Numerical Study Of Incompressible Flow Characteristics Through Butterfly Valve

H. F. Elbakhshawangy[1], O. S. Abd El-Kawi[1] and H. H. Sarhan[2]
(1): Reactors department, Atomic Energy Authority of Egypt, P.O. box: 13759 , Cairo, Egypt,
elbakhshawangy@yahoo.com, usama_ali35@hotmail.com
(2): Mechanical Power Engineering Dept., Faculty of Engineering, PortSaid University, Egypt,
Heshamsarhan66@gmail.com

***Abstract*—In the present study incompressible, viscous, and turbulent flow of water through a butterfly valve of 20 cm diameter was considered. The aim of the present study was to provide comprehensive study of the effect of inlet velocity on the variation of flow characteristics with valve positions. For this purpose, different values of inlet velocity of 1,2,3,4 m/s were applied and for each case of inlet velocity, flow characteristics parameters such as velocity profile, pressure distribution, turbulent kinetic energy, loss coefficient and flow coefficient were obtained at different valve opening angles of 30,45,60,75 deg. The results showed that more smooth and free of turbulence flow with small pressure drop across the valve was obtained either at large valve opening angel or small inlet velocity. So, to operate the valve at wide range of flow rate, it is not recommended to operate it at small opening angel. The loss and flow coefficients do not depend on the flow velocity, but strongly depends on the valve opening angel. The loss coefficient decreased whereas the flow coefficient increased by increasing the opening angel. Equations relating the loss and flow coefficients with opening angles for the studied valve size were obtained.**

Keywords—Numerical study; flow characteristics; butterfly valve

I. INTRODUCTION

A butterfly valve (Fig. 1) [1] is a type of flow control device that controls the flow of gas or liquid in a variety of processes and piping applications. These applications includes oil & gas transportation, power generation plants, petrochemical plants, water distribution, sewage and fire fighting applications. The three main components of a butterfly valve are the valve body, the valve disk and the shaft (stem) which is typically connected to an actuator. The shaft acts as the center of rotation for the disk, and turns the disk in a 90 degree motion from a position parallel to the fluid flow (fully opened position) to a position where the disk is perpendicular to the fluid motion (fully closed position). For that reason the butterfly valve belongs to the quarter turn valves family [1]. The butterfly valve is one of the most widely used valves. They are commonly used as on-off and controlling device or throttling valves (for flow or pressure control), in many industrial applications [2, 3, 4]. The most outstanding advantages of butterfly valves are simple mechanical assembly, light weight, low cost, their low pressure drop when they are in fully open position, and the quick and easy shift from a full open to a full closed position [5, 6].

In the past, scientists performed experiments to analyze butterfly valves flow characteristics in laboratories, which required a variety of equipments, lots of time and funds. With the fast progress in computer visualization and numerical analysis techniques and the development of Computational Fluid Dynamics (CFD), it has become possible, nowadays, to perform quick, accurate and advanced analysis using simulation technique [7]. In fact, numerical analysis has not only being used to study the flow characteristics of butterfly valves, their performance, but also to improve their design. Jeon et al. [8] performed numerical analysis on some kind of butterfly valves using commercial CFD code FLUENT. According to the valve opening, the pressure distribution, the flow coefficient, and the loss coefficient were studied. The influence of these design parameters on the valve performance were checked. Kim and Wu (1993) studied the flow pattern, velocity distribution and flow coefficient of butterfly valve through two-dimensional numerical analysis. Huang and Kim [9] used the commercial code FLUNT to study the velocity field and pressure distribution for three-dimensional incompressible flow around butterfly valves. Their results show the optimum design of the butterfly valve for stable flow regulation, smooth opening and shutting ability. Wojtkowiak and Oleskowicz [10] performed experimental and numerical investigations of butterfly valve flow characteristics, flow fields and pressure distributions. They compared their results with available literature data. On the basis of these results credibility of the used CFD model was estimated. Acceptable comparisons were reported. They proposed an improved flow characteristics equation. Song and Chul Park [11] used the commercial code ANSYS CFX 10.0 represents the flow behaviour and provide a three-dimensional numerical simulation of water around the butterfly valve and estimate the pressure drop, flow coefficient and hydrodynamic torque coefficient. It is the first step towards improving valve design. B. Prema, et al. [12] studied the effect of pressure drop, flow coefficient and flow behaviour across different valve design, and optimized design was suggested. The calculated flow coefficient for the optimized design shows 56.8% improvement compared to baseline design and the hydrodynamic torque coefficient which controls the opening and closure of the valve seems to be higher in the case of optimized valve design. Ghaleb Ibrahim et al. [13] implemented a numerical simulation for flow of water past over a butterfly valve using commercial fluid dynamics software FLUENT. Velocity profile, pressure distribution, turbulence kinetic energy and turbulence intensity are the parameters used to present the characteristic of flow. Their results showed that turbulence in flow starts at the edges of the valve disc and gets growing according to the specified case. These vortices and circulation region are generated always in downstream region behind the valve disc. 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. The aim of the present study was to provide comprehensive study of the effect of inlet velocity on the variation of flow characteristics with valve positions. To accomplish this purpose, different cases of inlet velocity have been studied. For each case of inlet velocity, flow characteristics parameters such as velocity profile, pressure distribution, and turbulent kinetic energy were obtained at different valve opening angles. Numerical simulation of flow of water through a butterfly valve was used to obtain the aforementioned parameters. The simulation technique was carried out using commercial fluid dynamics software ANSYS. A typical butterfly valve is shown in Fig. 1 [1].

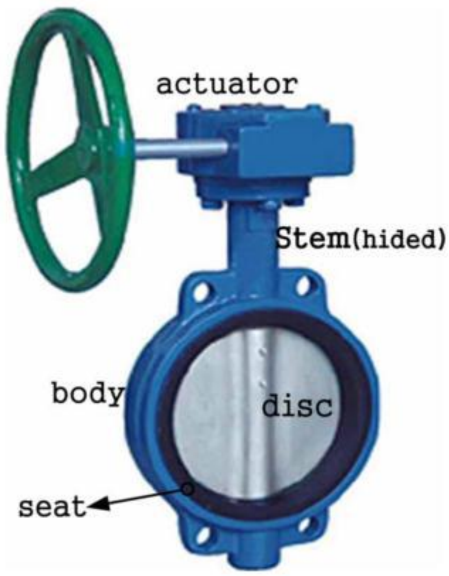

Fig.1: Butterfly valve

II. GOVERNING EQUATIONS

Incompressible, viscous, and turbulent fluid flow through the butterfly valve was considered. This type of flow is described by the Reynolds averaged Navier Stoke equations

(RANS). The (RANS) is considered to be a modification of the unsteady Navier-Stoke equations (NS) by introducing turbulent quantities in the form of the sum of average and fluctuating quantities in the standard (NS), and taking the average of the instantaneous quantities of the two sides of (NS) over a long period of time. So, the (RANS) for the case of two dimensional flows will be in the form:

Momentum equations:

$$\rho(\bar{U}\frac{\partial \bar{U}}{\partial x}+\bar{V}\frac{\partial \bar{U}}{\partial y}+\frac{\partial \overline{\grave{U}^2}}{\partial x}+\frac{\partial \overline{\grave{U}\grave{V}}}{\partial y}) =- \frac{\partial \bar{P}}{\partial x} +\mu(\frac{\partial^2 \bar{U}}{\partial x^2}+\frac{\partial^2 \bar{U}}{\partial y^2})$$

$$\rho(\bar{U}\frac{\partial \bar{V}}{\partial x}+\bar{V}\frac{\partial \bar{V}}{\partial y}+\frac{\partial \overline{\grave{V}^2}}{\partial y}+\frac{\partial \overline{\grave{U}\grave{V}}}{\partial x}) =- \frac{\partial \bar{P}}{\partial y} +\mu(\frac{\partial^2 \bar{V}}{\partial x^2}+\frac{\partial^2 \bar{V}}{\partial y^2})$$

Continuity equation:

$$\frac{\partial \rho \bar{U}}{\partial x}+\frac{\partial \rho \bar{V}}{\partial y}$$

Where $\bar{U}, \bar{V}$ are the average velocity components, ù, v̀ are the fluctuating velocity components in x and y directions respective. The (RANS) produce additional unknown quantities known as Reynolds or turbulent stresses which are the products of the fluctuating quantities. These new unknown quantities need to be modelled by additional equations containing known quantities to obtain closed form solution. The most commonly used models to achieve this purpose are: k-ε, k-ω, and Reynolds shear stress model. The k-ε turbulent model is used as it is more accurate and robust for valve flow as indicated by Mohammadi and Pironneau [14]. Unlike other turbulence models, k-ε model focuses on the mechanisms that affect the turbulent kinetic energy. The underlying assumption of this model is that the turbulent viscosity is isotropic, in other words, the ratio between Reynolds stresses and mean rate of deformation is the same in all directions. For isotropic turbulence: $\overline{\grave{U}^2} = \overline{\grave{V}^2}$.

III. CFD MODEL

III.I Numerical method

Solving the previous (RANS) equations analytically is so complicated, so these equations are solved by a finite element base technique. The commercial computer code ANSYS is used for this purpose. The k-ε turbulent model was chosen as discussed in the preceding section to be used by the computer code. The used release of the ANSYS program is 5.4. Solid modelling technique is used for the purpose of model generation. In solid modelling the geometric boundaries are described, control over the size and desired shapes of elements are established, and then the ANSYS program is constructed to generate all the nodes and elements automatically. The physical properties are supplied as fixed values. Single phase for water flow is considered, and the problem domain does not change.

III.II Model Geometry

The valve disk considered for the present study is of 20 cm in diameter, 6 cm in thickness with a pipe diameter of 20.6 cm. The gap of 6 mm between the valve disk and the pipe wall is filled with a rubber seal around the rim of the valve. This gap was not modelled as it is very small and was not expected to have a significant effect on the flow through the valve. According to the research of Huang and Kim [9] the upstream and downstream pipe lengths should be at least 2 and 8 disk diameters respectively. So, the upstream and downstream pipe lengths were set to be 70, and 250 cm respectively.

III.III Boundary conditions

Different values of inlet velocity were used to study the effect of inlet velocity on the variation of flow characteristics with valve opening, angles. These values are 1, 2, 3, and 4 m/s. Developed uniform velocity profile is assumed for each value, while the pressure at the outlet was set to zero Pascal. No-slip boundary conditions at the wall was set for valve disk and the pipe.

IIII. RESULTS AND DISCUSSION

A numerical technique using commercial computer code ANSYS was used to simulate the flow of water through a butterfly valve. Flow field around the valve has been studied for different opening angles of 30, 45, 60, and 75 of the valve and at different inlet velocity of 1, 2, 3, and 4 m/s. For each case the velocity profile, hydrodynamic pressure, and turbulent kinetic energy are obtained. As indicated before, the studied length around the valve is 2D and 8D upstream and downstream respectively.

IIII.I Velocity Profiles

Figures 2-5 showed the effect of inlet velocity on the velocity profiles for different valve opening angles. From these figures it can be shown that for each case of inlet velocity, the flow velocity remained nearly constant and equal to the inlet velocity until it approached the valve where the velocity decreased and reached zero at stagnation region on the valve body. The flow became more smooth and velocity profile more uniform by increasing the valve opening angle. For small opening angles and because of the reduction in the flow area between the pipe and the valve edges, jet flow was created and the velocity increases and reaches maximum values in this region (red and yellow colours in the figures). This maximum value increased with the increase in inlet velocity. These high velocities caused Vortices, eddies and recirculation flow on the downstream surface of the valve body. These

Vortices increased with the decrease in the opening angles. There was some turbulence then because of the mixing regions of different velocities until the flow approaches its inlet velocity value again. These results agreed with the results of Ibrahim et al. [13]. The downstream distance from the valve at which the flow regains its inlet velocity value became smaller by increasing the valve opening angle. By comparing the velocity profiles at opening angles in figures 2 to 5, the following results can be summarized. By decreasing the inlet velocity, the maximum velocity value which existed at the valve edges decreased, the difference between the maximum and minimum velocity inside the pipe decreased, the turbulence and secondary flow decreased, the velocity became more uniform, the flow more smooth i.e. free of turbulence and vortices and approach the case of flow around a stream body , and in case of turbulence created because of small opening angles, the downstream distance for the flow to regain its inlet velocity value became smaller.

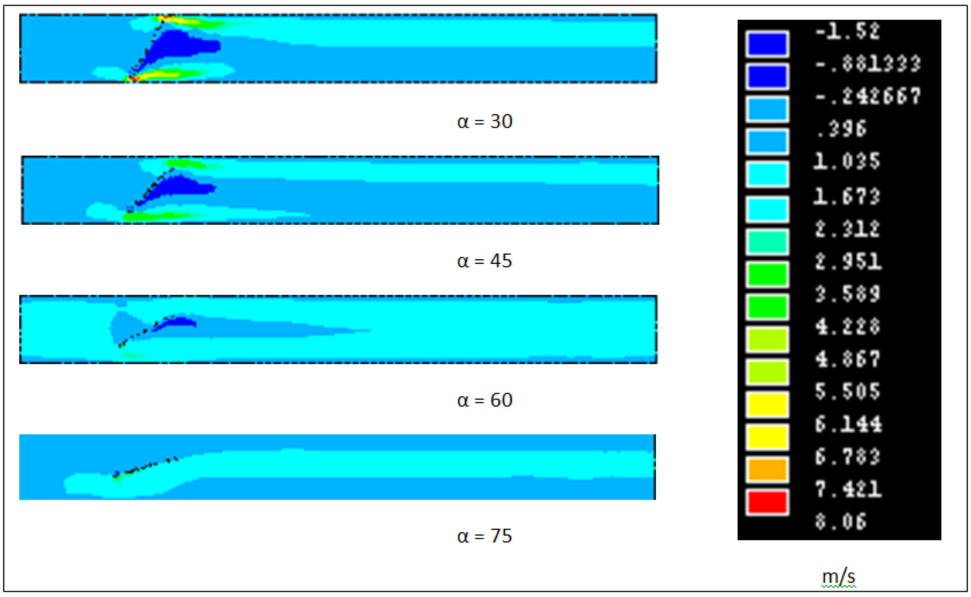

Fig.2: Change of velocity profile with opening angles for 1 m/s inlet velocity

So, it s clear that the effect of inlet velocity is that the opening angel at which turbulence starts to occur increases by increasing the inlet velocity. So, smooth flow is obtained either by decreasing the inlet velocity or increasing the opening angel.

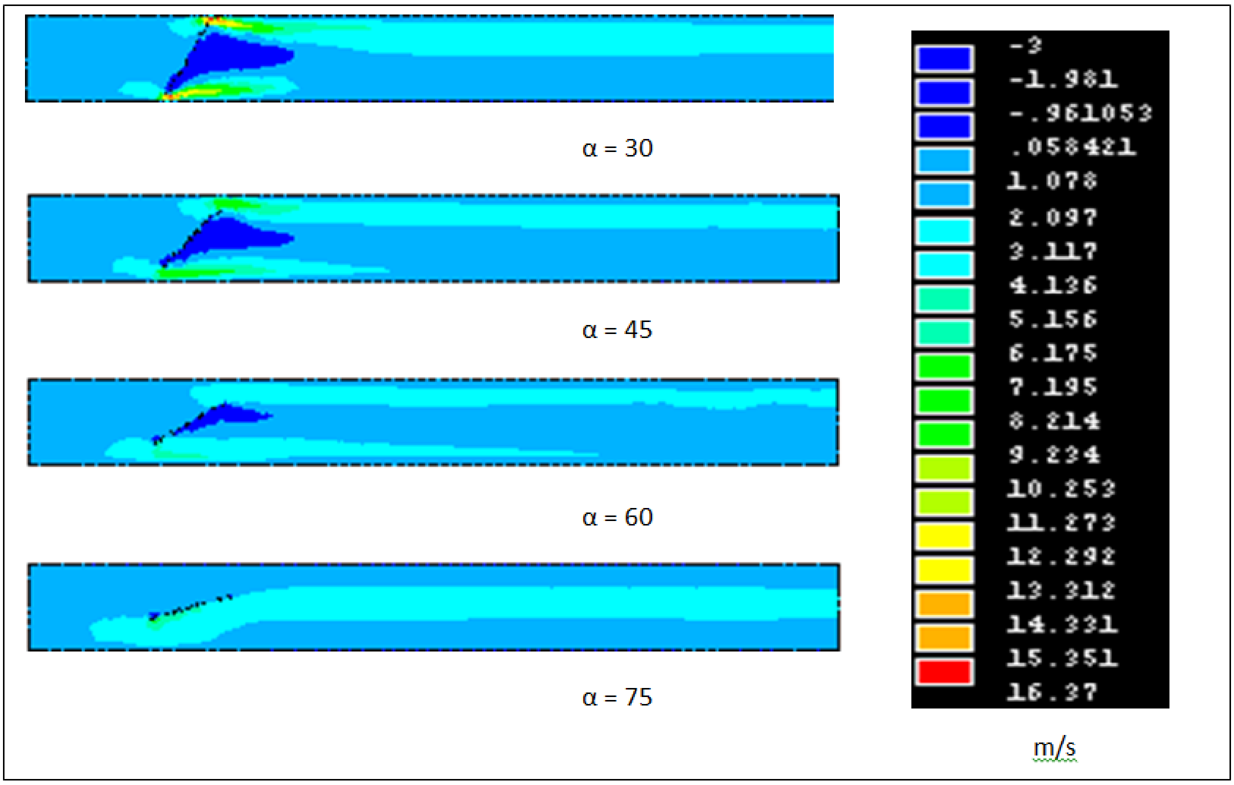

Fig.3: Change of velocity profile with opening angles for 2 m/s inlet velocity

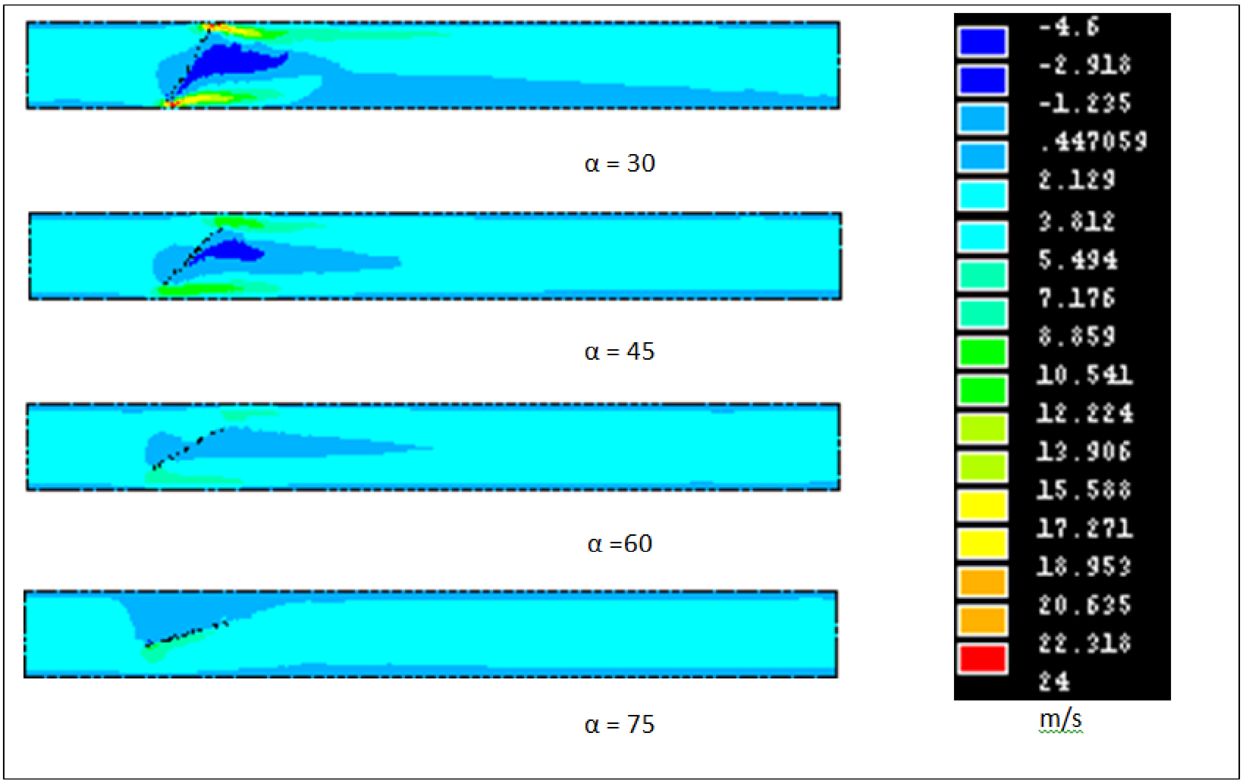

Fig.4: Change of velocity profile with opening angles for 3 m/s inlet velocity

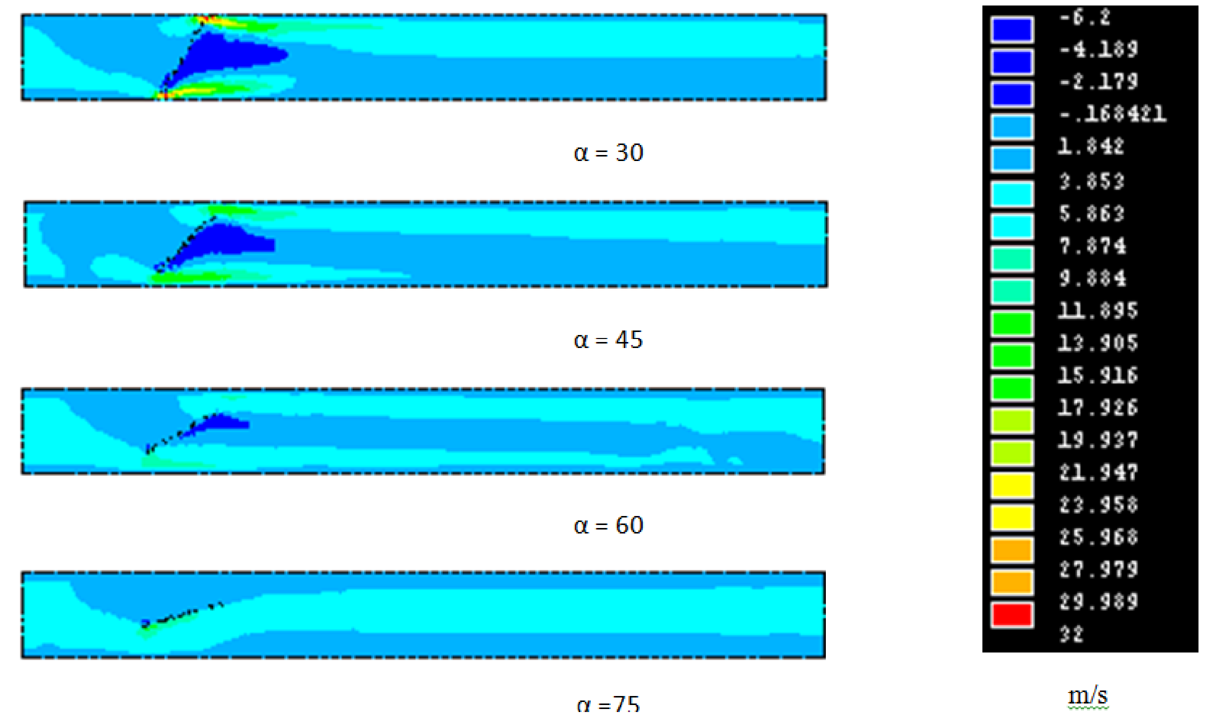

Fig.5: Change of velocity profile with opening angles for 4 m/s inlet velocity

IIII.II Pressure Distribution

Figures 6-9 showed the effect of inlet velocity on the dynamic pressure distribution for different valve opening angles. From these figures, it can be shown that the highest values of pressure were always upstream the valve disc and occurs and this was because the existence of the valve in the way of flow increases the hydraulic resistance to the flow and this increases the pressure upstream. Whereas, the pressure values significantly decreased at the valve edges where the velocity values are maximum in this region as discussed before. These results were also agree with the results of Ghaleb Ibrahim et al. [13]. So, by increasing the opening angel the pressure values in the upstream region decrease as the hydraulic resistance to the flow decrease Whereas the pressure values increased at the valve edges. For small opening angles, Small and negative pressure values existed in the downstream region immediately behind the valve, where recirculation flow and vortices existed. Getting far from the valve in the downstream region the pressure increased again. By increasing the valve opening angel, the pressure values throughout the flow field became closer, I.e. the pressure drop decreased, and regions of negative pressure values disappeared. The effect of inlet velocity was that by increasing the flow inlet velocity, the maximum pressure value which existed upstream the valve body increased, whereas the values at the valve edges where maximum velocity existed decreased, because of the increase in recirculation flow and vortices immediately downstream the valve body, the pressure values in this zone decreased, negative pressure values in this zone increased, and pressure difference around the valve increased at the same opening angel.

IIII.III Turbulent Kinetic Energy

Figures from 9 to 12 showed the effect of inlet velocity on the kinetic energy at different for different valve opening angles. From these figures it can be shown that, maximum values of turbulent kinetic energy existed at the valve edges, where maximum velocity existed, and decreased gradually in the downstream region until the kinetic energy became uniform away from the valve body. So, it was assured that there was turbulence at the valve edges and in the downstream region immediately behind the valve. These turbulence was attenuated by opening the valve. The effect of inlet velocity as shown from these figures was that, by increasing the inlet velocity turbulence became larger and more pronounced and turbulence extent behind the valve increased. So, degree of turbulence, and its extent were increased by either closing the valve at the same inlet velocity or increasing the flow inlet velocity at the same opening angel.

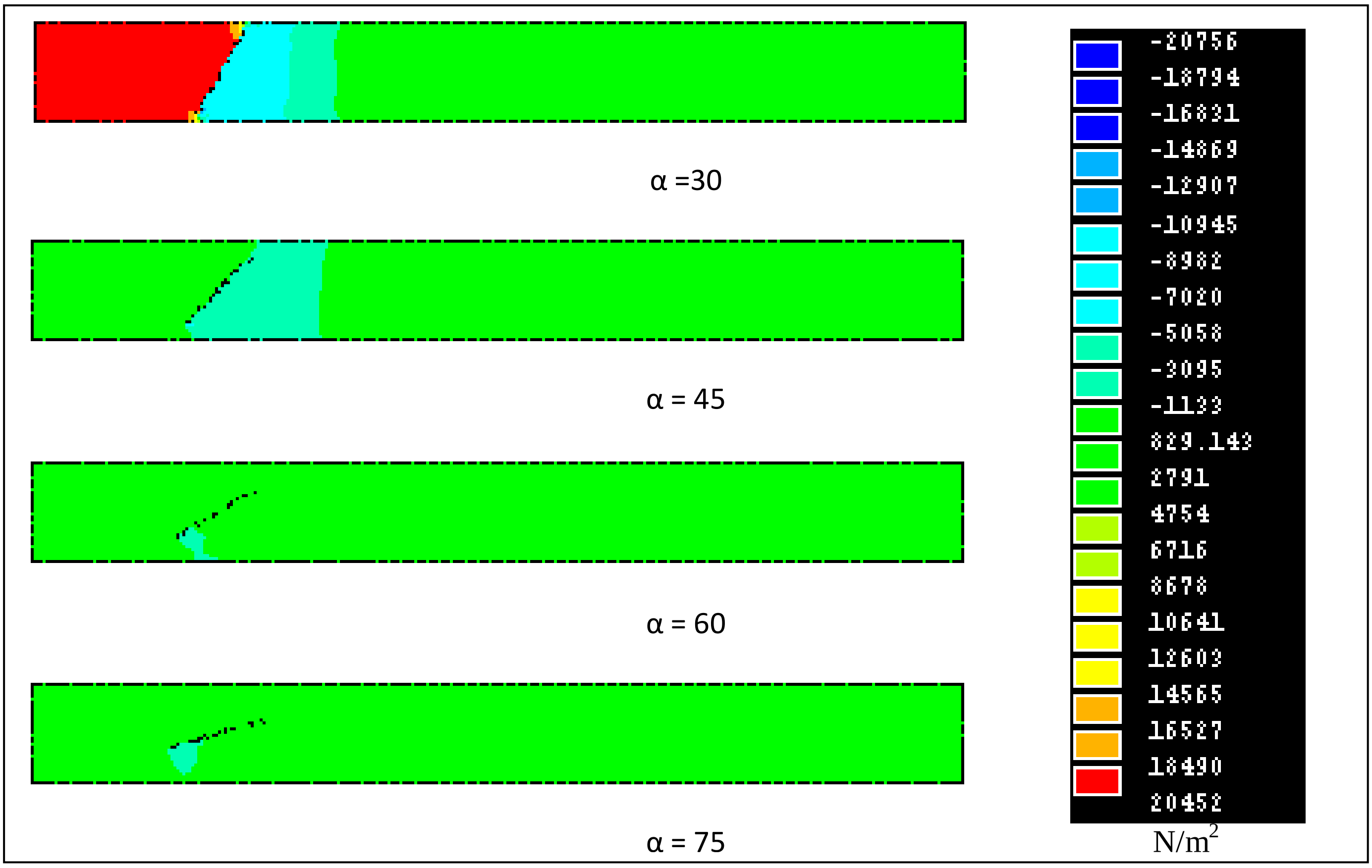

Fig.6: Change of pressure distribution with opening angles for 1 m/s inlet velocity

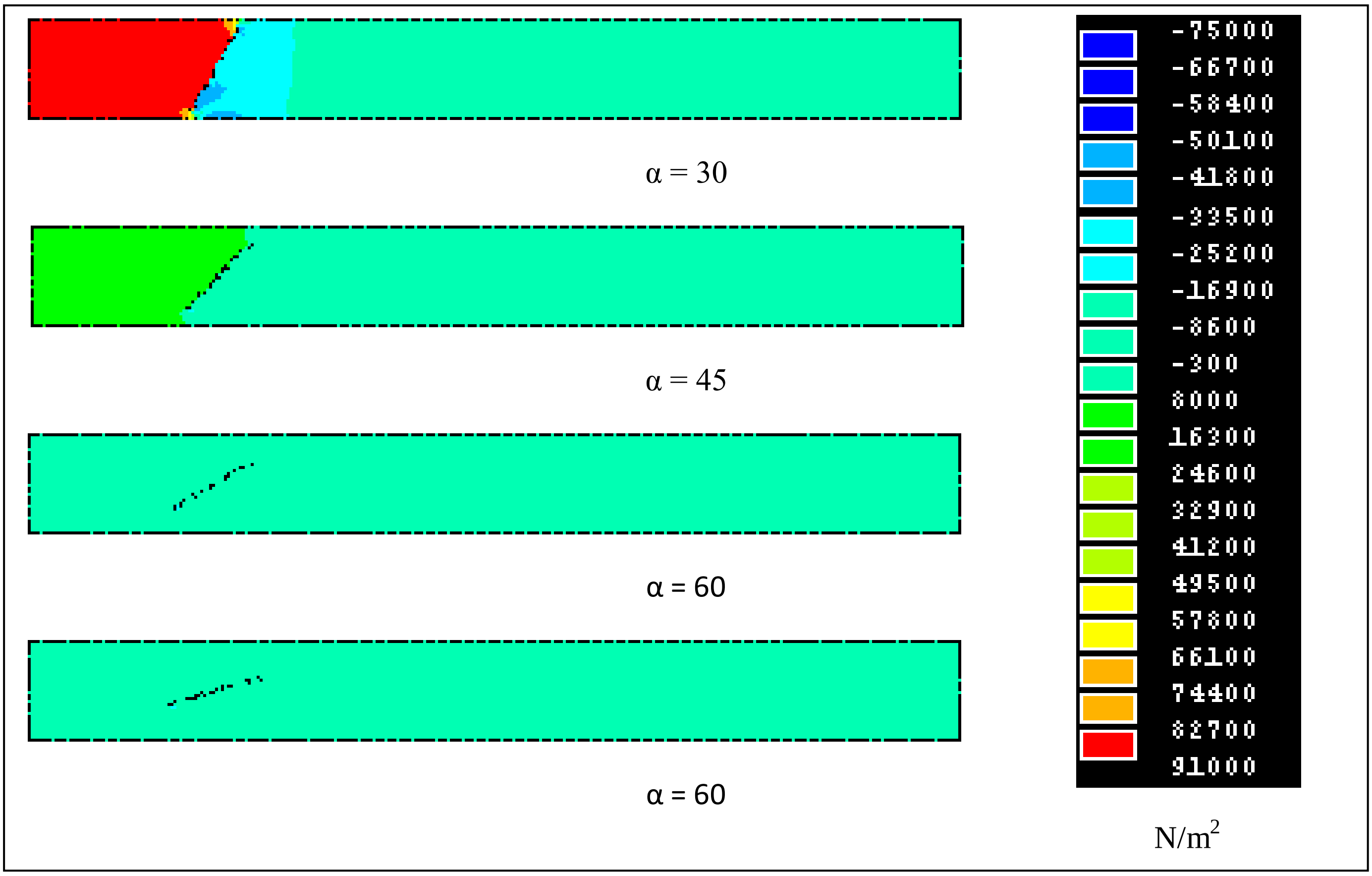

Fig.7: Change of pressure distribution with opening angles for 2 m/s inlet velocity

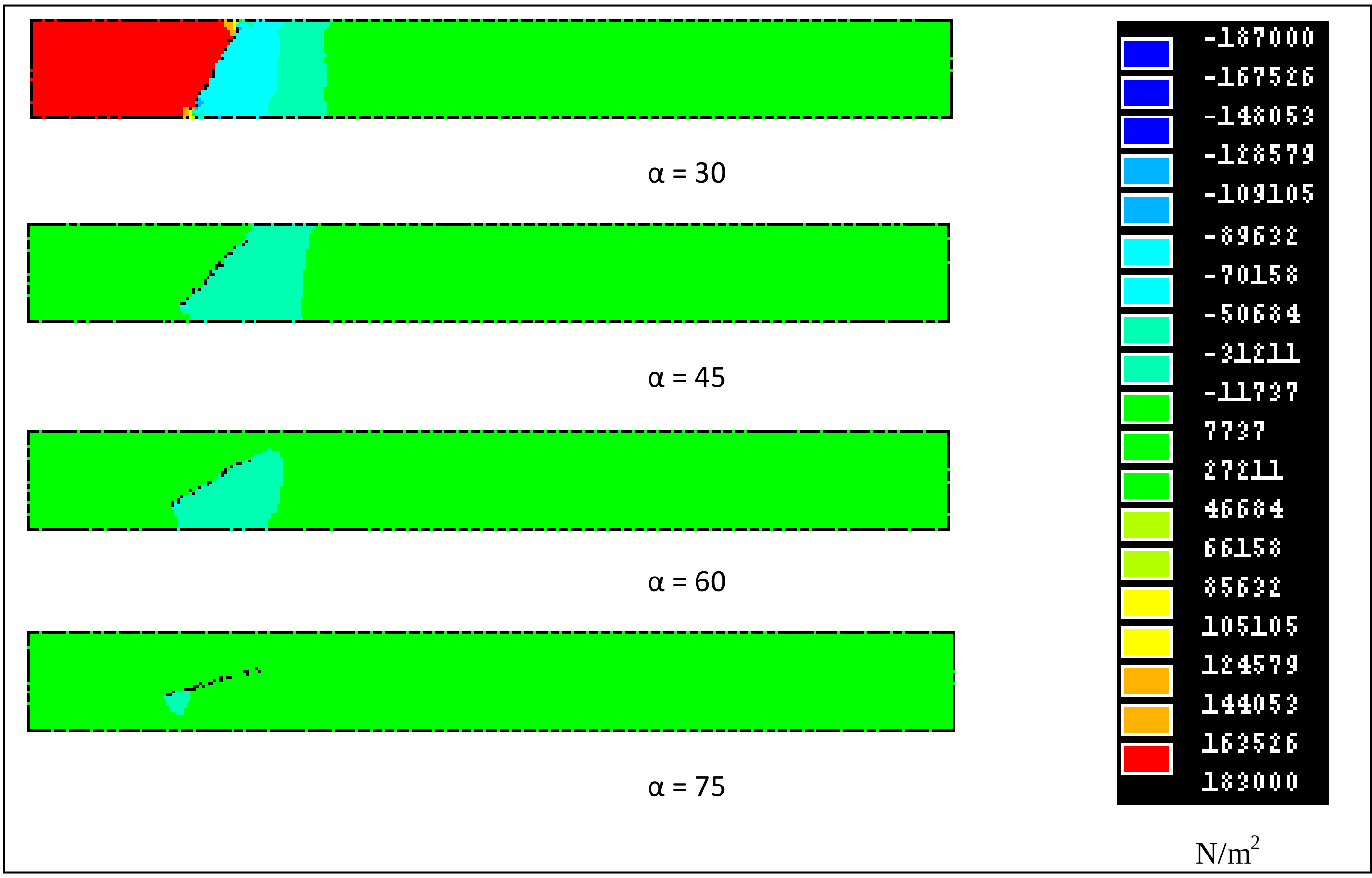

Fig.8: Change of pressure distribution with opening angles for 3 m/s inlet velocity

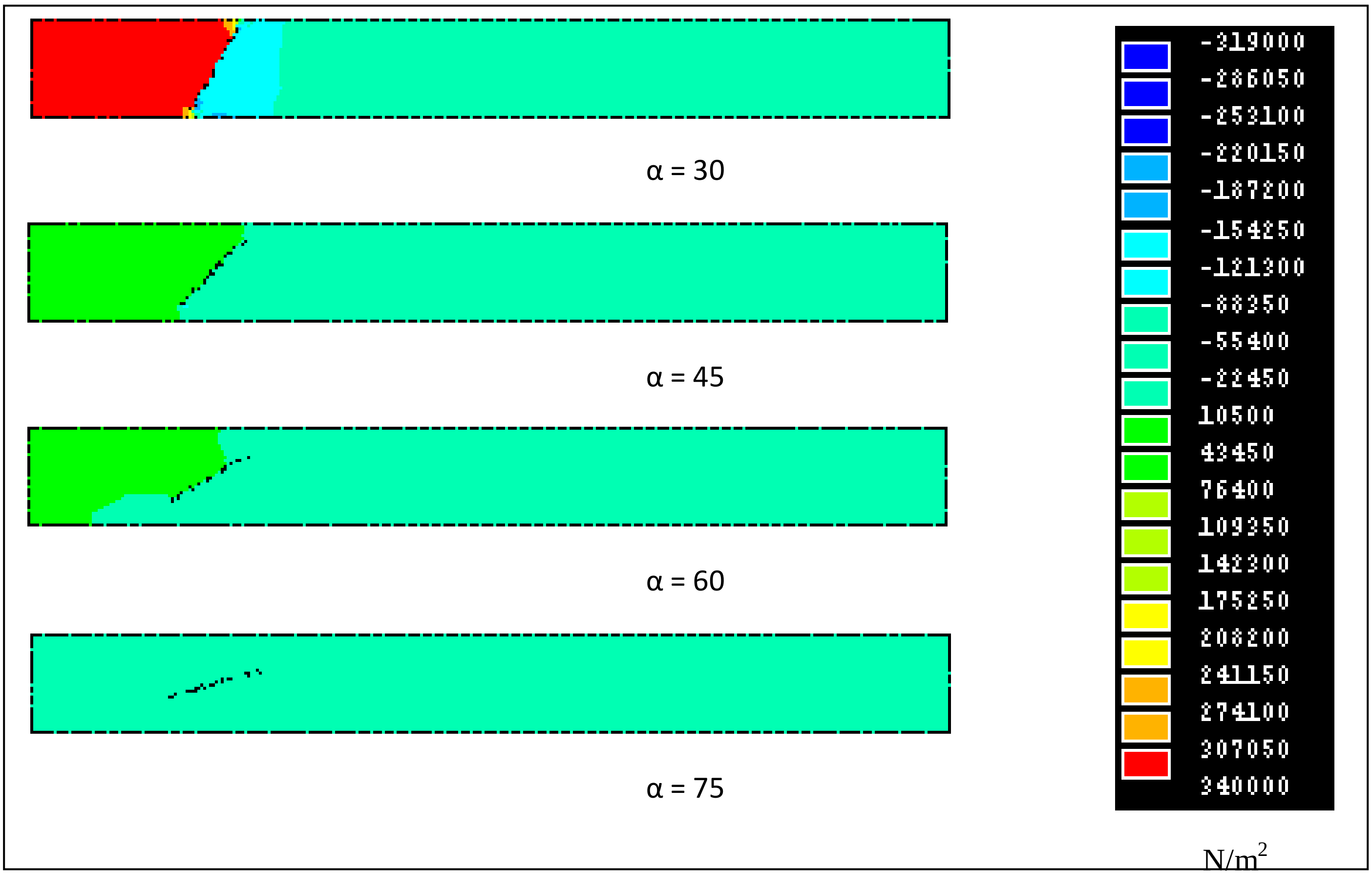

Fig.9: Change of pressure distribution with opening angles for 4 m/s inlet velocity

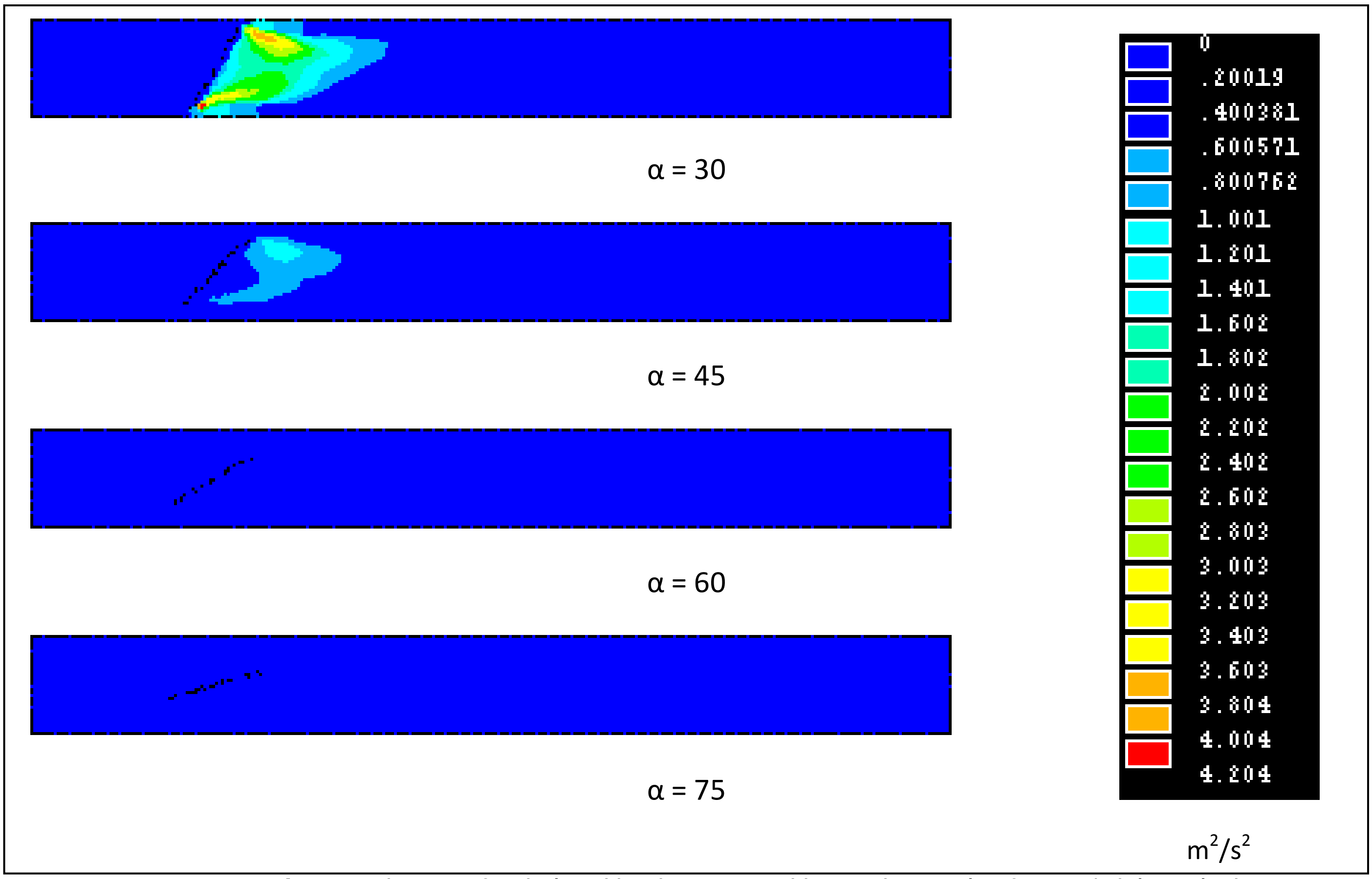

Fig.10: Change of turbulent kinetic energy with opening angles for 1 m/s inlet velocity

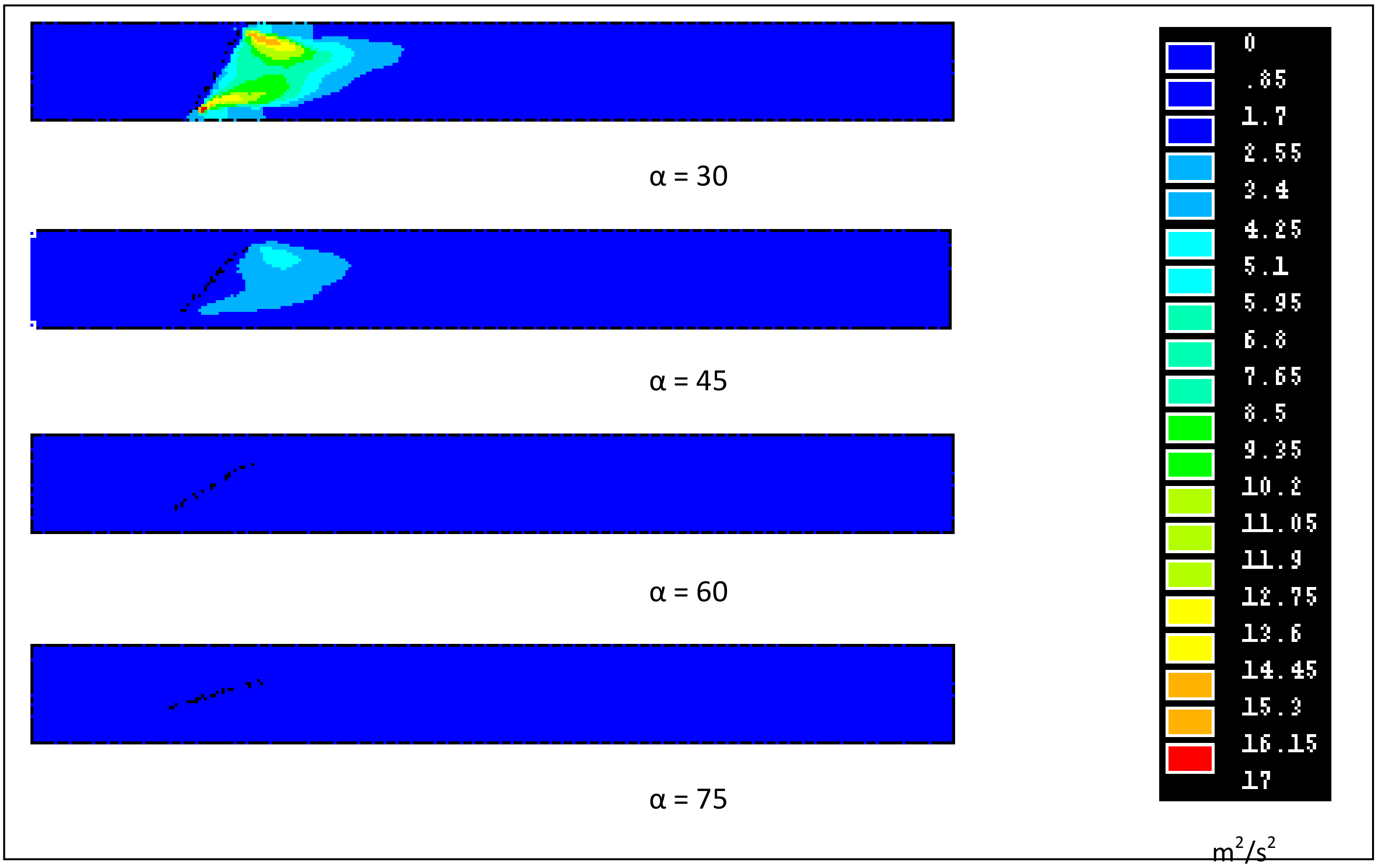

Fig.11: Change of turbulent kinetic energy with opening angles for 2 m/s inlet velocity

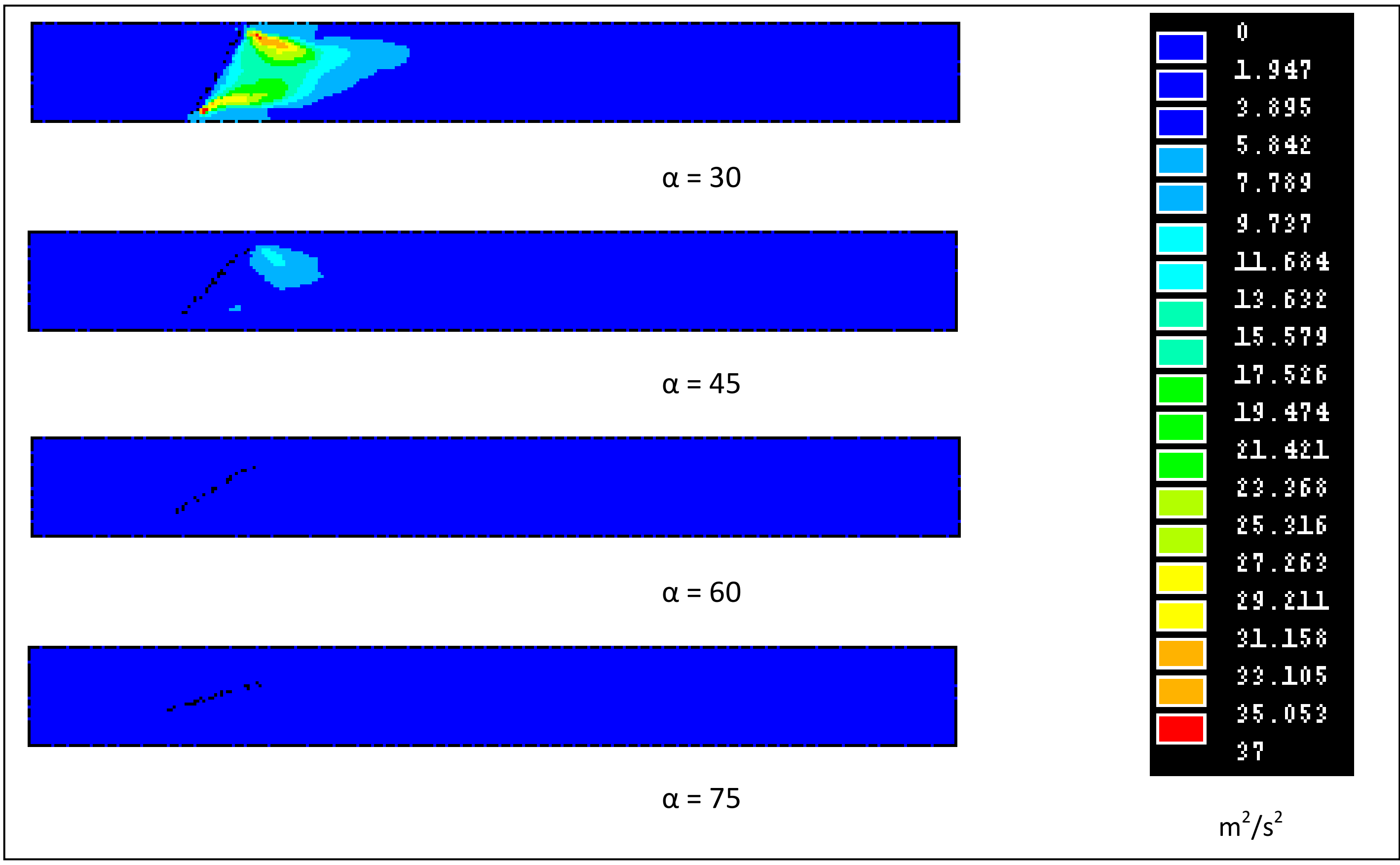

Fig.12: Change of turbulent kinetic energy with opening angles for 3 m/s inlet velocity

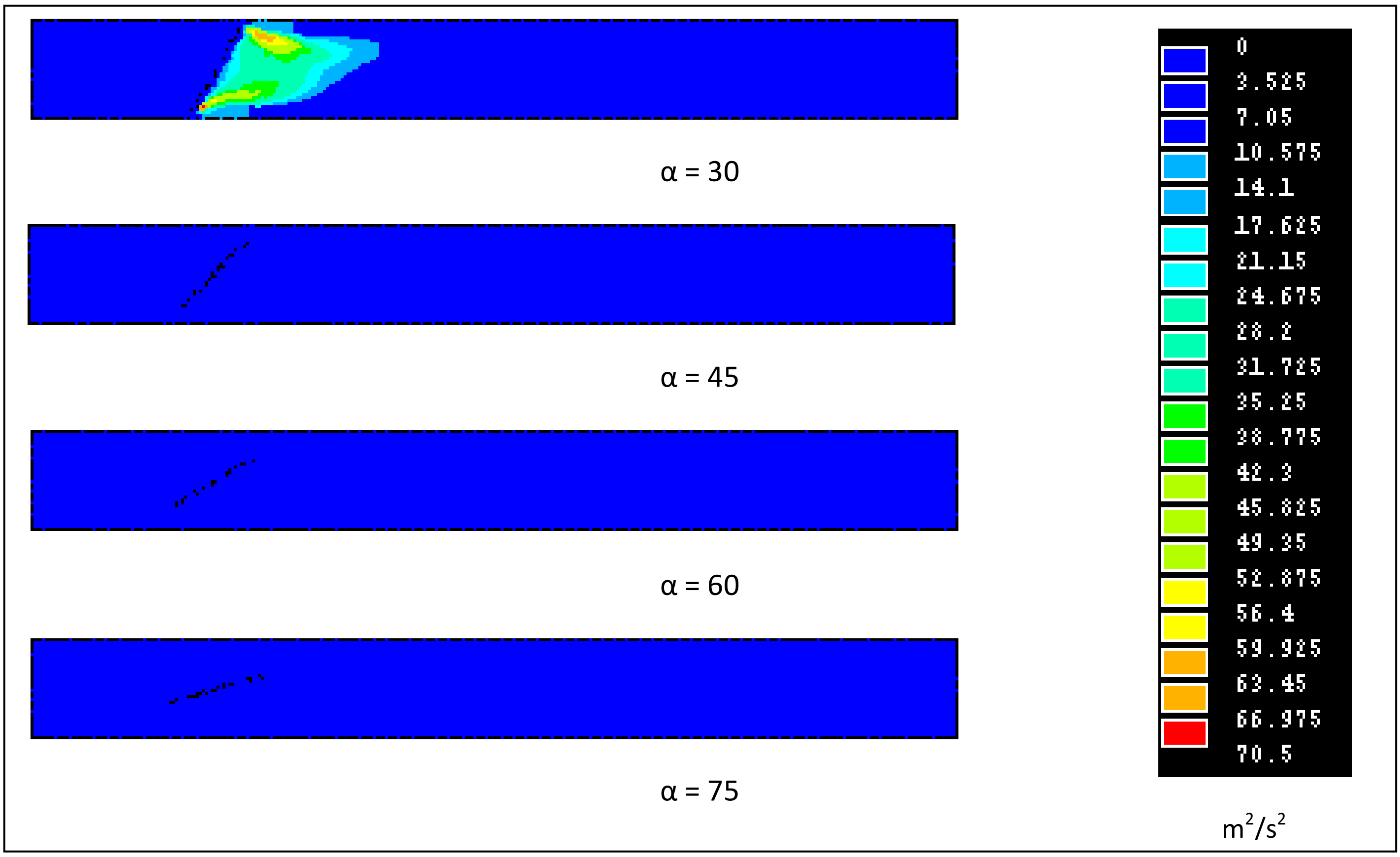

Fig.13: Change of turbulent kinetic energy with opening angles for 4 m/s inlet velocity

IIII.IIII The Loss Coefficient

The variation of loss coefficient K with opening angle at different flow velocity of 1,2,3, and 4 m/s is shown in Fig.14. From this figure, it can be shown that the dependence of the loss coefficient on the flow velocity is nearly negligible, but it is a strong function of the opening angel. The loss coefficient decreases by increasing the valve opening angel. The best fit line is also shown on the figure and the equation for this line and for the studied range of opening angle for this size is also shown on the figure.

IIII.IIIII The flow coefficient

variation of flow coefficient C_v with opening angle at different flow velocity of 1,2,3, and 4 m/s is shown in Fig.15. From this figure, it can be shown that the dependence of the flow coefficient on the flow velocity is negligible, but it strongly depends on the opening angel. The flow coefficient increases by increasing the valve opening angel. The best fit line is also shown on the figure and the equation of this line and for the studied range of opening angles for this size of butterfly valve is also shown on the figure.

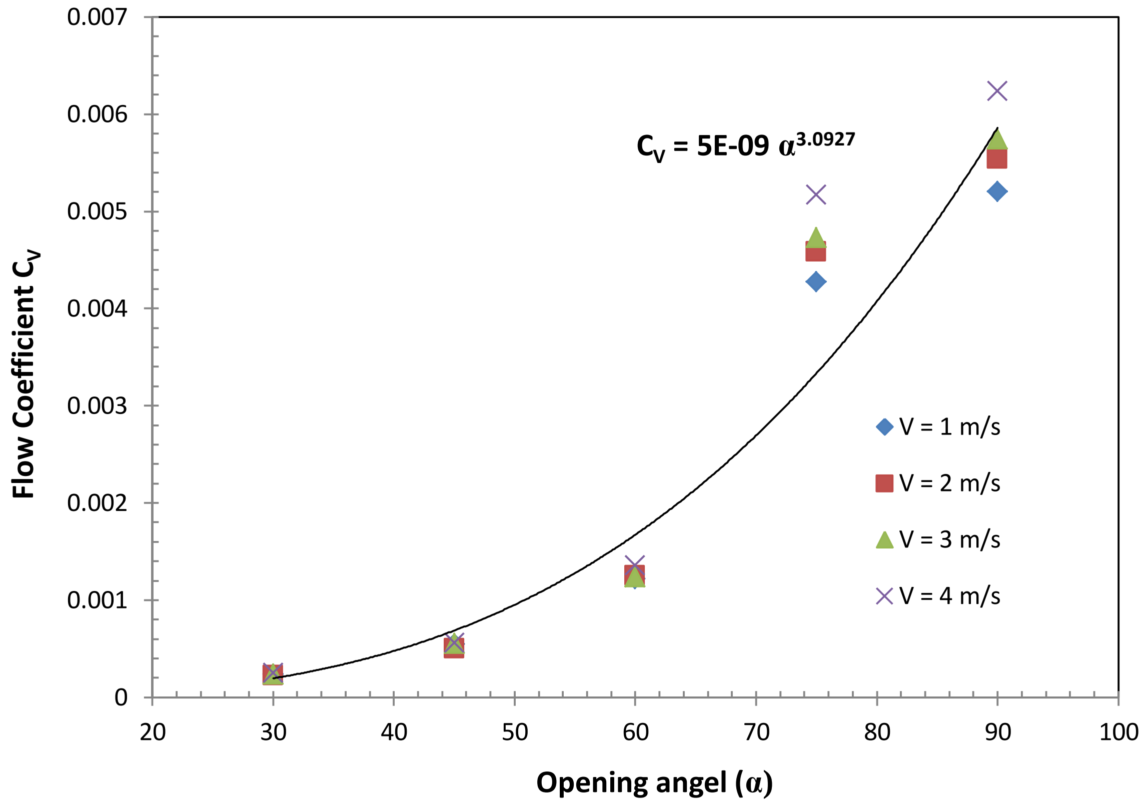

Fig.14: Variation of loss coefficient with opening angle at different flow velocities

Conclusion

1- By increasing the inlet velocity at the same valve opening angel, or decreasing the valve opening angel at the same inlet velocity, a turbulence region of Vortices, eddies and recirculation flow on the downstream surface of the valve body is created. The degree of turbulence and extent of turbulence region on the downstream region increase.

2- The effect of inlet velocity is that a more smooth, uniform and free of turbulence flow is obtained at smaller opening angel by decreasing the inlet velocity. So, for the valve to operate at a wide range of flow rate, it is not recommended to operate the valve at small opening angel.

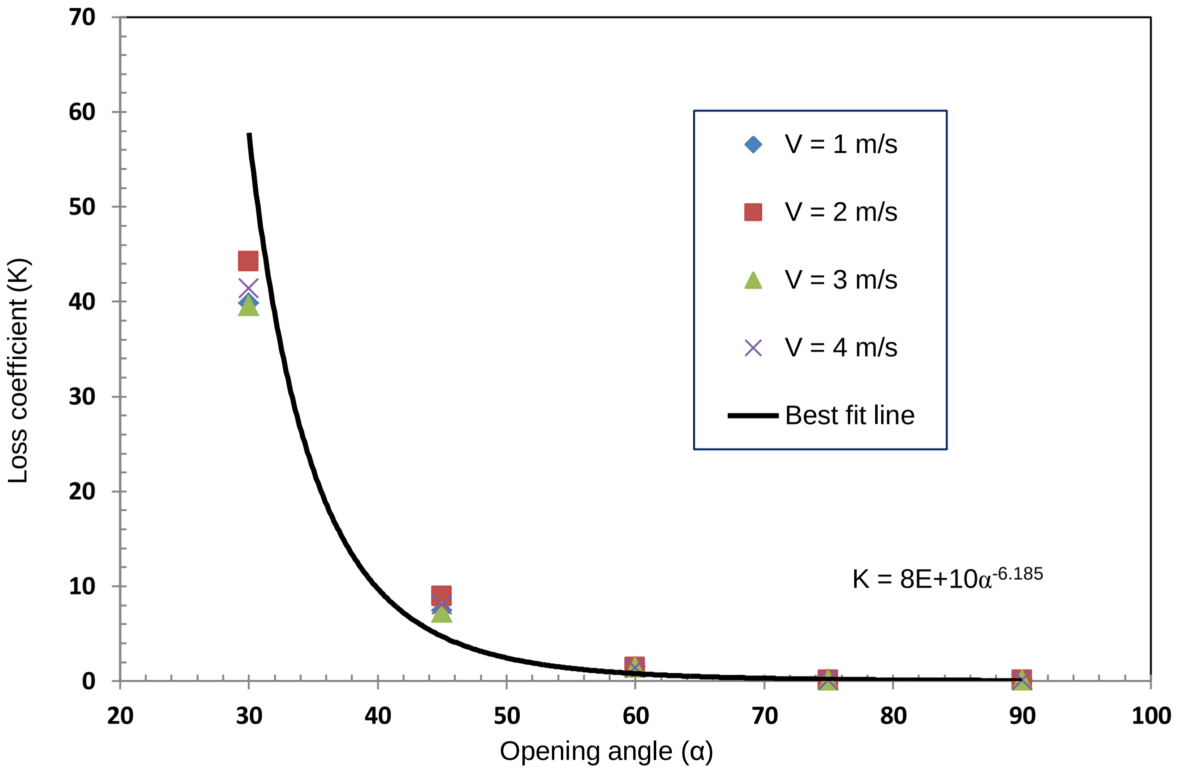

Fig.15: Variation of flow coefficient with opening angle at different flow velocities

3 The effect of flow inlet velocity is by increasing the inlet velocity, the maximum pressure value exists upstream the valve body increases, immediately downstream the valve body the pressure values in this zone decreases whereas negative pressure values in increases, and pressure difference around the valve increases at the same opening angel.

4 Turbulent kinetic energy and degree of turbulence are increased by either closing the valve at the same inlet velocity or increasing the flow inlet velocity at the same opening angel.

5 Free of turbulence more smooth flow with small pressure drop across the valve is obtained either at large valve opening angel or small inlet velocity. So, to operate the valve at wide range of flow rate, it is not recommended to operate it at small opening angel.

6 The loss and flow coefficients do not depend on the flow velocity, but strongly depends on the valve opening angel. The loss coefficient decreases whereas the flow coefficient increases by increasing the opening angel. The following equations relating the loss and flow coefficients with opening angel respectively for this size of butterfly valve are: $K = 8E{+}10\ \alpha^{-6.185}$, $C_V = 5E{-}09\ \alpha^{3.0927}$.

NOMENCLATURE

C_v	: Flow coefficient	Dimensionless
K	: Loss coefficient	Dimensionless
P	: Pressure	N/m^2
u	: Velocity component in x direction.	m/s
v	: Velocity component in y direction	m/s
x	: Distance in the transverse direction.	m
y	: Distance in the longitudinal direction	m

GREEK LETTERS

α	: Valve opening angel	deg
ρ	: Fluid density	kg/m^3
μ	: Fluid vescocity	Ns/m^2

References

1- A. Dawy, A. Sharara, A. Hassan, “A numerical investigation of the incompressible flow through a butterfly valve using CFD,” International Journal of Emerging Technology and Advanced Engineering, Volume III, Issue XI, November 2013.

2- P. Naseradinmousavi, , & C. Nataraj, “Nonlinear mathematical modelling of butterfly valves driven by solenoid actuators," Applied Mathematical Modelling, 35(5), 2011, 2324-2335.

3- M. Sandalci, E. Manchan, E. Alpman, and k. Kucukada, "Effect of the flow conditions and valve size on the butterfly valve performance, " J. of Thermal Science and Technology 30, 2, 103-112, (2010).

4- Z. Leutwyler, & C. Dalton, "A CFD study to analyze the aerodynamic torque, lift, and drag forces for a butterfly valve in the mid-stroke position," In ASME HT FED Summer 2004 Conference, Charlotte, North Carolina, 2004, July.

5- C. Kwuimy & C. Nataraj, "Modeling and dynamic analysis of a magnetically actuated butterfly valve," Nonlinear Dynamics, 70(1), 2012), 435-451.

6- S. Han, J. Ahn, K. Lee, S. Han, "Theoretical seismic analysis of butterfly valve for nuclear power plant, " Transactions of the Korean Society of Mechanical Engineers, A. Volume 36, Issue 9, September 2012, 1009-1015.

7- X. Song, L. Wang, S. Baek, & Y. Park, "Multidisciplinary optimization of a butterfly valve," ISA transactions, 48(3), 2009, 370-377.

8- S. Jeon, J. Yoon, & M. Shin, "Flow characteristics and performance evaluation of butterfly valves using numerical analysis," IOP Conference Series: Earth and Environmental Science (Vol. 12, No. 1, p. 012099). IOP Publishing, 2010, August.

9- C. Huang, & R. Kim, "Three-dimensional analysis of partially open butterfly valve flows," Journal of fluids engineering, 118(3), 1996.

10- K. Wojtkowiak and C. Oleskowicz-Popiel, "Investigation of butterfly Control valve Flow Characteristics," Foundations of Civil and Environmental Engineering, No. 7, pp-381, 2006.

11- Xue guan Song, Young Chul Park, "Numerical analysis of butterfly valve-prediction of flow coefficient and hydrodynamic torque coefficient," Proceedings of the World Congress on Engineering and Computer Science (WCECS), San Francisco, USA, October 2007, 24-26.

12- B. Prema, Sonal Bhojani, N. Gopalakrishnan, "Design optimization of butterfly valve using CFD," Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, India, December 2010, 16-18.

13- G. Ibrahim, Z. Al-Otaibi, H. Ahmad, "An investigation of butterfly valve flow characteristics using numerical technique," Journal of advanced science and engineering research, Vol.3, No.2, June 2013, 151-166.

14- B. Mohammadi and O. Pironneau, "Analysis of the k-Epsilon Turbulence Model," Research in Applied Mathematics", John Wiley& Sons Ltd (Import), August 1994.