Analysis of One-Way and Two-Way FSI Approaches to Characterise the Flow Regime and the Mechanical Behaviour during Closing Manoeuvring Operation of a Butterfly Valve

M. Ezkurra, J. A. Esnaola, M. Martinez-Agirre, U. Etxeberria, U. Lertxundi, L. Colomo, M. Begiristain, I. Zurutuza

Abstract**—**Butterfly valves are widely used industrial piping components as on-off and flow controlling devices. The main challenge in the design process of this type of valves is the correct dimensioning to ensure proper mechanical performance as well as to minimise flow losses that affect the efficiency of the system. Butterfly valves are typically dimensioned in a closed position based on mechanical approaches considering uniform hydrostatic pressure, whereas the flow losses are analysed by means of CFD simulations. The main limitation of these approaches is that they do not consider either the influence of the dynamics of the manoeuvring stage or coupled phenomena. Recent works have included the influence of the flow on the mechanical behaviour for different opening angles by means of one-way FSI approach. However, these works consider steady-state flow for the selected angles, not capturing the effect of the transient flow evolution during the manoeuvring stage. Two-way FSI modelling approach could allow overcoming such limitations providing more accurate results. Nevertheless, the use of this technique is limited due to the increase in the computational cost. In the present work, the applicability of FSI one-way and two-way approaches is evaluated for the analysis of butterfly valves, showing that not considering fluid-structure coupling involves not capturing the most critical situation for the valve disc.

Keywords**—**Butterfly valves, fluid-structure interaction, one-way approach, two-way approach.

I. Introduction

BUTTERFLY valves are widely used for both shutting off and throttling the fluid flow in a wide range of industrial applications such as gas, oil or water transportation, air admission in combustion engines or assisting blood circulation in artificial hearts, among others [1]–[3]. They have a simple structure, which consists of the disc, the shaft and the valve body [2], [4]. Their main advantage is that they can quickly bring the valve from the closed to the fully open position. Moreover, when they are fully open the pressure drop is very low [5], [6].

The failure of these components could be very dramatic as it may cause natural disasters, very costly breakdowns of the systems where they are integrated or even the loss of human lives [7], [8]. Therefore, it is essential to ensure the structural integrity of the valve components. In this regard, numerical simulation techniques have enabled the prediction of the behaviour of the fluid and the structural components during opening and closing operations.

The pressure drop in a pipeline through a butterfly valve changes during the closing manoeuvre, as it is shown in Fig. 1.

The maximum pressure drop is given at the fully closed position, and therefore, valves are typically designed to resist such condition where pressure is assumed to be constant [9]. However, considering the fluid and the structure interaction phenomena could provide further information about the system behaviour. The pressure distribution is not uniform on the disc surface as it rotates during operation and negative gauge pressure values may appear at the rear [10], [11], leading to uneven deflection of the disc. This behaviour is hard to predict without performing Computational Fluid Dynamics (CFD) analyses.

During 1970s and up to late 1980s, various authors carried out structural simulations of butterfly valves to dimension shaft-disc-bearings assembly [12]. These authors considered uniform pressure distribution over the disc surface in 2D and 3D models. In the early 1990s, CFD made possible to determine accurately the flow characteristics in butterfly valves depending on the disc position. Initial 2D models [13] resulted on reasonable preliminary results, and subsequent 3D models provided more accurate flow analysis [4]. These works were conducted with fixed disc positions, which involve steady-state analyses rather than transient. Even nowadays some authors work with 3D steady-state CFD models [5], [14]. In 2010, in order to obtain results that take into account dynamic effects, moving grids were implemented to accomplish transient analyses [2]. In addition, most researchers have studied the fluid field by means of CFD but with no analysis of the structure behaviour [6]. Nevertheless, fluid pressure may have great effect on the valve stress/strain distribution [9]. In this sense, the interaction between the fluid and the structure may be considered by means of one-way or two-way coupling approaches. In the one-way approach, pressures obtained from the CFD analysis are transferred to the structural model. In the two-way approach, not only fluid pressures are transferred to the structure, but also the fluid domain is updated as a result of the induced structural deformations. In the late 2000s, different authors performed

M. Ezkurra, J. A. Esnaola, M. Martinez-Agirre, U. Etxeberria and U. Lertxundi are with the Faculty of Engineering, Mondragon Unibertsitatea, Mondragon 20500 Spain (phone: +34 943794700; fax: +34 943791536; e-mail: mezkurra@mondragon.edu).

L. Colomo, M. Begiristain and I. Zurutuza are with Ampo Poyam Valves, Idiazabal 20213 Spain (phone: +34 943188000; fax: +34 943188130).

one-way fluid-structure interaction (FSI) simulations applied to butterfly valves with the aim of ensuring valve structural integrity or obtaining its optimum dimensions [6], [9]. Regarding two-way approaches, few applications are found related to valves [15], [16].

In this paper, classical structural analysis, one-way FSI approach and two-way FSI approach are analysed to dimension butterfly valves, in terms of flow characteristics and structural behaviour.

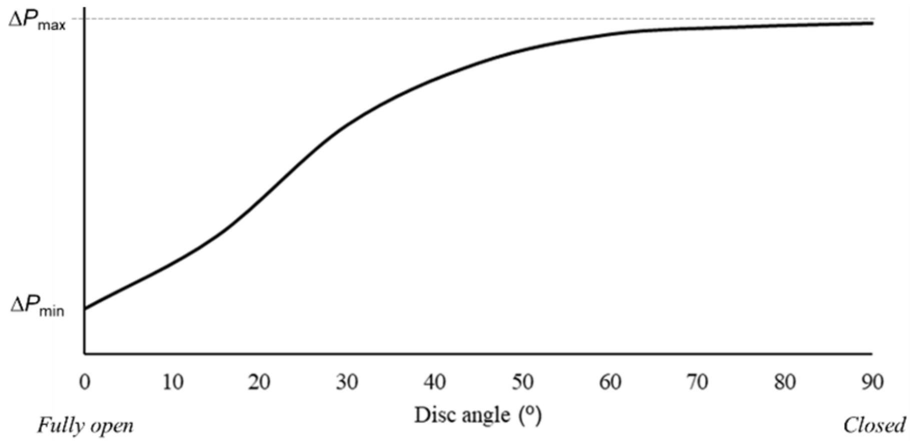

Fig. 1 Typical pressure drop evolution during butterfly valve closing

II. Fluid-Structure Interaction

FSI is the mutual interaction between a deformable structure and an internal or surrounding fluid flow. The fluid exerts pressure loads causing the structure to deform. At the same time, the fluid geometric domain is updated considering the structural deformations. FSI industrial applications can be found in automotive and aeronautical sectors (door seals, wings), biomechanics (design of heart valves), constructions (wind loading of structures), etc. [1], [15], [16]. FSI simulations can be classified as one-way coupled or two-way coupled as shown in Fig. 2.

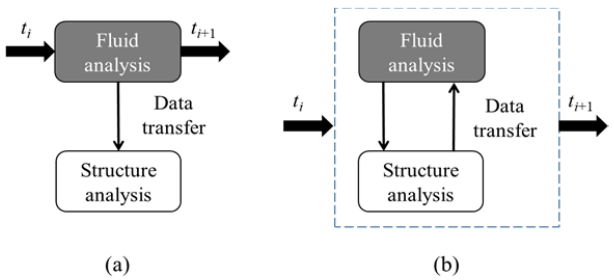

Fig. 2 (a) One-way and (b) two-way coupled FSI approaches

In one-way coupling, it is considered that fluid domain is hardly affected by the resulting small structural deformations. This allows CFD and structural analysis to be solved independently with unidirectional data transfer: only fluid pressure is transferred from CFD to structural domain. In two-way coupling, structural deformation due to fluid pressure affects the flow field and, therefore, fluid and structural domains must be solved simultaneously with bidirectional data transfer. Pressure is exported from CFD to structural analysis, and deformation is transferred from structural to CFD analysis to update the geometry of the fluid domain every coupling iteration, until both solutions converge [16], [17].

III. Computational Analysis Setup

A. Case Study

In order to compare different modelling approaches to dimension butterfly valves, a generic case study is selected. The simplified geometry consists of a 42 mm inner diameter pipeline and a disc of 40 mm diameter and 1.5 mm thickness. For the analysis, a closing operation of the butterfly valve is simulated, where the disc angle θ with respect to the horizontal axis varies from 0º (fully open) to 90º (fully closed). A relatively high closing velocity is considered in order to reveal the differences between transient and steady-state solutions. For that purpose, an operation time of 1.5 seconds is set, which results in an angular velocity of 1.05 rad/s.

B. Geometrical Model

The selected case study is modelled considering a half-symmetric geometry. Upstream and downstream pipe lengths of 3 and 15 times the pipe inner diameter are defined [4], respectively, as shown in Fig. 3.

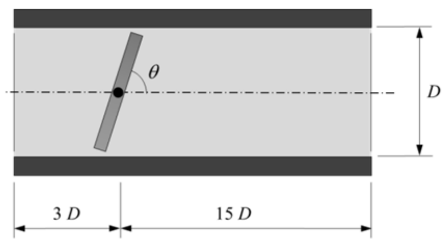

Fig. 3 Geometrical model of the butterfly valve

C. Considered Simulation Approaches

Different models are considered to dimension the disc of the butterfly valve. Specifically, the performed calculations are:

1) **Classical structural approach:** total pressure drop along the pipeline can be monitored by using manometers located upstream and downstream far enough from the disc (Fig. 1). Then, the disc is dimensioned for the maximum pressure drop which is identified for the fully closed position. The corresponding pressure drop value is applied as a hydrostatic uniform pressure over the front surface of the disc.
2) **One-way FSI coupling with steady-state CFD analysis results:** First, steady-state fluid simulations are performed for every 15º fixed disc positions during closing operation (15º, 30º, 45º, 60º, 75º and 90º). Then, the calculated pressures for each position are transferred to the static structural disc model.
3) **One-way FSI coupling with transient CFD analysis results:** First, a transient CFD simulation of the valve closing operation is carried out. Then, the corresponding pressures for 15º, 30º, 45º, 60º, 75º and 90º disc positions are transferred to the static structural disc model.
4) **Two-way FSI coupling:** Steady-state simulations are performed for 15º, 30º, 45º, 60º, 75º and 90º disc positions, simultaneously solving the fluid and the structural domains.

For all these approaches, the CFD and structural models are defined in the same way. The results of FSI approaches, which are closer to reality because they consider fluid and structural domain coupling, are compared to classical structural approach.

D. Fluid Analysis Model

CFD analyses are carried out in FLUENT v18.2 software. Liquid water is considered as the working fluid with a gauge pressure of 5 bar at the inlet and atmospheric pressure at the outlet. No-slip condition is selected for the walls.

To accomplish the transient CFD analysis a moving grid region is created, which consists of a 41 mm diameter sphere that surrounds the disc and rotates around the rotation axis (see Fig. 4). In steady-state calculations, the rotation of the moving grid is supressed.

After performing a mesh sensitivity analysis, an element sizing of 0.35 mm is established on the periphery of the disc and the sphere, with a maximum element size of 3 mm. In addition, inflation layers are defined both for the valve body and disc surfaces, achieving wall $y+$ values close to 1 as demanded by the selected SST k-ω turbulence model. As a result, the final model presents a total of 698,574 nodes and 2,875,021 elements. In the two-way coupling approach remeshing and smoothing algorithms are activated to update the CFD mesh due to the disc structural deformation.

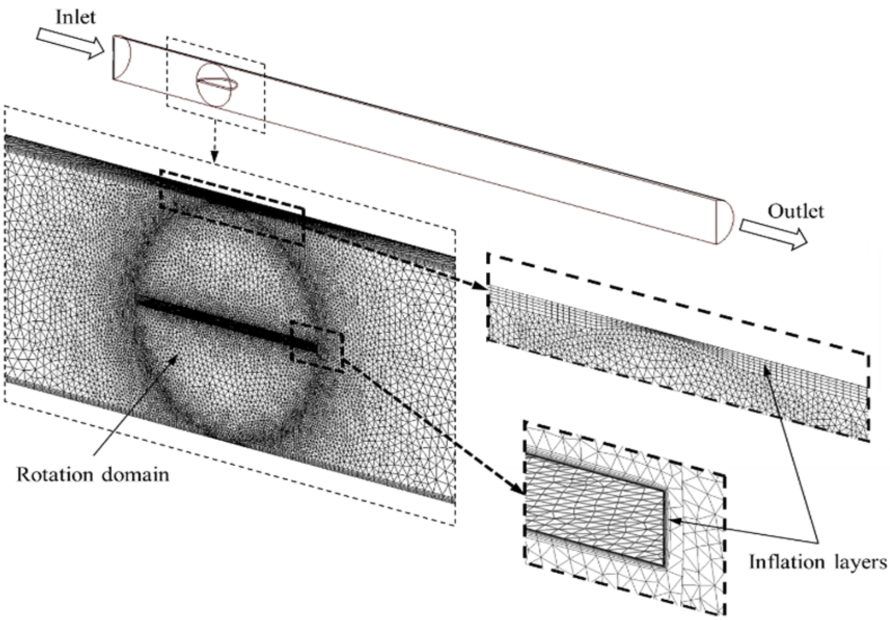

Fig. 4 Computational CFD model

E. Structural Analysis Model

Structural analyses are carried out in ANSYS Mechanical software. The disc of the butterfly valve is the unique component modelled for the structural analyses. A global mesh size of 0.35 mm is chosen, which leads to 142,438 nodes and 30,445 elements. Elastic aluminium alloy material properties are assigned to the disc, being the Young's modulus E = 70 GPa and the Poisson's ratio ν = 0.3. A small surface is defined to apply the boundary conditions that mimic the valve shaft constraints (Fig. 5). This implies to fix the disc in the corresponding angle, for a consistent pressure load transfer.

Nodes A and B are denoted for the analysis of results. They are both located at the midsurface of the disc in the symmetry plane. Node A is the one closest to the pipe inlet, and moves down as the disc rotates. Node B, conversely, is the one that rises as valve is closed.

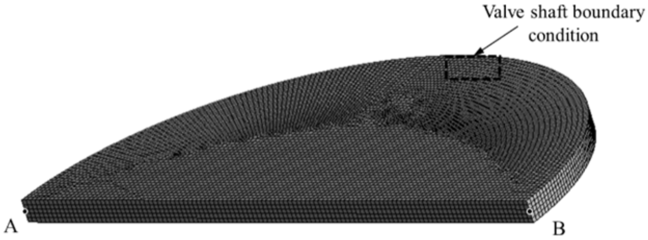

Fig. 5 Structural disc model

F. System Solving Procedure

FSI simulations are carried out by coupling CFD and structural simulations in ANSYS Workbench 18.2. The particular solving procedures related to each of the considered approaches are the following:

1) **Classical structural approach:** A static structural calculation is carried out just for 90º disc position.
2) **One-way FSI coupling with steady-state CFD analysis results:** CFD steady-state solutions are obtained for fixed disc positions between 15º and 90º, every 15º. The magnitudes of fluid pressure on disc surfaces are imported and applied over the ones in the structural domain. Static structural calculations are carried out, where the disc orientation corresponds to the one modelled in the CFD domain.
3) **One-way FSI coupling with transient CFD analysis results:** A CFD transient calculation is performed to simulate the 1.5 second closing manoeuvre. A time-step of 0.01 seconds is defined, which leads to 150 time-steps. A total of 200 iterations per time-step are established. Solutions every 15º of rotation are saved in order to transfer the corresponding pressure values to the static structural models of the disc, in the same way as in the second approach.
4) **Two-way FSI coupling:** Both CFD and structural solvers are coupled and synchronised to get converged solutions. Pressures are transferred from CFD to structural solver, and displacements from structural solver to CFD every coupling iteration. Pressure data transfer is ramped over 5 coupling iterations for a better convergence. A total of 60 coupling iterations are set. After performing the calculations, it is ensured that both domains and data transfers converge.

G. Post-Processing

1. CFD Domain Results

First, a qualitative analysis of the pressure and velocity fields is performed by monitoring their respective contour plots. Then, to quantify the effect of pressure in each approach, the resultant force perpendicular to the disc surface and its misalignment with respect to the rotation axis are determined. On the one hand, the resultant normal force (F_n) is presented normalised with respect to the one calculated with the classical structural approach (F_{n_str}), applying the pressure drop between upstream and downstream the pipeline for each disc position in order to be comparable. On the other hand, the misalignment as well as the consequent resultant moment, which are not considered in the classical structural approach, are also plotted along the closing operation.

Regarding velocity results, the valve flow coefficient C_V is determined, which is given by the flow capacity of the valve corresponding to a unit pressure drop at a certain opening position:

$$C_V = \frac{Q}{D^2\sqrt{\Delta P\,\rho}} \quad (1)$$

where Q is the mass flow rate in kg/s, ΔP is the total pressure drop in Pa, ρ is the density of the fluid in kg/m^3, and D is the valve diameter in metres. Therefore, C_V is a dimensionless magnitude. The obtained results are normalised with respect to the C_V achieved with the traditionally used steady-state CFD calculations, as flow coefficient cannot be calculated in the classical structural approach.

2. Structural Domain Results

Structural results are focused on valve disc deflection, defined as the perpendicular displacement with respect to the non-deformed disc geometry. Maximum deflection values are expected in the locations which are furthest from the disc rotation axis. Therefore, the deflections at nodes A and B (denoted in Fig. 5) are obtained at the specified disc positions along the valve closing operation. The deviations of the results at nodes A and B are calculated as the relative error with respect to the deflection achieved in classical structural approach, which considers maximum pressure drop at 90º disc position.

IV. Results and Discussion

A. Fluid Field Analysis Results

In our particular case study, pressure and flow velocity magnitudes are similar for all the considered approaches, as it can be seen in Fig. 6. However, there are appreciable differences on the contour distribution, which are more relevant in the interval between 15º and 45º. Additionally, the deflection of the disc can be observed in the fluid domain plots corresponding to the two-way coupled simulations, which has an impact on the fluid behaviour around the disc.

Fig. 7 shows (a) the resultant normal force on the disc and (b) the normalised normal force. First, it is observed that normal force on the disc surface is monotonically increasing all along the closing operation. Consequently, the force analysis could lead to identify the 90º case as the critical instant. The force magnitudes predicted by all the approaches are of similar magnitude. However, when normalising with respect to the classical mechanical approach (Fig. 7 (b)) differences among the studied methods are identified. It can be seen that for the one-way FSI approaches resultant normal force values on the disc are lower than the ones predicted by the classical structural approach. The differences are more notorious for small angles. On the other hand, the two-way FSI approach has the highest normal force at 15º and it is higher than the one-way approaches at 30º. However, it leads to the lowest normal force values in the rest of the disc positions.

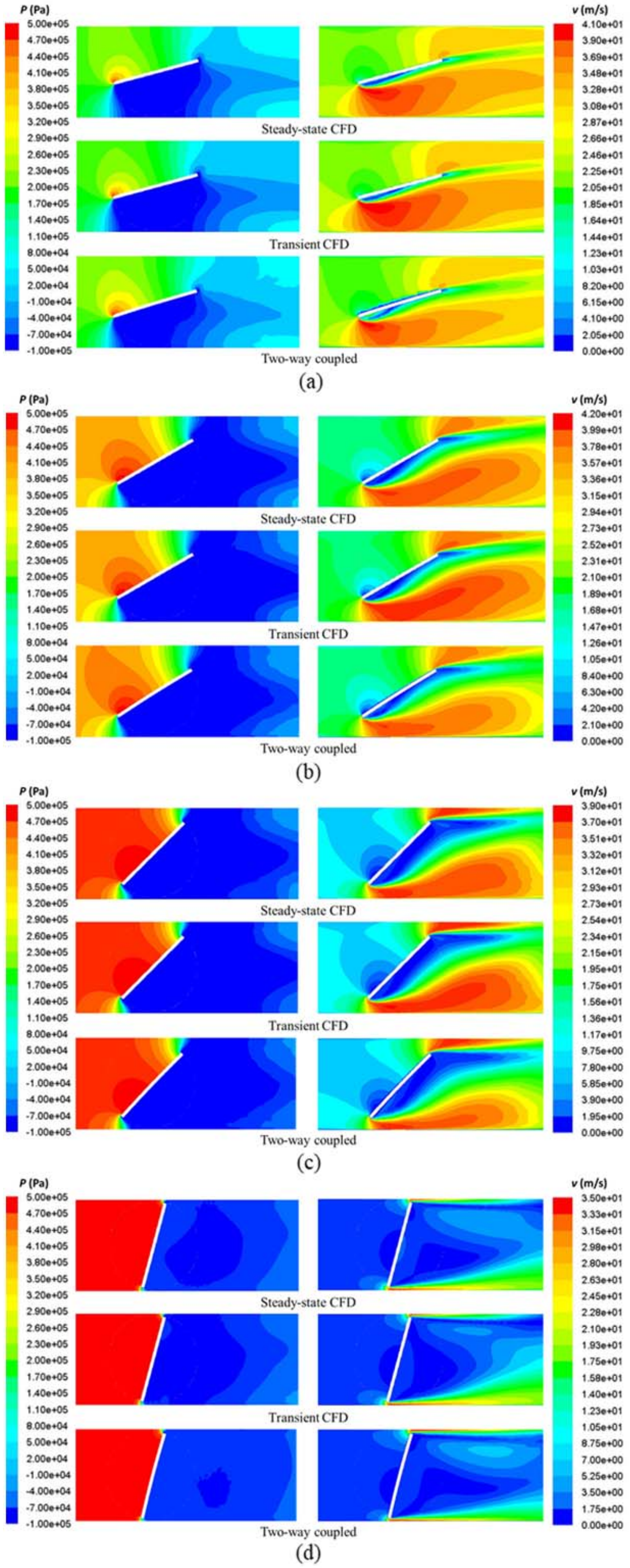

Fig. 6 Pressure and velocity contours for (a) $\theta = 15º$, (b) $\theta = 30º$, (c) $\theta = 45º$ and (d) $\theta = 75º$

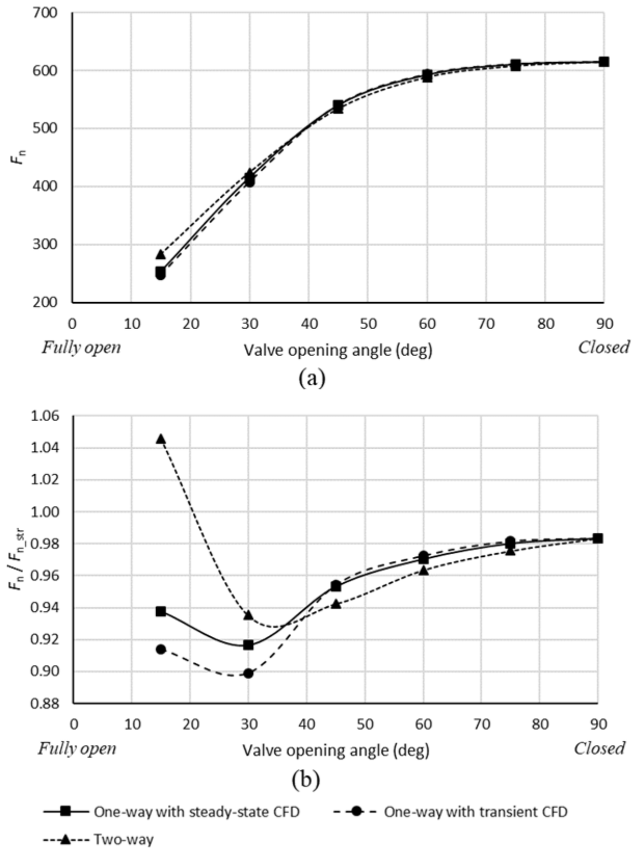

Fig. 7 (a) Normal force acting on the disc and (b) normal force with respect to the one calculated with the classical structural approach, during valve closing operation

In order to extend the structural analysis, Fig. 8 shows (a) the misalignment of the resultant normal force and (b) the resultant moment.

As it can be seen, the highest bending moment is given at low angles, being the maximum at roughly 30º position. This is attributed to the higher misalignment values observed at low angles, which compensate the lower normal force values.

Fig. 9 shows (a) the absolute flow coefficient and (b) the relative flow coefficient with respect to the one-way coupling with steady-state CFD approach.

The flow coefficient varies from a value between 1.4 and 1.5 in the fully open position to 0.09 when the valve is closed. C_V values in the one-way approach with steady-state CFD and the two-way approach are lower, due to both lower mass flow rate and higher total pressure drop.

In order to compare the flow coefficient among the different approaches, their corresponding C_V is normalised with respect to the C_V of the one-way approach with steady-state CFD results. As it can be seen, the one-way approach with transient CFD estimates the highest flow coefficient in all the positions. On the other hand, the two-way approach shows almost the same C_V as the one-way approach with steady-state CFD at 15º and 75º, but it is lower in the rest of the positions.

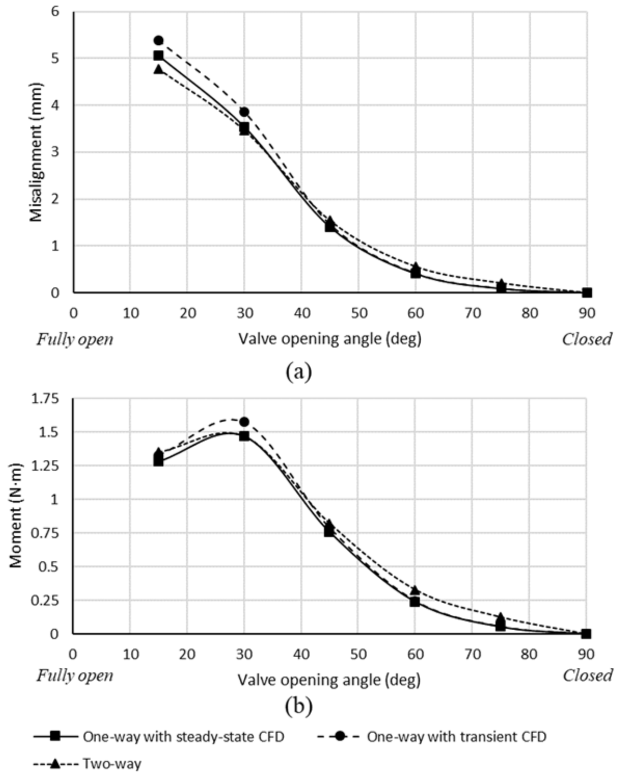

Fig. 8 (a) Resultant force misalignment and (b) resultant moment on the disc, during valve closing operation

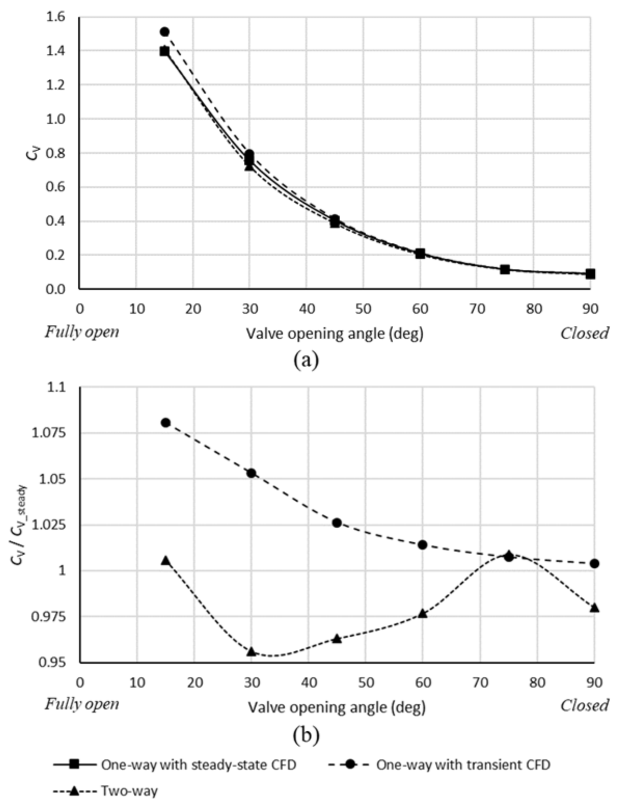

Fig. 9 (a) Flow coefficient C_V and (b) flow coefficient relative to the one-way coupling with steady-state CFD approach, during valve closing operation

B. Structural Analysis Results

The deflections at nodes A and B denoted in Fig. 5 are shown in Table I.

TABLE I
DEFLECTION OF NODES A AND B IN DIFFERENT ANGLES AND APPROACHES

	θ(º)	Node A		Node B	
		Deflection	**Deviation**	**Deflection**	**Deviation**
Classical	90	**1.48**	(Ref.)	**1.48**	(Ref.)
One-way with steady-state CFD	15	1.54	3.6%	0.23	-84.3%
	30	**1.96**	**32.1%**	0.50	-66.6%
	45	1.88	26.6%	1.15	-22.7%
	60	1.76	18.4%	1.54	3.6%
	75	1.73	16.3%	1.67	12.7%
	90	1.71	**15.1%**	**1.71**	**15.1%**
One-way with transient CFD	15	1.55	4.2%	0.20	-86.8%
	30	**1.99**	**34%**	0.42	-71.5%
	45	1.89	27%	1.13	-23.7%
	60	1.76	19%	1.54	3.6%
	75	1.73	16%	1.67	12.8%
	90	1.71	**15%**	**1.71**	**15.2%**
Two-way	15	1.49	0.3%	0.15	-90.0%
	30	**1.72**	**15.9%**	0.27	-81.6%
	45	1.54	3.6%	0.73	-50.7%
	60	1.37	-7.8%	1.04	-29.8%
	75	1.30	-12.5%	1.17	-21.1%
	90	1.24	**-16.3%**	**1.25**	**-16.0%**

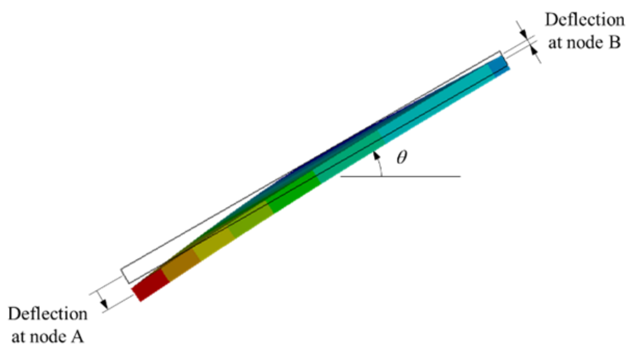

Fig. 10 True scale deformed valve disc vs. absolutely rigid disc for θ = 30º in one-way coupling with transient CFD approach

For each considered approach, the maximum deflection value is highlighted in bold. For the classical structural approach, the deflection is symmetric and its maximum is obtained in the fully closed position (90º), when the pressure drop gets its highest value. Regarding all the other approaches, deflection at node A is higher than at node B in all the disc positions along the closing operation. In addition, they all agree on the fact that maximum deflection at node A arises when butterfly is rotated 30º (see Fig. 10), which is consistent with the angle of maximum resultant moment identified in Fig. 8 (b). Furthermore, the predicted deflection at this position is higher than the one given by the classical structural approach. One-way coupling approaches estimate a 32% to 34% higher deflection, whereas the two-way approach concludes an almost 16% of increment. Therefore, it is proved that not only the absolute normal force value is relevant to dimension the valve disc, but also the resultant moment due to misalignment. In addition, even the deflection at the fully closed position is notoriously different for all the considered approaches: it is 15% higher than the classical structural approach in one-way simulations, and 16% lower in the two-way approach.

With respect to node B, maximum deflection is obtained at fully closed position according to all the considered methods. The deflection deviations measured with respect to the classical structural approach at this position, are practically the same as the ones estimated for node A.

V. CONCLUSION

Regarding fluid dynamic behaviour, all considered approaches present similar pressure and velocity values. However, appreciable variations in the contour distributions are found, which have a direct impact on the resultant mechanical loads transferred to the disc.

With regard to the disc dimensioning analysis, classical structural approach does not consider resultant normal force misalignment, and consequently, it is not able to identify neither the critical operating angle nor the maximum deflection value.

With respect to the applicability of FSI methods, one-way FSI approaches are able to identify the critical operating angle. However, as they do not update the fluid domain related to the disc deformation, differences in the pressure field lead to more conservative deflection values, which may be acceptable for a wide range of applications. Nevertheless, when optimisation is a key aspect, two-way FSI simulations should be performed in spite of their higher computational cost.

REFERENCES

[1] M. G. Al-Azawy, A. Turan, and A. Revell, "Investigating the impact of non-Newtonian blood models within a heart pump," *International Journal for Numerical Methods in Biomedical Engineering*, vol. 33, no. 1, 2017.

[2] L. Wang, X. Song, and Y. Park, "Dynamic analysis of three-dimensional flow in the opening process of a single-disc butterfly valve," *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, vol. 224, no. 2, pp. 329–336, 2010.

[3] E. Krimmer, P. Baur, C. Lorenz, B. Hezel, and H. Hickl, "Butterfly valve with injection-molded shaft," U.S. Patent No 6,901,942, 7 Jun. 2005.

[4] C. Huang and R. H. Kim, "Three-dimensional analysis of partially open butterfly valve flows," *Journal of fluids engineering*, vol. 118, no. 3, pp. 562–568, 1996.

[5] G. Ibrahim, Z. Al-Otaibi, and H. M. Ahmed, "An Investigation of Butterfly Valve Flow Characteristics Using Numerical Technique," *Journal of Advanced Science and Engineering Research Vol*, vol. 3, no. 2, pp. 151–166, 2013.

[6] X. Song, L. Wang, and Y. Park, "Analysis and optimization of a butterfly valve disc," *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, vol. 223, no. 2, pp. 81–89, 2009.

[7] D. W. Roldán, M. de Souza and G. Francisco, "Risk-Based Analysis of LNG Carriers Loading and Unloading Operations," in *The Twenty-second International Offshore and Polar Engineering Conference*, 2012.

[8] A. Göksenli and B. Eryürek, "Failure Analysis of Pipe System at a Hydroelectric Power Plant," *World Academy of Science, Engineering and Technology, International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering*, vol. 9, no. 9, pp. 1643–1646, 2015.

[9] X. G. Song, L. Wang, and Y. C. Park, "Fluid and structural analysis of a large diameter butterfly valve," *Journal of Advanced Manufacturing Systems*, vol. 8, no. 01, pp. 81–88, 2009.

[10] T. Sarpkaya, "Torque and cavitation characteristics of butterfly valves," *Journal of Applied Mechanics*, vol. 28, no. 4, pp. 511–518, 1961.

[11] T. Kimura, T. Tanaka, K. Fujimoto, and K. Ogawa, "Hydrodynamic characteristics of a butterfly valve—prediction of pressure loss characteristics," *ISA transactions*, vol. 34, no. 4, pp. 319–326, 1995.

[12] B. Goksel, J. Rencis, and M. Noori, "Finite element analyses of a butterfly valve assembly," *Journal of pressure vessel technology*, vol. 111, p. 197, 1989.

[13] R. Kim and N. Wu, "Numerical simulation butterfly valve fluid flow," in *proceedings of the FLUENT User's Group Meeting*, 1992, pp. 296–313.

[14] Z. Leutwyler and C. Dalton, "A CFD study of the flow field, resultant force, and aerodynamic torque on a symmetric disk butterfly valve in a compressible fluid," *Journal of Pressure Vessel Technology*, vol. 130, no. 2, p. 021302, 2008.

[15] R. Jaiman, H. Thomas, and F. Shakib, "Direct-Coupled Fluid-Structure Interaction for Automotive Applications," SAE Technical Paper, 2012.

[16] A. El Hami and B. Radi, *Fluid-Structure Interactions and Uncertainties: Ansys and Fluent Tools*. John Wiley & Sons, 2017.

[17] O. C. Zienkiewicz, R. L. Taylor, and P. Nithiarasu, *The Finite Element Method for Fluid Dynamics,* 7th ed. Butterworth-Heinemann, 2014.