Flow Analysis of the Environmental Chemical Reaction Processes at Power Plant in accordance with the Baffle Structure

Yeon-Tae Jeong*, Kwang-Beom Hur, Joon-Woo Gil

KEPCO Research Institute, Korea Electric Power Corporation, 105 Munji-Ro Yusung-Gu, Daejeon 34056, Korea

*Yeontae.jeong@kepco.co.kr

Abstract

In the area of environmental chemistry of power plant, flow analysis of the reactor with built-in impeller is a very important part from the perspective of the improvement of the efficiency of the entire process. As a wide range of methods are being proposed for the analysis of the flow pattern within the reactor, this study analyzed the flow within the reactor according to the baffle structure (height) installed on the internal wall of the reactor in order to improve the reaction efficiency through the inducing of the up and down stirring with the reactor. As the results of the execution of the flow analysis for each of a diverse range of cases by utilizing the Computational Fluid Dynamics (CFD) method, it was possible to confirm that the flow is markedly improved by inducing the up and down stirring among the reactants within the reactor if the baffle is elevated to the level below the water surface. In particular, as the results of the analysis of the general cases in which the baffle is elevated all 4 steps and the cases in which the baffle is elevated only 2 steps, elevating the baffle only 2 steps achieve the same effect as the elevating of the baffle by 4 steps. Therefore, it was possible to expect to improve the efficiency with out the need to increase the use of electric power substantially if the outcomes of this study is applied to the actual sites of power plants in the future.

Keywords: Environmental chemistry, thermal power plant, reactor

I. INTRODUCTION

In the area of the environmental chemistry of the power plant, Continuous Stirred Tank Reactor (CSTR) with impeller installed inside the tank is used widely for a diverse range of purposes including the storage, mixing and chemical reaction of the various raw materials [1]. The aforementioned reactor is characterized by its ease of operation and management due to the ease of feeding of the reactants while maintaining the appropriate retention time and possibility of application for a diverse range of purposes through the alteration of the reaction composition and operating conditions.

Summary of the cases of the reactors at the site of power plant [2][3].

A lot of researches related to the mixing efficiency of CSTR reactor has been carried out recently [4]-[7]. Among these, there are numerous researches on the optimization of the internal flow in accordance with the changes in the wide range of the impeller appearance-reactor configuration, and the appropriate changes in the wide range structures (baffle, draft tube, etc.) installed within the reactor that are being carried out [4].

Since the improvement of the flow inside the reactor is one of the most important factor that can improve the mixing efficiency of the reactor, the improvement of its performances can be deemed the most important factor that determines the productivity of the overall chemical reaction process. In this regard, this paper proposes the means of the optimal reaction process composition for the utilization in the area of the environmental chemistry of power plant by analyzing the mixing efficiencies in accordance with the diverse range of baffle structures by using the Computational Fluid Dynamics (CFD) method.

II. ANALYSIS

A. Process

The interesting configurations of the reactor in this study are as follows. The overall volume of the reactor was presumed to be 10 m³, which is the most frequently used size for the storage of the key raw substances in the area of environmental chemistry of power plant, while the format of the impeller installed within the reactor was presumed to be pitched paddle with 4-step blade format. The rotational speed of the impeller (RPM) was presumed to be 100 rpm in the analysis for this Study since impellers are most frequently operated at the rotational speed range of 50-100 rpm at the sites of power plants. Steady state without changes with the passage of time was presumed along with the assumption that there is absence of raw materials inputted from the outside of the reactor. The substance within the reactor is also presumed to be pure water for the simplifying of the system evaluated.

On the basis of the aforementioned Base conditions, the flow distribution of the chemical reactor in accordance with the configuration (size) of the baffles installed on the internal wall of the reactor was analyzed. Generally, the baffles are installed inside the stirring tank reactor to perform the role of inducing the up and down stirring at the time of mixing by impeller. However, such baffle induces outcomes that differ in accordance with the configuration of the reactor, number of rotations and the characteristics of the impeller. Therefore, it is highly important to determine the optimal baffle structure through precision analysis prior to the installation. In particular, if the baffle is excessively installed to the extent of being unnecessary, the pressure imparted on the impeller increases with ensuing increase in the electric power consumed. Accordingly, optimal baffle design is very...
In this regard, this study executed the flow analysis in accordance with the configuration of the baffle installed on the internal wall of the reactor for each of the cases. For this purpose, as illustrated in the Fig. 4, the method of elevating the baffle by 2 steps among the total of 4 steps and elevating it by all 4 steps in comparison to the baseline case was devised. The Fig. 4 below is the conceptual diagram for each of the cases.

**B. CFD Analysis**

In this study, the CFD interpretation for the analysis of the flow in accordance with the diverse range of baffle configurations within the reactor was executed on the basis of the commercially available flow interpretation package (provided by Fluent Inc.) [8].

In this study, the system being analyzed was limited to the CSTR that includes the impeller and the baffle in the reactor. The meshing work of the reactor for the CFD calculation used the meshing tool provided by Fluent Inc. The mesh finally generated through meshing was at the level of 500,000. Since the internal flow belongs to the domain of turbulent flow with sufficiently high rotation of impeller, the overall computation was made by applying appropriate turbulence model (k-epsilon model). The following hypotheses were set for the factors that do not impart significant influence on the final results in order to shorten the time required for the computation and to minimize the amount of computer memory being used in the flow interpretation:

1) Disregard the reaction: The effects of the reaction on the flow are excluded.
2) Consistency of the property of matter: Disregard the changes in the density/viscosity within the stirring tank in accordance with the reaction (presume that the internal aspect of the reactor is filled with water).
3) Apply the presumption of turbulent flow: Apply the k-epsilon model
4) Rotation analysis: Utilize the MFR technique of FLUENT

As the method of resolving the turbulence model, this Study
used the ‘k-epsilon model’, which is a general turbulence model solver. The standard transport equation in the standard k-epsilon model for the analysis of the reactor is as shown in Eq. 1 and 2.

\[
\frac{\partial}{\partial x_i}\left(\rho k\right) + \frac{\partial}{\partial x_i}\left(\rho\varepsilon\right) = \frac{\partial}{\partial x_i}\left(\frac{\mu + \frac{\mu_t}{\kappa}}{\kappa}\frac{\partial k}{\partial x_i}\right) + \frac{\partial}{\partial x_i}\left(\frac{\mu + \frac{\mu_t}{\kappa}}{\kappa}\frac{\partial \varepsilon}{\partial x_i}\right) + C_{\mu_k}\left(P_k + C_{\mu_k}P_2\right) - C_{\varepsilon_k}\frac{\rho}{k} + S_k + \rho \varepsilon
\]

(1)

\[
\frac{\partial}{\partial x_i}\left(\rho \varepsilon\right) + \frac{\partial}{\partial x_i}\left(\rho \varepsilon u_i\right) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\kappa}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{\mu_k}\left(P_k + C_{\mu_k}P_2\right) - C_{\varepsilon_k}\frac{\rho}{k} + S_k + \frac{\varepsilon}{k} \nabla^2 \varepsilon
\]

(2)

Here, ‘k’ is the turbulent kinetic energy while ‘\varepsilon’ is the rate of dissipation.

k-epsilon (k-\varepsilon) turbulence model is the most common model used in Computational Fluid Dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. The k-\varepsilon model has been effectively adopted especially for the calculation of recirculating flows in a reactor [9]. This model is the most widely used and validated turbulence model with applications ranging from industrial to environmental flows, which explains its popularity. It is usually useful for simplest turbulence model for which only initial and/or boundary conditions needs to be supplied [7] and can also be adopted for the free shear layer flows with relatively small pressure gradients as well as in confined flows where the Reynolds shear stresses are most important [10].

C. Boundary Conditions

For the CFD analysis, boundary conditions including geometry and operating conditions of present study are as shown in table 1. And using these data, CFD analysis of various baffle configuration was evaluated.

III. RESULTS AND DISCUSSION

CFD analysis was executed on the foundation of the contents proposed above. For the flow analysis in accordance of the configuration of the baffle for each of the Cases, the cross-section of the reactor is cut from various directions to make categorization of the flow pattern observed by means of the velocity components for analysis.

A. Velocity Magnitude

As the results of the analysis of the velocity magnitude & stream-line, it was confirmed that the up and down stirring was not harmonious with severance in the middle in the existing short baffle structure (the baseline case). In contrast, in the cases of the height of the baffle elevated close to the surface of the liquid, in both the case with elevation of the baffle by 4 steps (Case 2) and the case with elevation only by 2 steps (Case 1), up and down stirring was performed substantially better than the Baseline case, thereby confirming that the state of mixing within the reactor is improved.

B. Axial Velocity Magnitude

As the results of the observation of the distribution of the axial velocity for each of the cases presented above, the existing baffle structure (baseline case) is characterized by the severance of the upward flow, which rises along the baffle, at the bottom of the reactor, thereby resulting in failure to induce overall up and down stirring. In addition, the upward flow induced in the bottom of the blade of the 1st stage impeller inhibits the downward movement of the reactants generated in the upper portion. Therefore, it results in the shortening of the distribution of the average retention time since the reactor ends up using only the portion without being able to use the entire volume of the reactor in its reactions. In contrast, in both cases in which the baffle was elevated, strong axial velocity was observed around the baffle. This fact indicates that the overall stirring efficiency is increases due to strong up and down stirring within the reactor.

C. Tangential Velocity Magnitude

The following is the results of the observation of the distribution of the velocity in the tangential direction (or rotational direction). In the existing baffle structure (baseline case), the
rotational velocity was found to be relatively higher in the top of the reactor. When this is considered in combination with the distribution of the velocity in the axial direction, it signifies that the raw material, immediately after having been inputted into the reactor, fails to use all of the bottom portion of the reactor but rotates at high velocity only in the top portion prior to being discharged into the next stage of the process. On the other hand, in the case of having the baffle elevated (Case 1 and Case 2), it was possible to confirm that the tangential velocity is evenly distributed throughout the entire reactor.

D. Discussion

As the results of the analysis of the flow within the reactor in accordance with a diverse range of the baffle structures, it was confirmed that the internal flow distribution of the reactor is improved when the height of the baffle is elevated to the water surface. In particular, as the result of the analysis of the general case of elevating the baffle by all 4 steps available and the case of elevating the baffle only by 2 steps, it was possible to confirm that the effect achieved by elevating the baffle only 2 steps was the same as that achieved by elevating it by 4 steps. Therefore, it was possible to anticipate the improvement of efficiency without the substantial increase in the electrical power consumption by applying this finding.

IV. CONCLUSIONS

In this Study, the flow in the reactor in accordance with the baffle structure (height) installed on the internal wall of the reactor for the purpose of the improvement of the reaction efficiency through the inducing of up and down stirring was analyzed. Using a CFD analysis based on CSTR (reactor volume: 10 m³), it was confirmed that the internal flow distribution of the reactor is improved when the height of the baffle is elevated to the water surface. In particular, as the result of the analysis of the general case of elevating the baffle by all 4 steps available and the case of elevating the baffle only by 2 steps, it was possible to confirm that the effect achieved by elevating the baffle only 2 steps was the same as that achieved by elevating it by 4 steps. Therefore, it was possible to anticipate the improvement of efficiency without the substantial increase in the electrical power consumption by applying this finding.

REFERENCES

[1] Alena Kukukova et al., “A new definition of mixing and segregation: Three dimensions of a key process variable”, Chemical Engineering Research and Design, 87, 633-647, 2009.
[2] Diagnostic report on water treatment equipment of Seocheon thermal power plant, 5, 2006.
[3] Diagnostic report on water treatment equipment of Boryeong power plant, 5, 2005.
[4] Carsten Stemich and Lothar Speigel “Characterization and quantification of the quality of gas flow distributions”, Chemical.
[5] Rahimi, M., Kakekhani A.,Alsairafi AA., "Experimental and Computational Fluid Dynamic(CFD) Studies on Mixing Characteristics of a Modified Helical Ribbon Impeller," Korean J. Chem. Eng., 27(4), 1150-1158, 2010.
[6] Rahimi, MR., Azizi, N., and Hosseini, SH., "CFD Study of Hydrodynamic Behavior of a Vibrating Fluidized Bed Using Kinetic-frictional Stress model of Granular Flow," Korean J. Chem. Eng., 30(3), 761-770(2013). Engineering Research and Design, 89, 1392-1396, 2011.
[7] Henk Karrle Versteeg, Weeratunge Malalasekera, “An introduction to Computational Fluid Dynamics" The Finite Volume Method", Pearson Education Limited, ISBN 9780131274983, 2007.
[8] FLUENT/CFX, ANSYS, INC.
[9] Launder, B.E.; Spalding, D.B. “The numerical computation of turbulent flows". Computer Methods in Applied Mechanics and Engineering 3 (2): 269-289. 1974.
[10] P Bradshaw, “Turbulent Secondary Flows”, Annual Review of Fluid Mechanics 19: 53-74, 1987.