Simulation of a turbine flow meter with a cylindrical support fence using CFD

Sadegh Abbasnia [1,*] Alireza Abbasnia [2]

[1] Iranian Research Organization for Science and Technology (IROST), Department of Chemical Technologies, Azadegan Highway, P.O. Box 33535111, Tehran, Iran

[2] Department of Chemical Engineering, University of Tabriz, Tabriz, Iran

Corresponding author E-mail address: Sadeghabbasnia1981@gmail.com

Abstract

The effects of temperature and viscosity on flowmeters under various conditions were investigated in this study using Computational Fluid Dynamics (CFD) modeling on a turbine flowmeter with a cylindrical fence support. For a standard flowmeter geometry, a 3D computational model was created. As a suitable starting point for additional simulations, work started with a steady-state model. These data were entered into a transient model that featured a rotor swirling region. Considering the rotor as a rigid structure that can freely rotate around its axis of rotation, a fluid-structure interaction simulation was used to automatically adjust the rotor speed based on the torque applied to the rotor by the current. As a result, the simulation's ability to accurately represent both the transient response changes and the equilibrium rotor speed with respect to flow rates is improved. To produce multiples, a parametric analysis was carried out with these models.

Key words:

CFD, Flow meter, cylindrical support fence, viscosity, temperature

1. Introduction

A volumetric flow meter is a type of flow meter. Turbine meters are divided into three primary subcategories that comprise their basic design. These consist of the pulse pickup, internal assembly, and housing. The outer cover is called the housing. A rotor and bearing assembly on a shaft supported by a stator or fence make up the internal assembly. The liquid enters the flow meter chamber after traveling through the tube. As a result, the speed at which the rotor rotates is directly proportional to the amount of fluid that passes through it. Each blade is detected as it passes the sensor using a modulated radio frequency selective sensor or a magnetic carrier. The meter's output is generated at this frequency. After that, this output can be routed to additional electronics for processing. A turbine flow meter with a cylinder inlet fence is depicted in Figure 1.

Few models of turbine flowmeters are based on CFD simulations, even though many have been created over the years. At Tianjin University, a CFD model for a turbine flowmeter has been created [1]. The analysis of viscosity's impact on flowmeter performance was the aim of that work. ANSYS Fluent and the experimental data were used to create calibration curves for various viscosities. There are three areas in CFD simulation. For the purpose of the moving reference frame (MRF) simulation, two fixed regions with flow smoothers on either side of the rotor region and the peri-rotor region were set. After running the simulations, the torque was verified. The rotor speed was changed up or down to make up for any torque that the rotor may not have had. To achieve the zero torque condition, they employed a constant setting of ω. According to the findings of this study, variations in viscosity affect the flow velocity profile that enters the rotor and the wake flow that follows the upstream flow conditioner. Both the turbine flow meter's performance and the pressure distribution on the rotor blades are impacted by this. Tegtmeier et al. [2], performed a CFD analysis of viscosity effects on turbine flow meter performance. The geometry of a typical fuel transfer flowmeter was used to create a three-dimensional mesh. This entry belonged to a group of models. To provide a suitable starting condition for additional simulations, work started with a steady-state model. Ultimately a wide range of results and data were gained. Rochmanto et al. [3], performed the calibration of a turbine flowmeter intended for use with compressed natural gas (CNG). using a method that mimicked the kinematic viscosity property of compressed natural gas (CNG) in place of air. A regulating instrument in the calibration installation was adjusted to change the air's temperature and pressure in order to attain the same kinematic viscosity value as CNG during the calibration process. To compare with air, argon (Ar) was also employed, but with a change in working pressure to achieve comparable kinematic viscosity characteristics for both gases. The computation's output indicated that, in order to obtain comparable kinematic viscosity, the working pressure settings for argon and air were, respectively, 4 .5 bars and 3 .38 bars. Huang et al. [4], investigated using numerical simulations the effects of axial clearance variations on the performance characteristics of a dual-rotor flowmeter (DRT-FM), with calibration experiments serving as a means of validating the numerical results. As the clearance increases, the K factors of the various groups increase between

200 and 1600 L/h, according to the findings. Ryu Seo-Yoon and associates [5], created an experimental setup and established a numerical technique to simulate the impeller's rotational motion in response to the injection of water into the flowmeter, resulting in the development of a thorough performance evaluation technique [5]. A test setup was constructed to ensure measurement precision in response to variations in fluid pressure and temperature, as well as to confirm dependability in a range of flow circumstances, from low to high. Its validity was verified numerically using the six-degree of freedom method, as opposed to a moving reference frame, which numerically analyzes the turbine's rotational speed by backtracking it to match the flow conditions by providing it with a constant rotational speed. The impeller's acceleration motion could be directly numerically analyzed by reducing the cost of numerical analysis by mimicking water injection into the flowmeter, creating kinetic energy in the impeller based on the flow rate, and then rotating the impeller because of the generated torque. To confirm the applicability and dependability of the numerical technique, the results were compared with experimental data and evaluated in terms of angular velocity and pulse signal. The phenomenon observed during impeller rotation was explained by analyzing the impeller's aerodynamic performance. Orlova and associates. Examined a turbine flow meter about a system of hydrogen compressors in [6]. The fundamental ideas of linearity, accuracy, and turbine flow meters were discussed, along with calibration. The paper included a physical experimental testing scheme. One of the main elements influencing turbine flow meter performance is the fluid's viscosity. Experiments with numerical modeling were conducted for various fluids with varying viscosities. Moni and associates. [7] made use of the device's constant pressure loss, but He and co. used the convergent section of a V-Cone meter's permanent pressure loss to DP ratio [8]. Zhang along with others [9]. As well as Xu et al. [10] used a pair of distinct DP devices in series. Xue and associates. [11] Created theoretical models for the pressure gradient in the throat and convergent regions of an Extended-Throat Venturi Tube (ETV) and suggested several iterative and direct fitting algorithms to calculate the flow rate.

Some industrial instruments were simulated using CFD at [12–17]. The geometry of a typical flowmeter with a cylindrical inlet fence used in fuel transfer applications was used in this study to create a 3D network. This entry belonged to a group of models. For the purpose of providing a suitable starting condition for additional simulations, work started with a steady-state model. To shed light on the relationship between rotating and static structures, the model featured a rotating area surrounding the rotor. Better starting conditions were established by this simulation for use in later runs. These models were used in a parametric analysis to produce a number of calibration curves that illustrate the flow meter response in various scenarios. For every calibration curve, a number of simulations for a given fluid are run across a range of flow rates. A global calibration curve for the meter is created by compiling calibration curves for a range of viscosities. After extensive testing and calibration, this model generated calibration results that closely resembled the observed response. Full-flow field visualizations were generated by this simulation. Given the flow conditions and meter geometry, more precise velocity curves that more closely reflected the profile entering the rotor were generated. This can offer helpful information that could help with turbine flowmeter design and calibration.

2. Geometry and meshing

On a single-rotor flowmeter, the geometry is based. Using this flow meter in a five-inch pipe is intended to measure hydrocarbons. A flat, single-blade turbine makes up this device. Eight blades are present on the rotor of this flowmeter's standard model. Alternative iterations of this meter employ distinct rotors featuring anywhere from three to nine blades that vary in thickness and angle of attack. Bearings hold the rotor of this meter in place. The frequency of rotation of the rotor is measured using a radio frequency pickup. No additional drag is produced on the rotor by this kind of pickup. The rotor has two three-piece cylindrical flow straighteners at each of its two extremities. For simulation and computation, the volumetric flow range is set between 1 and 50 GPM. The software ANSYS WORKBENCH was used to generate the geometry. Figure 2 shows a simplified view of the geometry of the flow meter. A series of lines and planes are positioned at different points in order to get ready for a uniform post-simulation analysis. The velocity profile for each of the main solid structures can be seen from three different planes: the rotor, the outlet fence (also a stator), and the inlet fence. In order to collect velocity flow profile data, lines were also positioned at the same locations as the planes. With respect to the inlet and outlet fences, these lines are angled at a 45-degree angle. Lastly, a surface of revolution was employed to give a general perspective of the flow through the meter (refer to Fig. 3). The mesh is produced by ANSYS CFX Meshing in Fig. 4. Two primary sections make up the mesh. The area revolving around the turbine is one. The area surrounding the constant flow straighteners that are not rotating is the second region. The rotating area is surrounded by the stationary area. For every region, unstructured grids were employed. Using tetrahedrons, cutting cells, and assembly meshing techniques, the first meshes were produced in ANSYS Meshing. To provide basic settings and outcomes, these meshes were made quickly and simply. The hexahedral cells produced by the cut cell method are of excellent orthogonal quality (0.95 average), but the angular geometry is not sufficiently resolved. With ANSYS CFX Meshing, the final set of meshes was produced. These meshes are intended to capture the effects of the boundary layer on the rotor, with increased resolution in the target and swelling regions. With the help of this set, a mesh convergence study of the issue was conducted to identify the best mesh for minimizing execution time without sacrificing accuracy. The simulation employed multiple hydrocarbon fluids. Nine to twelve different flow rates were examined for every kind of fluid. For every data point, repetitions were made, and a calibration curve was created by averaging the results of these repetitions. Three sections make up the computational domain: the rotor region, the downstream straight pipe region, and the downstream flow conditioner; the upstream straight pipe region and the upstream flow conditioner. Moving reference frame (MRF) motion was selected for the rotor region, while fixed motion was selected for the other two regions. Two matching sliding interfaces connect the fixed areas and the rotating area. Using standard wall functions, the k-epsilon solver was the turbulence model utilized. Every case starts with the same parameters and boundary conditions. The middle of the flowmeter's operating range was used to match the inlet velocity to the flow rate. The only thing mentioned is output. To correspond with experimental findings, the output can also be modified for a given pressure drop. The rotational speed determined by the experiment is first approximated by adjusting the rotor speed. By varying the rotor speed, the torque on the rotor is stored as an output to guarantee that the system reaches zero torque equilibrium. To reduce the moment, the rotor speed was manually changed. In order to better simulate the interaction of the moving rotor with steady-flow smoothing fences and to model the effects of flow transients, transient models were developed and their initialization was based on the results of the steady-state models. A sliding reference frame is used to set this.

3. Results

3.1. Grid Unification

While utilizing a manageable amount of processing power, the mesh had to generate minimal simulation error. Between 500000 and over 30 million cells were employed in a range of mesh configurations. With a viscosity of 30 cSt, these meshes were run at the flow meter's midpoint (20

GPM). The calibration information for this flow meter was compared with the outcomes of these runs. Since the error and cell count were comparatively low, the grid with roughly 10 million cells was chosen. The following sections made extensive use of this grid. The case of the straight inlet fence employed a grid with comparable properties. Fig. 5 presents the outcomes.

3.2. Steady State Moving Reference Frame (MRF)

The tetrahedron assembly mesh was used in a steady-state ANSYS CFX simulation. Visual examination reveals that the flow operates as anticipated. The velocity in the stationary frame is displayed in Figure 6. It seems reasonable to assume that this flow field is correct. Wakes are visible developing behind the immobile structures. The inlet fence increases the flow's angle of attack by turning the flow into the rotor. This could increase the flow meter's operational range, particularly in the low-flow area. To find the rotor speed vs., the rotor speed was manually adjusted over a brief range. Scene curve depicted in Figure 7. The model was run with the rotor speed adjusted until a convergent solution was reached. This was carried out for a range of rotor speeds that were close to the predicted value. This illustrates the point at which the moment equals zero at 382 rad/s, or 63 Hz, rotor speed. The rotor speed needed to generate zero torque is quite similar to the 46 Hz experimentally determined value.

3.3. Transient Moving Reference Frame (MRF)

A transient solver was used to repeat this procedure. With the chosen flow conditions, the outcomes closely matched those of the steady-state solver. But by introducing a temporal component, it was possible to see the fleeting wakes from the static structures. The moment that resulted across the rotor was more variable than in the steady-state scenario because of the wakes' transient effects. To gain a better understanding of the transient interaction of the rotor on the static flow fences, a moving mesh was evidently required. Fig. 8 displays the results of the transient simulation. A restricted 6DoF UDF was employed in an attempt to capture both the transient interaction of the structures and to automatically bring the rotor to a steady state speed. The rotor speed in a steady state could be automatically solved for in this simulation. Additionally, the fleeting interactions between the structures were visible. Each side of the blades' pressure contour is measured as the solver gets closer to the steady state rotor speed. Higher pressure areas might be observed on the upstream or downstream side of the blade if the rotor were spinning too quickly or slowly. Both sides of the blades displayed pressure contours that were strikingly similar at a constant speed. Early in the simulation and at the end, the pressure on the rotor is shown in Figures 9 and 10. It is evident that at the beginning of the simulation, the rotor's torque is uneven, but by the end, it equalizes. There were a few noticeable spikes in the rotor moment when the flow rate was altered. This observation makes sense because the solution is modified by the solver to meet the new conditions. The rotor speed would then gradually change over several time steps. While the moment would not cross zero, it would approach it. The 6DoF solver's method of determining the rotor speed for the subsequent time step causes the moment to converge to a tiny positive or negative value. It was necessary to take duplicates at each set point because of the variation in the results, which is partially explained by this. Through determining the equilibrium flow rate across various flow rates, a calibration curve was created.

3.4. Hydrocarbon Flow Profiles

Furthermore illuminating the internal workings of the turbine flow meter are the velocity profiles, which are displayed in Fig. 11-16. The rotor's angle of attack can be increased by observing how the inlet fence's bends cause the flow to change. As a result, there is a greater reaction, particularly at the flow meter's low calibration curve. Within this laminar region, the stationary structure and rotor interaction is also observable. Through the inlet flow straightener's wake, the blades travel as the rotor rotates. Because of the uneven flow, this causes somewhat different torque values on the rotor.

3.5. Hydrocarbon Velocity Profiles

A major influence on the flow profile comes from the static structure ahead of the rotor. The calibration curve is significantly influenced by the flow profile that the rotor meets. The flow velocity profile, which in turn influences the pressure distribution on the rotor blades, is influenced by changes in fluid viscosity. The result is a change in the rotor's rotating velocity. Input data for various computational models, such as the Winchester model, can be obtained from the velocity profiles of each case. A uniform or fully developed annular flow is not what the velocity profiles in Figs. 17–19 represent. The wake flow originating from the inlet fence partially develops and impacts the flow. The corresponding radial location's rotational velocity will not coincide with the radial flow velocity, as this flow profile further demonstrates. That is to say, pressure drive occurs in the blade's center zone, while pressure drag occurs in the vicinity of the hub and tip.

3.6. Straight Fence Calibration Curve

Since the inlet fence's bend was eliminated in the straight fence simulation, the rotor is not being approached by a favorable incident flow. In order to enhance the flow meter's response in the low flow regime, this bend is added during the initial calibration process. Depending on that initial calibration, the bend may differ from one flow meter to the next. This example was included to highlight the model's capacity to depict the consequences of modifications to the flow meter's geometry. For this simulation, a single viscosity was employed. As shown in Fig. 20, the calibration curve for this instance is comparable to those of earlier cases, with the exception of a notable drop-off in response at the lowest end of the flow regime. This supports the logic behind bending the inlet fence to enhance the meter's functionality. The calibration curve's remaining portion remains lower than the hydrocarbon simulation from earlier. The reason for this is that there isn't enough swirl to raise the rotor's K factor.

3.7. Straight Fence Flow Profile

The flow passes through the inlet fence and reaches the rotor at a reduced angle of attack, according to the straight fence's velocity profiles. Part of the rotor speed reduction can be attributed to this decreased angle of attack. The leading edge of the blades has a zone where the flow velocity is higher. Because there is less to disturb the flow, the wakes from the inlet fence are smaller in this type of flow meter. Low-velocity fluid created a sizable wake due to the angled inlet fence. This suggests that, in comparison to an angled inlet fence, there will likely be less interaction between the rotor and the static structure in the case of a straight fence. Because of the uniformity of the flow, the torque on the rotor is more consistent throughout the turbine's full rotation (see Figs. 21 and 22).

4. Conclusion

The primary objective of this study is to improve the accuracy of computational modeling of a turbine flow meter using environmentally safe fluids and cylindrical support inlet and outlet fences. This modeling will allow for the simulation of the rotor response and flow across a wide range of the meters' operation. In every application, accurate measurements depend on flow meter calibration. The standard Stoddard solvent has been used for many hydrocarbon flow meter calibrations; however, this volatile calibration fluid puts the workers' health and the environment at risk. Although they can be used as acceptable substitutes, other safer calibration fluids, like a solution of propylene glycol and water, have the same kinematic viscosity as volatile fluids. However, there are worries about how using these substitute calibration fluids may affect accuracy. Despite extensive research, the fluid dynamics mechanism underlying the viscosity effect on turbine flow meter performance remains incompletely understood. The meter response and the geometry of the turbine meter calibration curve cannot be sufficiently explained by any precise physical model that has been verified by experimentation. Furthermore, the way that temperature-related variations in kinematic viscosity impact turbine meter calibrations is not well understood. There have previously been published physical models for the turbine meter calibration curve based on momentum and airfoil approaches; however, to increase accuracy, these models are augmented with experimental correction factors. This study examined the effects of viscosity on turbine flow meters across a wide range of operations using computational fluid dynamics (CFD) modeling in an effort to better understand and utilize the meters. A customary flow meter geometry was modeled computationally in three dimensions. With the help of the CFD model's time-varying mesh, the flow field can be used to observe how the rotation and static structures interact. Through manipulation of the fluid's density, viscosity, and flow rate, this model was utilized to assess the calibration of a turbine flow meter and determine the response, or steady state rotor speed. To give an appropriate starting condition for additional simulations, the work started with a steady-state model. An interim model was fitted with these findings. To offer insight into the interplay between the static and rotational structures, the transient model featured a rotating zone surrounding the rotor. New starting conditions for the subsequent simulation were generated by this one. Utilizing a fluid-structure interaction simulation, the rotor was treated as a rigid structure with free rotation around its own axis of rotation in order to automatically adjust the rotor speed based on the torque imparted on it by the flow. This indicates that when flow rates change, the transient response in simulation is closer to the actual transient response, and the equilibrium rotor speed is more accurately matched.

References

[1]- Guo, Suna, Lijun Sun, Tao Zhang, Wenliang Yang, and Zhen Yang. "Analysis of viscosity effect on turbine flowmeter performance based on experiments and CFD simulations." *Flow Measurement and Instrumentation* 34 (2013): 42-52.

[2]- Tegtmeier, Carl. "CFD Analysis of Viscosity Effects on Turbine Flow Meter Performance and Calibration." (2015).

[3]- Rochmanto, Budi, Hari Setiapraja, Ihwan Haryono, and Siti Yubaidah. "A study of kinematic viscosity approach with air as a gas medium for turbine flowmeter calibration." *Flow Measurement and Instrumentation* 95 (2024): 102490.

[4]- Huang, Fuji, Liang Yan, Chabi Christian Monsia, Yuxiang Han, Jiabao Liu, and Hao Zan. "Effect of Axial Clearance Variation on Dual-Rotor Flowmeter Performance." *Sensors* 24, no. 13 (2024): 4389.

[5]- Ryu, Seo-Yoon, Dong Gyu Yun, Hae Chan Kim, Cheolung Cheong, and Su Il Park. "Development of numerical and experimental performance evaluation techniques for high-reliability turbine flow meters of ice-making water-dispenser." *AIP Advances* 14, no. 7 (2024).

[6]- Orlova, S., T. N. Devdas, V. P. K. Vasudev, and S. Upnere. "Numerical Modelling of a Turbine Flow Meter Used as Part of the Hydrogen Compressor System." *Latvian Journal of Physics and Technical Sciences* 60, no. 6 (2023): 113-126.

[7]- Monni, Grazia, Mario De Salve, and Bruno Panella. "Two-phase flow measurements at high void fraction by a Venturi meter." *Progress in nuclear energy* 77 (2014): 167-175.

[8]- He, Denghui, Bofeng Bai, Jun Zhang, and Xianwen Wang. "Online measurement of gas and liquid flow rate in wet gas through one V-Cone throttle device." *Experimental Thermal and Fluid Science* 75 (2016): 129-136.

[9]- Zhang, Fusheng, Feng Dong, and Chao Tan. "High GVF and low pressure gas–liquid two-phase flow measurement based on dual-cone flowmeter." *Flow Measurement and Instrumentation* 21, no. 3 (2010): 410-417.

[10]- Xu, Ying, Chao Yuan, Zhenghai Long, Qiang Zhang, Zhenlin Li, and Tao Zhang. "Research the wet gas flow measurement based on dual-throttle device." *Flow Measurement and Instrumentation* 34 (2013): 68-75.

[11]- Zheng, Xuebo, Denghui He, Zhigang Yu, and Bofeng Bai. "Error analysis of gas and liquid flow rates metering method based on differential pressure in wet gas." *Experimental Thermal and Fluid Science* 79 (2016): 245-253.

[12]- Abbasnia, Sadegh, Zarrin Nasri, and Mohammad Najafi. "Comparison of the mass transfer and efficiency of Nye tray and sieve tray by computational fluid dynamics." *Separation and Purification Technology* 215 (2019): 276-286.

[13]- Abbasnia, Sadegh, Vali Shafieyoun, Mohammad Golzarijalal, and Zarrin Nasri. "Computational fluid dynamics versus experiment: an investigation on liquid weeping of Nye Trays." *Chemical Engineering & Technology* 44, no. 1 (2021): 6-14

[14]- Abbasnia, Sadegh, Zarrin Nasri, Vali Shafieyoun, and Mohammad Golzarijalal. "Nye tray vs sieve tray: A comparison based on computational fluid dynamics and tray efficiency." *The Canadian Journal of Chemical Engineering* 99 (2021): S681-S692.

[15]- Abbasnia, Sadegh, Qiming Jimmy Yu, Zarrin Nasri, and Ali Khojasteh. "Nye Tray Versus Sieve Tray: Experimental Study on Hydrodynamic Parameters." *Chemical Engineering & Technology* 45, no. 1 (2022): 110-118.

[16]- ABBASNIA, SADEGH, Vali Shafieyoun, and Zarrin Nasri. "CFD versus experiment: an investigation on liquid weeping of Nye tray." *Authorea Preprints* (2020).

[17]- Abbasnia, Sadegh, and Alireza Alireza Abbasnia. "Computational fluid dynamics comparison of two-equation turbulence models by studying hydrodynamic parameters in the distillation trays."