ali Mohamed Abushaala was born in Misurata /Libya, on June 8, 1982. He received Bsc degree in Mechanical Engineering from University of Misurata, in 2005. He got MSc degree in Mechanical Engineering from University of Huddersfield/UK in 2009. Moreover, he got PhD degree in Mechanical Engineering from the University of Huddersfield/UK in 2018, where he is currently lecturer in Department of Mechanical Engineering (ME) at Misurata University/Libya.