Article

Design of a Cyclone Separator Critical Diameter Model based on a Machine Learning and CFD

Donggeun Park [1], Jeung Sang Go [2,*]

[1] Department of Advanced Materials and Parts of Transportation Systems, Pusan National University, 2, Busandaehak-ro 63beon-gil, Geumjeong-gu, Busan, 46241, Rep. of Korea; dgpark94@pusan.ac.kr
[2] School of Mechanical Engineering, Pusan National University, 2, Busandaehak-ro 63beon-gil, Geumjeong-gu, Busan, 46241, Rep. of Korea; microd@pusan.ac.kr
* Correspondence: microd@pusan.ac.kr; (please send review result as follow that : dgpark94@pusan.ac.kr)

Abstract: This paper deals with the characteristics of the cyclone separator from the Lagrangian perspective to design important dependent variables, develops a neural network model for predicting the separation performance parameter, and compares the predictive performance between the traditional surrogate model and the neural network model. In order to design the important parameters of the cyclone separator based on the particle separation theory, the force acting until the particles are separated was calculated using the Lagrangian-based CFD methodology. As a result, it was proved that the centrifugal force and drag acting on the critical diameter having a separation efficiency of 50% were similar, and the particle separation phenomenon in the cyclone occurred from the critical diameter, and it was set as an important dependent variable. For developing a critical diameter prediction model based on machine learning and multiple regression methods, Unsteady-RANS analyzes according to shape dimensions were performed. The input design variables for predicting the critical diameter were selected as four geometry parameters that affect the turbulent flow inside the cyclone. As a result of comparing the model prediction performances, the ML model showed the 32.5 % of improvement rate of R^2 compared to the traditional MLR considering the nonlinear relationship between the cyclone design variable and the critical diameter. The proposed techniques have proven to be fast and practical tools for cyclone design.

Keywords: Cyclone separator; Computational fluid dynamics (CFD); Machine learning; Unsteady RANS; Critical Diameter;

1. Introduction

Cyclone separators with cheap and high separation performance have been mainly used to reduce emission generated in industrial and manufacturing processes. The cyclone flow has an outer flow that rotates along the wall to the bottom of the dust container and an inner flow that is reversed at the end of the dust container and discharged to the cyclone air outlet as shown Figure 1. The cyclone separates the contaminant particles by acting flow shear force in the outer flow and inner flow. The centrifugal force and drag force by the turbulence flow act on the particles injected at the cyclone inlet. The centrifugal force pushes the particle to the wall (the outer flow region) and the drag force pushes the particle to the cyclone center (inner flow region). In other words, when the centrifugal force acting on the particles is greater than the drag force, the particles are separated. In order to increase the separation performance of the cyclone, it is necessary to design the cyclone shape so that the centrifugal force acts more than the drag force on the particles of the smallest diameter possible. The turbulent behavior of cyclone is primarily influence by the size of the cyclone shape.

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 November 2020 doi:10.20944/preprints202011.0187.v1

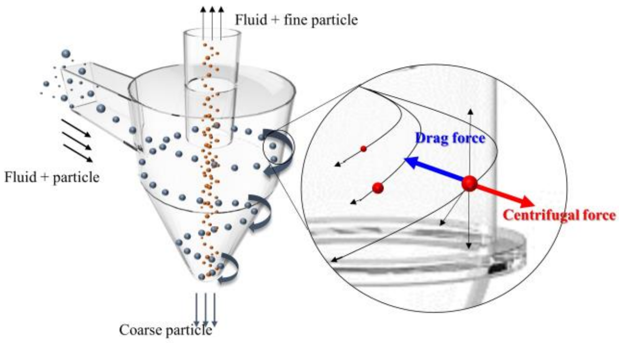


Figure 1. Geometry of the cyclone separator

Therefore, many studies have been conducted over the past decades to optimize the separation performance according to the shape of the cyclone. Early researchers developed an empirical equation based on the experimental values of particle separation according to various shape changes and the equation derived by the conservation law [1-3]. However, since the empirical equations are based on experimental values with uncertainty, the reproducibility of the results is poor.

With the rapid development of computing speed and numerical analysis techniques, many studies were conducted to predict cyclone flow by solving RANS equations based on commercial computational fluid dynamic (CFD) codes. The CFD studies have investigated the effect on separation performance by independently setting several cyclone geometric design variables [4-10]. For example, the separation efficiency and pressure drop were evaluated according to the inlet shape by using CFD [6], and the cyclone performance investigated according to the relationship between the outlet shape and the shape of the dust container [8,9]. The CFD method was used to analyze internal flow characteristics that cannot be derived through experiments. However, Because of the independent design variables, the design space is considered less and there is a possibility to derive local optimization. In addition, it takes a lot of computational cost to obtain the cyclone separation performance according to various shapes by using CFD.

In order to solve the local optimization problem and computing cost problem, the method combining CFD and surrogate modeling has been applied for the relationship between cyclone shapes and separation performance [11-16]. For example, the developed surrogate models such as ANN, RSM and GMDH algorithm showed reasonable predictive performance, and optimum design was performed by applying optimization algorithms such as genetic algorithms to the surrogate model [13]. However, the most optimization studies omitted the analysis of the cause of the optimal separation performance. For analyzing the optimization results, the tangential velocity distribution of the shape before and after optimization was compared based on the particle separation theory from Euler's point of view [13,14]. However, since the force acting on the particles differs according to the rotational trajectory, it is more appropriate to analyze it according to the trajectory position rather than to analyze it from the Euler perspective. In other words, it is necessary to analyze the force acting according to the particle trajectory in the cyclone from the Lagrangian perspective. When analyzing the cyclone separation performance from the Lagrangian perspective, the verified cyclone separation performance parameter can be newly considered as dependent variable for cyclone design.

This study has two purposes; (1) to analyze the characteristics of cyclones from the Lagrangian perspective by using CFD for designing and verifying important dependent variables that are different from previous studies; (2) to develop a machine learning prediction mode of the derived dependent variables; (3) to compare the prediction performance with the machine learning model

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 November 2020 doi:10.20944/preprints202011.0187.v1

and traditional surrogate model. This study identifies more rational dependent variables for cyclone design. In addition, it is possible to propose a fast and reliable process.

The flow chart of this study is summarized as shown in Figure 2. First, based on CFD, we analyze the characteristics of cyclones from the Lagrangian perspective to derive meaningful dependent variables. Next, a CFD data set of dependent variables is created to develop a machine learning model and a surrogate model. The data sets are created according to a design space with various combinations using the Design of Experiment method. The cyclone machine learning models are optimized for training parameters such as the learning rate, epoch etc. based on random sampling. Finally, the developed model evaluates the predictive performance compared to the traditional surrogate model, MLR.

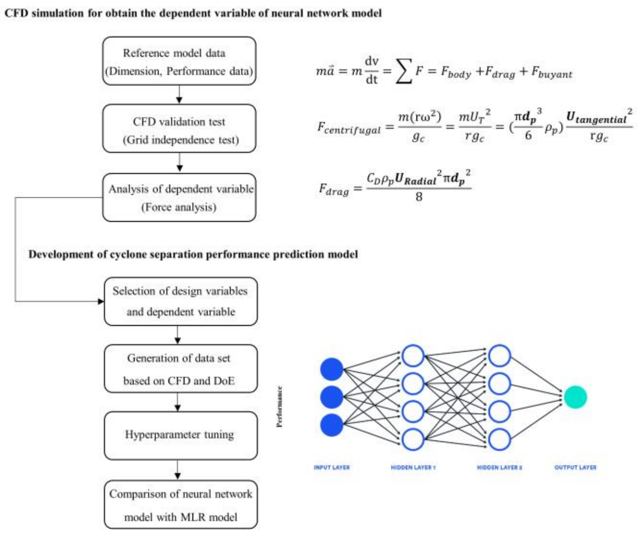


Figure 2. Research flow chart

2. Research Methods

2.1. Governing equation for numerical simulation

The numerical simulation was applied to reasonably obtain the dent variable of the separation performance for machine learning algorithm. The cyclone flow exhibits strong three-dimensional rotational turbulence. The 3D Reynolds averaged Naiver-Stokes equation are solved based on finite volume method. The Eq.1 represents the RANS equation.

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j} u_i' u_j' \tag{1}$$

where the p is pressure. the velocity components are decomposed into the mean velocity, $\bar{u}_i$ and fluctuating velocity, u_i', respectively. The u_i' is Reynolds stress. The additional process of Reynolds stress is required to solve the RANs equation. The various turbulence models have been used for solve the Reynolds stress term [17]. In order to obtain high-accuracy numerical analysis results, the turbulence model suitable for flow characteristics must be applied. The turbulence models have been successfully applied in many industrial fields. For detailed equations and explanations on the turbulence model, reference is made to the length limitation of this paper [17,18].

The behavior of fluid and solid particles was simulated complementarily for investigating the dynamic behavior of cyclone. In order to design rationally the cyclone separation performance parameters based on the Lagrangian method rather than the Euler method, which is one of the objectives of this study, it is necessary to trace the dynamic behavior of solid particles. The equation in Eq 2 represents the particle trajectory equilibrium equation.

$$\frac{d\vec{u}_p}{dt} = \mathrm{F_D}\left(\vec{u}_k + u_k' - \vec{u}_p\right) + \frac{\vec{\mathrm{g}}(\rho_\mathrm{p} - \rho)}{\rho_\mathrm{p}} + \mathrm{F} \tag{2}$$

where the term F_D is the drag force per unit particle mass, and F is an additional acceleration (force/unit particle mass). The u' in the turbulence transport equation influences the behavior of the particles. In this study, the commercial CFD code ANSYS 16.1 was used to solve Eq 1 and 2. The computational domain of cyclone is used as a cutcell type as shown Figure 3. The cyclone dimension of reference model that will be used for validating CFD was cited [19]. The near-wall treatment was achieved by using scalable wall functions considering the grid refinement with $y^+ < 11$. The Table 1 shows the boundary conditions for numerical analysis applied in this study. For CFD simulation, the SIMPLE algorithm, PRESTO!, Second order upwind scheme were used for pressure term, pressure-velocity term and turbulence kinetic and dissipation and momentum term, respectively. The criteria of residual values of the turbulence equation and other equation for assessing CFD convergence were set as 10^{-6} and 10^{-4}.

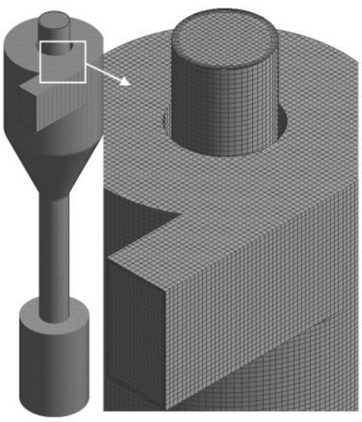


Figure 3. Computational domain of cyclone separators

Table 1. Boundary condition for CFD

Boundary condition	Values
Inlet velocity	800 (m^3/h)
Pressure drop	1 atm
Time step size	0.0001 sec
Number of time step	1500
Particle density	2770 kg/m^3

2.2. *Machine learning algorithm*

The computational cost is high when performing CFD simulations on various geometric design parameters to get the cyclone separation performance for the desired system. A machine learning model for cyclone separation performance has been developed for solving the time cost problem of cyclone design. The developed model can predict fast separation performance changing the various design combinations. The separation performance model was developed using the back propagation neural network model among machine learning algorithms. The neural network model predicts the output variable according to the new input variable by giving nonlinear characteristics to the relationship between the input design variable and the output variable. The structure of the neural network consists of several hidden layers between input and output variables. The layer consists of various nodes, and the node converts the linear combination of input variables into a sigmoid nonlinear form as shown in Eq. 3 and Eq. 4.

$$y_j^{(k)} = b_0 + \sum_{i=1}^{n} w_i x_i \tag{3}$$

$$y_{j\_out}^{(k)} = \frac{1}{1 + \exp\left(-y_j^{(k)}\right)} \tag{4}$$

Where k is layer number, j is node number and w_i is weight. The input variables are transferred to the hidden layer and calculated until the end of the output stage. Then, the weight of all nodes are updated repeatedly so that the error with the true value is minimized. This is called backpropagation process. That is, the parameters such as learning rate, epoch, batch size and number of hidden layers etc. must be optimized to make the minimum difference value between the true value and prediction value.

In this study, input variables of neural network model were set as four geometry types of cyclone to make a model for predicting cyclone separation performance. It was confirmed that four geometric shape out of many geometries have a great influence on the separation performance [12]. The combination of design variables for training and testing a neural network model was created based on the Desing of Experiment (DoE). The DOE enables to create a design area where a lot of information can be obtained with little data. The range of design variables is shown in the Table 2. The minimum and maximum bounds were created in a range where interference does not occur between shapes. The number of simulation combinations generated is 100. The data set consisted of a training set, a validation set, and a test set, with each percentage set to 70%, 10% and 20%. Since the train set and test set have different combinations, it is possible to evaluate the predictive performance of the generalized model with a new combination rather than the design variables used for training. The validation set is a set that checks whether over fitting occurs in the process of updating weights. The Figure 4 shows the design space of the train set and test set for the cyclone shape variable used in this study. Because the design space of the train set and the test set are different, the generalized performance tests can be performed. All BPNN calculations were carried out using PYTHON 3.6.

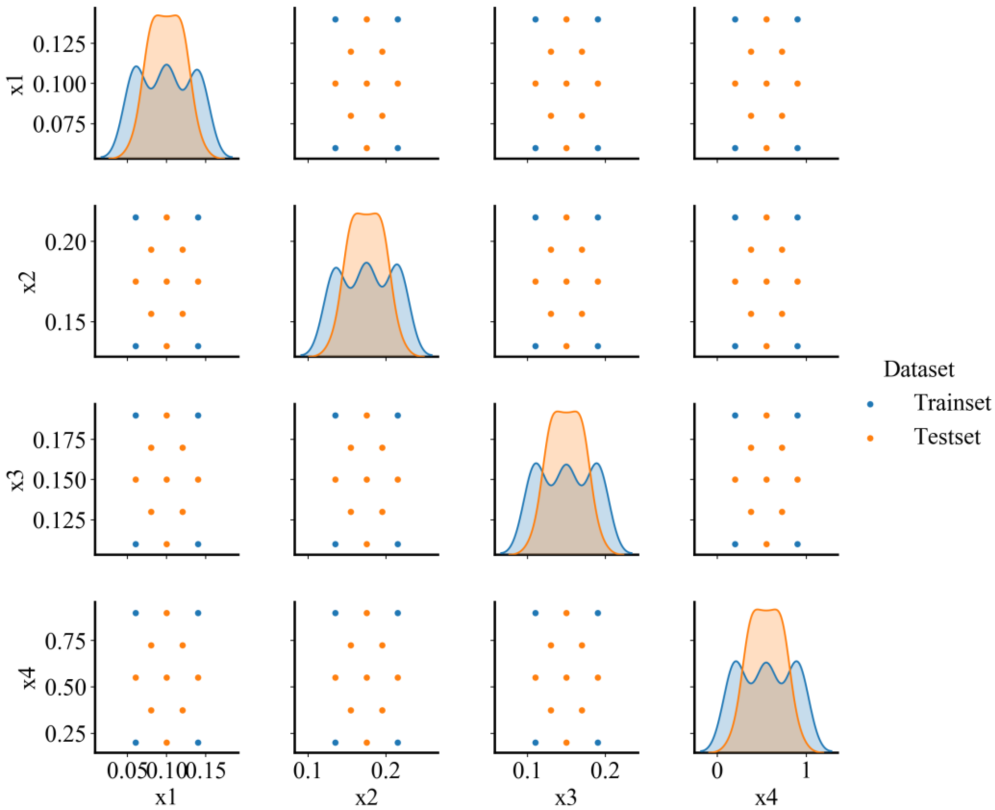


Figure 4. Design space for dataset; x1,x2,x3 and x4 is design variable of cyclone.

Table 2. Range of design variable of cyclone.

Boundary condition	Min(x/D[1])	Max(x/D[1])
Outlet diameter	0.275	0.475
Inlet width	0.15	0.35
Inlet height	0.3375	0.5375
Cone length	0.5	1.95

[1] D is 0.4 m

3. Results

3.1. CFD simulation result for validation

To achieve the purpose of this study, it is essential to validate the use of CFD. The experimental results and cyclone dimensions were cited [19,20] and compared with CFD results under the same conditions. To verify the validity of the CFD, a grid dependence test was performed. The grid dependence test not only helps the efficient use of computing cost, but also obtains a numerically optimized computational domain. The cited separation efficiency and pressure drop test data were compared with the CFD simulation results according to the grid size as shown **Table 3**. As the grid size decreases, the numerical values converge, showing an error within 2% of the referenced experimental values. Therefore, the maximum length of grid was selected as 6.5 mm in this study. The selected grid size is used as an input condition when obtaining data for CFD analysis for neural network modeling.

Table 3. Grid dependence test results.

Grid size (mm)	100	17.5	6.5	4.5	3.5
Separation efficiency (Error)	61.38 %	26 %	1.1 %	1.06 %	1.1 %
Pressure drop (Error)	37.5 %	7 %	0.5 %	0.49 %	0.51 %

In addition, in order to select an appropriate turbulence model that can simulate a cyclone strong rotational flow, the results of the velocity distribution experiment and the prediction results according to the turbulence model were compared. The experimental data and simulation data were compared with the results of the velocity distribution at the two locations (Y =0.77D, 0.345D) as shown Figure 5. When the K-e model was used, it showed an abnormal tangential distribution near the wall. The reason for this prediction is that the k-e model assumes anisotropic property for modeling the Reynolds stress term. When K-e model is applied for cyclone flow analysis, the outer flow and inner flow can be captured incorrectly. In contrast, RSM predicted a velocity distribution similar to the experimental results. RSM can properly simulate rotational flow through an isotropic assumption for Reynolds stress term. Therefore, in this study, RSM model was applied to capture the cyclone flow.

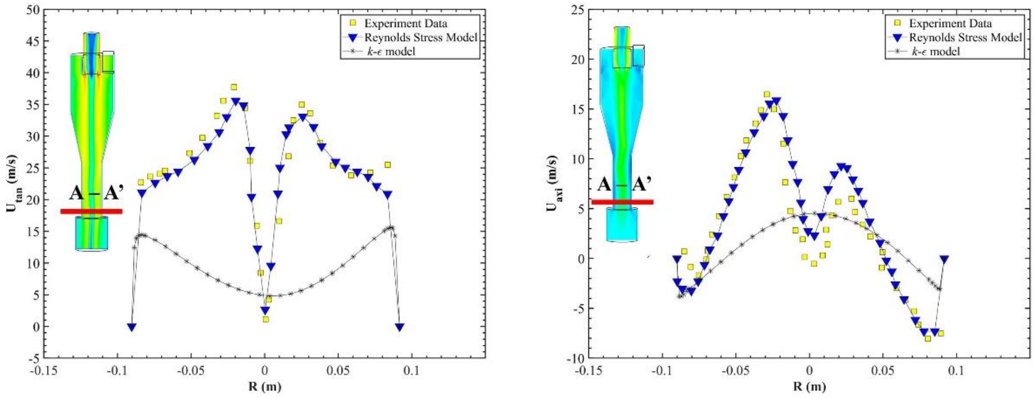


Figure 5. Comparison results of the velocity distribution experiment and the prediction results according to the turbulence model

3.1. CFD simulation for the dependent variable of cyclone

By analyzing the characteristics of cyclones from the Lagrangian perspective, important dependent variables different from previous studies were verified. In order to consider the average effect of particles of various sizes by using a ANSYS FLUET User Define Function code, the force acting on the particles was calculated until the particles were separated. The dynamic behavior results are very similar to the particle separation theory.

As shown Figure 6, In the case of 1um particles, the drag acting on the particles is superior to the centrifugal force, so the particles enter the inner flow region and are discharged through the upper outlet of cyclone. In the case of 1.5 um, the centrifugal force and drag act similarly, resulting in rotational motion within the cyclone. Eventually, it can be seen that the drag is slightly larger and the particles are rebound in the dust bin. The 5um particles have a larger centrifugal force than the drag force, so that the particles are collected in the dust bin.

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 November 2020 doi:10.20944/preprints202011.0187.v1

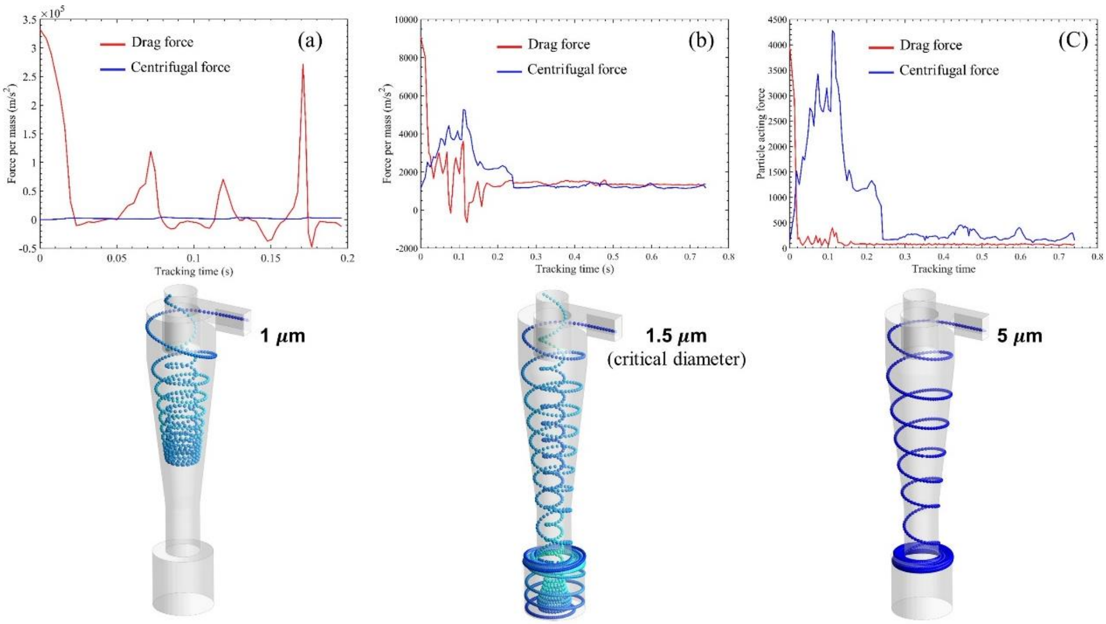


Figure 6. The force analysis acting on particle; (a) 1μm behavior, (b) 1.5 μm behavior, (c) 5 μm behavior.

To quantify this separation phenomenon, the forces acting during the separation time were averaged and compared with the separation efficiency curve as shown in the Figure 7. The difference in the equilibrium action of centrifugal force and drag occurs around the diameter with a separation efficiency of 50%. That is, the diameter with the same centrifugal force and drag force with 50% separation efficiency is the critical diameter that can explain cyclone particle separation. When analyzing the characteristics of the cyclone from the Lagrangian point of view rather than the Euler point of view, the critical diameter, in which centrifugal force and drag force act similarly, is an important design variable. The critical diameter was selected as a neural network output variable.

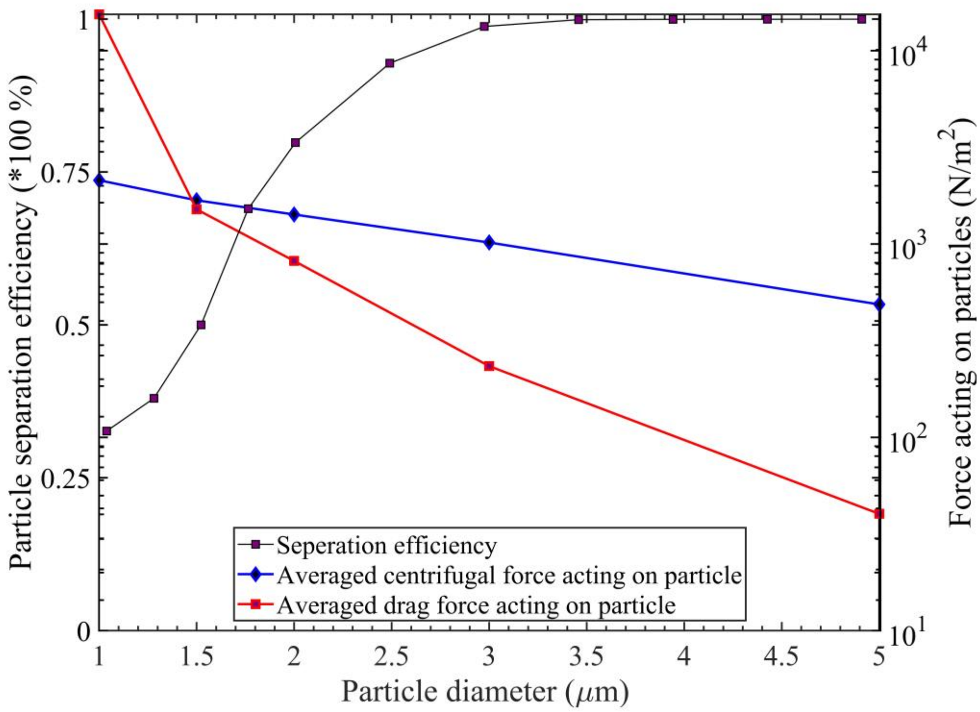


Figure 7. The averaged force results acting during the separation time with the separation efficiency curve

3.1. Cyclone performance prediction model development using neural network algorithm

The neural network method was applied to develop a cyclone critical diameter prediction model. To develop a cyclone separation performance model, various design combinations were made based on the DoE method and CFD. The hyperparameter tuning was performed with random sampling method to derive parameters that affect the prediction model. The number of sampling was set to 500 in order to consider diversity of design space. The result of parameter optimization is shown in the Figure 8. The Figure 8 shows the results predicted by a random combination of learning parameters. The right end of the Figure 8 is a quantification of prediction performance according to parameter combinations. The R square and mean normalized error were used as indicators for quantitative evaluation of the model. R square represents the degree of agreement between the true value and the predicted value, and the closer to 1, the higher the performance. MNE is an index that can objectively evaluate the model's performance. The optimal parameter combinations are summarized in the Table 4.

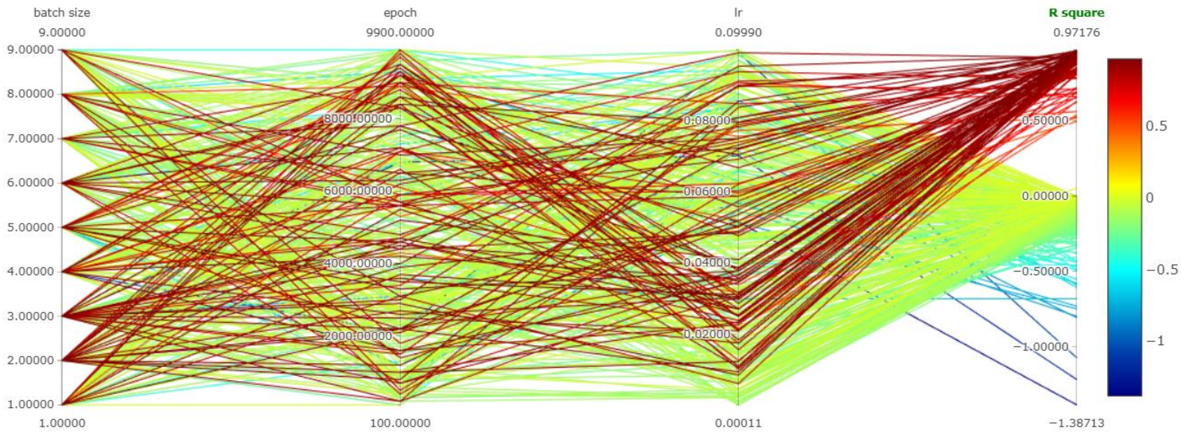


Figure 8. Hyperparameter tuning results

Table 4. The optimized learning parameter.

Optimized parameters	Values
Epoch	5200
Learning rate	0.00054
Batch size	2
Number of layer	5
Node	8/16/24/16/8

The model was developed using the train set based on the optimal learning parameter combination, and the performance of the neural network model was evaluated using the test set. To evaluate the predictive performance of the neural network model, the results of multi linear regression as reprehensively traditional surrogate model were compared. The result of comparing the prediction performance of the MLR model and the neural network model was shown in the Figure. 9 Unlike the MLR model, since the neural network model can create a complex nonlinear relationship between the cyclone design variable and the critical diameter, it shows better predictive performance than the traditional method. The results of expressing this quantitatively are shown in the Table 5. NN increased about 32.2% and 27.6% in R2 and MNE, respectively, compared to MLR. This shows very good prediction performance. Figure 10 represents the training results and prediction results by the NN and MLR, respectively.

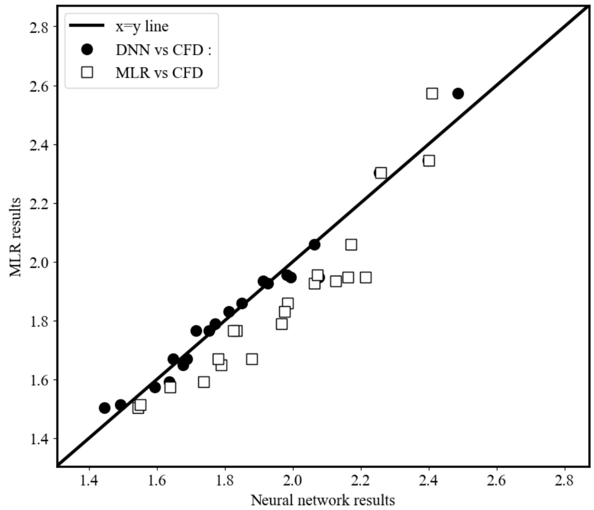
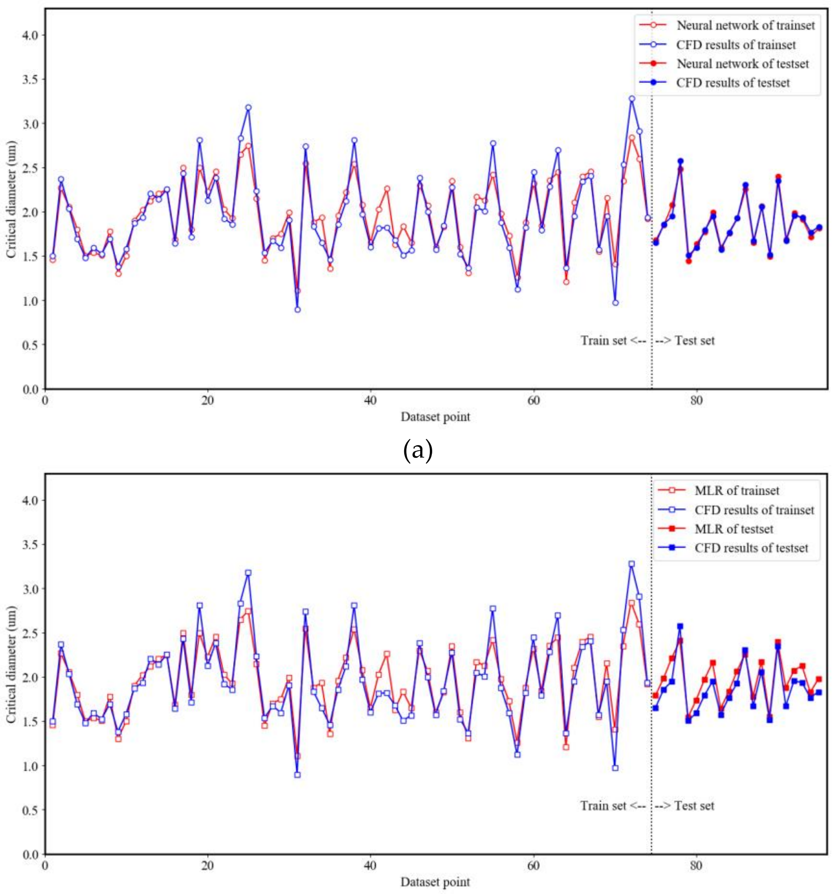


Figure 9. The result of comparing the prediction performance of the MLR model and the neural network model.



(a)

(b)

Figure 10. the training results and prediction results by the NN and MLR; (a) Neural network results; (b) Multi linear regression

Table 5. Neural network prediction model performance comparing with MLR results

Metric	MLR	NN	improvement
Mean normalized error	6.73	1.86	- 27.6 %
R^2	0.735	0.972	+ 32.2 %

4. Conclusion and Discussion

In the present study, the particle behavior characteristics in the cyclone were analyzed from the Lagrangian perspective through RANS analysis and particle force analysis for efficient and fast design of the cyclone. Based on the particle separation theory, it was clearly confirmed that the separation phenomenon occurred according to the force equilibrium action of the particles using CFD. In particular, it was demonstrated that the centrifugal force and the drag force are similar in the diameter with the separation efficiency of particles with 50 % separation efficiency. This indicates that it is an important dependent variable for cyclone design based on particle separation theory. Therefore, the critical diameter was applied to the neural network as the design dependent variable. The neural network model was developed by using CFD combinations that considered various design space based on the design of experimental method. The learning parameters of developed model showed sufficient distribution in the design space, and the neural network prediction model showed superior performance compared to the traditional MLR results.

In a future study, we plan to find a wider design area point based on the critical diameter ANN model and global optimization algorithm, or derive the optimal critical diameter, and investigate the generalization characteristics of the neural network model through experimental method.

Author Contributions: D.-G.P. and J.-S.G. analyzed the numerical data; D.-G.P. developed the neural network model and organized draft paper.

Funding: be supposed to add after first review process

Acknowledgments:

Conflicts of Interest: The authors declare no potential conflicts of interest with respect to the research, authorship and publication of this article

References

1. Iozia, D.L.; Leith, D. Effect of cyclone dimensions on gas flow pattern and collection efficiency. *Aerosol Sci. Technol.* **1989**, *10*, 491–500, doi:10.1080/02786828908959289.
2. Shepherd, C.B.; Lapple, C.E. Flow Pattern and Pressure Drop in Cyclone Dust Collectors: Cyclone without Inlet Vane. *Ind. Eng. Chem.* **1940**, *32*, 1246–1248, doi:10.1021/ie50369a042.
3. Avci, A.; Karagoz, I. Theoretical investigation of pressure losses in cyclone separators. *Int. Commun. Heat Mass Transf.* **2001**, *28*, 107–117, doi:10.1016/S0735-1933(01)00218-4.
4. Raoufi, A.; Shams, M.; Farzaneh, M.; Ebrahimi, R. Numerical simulation and optimization of fluid flow in cyclone vortex finder. *Chem. Eng. Process. Process Intensif.* **2008**, *47*, 128–137, doi:10.1016/j.cep.2007.08.004.
5. Wang, B.; Xu, D.L.; Chu, K.W.; Yu, A.B. Numerical study of gas-solid flow in a cyclone separator. *Appl. Math. Model.* **2006**, *30*, 1326–1342, doi:10.1016/j.apm.2006.03.011.
6. Misiulia, D.; Andersson, A.G.; Lundström, T.S. Chemical Engineering Research and Design Effects of the inlet angle on the flow pattern and pressure drop of a cyclone with helical-roof inlet. *Chem. Eng. Res. Des.* **2015**, *2*, 307–321, doi:10.1016/j.cherd.2015.06.036.
7. Ganegama Bogodage, S.; Leung, A.Y.T. CFD simulation of cyclone separators to reduce air pollution. *Powder Technol.* **2015**, *286*, 488–506, doi:10.1016/j.powtec.2015.08.023.
8. Elsayed, K.; Lacor, C. Numerical modeling of the flow field and performance in cyclones of different cone-tip diameters. *Comput. Fluids* **2011**, *51*, 48–59, doi:10.1016/j.compfluid.2011.07.010.
9. De Souza, F.J.; Salvo, R.D.V.; Martins, D.D.M. Effects of the gas outlet duct length and shape on the performance of cyclone separators. *Sep. Purif. Technol.* **2015**, *142*, 90–100, doi:10.1016/j.seppur.2014.12.008.
10. Ganegama Bogodage, S.; Leung, A.Y.T. CFD simulation of cyclone separators to reduce air pollution. *Powder Technol.* **2015**, *286*, 488–506, doi:10.1016/j.powtec.2015.08.023.
11. Elsayed, K.; Lacor, C. Modeling and Pareto optimization of gas cyclone separator performance using RBF type artificial neural networks and genetic algorithms. *Powder Technol.* **2012**, *217*, 84–99, doi:10.1016/j.powtec.2011.10.015.

12. Safikhani, H. Modeling and multi-objective Pareto optimization of new cyclone separators using CFD, ANNs and NSGA II algorithm. *Adv. Powder Technol.* **2016**, *27*, 2277–2284, doi:10.1016/j.apt.2016.08.017.
13. Park, D.; Cha, J.; Kim, M.; Go, J.S. Multi-objective optimization and comparison of surrogate models for separation performances of cyclone separator based on CFD, RSM, GMDH-neural network, back propagation-ANN and genetic algorithm. *Eng. Appl. Comput. Fluid Mech.* **2020**, *14*, 180–201, doi:10.1080/19942060.2019.1691054.
14. Sun, X.; Kim, S.; Yang, S.D.; Kim, H.S.; Yoon, J.Y. Multi-objective optimization of a Stairmand cyclone separator using response surface methodology and computational fluid dynamics. *Powder Technol.* **2017**, *320*, 51–65, doi:10.1016/j.powtec.2017.06.065.
15. Safikhani, H.; Hajiloo, A.; Ranjbar, M.A. Modeling and multi-objective optimization of cyclone separators using CFD and genetic algorithms. *Comput. Chem. Eng.* **2011**, *35*, 1064–1071, doi:10.1016/j.compchemeng.2010.07.017.
16. Elsayed, K.; Lacor, C. CFD modeling and multi-objective optimization of cyclone geometry using desirability function, Artificial neural networks and genetic algorithms. *Appl. Math. Model.* **2013**, *37*, 5680–5704, doi:10.1016/j.apm.2012.11.010.
17. ANSYS Inc. ANSYS FLUENT Theory Guide; ANSYS FLUENT-16.1; ANSYS Inc.: Canonsburg, PA, USA, 2018.
18. Tang, Y.; Guo, B.; Ranjan, D. Numerical simulation of aerosol deposition from turbulent flows using three-dimensional RANS and les turbulence models. *Eng. Appl. Comput. Fluid Mech.* **2015**, *9*, 174–186, doi:10.1080/19942060.2015.1004818
19. Obermair, S.; Woisetschläger, J.; Staudinger, G. Investigation of the flow pattern in different dust outlet geometries of a gas cyclone by laser Doppler anemometry. *Powder Technol.* **2003**, *138*, 239–251, doi:10.1016/j.powtec.2003.09.009.
20. Obermair, S.; Staudinger, G. The dust outlet of a gas cyclone and its effects on separation efficiency. *Chem. Eng. Technol.* **2001**, *24*, 1259–1263, doi:10.1002/1521-4125(200112)24:12<1259::AID-CEAT1259>3.0.CO;2-O.