ISBN 978-81-934174-9-2
8th International Conference on Innovations in Engineering, Technology, Computers and Applied Sciences (IETCAS-2017)
Bangkok (Thailand) Dec. 25-26, 2017

Low Reynolds Number Flow around Transversely Arranged Elliptic Cylinders in a Channel of Slit Nozzle

Young-Woo Son[1], Jong-Hwi Lee[1], Jong-Hyeon Shin[1] and Se-Myong Chang[2]

[1]Department of Mechanical Engineering, Kunsan National University, 54150, Korea

[2]School of Mechanical Convergence System Engineering, Kunsan National University, 54150, Korea

Abstract: *In this paper, the authors investigated a ploy-urethane flow for the heat protection of building construction material. In the industrial process for manufacturing, maintaining the uniform thickness is one of critical problem, and the even mixing of raw materials becomes a final goal of this research. There are important points under the consideration of fluid dynamics, which lie in their low Reynolds number between 10 and 100 where the flow regime has not been clearly understood. The obstacles such as elliptic cylinders in the parallel manner can enhance the uniformity of flow as well as high vorticity inside the flow field, which is very near from the creeping flow region, but somewhat different from Re < 1. Computational fluid dynamics (CFD) technique is used for the comparison of velocity distribution at the nozzle outlet and the overall lumped parameters such as pressure drop and maximum vorticity, etc. The result shows that a specific configuration of the gap interval for cylinders determines the optimum characteristics of design for the slit nozzle, and, in addition to these significant findings in engineering, the unordinary behavior of the low-Reynolds-number flow due to the effect of flow physics concerning the high viscosity of fluid.*

Keywords: *Elliptic Cylinder, Mixing Enhancement, Laminar Flow, Low Reynolds Number, Nozzle, CFD*

1. Introduction

Urethane foams are widely used for the adiabatic material thanks to it significant strength, endurance, elasticity, and heat cut-off capacity, etc. The hard urethane board in Fig. 1 is manufactured trough tour-step process such as mixing, reaction, injection, and cutting. A precise pump sends mixing liquids to the mixing chamber, and the mixed liquid is injected to a slowly moving conveyor belt, which starts to make foams, passing a pressure transducer, to be hardened to solid. The final process is cutting it in regular size. The temperature and mixing ratio are main parameters determining mechanical properties like density, strength, and hardness, etc [1]. Among them, the uniformity of flow is one of the most important factors for the better quality of urethane-foam boards.

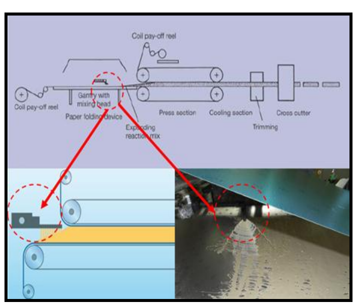
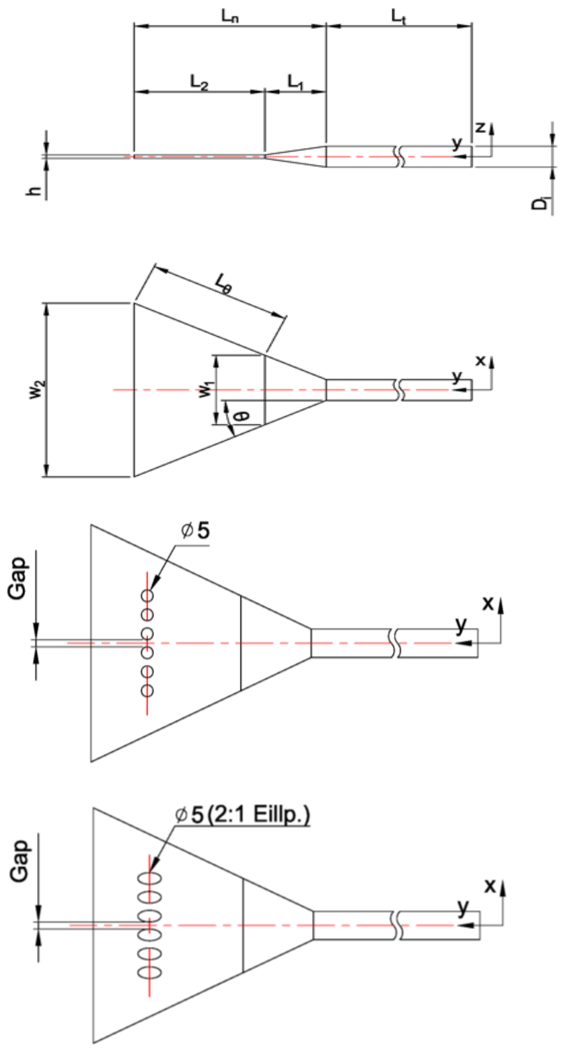


Fig. 1: The manufacturing process of polyisocyanurate boards



(a)

(b)

(c)

Fig. 2: The models used in this research: (a) the baseline model, (b) circular model, (c) 2:1 ellipse model.

However, or the flow uniformity in such low Reynolds number flow of 10 to 100 is not known. So far the micro slit nozzles of 50 μm gap, for example, has been investigated to produce uniform flow by changing the shapes, but the Reynolds number is ranged less than unity, so it lies in a creeping flow region [2]. There have been many researches about high Reynolds numbers more than 10^4, or turbulent flow, for electrical dust collectors. The effect of area ratio on the velocity profiles of both inlet and outlet is studied for a diffuser with given expansion angle [3]. The shape and arrangement of guide vanes are also studied about effects on the uniformity of flow [4].

In the preliminary research [5], the main element effecting on the flow diffusion was not the shapes but the distribution of bodies. Moreover, the experimental investigation in Ref. [6], although the range of Reynolds number is different from the interest of this research, shows clearly that the uniformity of cylindrical obstacles is enhanced if the smaller cylinders are used. Therefore, the configuration in Fig. 2(a-b) is used for baseline and circular model of this research, respectively. Additionally, the shape is modified to 2:1 long-axis ellipses in Fig. 2(c). The diameter of circles and the short-axis length of ellipses are all fixed to five millimetres, and the gap is the variable parameter. Table 1 is the list of all fixed parameters used in Fig. 2. The flow regime is in whole laminar one, and commercial software is used for the computation of flow uniformity and the performance of mixing.

TABLE I: Fixed Parameters; see Fig. 2(a).

Symbol	Value (unit)
L_t	400 (mm)
w_1	100 (mm)
w_2	40 (mm)
θ	25^o
L_θ	70.7 (mm)
L_n	94 (mm)
L_1	30 (mm)
L_2	64 (mm)
L_3	70 (mm)

2. Research Method

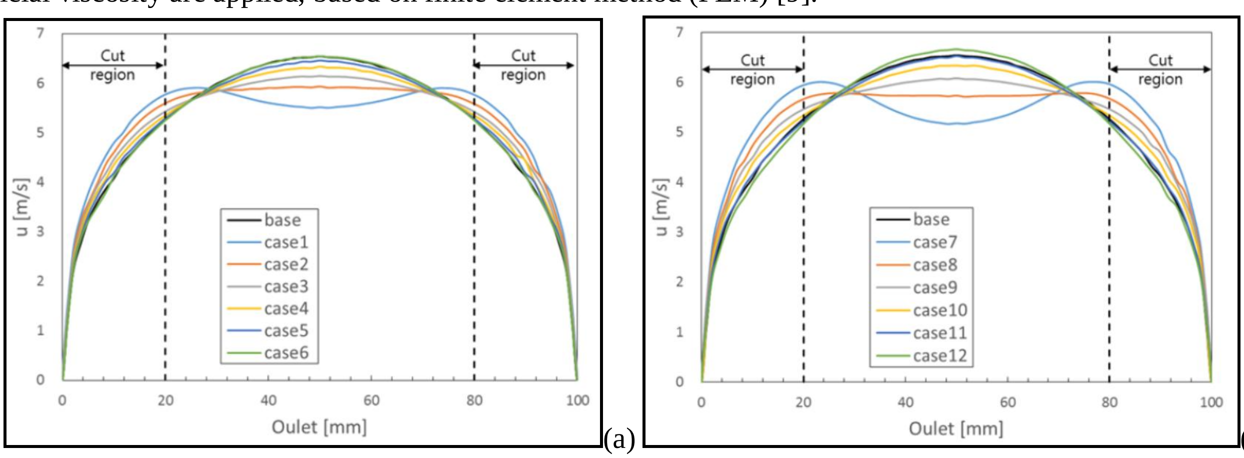
2.1. Cases of Computational Models and Grids

Six cylinders with blockage ratio of 30% are used as obstacles inside the baseline nozzle in Fig. 2(a), which are arranged transversely to the flow direction. The test cases are defined as the variable parameter of gap size from 2 to 7 mm with 1 mm increment for case 1 to 6 in circle models, and this scheme is repeated for case 7 to 12 in long-axis ellipse models where Fig. 2(b-c) are correspondent to case 2 and 8, respectively.

Tetrahedral grids are used in background, but prism layers are used near the nozzle wall, and five hexahedral layers are arranged at the wall near tubes. The sensitivity test is performed with baseline grids to achieve reasonable convergence of solutions. The total numbers of grids are 330k, and about 90% of which are concentrated near the nozzle section of injection.

2.2. Governing Equations and Boundary Conditions

For the three-dimensional incompressible flow, the governing equations are Navier-Stokes equations where the boundary conditions are categorized as fixed flow rate inlet, constant pressure outlet and no-slip wall conditions [7]. Flow properties are weight-averaged for the mixture ratio of raw materials: density and viscosity are easily calculated just from the linear combinations [8]. The 'Laminar Fluid Flow' module in COMSOL Multiphysics 5.2 is used for the numerical computation where FGMRES linearization and Petrov-Galerkin artificial viscosity are applied, based on finite element method (FEM) [9].



(b)

Fig. 3: The velocity distribution at the outlet: (a) circular models, (b) 2:1 ellipse models.

3. Result and Discussion

3.1. Flow Uniformity

Fig. 3(a-b) are the outlet velocity distribution for twelve cases described in section 2.1. The velocity distribution is changed with varied gap size between cylinders. The resistance of transverse obstacle array decreases the central velocity, but this effect is found smaller in the wide gaps. Therefore, it seems obviously that there must be an optimum gap size for this problem. The uniform profile in the central region is obtained with consideration of the effect of side walls in the nozzle. The simple Blasius profile is used in order to calculate their boundary-layer thickness [7]. By just cutting them, the RMS (root mean square) errors flow distributions from the mean values are computed as a lumped parameter for indicating flow uniformity [9]. The RMS error is plotted and compare with each other in Fig. 4 where the cutting of side-wall boundary layer will amplify the difference to judge the uniformity of flow. The less value of RMS error means better uniformity.

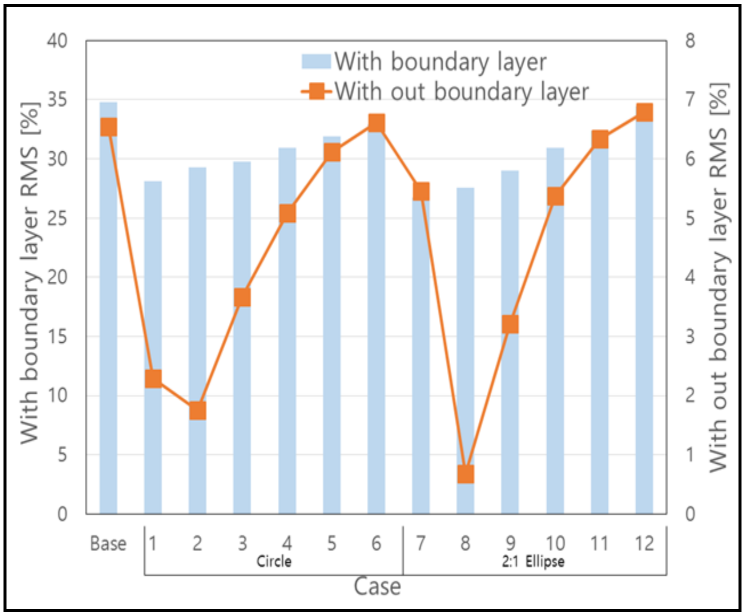


Fig. 4: The RMS errors for cases.

3.2. Mixing performance

Fig. 5 shows that the comparison of maximum vorticity and pressure drop for case 2 and 8 based on those of baseline model in Fig. 2(a). The mixture performance of maximum vorticity ratio (I_ς) is enhanced to 1.2 and 3.4 for circle and 2:1 ellipse, respectively. The shape of ellipse shows a lower RMS error in velocity profile in Fig. 3, and also better performance of mixing in Fig. 4. Therefore, the mixing performance is enhanced from the used of ellipse for the optimized gap size. As the pressure drop is only increased within 11% (see I_P in Fig. 5), the change of obstacle shape from circle to 2:1 ellipse seems economically reasonable.

The next question is why this difference is originated. Fig. 6 shows clearly that the vorticities at the wake have some difference. For the circle, the separated flow is detached from the boundary layer, and the vortices are rapidly diffused in the highly viscous flow at the wake, but they are not severely diffused in the boundary layer of 2:1 ellipse because the vortices are hold more inside the layer. Thus, the increase of maximum vorticity is explained for 2:1 ellipse model, and the flow at the outlet should be more uniform with less RMS error.

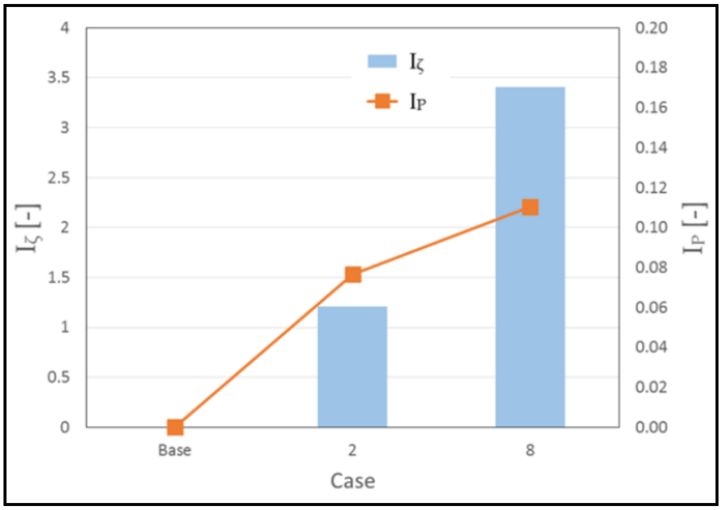


Fig. 5: Maximum vorticity (left axis) and pressure drop (right axis) for case 2 (circle shape) and case 8 (2:1 ellipse shape).

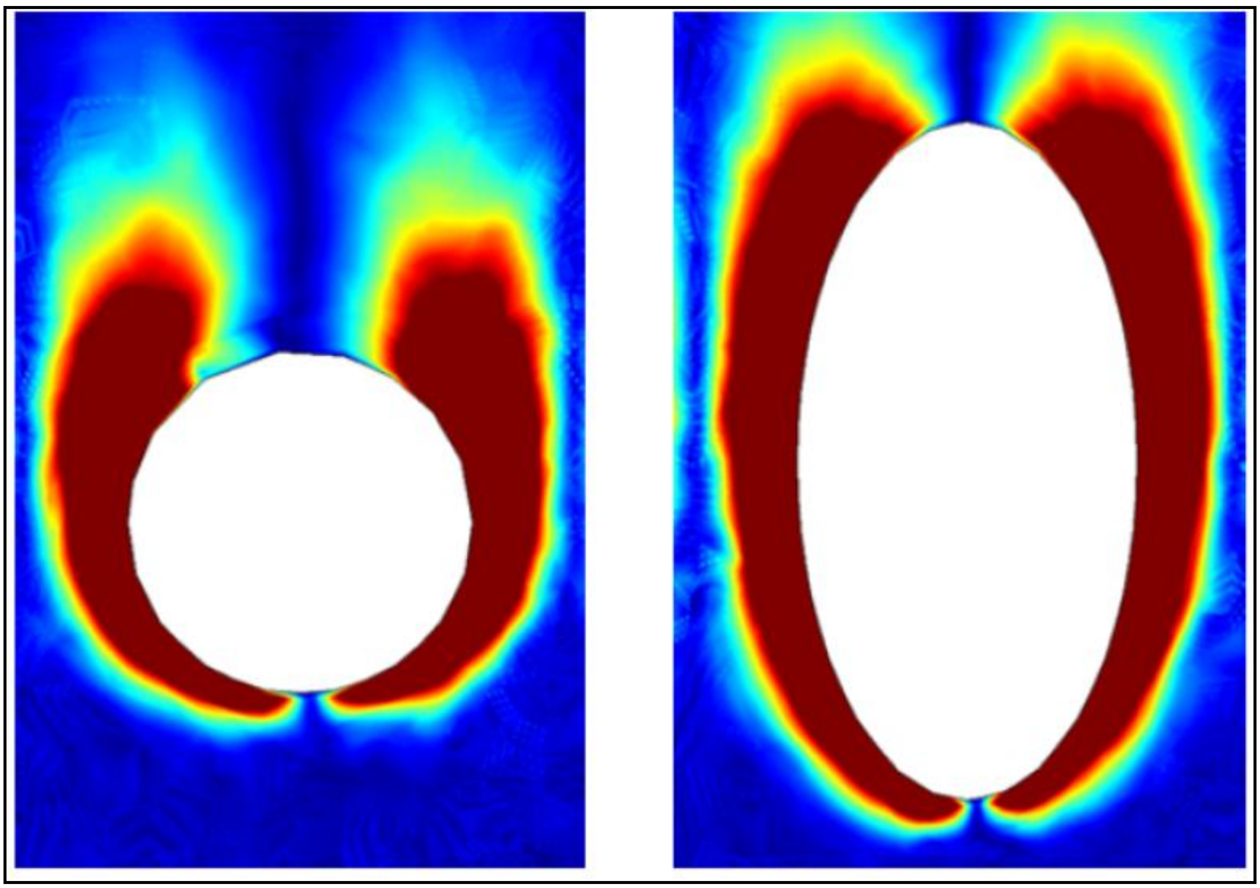


Fig. 6: Vorticity distribution around the bodies in the flow field: (left) circle, (right) 2:1 ellipse.

4. Acknowledgements

This research is supported by KETEP (Korea Institute of Energy Technology Evaluation and Planning, No. 20174010201350) financed by Ministry of Trade, Industry, and Energy in Korean government.

5. References

[1] Kim, H.S., Youn, J.W., "A Study on Foaming Characteristics of Polyurethane Depending On Environmental Temperature and Blowing Agent Content," *Transactions of Materials Processing*, Vol.18, No.3, pp. 256-261, 2009.

[2] Shon, B.C., Kwak, H.S., Lee, S.H., "A Study on Flow in a Slit Nozzle for Dispensing a Low-Viscosity Solution of Single-Walled Carbon Nanotubes," Korean Society of Computational Fluids Engineering Vol. 14, No. 1, pp. 78-85, 2009.

[3] Kwon, H.G., Park, S.H., Cho, H.H., Park, K.S., "Effect of Inlet Diffuser-Angle for Flow Uniformity of Industrial Electrostatic Precipitators," *Korean Journal of Air-Conditioning and Refrigeration Engineering*, Vol.18, No.4, pp. 328-334, 2006.

[4] Noh, K.W., Bae, S.J., Park, S.H., Kang, S.K., Lee, J.M. "Design of a Guide Vane for Improving Inside Flow Uniformity of Electrostatic Precipitator," *The Transactions of The Korean Institute of Electrical Engineers*, Vol. 62, No. 4, pp. 523-528, 2006.

[5] Son, Y. W., Chang, S.M., "CFD Analysis on the Nozzle of High Viscid Urethane Foam," Korean Society of Computational Fluids Engineering, Spring Meeting, Busan, Korea, May 20~22, pp102-103, 2017

[6] Barratt, D., Kim, T., "A Banked Wide-Angle Diffuser with Application to Electrostatic Precipitators," *Journal of Power and Energy*, Vol. 229, No. 1, pp .88~98, 2015.

[7] White, F.M., *Fluid Mechanics*, 5e, McGraw-Hill, 2009.

[8] Son, Y.W, Lee J.H., Chang, S.M, "CFD Analysis on the nozzle of high viscid raw material for urethane foam", *Journal of Computational Fluids Engineering*, Vol. 22, No. 3, pp. 79-85, 2017.

[9] Kim, H.T., Chang, S.M., Shin, J.H., Kim, Y.G., "The Feasibility of Multidimensional CFD Applied to Calandria System in the Moderator of CANDU-6 PHWR Using Commercial and Open-Source Codes," *Science and Technology of Nuclear Installation*, Vol. 2016, Article ID: 3194839, pp. 1~10, 2016.