Transactions of the Korean Nuclear Society Autumn Meeting
Pyeongchang, Korea, October 30-31, 2014

Computational Fluid Dynamic Pressure Drop Estimation of Flow between Parallel Plates

Hyung Min Son [a*], Soo Hyung Yang [a], Jong Hark Park [a]
[a]*Korea Atomic Energy Research Institute, Daedeok Daero 989-111, Yuseong Gu, Daejeon, 305-353, Korea*
[*]*Corresponding author: hyungmson@kaeri.re.kr*

1. Introduction

Rectangular cooling channel geometry has been a popular choice for heat transfer applications owing to its high surface area-to volume ratio. In nuclear industry, many research reactors are fueled by plate-type fuels which form narrow rectangular flow channels. In addition, many pool type reactors have forced downward flows inside the core during normal operation; there is a chance of flow inversion when transients occur. During this phase, the flow undergo transition between turbulent and laminar regions where drastic changes take place in terms of momentum and heat transfer, and the decrease in safety margin is usually observed. Additionally, for high Prandtl number fluids such as water, an effect of the velocity profile inside the channel on the temperature distribution is more pronounced over the low Prandtl number ones. Although the velocity distribution from the wall gives the pressure drop along the flow channel; the safety analysis code such as RELAP5 calculates the value from embedded experimental correlations based upon bulk flow conditions. This makes the checking of its pressure drop estimation accuracy less important, assuming the code verification is complete.

With an advent of powerful computer hardware, engineering applications of computational fluid dynamics (CFD) methods have become quite common these days. Especially for a fully-turbulent and single phase convective heat transfer, the predictability of the commercial codes has matured enough so that many well-known companies adopt those to accelerate a product development cycle and to realize an increased profitability. In contrast to the above, the transition models for the CFD code are still under development, and the most of the models show limited generality and prediction accuracy. Recently developed models such as one developed by Langtry and Menter (2005) exhibited greatly improved predictability and code adoptability, which became the default transition model for the widely used commercial code [1-3]. According to the literature, the formulation is still based upon experimental correlations tailored to predict the transition phenomena, and most of the works are based upon data from external flows over wing surface and flat plate. This tells us that some additional modification is required to make it more suitable for internal flows, as recently investigated by some researchers [4-7].

Unlike the system codes, the CFD codes estimate the pressure drop from the velocity profile which is obtained by solving momentum conservation equations, and the resulting friction factor can be a representative parameter for a constant cross section channel flow. In addition, the flow inside a rectangular channel with a high span to gap ratio can be approximated by flow inside parallel plates. Considering the above, in this study, a CFD calculation is carried out on the parallel plate channel geometry, and the pressure drop values are compared with those from literatures in terms of friction factors, and its applicability on the channel flow is discussed.

2. Analysis Geometry and Results

In this chapter, analysis geometry and boundary conditions are described along with applied assumptions.

2.1 Flow Channel Geometry

In this study, a two-dimensional (2D) simplification of the rectangular test section geometry taken from literature is used [8]. As shown in Fig 1, a thin slice is taken out from the test section which originally has 50 mm and 2.25 mm of width and thickness dimensions, respectively. The 2D channel flow is simulated by imposing symmetric boundary conditions to the sliced surfaces. As shown in the figure, two wall boundary conditions are placed which correspond to heater surfaces.

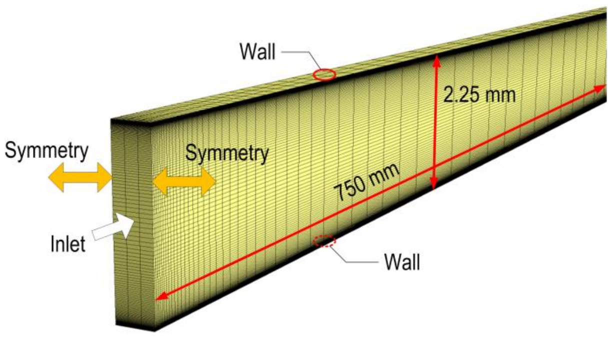

Fig. 1. Analysis geometry with boundary conditions and computational grids.

2.2 Analysis Assumption

In this analysis, *k-ω* based Shear Stress Transport (SST) turbulence model developed by Menter (1994) is solved using ANSYS® CFX code, based on the fact that the most of the validations of the above mentioned transition model has been done utilizing this model

[2,3,9]. In order to numerically solve the governing equations, geometry of interest is spatially descritized as shown in Fig 1. Resolution of the grids is kept highest near the walls to obtain the accurate flow distribution, where the nondimensional distance (y^+) of the first node from the wall is kept near one based on the guideline [3]. As a preliminary exercise, the steady-state analysis is done using isothermal water (at 298.1K, 1.5bar) as a working fluid, and the inlet velocity is controlled between 0.1 to 10 m/s to obtain flow conditions spanning from laminar to turbulent. Gravity is not considered due to isothermal forced convection assumption. For the laminar and turbulent region, the calculation is carried out until the convergence criteria (less than 10^{-5}) in terms of the root mean square (RMS) of residuals of the governing equations are met. For the transition region, less strict criteria (less than 10^{-4}) are applied.

2.3 Friction Factor Estimate

In order to see how well the code simulates the flow, the Darcy friction factor (*f*) values are compared with below correlations for the flow inside parallel plates, taken from literatures [10-12]. Along with these, well-known Colebrook and Blasius relations for pipe flow are also compared [13,14]. In this study, the hydraulic diameter of twice the channel thickness is used to calculate Reynolds number (*Re*) values.

for laminar flow:

$$f \cdot \mathrm{Re} = 96 \tag{1}$$

for fully turbulent flow (Churchill and Chan, 1994):

$$\sqrt{f/8} = 3.3618 - \frac{190.83}{\mathrm{Re}\sqrt{f/96}} + 2.5\ln\left\{\mathrm{Re}\sqrt{f/96}\right\} \tag{2}$$

for fully turbulent flow (Patel and Heat, 1969):

$$f = 0.1688/\mathrm{Re}^{1/6} \tag{3}$$

Figure 2 compares the calculated friction factors with existing correlations where we can observe that the calculated values are in reasonable agreement with values from the correlations. Based on the comparison, it is also seen that the transition region is estimated to exist approximately between Reynolds numbers from 4,000 to 10,000. These values are somewhat higher than the range (approximately 2,700~6,000) reported from the literature [12]. These differences may be coming from the presence of entrance effect, smooth wall assumption and different inlet turbulent intensity condition (assumed to be 5% in the analysis). However, the direct numerical solution study of the plane Poiseuille flow by Orszag and Kells (1980) shows that the transition can occur at Reynolds number at order 2,000 [15]. These suggest that the implemented transition model may require some adjustment to make it suitable for the channel flow, which is discussed in next subsection.

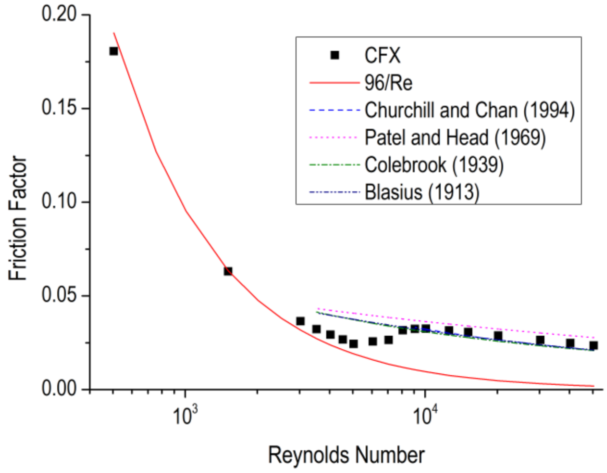

Fig. 2. Comparison of calculated friction factors with existing correlations.

2.4 Modification of Transition Region

The transition model utilized in this study solves two transport equations of intermittency (γ) and transition momentum thickness Reynolds number ($\widetilde{\mathrm{Re}}_{\theta t}$), which controls the production and destruction terms in the transport equation of turbulent kinetic energy (*k*) [1-3]. Equations (4) and (5) show the intermittency equation and destruction source term, respectively [2]. In this study, as a preliminary exercise, the destruction source term (E_γ) is adjusted by adjusting the value of the constant c_{e2} which controls the lower intermittency limit, and its effect on the transition region is seen.

$$\frac{\partial(\rho\gamma)}{\partial t} + \frac{\partial(\rho U_j \gamma)}{\partial x_j} = P_\gamma - E_\gamma + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_f}\right)\frac{\partial \gamma}{\partial x_j}\right] \tag{4}$$

$$E_\gamma = c_{a2}\rho\Omega\gamma F_{turb}(c_{e2}\gamma - 1) \tag{5}$$

Figure 3 shows the friction factor distributions with different value of c_{e2}, where it is observed that the applying smaller value resulted in the transition region in lower and narrower Reynolds number range (roughly 3,500~6,000)

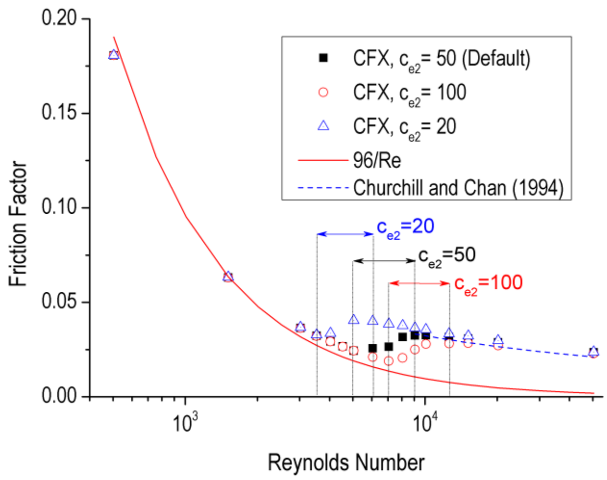

Fig. 3. Effect on friction factor from adjusted transition model.

In addition to the above, a turbulent friction factor for the c_{e2}= 20 case is fitted using least square method as shown in following relation with adjusted R^2 value of 0.998. The developed correlation is found to give values between that of Churchill and Chan (1994) and Patel and Head (1969).

for turbulent flow (6,000<Re<50,000):

$$f = 0.023 + 0.025e^{-6.774 \cdot 10^{-5}\mathrm{Re}} \quad (6)$$

3. Conclusions

The computational fluid dynamics simulation on the flow between parallel plates showed reasonable prediction capability for the laminar and the turbulent regime. In contrary, it is observed that the embedded transition model may require some adjustment to make it suitable for the internal flows. Preliminary sensitivity study on the transition modeling constant showed reduced value resulted in the lower and the narrower transition region in terms of Reynolds number. The turbulent friction factor correlation obtained in this study may require additional comparison study to gain the applicability. In addition, this analysis process can be readily extended to three dimensional rectangular flows to obtain more meaningful results and is left as a future study item.

ACKNOWLEDGEMENTS

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (NRF-2012M2C1A1026916).

REFERENCES

[1] R. B. Langtry, F. R. Menter, Transition Modeling for General CFD Applications in Aeronautics, 43rd AIAA Aerospace Sciences Meeting and Exhibit, Jan.10-13. 2005, Reno, NV.

[2] F. R. Menter, R. B. Langtry, and S. Volker, Transition Modeling for General Purpose CFD Codes, Flow, Turbulence and Combustion, Vol.77, p.277, 2006.

[3] ANSYS CFX-Solver Theory Guide, ANSYS, Inc., Canonsburg, PA, 2009.

[4] J. P. Abraham, E. M. Sparrow, and J. C. K. Tong, Breakdown of Laminar Pipe Flow into Transitional Intermittency and Subsequent Attainment of Fully Developed Intermittent or Turbulent Flow, Numerical Heat Transfer, Part B: Fundamentals: An International Journal of Computation and Methodology, Vol.54, p.103, 2008.

[5] J. P. Abraham, E. M. Sparrow, and J. C. K. Tong, Heat Transfer in All Pipe Flow Regime: Laminar, Transitional/Intermittent, and Turbulent, International Journal of Heat and Mass Transfer, Vol.52, p.557, 2009.

[6] W. J. Minkowycz, J. P. Abraham, and E. M. Sparrow, Numerical Simulation of Laminar Breakdown and Subsequent Intermittent and Turbulent Flow in Parallel-plate Channels: Effects of Inlet Velocity Profile and Turbulence Intensity, International Journal of Heat and Mass Transfer, Vol.52, p.4040, 2009.

[7] J. P. Abraham, E. M. Sparrow, J. C. K. Tong, and D. W. Bettenhausen, Internal Flows Which Transist from Turbulent Through Intermittent to Laminar, International Journal of Thermal Sciences, Vol.49, p.256, 2010.

[8] Y. Sudo, Core Heat Transfer Experiment for JRR-3 to be Upgraded at 20 MWt, Part I: Differences in Force Convection Heat Transfer Characteristics between Upflow and Downflow for Rectangular Vertical Flow Channel (in Japanese), Japan Atomic Energy Agency, Muramatsu, 1984.

[9] F. R. Menter, Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications, AIAA Journal, Vol.32, p.1598, 1994.

[10] R. K. Shah, A. L. London, Laminar Flow Forced Convection Heat Transfer and Flow Friction in Straight and Curved Ducts- A Summary of Analytical Solutions, Stanford University, Stanford, CA, 1971.

[11] S. W. Churchill, C. Chan, Improved Correlating Equations for the Friction Factor for Fully Turbulent Flow in Round Tubes and between Identical Parallel Plates, both Smooth and Naturally Rough, Industrial and Engineering Chemistry Research, Vol.33, p.2016, 1994.

[12] V. C. Patel, M. R. Head, Some Observations on Skin Friction and Velocity Profiles in Fully Developed Pipe and Channel Flows, Journal of Fluid Mechanics, Vol.38, p.181, 1969.

[13] C. F. Colebrook, Turbulent Flow in Pipes, with Particular Reference to the Transition Region between Smooth and Round Pipe Laws, Journal of the Institution of Civil Engineers, Vol.11, p.133, 1939.

[14] P. R. H. Blasius, Das Aehnlichkeitsgesetz bei Reibungsvorgangen in Flüssigkeiten, Forschungsheft, Vol.131, p.1, 1913.

[15] S. A. Orszag, L. C. Kells, Transition to Turbulence in Plane Poiseuille and Plane Couette Flow, Journal of Fluid Mechanics, Vol.96, p.159, 1980.