International journal of COMADEM
Vol # No # (January 2022) p.##-##

COMADEM

International Journal of Condition Monitoring and Diagnostic Engineering Management

Development of a method for the quantification of oil-water mixing in severe service control valves

Yunluo Yu [a], Muhammad Atif [a] *, Rakesh Mishra [a], Matthew Charlton [b], Andrew Limebear [b] and Aliyu Aliyu [c]

[a] *University of Huddersfield, HD13DH, United Kingdom*
[b] *Koso Kent Introl, Brighouse, HD6 1QG, United Kingdom*
[c] *University of Lincoln, LN6 7TS, United Kingdom*

* *Muhammad Atif. Tel.: +44 7565935968; email: Muhammad.atif@hud.ac.uk*

ABSTRACT

Control valves in the oil and gas industries frequently handle oil-water flow. Pressure drops and energy dissipation in the valve generate turbulence mixing in the flow which causes higher shear forces resulting in emulsification. This highly emulsified mixtures require a large size separator, extra time and chemical additives that increase the cost of the mixture flow and separation operations. Therefore, it is essential to quantify the extent of shear forces generated and degree of oil-water mixing resulting because of valves present in the oil transportation pipeline system and also to evaluate the performance of different valves. In this study, a novel quantification method is proposed to evaluate the extent local and global oil-water mixing within and because of valves. A widely used indicator; Mixing Coefficient (M_c) together with other novel proposed indicator Modified Mixing Coefficient (M_{mc}) and Velocity-involved Modified Mixing Coefficient (V_{mmc}), are used as indicators to evaluate the local oil-water mixing degree at various cross-sectional planes of the valve. The variances of M_c, M_{mc}, and V_{mmc} are used to evaluate the global oil-water mixing performance of the valve. A well validated computational fluid dynamics model is used to evaluate the oil-water mixing degree globally and locally caused by two valves equipped with tangentially oriented and directly oriented orifices respectively. It has been concluded that at higher inlet velocities, the tangentially oriented orifices can reduce the oil-water mixing better than directly oriented orifices. At lower velocities, however, the directly oriented orifices are providing better separation properties. When considering the in-situ oil volume fraction, the tangentially oriented orifices can always produce a higher change in in-situ oil volume fraction compared with directly oriented orifices. The quantification method and the comparison of tangentially and directly oriented orifices can be used to inform the control valve design in oil and gas industries.

Keywords: *Oil-water mixing degree; contrl valves; mixing coefficient; tangentially-oriented orifices; directly-oriented orifices.*
Article history: *Received ; Published .*

1. Introduction

Severe service control valves are widely used in various industrial applications such as chemical, power generation, hydrocarbon, and oil & gas industries [1-5]. Multiphase flows in these applications are frequently used where two immiscible liquids flow together in the same pipeline. For example, oil and water flow can be found in both upstream and downstream processes in the oil industry. The two immiscible liquids mix together as they pass through valves due to the turbulence and shear generated by the valve body, trim and sealings. The mixing can also happen in straight pipes because of intrinsic turbulence of the fluid flow. This highly mixed and emulsified mixture requires extra time and chemical additives that complicates the downstream separation process and increase the cost for the oil-water separation process [6-8].

Extensive work has been devoted to study the low-shear valves with the aims of achieving higher oil-water separation efficiency. Previous studies [9-13] quantified the oil/water mixing in valves using droplet sizes. Since predicting the maximum droplet size in oil-water flows is challenging and time-consuming, most research that employ maximum droplet sizes to quantify oil/water mixing are experimental investigations. Incorporating the CFD evaluation technique into the valve design and development processes, however, will save time and money as compared to experimental techniques. Monteiro and Petrobras [14] quantified the oil/water mixing by the Oil-in-Water (OiW) contamination at the outlet. They experimentally compared a low shear valve with a 1/4 inch conventional globe valve for water/oil separation and concluded that the low shear valve outperforms the conventional globe valve by achieving a reduction on OiW contamination in the range of 22-51%. They also reached the conclusion that the low shear valve's benefits are highly dependent on the process conditions and variables. Analysing local flow fluctuations and quantifying mixing behaviour at different process settings can contribute to the development of more efficient and robust design models of low shear valves development. Therefore, there is a need to establish a quantification approach for oil-water mixing in valves that can show both local and global oil-water mixing degrees in valves and can be complemented numerically and experimentally.

$$Mc = \sqrt{\frac{\frac{1}{n}\sum_1^n (c_i - \bar{c})^2}{\bar{c}(1-\bar{c})}} \quad (1)$$

The Mixing Coefficient (Mc), given by the equation 1, where n is the total sampling number, c_i the volume fraction at a position on the cross section considered and $\bar{c}$ the average of c_i, has been widely used to indicate the degree of mixing in multiphase applications [15-17]. Wu and Tsai [15] used Mc as an indicator to evaluate the performance of micromixers with different geometries and flow conditions. Their results show that the Mc of a newly proposed micromixer is better than that of the ordinary one, thus indicating a better mixing performance. Rasouli et al. [16] took Mc as the design objective and performed a multi-objective genetic algorithm for a curved micromixer to optimize its geometry. Their simulation results validated the accuracy and trend of the predicted objective functions for the optimum designs. Zhang et al. [17] experimentally studied the mixing degree and flow regime of T-jet reactors with various geometries and concluded that as the headspaces height increases, the mixing performance becomes lower and the engulfment of two inlet streams becomes weaker. The above expression however does not take into consideration the slip between the different phases. Considering the limitations of Mc in quantifying liquid-liquid mixing both numerically and experimentally, it is promising to quantify the oil-water mixing degree in valves using Mc.

$$Mmc = \sqrt{\frac{\frac{1}{n}\sum_1^n (c_i - \bar{c_0})^2}{\bar{c_0}(1-\bar{c_0})}} \quad (2)$$

$$Vmmc = \sqrt{\frac{\frac{1}{n}\sum_1^n (c_i v_i - \bar{c_0}\bar{v_0})^2}{\bar{c_0}\bar{v_0}(\bar{v_0} - \bar{c_0}\bar{v_0})}} \quad (3)$$

Modified Mixing Coefficient (M_{mc}), where ($\bar{c_0}$) denoting the average volume fraction at the start of mixing process. The other indicator is Velocity-involved Modified Mixing Coefficient (V_{mmc}), where v_i denoting the velocity at a position on the cross section considered and ($\bar{v_0}$) denoting the average velocity at the start of mixing process. It is hypothesised that the above three methods will be able to represent mixing quantitatively over a wide range of flow regimes.

Given this opportunity, this study has been designed to evaluate effectiveness of the quantification methods to measure the extent of oil-water mixing because of complex geometry associated with two different globe valves. This quantification can be used to compare the performance of low-shear valves numerically using Computational Fluid Dynamics (CFD). For this purpose, several cross-sectional planes are created at different position within the valve and the M_c, M_{mc}, and V_{mmc} at each plane are calculated to evaluate progression of mixing from inlet to the outlet as well as to establish relationship between local mixing and overall performance of the valve. The variances in the computed values of the M_c, M_{mc}, and V_{mmc} corresponding to different planes are utilised to quantitatively evaluate the global performance effectiveness of the valve.

2. CFD simulation of the oil-water flow in valves

2.1. CFD simulation

The numerical modelling of the globe valves having directly oriented and tangentially oriented orifices was conducted using the Ansys 2020 R1 software to simulate the extent of mixing of the oil and water flowing within the valve. The CFD simulation processes include pre-processing, solver setup and post-processing stages. In pre-processing stage, the geometry of the fluid domain is created and meshed. Then, the fluid properties, boundary conditions, turbulence modelling and convergence criteria are specified in solver setup stage. These steps are essential specially when conducting CFD modelling of mixture flow passing complex geometries. Finally, the results are visualized at the post-processing stage.

2.1.1. Pre-processing

The fluid domain created for the studied valve and its inlet/outlet pipes is shown in Figure 1. The valve used in this study is a globe valve that is widely used in oil & gas industry. The valve has a same internal diameter D at the inlet and outlet. As shown in Figure 1, the length of the inlet pipe is 2D, while the length of the outlet pipe is 6D. This length is determined as per the BS EN Standards [20-22]. The outlet pipe is longer than the inlet pipe to allow the fluid flow to be fully developed or stabilized before taking any measurements.

Meshing is an important step in any CFD analysis. In this step, the fluid domain is discretized into small parts (mesh elements). Some parts of the valves contain geometric details such as very small edges and faces, which are vital for the manufacturing of the valve. However, in CFD modelling process, their impact on the mechanics of the studied problem can be neglected. Moreover, these small edges and faces may unduly constrain the mesh generation, leading to a larger element number or a poor mesh. Therefore, in this study, virtual topology function of Ansys Workbench is used to ignore the small edges and faces of valve bodies in the mesh generation process, as shown in Figure 2.

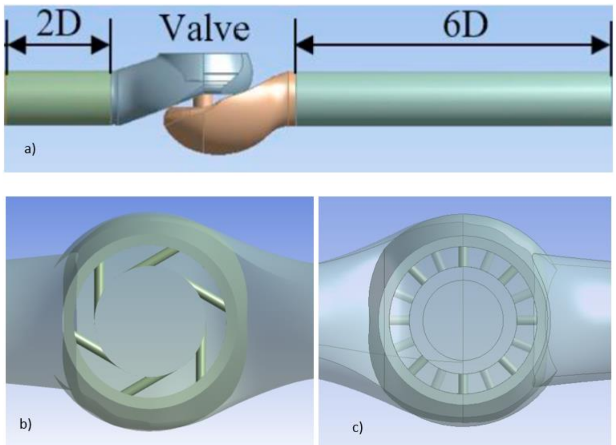

Figure 1. a) CAD model of the control valve/inlet and outlet pipe geometries b) tangentially oriented trim c) directly oriented trim

The number of mesh elements also affects the accuracy and computational time of the CFD simulation [23]. To accurately simulate the local flow within the valves, a large number of mesh elements, typically in the order of millions is required for effective 3-dimensional simulations. Moreover, at regions with strong curvature and/or flow gradients [24-25], such as the orifices, the number of mesh elements should be increased to ensure the accuracy of the simulation. Figure 3 depicts an example of the meshing of a valve. Two types of mesh grids are adopted in the fluid domain – hexahedral and tetrahedral mesh. At inlet and outlet pipes, where the geometry is less complicated, hexahedral mesh are adopted. At the main valve body, where the geometry is more complicated, tetrahedral mesh are used. The type of the mesh used in the fluid domain should also be selected based on the flow symmetry. It is widely agreed that tetrahedral mesh should be used for asymmetric flows, while hexahedral mesh should be used for symmetric flows to achieve more accurate results [24].

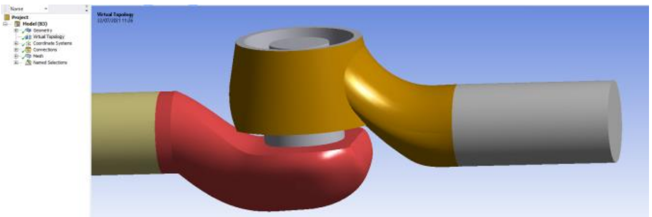

Figure 2. The virtual surfaces applied in the meshing process

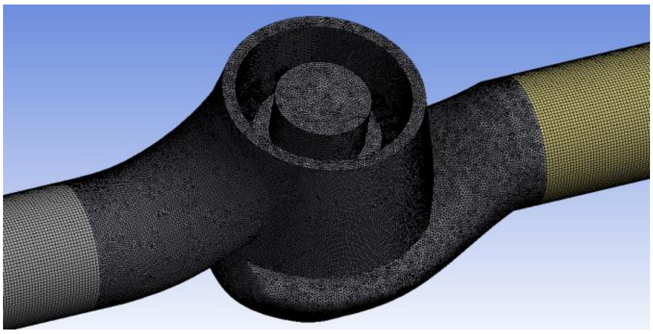

Figure 3. Hybrid mesh of the valve flow domain

2.1.2. Solver setup

Eulerian-Eulerian multiphase model has been widely used to simulate two-phase liquid-liquid flows where water is modelled as the primary phase and oil as the secondary phase [26-28]. Table 1 shows the inlet boundary conditions at various velocities and oil volume fractions. Atmospheric pressure is set at the outlet boundary and the no-slip condition is set at the walls. In solver settings, the SIMPLE pressure-velocity coupling scheme is used for single phase and the Phase-coupled SIMPLE scheme is used for two-phase simulation. RSM model is selected as the turbulence model. Convergence is monitored using the difference of the mass flow rate of the oil at inlet and outlet. All the simulations are conducted at steady state conditions. In the solver setup, the mesh can also be turned to Polyhedra to reduce the element number, as shown in Figure 4. After mesh change to Polyhedra, the element number reduced from 7.9 million to 2.8 million in this case.

Table 1. Inlet boundary conditions

Volume fraction (%)	Water inlet velocity (m/s)	Oil inlet velocity (m/s)	Mc
	1	1	0 (well-mixed)
5	4	4	0 (well-mixed)
	10	10	0 (well-mixed)
	1	1	0 (well-mixed)
20	4	4	0 (well-mixed)
	10	10	0 (well-mixed)
	1	1	0 (well-mixed)
40	4	4	0 (well-mixed)
	10	10	0 (well-mixed)

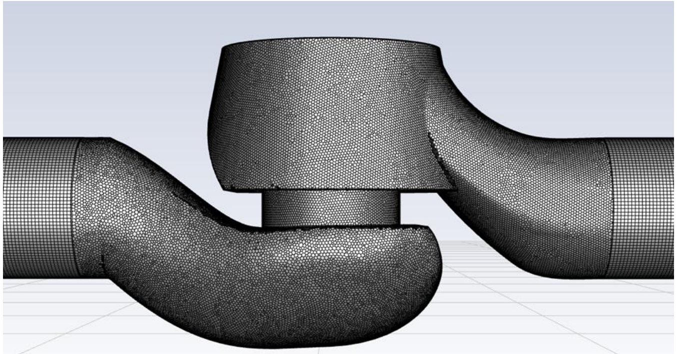

Figure 4. The converted Polyhedra mesh on valve body

2.2. CFD validation

To ascertain the accuracy of the CFD model developed in section 2.1, a comparison between the CFD predicted results and the experimental data is carried out. In this study, first, the flow coefficients (Cv) of the studied valve at various valve opening positions are predicted using the CFD model and compared with the designed value. The equation to calculate the Cv of valves is listed below [29-30]:

$$Cv = Q\sqrt{\frac{S.G.}{\Delta P}} \tag{4}$$

where Q is the mass flow rate, S.G. is the specific gravity of the fluid, ΔP is the differential pressure of the valve system.

Figure 5 shows the Cv of the control valve system at different Valve Opening Positions (VOP). It can be found that the Cv obtained by CFD modelling matches well with the designed Cv obtained from the data sheet of the manufacturer. The most part of the difference is caused by numerical convergence.

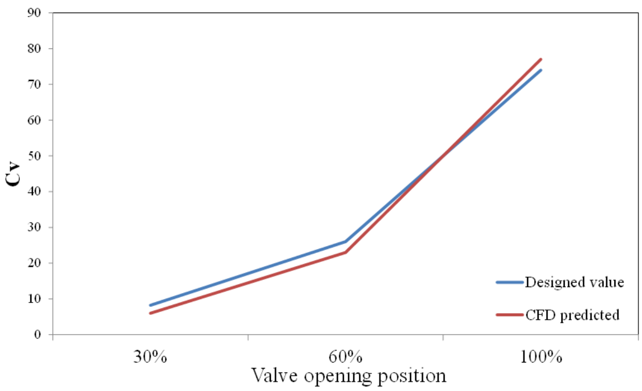

Figure 5. The comparison of the CFD predicted Cv and the designed Cv at different VOP

The experimental data of Soleimani [18] are used to verify the accuracy of the CFD model in predicting the oil-water distribution at various inlet velocities and oil volume fractions. The simulation results of Berrio et al. [19] are also compared in this study. Figure 6 depicts the geometry and boundary conditions of the simulated pipe in Ref. [18-19]. The diameter and other simulation parameters are given in Table 2 where the diameter and length of the pipe was kept constant. The measurement of water fraction was taken at different heights along the diameter of the pipe.

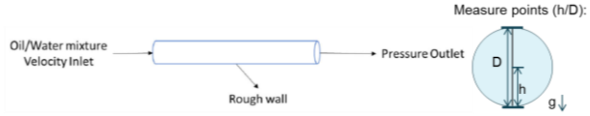

Figure 6. The pipe simulated in Ref. [18-19] for validation

Table 2. The simulated conditions in Ref. [18-19]

Pipe diameter	0.0254m	Interfacial tension	0.017 N/m
Pipe length	9.7m	Water density	1000 kg/m^3
Oil viscosity	0.0016 Pa.s	Water viscosity	0.001 Pa.s
Oil density	801 kg/m^3	Data acquisition location	8m of length

The results in Figure 7 represent variation in volume fraction at different h/D values. Inlet velocities of 3m/s, 2.12m/s and 1.25m/s and inlet water fractions of 60% and 46% have been simulated and the results are compared with the experimental data of Soleimani [18] and simulation data of Berrio et al. [19]. It can be found from Fig.7 that the CFD results obtained by the established model match well with the experimental and simulation results in Ref.[18-19] at various inlet velocities and water fractions. The accuracy of the established CFD methodology in predicting the oil-water interaction and distribution is thus validated. Using similar CFD methodology, a detailed quantitative analysis of the oil-water mixing within the valve can be carried out.

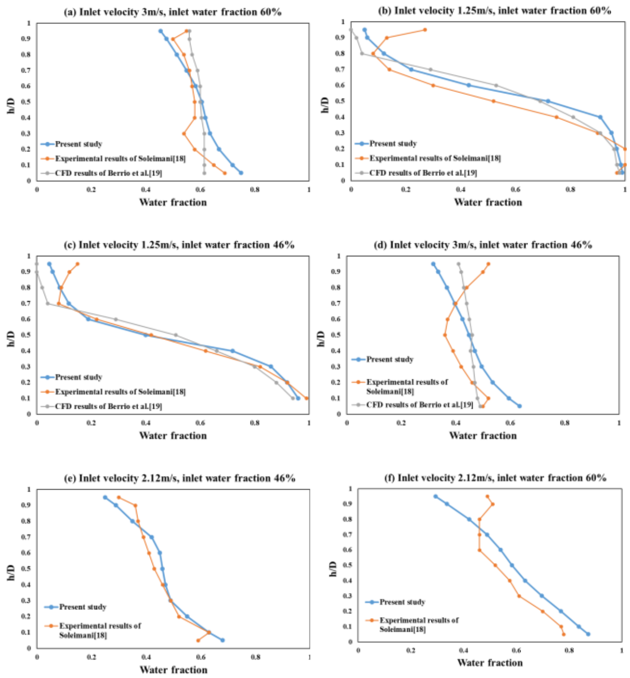

Figure 7. The comparison of the CFD results in present study with the experimental results in Ref. [18] and the simulation results in Ref.[19]

3. Quantification of oil-water mixing in valves

In this section, an evaluation process of the extent of oil-water mixing within globe valves having directly oriented and tangentially oriented orifices has been carried out using the proposed quantification method. The specification of both the valves is given in Table 3. Glove valve geometry was selected as they are commonly used in the oil industry regulating the oil-water flow in the pipeline and are expected to maintain constant conditions for the mixture before being transported to the oil-water separator. Extraction of the local flow data along the flow path including at cross-section of inlet/outlet pipes and valve body as shown in Figure 8 was carried using eleven local planes. Each plane is perpendicular to the direction of the fluid velocity. Planes 1, 4 and 11 are at the pipe inlet, seat of the valve, and pipe outlet respectively. The mixing parameters Mc along with other proposed indicators MMc and VMMc were computed on each local plane to evaluate the degree of local oil-water mixing within the valves. From Table 3, it can be found that when the orifices are tangentially oriented, the Cv of valve is reduced. Conditions in Table 1 were used as the inlet boundary condition of all simulations.

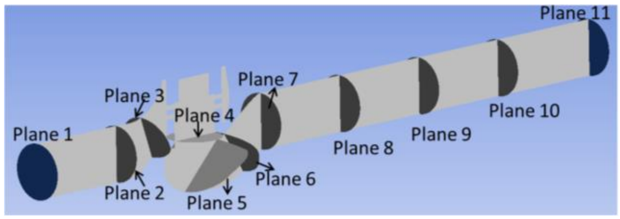

Figure 8. The local cross-sectional planes extracted in the globe valves to capture local flow

Table 3. The parameters of the globe valves studied in this paper

	Globe valve I	Globe valve II
Orifice arrangement	Directly-oriented	Tangentially-oriented
Orifice diameter	9 mm	9 mm
Orifice number	30	30
Inlet/Outlet diameter	4 inch	4 inch
Cv	77	61

Oil volume fraction of both valves is compared in the figure 9, where left side represents directly oriented globe valve I and right side represents tangentially oriented valve II. Comparison was carried at two different velocities 10 and 4m/s with two different oil volume fractions of 0.2 and 0.4. A well-mixed oil and water mixture is allowed to enter from the inlet side and the oil being lighter fluid starts floating and accumulating on top of the water in the inlet pipe. After passing through the orifices, the flow partially separates because tangentially oriented orifices are generating swirl motion. Water being a denser fluid as compared to oil, it moves towards the valve body and oil accumulates in the middle. As there are no swirl breakers or flow straightener equipped in the downstream pipe, the oil and water again partially start mixing in the downstream pipe. The size of the oil droplet reduces as it travels towards the downstream pipe. Directly oriented orifices valve on the other hand has high mixing of the oil and water as it passes through the trim. The higher concentration of the oil can be seen towards the valve body at 4m/s. At higher velocities in the directly oriented orifices, the flow is uniformly mixed and separation properties are very poor. The performance of the tangentially oriented orifices is, however, better as compared to the directly oriented orifices in terms of the oil-water separation.

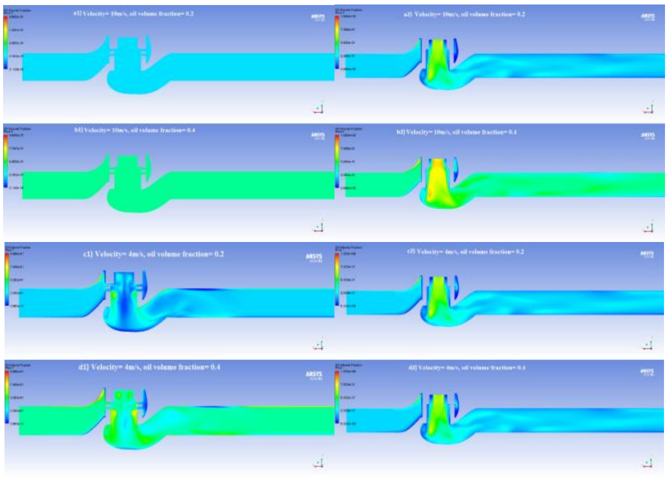

Figure 9. Distribution of oil volume fraction for globe valve with directly oriented (left pictures) and tangentially oriented (right pictures) orifices at 10m/s and 4m/s for 0.2 and 0.4 oil volume fractions

From Figure 10, the local separation effects can be quantified. It can be seen that at 1m/s, the extent of oil-water mixing in the two valves is similar globally but different locally especially within the valve body. When velocity is increased to 4m/s and 10m/s, the tangentially oriented globe valve II has a much higher M_c (indicating much better separation) than globe valve I at planes 4-5, which is because of the swirl formation by the tangentially oriented orifices. The flow rotates after passing through tangential orifices and both oil and water start partially separating. Water being dense fluid moves towards the walls and oil accumulate in the center of the rotating vortex. Thus, it can be concluded when using M_c as an indicator is quite effective in capturing local separation in the flows. For all the simulated oil volume fractions, tangentially oriented orifices seem to provide a better local oil-water separation performance although effects are small at low inlet velocity. Besides, it can also be found from Fig. 9 that a higher inlet oil volume fraction always leads to a better oil-water separation performance for the two valves at all the simulated velocities. Although M_c is a good indicator for analysing mixing behavior its inability to capture volume fraction history and velocity field information are major limitations.

A modified form of M_c has been developed, namely M_{mc}, to evaluate mixing. For this purpose, M_{mc} corresponding to planes 1-11 within the two valves are calculated and shown in Figure 11. Although trends are similar to that seen in figure 10, some additional information is captured by this indicator. It can be seen that at all the simulated velocities, the M_{mc} values corresponding to the planes 4-5 of globe valve II are much higher than that of globe valve I. Also, M_{mc} values are higher than M_c values indicating better sensitivity to the separation phenomena. For very/moderately mixed flow both the M_c and M_{mc} provide similar values. Both the M_c and M_{mc} indicators have established the superiority of the valve with tangentially oriented orifices in local oil-water separation performance at all the simulated inlet velocity and oil volume fractions. It also provides a clear picture of how the separation is affected by increasing the oil fraction. Higher oil fraction encourages a better flow separation and larger size of oil domains.

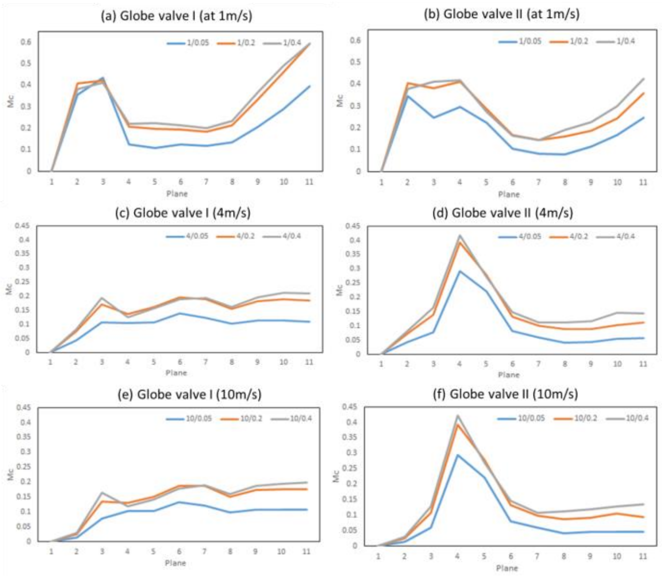

Figure 10. The M_c values at planes 1-11 of globe valve I and II, where the legend represents the inlet velocity of oil (m/s)/ the inlet oil volume fraction

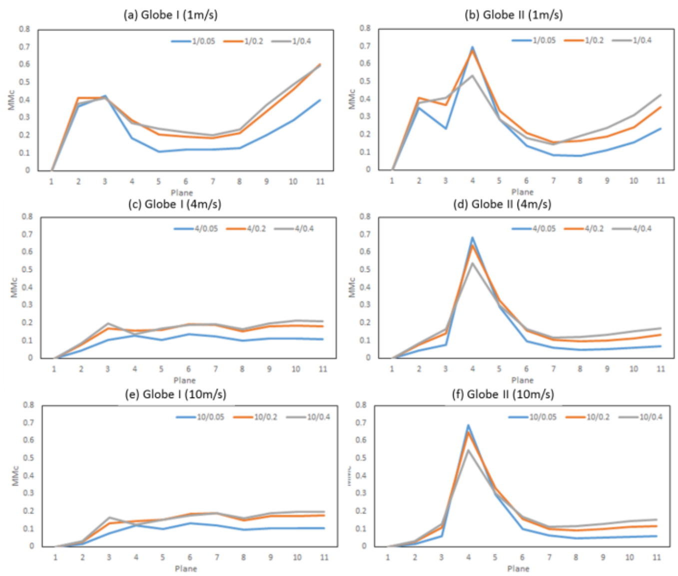

Figure 11. The M_{mc} values at planes 1-11 of globe valve I and II, where the legend represents the inlet velocity of oil (m/s)/ the inlet oil volume fraction

To incorporate velocity induced effects a new parameter V_{mmc} has been developed. The V_{mmc} indicator seems to significantly increase (up to three times) with the separation as compared to M_c and M_{mc}. It thus increases sensitivity to separation prediction significantly as compared to M_c and $M_{mc.}$ Typically, there are three factors that affect the mixing, namely, geometry, velocity and volume fraction (figure 12). This indicator includes local velocity field information in addition to the volume fraction and captures more local information on the flow mixing. It was found that at all the simulated speeds, the V_{mmc} of planes 4-5 of globe valve II are much higher than that of the globe valve I, indicating a larger change in the in-situ oil volume fraction caused by the fluid flowing. The quantification of the flow separation is much clear in terms of the proposed M_{mc} and V_{mmc} indicators as compared to the M_c.

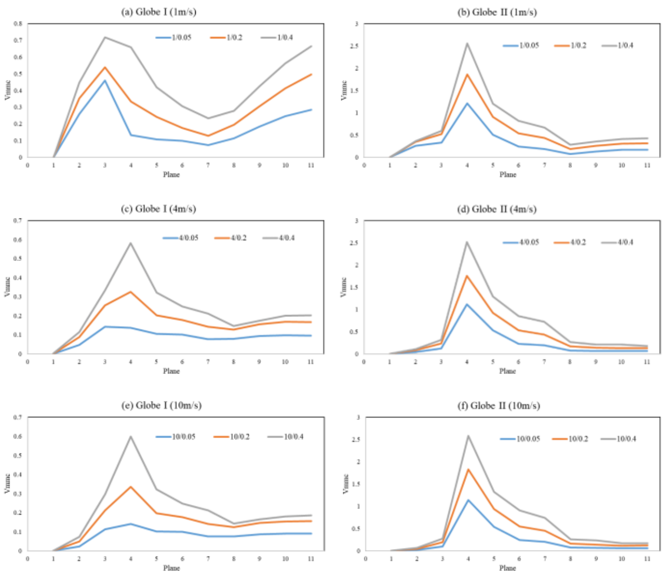

Figure 12. The V_{mmc} values at planes 1-11 of globe valve I and II, where the legend represents the inlet velocity of oil (m/s)/ the inlet oil volume fraction

After evaluating the local separation, an attempt has been made to develop a local indicator to quantify overall performance of the valve. To evaluate the global performance of the oil-water mixing of the valve, the variances of the M_c, M_{mc} and V_{mmc} values at planes 1-11 are calculated and shown in Table 4. From Table 4, it was concluded that the variances of the M_c, M_{mc} and V_{mmc} of globe valve II is always higher or very close to that of globe valve I at high inlet velocity (4m/s and 10m/s), which indicating that the oil and water are fairly separated in globe valve II. This is because the tangentially oriented orifices are generating swirl effect in the fluid and that promote separation of the oil and water because of density difference. At low inlet velocity (1m/s), the variances of M_c and M_{mc} are indicating that both valves have higher mixing and lower separation efficiency. The variances of V_{mmc} of globe valve II is always higher than that of globe valve I at all the simulated conditions, indicating a bigger change in in-situ oil volume fraction after the fluid passes through a valve. Therefore, when considering in-situ oil volume fraction change, the globe valve II always has a better performance than globe valve I.

Table 4. The sum of Mc, MMc, VMMc of planes 1-11 for globe valve I and II (the higher value is marked in red)

Boundary condition	Sum of Mc		Sum of MMc		Sum of VMMc	
	Globe I	Globe II	Globe I	Globe II	Globe I	Globe II
10/0.05	0.066	**0.096**	0.067	**0.181**	0.058	**0.304**
10/0.2	0.095	**0.127**	0.096	**0.184**	0.109	**0.537**
10/0.4	0.105	**0.125**	0.107	**0.157**	0.196	**0.802**
4/0.05	0.063	**0.098**	0.063	**0.182**	0.061	**0.294**
4/0.2	0.100	**0.126**	0.096	**0.186**	0.116	**0.519**
4/0.4	0.106	**0.130**	0.109	**0.156**	0.197	**0.779**
1/0.05	**0.207**	0.166	0.207	**0.221**	0.175	**0.330**
1/0.2	**0.272**	0.216	**0.262**	0.257	0.252	**0.519**
1/0.4	**0.270**	0.223	**0.264**	0.236	0.330	**0.737**

4. Conclusions

In this study, a well-validated CFD study was presented to evaluate the oil-water mixing degree within two severe-service control valves, with tangentially oriented and directly oriented orifices respectively. Three oil inlet volume fractions (5%, 20%, 40%) were tested at 1m/s, 4m/s, and 10m/s inlet velocities. Based on the established CFD model, a novel quantification method for the evaluation of extent of oil-water mixing within the control valves is proposed. The M_c combined with proposed mixing indicators M_{mc} and V_{mmc} are computed at eleven different local planes along the valve and the connecting pipes. The variances of the M_c, M_{mc} and V_{mmc} of the planes are used to evaluate the global separation performance of the valve. Higher values of the indicators are representing better flow separation. It was found that tangentially oriented orifices are generating less shear forces and improving the oil separation capacity of the valve. However, the CV of the tangentially oriented valve was comparatively lower than that of the directly oriented orifices globe valve. In directly oriented orifices valve, the fluids go through a large pressure drop in relatively smaller length and shear forces are much higher which promote the oil-water mixing. The proposed quantification method evaluates the local and global oil-water mixing of control valves and can be used to compare the performance of different valves. The V_{mmc} indicator captures the separation effects more strongly as compared to M_c and M_{cc} and therefore can be used as an indicator for both local and global separation behaviour within valves.

5. References

1. [1] Asim, T., Charlton, M. and Mishra, R., 2017. CFD based investigations for the design of severe service control valves used in energy systems. Energy Conversion and Management, 153, pp.288-303.
2. [2] Jacobson, W. and Zaghloul, M., 2017. Turbomachinery Control Valves-Sizing and selection. In Proceedings of the 46th Turbomachinery Symposium. Turbomachinery Laboratory, Texas A&M Engineering Experiment Station.
3. [3] Gao, Z.X., Yue, Y., Wu, J.Y., Li, J.Y., Wu, H. and Jin, Z.J., 2021. The flow and cavitation characteristics of cage-type control valves. Engineering Applications of Computational Fluid Mechanics, 15(1), pp.951-963.
4. [4] Xu, X., Fang, L., Li, A., Wang, Z. and Li, S., 2021. Numerical Analysis of the energy loss mechanism in cavitation flow of a control valve. International Journal of Heat and Mass Transfer, 174, p.121331.
5. [5] Qian, J.Y., Hou, C.W., Mu, J., Gao, Z.X. and Jin, Z.J., 2020. Valve core shapes analysis on flux through control valves in nuclear power plants. Nuclear Engineering and Technology, 52(10), pp.2173-2182.
6. [6] Raghavan, R. and Marsden, S.S., 1971. Theoretical aspects of emulsification in porous media. Society of Petroleum Engineers Journal, 11(02), pp.153-161.
7. [7] Eow, J.S. and Ghadiri, M., 2002. Electrostatic enhancement of coalescence of water droplets in oil: a review of the technology. Chemical Engineering Journal, 85(2-3), pp.357-368.
8. [8] Cunha, P.S.M.D., 2008. Mathematical Modeling of the Crude Oil Electrostatic Dehydration Process. Universidade Federal doRio de Janeiro, EQ, RJ, Brazil (Master's thesis), (in Portuguese).
9. [9] Paolinelli, L.D., Rashedi, A. and Yao, J., 2018. Characterization of droplet sizes in large scale oil–water flow downstream from a globe valve. International Journal of Multiphase Flow, 99, pp.132-150.
10. [10] Mitre, J.F., Lage, P.L., Souza, M.A., Silva, E., Barca, L.F., Moraes, A.O., Coutinho, R.C. and Fonseca, E.F., 2014. Droplet

breakage and coalescence models for the flow of water-in-oil emulsions through a valve-like element. Chemical Engineering Research and Design, 92(11), pp.2493-2508.

11. [11] Husveg, T., Bilstad, T., Guinee, P., Jernsletten, J., Knudsen, B. and Nordbø, H.T., 2009, May. A cyclone-based low shear valve for enhanced oil-water separation. In Offshore Technology Conference. OnePetro.
12. [12] Flanigan, D.A., Stolhand, J.E., Scribner, M.E. and Shimoda, E., 1988, October. Droplet size analysis: a new tool for improving oilfield separations. In SPE Annual Technical Conference and Exhibition. OnePetro.
13. [13] Paiva, A.P., 2004. Estudo da minimizac¸ão de erro das medic¸ões deconcentrac¸ão de emulsões por titrac¸ão Karl–Fisherutilizando-se projetos de experimentos. Universidade Federalde Itajubá, RJ, Brazil (Master's thesis)
14. [14] Monteiro, A.S. and Da Silva, F.S., 2019, October. Test Results of a Comparison between a Conventional and Low Shear Valve and Its Effect on Water/Oil Separation. In Offshore Technology Conference Brasil. OnePetro.
15. [15] Wu, C.Y. and Tsai, R.T., 2013. Fluid mixing via multidirectional vortices in converging–diverging meandering microchannels with semi-elliptical side walls. Chemical engineering journal, 217, pp.320-328.
16. [16] Rasouli, M., Mehrizi, A.A., Goharimanesh, M., Lashkaripour, A. and Bazaz, S.R., 2018. Multi-criteria optimization of curved and baffle-embedded micromixers for bio-applications. Chemical Engineering and Processing-Process Intensification, 132, pp.175-186.
17. [17] Zhang, J.W., Li, W.F., Xu, X.L., El Hassan, M., Liu, H.F. and Wang, F.C., 2020. Effect of geometry on engulfment flow regime in T-jet reactors. Chemical Engineering Journal, 387, p.124148.
18. [18] Soleimani A. 1999. Phase distribution and associated phenomena in oil-water flows in horizontal tubes [PhD thesis]. London: Imperial College London.
19. [19] Juan Carlos Berrio , Andres Pinilla & Nicolas Ratkovich (2020): CFD study of oil-water segregated and dispersed flow coalescence in horizontal pipes, Chemical Engineering Communications
20. [20] BS EN 60534-2-1, 2011. "Industrial-process Control Valves. Part 2–1: Flow Capacity – Sizing Equations for Fluid Flow under Installed Conditions.
21. [21] BS EN 60534-2-3, 1998. "Industrial-process Control Valves. Part 2–3: Flow Capacity – Test Procedures.
22. [22] BS EN 60534-2-5, 2003. Industrial-process Control Valves. Part 2-5: Flow Capacity - Sizing Equations for Fluid Flow through Multistage Control Valves with Interstage Recovery.
23. [23] Asim, T., Oliveira, A., Charlton, M., Mishra, R., May 2019. Improved design of a multistage continuous-resistance trim for minimum energy loss in control valves. Energy 174, 954–971.
24. [24] Versteeg, H.K., Malalasekera, W., 2007. An Introduction to Computational Fluid Dynamics: the Finite Volume Method, second ed. Pearson Education, Harlow, UK.
25. [25] Ferziger, J.H., Perić, M. and Street, R.L., 2002. Computational methods for fluid dynamics (Vol. 3, pp. 196-200). Berlin: springer.
26. [26] Parvini, M., Dabir, B. and Mohtashami, S.A., 2010. Numerical simulation of oil dispersions in vertical pipe flow. Journal of the Japan Petroleum Institute, 53(1), pp.42-54.
27. [27] Roudsari, S.F., Turcotte, G., Dhib, R. and Ein-Mozaffari, F., 2012. CFD modeling of the mixing of water in oil emulsions. Computers & chemical engineering, 45, pp.124-136.
28. [28] Walvekar, R.G., Choong, T.S., Hussain, S.A., Khalid, M. and Chuah, T.G., 2009. Numerical study of dispersed oil–water turbulent flow in horizontal tube. Journal of Petroleum Science and Engineering, 65(3-4), pp.123-128.
29. [29] Grace, A. and Frawley, P., 2011. Experimental parametric equation for the prediction of valve coefficient (Cv) for choke valve trims. International Journal of Pressure Vessels and Piping, 88(2-3), pp.109-118.
30. [30] Kang, C.W., Yi, C.S., Jang, S.M. and Lee, C.W., 2019. A Study of the Measurement of the Flow Coefficient Cv of a Ball Valve for Instrumentation. Journal of the Korean Society of Manufacturing Process Engineers, 18(3), pp.103-108.