SEMARAK ILMU PUBLISHING

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

A Numerical Comparison of 2D and 3D CFD Modelling for Contraction and Expansion Geometries with an Emphasis on Solid Particles Erosion

Faical Baghdadi[1], Meftah Hrairi[1,*], Waqar Asraar[1]

[1] Department of Mechanical and Aerospace Engineering, Faculty of Engineering, International Islamic University Malaysia, PO Box 10, 50728 Kuala Lumpur, Malaysia

ARTICLE INFO

Article history:
Received 8 June 2023
Received in revised form 5 July 2023
Accepted 9 August 2023
Available online 21 January 2024

Keywords:
Contraction expansion; 2D Vs 3D; Erosion; Particles tracking

ABSTRACT

In this study, erosion patterns and magnitude are compared between the outputs of 2D and 3D CFD models in contraction and expansion geometries. ANSYS Fluent software was used to model a circular cross-section geometry with a contraction and the results were compared to published experimental data. The simulation findings showed that there is good agreement between the 2D and 3D CFD models and the experimental data in terms of fluid flow properties such as velocity profiles and magnitude. It also demonstrated that the 2D and 3D CFD models' representations of erosion patterns and magnitudes are equivalent. The 3D CFD simulations were able to provide more information than the 2D CFD simulations, particularly in terms of erosion distribution over the entire geometry.

1. Introduction

Solid particle erosion is a phenomenon that occurs in a variety of industrial applications, such as oil and gas pipelines, especially in fitting like bends and chokes. The erosion of solid particles on the surface of these components can cause significant damage, leading to decreased efficiency, increased maintenance costs, and even failure as highlighted by Abdulla [1]. One particular area of interest is the erosion in contraction and expansion geometries which are commonly found in piping systems and are known to cause increased erosion due to changes in flow velocity and direction.

Many past studies have investigated solid particles erosion in contraction and expansion geometries using experimental and numerical methods especially in oil and gas industry either for surface pipe fittings or downhole equipment for oil and gas wells. Xu *et al.,* [2] investigated the failure in abrupt expansion in fracturing tubing in gas wells and found that numerical results were in good agreement with field data. At the front edge of a short, abrupt constriction following expansion, the maximum erosion rate was observed which quickly diminished downstream. Additionally, a quite significant erosion rate was observed at the reattachment region. In a similar application, Cheng *et al.,* [3] conducted an experiment to study the erosion in sudden contraction during fracturing

* *Corresponding author.*
E-mail address: meftah@iium.edu.my (Meftah Hrairi)

https://doi.org/10.37934/cfdl.16.6.157168

operation in oil or gas wells using non Newtonian fluids and found that the highest erosion occurred at the edge of contraction and that the erosion rate is mostly affected by fluid flow velocity, followed by particle concentration and diameter ratio. Similar observations of erosion location and magnitude were obtained by Duarte *et al.,* [4] which studied Dynamic mesh approaches for eroded shapes including a contraction geometry. Darihaki *et al.,* [5] presented a novel approach to predict erosion in oil and gas wells that includes expansion and contraction geometry. A three phases unsteady well modeling software was used to get fluid properties and velocities, which were the input for a numerical CFD software to obtain the particles erosion characteristics. It was found that higher erosion is located near the wellhead and that erosion in gas wells is more severe than oil wells, especially if no liquid film is present.

Numerical simulations, particularly using Computational Fluid Dynamics (CFD), have become an increasingly popular method for studying solid particle erosion. CFD allows for a more detailed analysis of the flow and erosion behavior, and can be used to investigate a wide range of parameters. A number of studies have employed both 2D and 3D CFD models to investigate erosion in geometries that involve contraction and expansion. Li [6], for instance, utilized 2D model to compare the results obtained from two different software programs and found that OpenFoam's results were in line with those of Ansys Fluent. 2D model used as well by Zang *et al.,* [7] to examine the impact of various CFD parameters on erosion prediction in contraction expansion including turbulence model, wall treatment, and first layer thickness. They suggested a quick fix for the current CFD model for gas flow cases. Darihaki *et al.,* [8] employed a 3D model to explore erosion in gradual contraction and found that the mesh structure can influence particle tracking. They advised using a circular core mesh to prevent any unphysical phenomena when tracking particles and concluded that the central grid size has less of an impact on erosion than the axial grid. 3D model used as well by Agrawal *et al.,* [9] to investigate the impact of dynamic erosion and shape change on contraction geometry and determined that CFD is a useful tool for predicting changes in erosion due to changes in geometry, particularly for sharp corners.

Although researchers used both 2D and 3D models to study erosion in contraction and expansion geometries, there was no clear preference for specific model based on experimental or numerical studies. Furthermore, different researchers studied the same phenomena using either 2D or 3D models. For example, Prasad *et al.,* [10] compared the flow in sudden contraction and enlargement with and without rounding using 2D models, while Darihaki *et al.,* [11] studied the same effect using a 3D model focusing on the change in erosion magnitude and profile due to the rounding effect.

Up to the author knowledge the work of Tsai *et al.,* [12] was the only reference that conducted a study to evaluate the capabilities and limitations of 2D and 3D CFD models for simulating the flow field in a sudden expansion. They have studied the flow in a microchannel and found that the experimental flow visualization results were consistent with the 3D numerical predictions. They recommended using the 2D simulation method only for predicting flow behavior in sudden expansion microchannels with high aspect ratios, as this was the area where they observed good agreement between the 2D, 3D, and experimental data.

There are some studies that have compared the performance of 2D and 3D models for the field of particle transport, but for different geometries. However, there was no general tendency for which model to use, as some studies found good agreement between the two models, while others found a big discrepancy between their results. For instance; Li *et al.,* [13] investigated the differences between 2D and 3D simulations of several Circulating Fluidized Bed (CFB) risers. The study found that while 2D simulations are widely used due to their lower computational cost, there were significant differences between the results of 2D and 3D simulations. For example, in one case, the 2D simulation under-predicted the pressure gradient compared to the 3D simulation and in another case it overpredict it. They recommended that 2D analysis to be used only for qualitative studies. Upadhyay *et al.,* [14] evaluated the prediction capability of 2D and 3D gas-solid flow simulation in a lab-scale CFB riser section. They found that the 3D simulation can accurately predict the axial solid holdup profile, while the 2D simulation underestimates or overestimates the solid holdup depending on the flow conditions. They also found that the 2D simulation fails to capture the high solid holdup near the riser exit and riser bottom dense region, which are important for erosion prediction. In the contrary, Cammarata *et al.,* [15] for similar study in RFB found the results of both 2D and 3D simulations showed reasonable consistency in terms of bed expansion and fluid bed voidage, indicating the usefulness of 2D simulations for sensitivity analysis. However, 3D simulations were found to be more important as they provided a more realistic representation of the fluidization behavior being investigated. Similarly, Salehi and Rahimi [16] study found that both qualitative and quantitative results on bed expansion and fluid bed voidage showed reasonable consistency between 2D and 3D simulations, thus underpinning the relevance of using 2D simulations for sensitivity analysis. However, the results also showed the major importance of performing 3D simulations as these provided a more realistic physical behavior of the fluidization investigated.

One of the main issues with using 3D CFD models for solid particle erosion in contraction and expansion geometries is the computational cost, as these models require significantly more computational resources than 2D models. However, 3D models can provide more detailed information about the flow field and particle trajectories, which can be important for accurately predicting the erosion rate. The purpose of this paper is to compare 2D and 3D CFD models in predicting flow characteristics in contraction and expansion geometries, with a particular emphasis on the solid particle erosion mainly erosion profile, patterns and magnitude. The results of the study will provide valuable insights into the accuracy of 2D and 3D models for predicting the erosion rate in these geometries, and will highlight the importance of using either of the models for accurate predictions of erosion rate in complex geometries.

2. Methodology

2.1 Problem Formulation

The geometry of the simulated problem is shown in Figure 1. The geometry studied has an inlet section of a length of 22.7 D_1 (Inlet pipe diameter) and contraction section of a length 20 D_1 similar to the original experiment dimensions. The inlet diameter D_1 is 100 mm and the outlet diameter D_2 is 57.7 mm which gives a contraction ratio of 0.576.

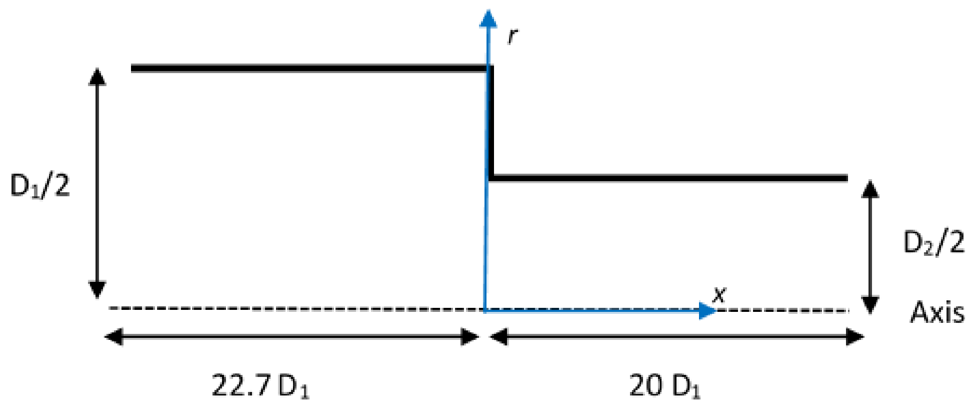

Fig. 1. Schematic of axisymmetric sudden contraction

2.2 2D CFD Model
2.2.1 CFD model

Using the commercial Ansys Fluent software, the current case was numerically solved for an incompressible water flow with a density of 1000 kg/m^3 and a viscosity of 0.001 kg/ms. The pressure-velocity coupling SIMPLE algorithm and the second order upwind discretization approach were chosen. For its well-known performance in internal flows, the realizable K ε turbulence model was utilized, and the standard wall function was used while ensuring that all Y+ are below 100. Regarding the boundary conditions, a uniform velocity V_0 is imposed at the inlet with 3.6% turbulence intensity computed based on the Reynolds number, which was calculated based on the inlet velocity V_0, the inlet diameter D_1, the water density, and viscosity, and found to be 1.54 E^5.At the downstream end of the contraction section, a pressure outlet condition is imposed, and the surfaces of the inlet and outlet pipes are considered as a stationary no-slip smooth wall. Since this is an axisymmetric geometry, only half of the geometry was modeled and an axis boundary was set at the symmetry axis of the geometry.

2.2.2 Grid dependency test

The realizable k ε model was used to assess four mesh configurations for grid dependency at different levels of refinement based on the radial dimension of the cells. Four refinements and meshes were compared M1: 0.1*D_1, M2: 0.05*D_1, M3: 0.025*D_1, and M4: 0.01*D_1. The velocity profile at an axis separated by 0.1*D_1 from the contraction step was utilized to compare the selected meshes. The numerical results, which are represented graphically in Figure 2, demonstrates that mesh refinement beyond grid M3 has no impact; in other words, M3 and M4 nearly perfectly predict the same axial velocity. However, in order to get better results for the current study, M4 was chosen as the appropriate grid resolution which has 84k cells.

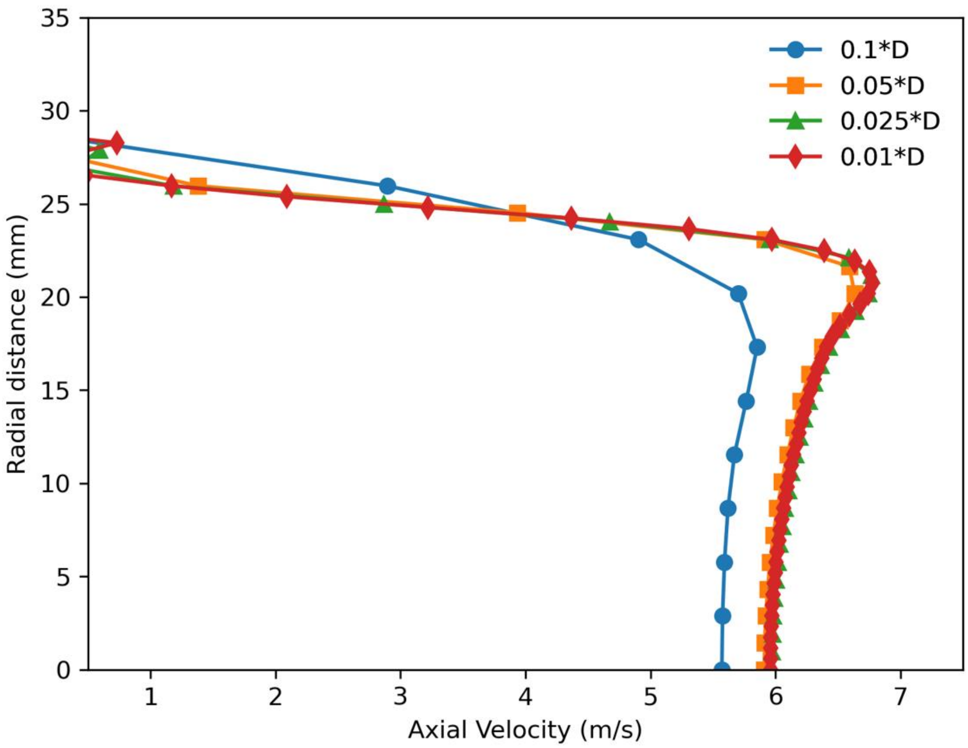

Fig. 2. Grid mesh comparison for dependency test

2.3 3D CFD Model

The same geometry as shown above was modeled using 3D CFD model. Figure 3 shows part of the 3D geometry with the proposed structured mesh generated using the ANSYS ICEM CFD meshing software. Same numerical setup and boundary conditions as the 2D model were used. Similar grid dependency study was conducted and results of a fine mesh of 914K cells which is more than ten times of the 2D mesh in term of grid size

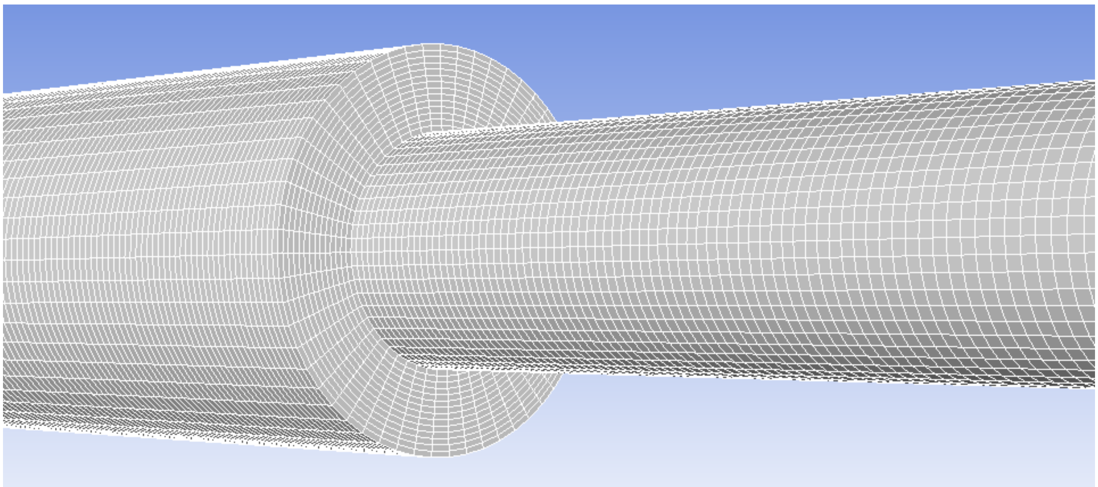

Fig. 3. 3D geometry with structured mesh

3. Results

3.1 CFD Validation for 2D Model

3.1.1 Comparison of the centreline velocity with experimental data

After choosing an appropriate mesh, the centreline velocity which is the velocity at the geometry's symmetry axis was used to compare the numerical results with the experiment data in Ref. [17]. A good agreement between the predicted velocity profile by the CFD and the experimentally acquired velocities can be seen in the plot of the normalized centreline velocity in Figure 4. It should be noted that the majority of the experiment measurement points were located near the contraction step's exit to better understand how velocity changed as the diameter contracts.

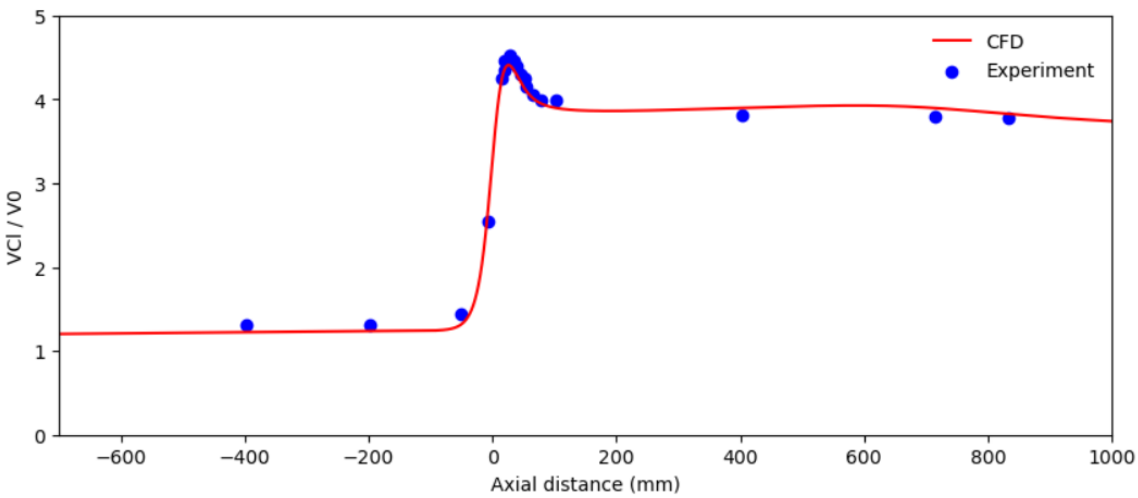

Fig. 4. Comparison of normalized centreline velocity: CFD Vs Experiment

3.1.2 Comparison of axial velocity with experimental data

The predicted CFD velocities and the experiment axial velocity at various places from the contraction step in the inlet and outlet contraction pipe were compared. Positions 1 and 2 are at the inlet pipe, respectively, at a distance of 4D and 0.1D from the contraction step, whereas positions 3 and 4 are within the contraction zone immediately following the contraction step, at a distance of 0.125D and 0.2D. The comparison reveals good agreement between the experiments and the predicted velocity profiles from the CFD 2D model as shown in Figure 5. The fact that some of the mistake is related to the digitalization of the velocity profiles from the original publication should be emphasized.

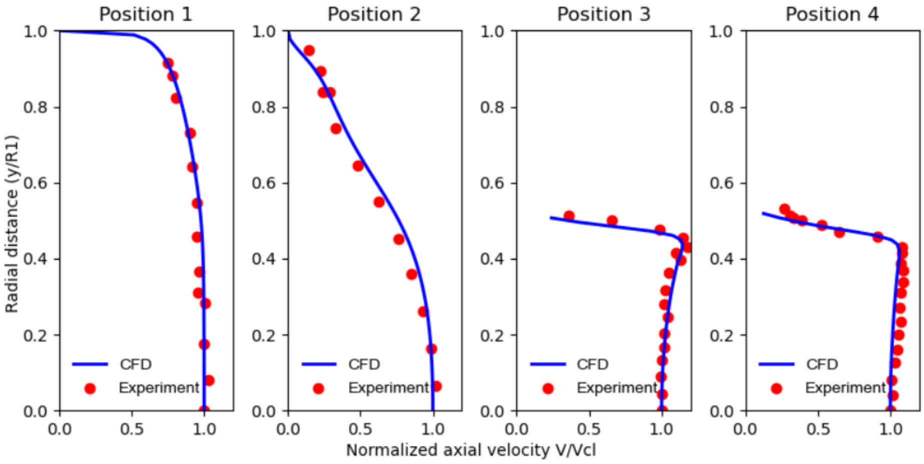

Fig. 5. Comparison of normalized axial velocity CFD Vs experiment

3.2 Comparison between 2D and 3D CFD Models
3.2.1 Velocity analysis

As it had been shown in previous section, the simple 2D simulated geometry gave good results in comparison of real experiment data. To check if there are any differences between the 3D and 2D model the velocity profiles at the previous studied locations were compared for both models and plotted in Figure 6 which shows that both models predict similar velocity profiles except for a slight difference in position 3.

The velocity contours from the 3D model on both the Y and Z planes were shown together to help explain why the axisymmetric 2D model gave very good and comparable results to the 3D model. This similarity as shown in Figure 7, explains why 2D models can present 3D geometries if they are symmetrical.

3.2.2 Erosion prediction

In this section solid particles erosion patterns and magnitude will be compared between the 3D and 2D CFD numerical simulations. Both erosion computation and particle tracking for the two models have been compared using the same geometry used above. A 300-micron solid particle with a density of 2650 kg/m3 that is comparable to sand produced with the main fluid stream in the oil and gas industry was injected at a rate of 1 g/s for the erosion calculation. The Mclaury erosion model was used to calculate erosion, and steel with a density of 7800 kg/m^3 was selected as the wall material.

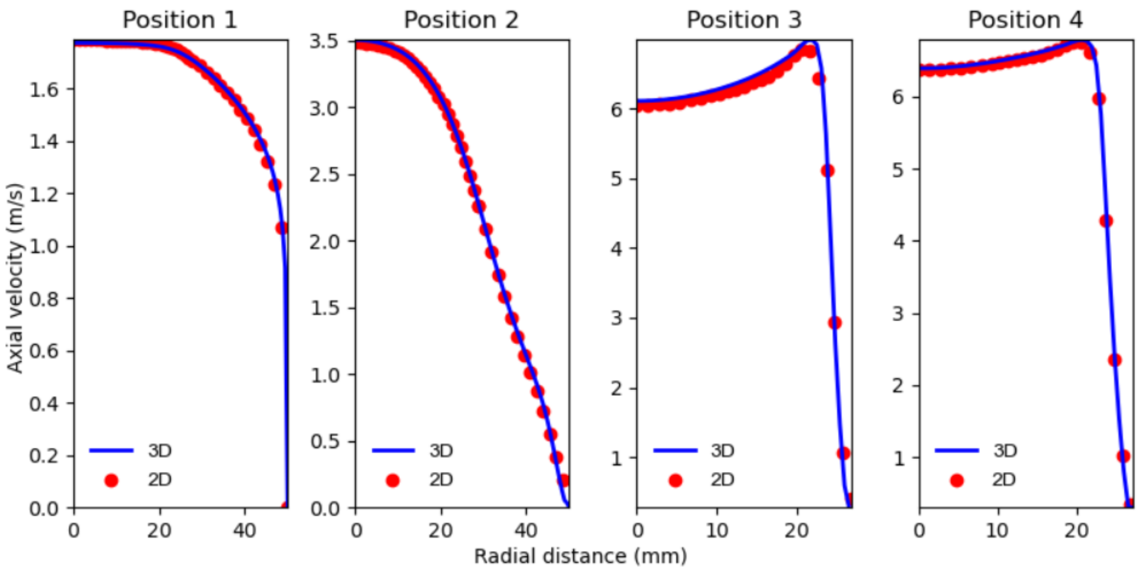

Fig. 6. Axial Velocity comparison 2D CFD model Vs 3D

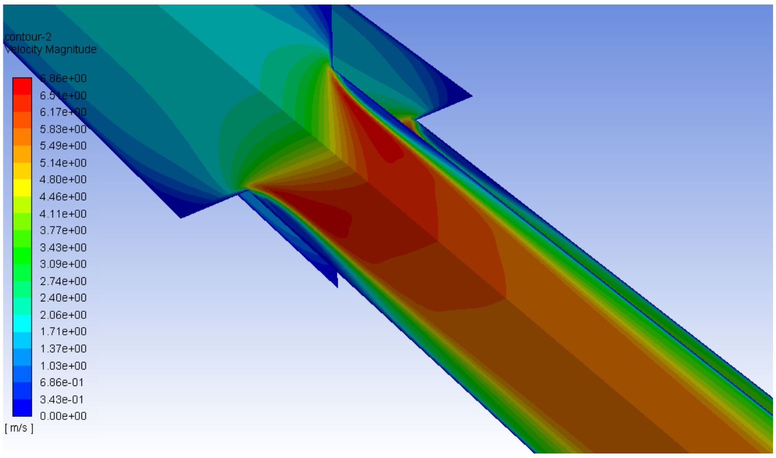

Fig. 7. Velocity contours from 3D model on Y and Z planes

3.2.2.1 Particles tracking

A large number of particles, approximately 100 000 particles, were injected for each model to avoid the effects of the number of particles tracked on erosion. Figure 8 displays an example of particle trajectories for both models which shows that the particle paths observed in the 2D and 3D models are quite similar.

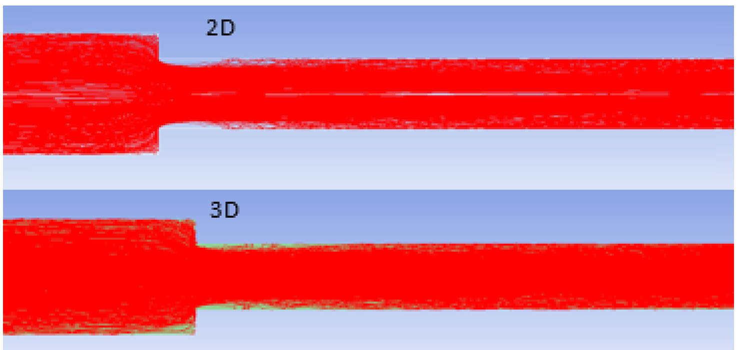

Fig. 8. Particles tracking with 2D and 3D CFD models

3.2.2.2 Erosion at the contraction step

In 3D simulations, erosion patterns are more observable as demonstrated in Figure 9. In the step of the contraction, there are substantial erosion rates that are more than fifty times higher than the surrounding area. With a few small asymmetrical patterns, the contraction step's erosion pattern is nearly symmetric throughout. These symmetric patterns provide additional proof that a 2D model could be used to model axisymmetric geometries.

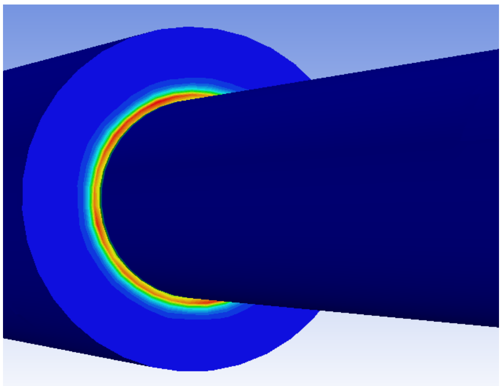

Fig. 9. Erosion pattern at the contraction step (3D model)

To illustrate more on the erosion pattern symmetry on the 3D model, the erosion rate along two radial lines was taken and compared as shown in Figure 10. As it can be seen, the erosion rate in two different radial cuts on the contraction step is almost symmetric.

The comparison between the 2D and 3D erosion rate at the contraction step is plotted in Figure 11 and shows a good agreement in term of pattern as well as max erosion rate recorded at the step. This is another good indication that supports the use of 2D model confidentially to represent erosion at the contraction geometry.

3.2.2.3 Erosion at the contraction outlet pipe

The erosion at the contraction outlet pipe shows slight asymmetric pattern as demonstrated in Figure 12 (a) and (b) which are the front and back views of the geometry.

As opposite to the 2D model where one erosion value will represent all circular region, the 3D model captures all the changes in that section. For illustration, the erosion trend on a circle corresponding to the perimeter of the contraction region at a location of 4D from the contraction step is shown in Figure 13 which demonstrates different erosion rates averaged at 0.027 mm/year.

To illustrate on the symmetry of erosion pattern on the contraction outlet walls, the first and second half of the cylinder were checked against each other and plotted in Figure 14 which shows an almost symmetric erosion pattern between the two halves of the cylinder.

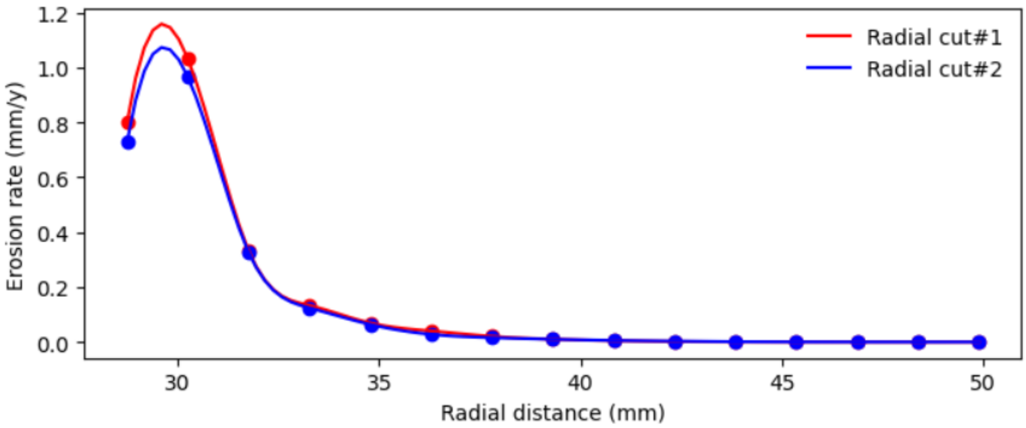

Fig. 10. Erosion pattern symmetry at the contraction step (Comparison between two radial cuts)

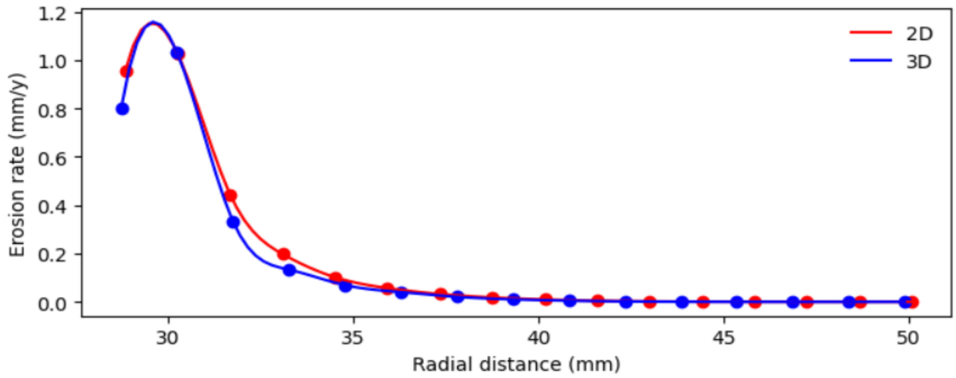

Fig. 11. Erosion comparison at the contraction step (2D Vs 3D CFD model)

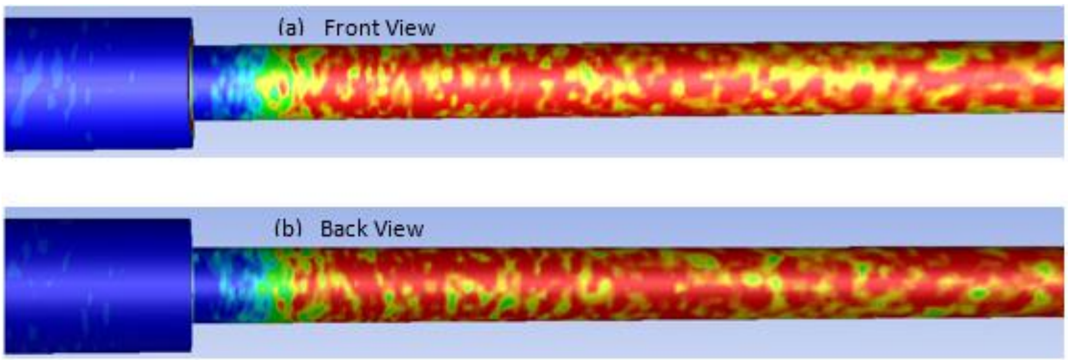

Fig. 12 Erosion pattern at the contraction outlet pipe; (a) front and (b) back view (3D) model)

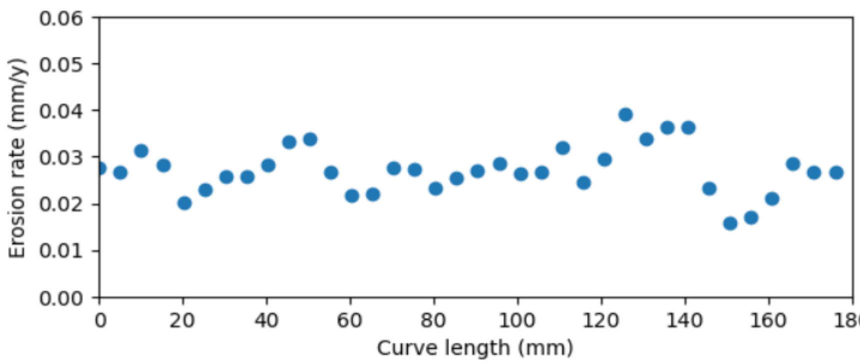

Fig. 13. Erosion pattern at perimeter, 4D from the contraction step (3D model)

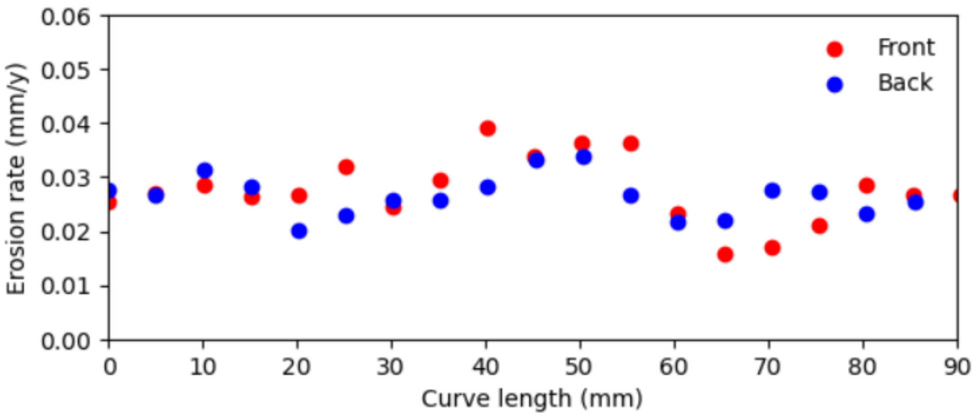

Fig. 14. Erosion pattern symmetry at a perimeter located 4D from the contraction step (front and back of the pipe)

To compare the erosion rate at the contraction outlet and the symmetry of the erosion pattern in the 3D model, the erosion alone two opposite lines on the 3D section were taken and compared against each other and plotted in Figure 15 which shows a good agreement in the erosion magnitude as well as erosion pattern between the two sections which are on opposite direction on the geometry studied.

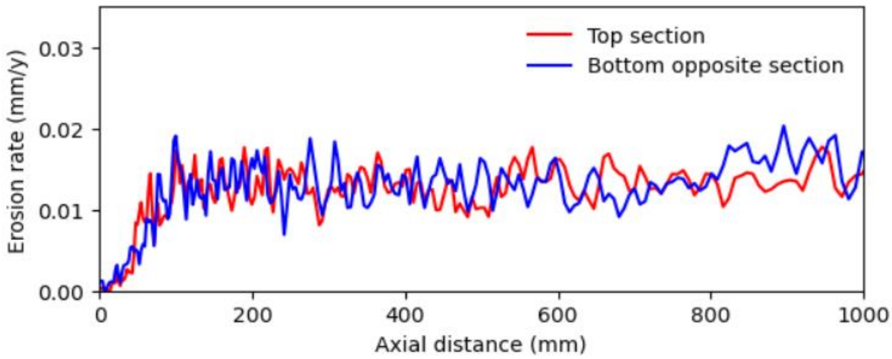

Fig. 15. Erosion pattern at the contraction outlet (top Vs bottom section)

Finally, the 2D and 3D model erosion rates were compared against each other and plotted in Figure 16 which gives a clear indication of the possible use of 2D model to represent 3D axisymmetric geometries as it shows very good agreement between the two models.

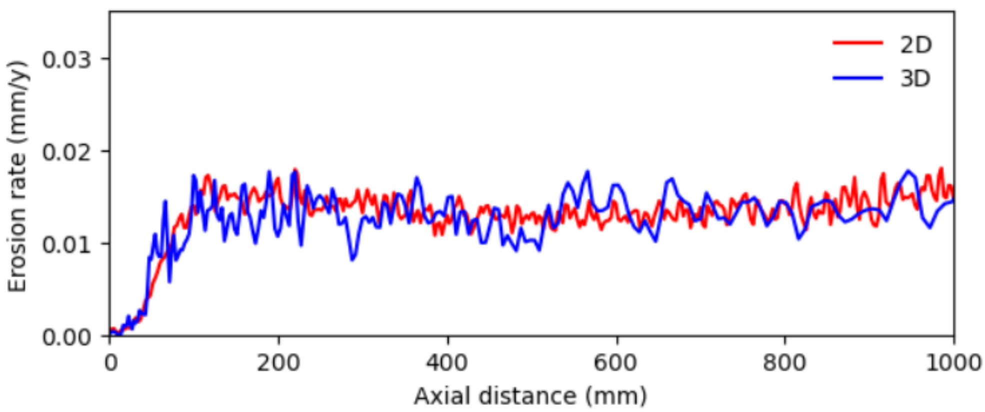

Fig. 16. Erosion comparison at the contraction outlet (2D Vs 3D CFD model)

4. Conclusions

In conclusion, the comparison of 2D and 3D CFD models in contraction and expansion geometries shows that 2D model can provide accurate predictions of flow and erosion characteristics as compared to the 3D model and experimental data. However, 3D CFD simulations provide more detailed information about erosion patterns, which can be important in some applications.

When simulating symmetric shapes, 2D axisymmetric modelling can be used to simplify the computational domain and utilize less computing power, but it ignores any potential 3D effects. A more exact simulation is provided by 3D modelling, but this takes more computing power. The decision between the two methods is based on the amount of simulation accuracy necessary and the available processing resources.

References

[1] Abdulla, Akar. "Estimating erosion in oil and gas pipe line due to sand presence." (2011).

[2] Xu, Yan, Zunce Wang, Sen Li, Fengxia Lv, Yuejuan Yan, and Houzhen Wen. "Analysis of Erosion and Failure in the Sudden Expansion Fracturing Tubing of Deep Gas Wells." In *International Conference on Offshore Mechanics and Arctic Engineering*, vol. 49149, pp. 293-299. 2010. https://doi.org/10.1115/OMAE2010-21049

[3] Cheng, Jiarui, Yihua Dou, Jiding Zhang, Ningsheng Zhang, Zhen Li, and Zhiguo Wang. "Experimental Study on Particle Erosion Failure of Abrupt Pipe Contraction in Hydraulic Fracturing." *Journal of Failure Analysis and Prevention* 18 (2018): 382-391. https://doi.org/10.1007/s11668-018-0409-5

[4] Duarte, Carlos Antonio Ribeiro, and Francisco José de Souza. "Dynamic mesh approaches for eroded shape predictions." *Wear* 484 (2021): 203438. https://doi.org/10.1016/j.wear.2020.203438

[5] Darihaki, Farzin, Ebrahim Hajidavalloo, Amir Ghasemzadeh, and Gholam Abbas Safian. "A localized sand erosion prediction approach for multiphase flow in wells: application for sudden-expansions." *Particulate Science and Technology* 39, no. 8 (2021): 954-970. https://doi.org/10.1080/02726351.2021.1871990

[6] Li, Qi. "Erosion Prediction on Contractions and Expansions Based on Computational Fluid Dynamics (CFD)." PhD diss., University of Tulsa, 2015.

[7] Zhang, Jun, Farzin Darihaki, Yeshwanthraj Rajkumar, Soroor Karimi, and Siamack A. Shirazi. "Erosion in Sudden Contractions and Expansions." In *NACE CORROSION*, pp. NACE-2019. NACE, 2019. https://doi.org/10.1016/j.wear.2019.01.106

[8] Darihaki, Farzin, Jun Zhang, and Siamack A. Shirazi. "Solid particle erosion in gradual contraction geometry for a gas-solid system." *Wear* 426 (2019): 643-651. https://doi.org/10.1016/j.wear.2019.01.106

[9] Agrawal, Madhusuden, Samir Khanna, Ardjan Kopliku, and Tim Lockett. "Prediction of sand erosion in CFD with dynamically deforming pipe geometry and implementing proper treatment of turbulence dispersion in particle tracking." *Wear* 426 (2019): 596-604. https://doi.org/10.1016/j.wear.2019.01.018

[10] Prasad, S. S., G. Satish, and G. Panduranga. "Comparison of Flow Analysis through Sudden Contraction and Enlargement of Pipes by Providing Smooth Corners." *Int. J. Eng. Trends Technol* 25 (2015): 205-211. https://doi.org/10.14445/22315381/IJETT-V25P238

[11] Darihaki, Farzin, Ebrahim Hajidavalloo, Amir Ghasemzadeh, and Gholam Abbas Safian. "Erosion prediction for slurry flow in choke geometry." *Wear* 372 (2017): 42-53. https://doi.org/10.1016/j.wear.2016.12.008

[12] Tsai, Chien-Hsiung, Han-Taw Chen, Yao-Nan Wang, Che-Hsin Lin, and Lung-Ming Fu. "Capabilities and limitations of 2-dimensional and 3-dimensional numerical methods in modeling the fluid flow in sudden expansion microchannels." *Microfluidics and Nanofluidics* 3 (2007): 13-18. https://doi.org/10.1007/s10404-006-0099-2

[13] Li, Tingwen, Sreekanth Pannala, and Mehrdad Shahnam. "2D versus 3D CFD Simulations of Circulating Fluidized Bed Risers." In Multiphase Flow Workshop, 2012. https://doi.org/10.1016/j.powtec.2014.04.008

[14] Upadhyay, Mukesh, Myung Won Seo, Parlikkad Rajan Naren, Jong-Ho Park, Thanh Dang Binh Nguyen, Kashif Rashid, and Hankwon Lim. "Experiment and multiphase CFD simulation of gas-solid flow in a CFB reactor at various operating conditions: Assessing the performance of 2D and 3D simulations." *Korean Journal of Chemical Engineering* 37 (2020): 2094-2103. https://doi.org/10.1007/s11814-020-0646-7

[15] Cammarata, Luca, Paola Lettieri, Giorgio DM Micale, and Derek Colman. "2D and 3D CFD simulations of bubbling fluidized beds using Eulerian-Eulerian models." *International Journal of Chemical Reactor Engineering* 1, no. 1 (2003). https://doi.org/10.2202/1542-6580.1083

[16] Salehi, M. A., and R. Rahimi. "COMPARISON BETWEEN 2D AND 3D TRANSIENT FLOW SIMULATION OF GAS–LIQUID DYNAMICS IN TWO-PHASE CYLINDRICAL BUBBLE COLUMN REACTORS BY CFD." *Int. J. Chem. Sci.* 6 (2009): 857-872.

[17] Bullen, P. R., D. J. Cheeseman, and L. A. Hussain. "A study of turbulent flow in pipe contractions." *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering* 210, no. 3 (1996): 171-180. https://doi.org/10.1243/PIME_PROC_1996_210_312_02