CHEMICAL ENGINEERING TRANSACTIONS
VOL. 113, 2024

A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.cetjournal.it

Guest Editors: Jeng Shiun Lim, Bohong Wang, Guo Ren Mong, Petar S. Varbanov

ISBN 979-12-81206-14-4; **ISSN** 2283-9216

DOI: 10.3303/CET24113109

RANS Equation-Based Gas Cyclone Separator CFD Simulation: an Appropriate Time Step Size

Piyawut Thongnoi[a], Walairat Chandra-ambhorn[a], Benjapon Chalermsinsuwan[b], Santi Wattananusorn[a,*], Patthranit Wongpromrat[a], Eakarach Bumrungthaichaichan[a]

[a]Department of Chemical Engineering, School of Engineering, King Mongkut's Institute of Technology Ladkrabang, Bangkok 10520, Thailand
[b]Department of Chemical Technology, Faculty of Science, Chulalongkorn University, Pathumwan, Bangkok 10330, Thailand
santi.wa@kmitl.ac.th

The present work answers a question for Reynolds-Averaged Navier-Stokes (RANS) equation-based gas cyclone separator computational fluid dynamics (CFD) simulation: "How small is the appropriate time step size?". The gas cyclone separator CFD models were developed with appropriate near-wall grid sizes and the proper numerical scheme set suggested by our previous works. The quadratic pressure-strain Reynolds stress turbulence model was used. The mean velocity profiles simulated by time step sizes of 0.0001 s, 0.0005 s, 0.00075 s, and 0.001 s were compared to the reference experimental data. Moreover, the hit-rate validation and statistical analysis were assessed to achieve the guideline for specifying the proper time step size. The results revealed that the time step sizes of 0.0001 s and 0.0005 s provided better mean axial velocity prediction accuracy than others. From hit-rate validation and statistical analysis, it can be finally concluded that the time step size smaller than 1/1470 of the residence time is essential for predicting mean flow inside the gas cyclone separator.

1. Introduction

Computational fluid dynamics (CFD) is an important method for studying gas cyclone separators because it can predict reliable flow phenomena with inexpensive investigation costs. Due to these benefits, many researchers have employed CFD to improve gas cyclone separator performances, including pressure drop and collection efficiency. The prediction accuracy of gas cyclone separator CFD simulations depends on the developed CFD models. Therefore, various CFD modelling parameters, such as grid, physical models, spatial discretization methods, etc., were tested to obtain the appropriate gas cyclone separator CFD model.

Bumrungthaichaichan (2022) introduced the near-wall grid size estimation method for gas cyclone separator CFD simulations by considering five gas cyclone separator geometries. All gas cyclone separator CFD models were developed by the Reynolds stress model (RSM) with linear pressure-strain sub-model. The simulated results confirmed that the proposed near-wall grid size estimation method enhanced the mean flow prediction accuracy because the near-wall computing nodes were in the valid region of the wall functions.

Later, in 2023, Bumrungthaichaichan (2023) investigated the suitable numerical schemes for gas cyclone separator and hydrocyclone CFD simulations by considering two numerical scheme sets, including QUICK numerical scheme (QNS) set and mixed numerical scheme (MNS) set. A comparison between these numerical scheme sets was presented in Bumrungthaichaichan (2023). The results indicated that only QNS could preserve mean flow similarity inside six gas cyclone separators. Moreover, the mean flow prediction accuracy of gas cyclone separator CFD simulations was improved when the RSM with quadratic pressure-strain sub-model was adopted.

At first sight, information on physical and numerical modellings suggested by Bumrungthaichaichan (2022, 2023) is seemingly adequate to develop the appropriate Reynolds-Averaged Navier-Stokes (RANS) equation-based gas cyclone separator CFD model. However, a criterion for setting the time step size of transient CFD simulations of gas cyclone separators is still unclear. To the best of our knowledge, we know that the time step

Paper Received: 30 August 2024; Revised: 17 September 2024; Accepted: 24 October 2024
Please cite this article as: Thongnoi P., Chandra-Ambhorn W., Chalermsinsuwan B., Wattananusorn S., Wongpromrat P., Bumrungthaichaichan E., 2024, RANS Equation-Based Gas Cyclone Separator CFD Simulation: an Appropriate Time Step Size, Chemical Engineering Transactions, 113, 649-654 DOI:10.3303/CET24113109

sizes for RANS equation-based gas cyclone separator CFD simulations should be a tiny fraction of the residence time (Chuah et al., 2006). The residence time of a gas cyclone separator is generally defined as a ratio of gas cyclone separator volume to volumetric flow rate. Here, one question arises: "How small is the proper time step size of RANS equation-based gas cyclone separator CFD simulation?". Therefore, the suitable time step size investigation is another concerning issue for gas cyclone separator CFD simulation that is worth further study.
In the present work, the mean velocities, including mean tangential velocity (U_t) and mean axial velocity (U_z), simulated by time step sizes of 0.0001 s, 0.0005 s, 0.00075 s, and 0.001 s, which approximately correspond to 1/7380, 1/1470, 1/980, and 1/738 of the residence time of the gas cyclone separator (0.7354 s), were compared to the experimental data of 0.29 m-diameter Stairmand gas cyclone separator of Hoekstra (2000) to investigate the appropriate time step size for RANS equation-based gas cyclone separator simulation. It is noted that for the present work, the normalized mean tangential velocity (U_t/U_{in}) and normalized mean axial velocity (U_z/U_{in}) profiles (where U_{in} is the inlet velocity) for the time step size of 0.0001 s simulated by fine grid resolution (756,788 cells) reported by Bumrungthaichaichan (2023) were employed for investigating the proper time step size. The time step size of 0.0001s was commonly used for simulating a 0.29 m-diameter Stairmand gas cyclone separator as reported by the previous works (Shukla et al., 2011; Pechmanee et al., 2021). In order to warrant fair comparisons between the present CFD results and the previous work of Bumrungthaichaichan (2023), the CFD modelling of the present work was identical to that of Bumrungthaichaichan (2023). The complete details of the present gas cyclone separator CFD model are described in section 2.

2. Gas cyclone separator CFD modelling

2.1 Gas cyclone separator and its operating conditions

In this paper, a 0.29 m-diameter Stairmand gas cyclone separator with a 0.58 m-height dustbin of Hoekstra (2000), which is an industrial scale, i.e., cyclone barrel diameter (D_b) = 0.25−1.5 m (Hoekstra, 2000), was considered. The dimensions of the considered gas cyclone separator are given in Figure 1. The air density (ρ) and viscosity (μ) were 1.096 kg/m^3 and 1.81623×10^{-5} kg/(m·s), respectively. The air inlet velocity was 16 m/s, which corresponds to the cyclone barrel diameter-based Reynolds number ($Re_{Db} = D_b U_{in} \rho/\mu$) of 2.8×10^5.

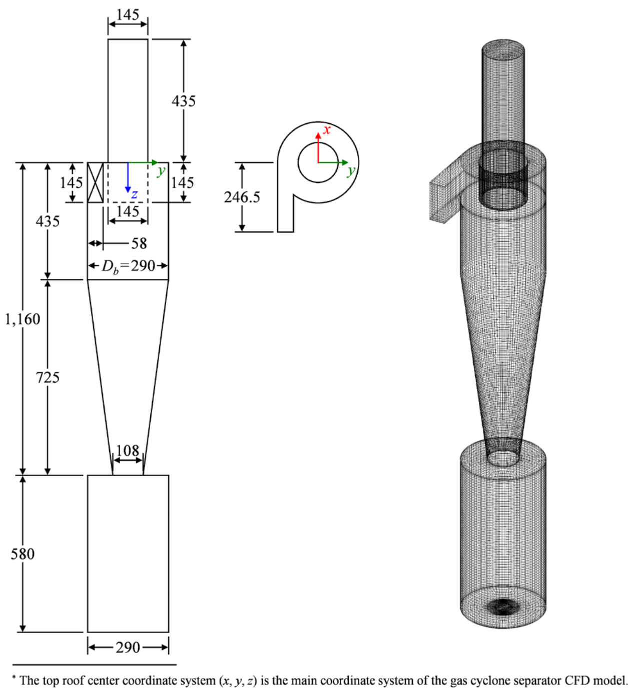

Figure 1: Gas cyclone separator schematic (Adapted from Bumrungthaichaichan (2023)) and grid generation

2.2 Computational domain and grid generation

The computational domain of the 0.29 m-diameter Stairmand gas cyclone separator was manually divided into several blocks to contain hexahedral grids generally used to prevent truncation errors. The near-wall grid sizes and grid qualities were identical to those of the previous works of Bumrungthaichaichan (2022, 2023). The grid generation of the considered gas cyclone separator is also shown in Figure 1.

2.3 Governing equations

The Reynolds-Averaged Navier-Stokes equations were resolved to obtain the mean flow fields inside the gas cyclone separator. The RSM turbulence model with quadratic pressure-strain sub-model was adopted as suggested by the previous work of Bumrungthaichaichan (2023). Many previous works have reported these governing equations; therefore, they were not repeated in this work.

2.4 Physical and numerical modellings

In order to warrant fair result comparisons, the present CFD model setups of ANSYS FLUENT were identical to those of Bumrungthaichaichan (2023) and can be described as follows. At the inlet, the velocity-inlet boundary condition type and uniform inlet velocity of 16 m/s were specified. The turbulence kinetic energy of 0.5845 m^2/s^2 and turbulence dissipation rate of 18.0865 m^2/s^3 were also imposed at the inlet section. Furthermore, the normal Reynolds stresses of 0.3897 m^2/s^2 and zero shear Reynolds stresses were set at the inlet. The outflow boundary condition type with a flow rate weighting of 1 was specified at the gas cyclone separator outlet. At the wall boundaries, the no-slip boundary condition was used. The scalable wall functions were employed for near-wall treatment.

From the QUICK numerical scheme set suggested by Bumrungthaichaichan (2023), the pressure-velocity coupling scheme was the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm. The discretization scheme for pressure was PRESTO! (Pressure Staggering Option). Other spatial discretization schemes were QUICK (Quadratic Upstream Interpolation for Convective Kinematics). The second-order implicit temporal discretization scheme was adopted. Scaled residuals of 10^{-5} were employed for all transport equations. The tangential velocity at the point 0.16 m below the cyclone top roof center (0.015 m below the vortex finder) was considered to ensure that the flow fields inside the gas cyclone separator exhibited semi-periodic behaviors. The normalized mean tangential velocities predicted by different time step sizes are represented in Figure 2.

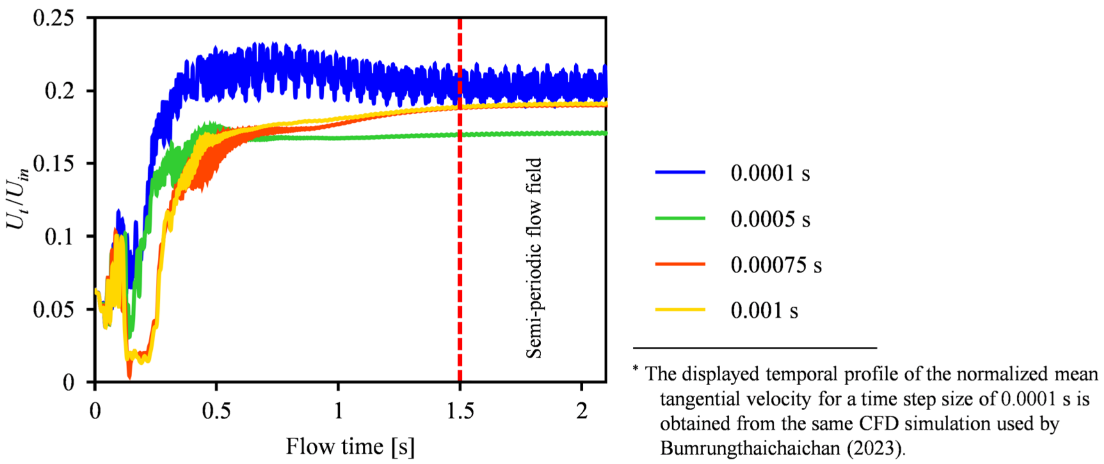

Figure 2: Normalized mean tangential velocities at the point 0.16 m below the cyclone top roof center predicted by different time step sizes

In Figure 2, the predicted results of different time step sizes reveal that the temporal profiles of the normalized local tangential velocity for these simulations are different. The semi-periodic flow fields of all CFD simulations can be observed at a flow time of approximately 1.5 s. Hence, the time-averaged flow properties obtained by flow time ≥2 s of all cases were appropriate for considering the suitable time step size for RANS equation-based gas cyclone separator CFD simulation because of the obtained semi-periodic flow fields.

2.5 Grid independence study and model validation

For grid independent solution study, the normalized mean tangential velocity and normalized mean axial velocity profiles at $z = 2D_b$ for a time step size of 0.0001 s simulated by coarse (245,356 cells) and medium (426,552 cells) grid levels of the present work were compared to those of fine grid level reported by Bumrungthaichaichan (2023) as shown in Figure 3. It is noted that the total number of cells for three different grid levels was identical to the previous works of Bumrungthaichaichan (2022, 2023). Simultaneously, the results predicted by three grid resolutions were also compared to the experimental data of Hoekstra (2000) to achieve model validation as represented in Figure 3. Moreover, the discretization error bars on normalized mean velocity profiles were obtained by grid convergence index to represent numerical uncertainty as depicted in Figure 3.

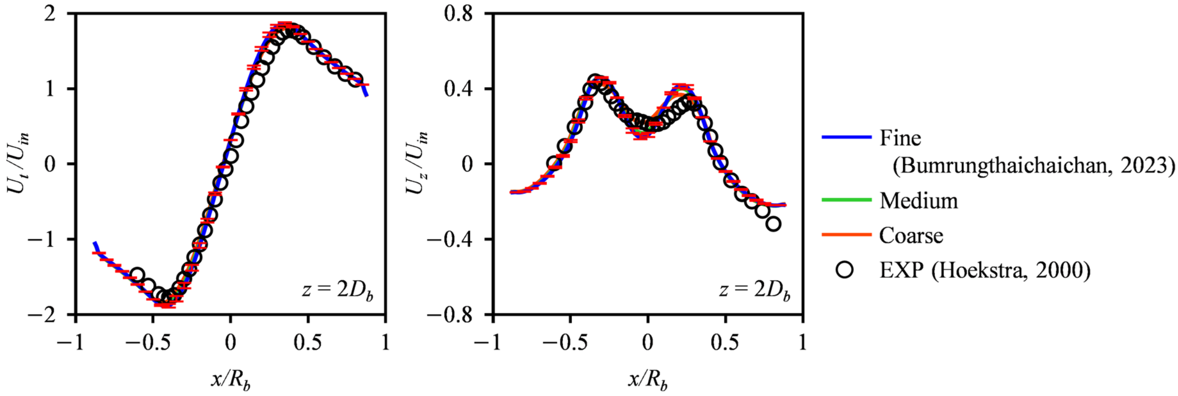

Figure 3: Profiles of (left) normalized mean tangential velocity and (right) normalized mean axial velocity obtained by CFD simulations of three different grid resolutions and experimental data of Hoekstra (2000)

Figure 3 reveals that the normalized mean velocity profiles of three different grid resolutions are similar, except for the normalized mean axial velocity profile simulated by coarse grid level. Furthermore, the normalized mean velocity profiles predicted by the medium grid level of the present work and the fine grid level of Bumrungthaichaichan (2023) are in good agreement with the measured profiles of Hoekstra (2000), especially for the normalized mean tangential velocity.
According to these results, it can be summarized that grid independence is obtained by the medium grid level. It is noted that the grid independence summary of this work is similar to the grid independence study summaries of linear pressure-strain RSM-based gas cyclone separator CFD models co-operated with MNS (Bumrungthaichaichan, 2022) and QNS (Bumrungthaichaichan, 2023). Moreover, from model validation, the results indicated that the quadratic pressure-strain RSM-based gas cyclone separator CFD model co-operated with QNS and medium grid level is adequate for predicting mean velocities inside the gas cyclone separator. However, in order to prevent any uncertainties, the fine grid level (756,788 cells) was selected to investigate the proper time step size for simulating the gas cyclone separator.

3. Results and discussion

In order to investigate the appropriate time step size for RANS equation-based gas cyclone separator CFD simulation, the normalized mean velocity profiles at $z = 2D_b$ obtained by CFD simulations were compared to those measured by Hoekstra (2000) as shown in Figure 4. The hit-rate (q) validation test, which was successfully used by Schlünzen et al. (2004) for analyzing prognostic microscale wind field models, was also applied for normalized mean axial velocity. In addition, for statistical analysis, the normalized mean square error (NMSE), fractional bias (FB), and correlation coefficient (R) were reported as shown in Figure 5, similar to the previous work for airflow over an array of cubes of Santiago et al. (2007). Furthermore, the relative computing performance was considered as depicted in Figure 6. These results are represented in the latter sub-sections.

3.1 Mean velocities

Figure 4 reveals that the predicted normalized mean tangential velocity profiles are similar and agree well with the measured data of Hoekstra (2000). For normalized mean axial velocity, the predicted profiles are different but exhibit M-shaped profiles similar to the experimental data of Hoekstra (2000). The profile peaks simulated by time step sizes of 0.0001 s and 0.0005 s are broader and in better agreement with the previous work of Hoekstra (2000) than other CFD simulations. Here, it can be stated that all CFD simulations can preserve the flow similarity of mean velocity components. The discrepancy in the normalized mean axial velocity between CFD simulations and experimental data of Hoekstra (2000) is more obvious than that of the normalized mean tangential velocity; therefore, the hit-rate validation and statistical analysis for normalized mean axial velocity were only assessed in sub-section 3.2.

3.2 Hit-rate and statistical analysis

In this work, the relative deviation (RD) of 0.25 and absolute deviation (AD) of 0.64 m/s, which was computed by inlet turbulence intensity (4%) times inlet velocity, were used to obtain hit-rates for different time step sizes. In Figure 5, the hit-rates of time step sizes of 0.0001 s and 0.0005 s are greater than the limit for hit-rate validation, i.e., the hit-rate is greater than 0.66, given by Schlünzen et al. (2004) because of low Courant numbers of 0.32 and 1.58 for time step sizes of 0.0001 s and 0.0005 s, respectively. From NMSE values in

Figure 5, the lower NMSE values of time step sizes of 0.0001 s and 0.0005 s indicate that these models show higher accuracy in mean axial velocity prediction than others. The negative values of FB in Figure 5 reveal that all CFD models overpredicted mean axial velocities. Moreover, in Figure 5, correlation coefficients for different time step sizes are greater than 0.9. These obtained correlation coefficients confirm very strong correlation between the predicted and measured results (Schober et al., 2018).

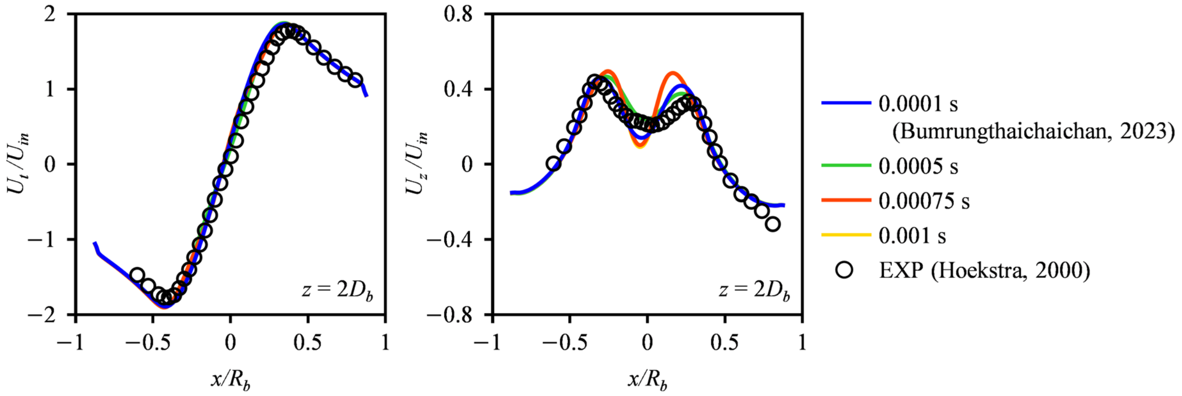

Figure 4: Profiles of (left) normalized mean tangential velocity and (right) normalized mean axial velocity obtained by CFD simulations for different time step sizes and experimental data of Hoekstra (2000)

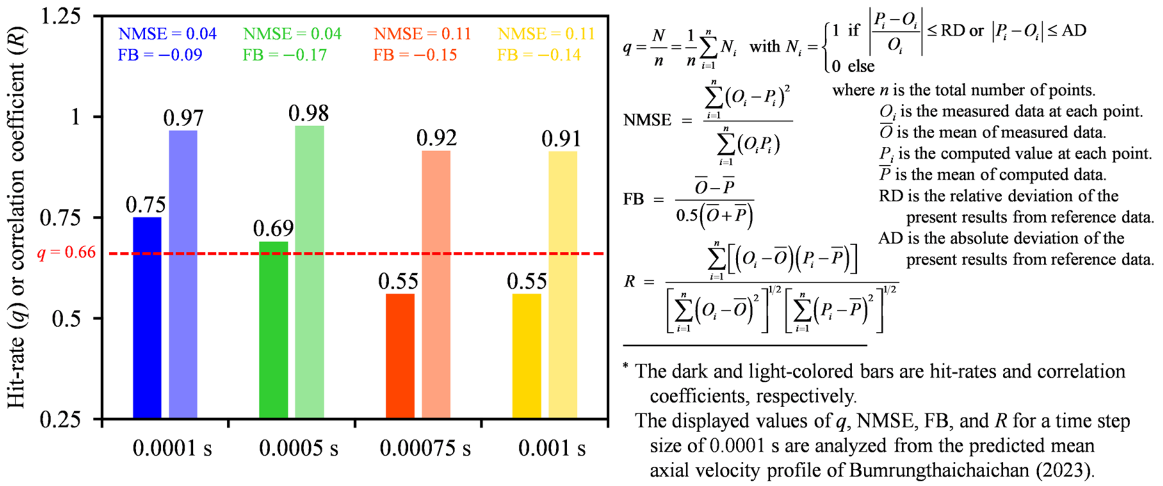

Figure 5: Hit-rates, normalized mean square errors, fractional bias values, and correlation coefficients of mean axial velocity for different time step sizes

3.3 Relative computing performance

In the present work, the computational times for different time step sizes were also considered using relative computing performance as suggested by Bumrungthaichaichan (2023). The relative computing performance was adopted to represent the speed-up simulations with time step sizes <0.0001 s as shown in Figure 6. From Figure 6, it can be seen that the time step sizes of 0.0005 s and 0.00075 s show speed-up simulations with relative computing performance of 1.51 and 1.87, respectively. Surprisingly, the time step size of 0.001 s can speed up the simulation to the relative computing performance of 9.42.

When these results were considered all at once, although the time step size of 0.001 s can significantly reduce the computational time, this time step size provides low hit-rate and NMSE values. In general, the hit-rate validation is obtained when the hit-rate is greater than 0.66. From this criterion, the time step sizes of 0.0001 s and 0.0005 s provide hit-rate validations. Moreover, although the statistical analyses of these time step sizes are insignificantly different, the hit-rate of time step size of 0.0001 s is 8% higher than that of time step size of 0.0005 s. According to this discussion, the proper time step size should be smaller than 1/1470 of the residence time of the gas cyclone separator, which can be considered as the guideline for specifying the appropriate time step size for RANS equation-based gas cyclone separator CFD simulation.

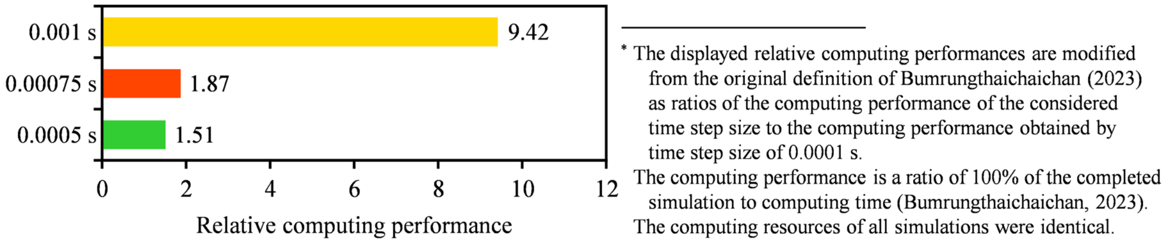

Figure 6: Relative computing performances for time step sizes of 0.0005 s, 0.00075 s, and 0.001 s

4. Conclusions

In this work, the question for gas cyclone separator CFD simulation: "How small is the appropriate time step size?" has been answered by comparing the simulated normalized mean velocity profiles of four different time step sizes with experimental data of Hoekstra (2000) and considering the hit-rate and statistical analysis. Due to the appropriate near-wall grid sizes, suitable numerical scheme set, and quadratic pressure-strain Reynolds stress turbulence model, all CFD models developed by four different time step sizes preserve the flow similarity of the mean velocity components. From hit-rate and statistical analyses, it can be finally concluded that the time step size of <1/1470 of the gas cyclone separator's residence time is essential for simulating the gas cyclone separator.

Nomenclature

D_b – barrel diameter, m
R_b – barrel radius, m
Re_{Db} – barrel diameter-based Reynolds number, -
U_{in} – inlet velocity, m/s
U_t – mean tangential velocity, m/s
U_z – mean axial velocity, m/s
x, y, z – Cartesian coordinates
μ – fluid viscosity, kg/(m·s)
ρ – fluid density, kg/m^3

Acknowledgments

This work was supported by King Mongkut's Institute of Technology Ladkrabang (Grant no.: 2565-02-01-023).

References

Bumrungthaichaichan E., 2022, How can the appropriate near-wall grid size for gas cyclone CFD simulation be estimated?, Powder Technology, 396, 327–344.

Bumrungthaichaichan E., 2023, A note of caution on numerical scheme selection: Evidence from cyclone separator CFD simulations with appropriate near-wall grid sizes, Powder Technology, 427, 118713.

Chuah T.G., Gimbun J., Choong T.S.Y., 2006, A CFD study of the effect of cone dimensions on sampling aerocyclones performance and hydrodynamics, Powder Technology, 162(2), 126–132.

Hoekstra A.J., 2000, Gas flow field and collection efficiency of cyclone separators, PhD Thesis, Delft University of Technology, Netherlands.

Pechmanee P., Namkanisorn A., Wattananusorn S., Bumrungthaichaichan E., 2021, CFD simulations of high efficiency gas cyclones: An influence of dustbin geometry, Computer Aided Chemical Engineering, 50, 529–534.

Santiago J.L., Martilli A., Martín F., 2007, CFD simulation of airflow over a regular array of cubes. Part I: Three-dimensional simulation of the flow and validation with wind-tunnel measurements, Boundary-Layer Meteorology, 122, 609–634.

Schlünzen K.H., Baechlin W., Brünger H., Eichhorn J., Grawe D., Schenk R., Winkler C., 2004, An evaluation guideline for prognostic microscale wind field models. In: 9th International Conference on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Garmisch-Partenkirchen, June 1–4, Germany.

Schober P., Boer C., Schwarte L.A., 2018, Correlation coefficients: Appropriate use and interpretation, Anesthesia & Analgesia, 126(5), 1763–1768.

Shukla S.K., Shukla P., Ghosh P., 2011, Evaluation of numerical schemes using different simulation methods for the continuous phase modeling of cyclone separators, Advanced Powder Technology, 22(2), 209–219.