

The Accuracy Degree of CFD Turbulence Models for Butterfly Valve Flow Coefficient Prediction

Mohammed M. Said^{1,*}, Hossam S.S. AbdelMeguid², Lotfy H. Rabie²

¹GUPCO - Gulf of Suez Petroleum Co., 270 Palestine St. 4th Sector, New Maadi, Cairo, Egypt

²Mechanical Power Engineering Department, Faculty of Engineering, Mansoura University, El-Mansoura 35516, Egypt

*Corresponding author: saidmm@gupco.net

Abstract Although engineers are mainly interested in the prediction of mean flow behavior, the turbulence cannot be ignored, because the fluctuations give rise to the extra Reynolds stresses on the mean flow. These extra stresses must be modeled in commercial CFD by selecting convenient turbulence model. The flow inside the control valve is complex and the control valves performance is precisely evaluated by determining the valve coefficient named, flow coefficient. Hence, aim of the present study is to investigate the effect of turbulence model type on the solution accuracy for the valve disk angles 40° and 60° as well as to implement the degree of agreement between experimental and numerical results. The numerical verification has been investigated by FLUENT 6.3 and the valve is meshed by GAMBIT 2. The mesh independent test has been carried out only by standard k- ϵ to evaluate the mesh effectiveness and attain the best accuracy. Among from these several turbulence models which have been studied here are standard k- ϵ , realized k- ϵ , k- ω , and RSM. Butterfly valve, STC model and (DN 50) diameter is chosen to be the test specimen in this research. The results showed that, there is no general turbulent model that can deal successfully with all cases. Numerical and experimental results are in general in good agreement, however are different in details, and showed that, RSM model is the most efficient numerical solver when applied to butterfly valve flow coefficient evaluation. For the future, a significant amount of work still needs to be undertaken in experimental unsteady butterfly valve flow analysis with RSM numerical model.

Keywords: CFD, butterfly valve, valve coefficients, turbulence, cavitation

Cite This Article: Mohammed M. Said, Hossam S.S. AbdelMeguid, and Lotfy H. Rabie, "The Accuracy Degree of CFD Turbulence Models for Butterfly Valve Flow Coefficient Prediction." *American Journal of Industrial Engineering*, vol. 4, no. 1 (2016): 14-20. doi: 10.12691/ajie-4-1-3.

1. Introduction

Butterfly control valves are sized according to the valve coefficients at different disk angles (α). Therefore, it is very important to know in which conditions the butterfly valves exhibit high performance relative to their angles. Misconception in sizing butterfly valves can destroy the flow continuity and change the physical performance. In many cases, it results in undesirable effects such as intensive noise and vibration, which can limit the life expectancy of the valve. Since it is difficult to understand experimentally the flow characteristics in detail because the flow around the valve has a complex 3D structure, CFD provides local information by taken into account all the variables as, pressure and velocity around the control valve disk. CFD analysis can reveal the complex flow structure around the valve, which the experiments hardly provide. Even otherwise, experimentation needs to be supplemented with CFD analysis because of the complex geometry as well as complexities like turbulence during flow through a valve. It is emphasized in the this study that, eddy viscosity closures, such as the k- ϵ , k- ω , and RSM models, have been commonly adopted for computing turbulent flows in practical applications since

they are relatively robust models. The experiments are carried out in the laboratory of the engineering faculty of 'Mansoura University'. An experimental set-up is built likewise ANSI/ISA-S75.02-1996 pertaining to control valve capacity test procedure. The numerical solution has been carried out by FLUENT 6.3 at valve disk angles 40° and 60° and GAMBIT 2.4 is used as a meshing tool. [1] studied the performance of butterfly valve of different disks and the flow characteristics using CFD. The results showed that the flow pattern associated with a double disk is more complex compared to a single disk type due to formation of recirculating eddies at the rear of the valve disk. Moreover, the results illustrated that the disks hydrodynamic behavior and dynamic torque coefficient are affected by the shape of the disk geometry. [2] studied numerically 1000 mm diameter butterfly valve using COSMOS FLOWWORKS software. The results revealed that the valve disk surface roughness has an insignificant effect on the disk opening torque. [3] utilized FLUENT 6.0 to predict the pressure profile on the butterfly valve disk at angles 30°, 45°, and 60°. The numerical results depicted that for certain disk angles, significant fluctuations in the torque are presented and cause severe vibrations to the piping system. [4] studied the fluid flow properties in a large butterfly valve using fluid structure interaction (FSI) to determine whether it can work safely

or not. The results of FSI suggested that large butterfly valve should not be fixed at a low opening angle, and also the improvement of butterfly valve design is conducted in this study. [5] studied the design optimization using CFD for butterfly disk. The result clarified that the flow coefficient increases by 56.8 % after redesigning the stem by the optimized design. The valve manufactures present their products with the valve coefficients which are the major target in the case of good sizing and selection process. [6] investigated the numerical calculations results for flow through a ball valve, which are based on the concept of [7] experimental data. In their study comparison of flow pattern at several opening angles was investigated. Furthermore, the pressure drops behind of the ball valve and formations of vortex flow after the valve section have been analyzed. [8] performed numerical analyses by applying CFD code, FLUENT, to obtain the solution of the turbulent flow field through a globe valve in different openings. The flow control valves in high velocity oxygen systems for different openings are simulated for turbulence and eddy dissipation. The influence of pressure, flow rate and opening of the valve on the rise in temperature and eddy dissipation rate is also obtained for compressible flow range. The simulation for turbulence is done by k- ϵ and k- ω models and the results have been compared. [9] reviewed the stationary and non-stationary characteristics of attached turbulent cavitating flows around solid objects. [10] investigated a comparison study of 48 inch butterfly valve's experimental performance coefficients using CFD in an incompressible fluid at Reynolds numbers ranging approximately between 10^5 and 10^6 . It was found that for mid-open disk angles ($\alpha = 30^\circ - 60^\circ$), CFD was able to appropriately predict common performance coefficients for butterfly valves. For lower valve angles ($\alpha = 10^\circ - 20^\circ$), CFD simulations failed to predict those same values, while higher valve angles ($\alpha = 70^\circ - 90^\circ$) gave mixed results. [11] studied the turbulent flow of water through a butterfly valve of 20 cm diameter. The results showed that the flow is smoother and free of turbulence at small pressure drop across the valve either at large valve opening angle or small inlet velocity. [12] implemented a numerical simulation for flow of water past over a butterfly valve using commercial fluid dynamics software FLUENT. It was found that the flow has a small effect with increasing closing angle till it reaches 55° , where the flow around the valve started to become highly turbulent. [13] applied a CFD analysis on a double eccentric butterfly valve disk to demonstrate the validity of a topology optimization approach. They determined the shape of the disk using topology optimization and also they compared the initial design with the optimal design. It is found that the pressure drop decreases by 8% and torque reduces by 5% in the modified design compared to the initial design. Also, the disk volume is reduced by 10% by using a modified design. Moreover, it was found that the disk needs fast and accurate shape design process for efficient design satisfied in valve performance. [14] adopted SST turbulence model to compare the butterfly valve torque values with the experimental results. The results showed that the inlet Reynolds number has quantitative influence on the flow field ; with the rise of the Reynolds number, the torque value increases and the size of separation regions in the downstream reduces. [15] presented a diagram of butterfly valve with a diameter of

100 mm and was operating in cavitation. Seven evolution stages of cavitation and choking flow rates are determined. The diagram can be utilized to determine the operating conditions in different stages of cavitation evolution.

2. Theoretical Model

With the modern evolution of computational methods it has become increasingly viable to consider more direct numerical methods for the solution of control valve fields. The numerical model is restricted in the study to grid generation via GAMBIT 2.4 and mesh dependence test. FLUENT 6.3 is adopted as a solver and pre-processing tool. For all turbulence equations mentioned in this study we solve for the unknowns of interest, namely the butterfly flow coefficient.

2.1. Mathematical Model

The flow through the butterfly valve is wholly turbulent as the mean Reynolds numbers, R_D range from 5,000 to 15,000 for the present experiments as well as the flowing liquid is water. Thus the analysis is confined so that the basic equations to be solved are the steady state continuity and the Navier-Stokes equations for incompressible and Newtonian water liquid.

2.1.1. Governing Equations

The governing equations of fluid dynamics commonly referenced for incompressible isothermal fluids in CFD include Navier-Stokes equations [10]. The solver employs a pressure based algorithm and a finite volume approach to solve the fluid flow equations for the near field close to the butterfly disk and the far field. Applying Newton's second law to a volume of fluid gives the momentum Eq. (1) [10].

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j} (u_j u_i) = -\frac{1}{\rho} \frac{\partial}{\partial x_i} p + \nu \frac{\partial^2 u_i}{\partial x_j^2}. \quad (1)$$

2.1.2. Steady State Turbulence Model

Turbulent flows exist in nature as well as in industrial environments, where they represent the vast majority of flows in practical applications. The prediction of their properties is therefore crucial to many scientific and engineering activities, [16]. One way of accounting for the randomness introduced by turbulence is to consider of the flow as being composed of a mean value and fluctuating component. This is known as Reynolds decomposition. One approach used to solve Navier-Stokes equations includes focusing on the effects of turbulence on mean flow properties by using what is called Reynolds-Averaged Navier-Stokes (RANS). RANS equations are expressed, in index notation form, Eq. (2) [10].

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial}{\partial x_j} \left(-\bar{p} \delta_{ij} + \mu \frac{\partial \bar{u}_i}{\partial x_j} + \tau_{ij} \right) \quad (2)$$

where, \bar{u}_i , is the mean average velocity vector. RANS equations are unsolvable without defining a turbulence closure model. Basically in the present study, for the turbulence closure, standard k- ϵ , realized k- ϵ , k- ω and

RSM are used as closure. [17] utilized FLUENT to simulate turbulent flows in a butterfly valve, in which the k-ε model was employed for turbulence consideration.

2.1.3. Grid Generation

Both the accuracy of a solution and its cost in terms of necessary computer hardware and calculation time are dependent on the fineness of the grid, [18]. The computational domain is meshed via GAMBIT 2.4. GAMBIT, which is used as a basis for simulations run in FLUENT standard k-ε model. The generated mesh has been repeated for different mesh types and sizes, the best efficient mesh method for converging solution is executed

using unstructured (tetrahedral) and T-grid type. The studied flow volume includes the valve disk and outside this disk the grid is slightly expanded toward the ISA domain of the pipe attached to the valve disk. Mesh density at curved surface of the disk is finer than other surfaces, as well as fairly coarse in the far field, i.e., mesh of higher density is generated in the area around the valve disk. The final meshes are generated for disk angles 40°, 60° and 70°. An illustration of the mesh for disk angle 40° is shown in Figure 1. Locally re-fined numerical grids of high densities, for all disk angles, range around 1.7x10⁶ elements.

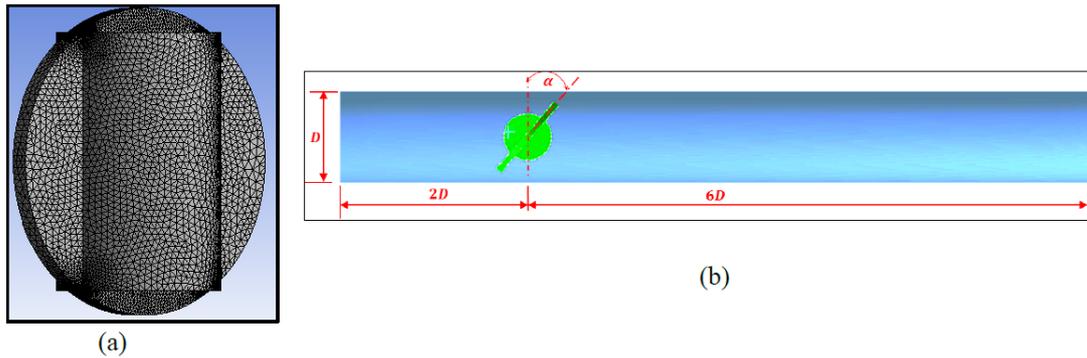


Figure 1. (a) Butterfly valve disk tetrahedron mesh, (b) the flow domain extended 2D and 6D upstream and downstream the valve respectively for disk angle ($\alpha=70^\circ$)

2.1.4. Mesh Dependence Test

Mesh dependence test is performed for 3D butterfly valve at 60° disk angle. The converging criterion is established when the numerical solution obtained for the inlet pressure on different grids agrees to within a level tolerance of 0.001. Four trails range from a coarse to fine mesh are performed, where the number of mesh elements, N is increased gradually with avoiding skewness and aspect ratio violation till defining the number of elements where the solution is independent of the mesh density. As illustrated Table 1, and after performing four trails, as the

grid resolution is refined, and the number of grid points is increased; the error in the numerical solution decreases and the result obtained for cell resolution around 1.7x10⁶ is adopted in the present study.

Table 1. Mesh dependence test for disk angle 60°

No	Mesh dependence test(60°)			P ₁ kPa	Error%	Time Hour
	No. of cells	No. of faces	No. of nodes			
1	275,034	581,397	62,066	5.6122	-	4
2	533,055	1,111,892	112,717	5.1917	8	6
3	1,188,539	2,476,541	249,793	5.0814	2	8
4	1,774,814	3,703,369	375,994	5.0763	0.1	10

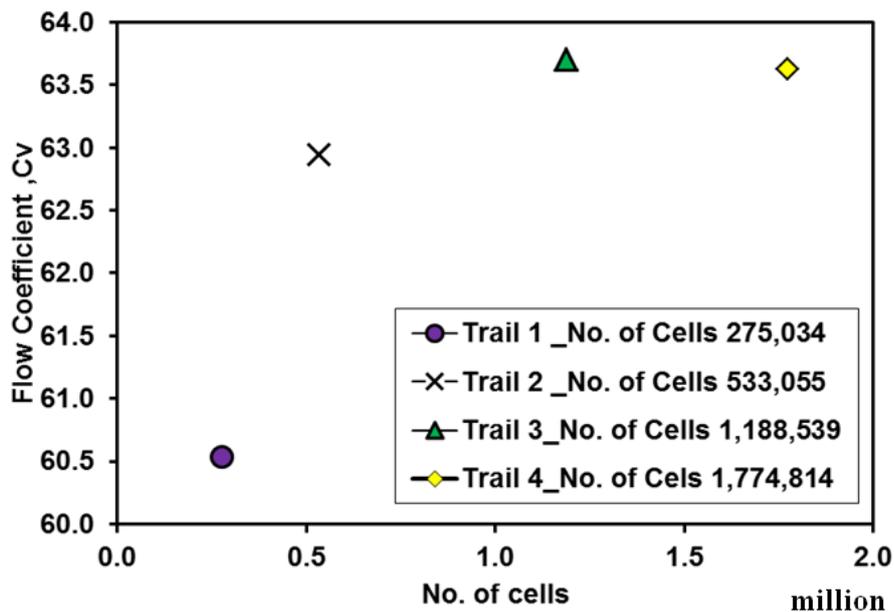


Figure 2. Variation of, Cv, with mesh density for disk angle 60°

The flow coefficient, C_V , is calculated from the numerical results of the different mesh resolutions and the error percentage between successive trials is depicted in Table 2. The result as shown in Figure 2 implies that, the values of, C_V , for trial 3 and 4 are indistinguishable.

Table 2. FLUENT 6.3 fixed entries and boundary conditions data.

Variables	Value
Inlet pressure, P_1	68.94 kPa
Outlet pressure, P_2	0 kPa
Turbulence intensity, I	4 %
Density, ρ	998.2 kg/m ³
Kinematic Viscosity, ν	1.13x 10 ⁻⁶ m ² /s
Hydraulic diameter, D	0.049 m

2.2. Numerical Model Boundary Conditions

A large source of uncertainty in CFD modeling can result from poor representation of boundary conditions, [3]. Also some flow problems can be very sensitive to apparently minor changes in the boundary conditions or

problem geometry. In the numerical model, water and flow properties as density, viscosity, inlet temperature, and inlet and outlet pressures are held unchanged. Moreover, the turbulent intensity, I , is 4%. The defined boundary conditions for the present study are varnished in Table 2. At solid boundaries, the no-slip condition is applied for 40°, 60°, and 70° disk angles.

2.3. Experimental Set Up

2.3.1. Test Rig Description

Experimental test rig is installed in the Hydraulic Lab of Faculty of Mechanical Engineering, Mansoura University. The control valve test rig and the flow coefficient estimation procedure are covered by ANSI/ISA-S75.02-1999, [19]. The test rig is schematically shown in Figure 3. The tested butterfly valve has been addressed by no. 4 as shown in Figure 3 is a wafer style, class #150, DN50 (0.049 m) diameter and has the same diameter as the upstream and downstream connected pipes. The manufacture is STONETOWN Company and the model is STC.

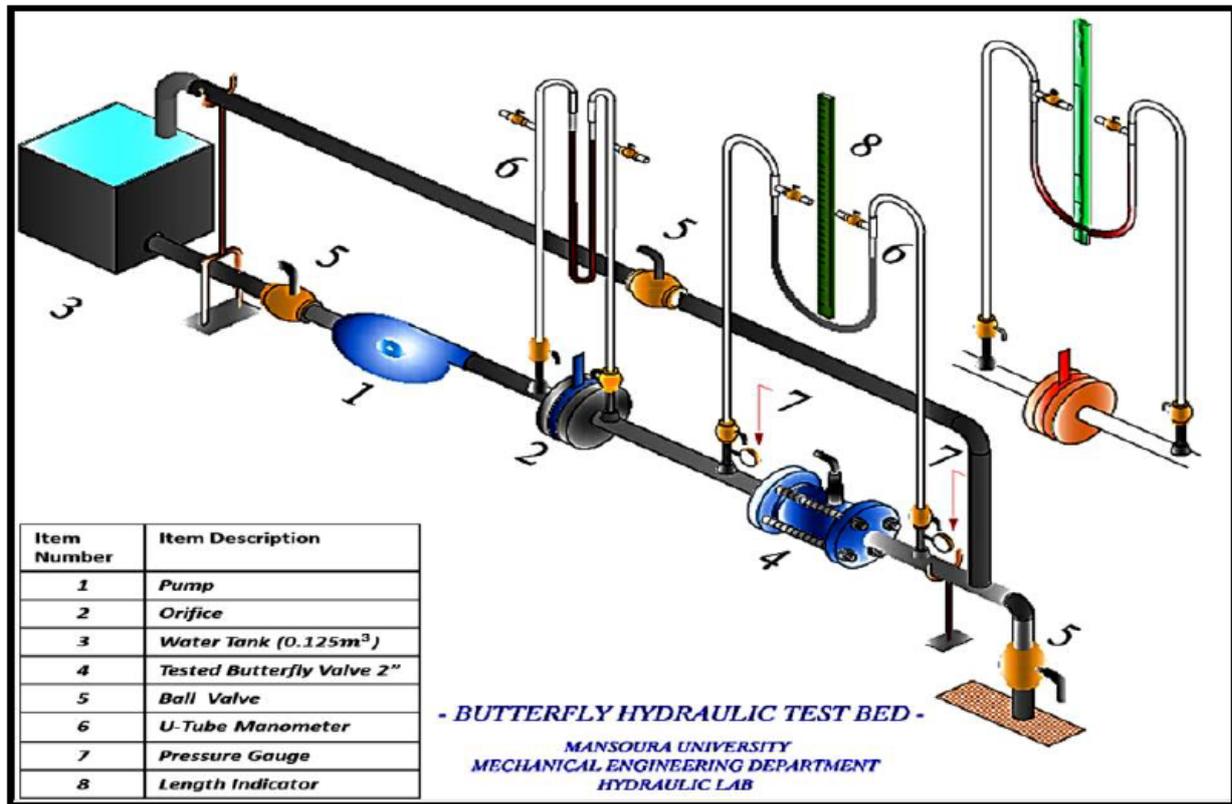


Figure 3. Butterfly valve test rig erected according to ISA standard S75.02-1999

3. Results and Discussions

This section is devoted to determine the confidence level of accuracy associated with the type of the turbulence model to use in the field of butterfly valve application and generally in hydraulic systems. Disk angles 40°, 60°, and 70° have been analyzed in order to account for numerical quantitative results emerging from four turbulence models named Standard and Realizable k-ε, k-ω, and RSM.

3.1. Disk Angle 40°

For disk angle 40°, numerical turbulence models results are plotted in Figure 4 and relative errors are tabulated in Table 3 for Standard k-ε (S.k-ε), Realizable k-ε ((R.k-ε), k-ω, and RSM. It is apparent from Table 3 that all results of turbulence models are over predicted relative to the experimental, and all models results have different degree of agreement with the experimental. As can be seen from the Figure 4, the closest numerical model accuracy to the experimental is RSM. The S.k-ε introduces better results than k-ω.

Table 3. Errors for different single phase turbulence models for disk angle 40°

Model	RSM	Standard k-ε	k-ω	Realized k-ε	Experimental
Numerical, C _v	30	32	34	38	24
Error%	25	33	41	58	-
Run time(hours)	14	7	8	8	-

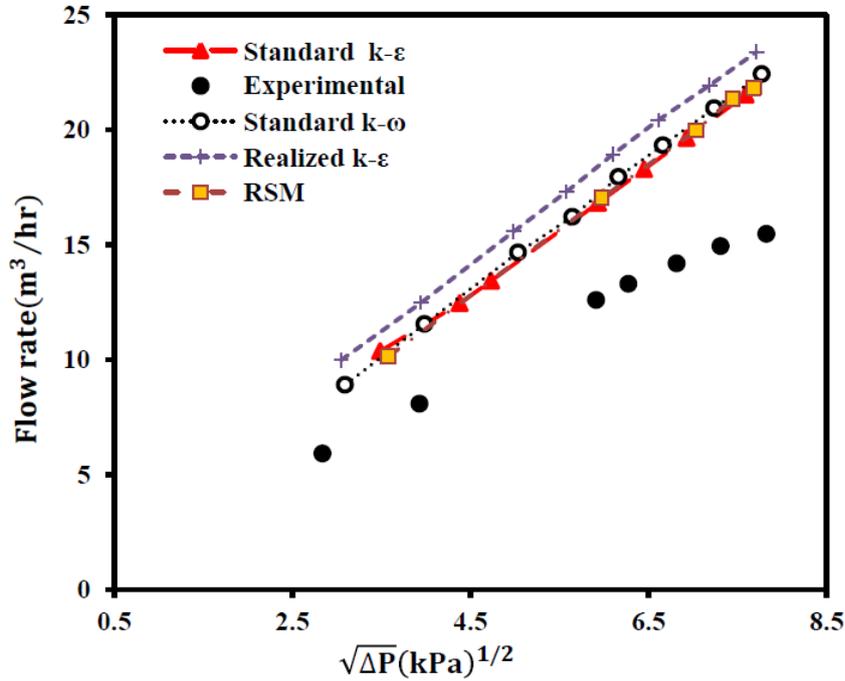


Figure 4. Experimental and numerical flow capacity curves for (α=40°)

S.k-ε is more effective than R.k-ε in studying large turbulence scales and eddy flow regime areas, as found in a small throttling passage in control valves. Furthermore, care must be taken while using these models in similar cases. Other turbulence models have been omitted due to including equations themselves are not stable, and computer time is significantly more than with the other models. S.k-ε can be regarded as the model with acceptable, reasonable accuracy, and less run times compared to other models.

3.2. Disk Angle 60°

By comparing Figure 5 with Figure 4, it is noted that the turbulence models haven't the same sort of accuracy. The degree of accuracy not only depends on the turbulence model but also on the geometry of the valve. Clearly, the question of which model is the best accurate is not straightforward one.

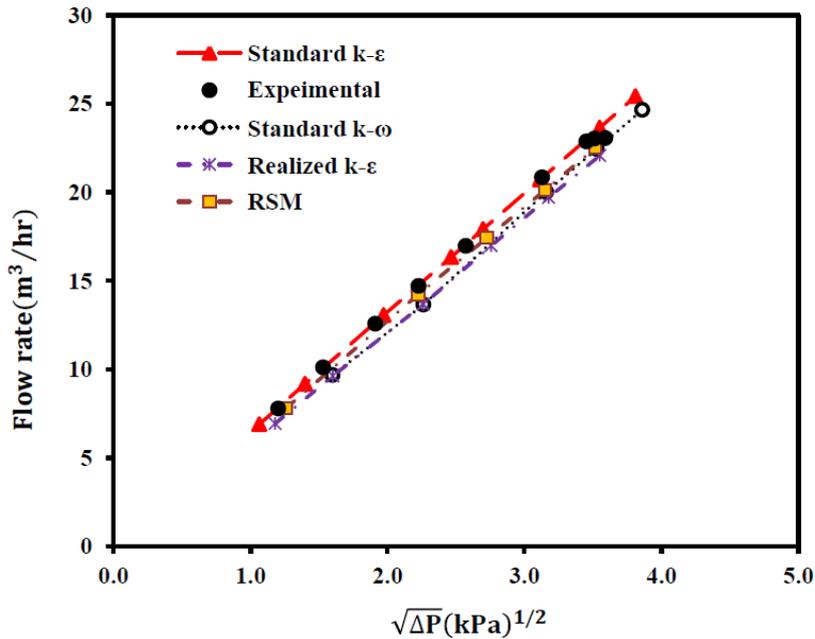


Figure 5. Experimental and numerical flow capacity curves for (α = 60°)

Many experts argue that the RSM is the only viable way forward towards a truly general-purpose. But recent advances in the area of non-linear $k-\epsilon$ model is very likely to reinvigorate research on two-equation turbulence models [18]. Although, in this case the RSM is the most accurate model for disk angle 40° , the S.k- ϵ and experimental curves being indistinguishable on the scale of Figure 5 and is the most accurate curve for disk angle 60° . Hence, it is fairly general to use RSM in variant of different geometries after it can be readily adapted.

3.3. Disk Angle 70°

In order to verify the dependency degree of the numerical result accuracy with the type of turbulence model, the solution for disk angle 40° and 60° is repeated

for disk angle 70° . As depicted in Figure 6 both $k-\omega$ and RSM models has the same contingency with the presence of the same pressure gradient. However at low pressure gradient the $k-\omega$ model is overestimate the experimental results and tends to be underestimate the results at higher pressure gradient. But the RSM is independent of pressure gradient and always is underestimate the experimental results. Consequently, comparison of $k-\omega$ and RSM models reveals that the $k-\omega$ model overweigh the RSM results with regard to pressure gradient. The R.k- ϵ and S.K- ϵ results are identical and agree well with the experimental results. However the R.k- ϵ errors increase in such an overestimation mode at higher pressure gradient. The S.k- ϵ gives the respectable agreement and is the refined model in an average sense.

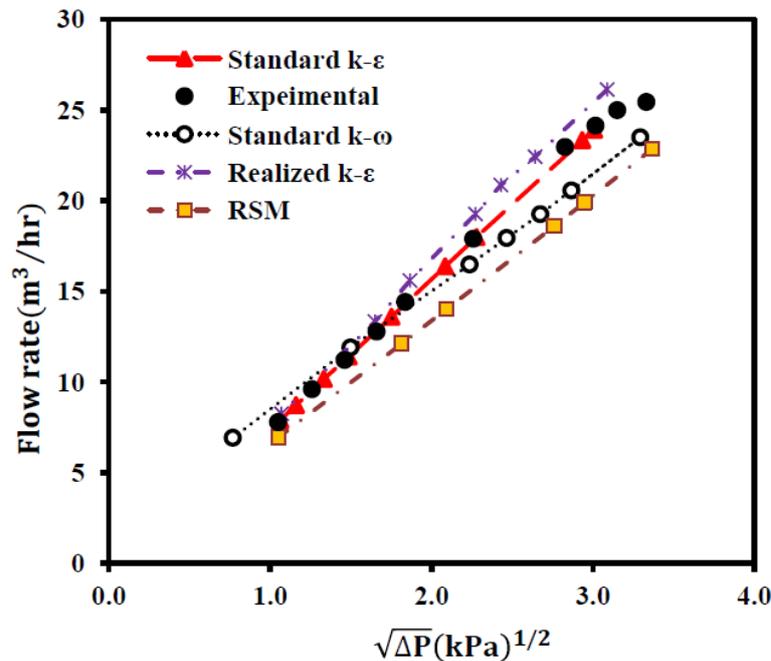


Figure 6. Experimental and numerical flow capacity curves for ($\alpha = 70^\circ$)

One of the primary observation that has been emerged from Figure 4, Figure 5, and Figure 6 comparison is that both R.k- ϵ and RSM model results are quite sensitive to valve disk opening. In general, the comparison with experimental yields good agreement at moderate disk angle. However at low disk angle, both models yield overestimate trend. At higher disk opening both models lead to increased errors with a discrepancy behavior. Although R.k- ϵ is overestimate the experimental results, in contrast, RSM model reveals underestimation behavior.

4. Conclusions and Recommendations

4.1. Conclusions

Experimental and 3D numerical simulation for (DN 50) butterfly valve at disk angles 40° , 60° , and 70° are presented in this study to evaluate the valve performance coefficient. Numerical model is meshed by GAMBIT 2.4 and tested before it is validated by experimental results. The results can be summarized as the following:

1. In dealing with numerical simulation by FLUENT 6.3, it highly appreciated that the user

of numerical solvers must exercise his own judgment to determine the appreciate turbulence model for each studied case to obtain acceptable results.

2. Disk angles 40° , 60° , and 70° are selected as worked examples to compare numerical solutions with experimental results and investigate the degree of accuracy of different turbulence models. As RSM model is the most accurate turbulence with long run computation times compared with S.k- ϵ model, some degree of compromise is required between good accuracy and short run time in case of selecting appropriated turbulence model.
3. The predicted butterfly valve flow coefficient via RSM model is aligned will with experimental data at small disk angles. However the highest agreement at disk angles 40° , 60° , and 70° is obtained by the aid of S.k- ϵ . Although $k-\omega$ model behavior for prediction tends to be underestimate the experimental results at higher disk angles, it introduces a well agreement independently of higher pressure gradients compared with the other turbulence models.

4.2. Recommendation

While efforts have been made to increase butterfly valve performance by numerical technique, a unique turbulence model for well prediction of flow coefficient to cover all operating ranges has not been realized to date. Due to relative errors at low disk angles and long computational time, more robust numerical solution with different meshes and higher computation resources to define the effect of the meshing types on the accuracy and the running time may be studied in future work.

List of Symbols

C_v	Flow coefficient [$\text{m}^3/\text{hr. kPa}^{1/2}$]
D	Valve disk diameter [m]
I	Turbulent intensity [%]
N	Number of mesh cells [-]
\bar{p}	Mean pressure [kPa]
P_1	Inlet static pressure [kPa]
P_2	Outlet static pressure [kPa]
R_D	Reynolds Number [-]
u	Velocity [m/s]
\bar{u}	Mean Velocity [m/s]
u_i, u_j	Velocity tensor [m/s]
τ_{ij}	Reynolds shear stress tensor [m^2/s^2]
x_i	Cartesian coordinates

Greek symbols

α	Butterfly valve disk angle [degree°]
ε	Energy Dissipation Rate [m^2/s^3]
ν	Kinematic viscosity [m^2/s]
ρ	Fluid density [kg/m^3]
ω	Turbulence frequency [1/s]

Abbreviations

ANSI	American National Standards Institute
CFD	Computational Fluid Dynamic
DN	Diameter Nominal
FSI	Fluid Structure Interaction
ISA	International Society of America
RANS	Reynolds-Averaged Navier-Stokes
RSM	Reynolds Stress Model

References

- [1] Jeon, S.Y., J.Y. Yoon, and M.S. Shin *Flow Characteristics and Performance Evaluation of Butterfly Valves using Numerical analysis*. Earth and Environmental Science, 2010. IAHR Symposium on Hydraulic Machinery and Systems.
- [2] Farid, V.T., et al., *Analysis of the Hydrodynamic Torque Effects on Large Size Butterfly Valves and Comparing Results with AWWA C504 Standard Recommendations*. Journal of Mechanical Science and Technology, 2012.
- [3] Leutwyler, Z. and C. Dalton *A CFD Study to Analyze the Aerodynamic Torque, Lift, and Drag Forces for a Butterfly Valve in the Mid-Stroke Position*. ASME 2004 Heat Transfer/Fluids Engineering Summer Conference, 2004: p. Paper HT-FED04-56016.
- [4] Wang, L., X. Song, and Y. Park *The Improvement of Large Butterfly Valve by Using Numerical Analysis Method*. International Conference on Fluid Mechanics (FLUIDS'09), 2009. Proceedings of the 6th WSEAS: p. 75-79.
- [5] Prema, B., S. Bhojani, and N. Gopalakrishnan *Design Optimization of Butterfly Valve using CFD*. Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, 2010(Paper FMFP10-FP-15.).
- [6] Shirazi, N.T., G.A. Reza, and H.A. Gholam *Analysis of the Ball Valve Performance in Presence of Cavitation*. Life Science Journal, 1460-1467, 2012
- [7] Chern, M.J. and C.C. Wang *Control of Volumetric Flow-Rate of Ball Valve Using V-Port*. Journal of Fluids Engineering, 2004. Vol. 126: p. 471-481.
- [8] Aditi, O., G. Sudipto, and C. Kanchan *CFD Modeling of Globe Valves for Oxygen Application*. 16th Australasian Fluid Mechanics Conference, Australia, 2-7 December 2007, 2007.
- [9] Guoyu, W., et al., *Dynamics of attached turbulent cavitating flows*. Progress in Aerospace Sciences 37 2001. (2001) 551–581.
- [10] Toro, A.D., *Computational Fluid Dynamics Analysis of Butterfly Valve Performance Factors*. Master of Science in Mechanical Engineering, 2012.
- [11] Elbakhshawangy, H.F., O.S. Abd El-Kawi, and H.H. Sarhan *Numerical Study of Incompressible Flow Characteristics Through Butterfly Valve* Journal of Multidisciplinary Engineering Science and Technology, 2015. Vol 2 (Issue : 7, July-2015).
- [12] Ibrahim, G., Z. Al-Otaibi, and H.M. Ahmed *An Investigation of Butterfly Valve Flow Characteristics Using Numerical Technique*. Journal of Advanced Science and Engineering Research, 2013. Vol 3, No 2 June (2013) 151-166.
- [13] Jun-Oh, K., et al., *Structural Design Strategy of Double-Eccentric Butterfly Valve using Topology Optimization*. World Academy of Science, Engineering and Technology 66 2012, 2012.
- [14] Hui, W., Y. Liu, and S. Wang *Numerical Simulation on Turbulent Flow in a Daul-Elbow Channel Behind Butterfly Valve*. Journal of Chinese Society of Power Engineering 34(6): 450-457. June 2014, 2014.
- [15] Băran, G., D. Bunea, and G.I. Oprina *On Cavitation and Cavitation Damage at Butterfly Valves* Scientific Bulletin of the "Politehnica" University of Timisoara Transactions on Mechanics Tom 52(66), Fascicola 6, 2007, 2007.
- [16] Mandica, S., *Robust Computations of Turbulent Flows*. Doctor of Philosophy of the University of Bath (UK) Department of Mathematical Sciences, 2006.
- [17] Huang, C. and R.H. Kim, *Three-Dimensional Analysis of Partially Open Butterfly Valve Flows*. Journal of Fluids Engineering, 1996. 118(3): p. 562-568.
- [18] Versteeg, H.K. and M. Malalasekera *An Introduction to Computational Fluid*. 2007 ed. 2007.
- [19] ISA, *ISA Standards, Recommend Practice and Technical Reports*. 1999. Control Valves -Vol. 3.