Simulation of fluid around parallel-column by LES

Lei Cheng*, Zhenwei Zhang, Xinyu Zhao, Xiaoming Zhang
Zhuhai College of Jilin University, Zhuhai, China

*Corresponding author e-mail: cl@jluzh.edu.cn

Abstract. To figure out the distributions of the parallel-column’s field, this article used LES to study parallel-column. This article demonstrate the rule of how the parallel-column field changes with P/D (P: center spacing D: diameter), focused on the how the Bluff-body’s P/D affect to the speed field & pressure field. The simulation showed the field changed as the P/D ratio rise, when P/D ratio reach 0.2-0.6 the field show as Von Karman vortices, 0.8-1.0 show as parallel field, when it got large than 1.2 will be show as vortex side flow. The optimum working point is between 0.8-1.0, the air flow in this area’s subgrid Turbulent Viscosity is smaller the vortex strength of tail also weak.

1. Introduction
Column spoiler played a very important role in chemistry industry, it can optimize the design of the tunnel or the devices to keep safety and guidance as well [1]. So far there are a lot of research on the column spoiler, the research shows from as common as Reynolds number to Flow mechanism calculation [2-6]. But as for flow field for parallel-column in micro area is not easy to read the article about. This article combine CFD simulation to the actual problems, this article focus on diameter of the tunnel how the center spacing affect the pressure field and the speed field.

2. Method of solution
This article focus on how the inside structure affect the stirred reactor, use the SIMPLE calculation based on speed field pressure field combination (Doormal & Raithby, 1984), considered the speed field and pressure field were two separated processes, according to the law of mass conservation we can calculate the pressure field by using the speed field which is already known. 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 LES to Simulation of flow around parallel column, though out the law of mass conservation and the law of momentum conservation we were able to know equations during the stable. The Continuity equation, momentum equation, energy equation of each phase will be as follow:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = S_n
\]

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i
\]
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 $10^{-5}$, 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 $10 \text{ m} \cdot \text{s}^{-1}$ at room temperature and normal pressure input.

### 3. CDF modelling

The size and the parallel-column structure shown in the Figure 1, simulate size is $70 \text{ mm} \times 50 \text{ mm} \times 30 \text{ mm}$ around the parallel-column, this article used the air as the simulation object (under room temperature and standard `air pressure`). The volume is very small use an unstructured grid to divide the whole structure, scale factor valued 0.1.

![Figure 1. size and structure of the parallel column area](image)

In this article we define a variable which is $P/D$. The variable value in this article are shown as Table 1.

| NO. | P(center spacing)/ mm | D(diameter)/ mm | P/D |
|-----|------------------------|-----------------|-----|
| 1   | 2                      | 10              | 0.2 |
| 2   | 4                      | 10              | 0.4 |
| 3   | 6                      | 10              | 0.6 |
| 4   | 8                      | 10              | 0.8 |
| 6   | 10                     | 10              | 1.0 |
| 7   | 12                     | 10              | 1.2 |
Split schematic show as follow in fig.2. Subgrid Turbulent Viscosity detection zone is from x=24 mm to x=60 mm separated into 19 layers, each layer spaced 2 mm, the separation will be demonstrated as figure 2.

![Figure 2 detection zone schematic diagram](image)

4. Results and discussions
Simulation of parallel-column’s pressure field and speed field demonstrate as follow figure.3 figure.4 and figure.5

![Figure 3 Instantaneous pressure field when P / D = 0.2, 0.4, 0.6, 0.8, 1, 1.2 (z=15 mm)](image)

![Figure 4 Instantaneous contours of vorticity when P / D = 0.2, 0.4, 0.6, 0.8, 1, 1.2 (z=15 mm)](image)
As the simulation shows, when P/D value between 0.2-0.6 air flow is mainly affected by viscous forces when the fluid flows through the parallel-column the Blunt body area is enlarged to include the area where the two column are located. Consider the parallel-column as a unibody, Von Karman vortices appeared obvious at the tail; When the P/D value reach 0.8-1.0 the field at smooth and steady state start to show parallel flow characteristics means the force between the column can cancel each other, inverse gradient of the flow is smaller so the bounder layer separation phenomena become weak, so the overall tend to be flow smoothly. Doesn’t interfere each other at the tail; When the P/D value reach 1.2, it appeared obvious at the tail that the pressure are losing lead to some part appeared to biased flow, the flow at the tail start to unstable as well.

At proximal part P/D value doesn’t affect the Von Karman vortices a lot, trend to gradually decreasing, means P/D affect a little to degree of confusion around the parallel-column in proximal. It tend to rise at the main part which means parallel-column mainly affected in this part. Easy to find from the figure that when P/D value reach between 0.8-1.0 has the most stable flow. At the expansion area overall tend to decline no mater how P/D value changed, so the result is considered meet the fact of our simulation.
Figure 7 different proximal area to P/D Von Karman vortices value schematic diagram

From the diagram is not hard to tell, at the proximal area the fluid is much stable. But at the same time when X reach 26mm the Von Karman vortices reach its maximum, so when X=26mm the fluid is most unstable. Also we can learn from the figure 3 fluid is in the low air pressure area which means the fluid at the proximal area is in dynamic stable form.

So do more analysis on the flowing form of this area can help to optimize the structure and the vibration as well in the future development.

5. Conclusion

(1) Under air as medium the P/D value has more significant influence on the flow around the parallel column, as the P/D value more than 0.8 air flow as Von Karman vortices; when the value reach between 0.8-1.0 air flow as parallel fluid; when the value go above 1.0 air flow as vortex side flow.

(2) Fluid form can be distinguish as three sectors: proximal, main and expansion. The influence of P/D value mostly reflect on the main area.

Acknowledgments
We are grateful to Key platforms and major projects of guang dong provincial education department (No.2017KTSCX212);

References
[1] SUMNER D. Two circular cylinders in cross-flow: a review [J]. Journal of Fluids & Structures, 2010, 26(6): 849
[2] CANTWELL B J, COLES D. An experimental study of entrainment and transport in the turbulent near wake of a circular cylinder [J]. Fluid Mech, 1983, 136(11): 321
[3] CHANG K S, SONG C J. Interactive vortex shedding from a pair of circular cylinders in a transverse arrangement [J]. International Journal for Numerical Methods in Fluids, 2010, 11(3): 317
[4] NISHIMURA H, TANIIKE Y. Aerodynamic characteristics of fluctuating forces on a circular cylinder [J]. Journal of Wind Engineering & Industrial Aerodynamics, 2001, 89(7): 713
[5] ACHENBACH E, Distribution of local pressure and skin friction around a circular cylinder in cross-flow up to Re = 5 × 106[J]. Fluid Mech. 1968, 34(4):625
[6] NORBERG C. Fluctuating lift on a circular cylinder: review and new measurements [J]. Journal of Fluids & Structures, 2003, 17(1)