Modeling of Check Valve Using Computational Fluid Dynamics

Aly M. EL-Zahaby[a], Gamal I. Sultan[b], Abdulkader S. Bekhatro[a], Ibrahim Ali[a]

ABSTRACT - When a check valve closes, its disc slams on the valve seat, this slam may cause many problems like noise, vibration and seat wear. The main reason of slam problem is the great hydrodynamic force acting on the valve disc near and at closing position. In order to reduce the impact force between the valve disc and its seat when closing, some modifications are applied to the valve disc geometry to reduce the drag force acting on it by the system moving back flow and consequently decrease the impact force.

The fluid flow across original and modified valve discs during closing mode are modeled using CFD package:"SOLIDWORKS Flow Simulation, SWFS" at different closing angles till closing position. The results of SWFS are validated by comparing with other CFD simulations, where SWFS results lies between the other results.

Comparison between the results of original and the modified valve disc models shows that the hydrodynamic force acting on the modified valve disc is reduced, thus the slam force between the disc and the seat is also reduced. The modified valve disc shows a good performance concerning valve flow coefficient at fully open position with reduction of force acting of valve disc near closing position by about 40%, when compared to the original valve disc.

Keywords - CFD, check valve, drag, slam.

1. INTRODUCTION

1.1. Check Valve Principle of Operation

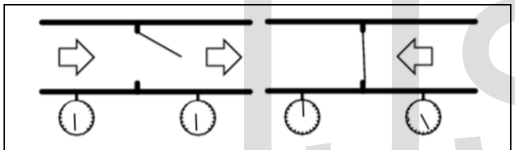


Fig. 1 Basic check valve function

Check valves are devices that allow fluid flow in one direction and prevent fluid flow in the opposite direction, by use of a moving plug (disk, wedge, plate, ball, etc) opening when forward flow begins in the pipeline, Fig. 1, [1]. A check valve is commonly used where reverse flow in a pipeline is not allowed. Reverse flow can occur after shutdown of a pump or closure of a control valve. The occurrence of reverse flow may cause damage to seals and brush gears of pumps and consequently drain tanks and reservoirs. Due to the physical properties of a check valve such as friction and inertia the valve will not close immediately and some reverse flow will still occur, [2].

1.2. Major Check Valve Types

The most common check valve designs are: swing, lift, tilting disc, and double disc designs, [3], Fig. 2.

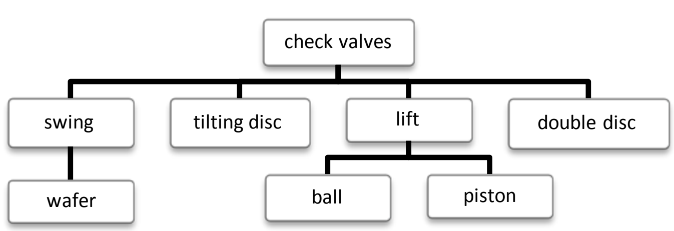


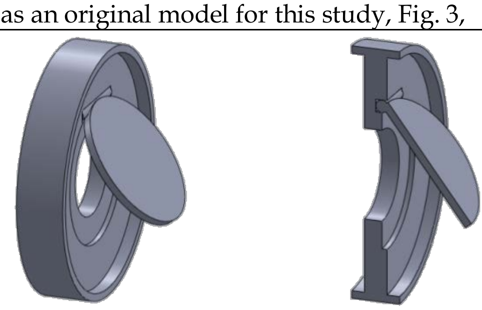
Fig. 2 Common check valve types

1.3. Check Valve Slam and Its Reduction

Simply, slam is the impact force between the valve disc and the valve seat due to rapid closure of the valve disc. Many things causes the valve disc to close rapidly such as the inertia of the valve disc, hydrodynamic force acting on the disc due to the back flow, and water hammer which forms a vacuum pressure zone causing a sudden large suction to the disc towards its seat. Valve slam can be overcome by using soft metal for valve disc seat, one of the water hammer reduction techniques or/and reducing the closing force by decreasing the hydrodynamic drag force acting on the valve disc during its closing, which is considered in this paper.

2. SCOPE OF THE WORK

The CFD modeling is applied on a wafer type swing check valve, so a 2" valve made by "JD Controls®", [4], is picked and drawn as an original model for this study, Fig. 3,



a) 3D view b) vertical cross section

Fig. 3 Original model of the wafer check valve

ISSN 2229-5518

The fluid flow across the valve disc during closing at different disc positions is modeled using CFD package:"SOLIDWORKS Flow Simulation, SWFS". The internal design geometry of the check valve disc will be modified in order to reduce drag on the moving disc and thus the force causes its slam.

Now the original model is drawn and imported into the CAD software, modifications that used for drag reduction in external flow problems are applied willing to reduce fluid force acting on the disc.

The common ways used in external flow problems to reduce drag over bodies such as rounding or chamfering sharp edges of the immersed body, adding frontal volumes to help in penetrating the flow and adding back trailers to reduce wakes formations are used.

3. MATHEMATICAL MODELING

The CFD software package "SWFS" uses the conservation laws of mass, angular momentum and energy conservation in the Cartesian coordinate system, [5], for solution of fluid flow across valve disc rotating about an axis passing through the coordinate system's origin. The software uses the mentioned equations to obtain the fluid flow velocity and pressure in the considered flow field across the valve disc during closing at different disc positions, Fig. 4.

At any angle θ°, the CFD software gives: torque and force acting on the valve disc. The solution starts with beginning of valve closure at θ_{max}= 75°, where both angular velocity and acceleration equal zero.

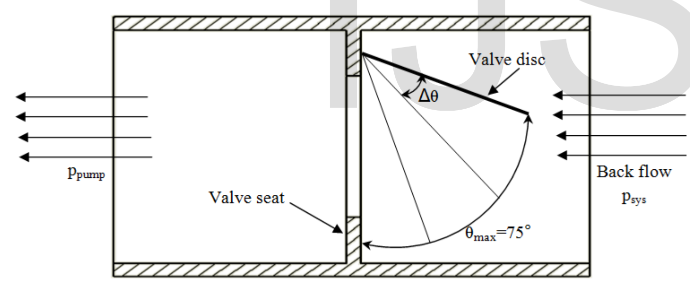


Fig. 4 Calculating scheme during valve closure at different disc positions

The obtained velocity and pressure are used to obtain force acting on the valve disc as follows:

$$\overline{F_{hyd}} = \overline{F_f} + \overline{F_p}$$

Where:

$$F_f = \int \tau_w . ds \qquad \text{, friction force}$$

$$F_p = \int p . ds \qquad \text{, pressure force}$$

And:

τ_w : Fluid's shear stress at the disc surface element of ds area.

p : Static pressure acting on the disc surface element of ds area.

Torque acting on the disc about its hinge is calculated using the relation:

$$T = (\oiint_{disc} (F_f + F_p) . dr) + (W \sin\theta \times L)$$

Where:

r : Arm of the hydrodynamic force

W: Apparent weight of the disc

L: Arm of the weight component normal to the disc

The angular acceleration of the disc is also evaluated as follows:

$$\alpha = \frac{T}{I}$$

Where:

α : Angular acceleration of the moving disc

T : Torque acting on the moving disc.

I : Moment of inertia of the disc

4. RESULTS AND DISCUSSIONS

4.1. Model Validation

Before starting simulation and obtaining results, validation of the "SOLIDWORKS Flow Simulation" (SWFS) CFD package is assured. A swing check valve is tested using the CFD package (SWFS), Fig. 5, under boundary condition of 1 [bar] outlet pressure and a flow velocity of 3 [m/s], where the torque acting on the valve disc is evaluated.

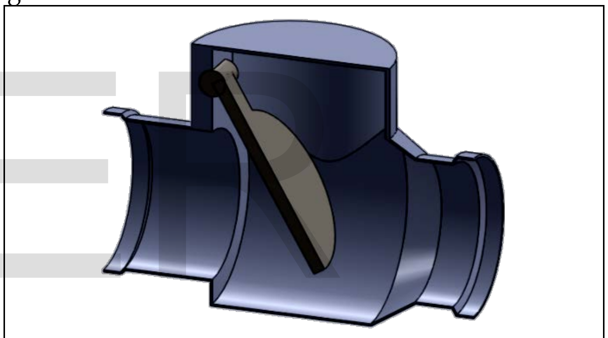


Fig. 5 Tested swing check valve by Martin Turesson

Martin Turesson, [6], investigated the same valve at the same operating conditions, using 1D hydraulic transients simulation code "RELAP5" and a CFD software "FLUENT". The obtained results of torque acting on the considered valve disc of both Martin Turesson and SWFS are presented in Fig. 6.

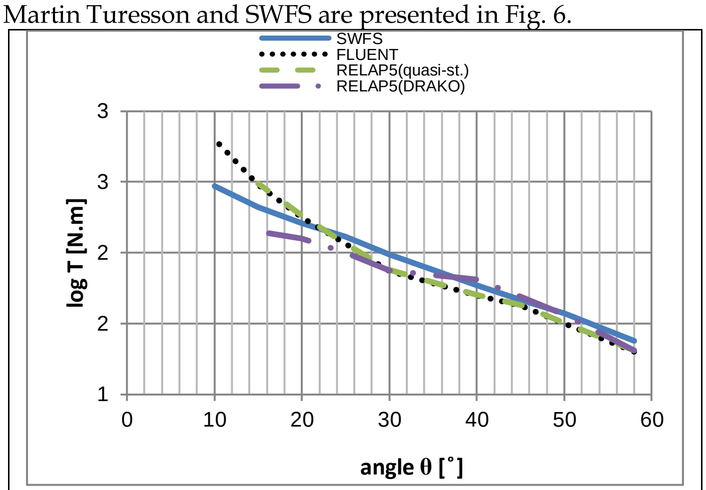


Fig. 6 Comparison between SWFS and results of Martin Turesson

The comparison of SWFS model with Martin Turesson models shows that the results of SWFS model lie between the results

---

- *Authors (a), Mechanical Power Engineering Department, Faculty of Engineering, Tanta University, Egypt.*
- *Authors (b), Mechanical Power Engineering Department, Faculty of Engineering, Mansoura University, Egypt.*
- *Corresponding author, E-mail: eng.ibrahim86@gmail.com*

ISSN 2229-5518

of Martin Turesson models. Therefore the results of SWFS model are validated and can be used successfully for evaluating the characteristics of the original and modified valves.

4.2. Original 2" Valve Disc

The CFD software "SOLIDWORKS Flow Simulation" is applied for static simulation of the basic configuration of the wafer type 2" swing check valve (original valve), Fig. 7,

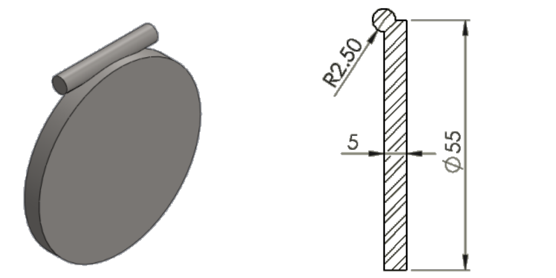


Fig. 7 Original valve disc model, dims. in [mm]

The 2" valve disc is tested during its closing travel under the following boundary conditions:
- Fluid flow: water with density ρ =998.2 [kg/m^3]
- Inlet pressure (system back pressure): p_{sys} = 6 [bar]
- Outlet/pump stoppage pressure: p_{pump}= 1 [bar]
- Ambient temperature T= 293.2 [K]
- Adiabatic wall thermal condition with zero surface roughness

Mesh and Mesh Size Selection:

The SWFS provides eight levels of mesh refining with the ability of adding dense local meshes in the critical areas of concern, [7], original and modified valve discs are tested at different disc angles with several mesh densities. Mesh test gives that the fifth level of mesh refining is enough for original and modified valve models almost for all disc positions where the deviation with the sixth level of refinement is about 0.07% to -0.03014%. For original valve disc at fifth level of mesh refining the number of cells varying from 21330 to 27879, Fig. 8. So the fifth level of mesh refining is enough for giving acceptable results.

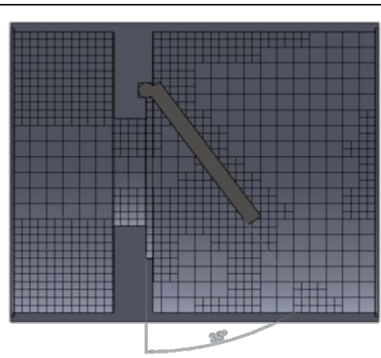


Fig. 8 Mesh size at fifth level of refinement of the original valve disc at θ=35°

The obtained results of forces and torques acting on the valve disc are shown in Fig. 9, and Fig. 10. Results show that the force on the disc increases with the decrease of the disc angle, since the projected area facing the flow increases and thus the drag on the disc increases as well.

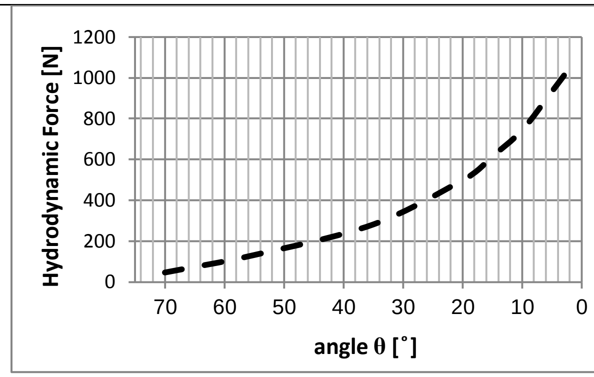


Fig. 9 Force acting on the disc at different angles

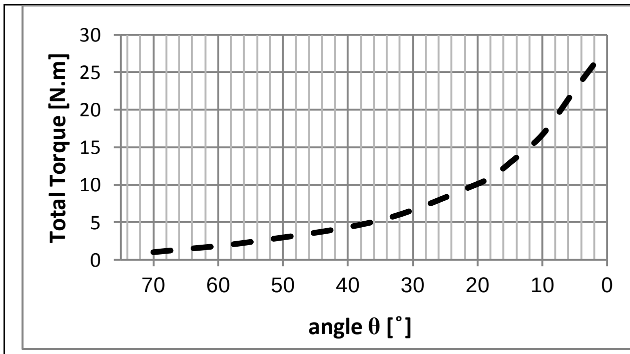


Fig. 10 Torque acting on the disc at different angles

4.3. Check Valve Suggested Modifications

The original valve disc geometry is modified in order to reduce the drag force acting on it so as to reduce slam force between the valve disc and its seat. This modified valve disc will is also tested using SWFS and results of force and torque acting on both disc models at several disc angles will be compared.

The valve disc modification is based on applying external flow drag reduction techniques to the valve disc with investigating their effect on the closing force/torque acting on it. The suggested modifications are tested at a high drag disc position, let it be at disc angle θ°=2°, where twenty nine modifications are applied.

Some of these modifications are promising and gave a good drag reduction but others are not, e.g. adding frontal 20 mm height conical volume as shown in Fig. 11, decreases the drag force by only 0.001% on the valve disc.

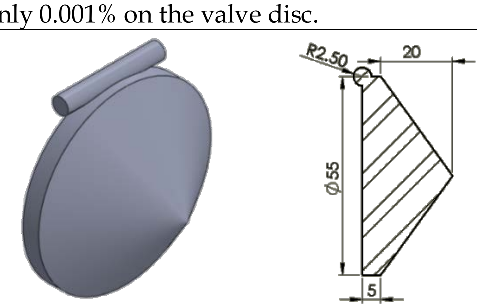


Fig. 11 Modified valve disc with frontal 20 mm height conical volume

On the other hand adding back fairing with 36 mm diameter, 30 mm height, as shown in Fig.12, decreases the drag force on the valve disc by 44.21%.

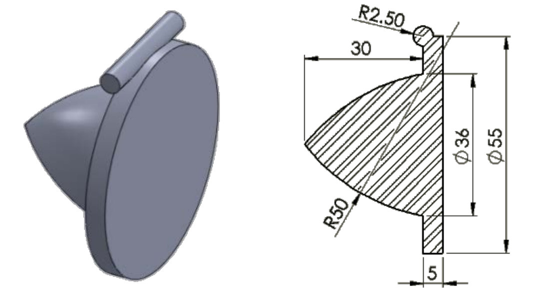


Fig. 12 Modified valve disc
with 36 mm diameter, 30 mm height back fairing

The final modified valve disc, after considering all effective modifications: back fairing, front chamfer and back fillet, is shown in Fig. 13, with 44.4% drag reduction.

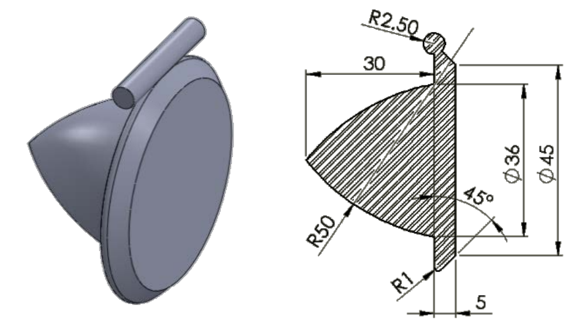


Fig. 13 Final modified valve disc, dims. in [mm]

4.4. The Modified Valve Disc

The fluid flow across the modified valve disc is also modeled using SWFS package at the same boundary conditions used for the original valve disc. The resulted force and torque acting on the final modified valve disc at different angles are presented in Fig. 14 and Fig. 15, respectively.

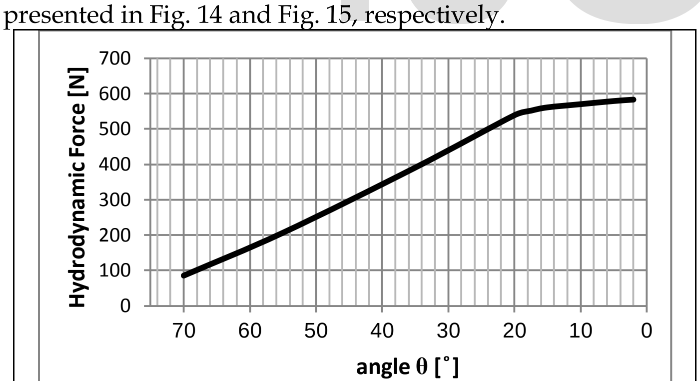


Fig. 14 Force acting on the modified valve disc at different positions

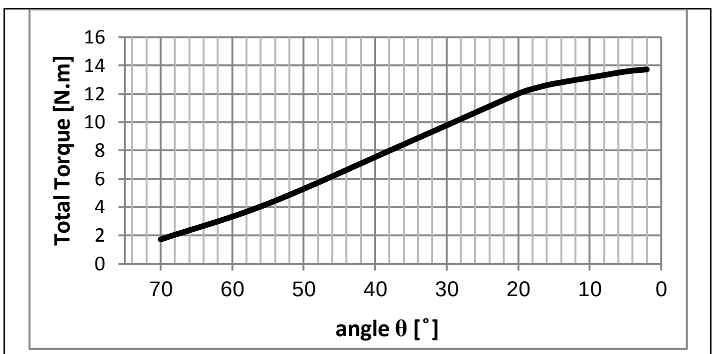


Fig. 15 Torque acting on the modified valve disc at different positions

4.5. Comparison Between Original and Modified Valve Discs

Figure 16 shows the variation of force values along disc travel for both original and modified valve discs, while Fig. 17, presents the difference between force values at specific disc angles.

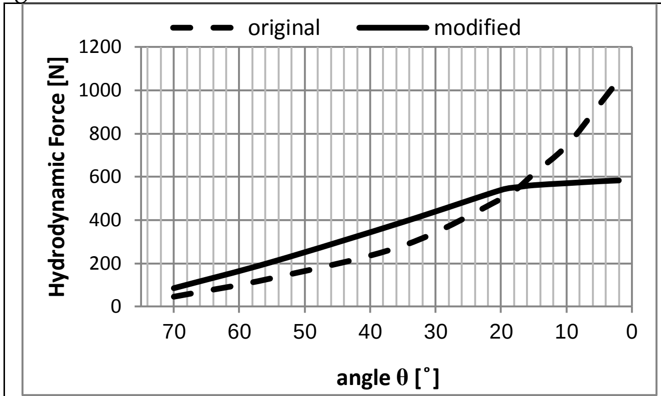


Fig. 16 Variation of force values along disc travel
for both original and modified valve disc

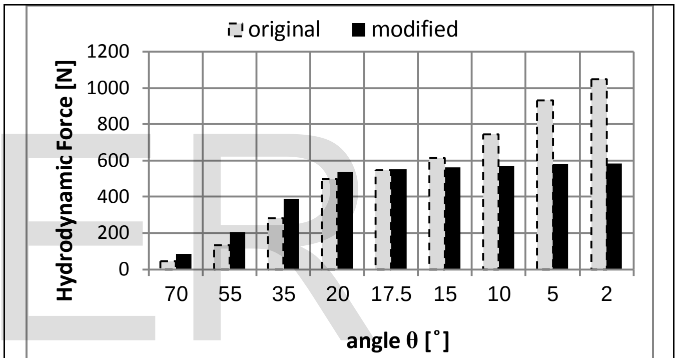


Fig. 17 Difference between force values at specific disc positions

Figure 18 shows the variation of torque values along disc travel for both original and modified valve discs, while Fig. 19 presents the difference between torque values at specific disc angles.

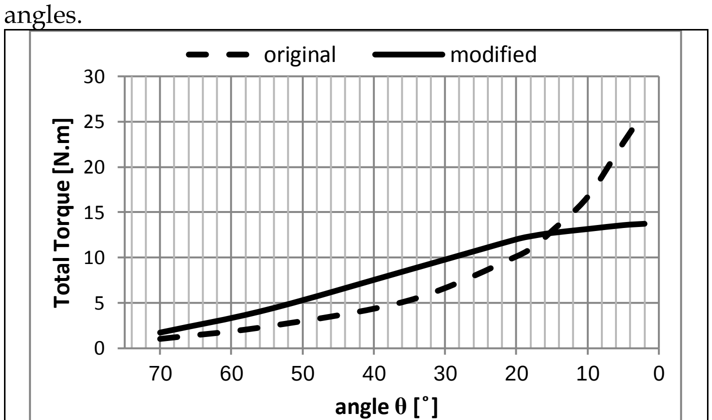


Fig. 18 Variation of torque values along disc travel
for both original and modified valve disc

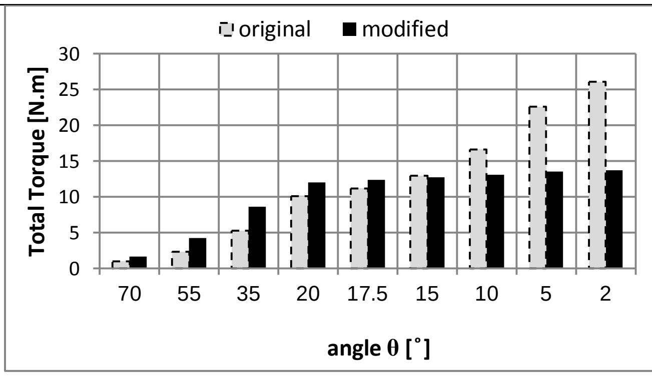


Fig. 19 Difference between torque values at specific disc positions

As pre-illuminating conclusion, the modified model performance is better than the original one, especially in the last 20 degrees before the disc reaches the seat, where force and torque in the disc are getting lower, which means that the slam of the disc on its seat is reduced.

The percentage change of force and torque along the disc travel due to the applied disc modification are shown in Fig. 20, and Fig. 21, for force and torque respectively.

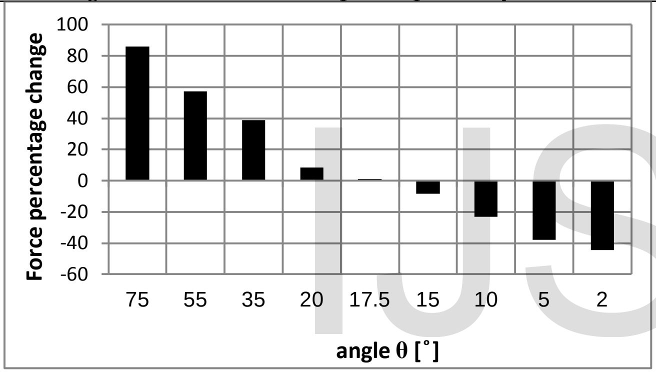


Fig. 20 Percentage change in force at different disc positions.

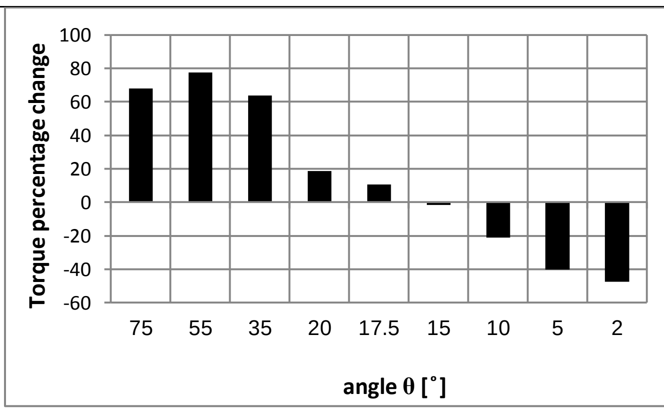


Fig. 21 Percentage change in torque at different disc positions.

4.6. Valve Flow Coefficient ,K_v

Valve flow coefficient is a constant related to the geometry of a valve, for a given travel, that can be used to establish flow capacity, [8]. It is a common way for valve selection and comparing between valves.

The flow coefficient for check valves is measured with the valve fully open, [9, 10] because if the disc is not be fully open it might be unstable where disc instability may cause valve wear, requiring increased maintenance or even failure of the check valve. The larger the K_v, the larger the flow at a given pressure differential and the more efficient is the valve.

For metric units, the flow coefficient K_v is the volume flow rate in cubic meter per hour at temperature of 20° that will pass through a fully opened valve with a pressure difference of 1[bar] across it, K_v can be calculated as follow, [11]:

$$K_{vv} = V^{\circ} \sqrt{\frac{SG}{\Delta p}}$$

Where,
V° : Volume flow rate [m^3/h]
SG : Specific gravity of the fluid [-]
Δp : Pressure difference across the valve [bar]

Calculating Kv for both original and modified valve disc at Δp=1[bar] gives results presented in Fig.22:

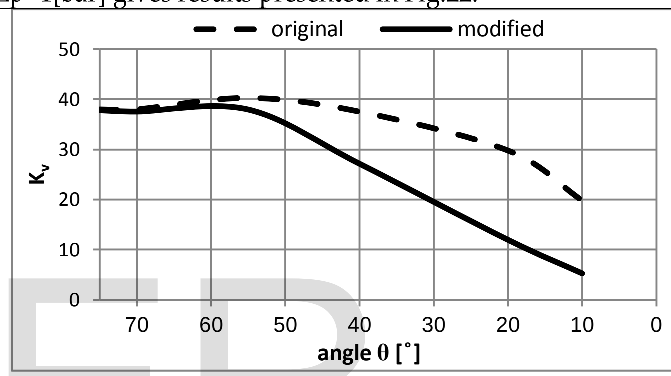


Fig. 22 Flow coefficient values along valve disc travel

Valve flow coefficient K_v slightly decreased by 0.44% at θ= 75°, the operating position where the valve is fully open.

Moreover, it can be noticed from Fig. 22, when the valve disc is in the transient position, the K_v for the modified disc is less than the original one, which means the volume flow rate passing through it would be smaller, which is also an advantage.

4.7. Valve Disc Angular Acceleration

The valve disc angular accelerations during closing of both original and modified valve discs are represented in Fig. 23.

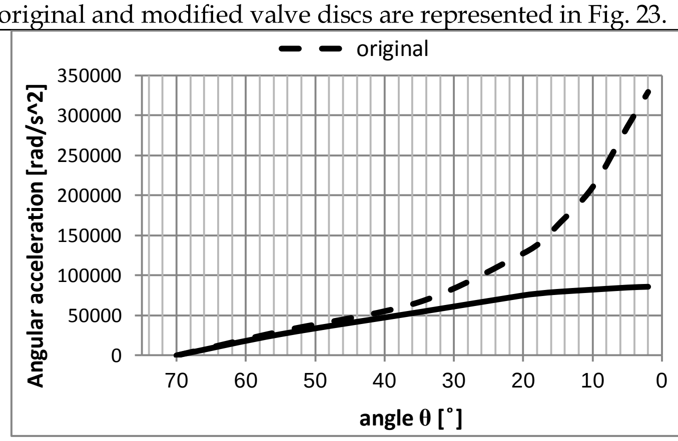


Fig. 23 Angular acceleration for both original and modified valve discs during closing

Figure 23 shows that the angular acceleration of the modified valve disc increases with smaller rate compared to the original valve disc which reduces the slam effect.

4.8. Higher Pressure Difference Case

The same simulation tests are applied to both original and modified valve disc models but at higher pressure difference of 12[bar] to check the efficiency of such modification at higher pressure difference. Figures 24 and 25 show the percentage change of force and torque due to valve disc modification for low and high pressure difference at different disc positions. The obtained results show that the performance of the valve is almost the same, no matter the operating pressure difference.

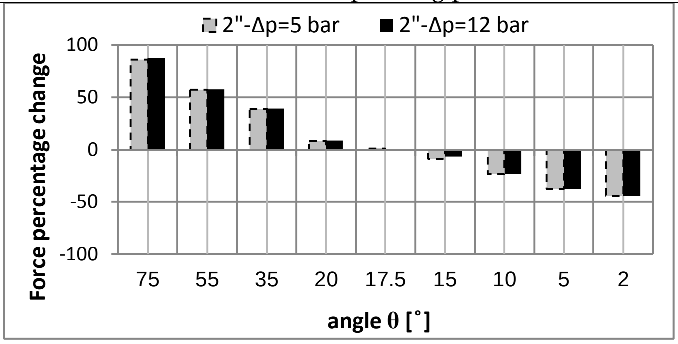


Fig. 24 Percentage change in force due to valve disc modifications for low and high pressure difference at different disc positions

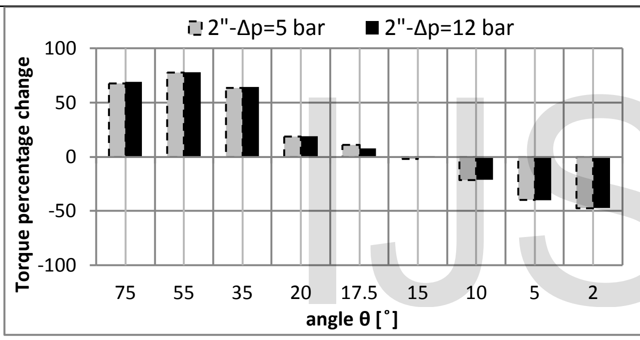


Fig. 25 Percentage change in torque due to valve disc modifications for low and high pressure difference at different disc positions

4.9. Larger Scale Case

Other tests are done to apply the modification on larger scale valves of 4" diameter. Figures 26 and 27, show slight percentage decrease in force and torque at disc angles near closing position, which enhances modification efficiency for large scale valves.

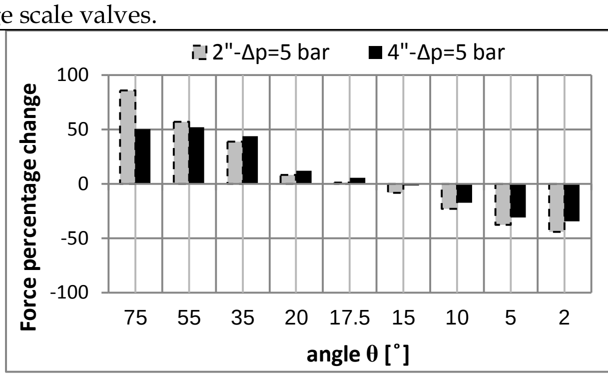


Fig. 26 Percentage change in torque due to valve disc modifications for both small and large scale cases at different disc positions

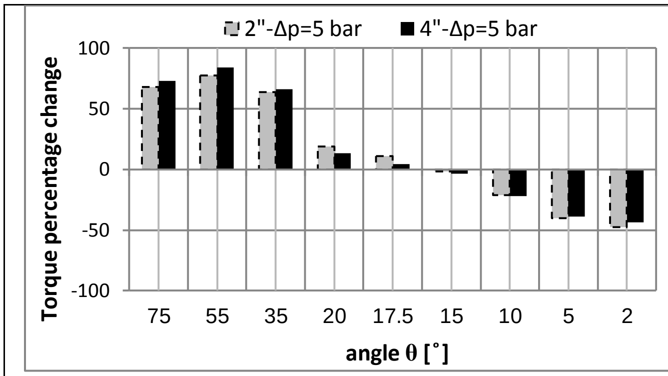


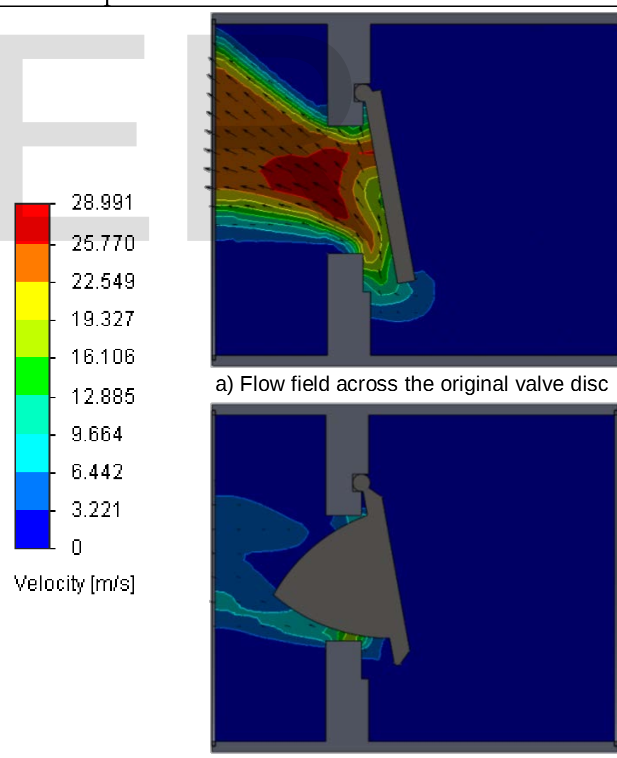
Fig. 27 Percentage change in torque due to valve disc modifications for both small and large scale cases at different disc positions

4.10. Flow Visualization

One of the advantages of CFD software that it can visualize the fluid flow across the tested valve, from which the performance improvement of the modified valve disc compared with the original one at small disc angles (let it be θ=10°) can be detected.

4.1.1. Velocity visualization

Figure 28 shows the velocity contours, isolines and vectors in meridonal plane for the fluid flow across the valve discs.



a) Flow field across the original valve disc

b) Flow field across the modified valve disc

Fig. 28 Velocity visualization across original and modified valve discs

Figure 28 a, shows a high velocity zone behind the original valve model (red area) which is not exist in the modified valve disc model. According to Bernoulli's equation that high velocity zone has a low pressure that will create suction region for the original valve disc towards the closing direction causing valve disc slam.

4.1.2. Pressure visualization

Different views of pressure visualization around original and modified valve disc models are shown in Fig. 29.

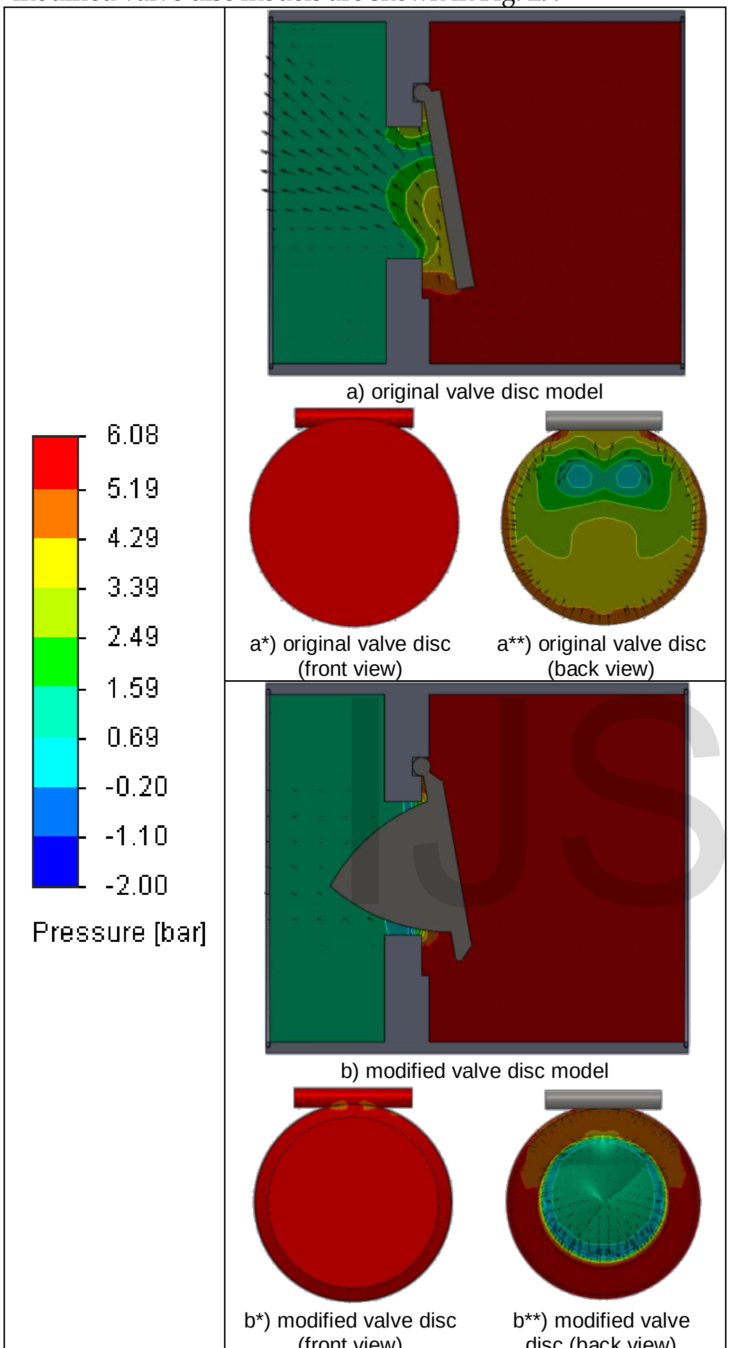


Fig. 29 Pressure flow visualization.

From Fig. 29 above, one can notice that pressure on frontal face of both original and modified discs is almost the same and appears in the same red color degree, but on the back face of the original valve disc, Fig. 29 a**, the pressure value goes down (less red areas and more yellow, green and even blue) while the modified valve disc back face, Fig. 29 b**, still has large pressure area (red annular area) than the original valve disc, hence the pressure difference across the modified valve disc is smaller than across the original one. The obtained results mean that the modified valve disc will suffer less suction towards closing direction, thus less closing force on it.

4.1.3. Vorticity visualization

Different views of vorticity visualization around original and modified valve disc models are shown in Fig. 30.

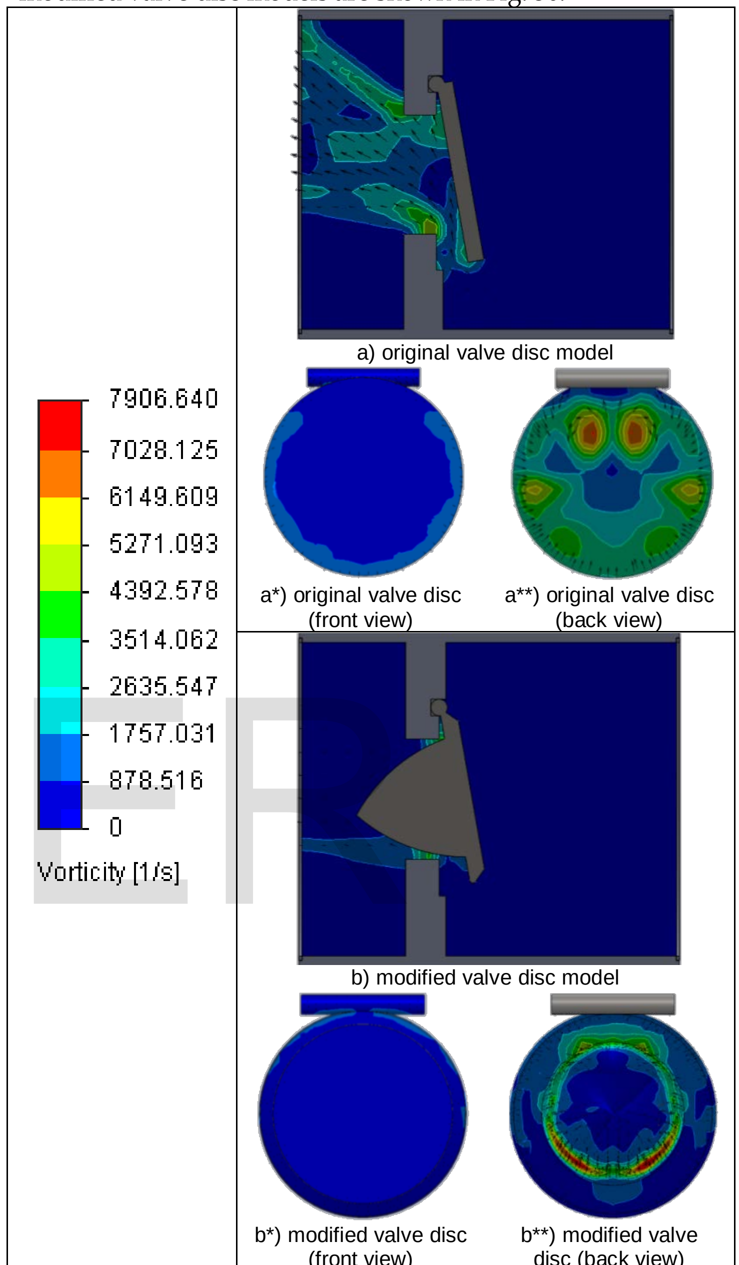


Fig. 30 Vorticity visualization.

Figure 30 a** shows two vortices formed on the back face of the original valve disc. Figure 30 b** shows two vortices formed on the back fairing of the modified valve disc and not on its back face. Less vorticity on the back face means less pressure losses and less pressure drop, that means also less pressure difference across both sides of the modified valve disc than the original one, which means that the modified valve disc will suffer less suction towards closing direction, thus less closing force on it.

5. CONCLUSIONS AND FUTURE WORK

The CFD package "SOLIDWORKS Flow Simulation, SWFS" is used efficiently to model the fluid flow through wafer type check valve during closing mode. The obtained results are validated by comparing with results of another CFD techniques, "FLUENT" and "RELAP5" simulation code, where good agreement exists.

Some of the external flow drag reduction techniques are useful in reducing forces on immersed bodies in internal flow devices like adding fairing or trailers and smoothing or filleting sharp edges as proved in the modified disc.

The modified valve disc gives good slam performance, as:

- The force acting on the valve disc decreases by about 40% at disc angles below 10°, i.e. just before the valve disc hits its seat and this is a modification advantage.
- Also the modified valve disc shows a good performance concerning valve flow coefficient at normal operating conditions when the disc is fully opened at 75° where the flow coefficient for the modified valve disc is almost the same as the original valve disc (0.44 % lower).
- Flow visualization abilities in CFD shows less wakes formation behind the modified valve disc than the original one, which permits smoother valve disc closing.

Another significant conclusion about this geometrical modification is that applying higher pressure difference (Δp = 12 bar instead of 5 bar) and larger valve scale (4" instead of 2") gives almost the same performance efficiency near closing portion.

Although the "static simulation" used in this research gives a good indication about the performance of the original and the modified check valves, but future work should be given in studying the dynamic performance of the valve disc using CFD system coupling techniques to get more information about disc kinematics.

REFRENCES

[1] Jansson L. and Lovmark J., "An Investigation of the Dynamic Characteristics of a Tilting Disc Check Valve Using CFD Analyses", Master thesis, Chalmers University of technology, Sweden, 2013.

[2] Rahmeyer W., "Dynamic Flow Testing of Check Valves", Nuclear Industry Check Valve Group Winter meeting, St. Petersburg, Florida, 1996.

[3] Xu H., Guang Z. M., Qi Y.Y.," Hydrodynamic Characterization and Optimization of Contra-Push check Valve by Numerical Simulation", Journal of Elsevier Ltd., Annals of Nuclear Energy, 2011.

[4] J.D. Controls Manufacturing, Products brochure, 2011.

[5] SOLIDWORKS Corporation, "SOLIDWORKS Flow Simulation 2012 Technical Reference", Technical paper, 2012.

[6] Turesson M., "Dynamic Simulation of Check Valve Using CFD and Evaluation of Check Valve Model in RELAP5 ", Master thesis, Chalmers University of technology, 2011.

[7] SOLIDWORKS Corporation," Advanced Boundary Cartesian Meshing Technology in SOLIDWORKS Flow Simulation", Technical paper, 2013.

[8] "Control Valve Handbook" 3rd edition, Fisher Controls International, Inc, 2001.

[9] "Design and Selection Criteria of Check Valves", Val-Matic Valve and Manufacturing Corp., 2011.

[10] "Masoneilan Control Valve Sizing Handbook", Masoneilan dresser, valve division, 2001.

[11] "About Flow Coefficient", FNW Corporation, 2012.

NOMENCLATURE

A	Area [m^2]
C_d	Drag coefficient
CAD	Computer aided drawings
CFD	Computational Fluid Dynamics
F	Force [N]
m	Mass [kg]
p	Pressure [N/m^2]
T	Torque [N.m]
V	Volume [m^3]
W	Weight [N]
ρ	Density [kg/m^3]
θ	Angle [°]
α	Angular acceleration [rad/s^2]