Study on the stirred blade mounting height-to-liquid level-ratio related to the law of flow field inside the stirred reactor

Lei Cheng1*, Zhenwei Zhang1, Xinyu Zhao1, Qia Lin1

1Zhuhai College of Jilin University, Zhuhai, China
*Corresponding author e-mail: cl@jluzh.edu.cn

Abstract. To figure out the law of the height of stirred blade installed inside the reactor affect the fluid inside, this article used CFD to Numerical simulation of the liquid filled area of the stirred reactor, then obtain the distribution of the liquid velocity field inside the reactor. Through the comparison of calculation. Consider the weak area of stirred mainly concentrated above the stirred blade, when stirred blade were installed at the stirred blade mounting height-to-liquid level-ratio reach to 40%, it means the mixing level reach maximum, according to the calculation the average turbulence kinetic energy is 0.003407-40 0.002994-45, and standard deviation of turbulence kinetic energy is able to stable between 0.001503-40 to 0.001275-45.

1. Introduction
Stirred reactor is used for chemical reactions, the agitating device is one of the most important parts. the difference of mixed effect inside the stirred reactor seriously affected the productivity of the production, Although currently there are a numbers of scholars do a lot of research on the structure of the stirred reactor (Kasat & Khopkar, 2008)but most of all were based on the fundamental calculation and the tradition calculation. The reactor itself were very complicate, use the tradition way to calculate, which means waste of times and money and using old method is very hard to optimize the design of reactor. CFD (Computational Fluid Dynamics) technique applied in Equipment optimization simulation of aviation, spaceflight, and mechanical design (Tabib & Joshi, 2008; Liu & Keplinger, 2018), can not only save time and experimental cost. Even make it more Intuitive to see the result. So this paper use CFD to Study how the stirred blade mounting height-to-liquid level-ratio related to the law of flow field inside the stirred reactor.

2. Method of solution
This article focus on how the inside structure affect the stirred reactor, use the SIMPLE calculation based on speed-filed pressure-filed combination (Doormal & Raithby, 1984). This article considered the speed-filed and pressure-filed were two separated processes, according to the law of Conservation of mass we can calculate pressure-filed by speed-filed which is know If it didn’t meet the need of the law of Conservation the correction to pressure were needed. at the same time speed were also corrected when the speed were corrected considered each correction to speed value don’t effect each other. Then calculate though iteration repeated to get each value for each grid.

This article used standard k-epsilon model to simulate the fluid inside the reactor, though out the mass conservation and momentum conservation we were able to know equations during the stable calculation. The Continuity equation, momentum equation, energy equations of each phase will be as follow:
\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho u_i \right) = S_m
\]

(1)

\[
\frac{\partial}{\partial t} \left( \rho u_i \right) + \frac{\partial}{\partial x_j} \left( \rho u_i u_j \right) = -\frac{\partial P}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i
\]

(2)

\[
\tau_{ij} = \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] - \frac{2}{3} \frac{\partial u_i}{\partial x_j} \sigma_{ij}
\]

(3)

\[
\frac{\partial}{\partial t} \left( \rho E \right) + \frac{\partial}{\partial x_j} \left( \rho u_j \left( \rho E + p \right) \right) = -\frac{\partial}{\partial x_i} \left( k_{eff} \frac{\partial T}{\partial x_i} \right) - \sum_{j} h_j J_j + u_j \left( \tau_{ij} \right)_{eff} + S_k
\]

(4)

Considering the quality of the grid structure of first sate wind-faced and First Order Upwind were absolute stable, so we used the First Order Upwind to control the discreteness methods, while other inside wall will be use the standard wall function. Considered Gravity, drag, lift, resistance which were equal phase interaction and Turbulent dissipation rate. During calculation we design the residual meet the need of energy equation to be less than $10^{-6}$, residual of the component transportation equation to be less than $10^{-3}$, other residual of related equation to be less than $10^{-4}$, we believed that the calculation were considered to be achieve convergence condition. In this article we took water as fluid as our research object. The outer condition is Import flow in the speed of 28 L·h⁻¹ at room temperature and normal pressure input.

3. CDF modeling

The size of the reactor is shown in the figure1 (Ignore the space occupied by the turbine blade to establish a model). we choose the disc turbine blade as the inside agitator, the height of turbine blade is installed in variable value as follows tab.1.

![Figure 1: size and structure of the stirred reactor](image)

In this article we define variable which is B-to-L (the stirred blade mounting height t divided by liquid level, hereafter ‘B-to-L’). The stirred blade mounting height-to-liquid level-ratio. Establish an experimental group model as shown in tab.1., define fluid heigh as $10^2$ m 0 mm, the B-to-L value distributed as follow (tab.1.)
Table 1. stirred blade installation height and the corresponded B-to-L value

| NO. | The impeller mounting height/mm | liquid level/mm | B-to-L |
|-----|---------------------------------|-----------------|--------|
| 1   | 20                              | 100             | 20%    |
| 2   | 30                              | 100             | 30%    |
| 3   | 40                              | 100             | 40%    |
| 4   | 50                              | 100             | 50%    |

Measuring point positions and naming is shown in the tab.2.

Split schematics show as follow in fig. 2. Since the volume was very small and the structure of blade was complex, so using an unstructured grid to divide the whole structure, scale factor valued 0.1, stirred blade and some moving parts were divided as 0.05 grid.

Figure 2. grid separation schematic diagram (left split, right combination)

Table 2. computational grid and solution domain of stirred reactor

| Numble    | x  | y  | z  | NO.     | x  | y  | z  |
|-----------|----|----|----|---------|----|----|----|
| Point-40-20 | 0  | 20 | 40 | Point-45-20 | 0  | 20 | 45 |
| Point-40-30 | 0  | 30 | 40 | Point-45-30 | 0  | 30 | 45 |
| Point-40-40 | 0  | 40 | 40 | Point-45-40 | 0  | 40 | 45 |
| Point-40-50 | 0  | 50 | 40 | Point-45-50 | 0  | 50 | 45 |
| Point-40-60 | 0  | 60 | 40 | Point-45-60 | 0  | 60 | 45 |
| Point-40-70 | 0  | 70 | 40 | Point-45-70 | 0  | 70 | 45 |
| Point-45-80 | 0  | 80 | 40 | Point-45-80 | 0  | 80 | 45 |
| Point-45-90 | 0  | 90 | 40 | Point-45-90 | 0  | 90 | 45 |

4. Results and discussions

It can be found from figure 3 and 4, inside the reactor x=0 mm section the velocity vector options is y component z component and the value of speed. The stirred active area is mainly concentrated in the area where the stirred blade is located. By comparing the velocity vector distributions of each group, it can be further clarified that the stirred ability is directly related to the impeller-to-liquid level-ratio. Not hard to tell from figure 4 inside the reactor x=0 mm section the velocity vector options is y component z component and the value of speed. The stirred active area is mainly concentrated in the area where the stirred blade is located. By comparing the velocity vector distributions of each group, it can be further clarified that the stirred ability is directly related to the impeller-to-liquid level-ratio.
Figure 3. Velocity distribution for a 3D simulation (x-coordinate=0, the velocity vector options is y component & z component).

Figure 4. Velocity cloud pictures distribution for a 3D simulation (x-coordinate=0, the velocity vector options is y component & z component) with different B-to-L:
A: 20%, B: 30%, C: 40%, D: 50%

We can tell from figure 5 each points appears to be top offset when the critical range reach 30%, this became more obvious after critical range reach 30%. When it reached 50%, the maximum interval is shifted to the bottom. This phenomena is similar to what happened in figure 5 Point 40-50 and point 45-50. However, the deviation amplitude shows an increasing trend at point 40-50 and point 45-50, also means range between 30-40% have the highest mixed efficiency in another world the efficiency of stirred blade is the highest. In another word the efficiency of stirred blade between 30-40% is the highest. At 40%, the average value of the full-charge arithmetic shows that the fluid kinetic energy distribution is more uniform, and the state of the reactor is closer to the steady state.

Figure 5. point 40 series (the left) and point 50 series (the right).
Different B-to-L value correspond different turbulence kinetic energy value.
Figure 6. Average of turbulence kinetic energy and the standard deviation of turbulence kinetic energy

5. Conclusions
(1) Provided a method for establishing a CFD simulation reactor model
(2) When B-to-L value reached 40% stirring performance reach maximum.

Acknowledgments:
We are grateful to Research on stirred reactor structure of flow field based CFD simulation (No. 2018XJCQ065)

References
[1] Kasat G.R, Khopkar A.R, V.V. Ranade, A.B. Pandit. CFD simulation of liquid-phase mixing in solid–liquid stirred reactor. J. Chemical Engineering Science, 2008, 63(15).
[2] Tabib, M.V., Roy, S.A., Joshi, J.B., 2008. CFD simulation of bubble column—an analysis of interphase forces and turbulence models. Chem. Eng. J. 139 (3), 589–614.
[3] Liu Liu, Olga Keplinger, Tian Ma, Thomas Ziegenhein, Natalia Shevchenko, Sven Eckert, Hongjie Yan, Dirk Lucas. Euler-Euler simulation and X-ray measurement of bubble chain in a shallow container filled with liquid metals. J. Chemical Engineering Science, 2018, 192.
[4] Van Doormal, J.P & Raithby G.D, (1984). Enhancements of the SIMPLE method for predicting incompressible fluid flows. Numerical Heat Transfer, 7, 147-163.
[5] Mohsen Ostad Shabani, Ali Mazahery, Mehdi Alizadeh, Ali Asghar Tofigh, Mohammad Reza Rahimpour, Mansour Razavi, Alireza Kolahi. Computational fluid dynamics (CFD) simulation of effect of baffles on separation in mixer settler. J. International Journal of Mining Science and Technology, 2012, 22(05): 705-708.
[6] Xinyu Yu, Tianwen Chen, Qi Zhang, Tiefeng Wang. CFD simulations of quenching process for partial oxidation of methane: Comparison of jet-in-cross-flow and impinging flow configurations. J. Chinese Journal of Chemical Engineering, 2018, 26(05): 903-913.